Final Project
March 21, 2018 (Wed)

ME I6500
Prof. Kenneth Gollins

Wall Shelf

Dong, Yuhao

Gao, Chen

Huang, Mian

Jiang, Zijian

TABLE OF CONTENT

Introduction

According to the U.S. Consumer Product Safety Commission, there was one accident report for every two weeks between 2010-2017 about failing furniture causing kids under 7 years old injuries and even death (Knowles and Pistone, 1). These accidents included furniture tipping over, wall slipping, and part fracturing. Some accidents could be prevented by parents instructing children not to climb over the furniture, but some accidents could unexpectedly happen due to mistakenly use of the product due to the information deficiency. The failing of wall mounted appliances due to over loading was a typical example. Thus, deficiency of using information could be very detrimental.

Different kinds of loading situations could cause different stress situations on the product. Thus, engineers and manufacturers should always be responsible for providing enough information of product usage. However, when we searched online, we noticed that most of the sellers tend to provide the value of the maximum load only. As engineering students, we noticed that this was not enough because the stress situations might have big differences under different loading situations even though the applied load values were the same. Therefore, for this project, we picked a wall mounted book shelf, which had the maximum load value provided from Amazon, and tried to investigate if different situation of maximum loading would cause the product failure. This failure referred to situations that the frame deformed and caused objects to fall. Moreover, through analyzing the stress results in those three situations, we wanted to prove that big differences in different stress situations could be caused by different loading situations although the applied value was the same; we further suggested that detailed information of caution of usage should be provided by the design engineers. Also, we learned a lot from the reflective project reading assignments, so we want to taste about the real-world engineering.

Development

	
Figure 1 - Book shelves	Figure 2 - Components

The bookshelf, shown in the Figure 1, we picked was “Roo Lee Upgraded 2-Tier Floating Wall Mount Shelves”. As showed in the Figure 2**,** the book shelf was constructed by two wood plates and two metal frames. Each frame was mounted on the wall by using 2 wall plugs and screws. Each wood plate was mounted on the shelves by 4 small screws. The product description showed that the maximum load the bookshelf could take was 45 lbs. However, there was no information indicates how the shell could sustain the maximum load, and there were many situations (conditions) consumers could use the shelf. Did the load had to be applied relatively uniform on one level of the shelf as the Figure 1 showed? Was the maximum load able to be applied on only one level safely? Or the load could be concentrated in the middle of the plate? What about the load concentrated on one end of the plate? What were the differences in stress situation between these load situations? These were the questions we wanted to find out. The project would be focused on the metal frame part to see if the metal frame would fail under different loading conditions of seller-indicated maximum load. As the structure was welded and not simple as one-whole-beam, frame width was thin, and the load was applied on the wood plates and then transferred to the metal frame, we couldn't find stress analytical equation from previous classes, and we had to use Finite Element Method in SolidWorks Simulation. The following sections would discuss about the solid modeling process, boundary condition considerations, loading situations explanation, meshing technique, improvement design, and data analysis.

Modeling

For the Solid modeling process, our primary goal was to make sure the dimensions of our SolidWorks visual model were close to the dimensions of physical model, so the simulation could produce an accurate result. In order to achieve this goal, we performed our measurements several times to minimize the human error. We were also using precise tools such as digital caliper to measure our physical model, and these tools could further help us to improve the accuracy of the SolidWorks model we created.

Our book shelf contained one pair of metal frames and one pair of wood boards. Since two items in each pair had the same sizes, we could only measure one item per pair. First, we used roll ruler to measure the long sides such as the length and the width. Next, we used digital caliper to measure the short sides such as the thickness. In fact, both metal frame and wooden board were designed to had uniform dimension. Thus, we concluded the items had uniform dimensions in length, width and thickness.

The measurement results were shown in the following table:

	Manual Measurement
Shelf Dimensions	142mm L
	396mm H
Wood Dimensions	600mm L
	146mm W
	15mm T

Table 1 - measurement results

Once we collected all the measurement, we started to model the following two parts.

Part 1: The metal frame

First, we created the side projection of the metal frame based on our measurement. We noticed that the slanted beam contains most of the geometric characteristics of the frame and would significantly influence the stress analysis. Therefore, to make sure the measurements were correct enough to yield accurate slanted angel, we took a picture of the side view and imported it into SolidWorks for matching. Next, we extruded the body part of metal frame. Then, we extruded the support bracing of metal frame. For the holes, we drew two circles on the upper level of the shelf then used extrude cut to create holes on both level of the shelf. Finally, we drew two ellipses on the back of the shelf and created two ellipses holes.

Part 2: The wood plate

Our wood plate was just a simple rectangular plate. Thus, we first drew a 600mm x 146mm rectangle. Then, we simply extruded by 15mm.

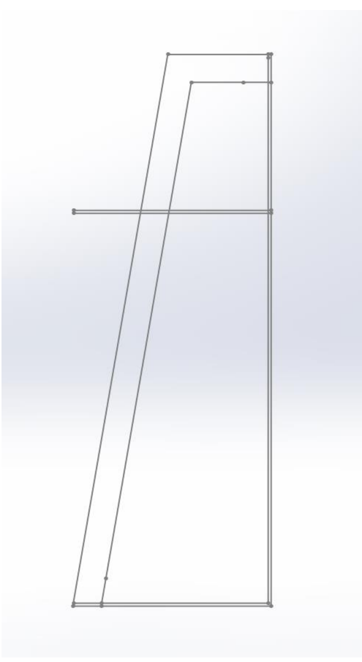
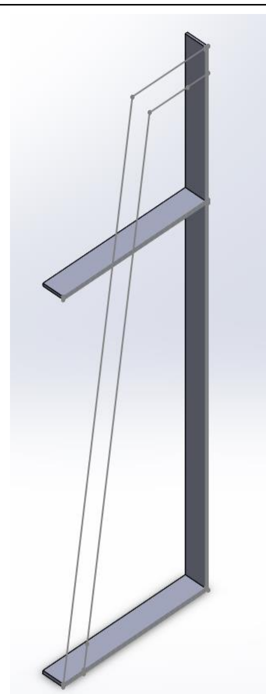
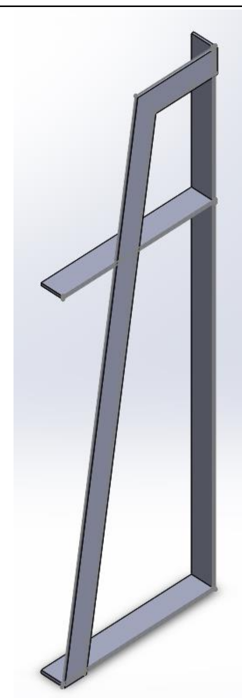
		

Figure 3 - sketch of the metal part of the bookshelf	*Figure 4 - extrusion of the body part of metal frame*	*Figure 5 - extrusion of support bracing*

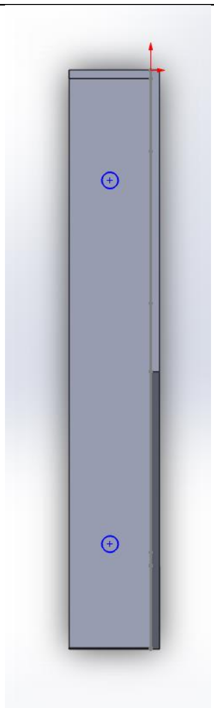
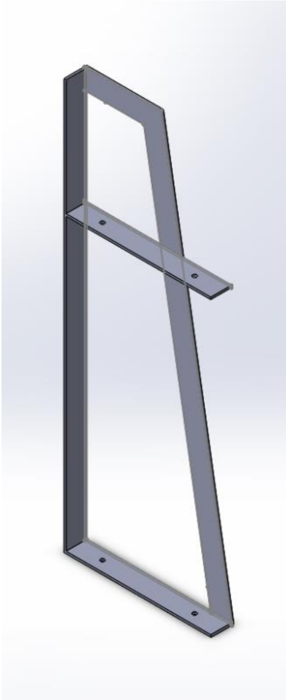
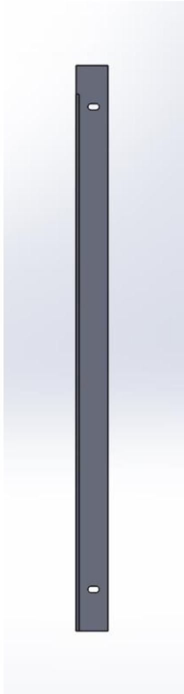
		

Figure 6 - two holes	*Figure 7 - used extrude cut to create holes*	*Figure 8 - back hole*

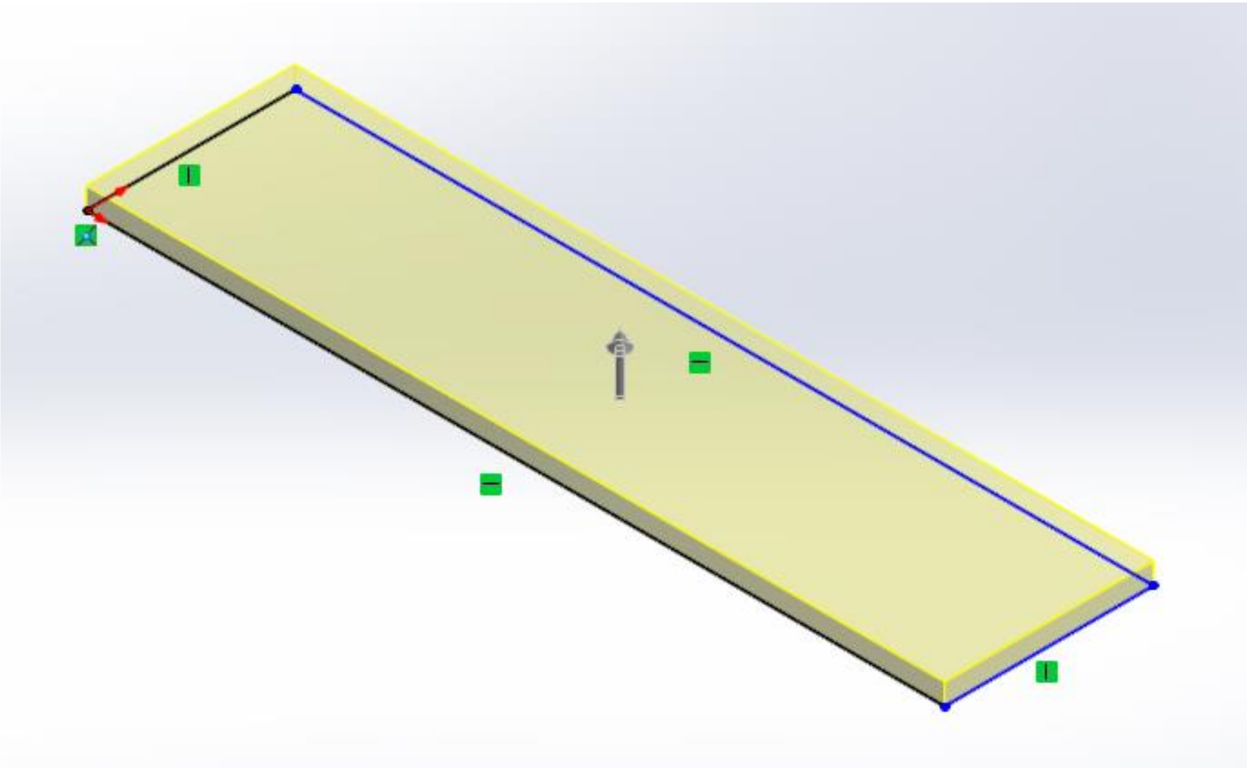


Figure 9 - The modeling of wooden plate—Drawing of rectangle and extrusion

The last part of our modeling was to assemble the metal frames and the wooden plates.

First, we created a reference plane 300mm (half of wooden plate length) away from the Right Plane. Then, we applied Mirror to the reference plane and the metal frame, which created a metal frame in the opposite side.

