ME422

2016-17 Winter Quarter

Dr. Jones

Final Project

ANSYS Memo

Thaddeus Hughes

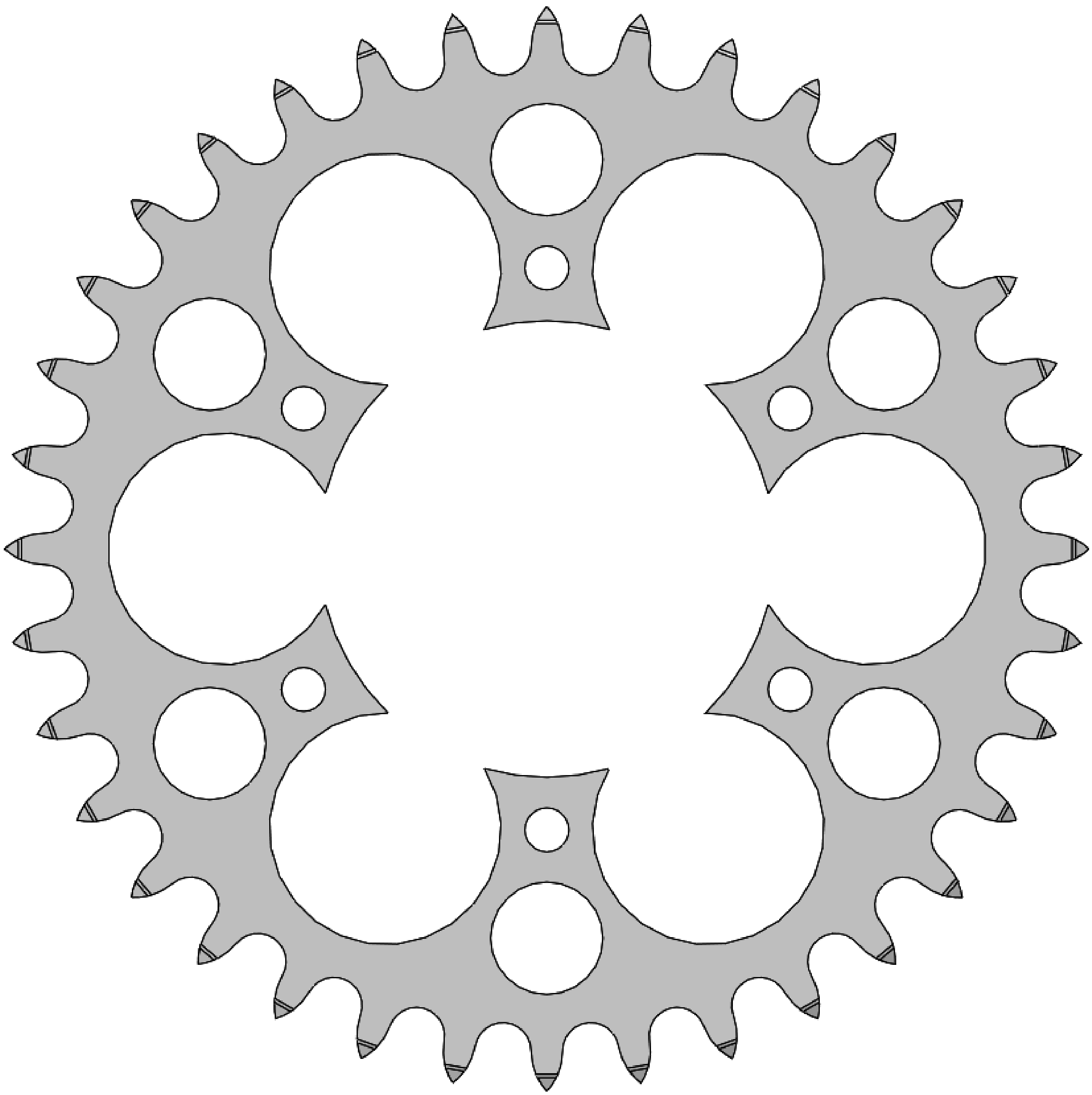
CM 2411

Revised: February 16, 2017

1: Introduction: Background and goals

Cars in the Formula SAE competition frequently use a chain drive as a final drive. A robust sprocket is imperative to not only having any driving functionality, but also smooth power transmission. Additionally, reducing system mass is always a goal, especially in rotational mass. Larger mass decreases vehicle agility.

A static structural finite element analysis of one such sprocket loaded via chain has been performed, as well as a brief design study to determine a pocketing strategy that will decrease mass while still keeping structural integrity. First, two validation models are used and results from them are compared to analytic solutions.



Engineering Data is linked throughout all models. 7075-T651 aluminum data was found from http://www.matweb.com/search/DataSheet.aspx?MatGUID=4f19a42be94546b686bbf43f79c51b7d, and 4340 Steel data was found from http://www.matweb.com/search/DataSheet.aspx?MatGUID=948f14a1d835445ab11c681ddb1bc2c6. The following properties have been used for each material throughout this analysis:

Material	4340 Steel	7075-T651 Aluminum
Young's Modulus (ksi)	29000	10400
Poisson's Ratio	0.29	0.33
Yield stress (psi)	141000	67000
Density (lb/in^3)	0.284	0.102

Table 1.1: Material properties used throughout analysis

2: Validation Model: Chain pulling on pinned hole

The first validation model used is to verify the setup and usage of multiple components and contacts in the FEA model. This is important, as using a model that includes the chain will spread force across multiple teeth in the sprocket.

The model used is shown below, and consists of two chain links (made of 4340 steel) in blue and green, and a block with a pin hole through it (made of 7075-T651 Aluminum). SOLIDWORKS CAD Models can be found at https://goo.gl/91XEMf and drawing files for these components can be found attached at the end of this document. The goals of this validation model are to ensure:

1. Stress concentration produced on the pink block matches that of a static analysis
2. Stresses generated within the links match that of a static analysis
3. Find a mesh size that doesn't take too long to compute while giving results that are within 3% of analytic solution

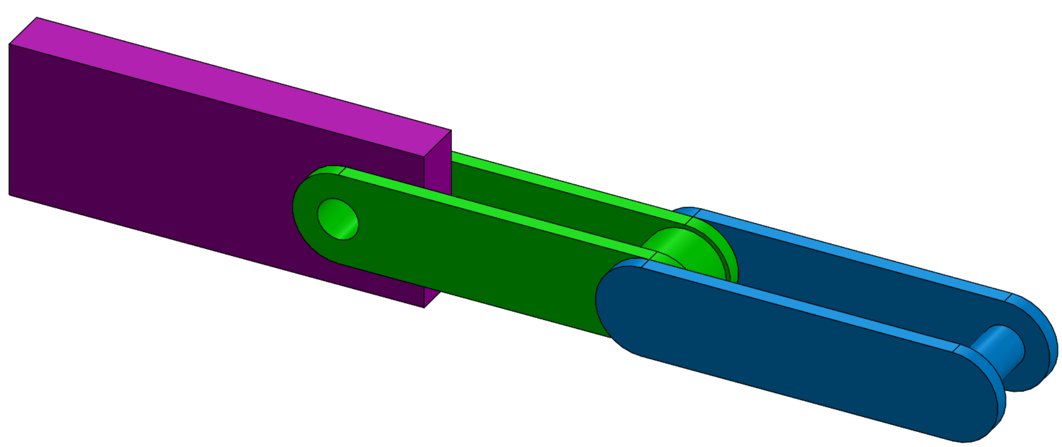


Figure 2.1: Overview of assembly used for validation model

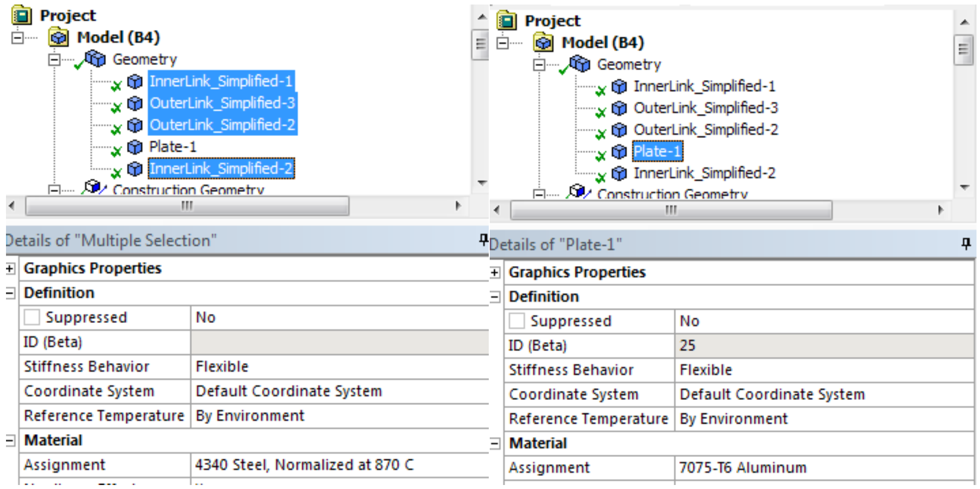


Figure 2.2: Unmodified screenshots of material assignment in analysis.

There are two frictional contacts used between bodies: Friction between links is 0.16, and friction between the links and the aluminum is 0.6. (http://www.engineeringtoolbox.com/friction-coefficients-d_778.html) These contacts can be added in the Model tree by going to Connections->Contacts->(right click)->Insert->Manual Contact Region .

The contact between the plate and the link, shown in figure 2.3, connects the sides of the plate touching the link and the inner hole to the sides of the link touching the plate and the outside of the link.

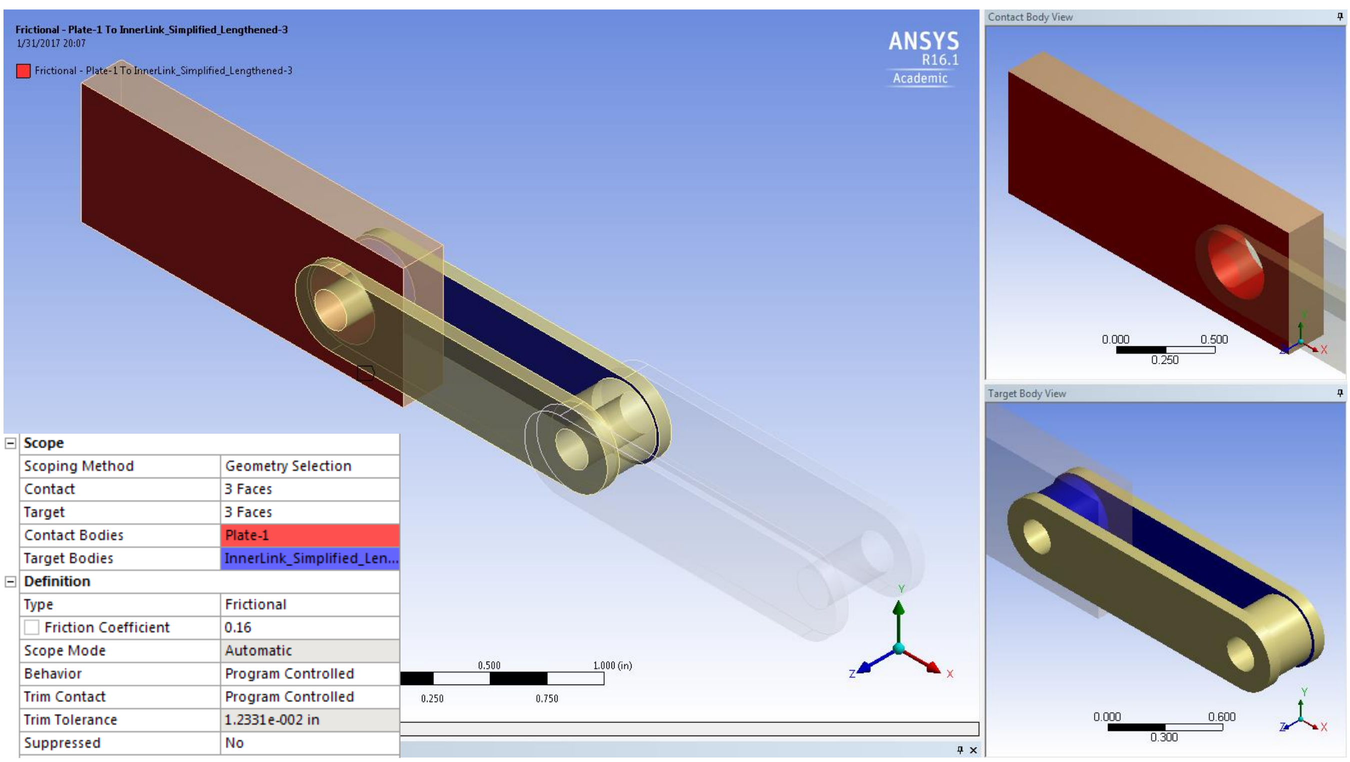


Figure 2.3: Contact between plate and link. Left window: Overview. Top right: Cutaway view of contacts on plate (in red). Bottom right: Cutaway view of contacts on link (in blue).

The contact between the inner and outer link, shown in figure 2.4, connects the sides of the link touching the link and the inner hole to the sides of the link touching the plate and the outside of the link.

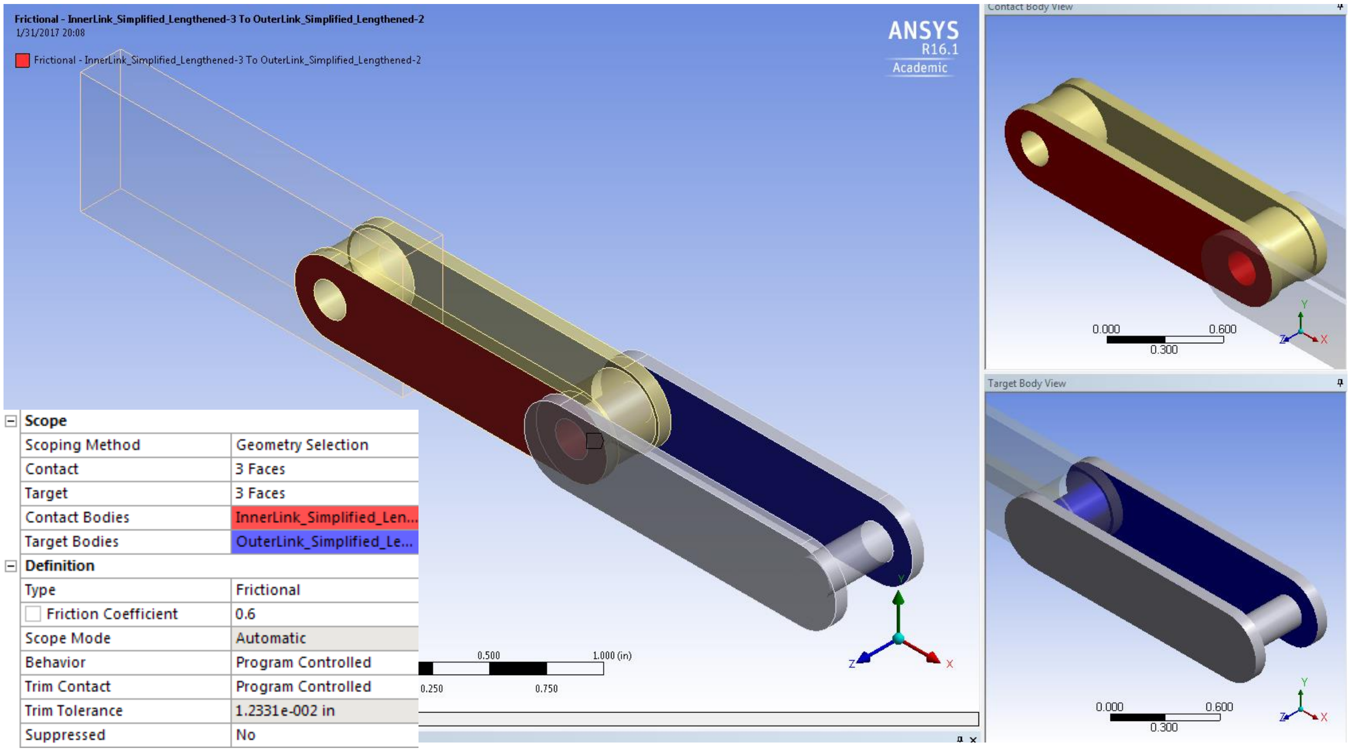


Figure 2.4: Contact between links. Left window: Overview. Top right: Cutaway view of contacts on inner link (in red). Bottom right: Cutaway view of contacts on outer link (in blue).

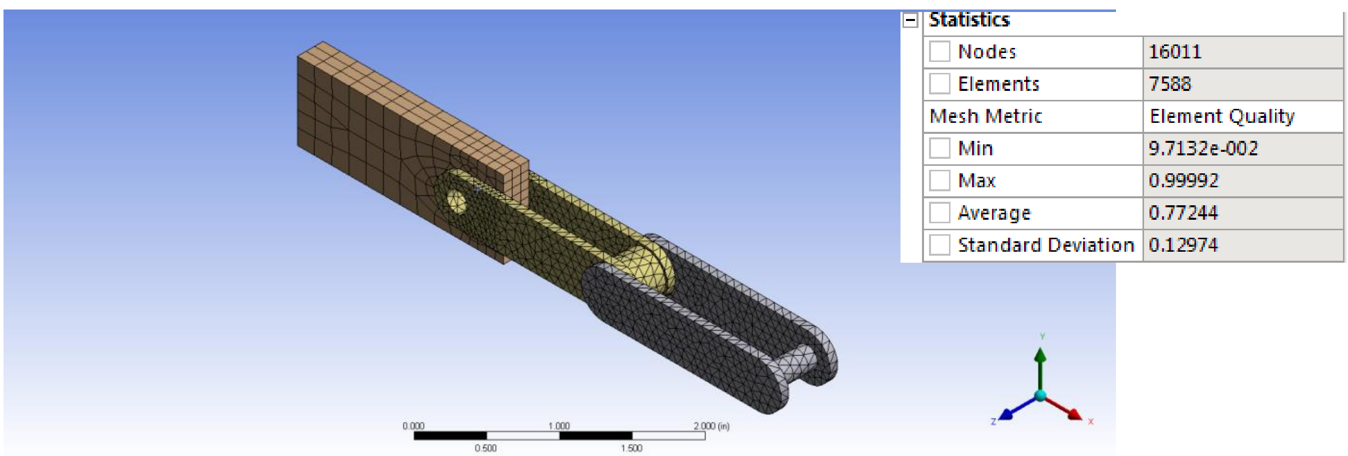
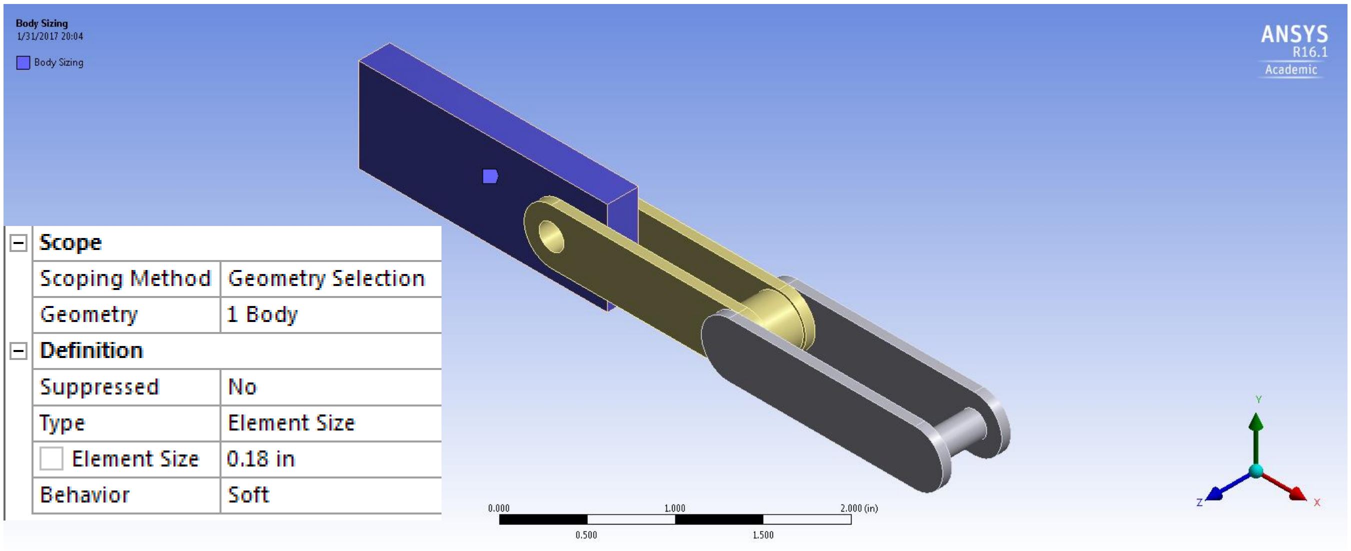


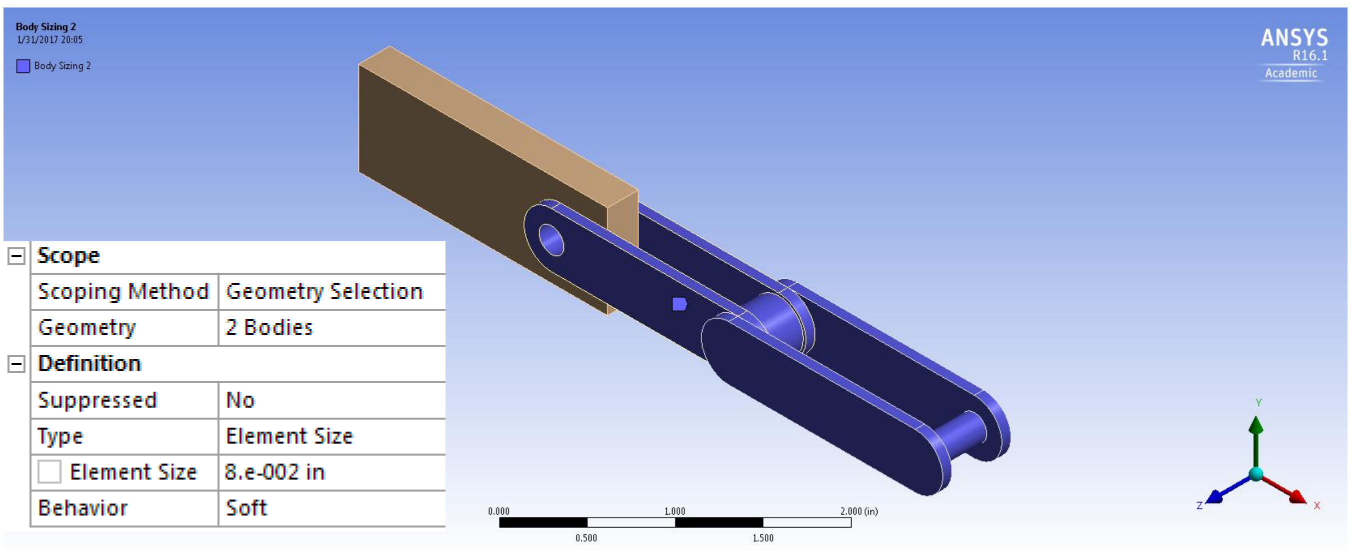
Figure 2.3: Mesh and quality statistics

Although never explicitly requested, the plate uses a combination of quadrilateral and triangular elements, and the links have triangular elements. The model's mesh is set by body sizing for the links and the plate with hole as shown in figure 2.4. After that a contact sizing is set as shown in figure 2.6. A mesh convergence study was conducted as shown below in Table 2.1, along with analytic results to validate against. Although mesh ID 3 is the finest mesh, a goal of this study is to determine a mesh size which is also computationally efficient. Mesh ID 4 has similar error to the analytic solution as mesh 3, but computation time is nearly halved. This will be important, as this validation model takes about 2 minutes to solve already; the final model will take even longer. Although higher meshes appear to also converge within the needed bound, I've erred on the side of a finer mesh to aid convergence in analysis later.



Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	0.18 in
Behavior	Soft

Figure 2.4: Body sizing for plate



Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Suppressed	No
Type	Element Size
Element Size	8.e-002 in
Behavior	Soft

Figure 2.5: Body sizing for chain links

Contact Region	Frictional - Plate-1 To InnerLink_Simplified_Lengthened-3
Definition	
Suppressed	No
Type	Element Size
Element Size	5.e-002 in

Figure 2.6: Contact sizing for plate-link contact

Mesh ID	Link Mesh Size (in)	Plate Mesh Size (in)	Contact Sizing (in)	Max Normal stress around pin hole (psi)	% Error in stress	Max Normal Stress in Link (psi)	Min Normal Stress in Link (psi)	Avg Normal Stress in Link (psi)	Error in stress
1	0.25	0.5	0.15	23982	19.29%	16056	14435	15245.5	0.62%
2	0.1	0.25	0.06	29629	0.29%	15729	14700	15214.5	0.42%
3	0.1	0.25	0.04	28959	2.54%	15859	14546	15202.5	0.34%
4 (finest)	0.075	0.15	0.04	29001	2.40%	15860	14562	15211	0.40%
5 (final)	0.08	0.18	0.05	29628	0.29%	15853	14538	15195.5	0.29%
Analytic solution				29714	-	-		15151	-

Table 2.1: The mesh convergence study for the validation model. Sample readings can be found in figures 2.8 and 2.9.

A fixed support is applied to the left side of the block, and a 1000 lbf force is applied to the pin of the rightmost link. This places the model in tension.

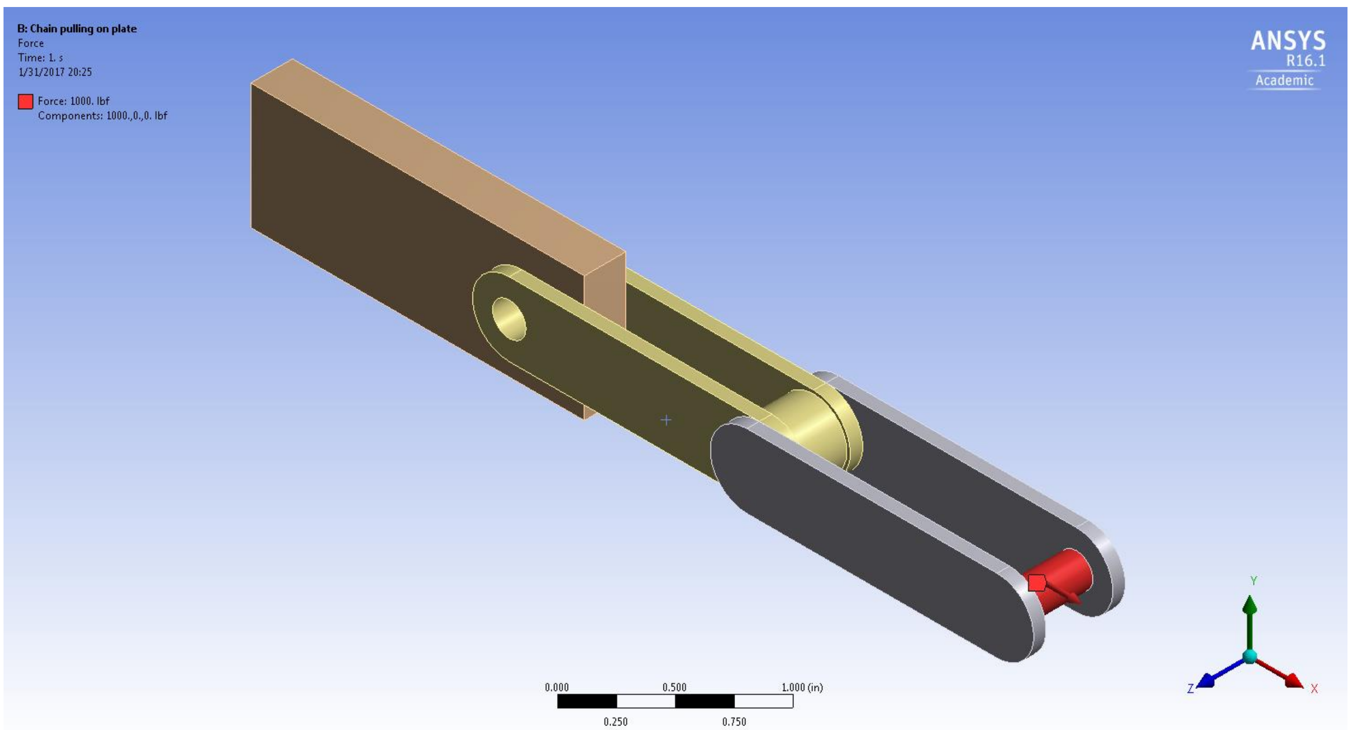


Figure 2.7: Tension force applied to outer link

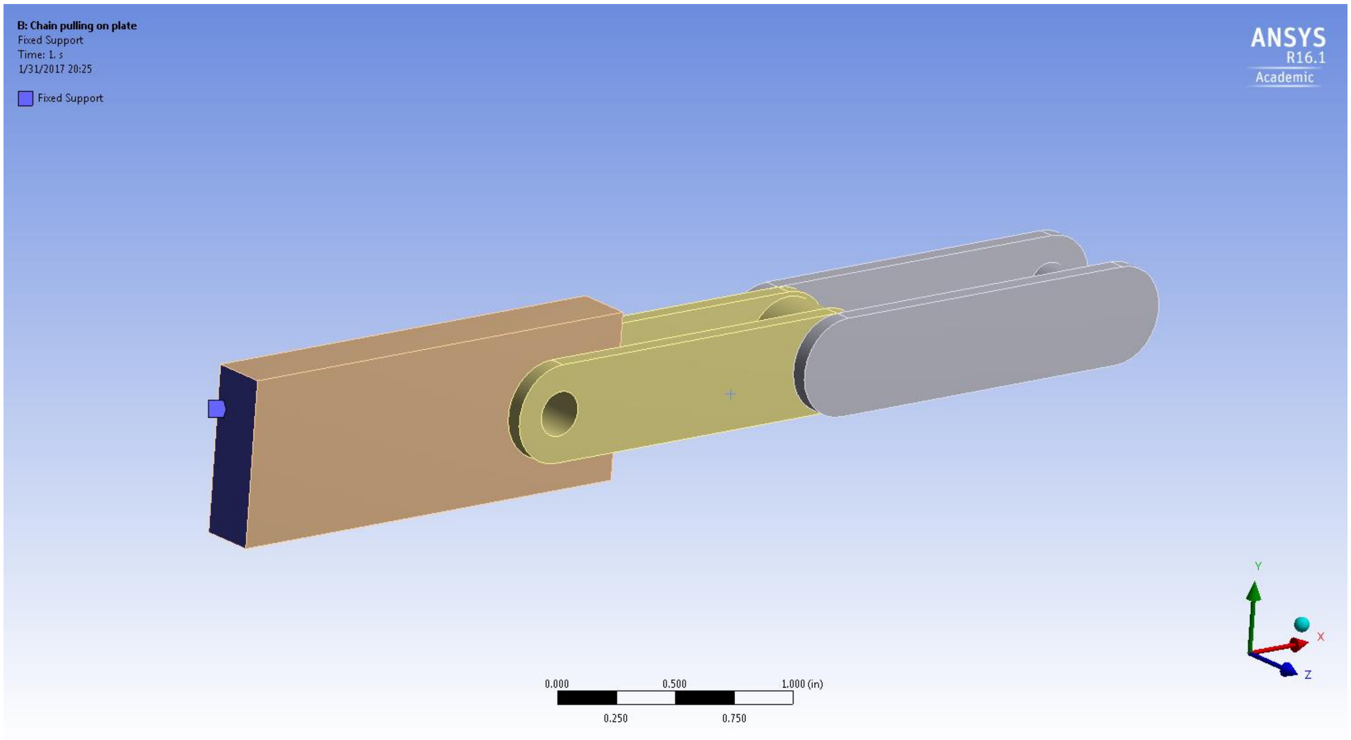


Figure 2.8: Fixed support on left end of plate

Normal stress in the +x direction for the link and pin hole are shown below.

Note the linear stress distribution within the link. This should be expected, but because of the contacts used, obtaining an analytic solution for the slope is difficult. As such, we will only compare the average stress in the link. As noted in table 2.6, the results agree within 0.3%, well below the 3% requirement.

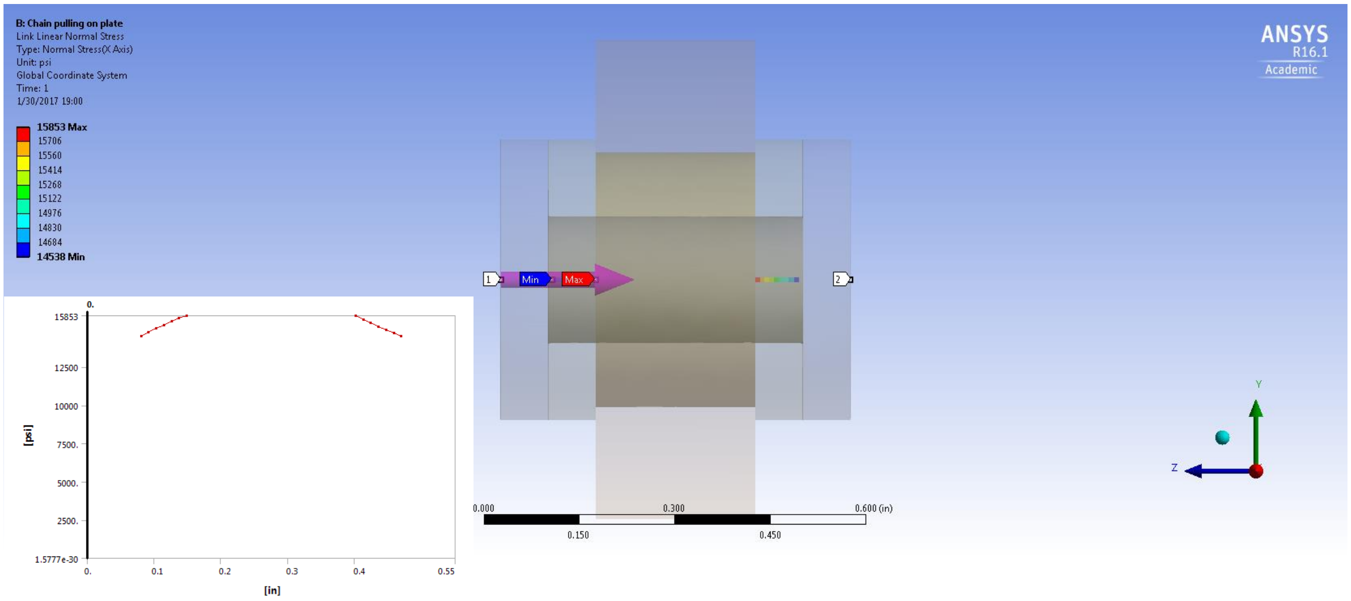


Figure 2.9: Stress in middle of inner link, along a line through the midplane. Min: 14538 psi, Max: 15853 psi, Average: 15195.5 psi

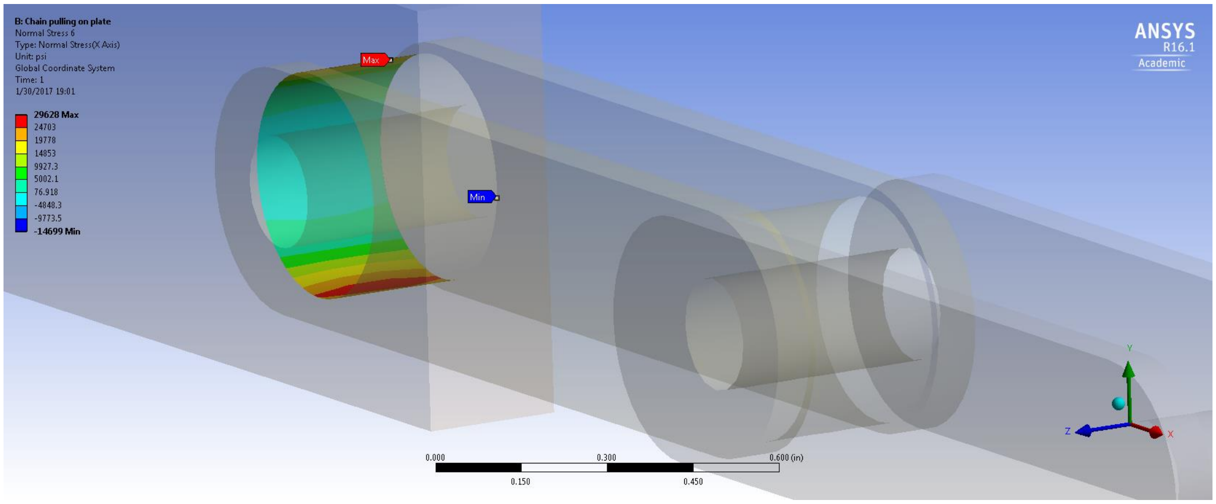


Figure 2.10: Normal stress in +x direction around hole in plate. Max: 29628 psi

Analysis Data Management	
Solver Files Directory	C:\Users\hughest1\EngineeringWorkspace\ME422\FSAE_Sprocket\FSAE_Sprocket_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	Bin

TABLE 13
Model (B4) > Analysis

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	71.6 °F
Generate Input Only	No

Figure 2.11: Solution confirmation and simulation authorship confirmation

3: Validation Model: Pulling on plate with compression only support

This second validation model used is to verify the setup and usage of compression-only supports around the bolt pattern for the sprocket. The model appears split to enforce a symmetry constraint which will assist in convergence for this validation model, however, this will not be needed on the final model as there will be multiple pin holes. Aside from this split, the part is identical in geometry to the plate with hole used in the first validation model.

The model used is shown below. SOLIDWORKS CAD Model can be found at https://goo.gl/l3HrOR and drawing files for this component can be found attached at the end of this document. The goals of this validation model are to ensure that the stress concentration around the pinned hole matches that of the analytic solution (which should be the same as this component from part 2).

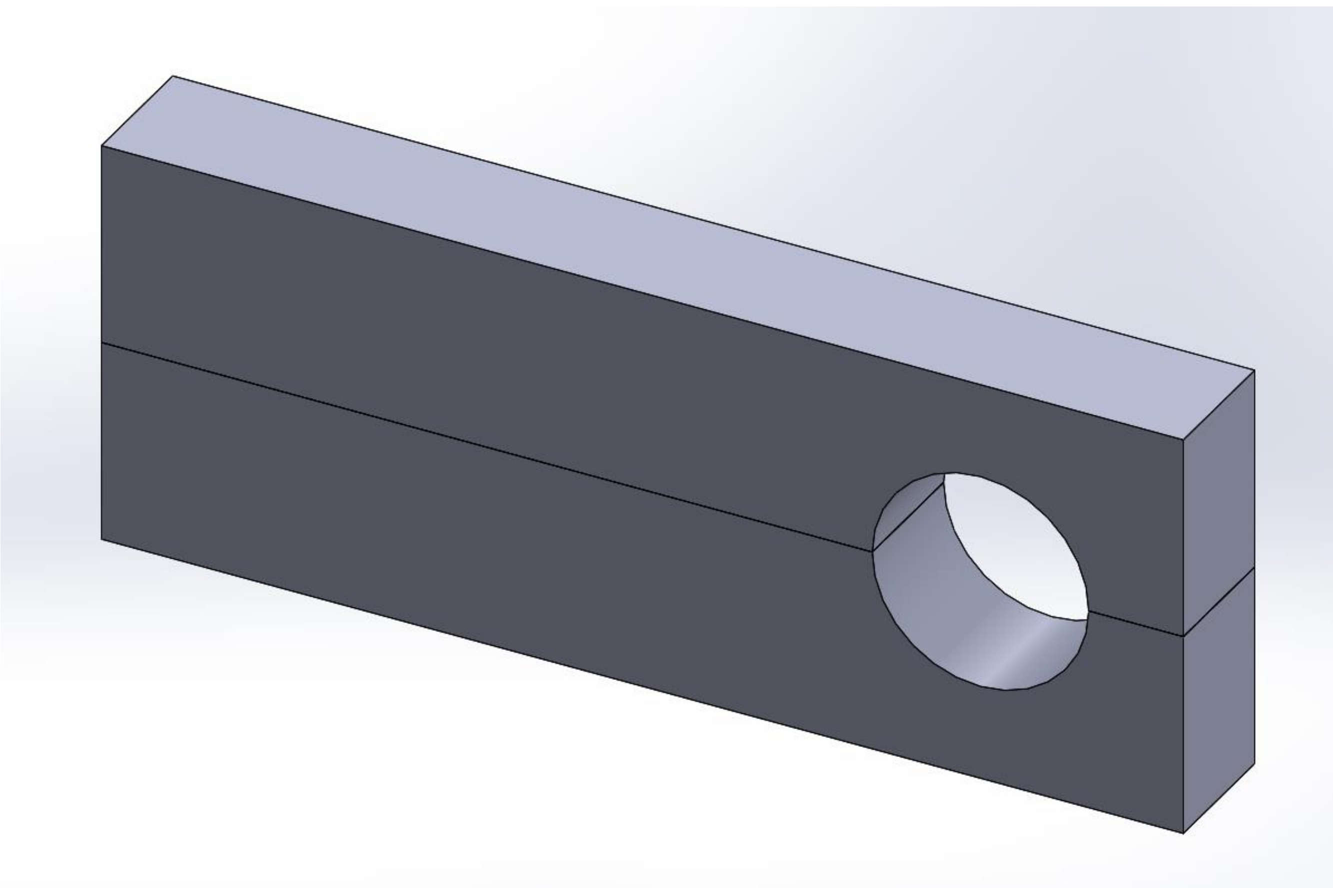


Figure 3.1: Isometric view of part used for validation model 2

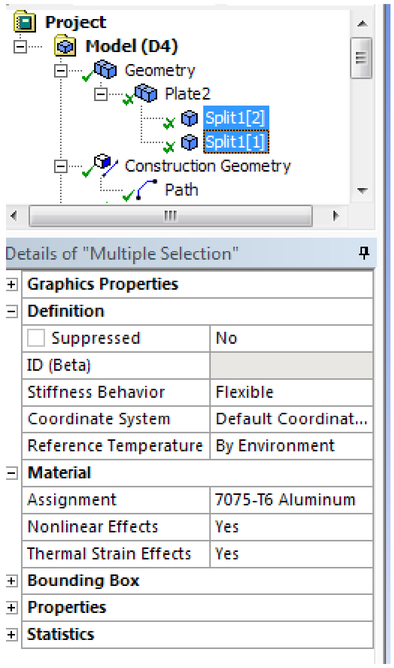


Figure 3.2: Unmodified screenshot of material assignment in analysis.