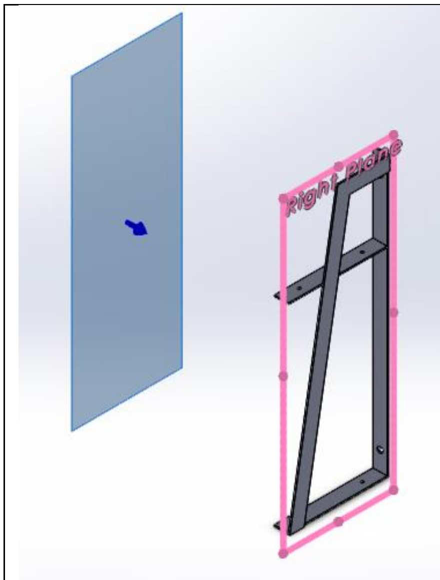


Figure 10 – Reference plane

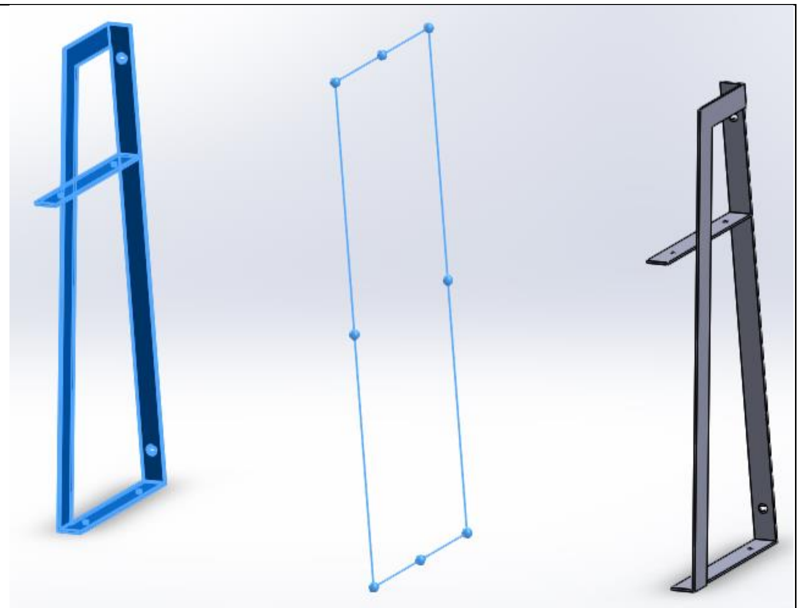


Figure 11 – Mirror to the reference plane & the metal frame

Next, we imported two wooden plate.

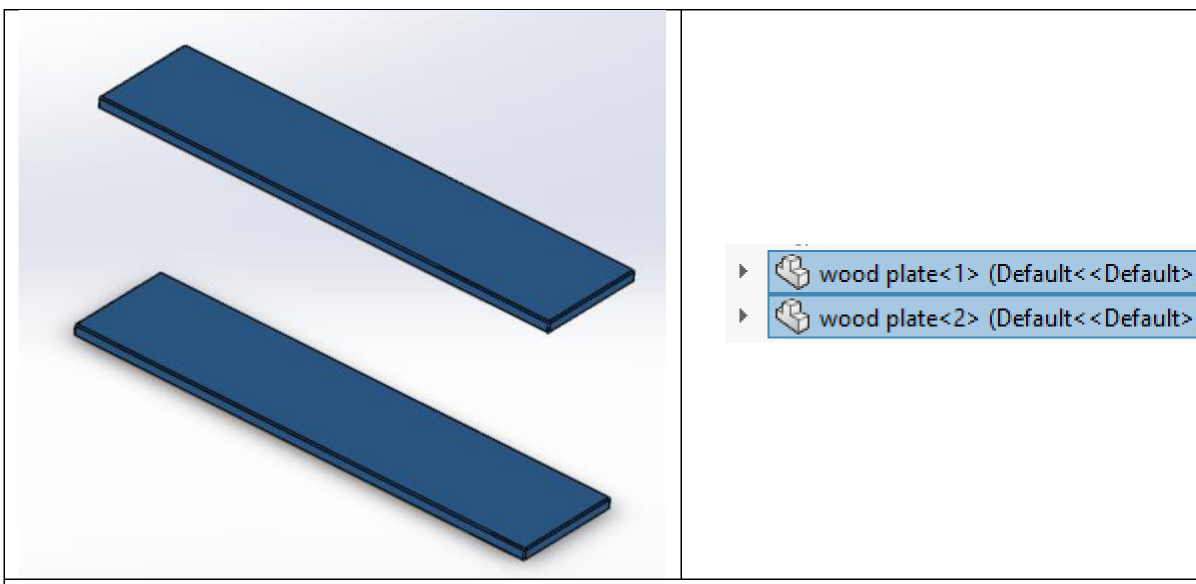


Figure 12 – Two wooden plate

In order to combine wooden plates and metal frames together, we applied the following mating:

-Applied Coincident to the bottom of wooden plates with the corresponding plate holder of the metal frame.

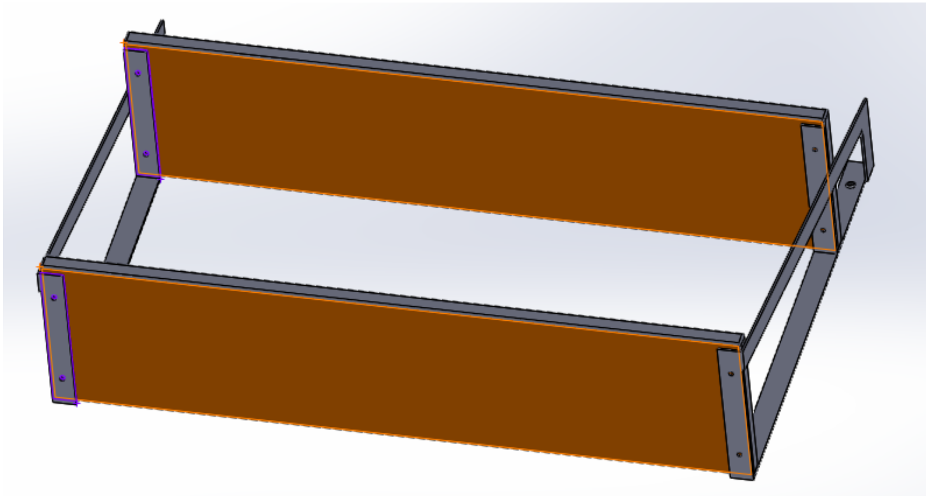


Figure 13 – Mating relationship between bottom of wooden plates and metal frame

-Applied Coincident to the side of wooden plates with the corresponding side of the metal frame.

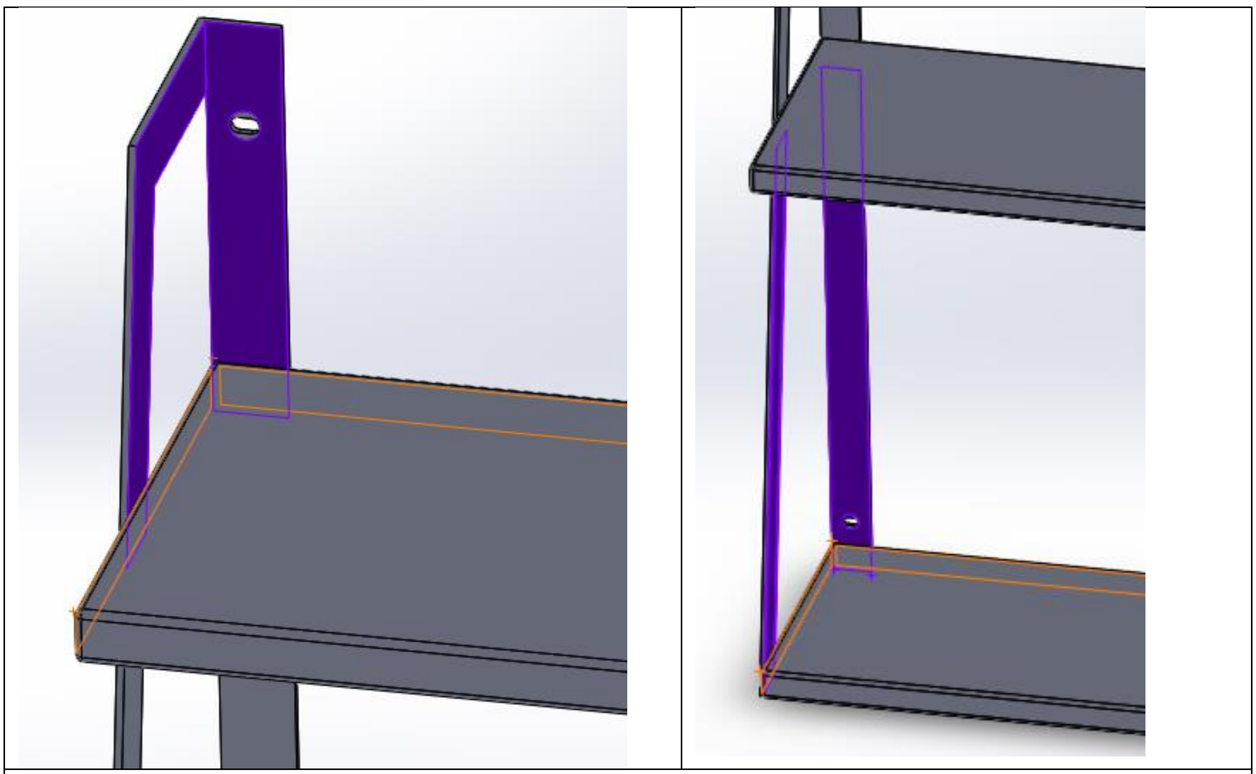


Figure 14 - Mating relationship between side of wooden plates and metal frame

This solid model was straight forward and no features were complicated enough to reduce the accuracy.

Materials

We purchased our book shelf on Amazon. However, it only showed that the frames were made of metal, and the plate were made of wood without specify the materials, so we had to find the best fit materials. For the frames, the material we chose was **AISI 1020**. The material we chose for the wood plate was **balsa**. The material properties we needed were the elastic modulus, Poisson's ratio, and the yield strength, which was showed in the Figure 15 below.

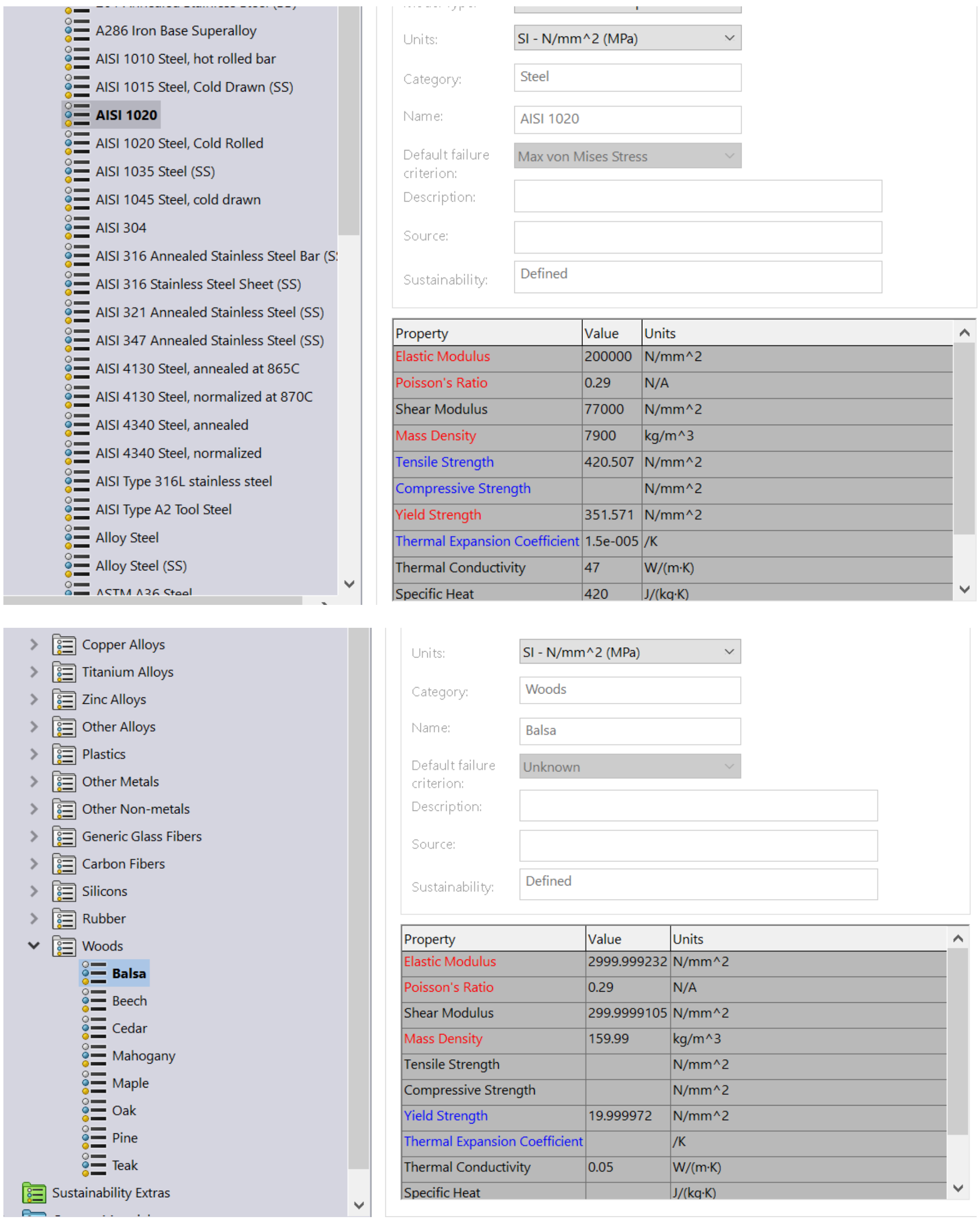


Figure 15 – Material Property of AISI 1020 & balsa

We first assumed that the part was made of steel. Then we took the part to the ME 461 laboratory and used Rockwell HRB to measure the hardness of the material. The data of HRB was around 65, which was close to the number we measured in ME 461 LAB steel 1020. We inferred that the material was steel 1020.

For wood, we chose Balsa for simulation because balsa had a very low density, and the wood we had was very light; also, Balsa was the only one had complete data in the SolidWorks. To verify that, we compared the weight with two material. We first measured the weight of the wood plate: 223 g. Then we use Balsa density: 159.99 $\frac{kg}{m^3}$ from AZOM to calculate the weight of the same dimensions wood plate:

$$V = 0.6 \times 0.146 \times 0.015 = 0.001314\, m^3$$

$$M = V \times \rho = 0.001314 \times 159.99 = 210\, g$$

The wood was very light, and the weight we obtained from calculation and measuring matched. Since our main object was the metal frame and the anisotropic property of the wood, we didn't look at the deformation on the wood plate.

Since the AISI 1020 was a ductile material, among the three failure theories we learned in class, the Von Mises failure theory would be the best fit. Again, because of the complexity and deficiency learning, no analytical calculation could be provided and we would compare the results to AISI yield strength for safety judging.

The description provided that the maximum allowable weight was 45 lbs. However, it didn't give us any information about how to distribute the object was safe. Therefore, we examined extreme cases based on real life possibilities, which could be broke into the following three cases:

Situation 1: One point (concentrated at one point) 200 N force in the middle of plate. This would simulate the daily usage of shelf, where people would place heavy objects (award trophy, wooden statues) at the center of wooden plate. The was conducted to verify the safety if the shelf would be safe when maximum load placed in the middle.

Situation 2: Uniform 200 N force on the plate surface. This would simulate the daily usage of shelf where people would evenly place light objects (books, toys) at the wooden plate. This was the most normal usage situation and conducted to verify if the maximum 45 lbs was possible in normal daily use.

Situation 3: Uniform 200 N force concentrated on the one side of the plate. This would simulate the not careful usage of shelf, where people accidently placed heavy objects (award trophy, wooden statues) at the edge of wooden plate. This was an extreme case. Even though people won't purposely intend to do so, the situation might happen due to careless mistakes. For example, if the person was trying to clean the wood and so put all the stuff on one side.

For each case, we would run few simulations first with middle fine mesh first. This helped us to identify the location of the possible maximum stress under each situation. Then we would run the simulation with standard mesh, which had the global mesh size of 6 mm and local refinement on the potential location with less than 3 mm mesh size. The restraint boundary conditions would be the same for all 3 cases and the load conditions would be different for each case.

First, we would apply force on the upper level plate only. Then, we would shift the force to the lower level plate. Lastly, we would split the same force on both upper and lower level plate.

Boundary Condition

The model we had for this project contains 4 parts, which were the left frame, the right frame, the upper wood plate, and the lower wood plate. When we were trying to run the simulation, we thought we could remove the wood plates, and use one frame part to do the simulation because we were focusing on the frame part, and everything else seems not important toward the results. The first step for running the simulation was to set up the correct boundary conditions. After the first group meeting, we thought there were 2 restraints and one force. Again, we neglect the weight of the wood plate. The back of the frame should be fixed because there was a wall behind of it. Also the hole for the crews should be fixed as well. The force should be uniformly applied to the part of the frame where holds the wood plate. However, when we ran the simulation, the result we got did not meet with our expectation. The outer edge of the part that holds the wood plate deformed more than anywhere else. Also the bottom part deformed too much as well. We realized that the boundary conditions we had are not correct.

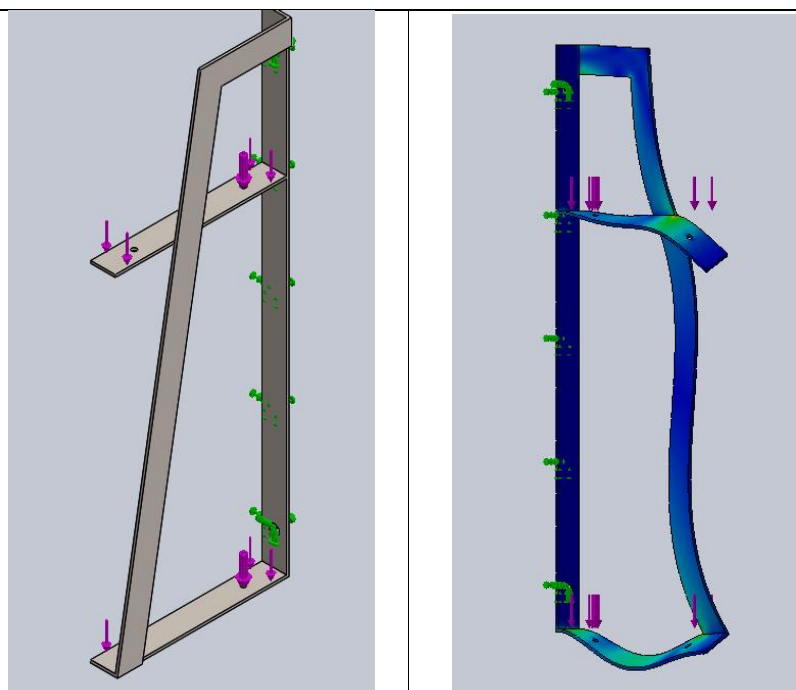


Figure 16 – Simulation result

Then, we tried to use virtual wall, which served as a "wall." The frame can't bend into the wall, but it could bend to the other direction (out of the wall). Also, the virtual wall did not prevent the wall moving or deforming in the z axis (up or down), which was more fit to the reality. We also added another restraint around the screw holes because the screw heads also prevent the frame from deforming. However, the new boundary conditions failed as well. It ran for more than 14 hours and couldn't show the result (Figure 17).

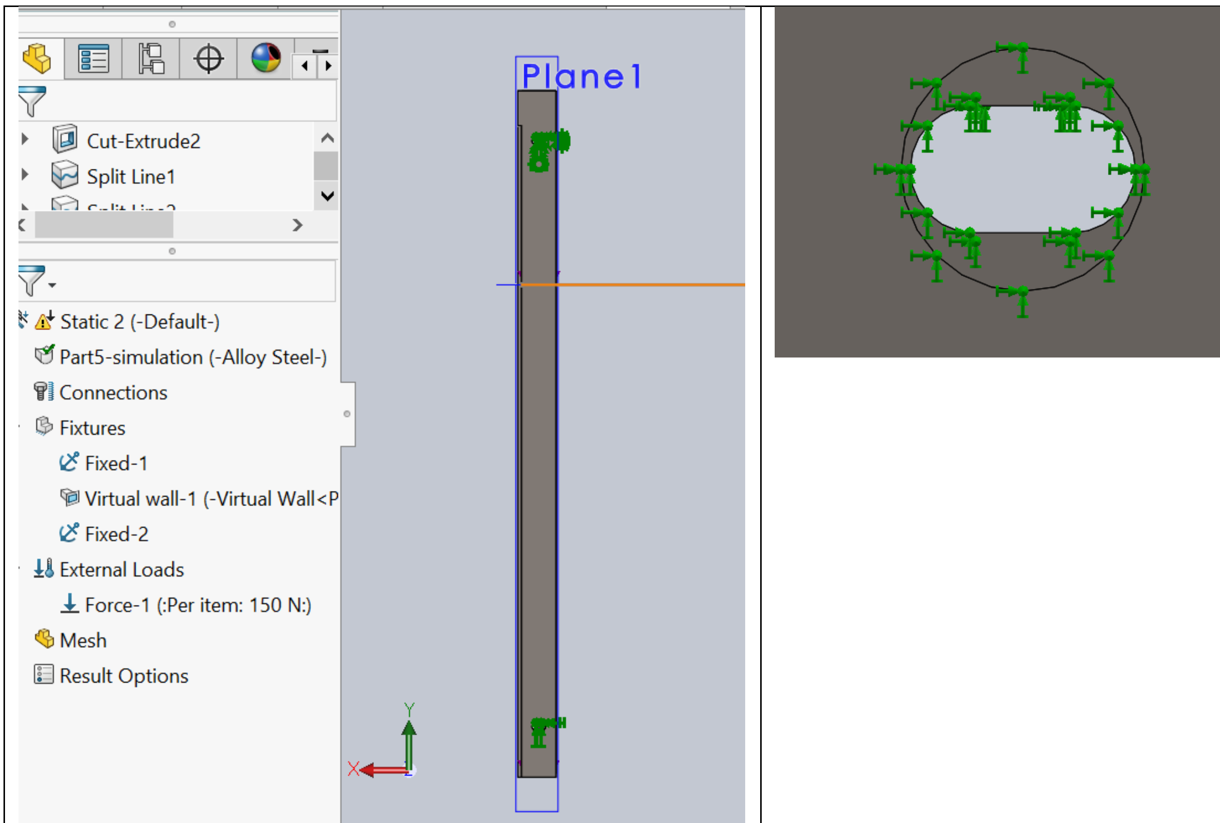


Figure 17 – Left: Virtual wall, Right: restraint around the screw holes

Next, we used the advanced fixture "On Flat Faces" for the wall. We selected the back surface of the frame, chose the direction "into the wall," and set the translation equal to zero. This meant the frame cannot bend into the wall because the translation was equal to zero. We also used the same method to the surface around the screw holes. However, the situation did not get better. The deformation was almost identical as the first one (with "fix geometry").