There is one bonded contact used between the two split halves.

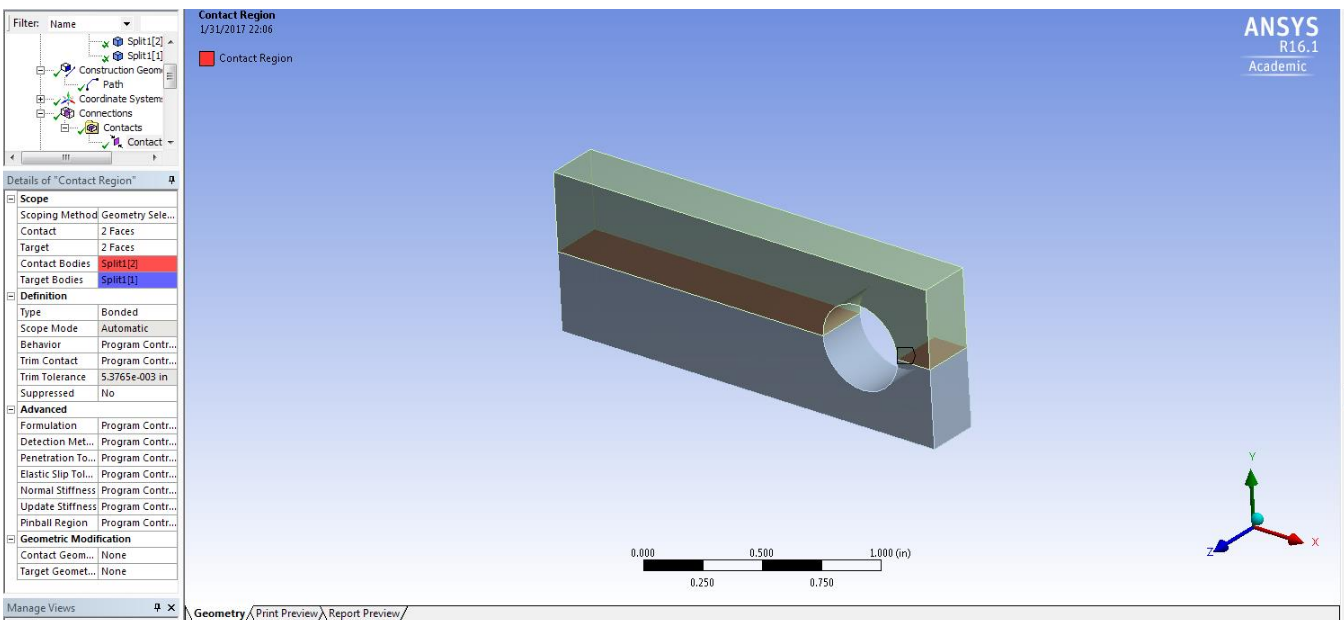


Figure 3.3: Contact between plate halves

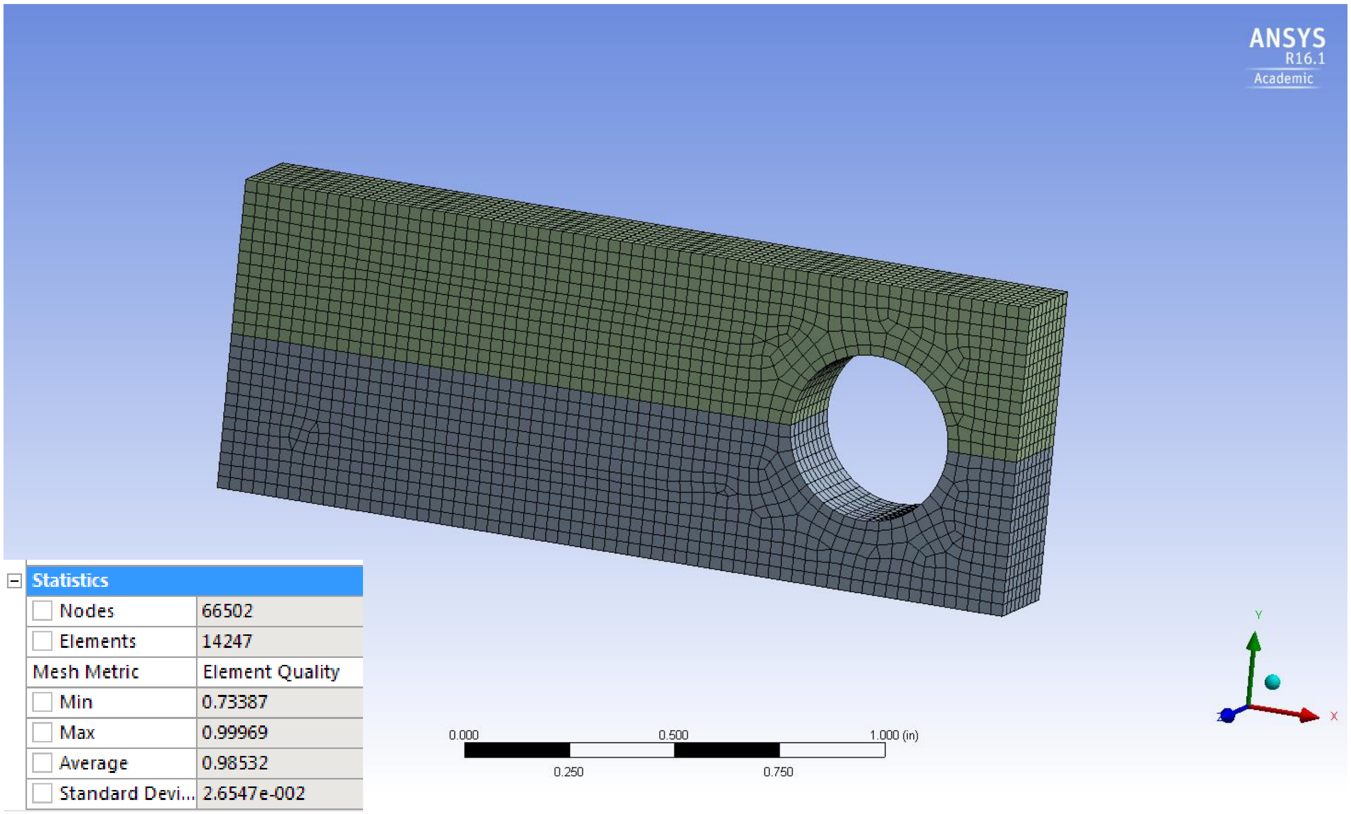


Figure 3.4: Mesh and quality statistics

Although not asked for explicitly, the model uses quadrilateral elements. The model's mesh is set by body sizing. After that a face sizing is set. A mesh convergence study was conducted as shown below, along with analytic results to validate against. Mesh ID 5 is the final mesh chosen due to convergence to under 3% error while still computing quickly.

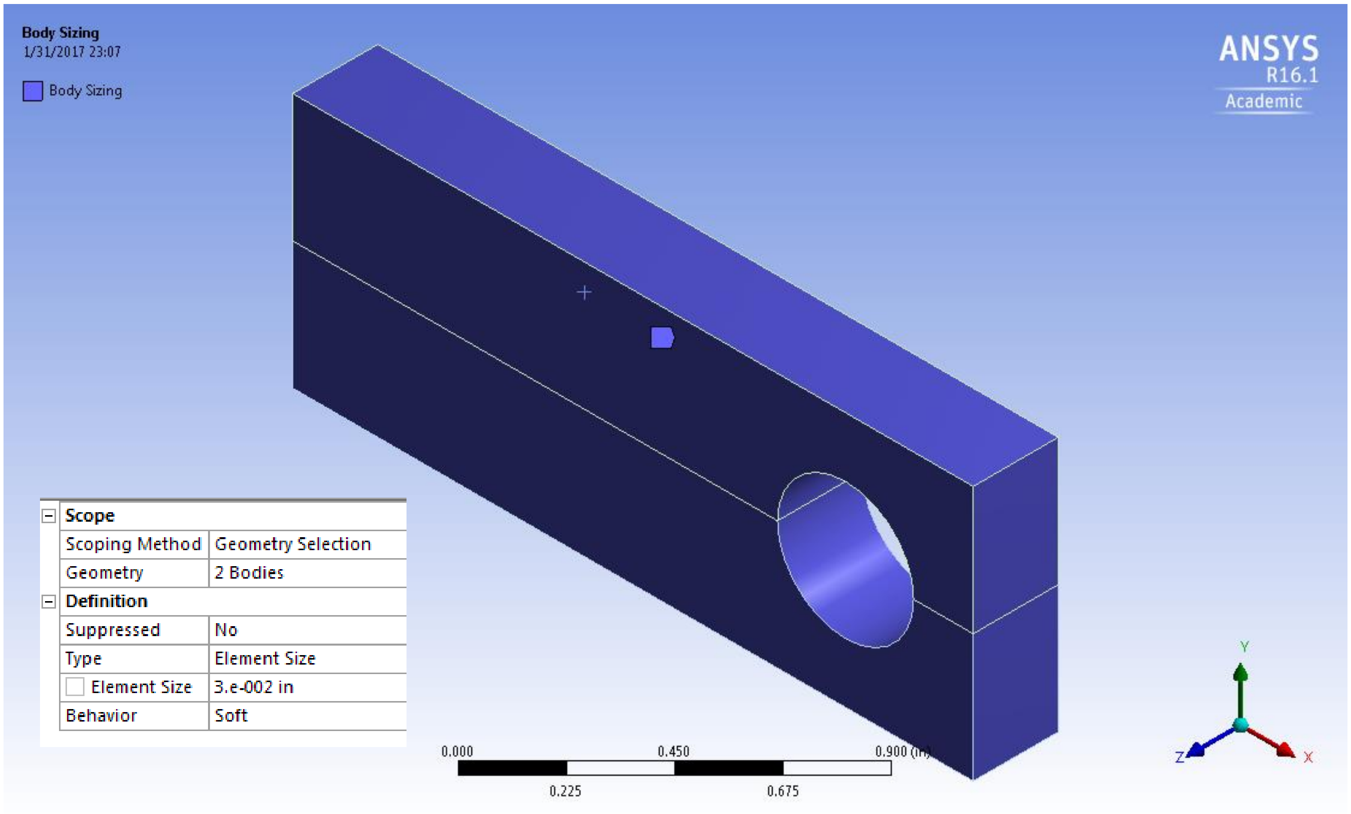


Figure 3.5: Body sizing for plate: 0.003 in

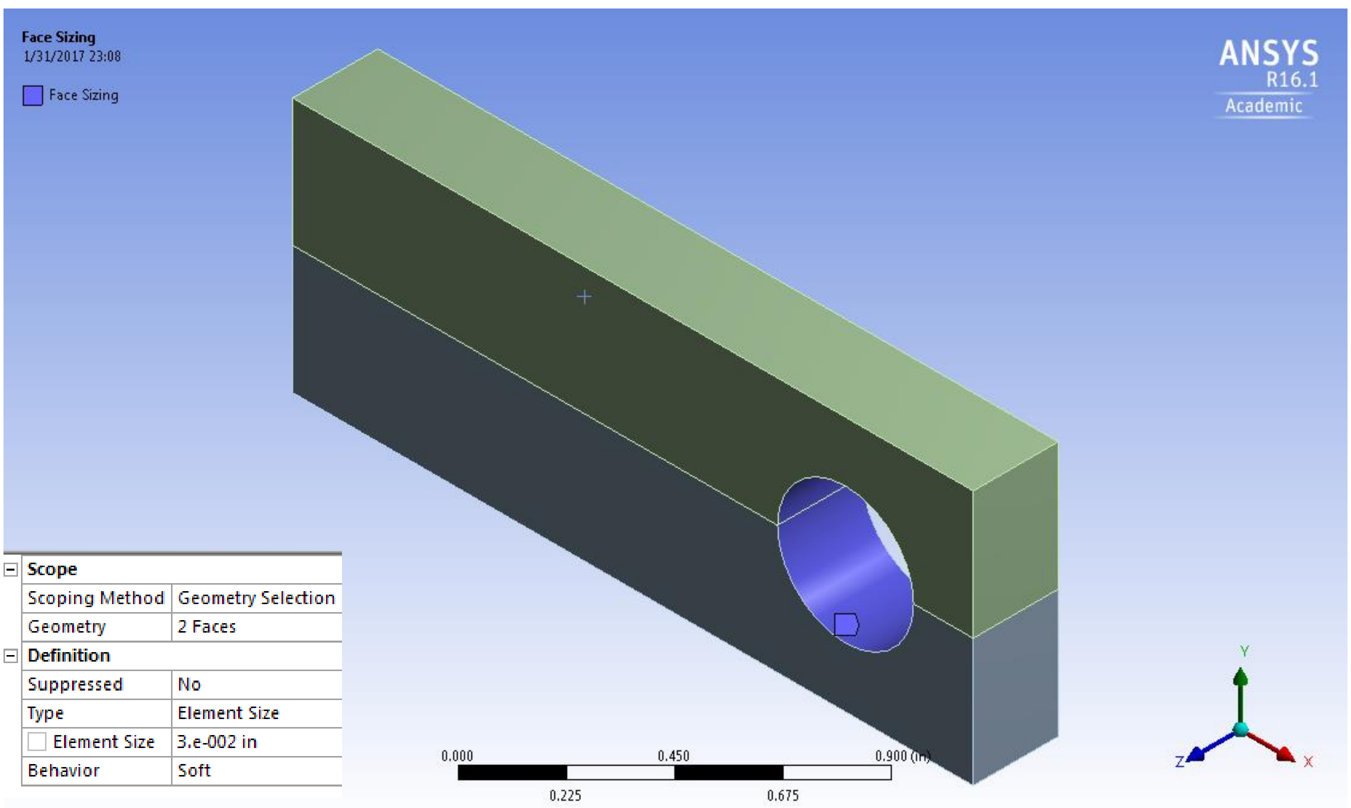


Figure 3.6: Face sizing for pin hole: 0.002 in

Mesh ID	Body Mesh Size	Face Mesh Size	Max Normal stress around pin hole	% Error
1	0.1	0.1	24925	16.12%
2	0.1	0.05	24154	18.71%
3	0.1	0.03	26382	11.21%
4	0.05	0.03	26350	11.32%
5	0.03	0.03	28994	2.42%
6	0.03	0.02	29512	0.68%
Analytic Model Prediction			29714	

Table 3.1: The mesh convergence study for the validation model. Sample reading can be found in Figures 3.10.

A compression-only support is applied to the left side of the block, and a 1000 lbf force is applied to the left of the link. This places the model in tension. Additionally, one of the faces at the split of the part is constrained to zero y displacement; enforcing model symmetry and helping solution stability.