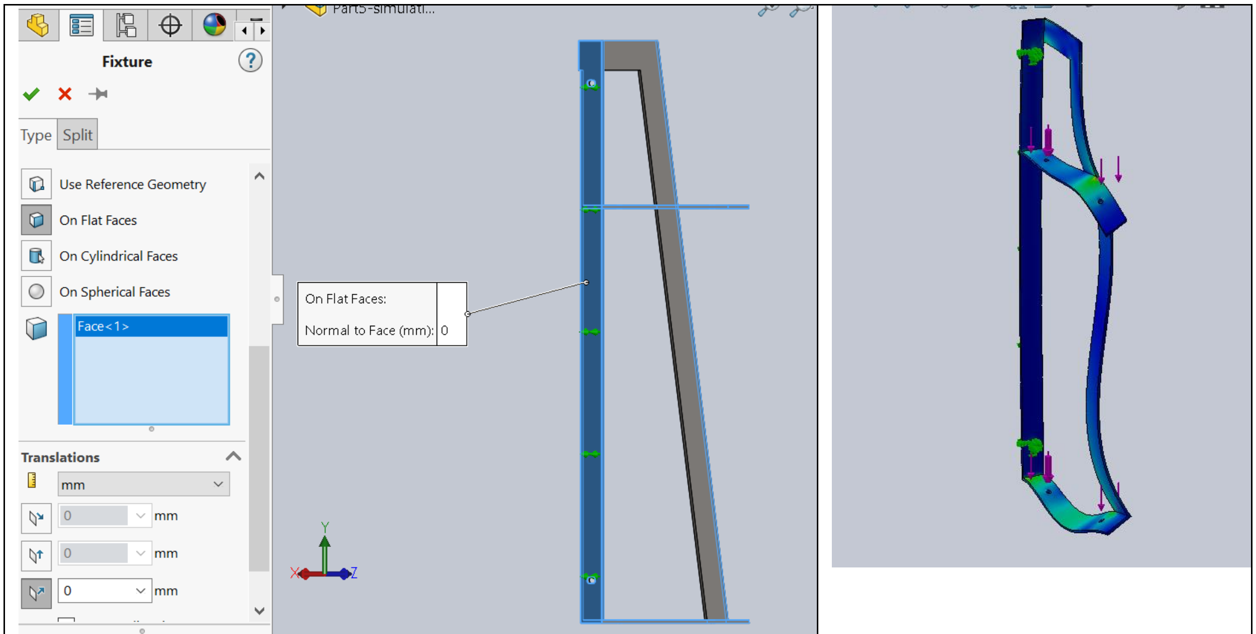


Figure 17 – Left: On Flat Faces, Right: Simulation result

We made a conclusion that it was not about the restraints. It was because of the load we applied. The load was not uniformly applied to the frame. Instead, the load on the outer edge should not be that great. The part that connected to the vertical frame should take more load.

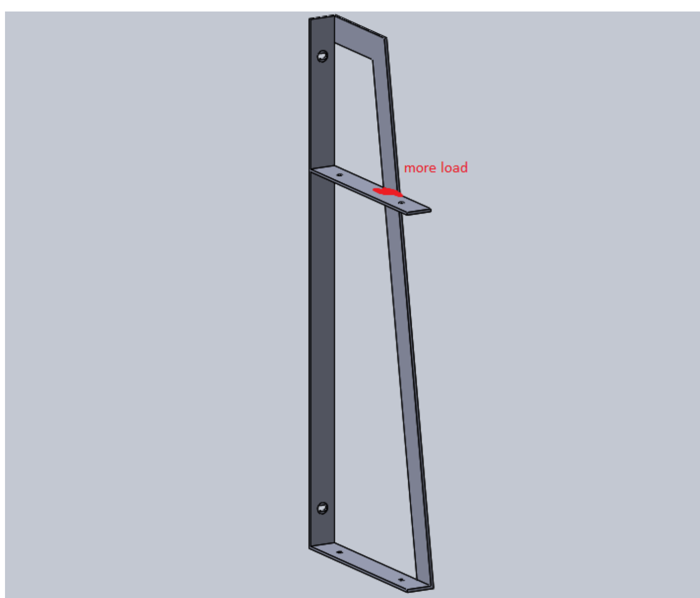
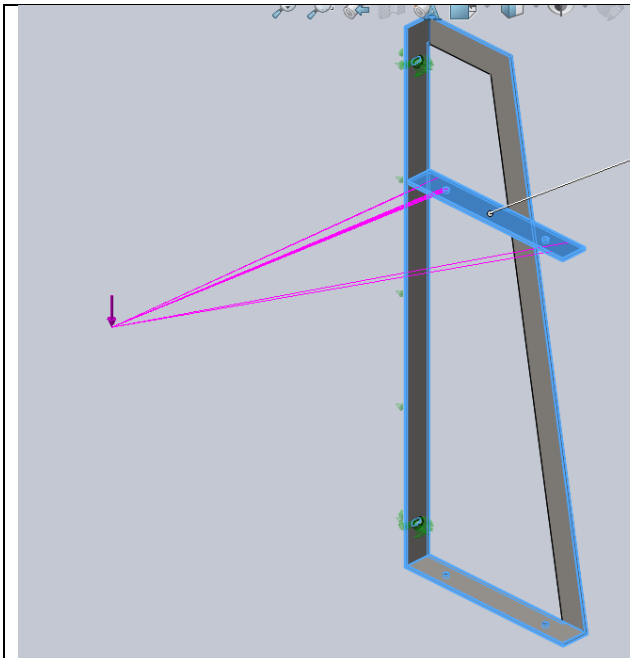


Figure 18 –Applied load

Then we tried to change the type of load that applied to the frame. The load type we chose was “Remote load (single point).” The load was applied to one point, and we could choose the location of the load. We assumed the load was in the center of the wood plate, and led the load apply to the part we have. In this situation, the deformation was also not correct.



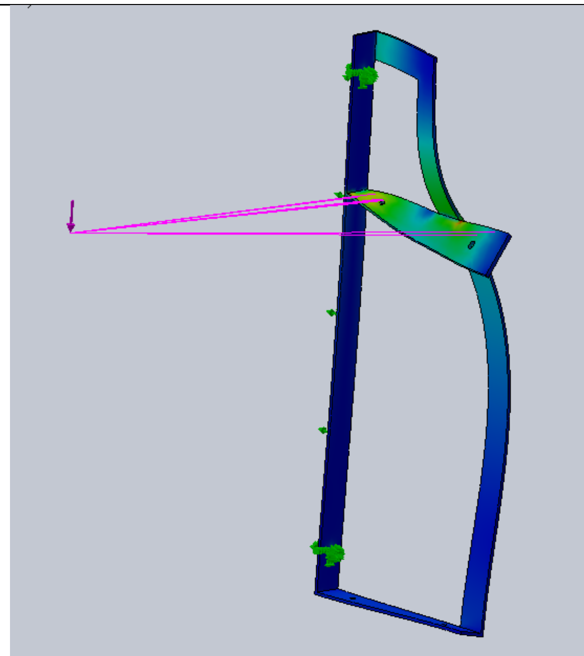


Figure 19 – Left: Remote Load, Right: Simulation result

We realized that the force was applied to the wood plate first, and the wood plate transmitted the force to the frame. We could not directly apply the load to the frame. Therefore, we need to use the entire assembly for our simulation.

We applied the same boundary conditions (fixed with the hole, on flat planes for the wall), and applied a uniform force to the wood plate. The deformation looked correct this time. In addition, in order obtain a converging stress, we applied 2mm fillet to the edge of wooden plate (which was 90 degree).

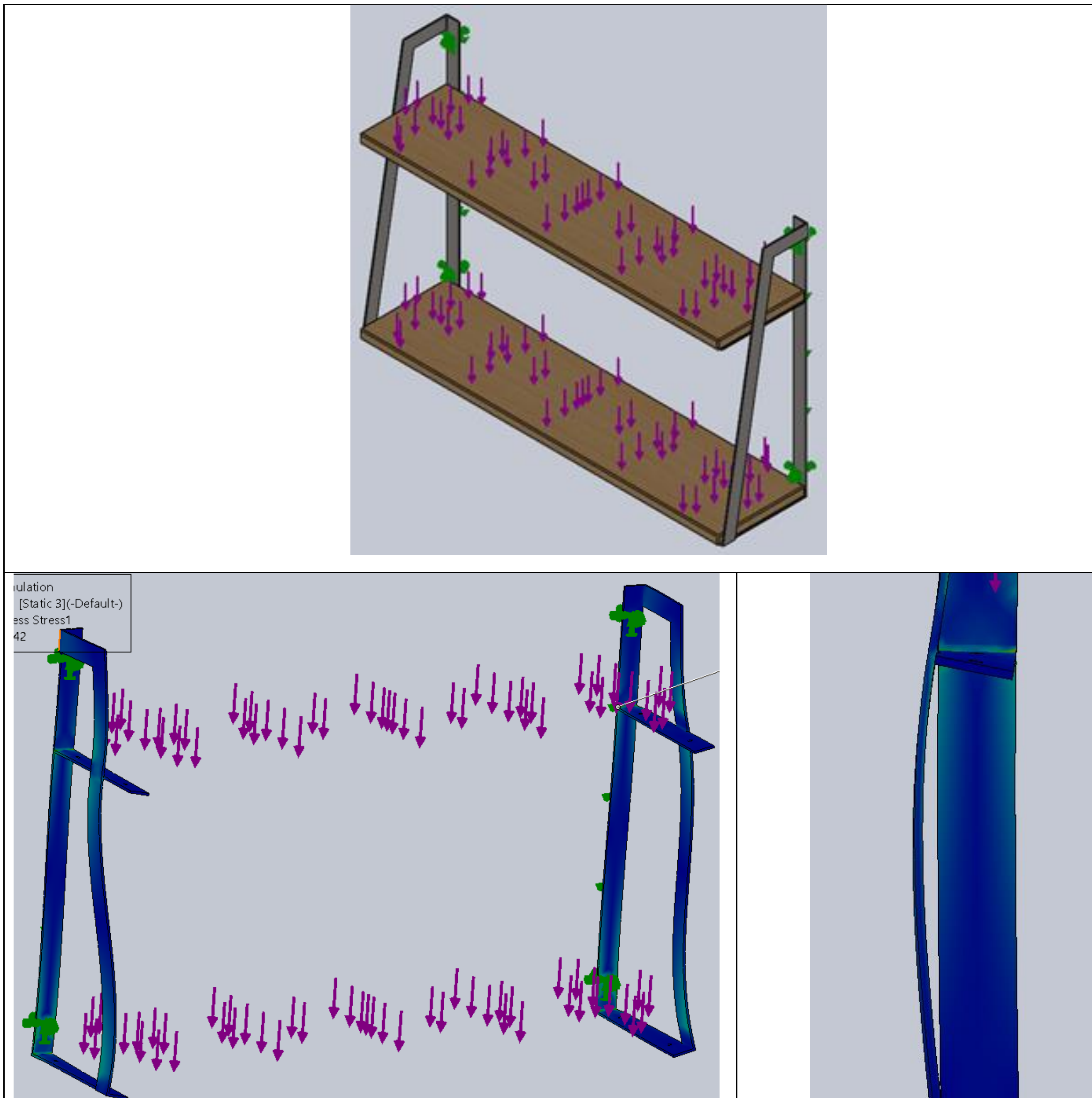


Figure 19 – Left: Remote Load, Right: Simulation result

There were 3 different cases we thought about. The first one was the load was uniformly distributed to the plate. The second one was the load was on one end of the plate. The third one was the load was in the center of the plate (single point). We tried to compare which one was the worst case. Moreover, in each case, we tried different conditions. 1. What happened if the load was on the upper plate only 2. What happened if the load was on the upper plate only 3. What happened if the load was applied to both plates.

Situation 1

We first ran a few simulations with global element size only, and uniform loading on the upper plate. The maximum stress kept diverging, and the maximum stress location jumped between L-shape edges and fixing holes. We noticed that this might be caused by that we created the solid modeling with 90 degree shapes. This would produce weird calculation in the SolidWorks. The original products were created by welding, and was not 90 degree between the edges. Therefore, to obtain the correct value of maximum stress, we did a sensitivity test as FEM2 by applying fillets on the 90 degree edges. The graph was shown below (Figure 20). The filets where the maximum stress start to converge was r =2 mm.

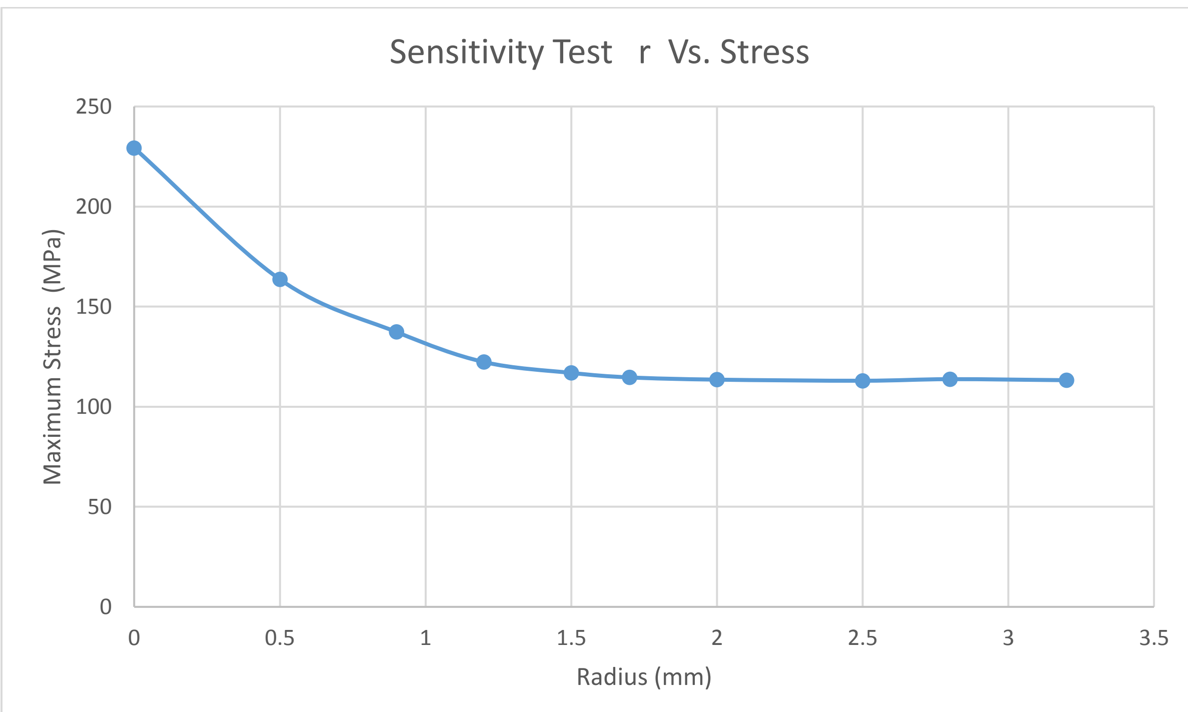


Figure 20 – Sensitivity test

For the remote load (load in the center of the wood plate), we tried three different cases. The first one was the load on the upper wood plate only, the second one was the load split on both plates, and the third one was load on the lower wood plate only (Figures 21, 22, 23). By comparing these three cases, we could see that which was the worst situation, and what were the differences between each condition. For the first case, the global mesh size we chose was 6 mm. And we tried 9 different local element sizes from 5 decreased to 1.6. After we ran the simulation, we knew that the maximum stress would probably located at the edge of the frame (Figure21, 24), so we chose the local element location to be around there. We chose the corner surfaces, the side surfaces, and the surfaces that holds the plate (Figure 25).

However, when we ran the simulation for the third case, which was the load on the lower plate, initially the maximum stress was on the edge. As we decreased the mesh size, the location of the maximum stress changed. It went to the screw hole. It might because of the split line we drew (Figure 26). The split line might affect on the calculations and cause some errors. Also it's possible that the location of the maximum stress was correct, and it should be there. However, since we were focusing more on the edges, we chose the 2nd largest maximum stress that located at the edge for our results. Moreover, since it's possible that the maximum stress was located on the screw hole, we would also improve the design of the screw hole as well, and we would discuss about the improvement in later sections.

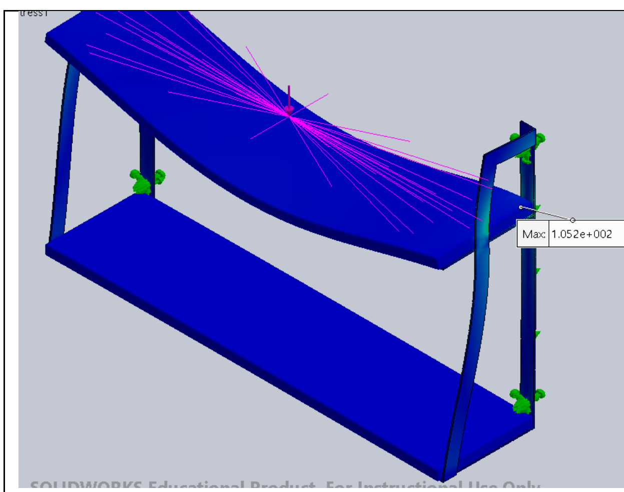


Figure 21: Load on top plate

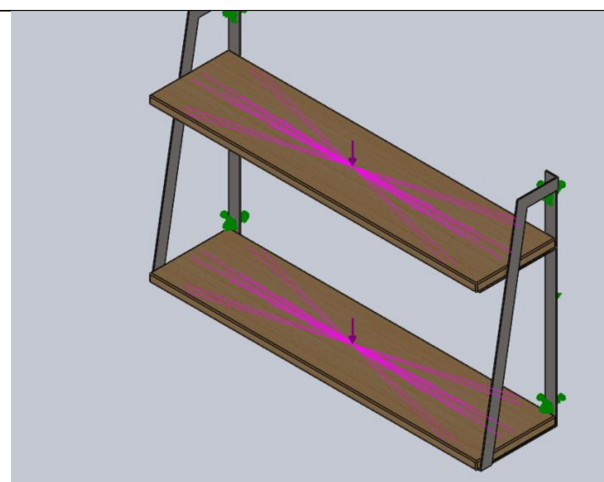


Figure 22: Load on both plate

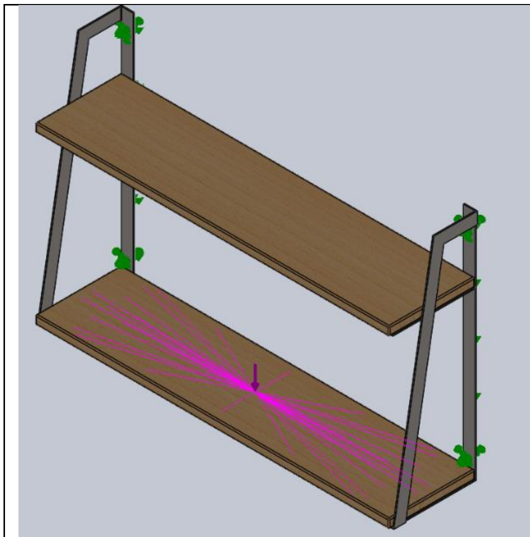


Figure 23: Load on bottom plate

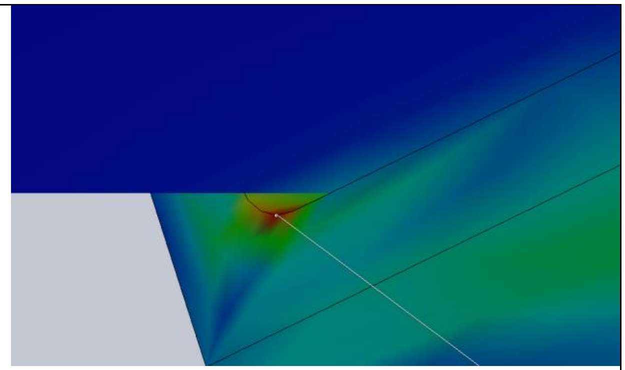


Figure 24: Maximum stress location

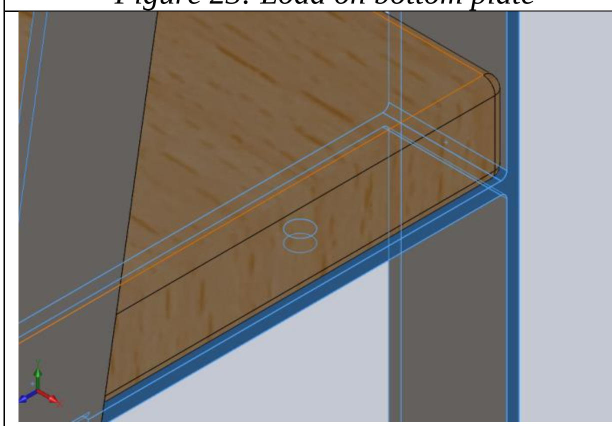


Figure 25: The surface holding the plate

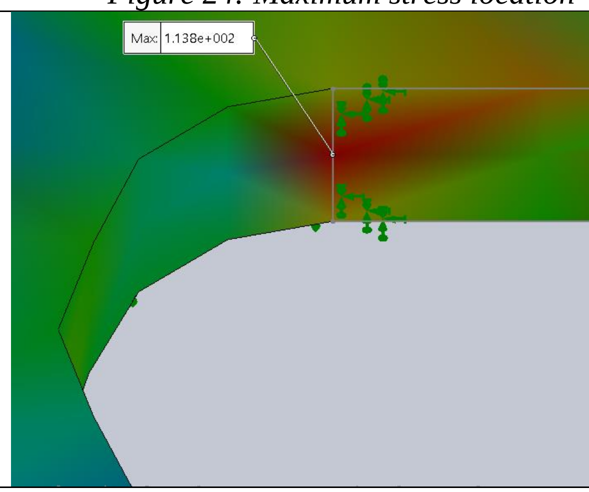


Figure 26: Maximum stress in the hole

Situation 2

For the uniform force distributed on the wooden plate, we tried three different cases. The first one was the load on the upper wood plate only, the second one was the load split on both plates, and the third one was load on the lower wood plate only (Figures 27, 28, 29). By comparing these three cases, we could imagine that which was the worst situation, and what were the differences between each condition. For the first case, the global mesh size we chose was 6 mm. And we tried 7 different local element sizes from 5 decreased to 1. After we ran the trial simulation with 9mm global mesh size, we observed that the maximum stress was located at the edge of the frame (Figure27, 30), so we applied the split line to cut the small area around the edge of the frame (Both top and bottom, then applied the local refinement around there (Figure 31, 32, 33, 34). Since adding split line was changing the model, so we renewed mating and boundary with the updated geometry.