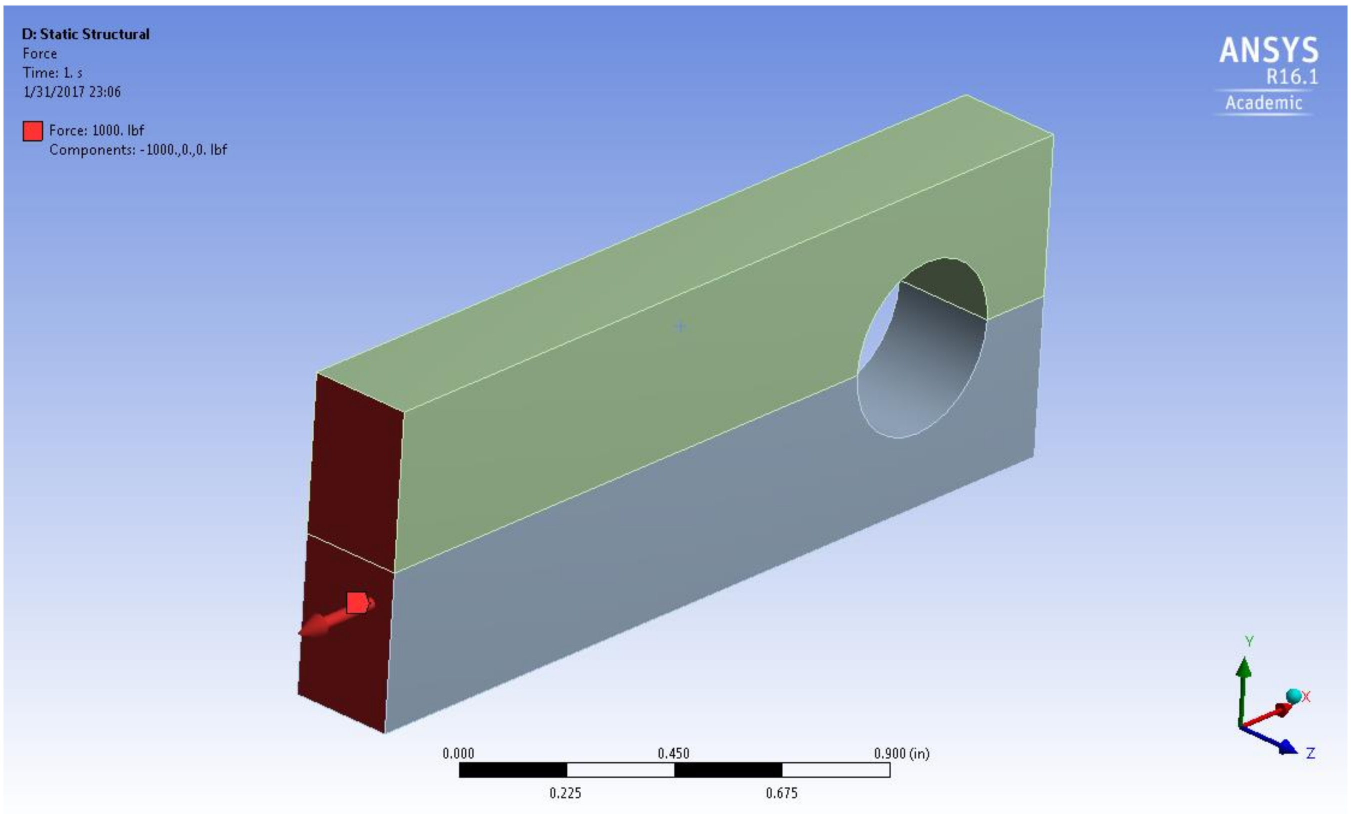


Figure 3.7: Tension force applied to left face of plate

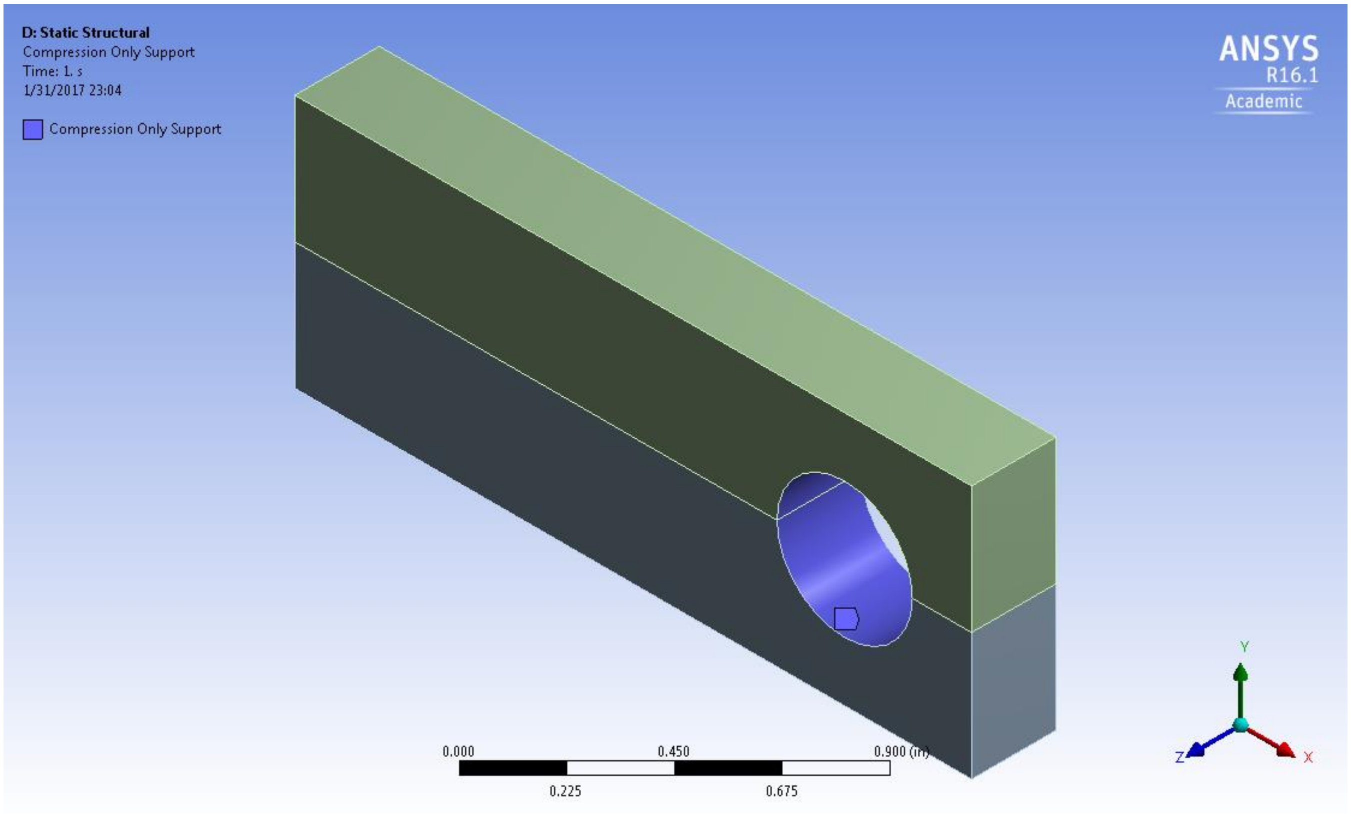


Figure 3.8: Compression-only support for pin hole

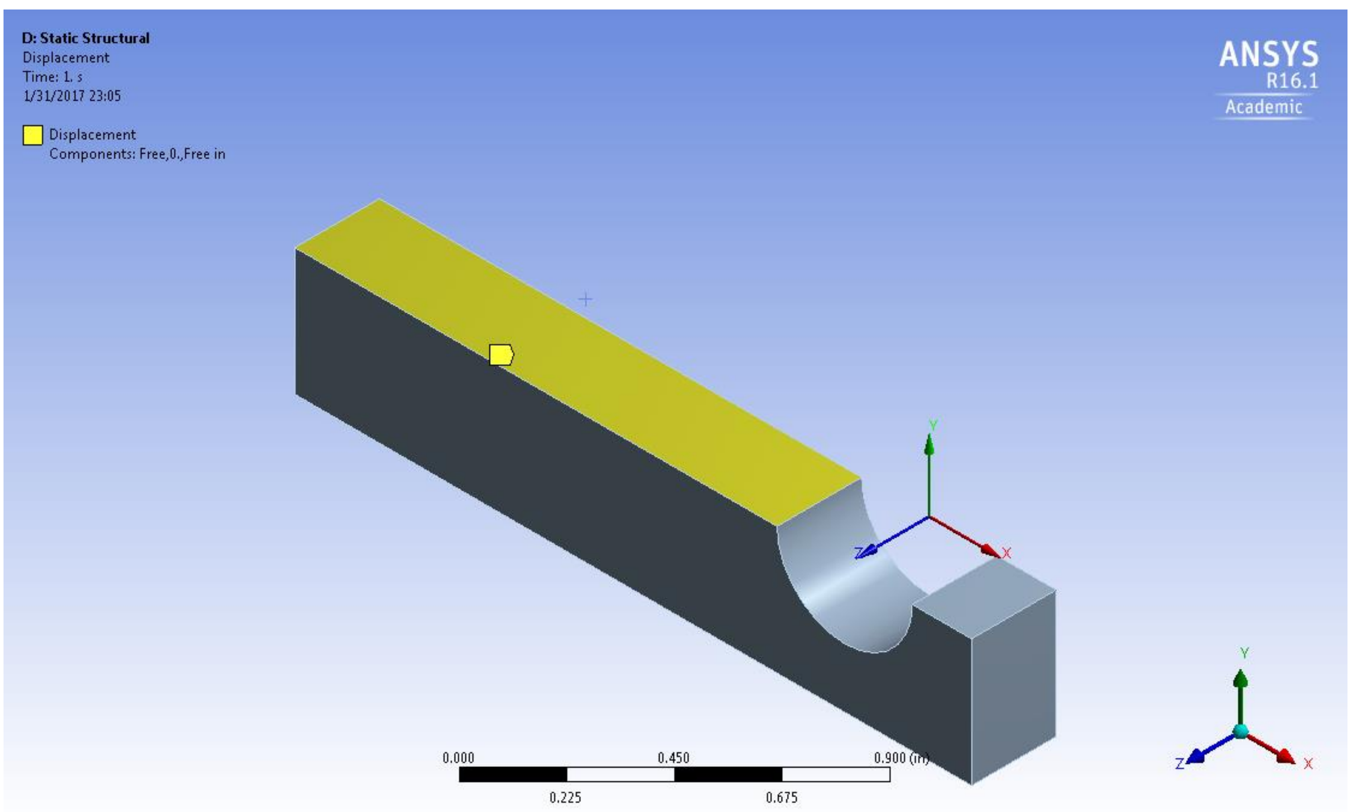


Figure 3.9: Symmetry support for middle of part

The normal stress in the +x direction is shown below. As expected, the maximum normal stress occurs around the pin hole.

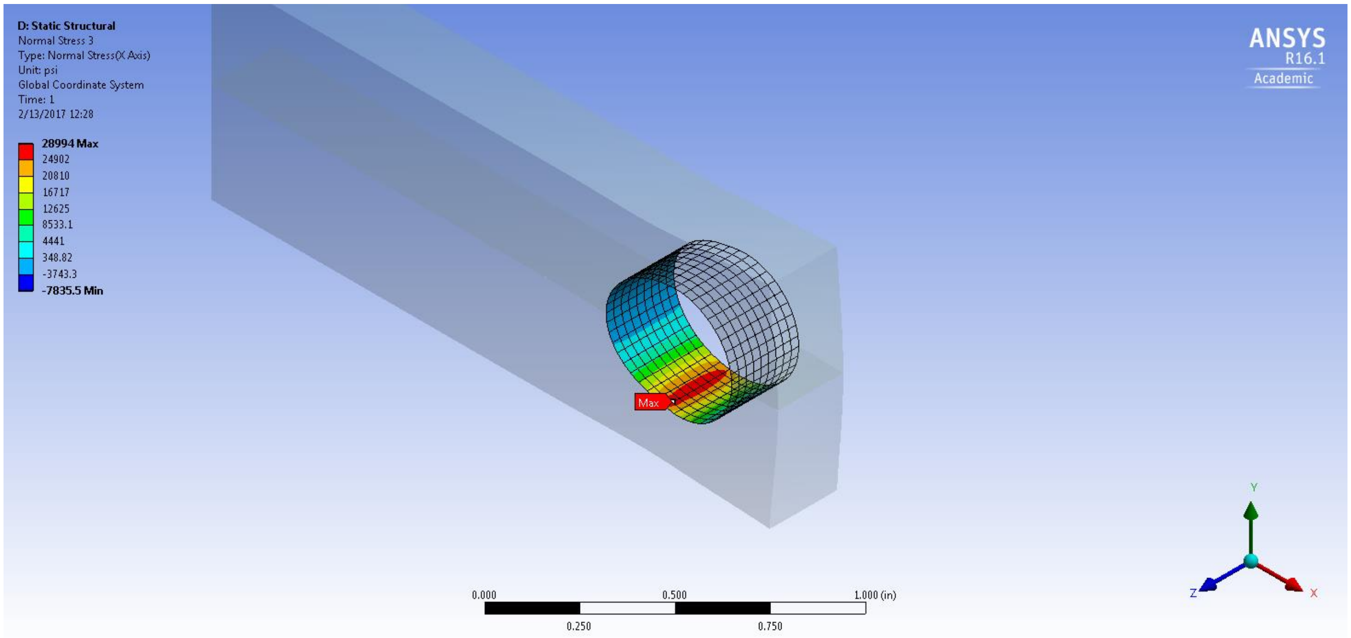


Figure 3.10: Stress in plate. Max stress around pin: 28994 psi

Analysis Data Management	
Solver Files Directory	C:\Users\hughest1\EngineeringWorkspace\ME422\FSAE_Sprocket\FSAE_Sprocket_files\dp0\SYS-2\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	Bin

TABLE 13
Model (D4) > Analysis

Object Name	*Static Structural (D5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	71.6 °F
Generate Input Only	No

Figure 3.11: Solution confirmation and simulation authorship confirmation

4: Final Model, Sans Pockets

This phase of the analysis is to determine a mesh size suitable for the final simulation by conducting a mesh convergence study on an unpocketed sprocket.

The model used is shown below, and consists of many chain links (made of 4340 steel) wrapped around a sprocket (made of 7075-T651 Aluminum). SOLIDWORKS CAD Models can be found at https://drive.google.com/file/d/0B_lA0xR4_viqRzFLeXNUUkpIYTA/view?usp=sharing. The model has been set up this way to model the distributed load the chain provides.

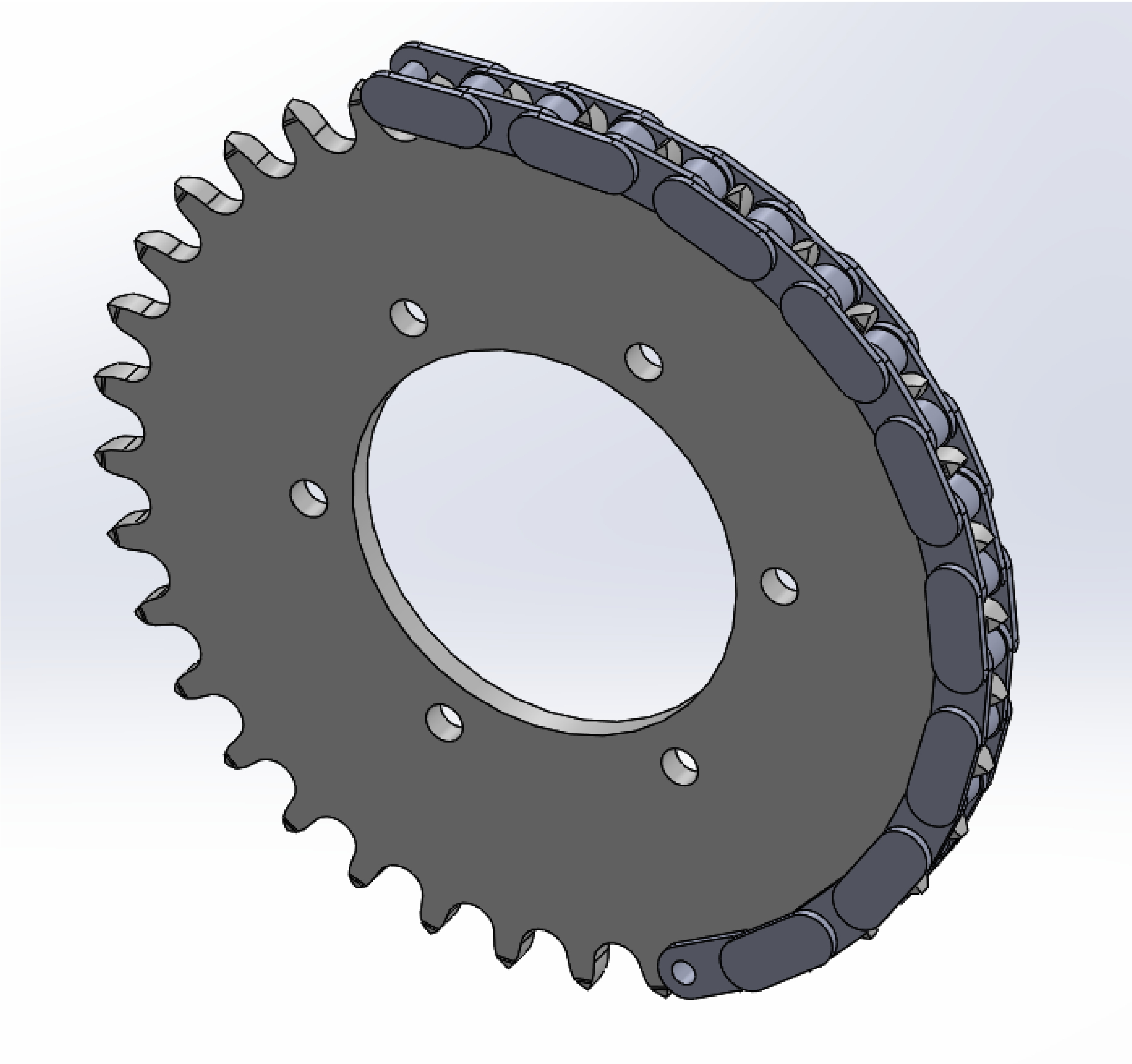


Figure 4.1: Overview of assembly used for mesh convergence model

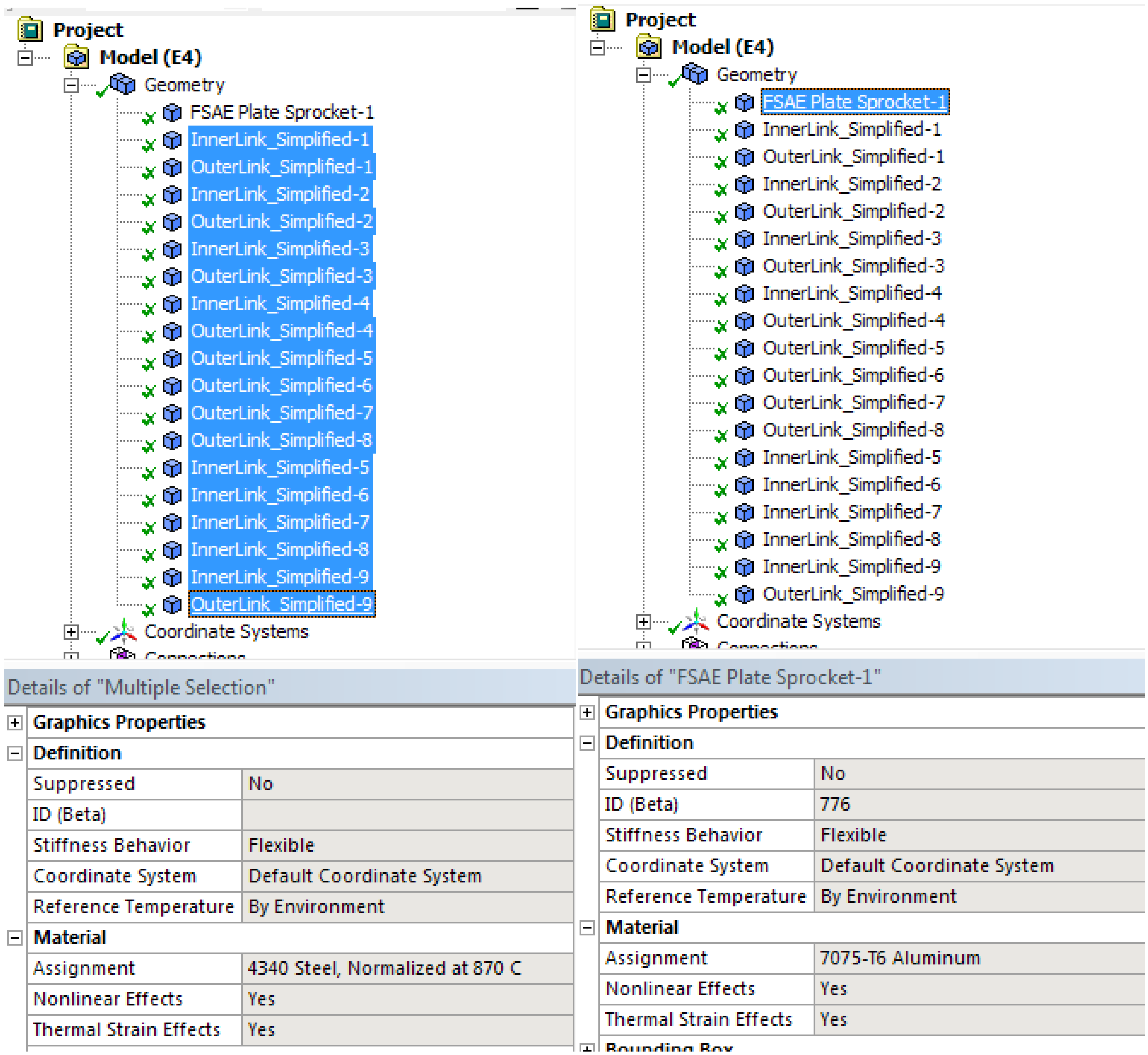


Figure 4.2: Unmodified screenshots of material assignment in analysis.

There are two types of contacts used between bodies: Friction between links is 0.16, and friction between the links and the aluminum is 0.6. (http://www.engineeringtoolbox.com/friction-coefficients-d_778.html) These contacts can be added in the Model tree by going to Connections->Contacts->(right click)->Insert->Manual Contact Region. Because there are so many links, only examples of these two contact types are shown.

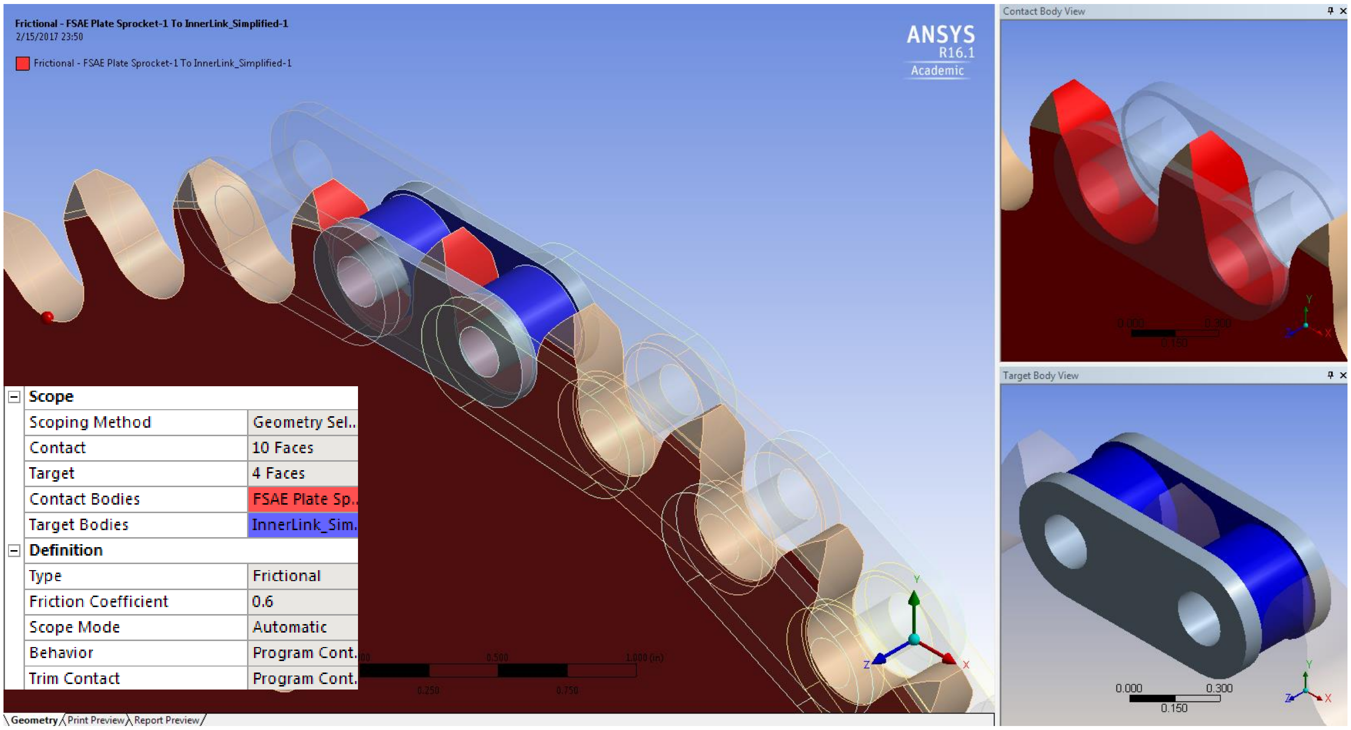


Figure 4.3: Contact between sprocket and inner link. Left window: Overview. Top right: Cutaway view of contacts on sprocket (in red). The contact points in red are the sides of the sprocket and the interior faces of the sprocket teeth contacting the inner link. Bottom right: Cutaway view of contacts on inner link (in blue).

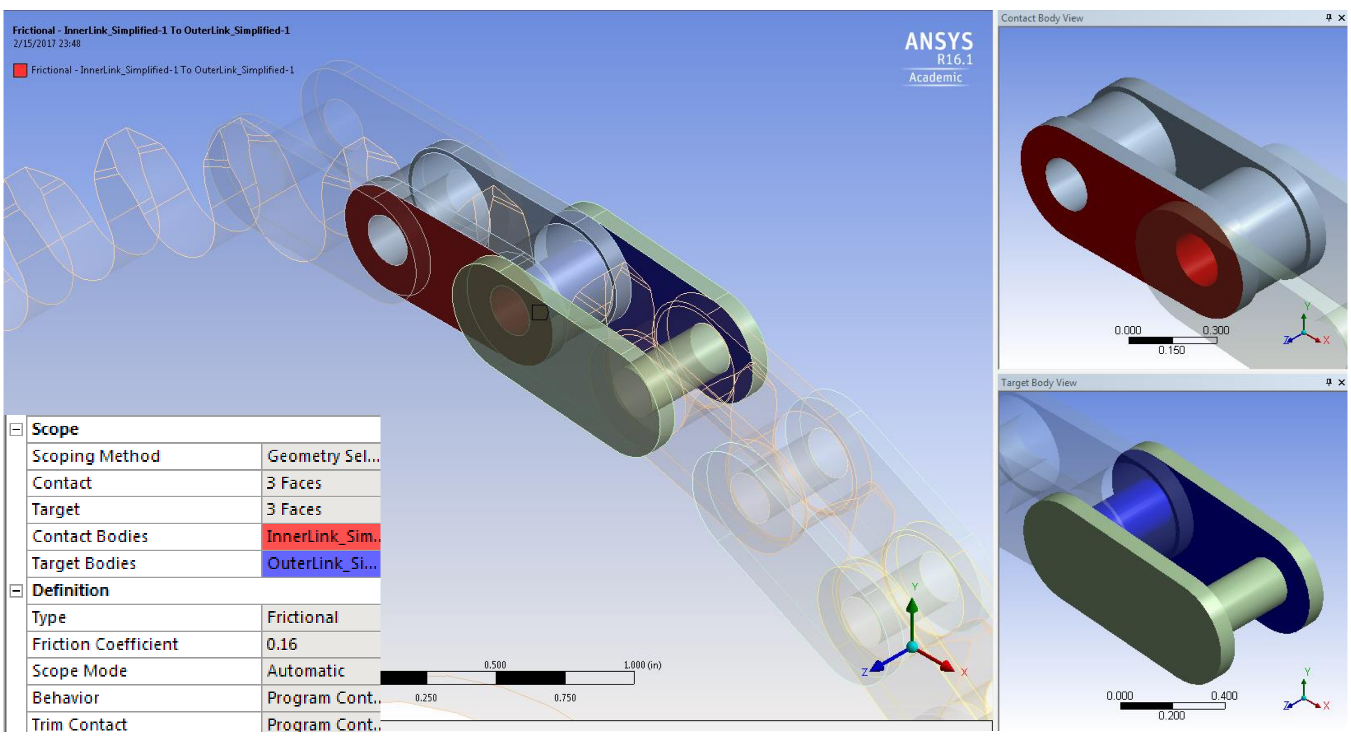


Figure 4.4: Contact between links. Left window: Overview. Top right: Cutaway view of contacts on inner link (in red). Bottom right: Cutaway view of contacts on outer link (in blue).

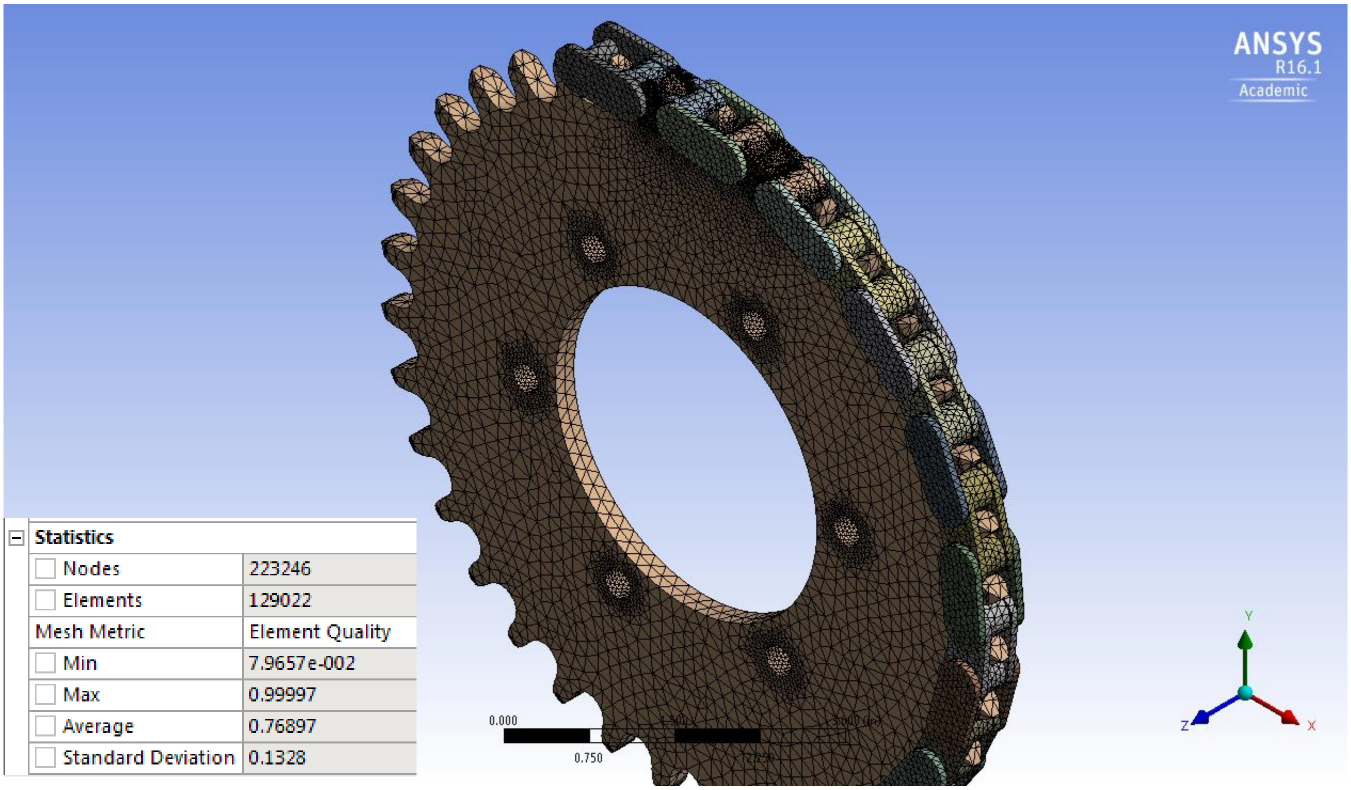


Figure 4.5: Mesh and quality statistics

Although never explicitly requested, the entire model uses triangular elements. The model's mesh is set by body sizing for the plate to 0.15 in as found to be a good sizing from previous validation models. The links' sizing was set to 0.08 in as found to be a good sizing from previous validation models. A refinement of level 2 is added around the bolt holes. After that a contact sizing is set for the first two inner links that contact the sprocket as shown in figure 4.9. A mesh convergence study was conducted as shown in Table 4.1. Even though mesh 7 is the finest, it showed significantly longer solving times (~10 minutes) while not offering much convergence. As such, mesh 6 was selected as the final mesh.

Mesh ID	Sprocket Body Sizing (in)	Chain Body Sizing (in)	First link Contact Sizing (in)	Second Link Contact Sizing (in)	Bolt hole refinement level	Max Sprocket Stress (psi)	Change from previous mesh	Max Stress at bolt hole (psi)	Change from previous mesh
1	0.15	0.08	0.08	0.08	1	23581		12689	
2	0.15	0.08	0.05	0.05	1	19632	-20.12%	12543	-1.16%
3	0.15	0.08	0.05	0.05	2	19656	0.12%	12483	-0.48%
4	0.15	0.08	0.04	0.04	2	36743	46.50%	12528	0.36%
5	0.15	0.08	0.03	0.03	2	28091	-30.80%	12568	0.31%
6	0.15	0.08	0.025	0.025	2	28796	2.45%	12577	0.07%
7	0.15	0.08	0.02	0.02	2	28532	-0.93%	12622	0.36%

Table 4.1: The mesh convergence study for the validation model. Sample readings can be found in figures 4.13 and 4.15. Final mesh: ID 6

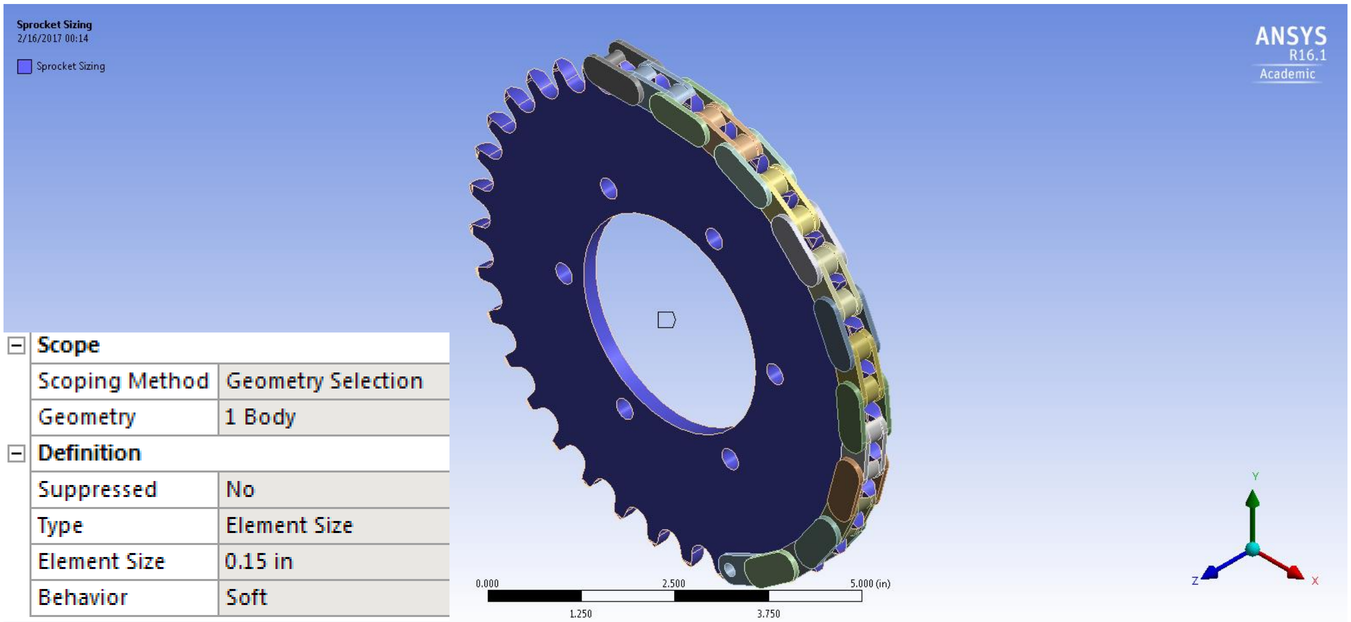


Figure 4.6: Body sizing for plate

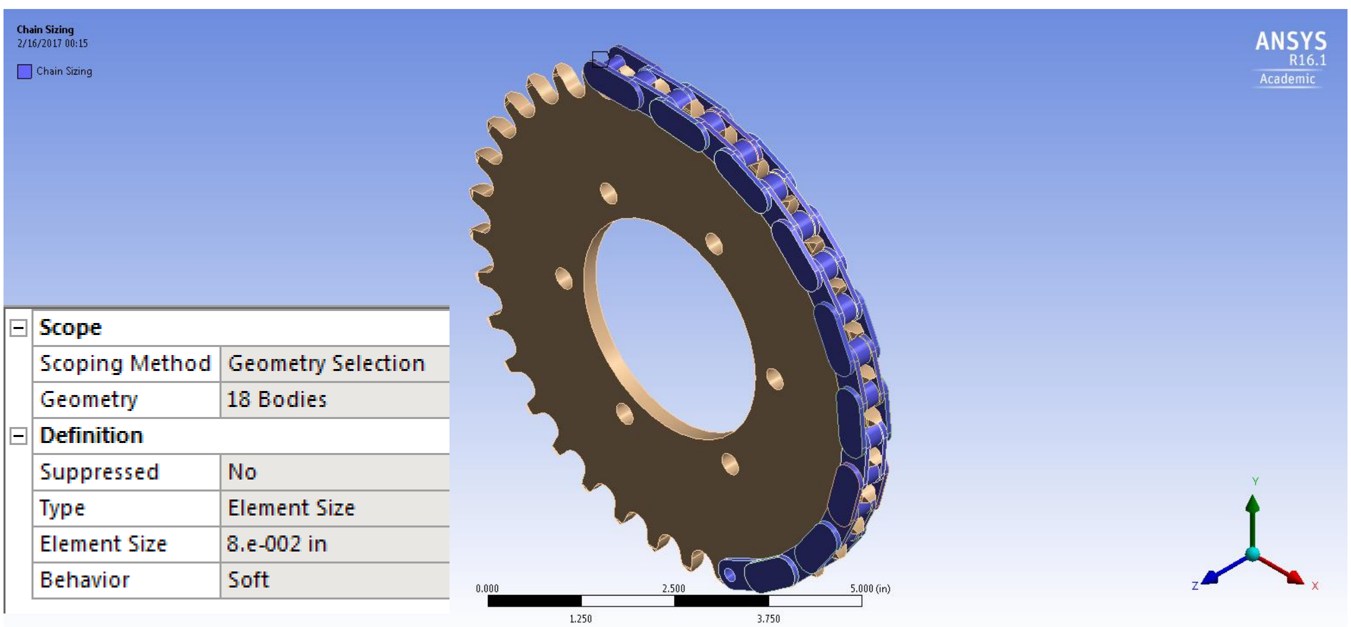


Figure 4.7: Body sizing for chain links

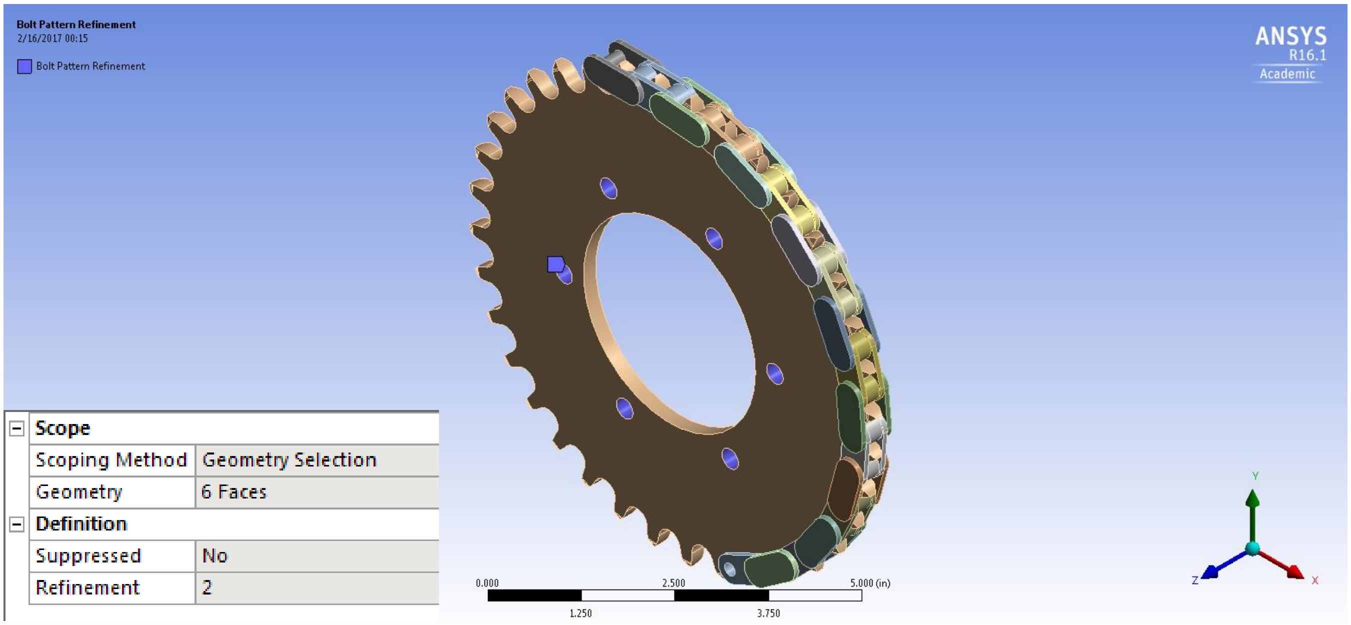


Figure 4.8: Bolt hole refinement

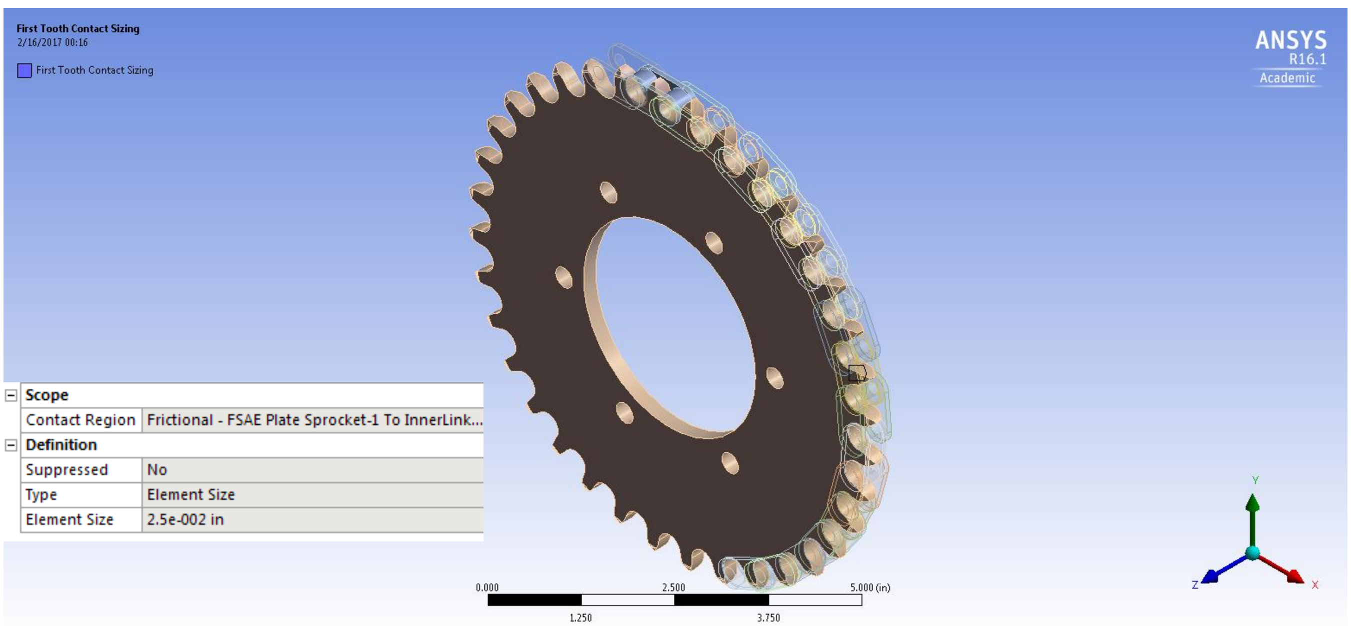


Figure 4.9: Contact sizing for first inner link. Same setup applied for second inner link as well.

A set of compression-only supports are applied to the bolt holes and a 1000 lbf force is applied to the pin of the rightmost link. This 1000 lbf force has been previously calculated by RoseGPE as a chain force induced by impending tire slip; this is worst case. Additionally, z-direction displacement constraints are added to the bolt holes to aid in convergence.