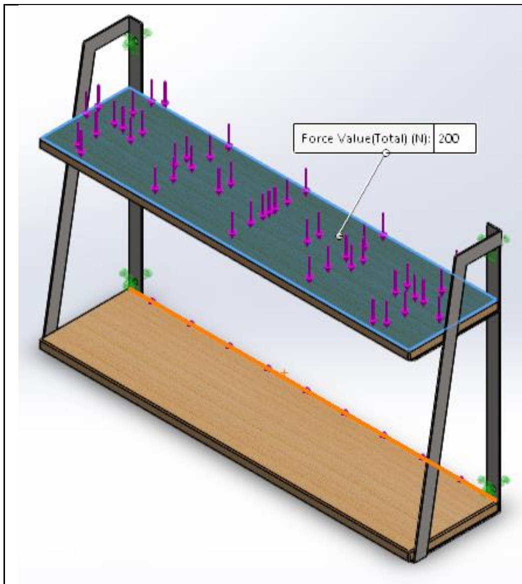


Figure 27: Load on top plate

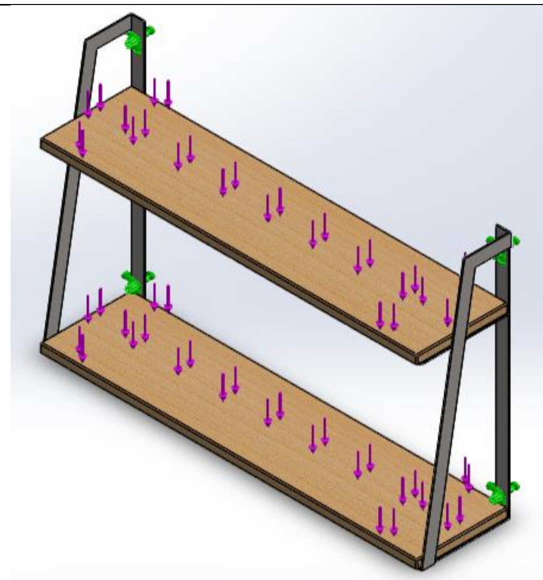


Figure 28: Load on both plate

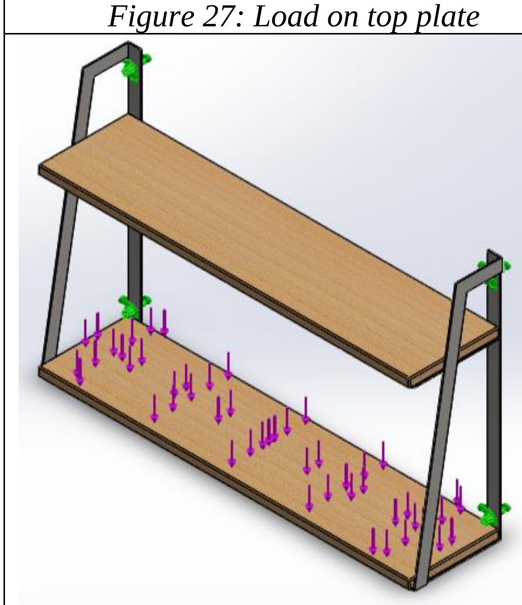


Figure 29: Load on bottom plate

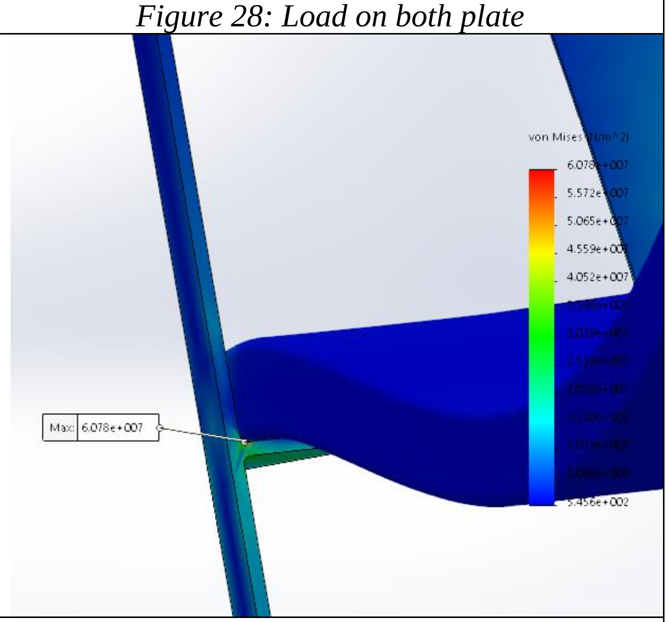


Figure 30: Maximum stress location

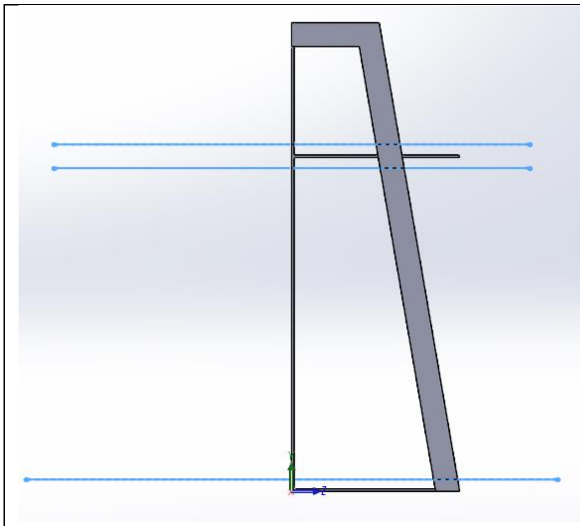


Figure 31: The surface holding the plate

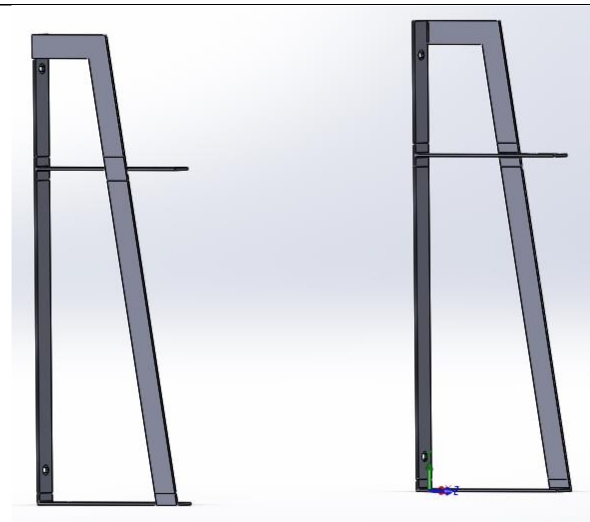


Figure 32: Maximum stress in the hole

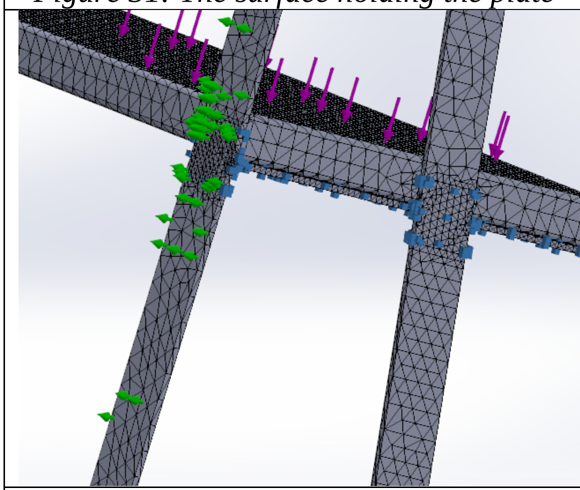


Figure 33: Local refinement - top

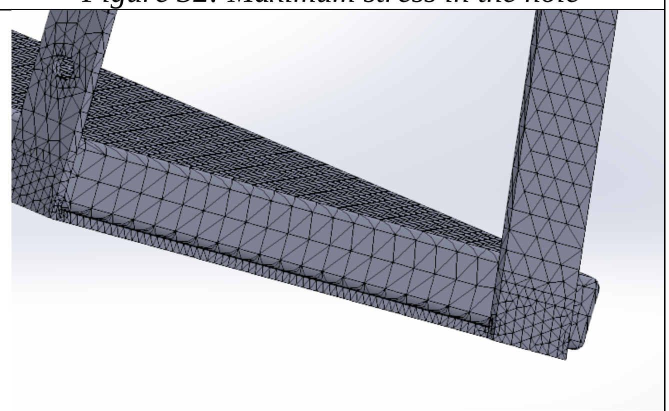


Figure 34: Local refinement - bottom

Situation 3

In this situation, as mentioned before, we would investigate if the frame would fail under the maximum loading of one side. The 200 N would be loaded on the first level first and then the loading on the second level would be investigated. To achieve the boundary conditions, we applied split line on the wood plate with a square of 10 x 90 cm as shown in the Figure 35. The 200 N boundary conditions would be loaded uniformly in the square as shown in the Figure 36 and 37. First, we run a few simulations with the upper plate load and found that the possible maximum stress locations were in the upper hole of the side of loading, the L region of the beam (including the fillet) and the crossing regions around the loadings. We performed global element size of 6 mm and local refinements in these location from 3 to 0.6 mm.

For the bottom plate loading case, which was similar to the upper plate case, the first few times of simulation indicated the sensitive region at the lower holes of the loading side, the L shape beam and the fillets. The first set of simulation provided diverging maximum stress within the holes as the case 1 and 2. The stress could possibly be in the hole, but the stress locating on the transition tangent edge as shown in Figure 38. This stress situation was more likely caused by geometry influence of solid modeling. Therefore, we tried to reduce the fixture area within the hole, and the second set of the simulation provides the maximum stress on the edges still. Therefore, we concluded to ignore the stress on the edges and focus on the location having the second maximum stress, which located on the filet of lower bottom. We then run a simulation of 6 mm global element size and 3 to 1.25 mm local refinement on the fillets.

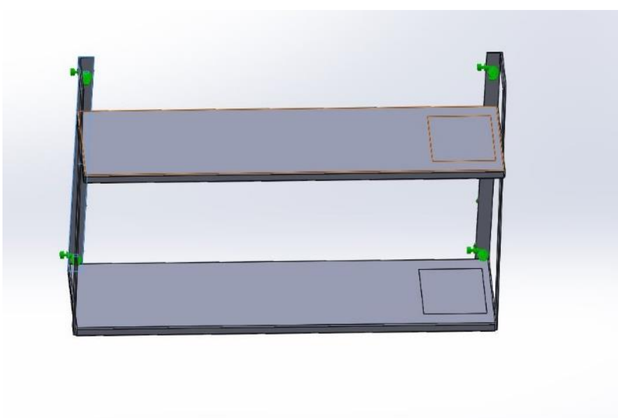
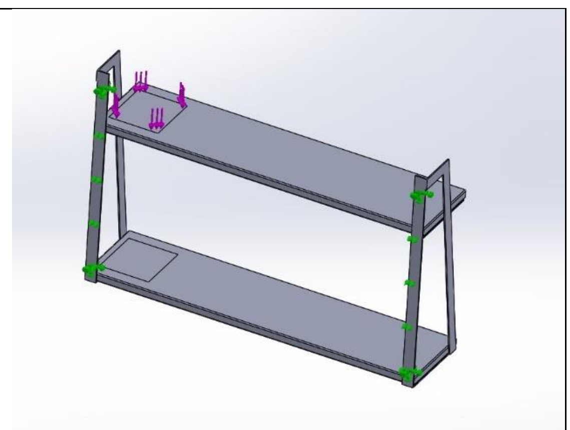
	