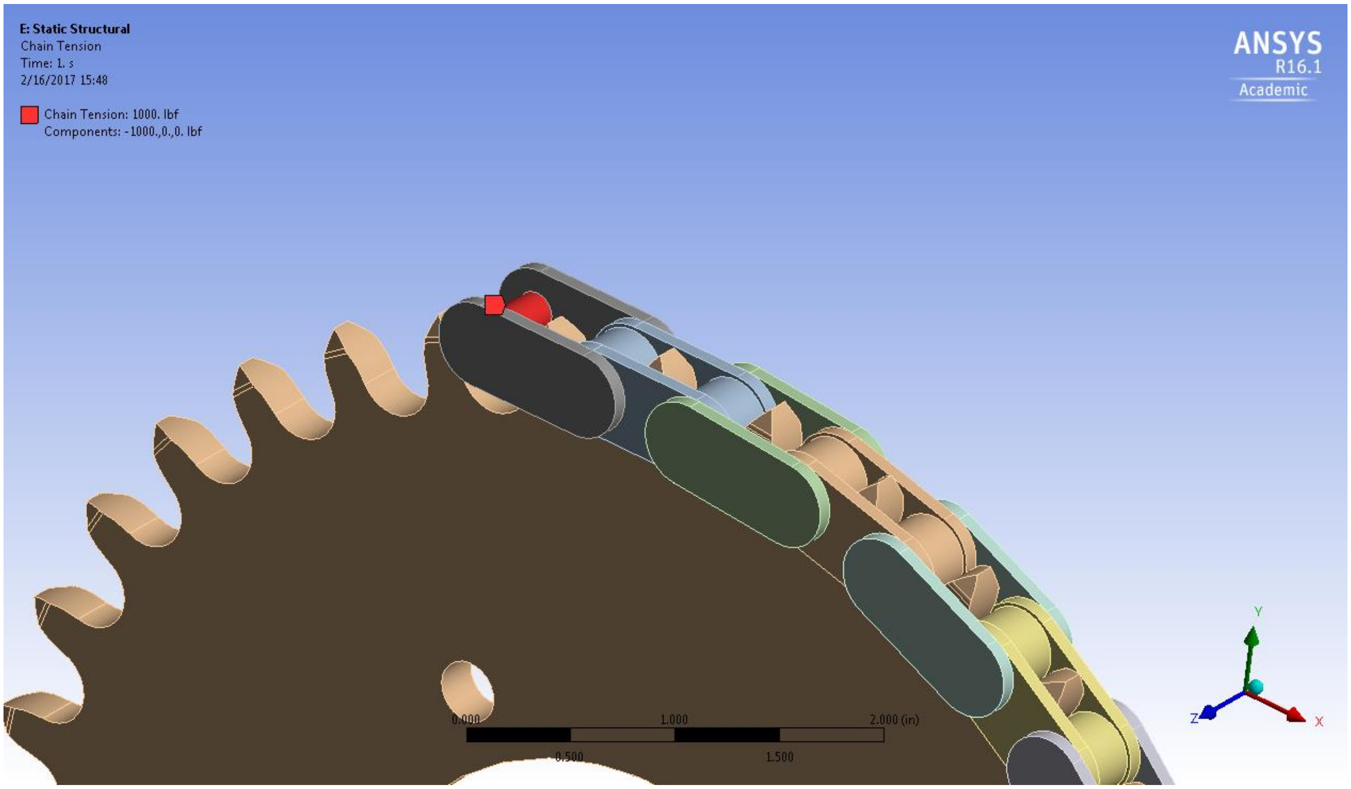


Figure 4.10: Tension force applied to outer link: 1000 lbf

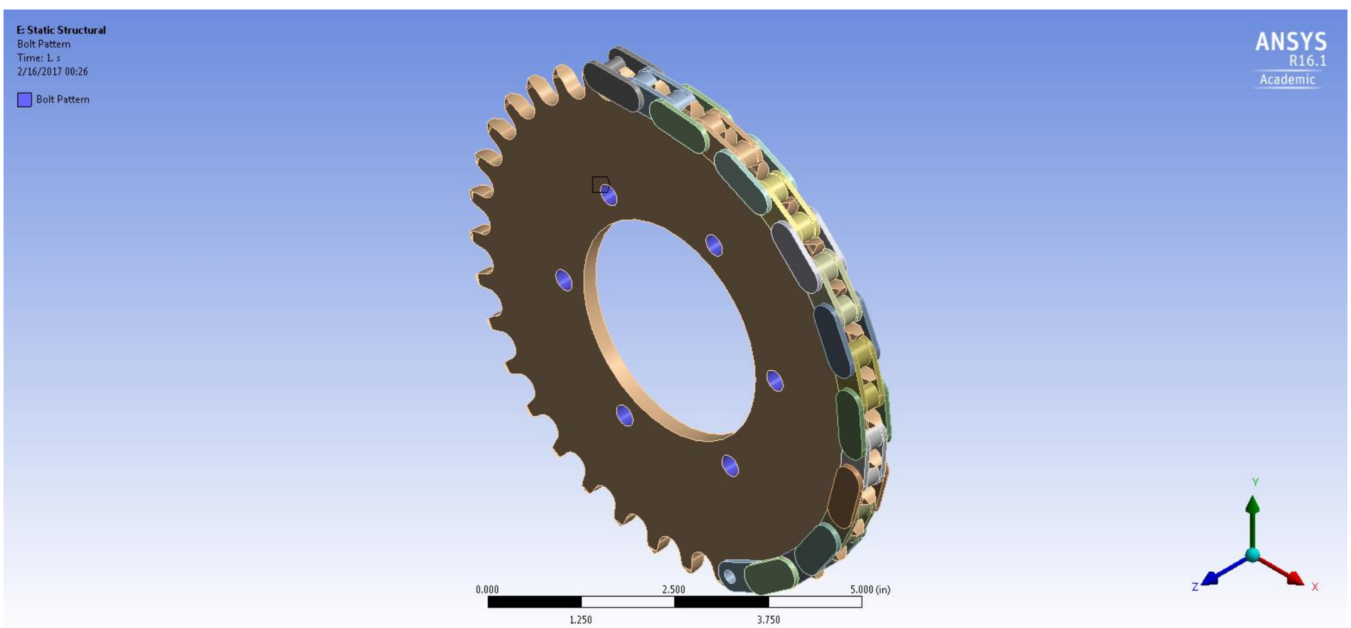


Figure 4.11: Compression-only supports for bolt holes of plate

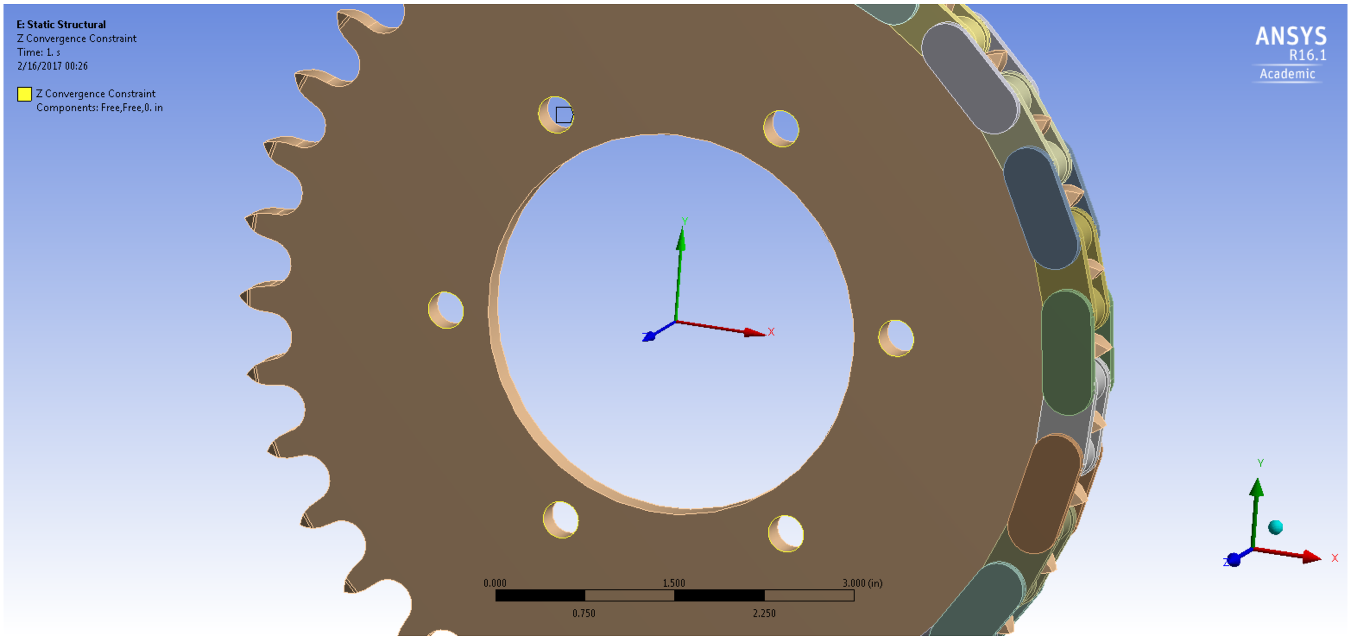


Figure 4.12: Z-direction displacement constraints to aid convergence

Von-Mises stress for the sprocket, as well as results scoped to the bolt holes are shown below. There appears to be a stress singularity, but this is somewhat expected, since this is where the link contacts the tooth of the sprocket, acting like a beam and producing more stress at this point. Plastic deformation may not actually be a concern in this application, since there are redundant teeth and more to take the load; wear-in is anticipated in sprockets. Still, preventing this point from yielding too much is important to part longevity and smooth operation.

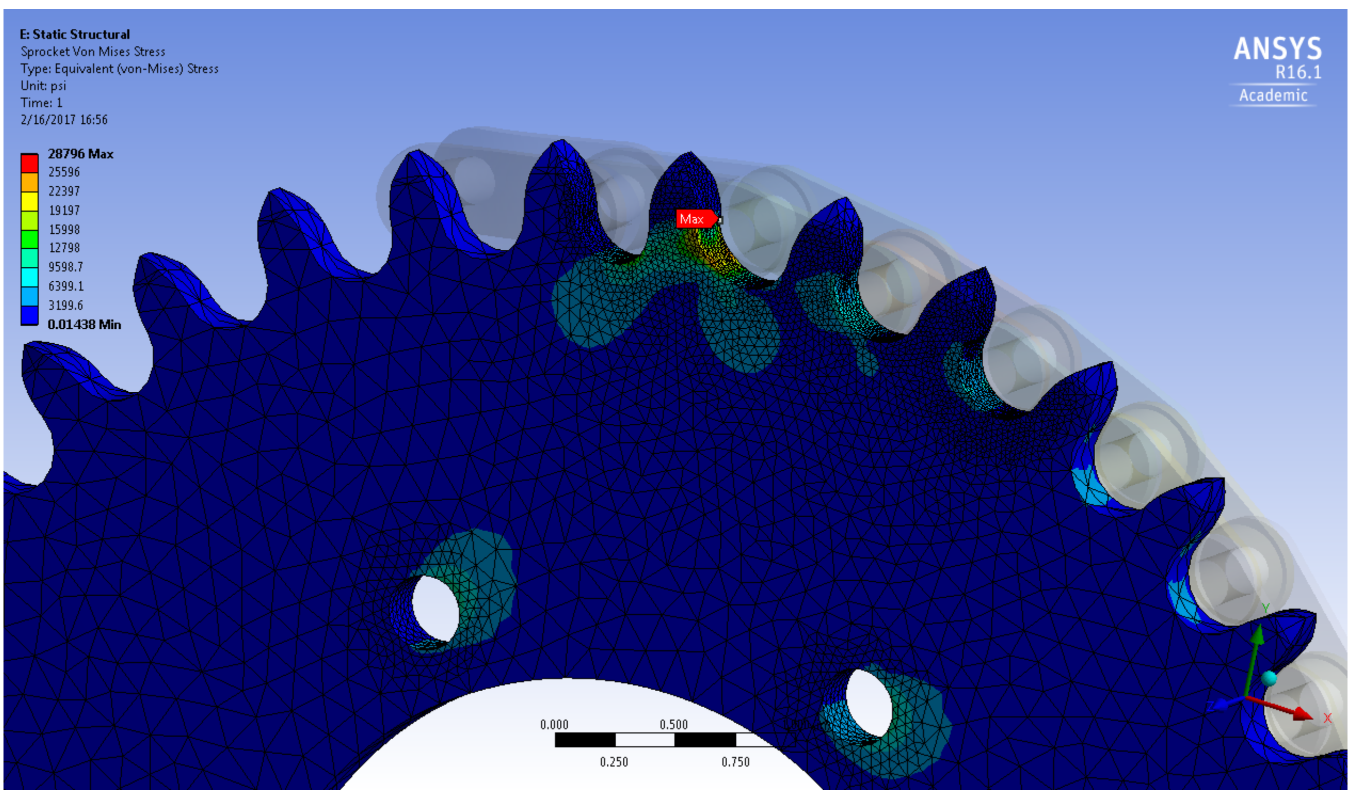


Figure 4.13: Von-Mises equivalent stress for entire part. Max: 28796 psi

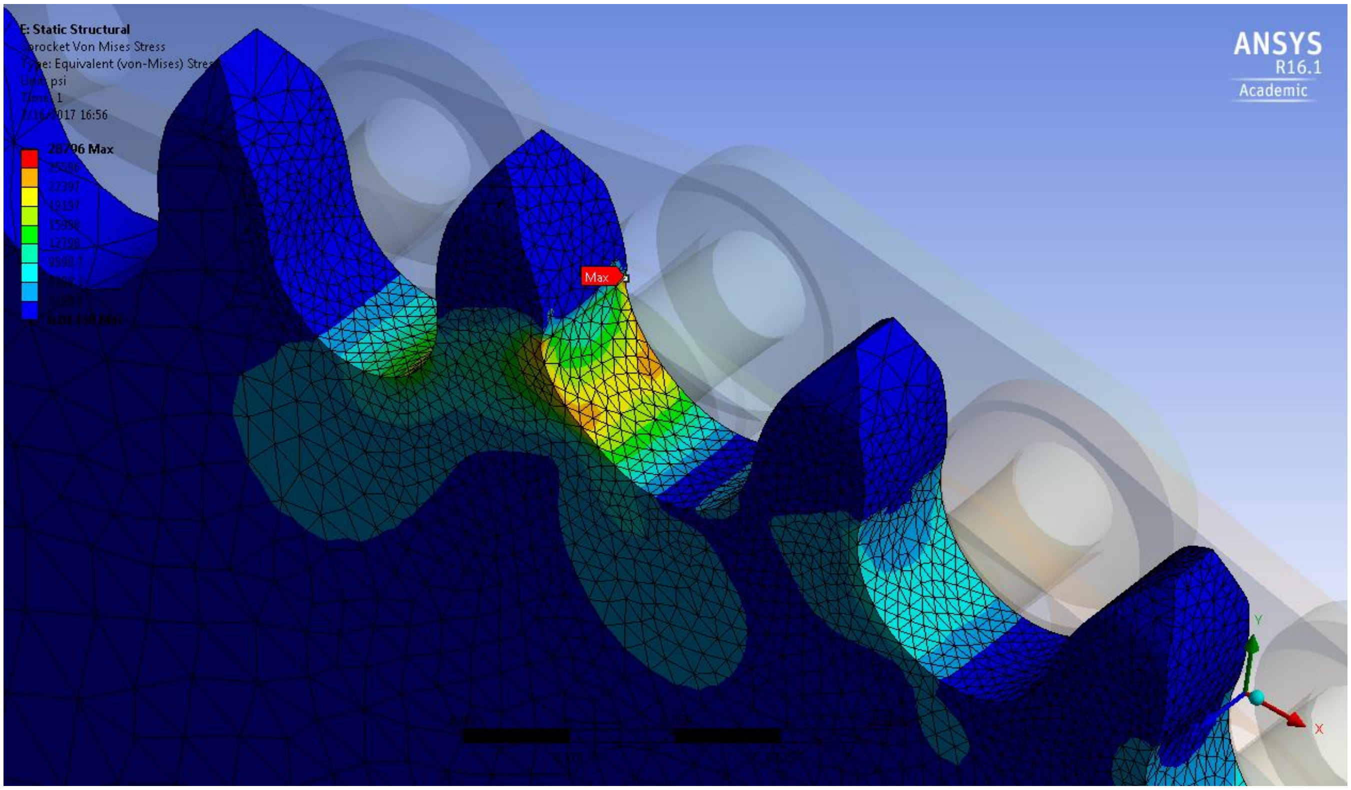


Figure 4.14: Von-Mises equivalent stress, zoomed in on first tooth. Max: 28796 psi

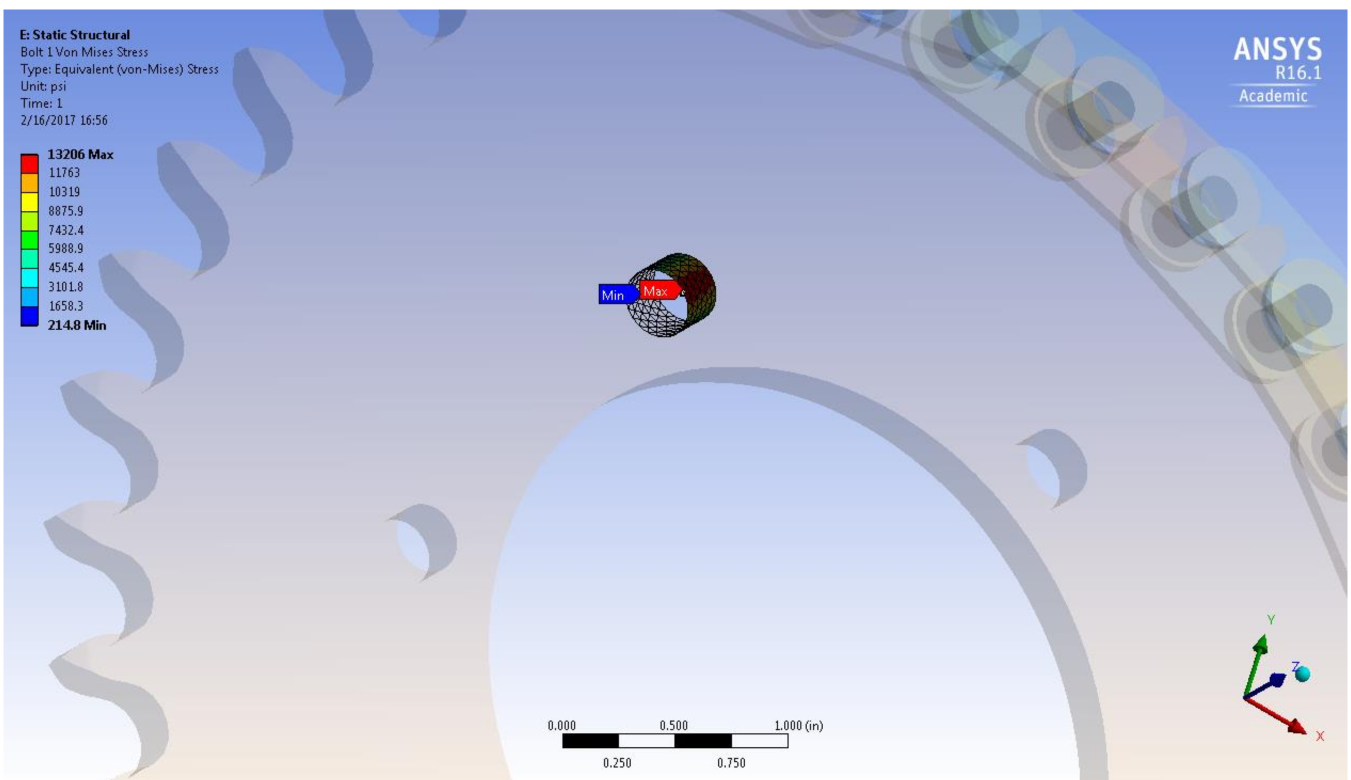


Figure 4.15: Von-Mises equivalent stress scoped to upper left bolt hole. Max: 13206 psi

Analysis Data Management	
Solver Files Directory	C:\Users\hughest1\EngineeringWorkspace\ME422\FSAE_Sprocket\FSAE_Sprocket_files\dp3\SYS-4\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	Bin

TABLE 15
Model (E4) > Analysis

Object Name	*Static Structural (E5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	71.6 °F
Generate Input Only	No

Figure 4.16: Solution confirmation and simulation authorship confirmation

5: Pocketing

With final mesh size and model setup determined, pockets can be added. The setup from section 4 has been duplicated, but this time, with a sprocket with holes in it for lightening purposes. The final assembly can be found at https://drive.google.com/file/d/0B_lA0xR4_viqRndSTXEwdTM4dE0/view?usp=sharing.

The goal of this section is to remove as much weight as reasonably possible (keeping with the general pocketing strategy demonstrated in figure 5.2) while a factor of safety of 1.5 against yield (meaning maximum stress must not exceed 44.6 ksi).

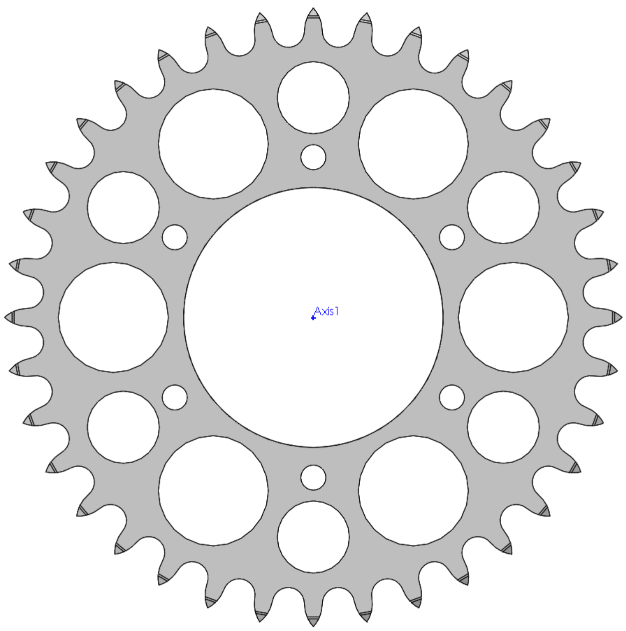


Figure 5.1: Example pocketed sprocket

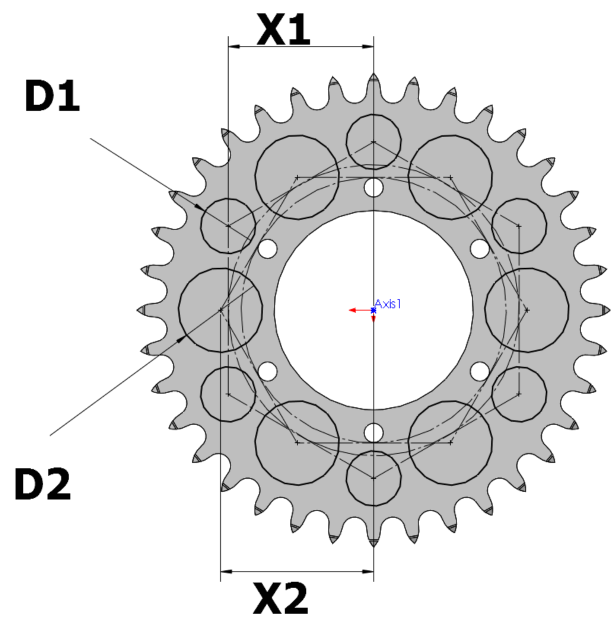


Figure 5.2: Pocketing geometry, and dimensions which will be modified.

This was accomplished by running successive iterations of the simulation and learning what material accomplished what. Most material really doesn't do much, but it serves to stiffen up and provide support to the sprocket teeth. This means that most of the material around the bolt circle can be removed. The final design reflects this, with the pocket removing material from around the bolt circle. This meant a large amount of material was removed; and a refinement was added around the pocketed area seeing most stress to study the stress here, and also to provide enough element thickness so that the thin sections act like proper beams rather than overly rigid.

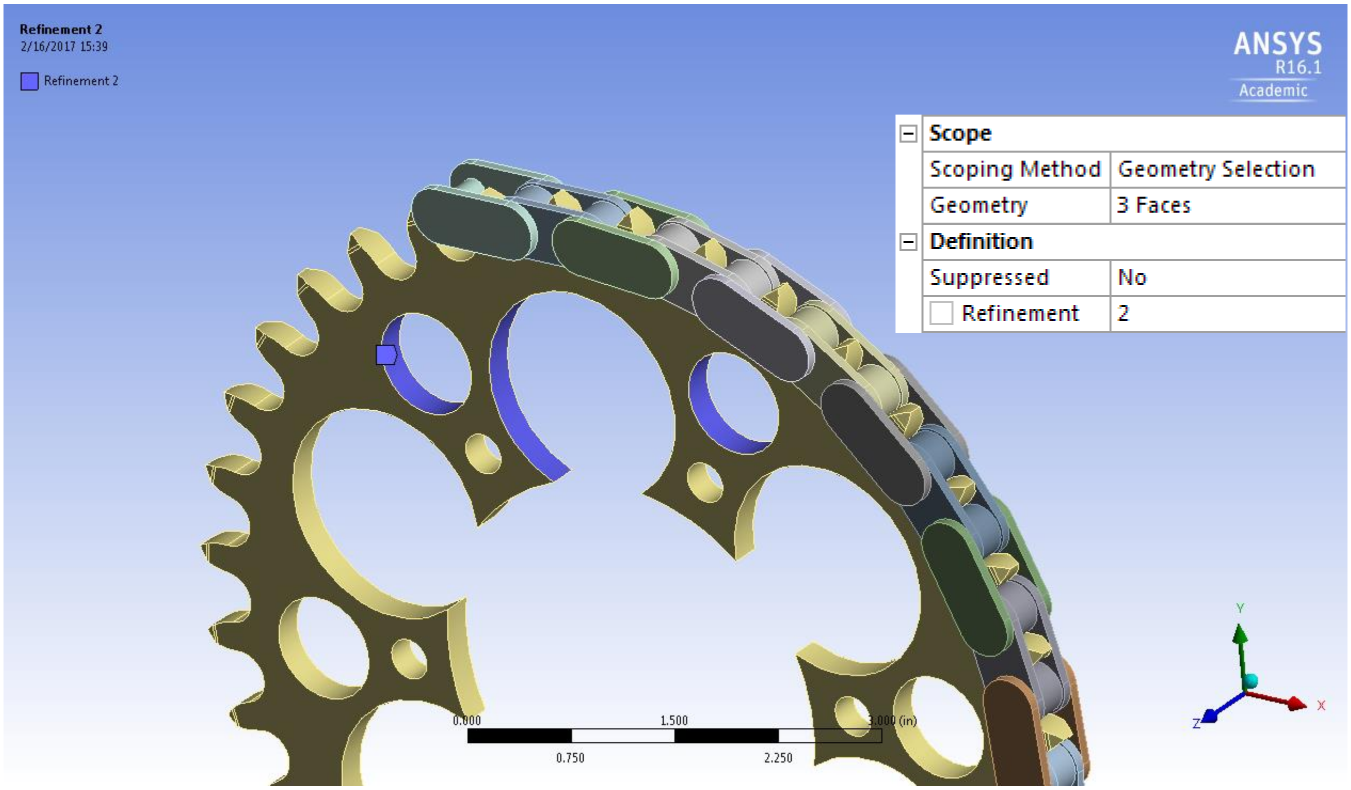


Figure 5.3: Refinement added to study stress around pockets

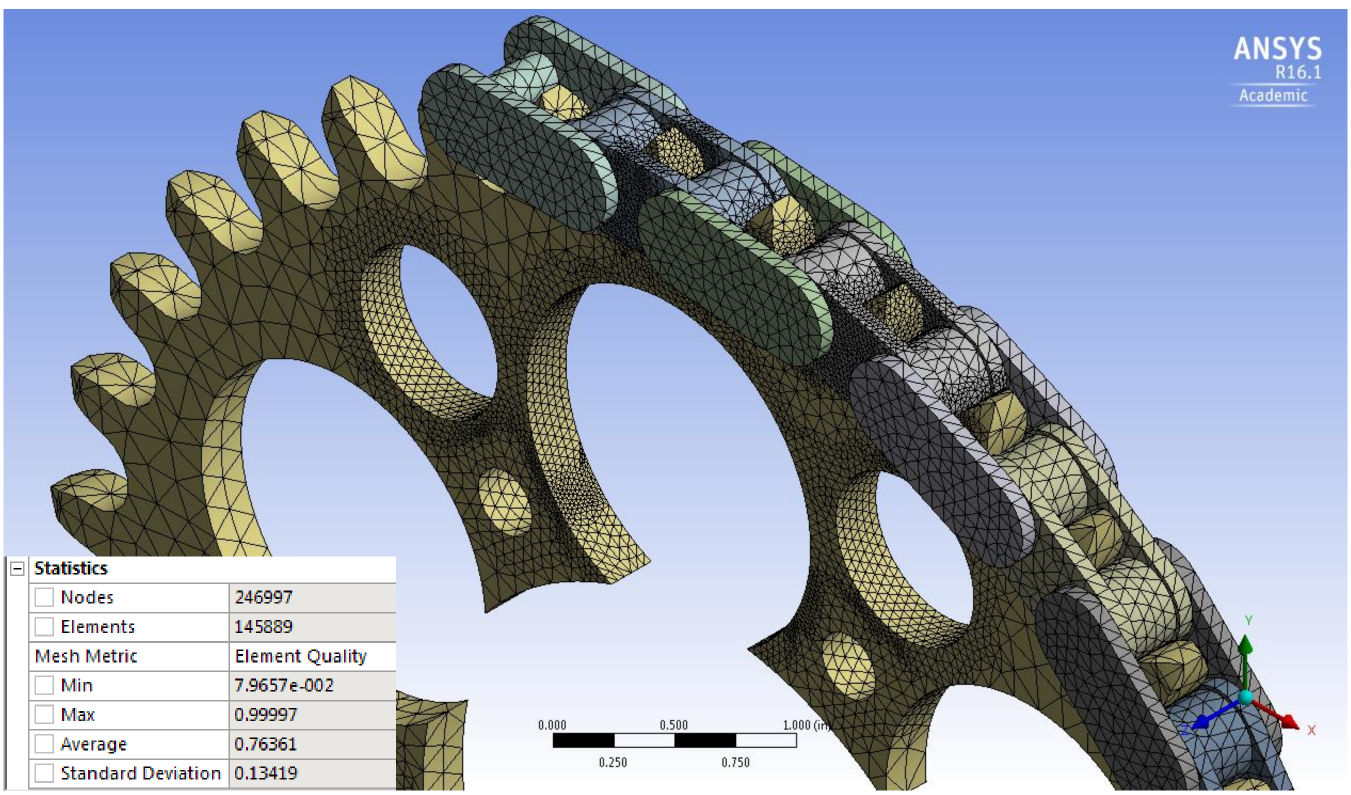


Figure 5.4: Final mesh used in analysis. Aside from the refinement shown in Figure 5.3, this is identical to the mesh used in Figure 4.

D1	X1	D2	X2	Volume	Density	Mass	Pct wt of original	Maximum stress in sprocket	FOS
UNPOCKETED				7.092	0.112	0.794	100%	28796	2.3267
0.750	2.300	1.250	2.400	4.789	0.112	0.536	68%	34676.000	1.9321
0.900	2.500	1.375	2.375	4.165	0.112	0.466	59%	37044.000	1.8086
0.800	2.400	1.650	2.300	3.558	0.112	0.398	50%	41499.000	1.6144

Table 5.1: Design study for stress in the sprocket. Dimensions specified in Figure 5.2. Sample reading of maximum stress can be found in Figure 5.5.

Interestingly, the maximum stress still occurs in the same location; around the tooth for the pocketed design. However, the magnitude of the stress has increased, with no changes made to the sprocket.

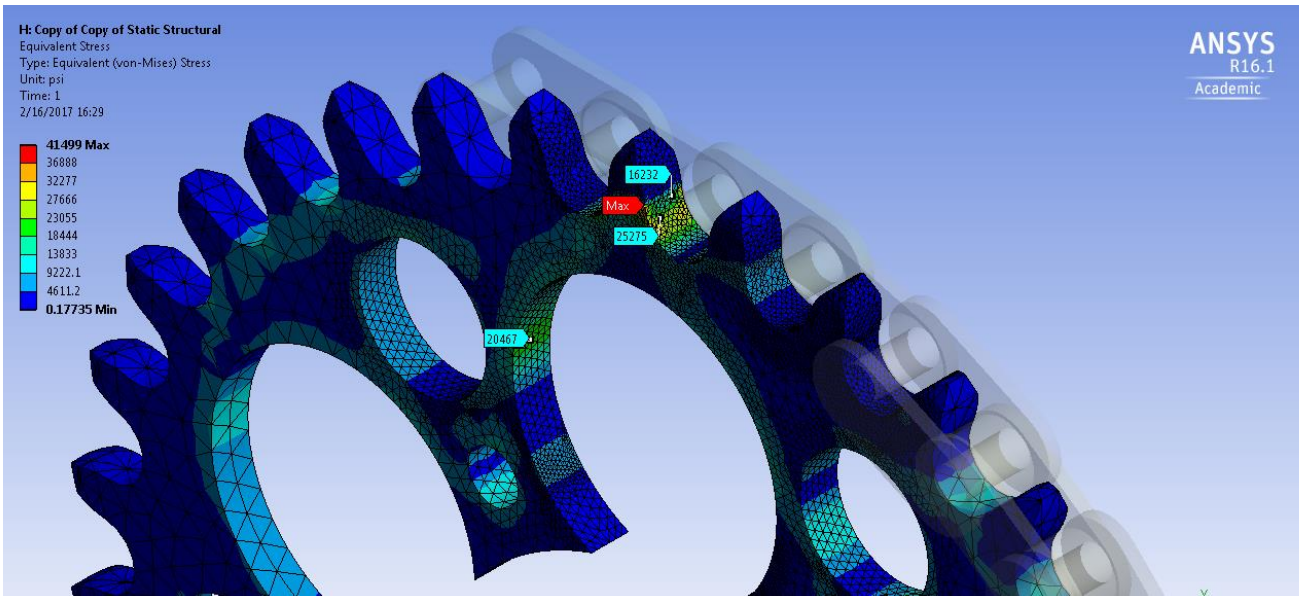


Figure 5.5: Overview of Von-Mises stress in part. Maximum stress: 41419 psi

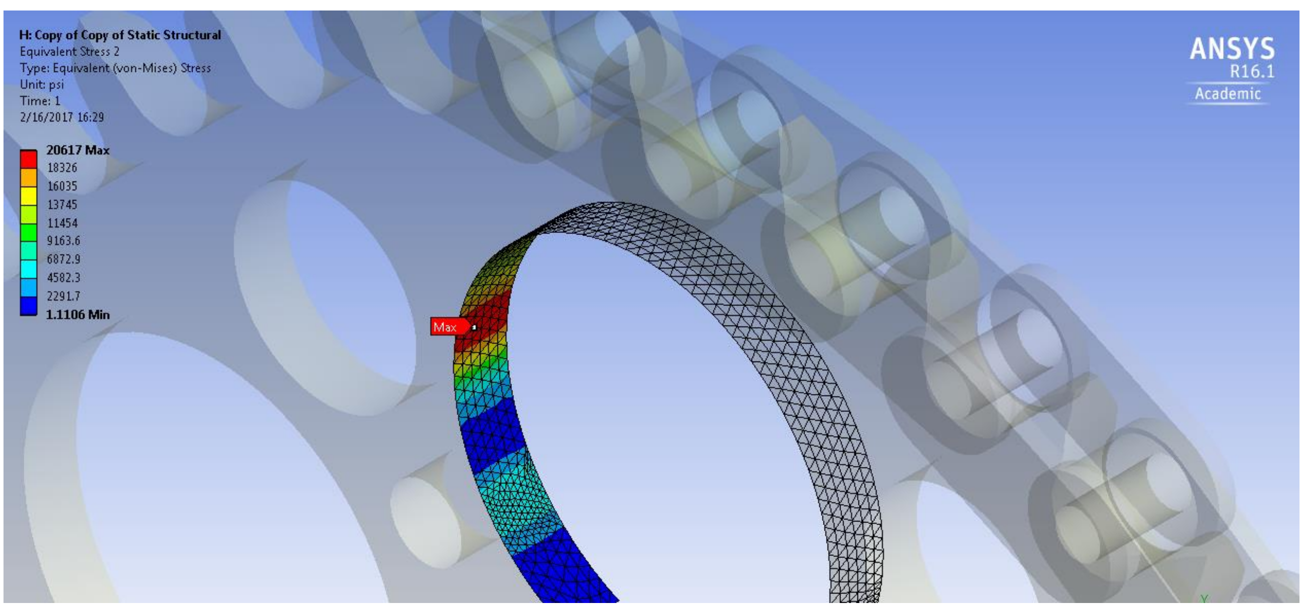


Figure 5.6: Von-Mises stress scoped to large pocket. Maximum stress: 20617 psi

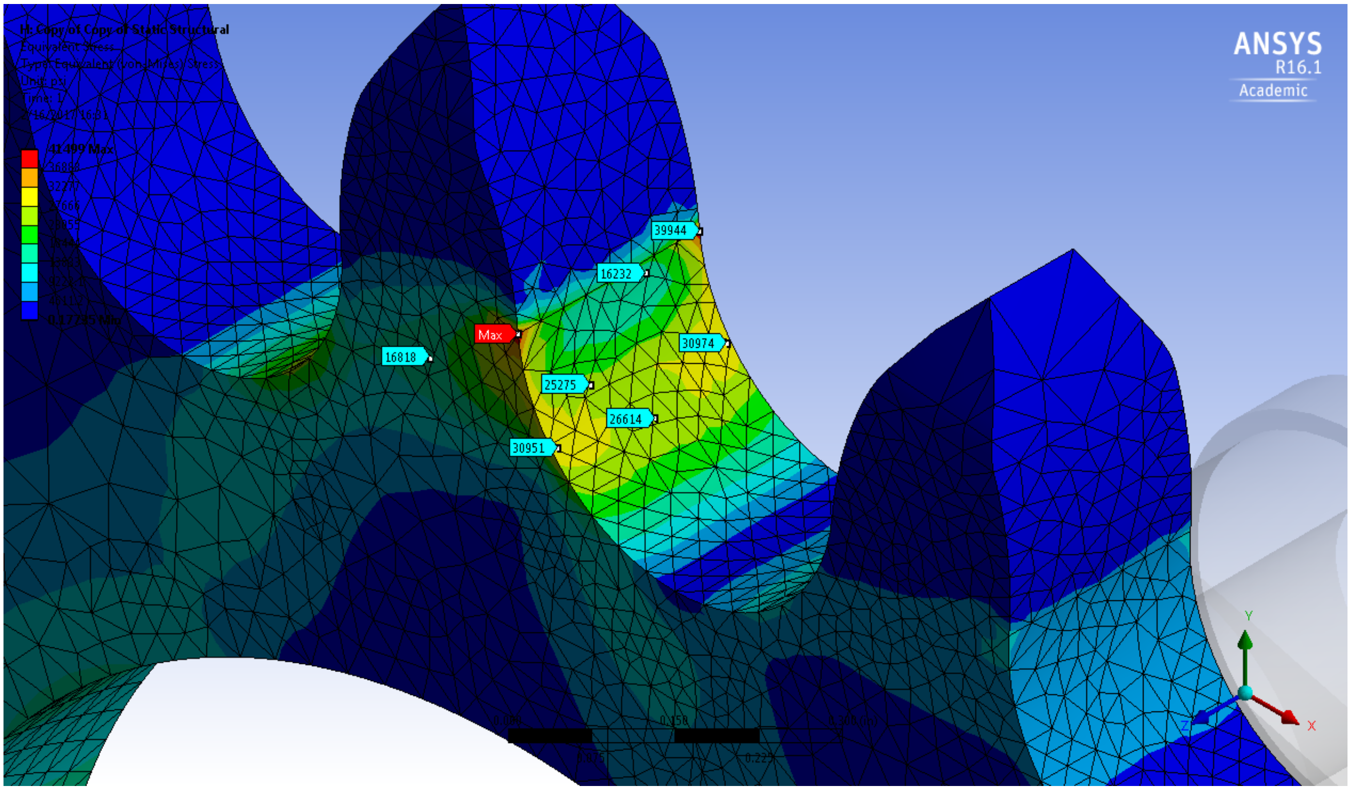


Figure 5.7: Von-Mises stress, zoomed in around first tooth. Maximum stress: 41419 psi

Analysis Data Management	
Solver Files Directory	C:\Users\hughest1\EngineeringWorkspace\ME422\FSAE_Sprocket\FSAE_Sprocket_files\dp5\SYS-4\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	Bin

TABLE 13
Model (H4) > Analysis

Object Name	*Static Structural (H5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	71.6 °F
Generate Input Only	No

Figure 5.7: Solution authorship verification

6: Conclusion

By using this analysis, mass of the sprocket was reduced by 50%, or 3/8ths of a pound, while still keeping the part with a factor of safety against yield over 1.5 by a Von-Mises stress criterion, as seen in Table 5.1.

It is imperative to note this is for one angle that chain is pulled at. If the sprocket was oriented at a different angle, different stress would be induced. As such, a study should be done to determine at what orientation maximum stress occurs at.

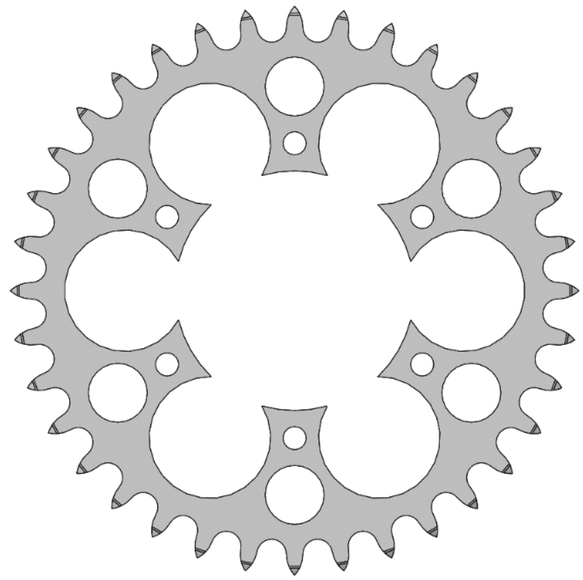


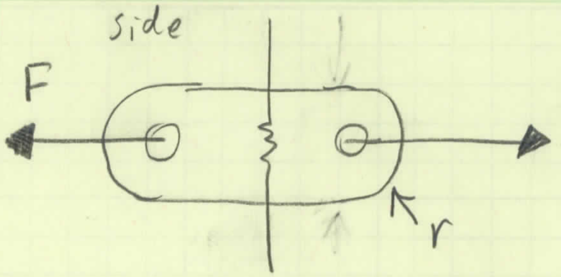
Figure 6.1: Final sprocket, pocketed

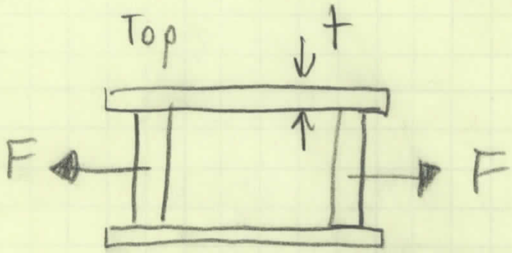
7: Lessons Learned

This project proved to be a valuable learning experience in both practice and fundamentals. Frictional contacts can tend to be messy, and when I had set up a simulation with low frictional values, mesh dependence was rampant. I also experienced the convergence issues nonlinear contacts such as compression-only supports present.

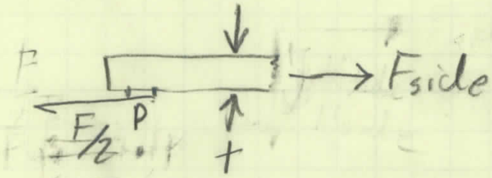
This model was very large, and I learned how to use parameters in ANSYS to run convergence studies. These would normally take hours being babysat, but with parameters employed, a table of design points can be set up, and then queued to solve, say overnight while sleeping with no need to be interfered with. This, however, showed the importance of correct model setup, as I once wasted an entire night doing such a convergence study with incorrect frictional parameters.

Something I should have done, as I went forward, would be to examine the effects of removing chain on convergence, and also defeaturing the sprocket to reduce computational effort in both meshing and solving. On a larger problem, and more thorough study, this would be non-optional.