Figure 35: The square area on plate	*Figure 35: Load on top plate*

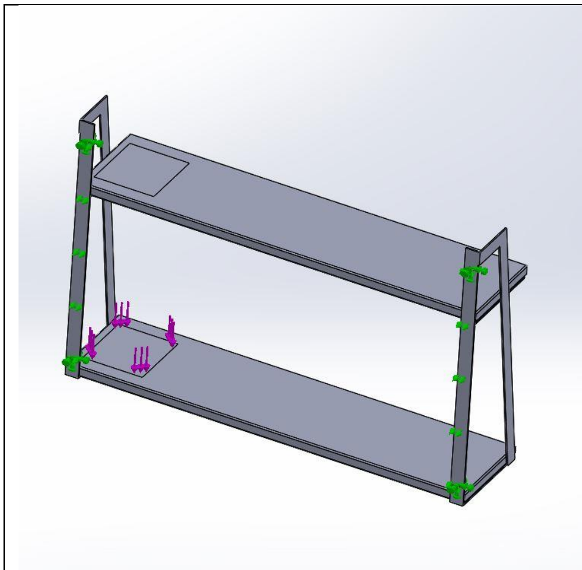


Figure 37: Load on bottom plate

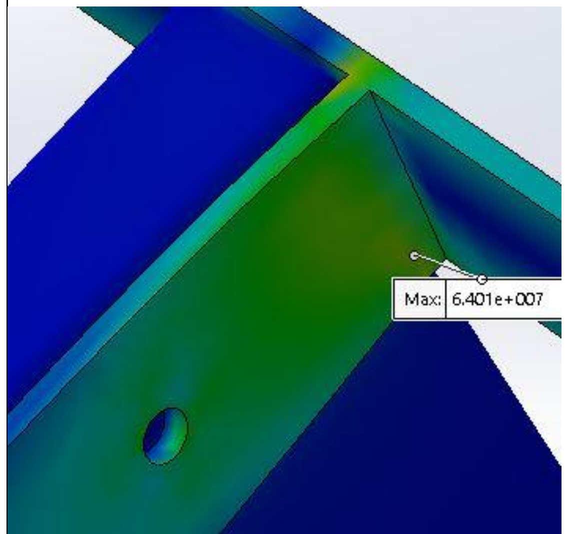


Figure 38: Maximum stress location

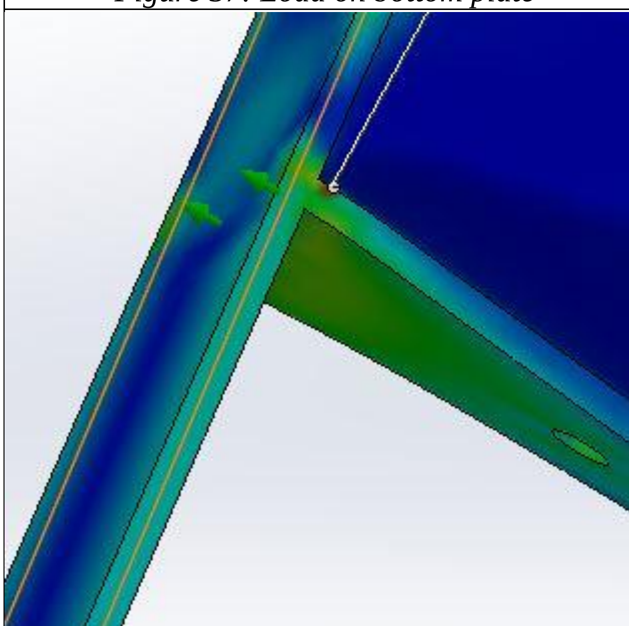


Figure 39: Maximum stress location

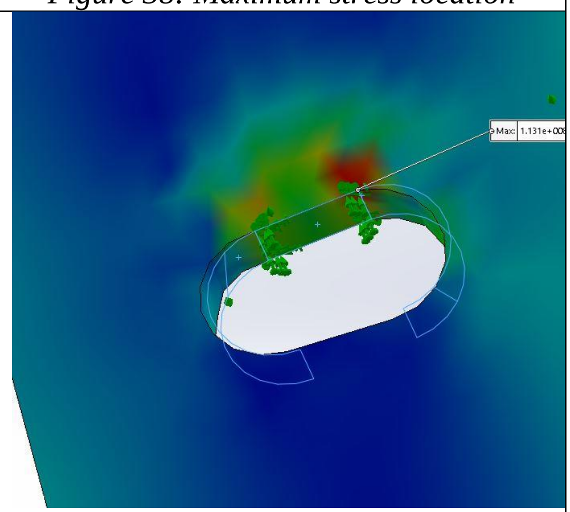


Figure 40: Maximum stress in the hole

Data Analysis:

Once we had finished our stimulation, we were moved on to the analysis. As we mentioned earlier in the assumption, for this project, we would analyze three Situation:

1. One point 200 N force in the middle of plate.

2. Uniform 200 N force on the plate surface.

3. Uniform 200 N force concentrated on the one side of the plate.

For comparison purpose, in each case, we would run three different force conditions. First, we would apply force on the upper level plate only. Then, we would shift the force to the lower level plate. Lastly, we would split the same force on both upper and lower level plates.

Our results were shown in the next few pages.

Situation 1: One point force in the middle of plate.

The following tables and graphs were the various size of mesh that we applied to the model.

Table 2: 200 N Force on the upper plate

Case #	$\sigma_{Von\ Mises}$	Displac.	Element	nodes	DOF	Global	Local	time
	MPa	*mm*	*10^3*	*10^3*	*10^4*	*mm*	*mm*	*sec*
1	105.2	1.573	99.800	163.355	48.996	6	-	12
2	110.9	1.575	102.864	168.284	50.474	6	5	13
3	97.63	1.575	106.862	175.184	52.544	6	4	13
4	103.6	1.578	115.704	189.826	56.937	6	3	14
5	109.7	1.577	126.705	207.035	62.0	6	2.5	17
6	116.7	1.579	140.114	229.804	68.930	6	2	17
7	117.8	1.579	153.557	250.026	74.997	6	1.8	19
8	117.3	1.578	173.943	281.249	84.364	6	1.6	22
9	116.9	1.579	185.611	299.174	89.738	6	1.5	23

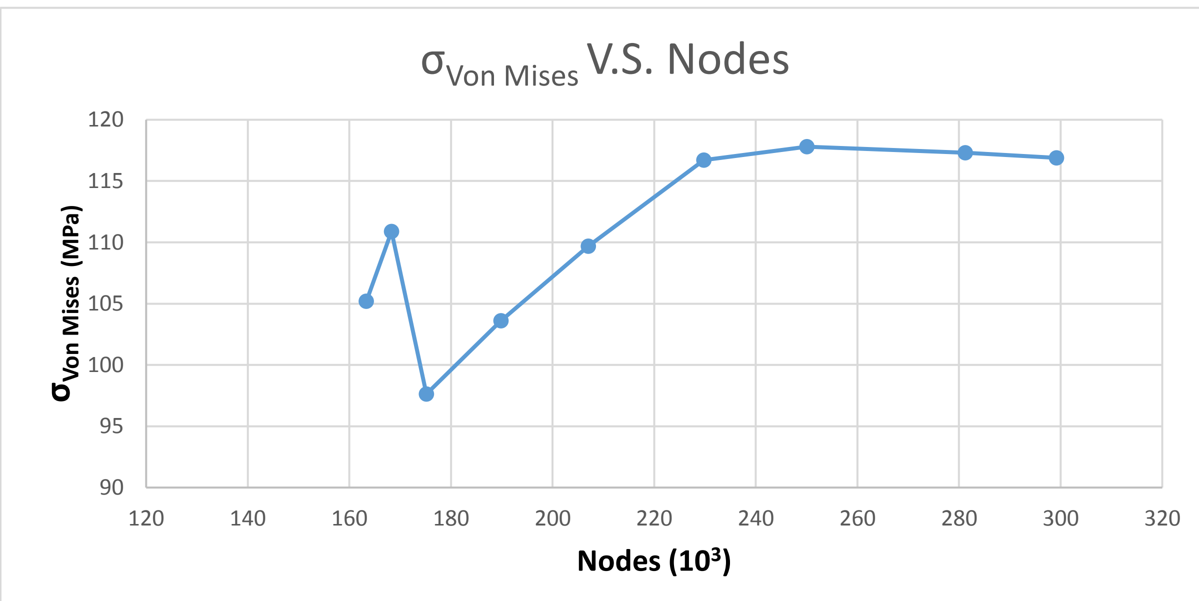


Figure 41: The value was converging to 169MPa

Table 3: 200 N Force on the lower plate								
Case #	**$\sigma_{Von\ Mises}$**	**Displac.**	**Element**	**nodes**	**DOF**	**Global**	**Local**	**time**
	MPa	mm	10^3	10^3	10^4	mm	mm	sec
1	116	1.735	102.356	166.851	50.044	6	-	13
2	111.6	1.736	104.221	169.592	50.867	6	5	13
3	101.9	1.736	107.031	174.300	52.279	6	4	14
4	111.1	1.735	115.423	186.594	55.967	6	3	15
5	103.2	1.737	135.562	217.148	65.134	6	2	15
6	105	1.737	144.773	230.225	69.057	6	1.8	18
7	104.7	1.737	160.874	252.715	76.104	6	1.6	20
8	104.9	1.737	169.806	265.409	79.612	6	1.5	21

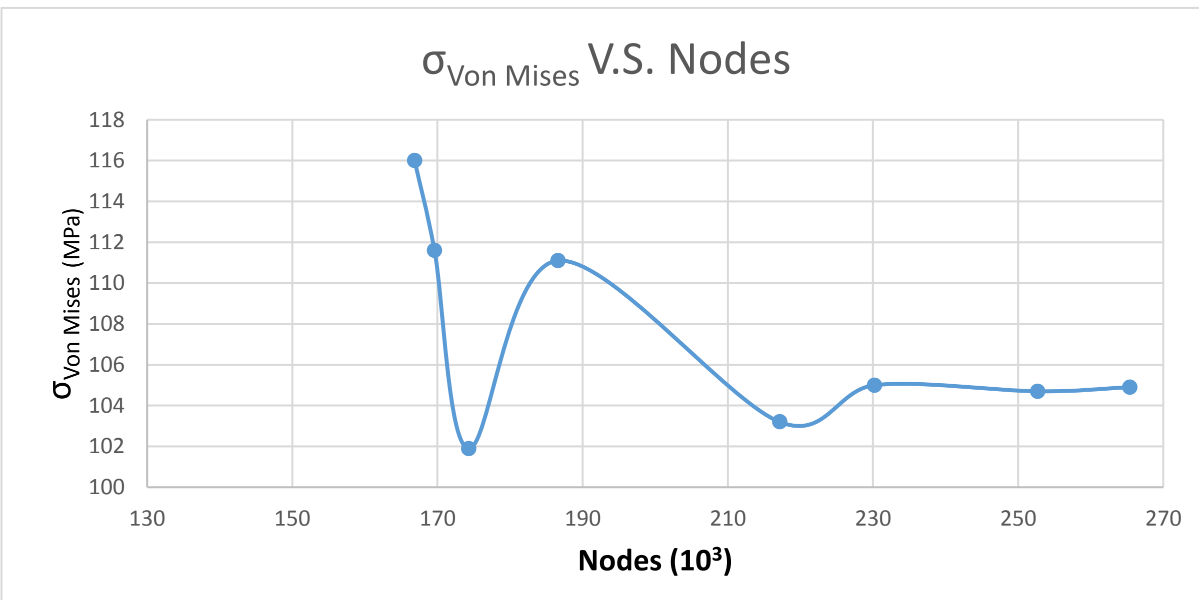


Figure 42: The value was converging to 104Mpa

Table 4: 100 N Force on both lower and upper plate								
Case #	**$\sigma_{Von\ Mises}$**	**Displac.**	**Element**	**nodes**	**DOF**	**Global**	**Local**	**time**
	MPa	mm	10^3	10^3	10^4	mm	mm	sec
1	59.08	0.8424	102.944	167.881	50.354	6	-	13
2	61.45	0.8426	105.553	172.235	51.66	6	5	13
3	54.26	0.8424	109.902	179.874	53.951	6	4	14
4	57.93	0.8425	122.201	199.898	59.959	6	3	16
5	59.29	0.8426	155.295	253.604	76.070	6	2	20
6	61.96	0.8426	173.087	280.823	84.236	6	1.8	20
7	61.29	0.8426	203.598	326.971	98.081	6	1.6	22
8	61.35	0.8426	220.758	353.449	106.021	6	1.5	23
9	61.23	0.8426	241.030	385.164	115.496	6	1.4	26

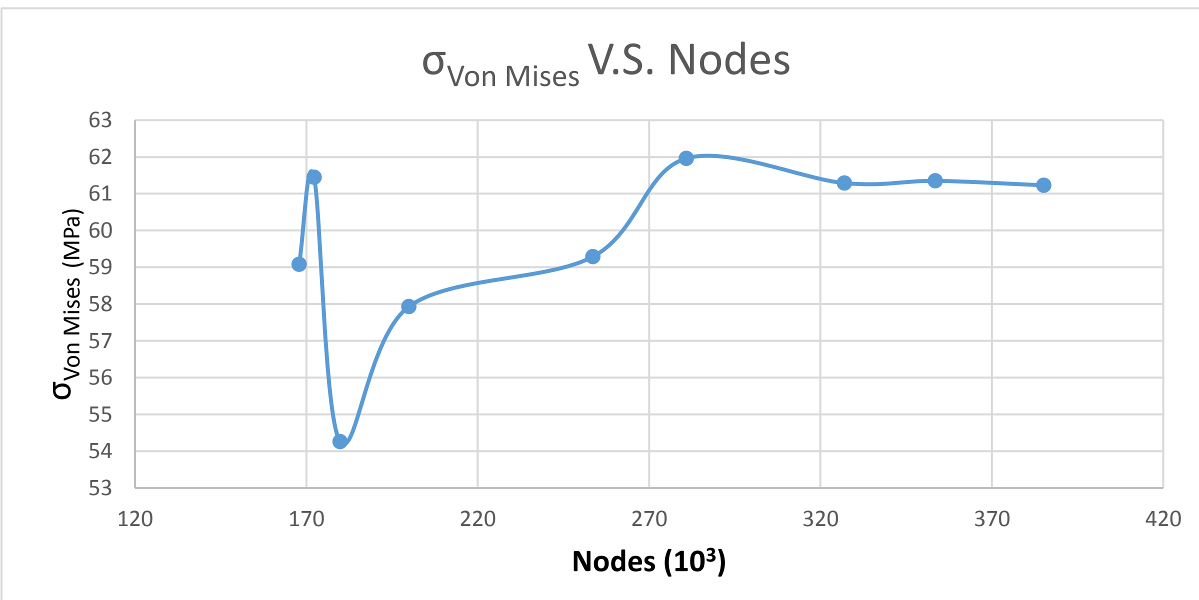


Figure 43: The value was converging to 69MPa

Situation 2: Uniform force on the plate surface.

The following tables and graphs were the various size of mesh that we applied to the model.

Table 5: 200 N Force on the upper plate								
Case #	**$\sigma_{Von\ Mises}$**	**Displac.**	**Element**	**nodes**	**DOF**	**Global**	**Local**	**time**
	MPa	*mm*	*10^3*	*10^3*	*10^4*	*mm*	*mm*	*sec*
1	108.1	1.553	34.997	61.969	18.58	9	-	11
2	106.6	1.556	72.721	120.319	36.085	7	-	17
3	99.43	1.574	99.783	163.459	49.027	6	-	24
4	116.4	1.545	129.071	216.628	64.970	6	5	68
5	115.0	1.539	133.086	224.261	67.260	6	3	82
6	114.3	1.553	174.115	291.257	87.359	6	2	184
7	113.8	1.535	325.227	530.421	159.108	6	1	395

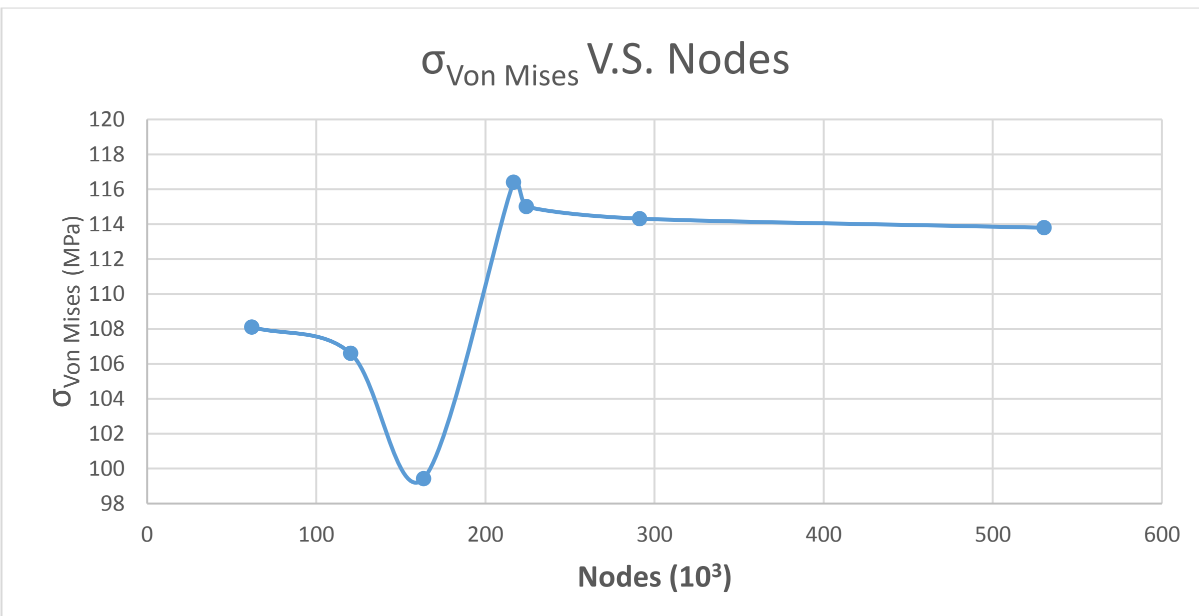


Figure 44: The value was converging to 113MPa

Table 6: 200 N Force on the lower plate								
Case #	**$\sigma_{Von\ Mises}$**	**Displac.**	**Element**	**nodes**	**DOF**	**Global**	**Local**	**time**
	MPa	*mm*	*10^3*	*10^3*	*10^4*	*mm*	*mm*	*sec*
1	131.3	1.715	35.037	62.035	18.6	9	-	14
2	121.9	1.727	72.721	120.319	36.085	7	-	30
3	119	1.737	99.783	163.459	49.027	6	-	32
4	101.5	1.691	137.639	229.626	68.87	6	5	98
5	117.1	1.687	141.654	237.259	71.16	6	3	84
6	101.6	1.695	174.115	291.257	87.359	6	2	185
7	100.45	1.675	321.726	525.021	157.488	6	1	343

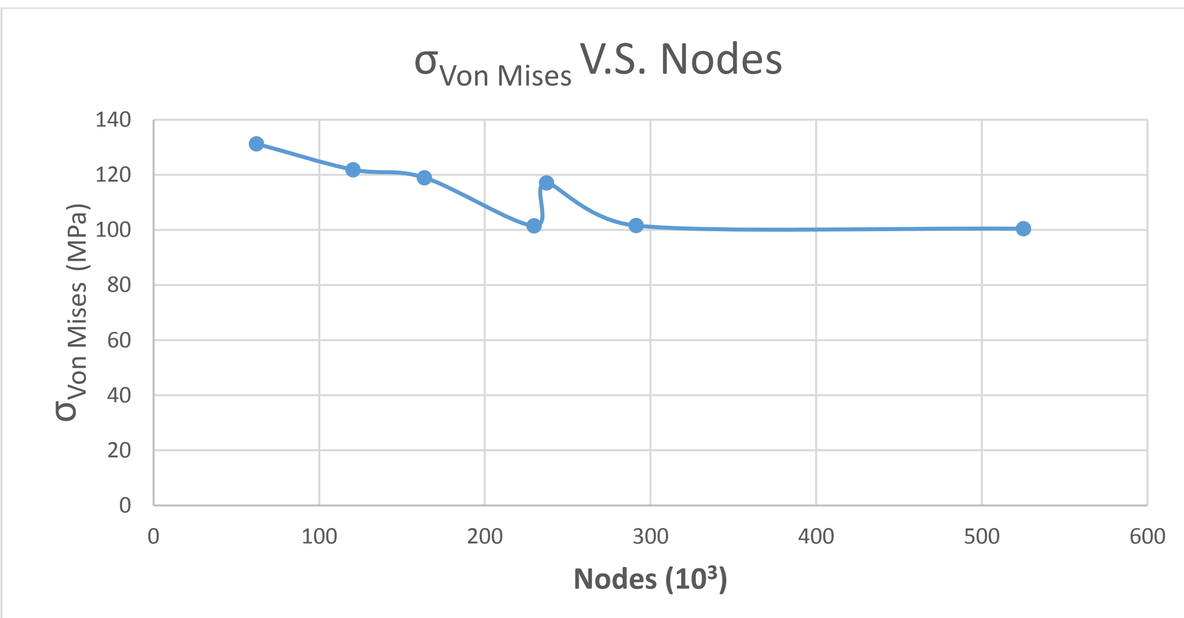


Figure 45: The value was converging to 100MPa

Table 7: 100 N Force on both lower and upper plate								
Case #	**$\sigma_{Von\ Mises}$**	**Displac.**	**Element**	**nodes**	**DOF**	**Global**	**Local**	**time**
	MPa	*mm*	*10^3*	*10^3*	*10^4*	*mm*	*mm*	*sec*
1	56.29	0.833	34.997	61.969	18.58	9	-	13
2	55.24	0.838	72.721	120.319	36.085	7	-	22
3	56.51	0.843	99.783	163.459	49.027	6	-	33
4	61.88	0.837	129.071	216.628	64.970	6	5	68
5	61.09	0.835	141.666	237.275	71.165	6	3	88
6	60.57	0.839	174.148	291.297	87.371	6	2	187
7	60.56	0.835	349.525	568.182	170.437	6	1	685

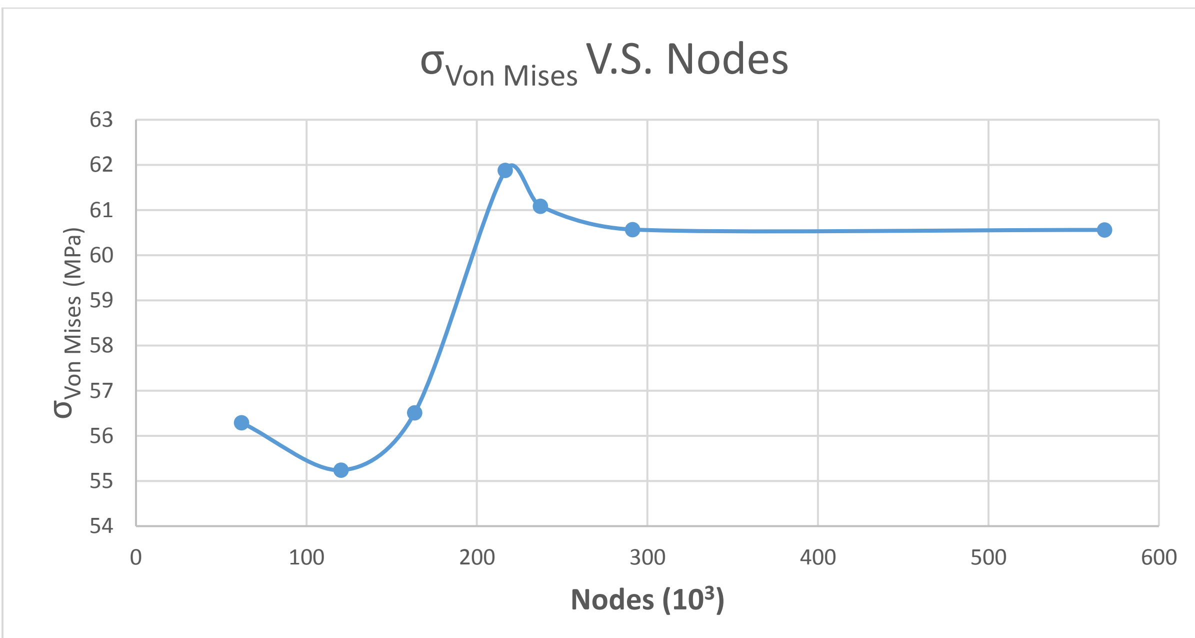


Figure 46: The value was converging to 60.5MPa

Situation 3: Uniform force concentrated on the one side of the plate. The following tables and graphs were the various size of mesh that we applied to the model.

Table 8: 200 N Force on the upper plate

Case #	**$\sigma_{Von\ Mises}$**	**nodes**	**Global**	**Local**
	MPa	*10