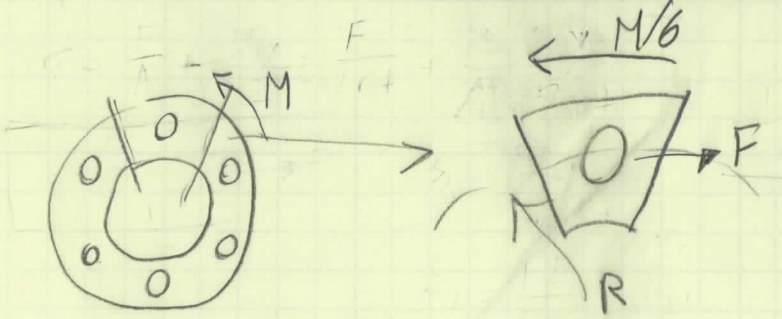
Cut away:



$$\Sigma F = F_{side} - \frac{F}{2} = 0 \Rightarrow F_{side} = \frac{F}{2}$$

$$\sigma_{Avg} = \frac{F_{side}}{A}$$

$$A = 2rt$$



$$\Sigma M_{center} = 0$$

$$\frac{M}{6} - FR = 0$$

$$F = \frac{M}{6R}$$

Validation of stress inside link

```
> restart
> t:=0.075; r:=0.22; F:=1000;
```

$$t := 0.07500$$

$$r := 0.22000$$

$$F := 1000 \tag{1.1}$$

```
> y:=pos
```

$$y := pos \tag{1.2}$$

```
> A_link:=2*r*t
```

$$A\_link := 0.03300 \tag{1.3}$$

```
> I_link:=2*r*t^3/12
```

$$I\_link := 0.00002 \tag{1.4}$$

```
> Fside_link:=F/2
```

$$Fside\_link := 500 \tag{1.5}$$

```
> sigma:=Fside_link/A_link
```

$$\sigma := 15151.51515 \tag{1.6}$$

```
> sigma_FEA:=(13344+16612)/2
```

$$sigma\_FEA := 14978 \tag{1.7}$$

```
> sigma/sigma_FEA
```

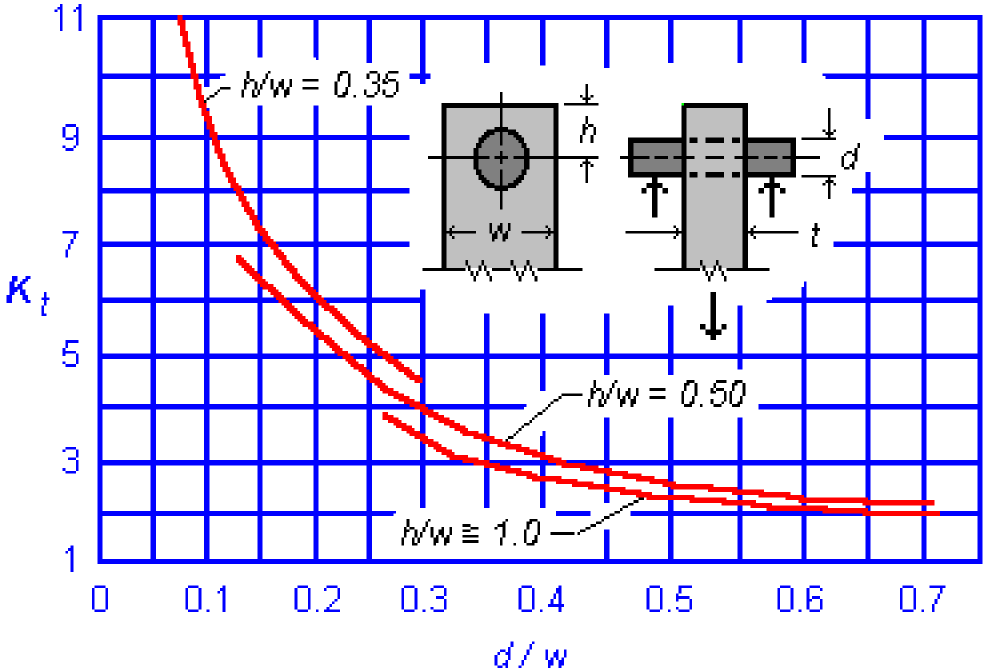
$$1.01158 \tag{1.8}$$

Validation of stress produced on pin hole

```
> restart
```

Chart via: http://analysischamp.com/StressBook/TABA2512.htm

>



> F:=1000

$F := 1000$ **(2.1)**

> w:=0.75

$w := 0.75000$ **(2.2)**

> h:=w/2

$h := 0.37500$ **(2.3)**

> d:=0.4

$d := 0.40000$ **(2.4)**

> t:=0.25

$t := 0.25000$ **(2.5)**

> h/w

0.50000 **(2.6)**

> d/w

0.53333 **(2.7)**

From chart

> Kt:=2.6

$Kt := 2.60000$ **(2.8)**

> A:=(w-d)*t

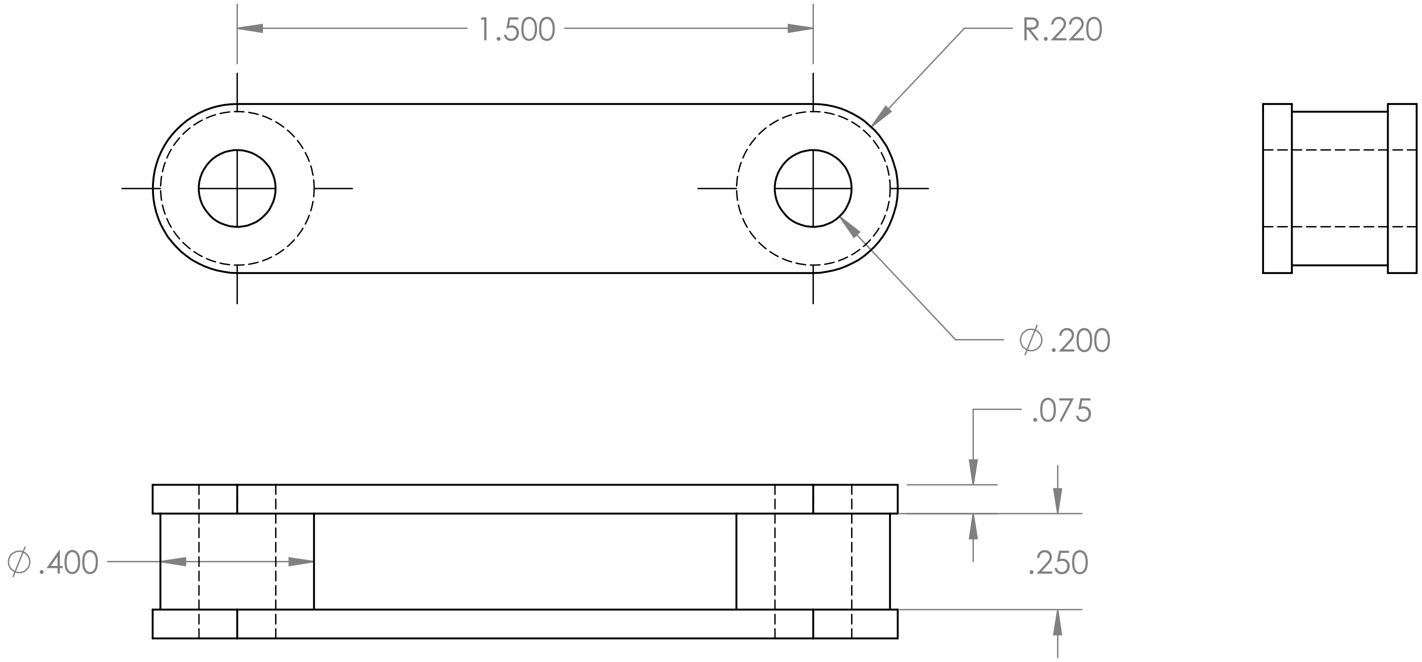
$A := 0.08750$ **(2.9)**

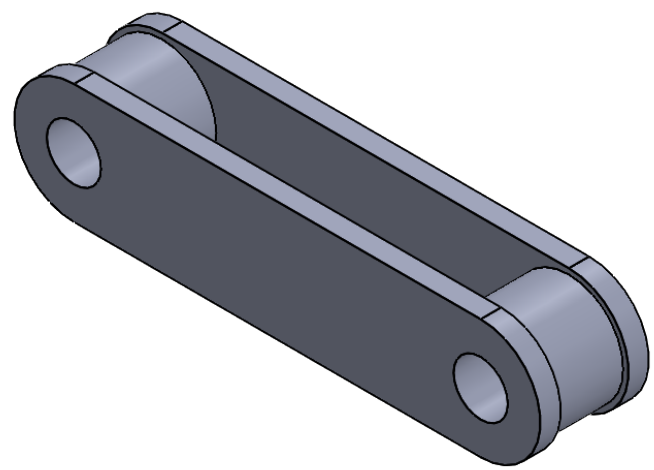
> sigma0:=F/A

$\sigma0 := 11428.57143$ **(2.10)**

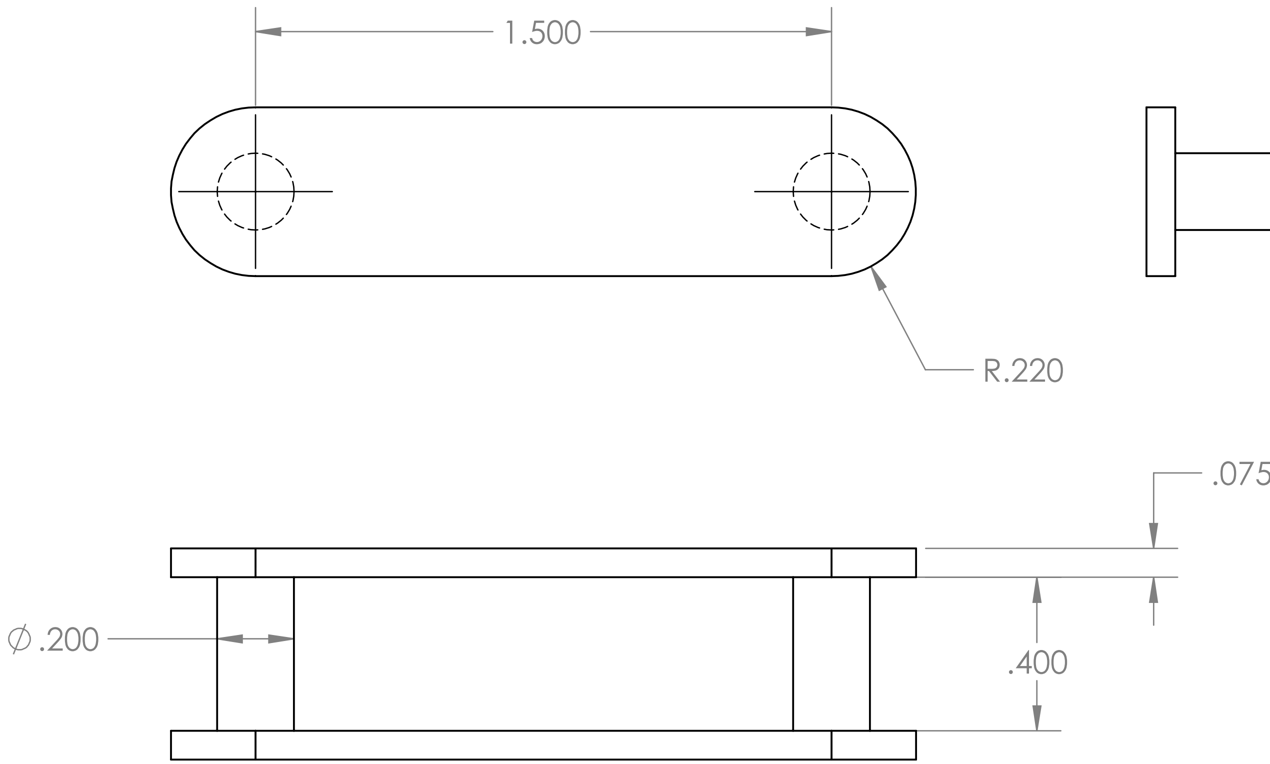
> sigma:=sigma0*Kt

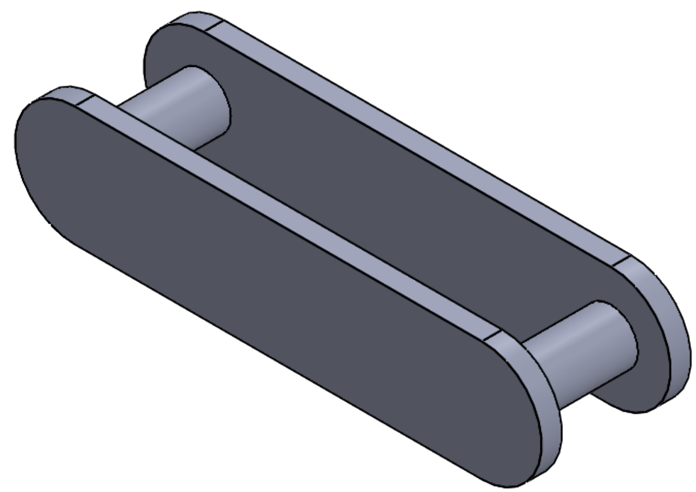
$$\sigma := 29714.28572 \tag{2.11}$$





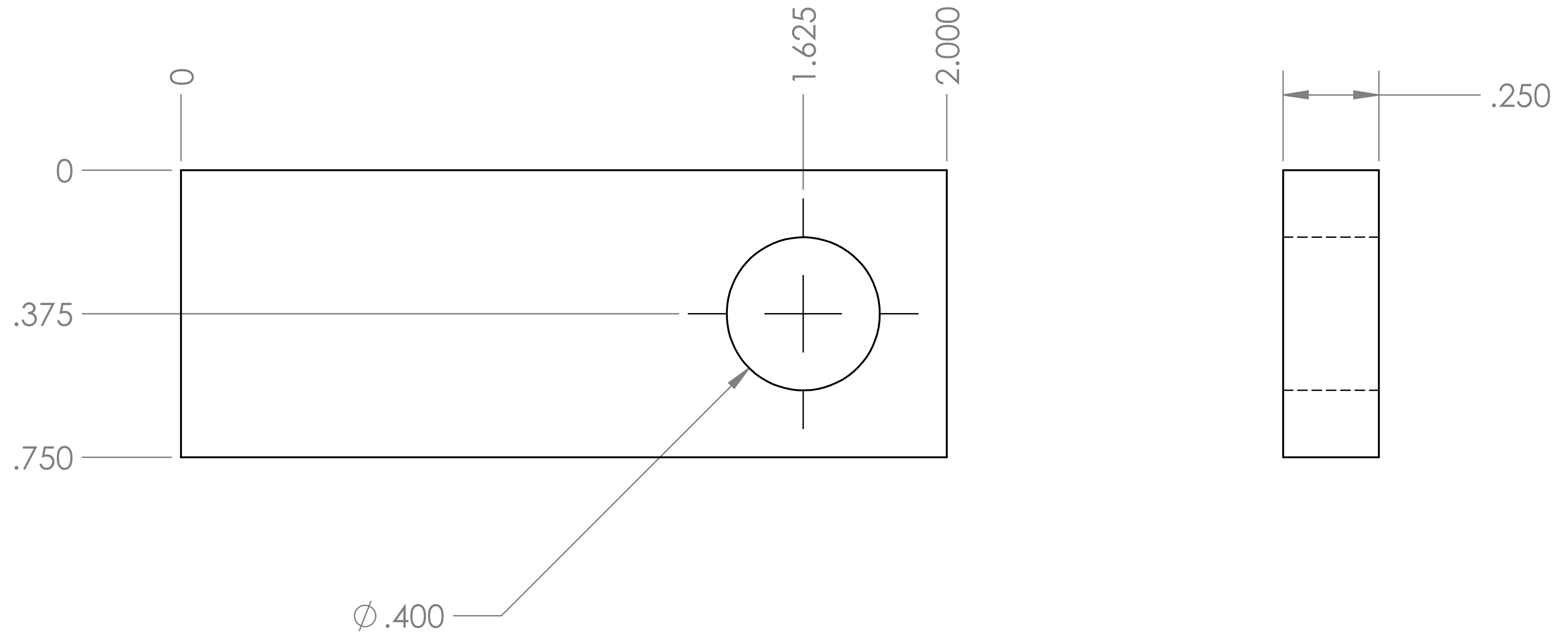
Dimensions are inches and degrees	InnerLink_Simplified_Lengthened	
Default Tolerances	Material:	
.X ±0.1 .XX ±0.01 .XXX ±0.002 Angles ±2	Drawn by: hughest1	
	Scale: 2:1	Revision:

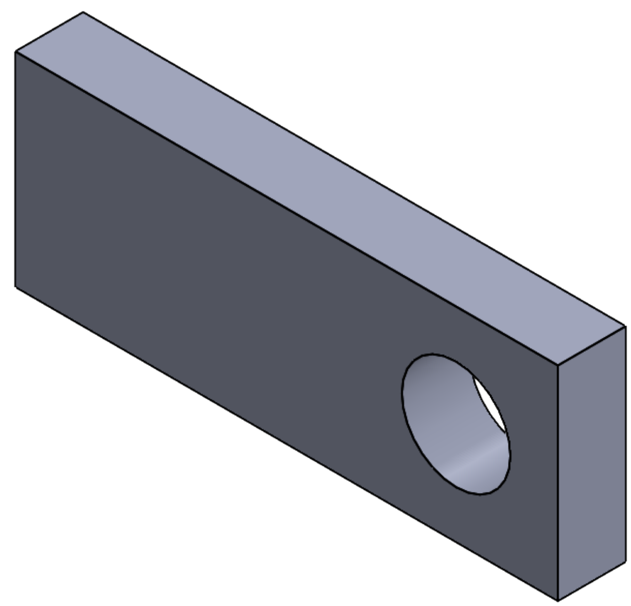




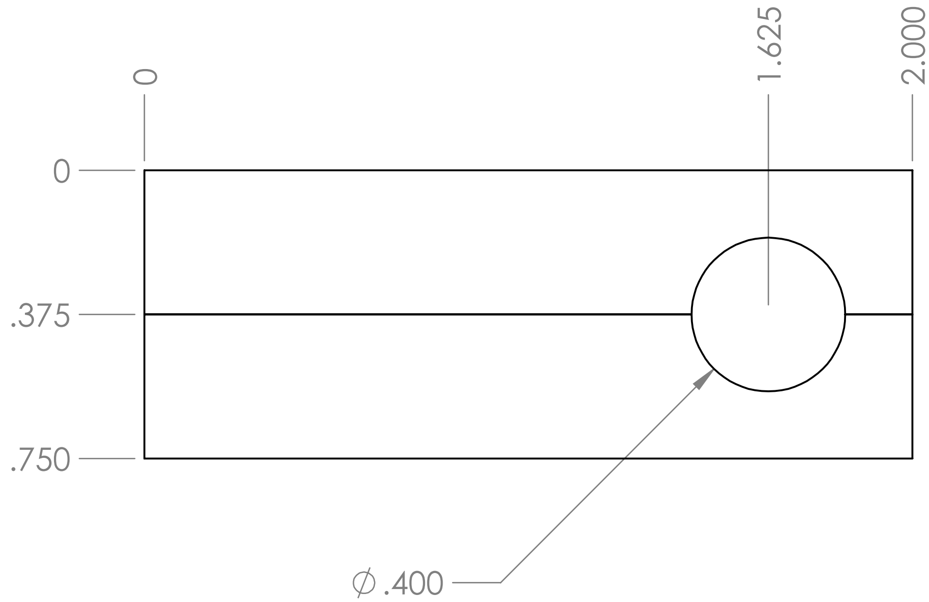
Dimensions are inches and degrees	OuterLink_Simplified_Lengthened	
Default Tolerances	Material:	
.X ±0.1		
.XX ±0.01	Drawn by: hughest1	
.XXX ±0.002		
Angles ±2	Scale: 2:1	Revision:

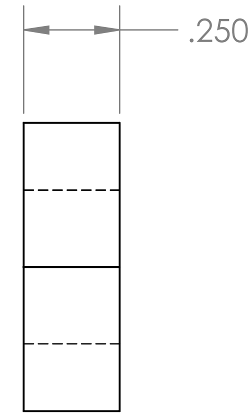
This space left for that "ACADEMIC USE ONLY" watermark

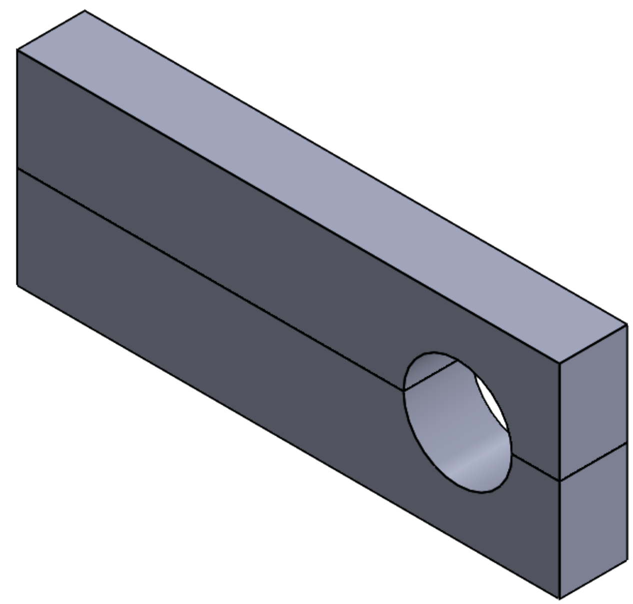




Dimensions are inches and degrees	Plate	
Default Tolerances	Material:	
.X ±0.1	Drawn by: hughest1	
.XX ±0.01		
.XXX ±0.002	Scale: 2:1	Revision:
Angles ±2		







Dimensions are inches and degrees	Plate2	
Default Tolerances	Material:	
.X ±0.1	Drawn by: hughest1	
.XX ±0.01 .XXX ±0.002 Angles ±2	Scale: 2:1	Revision: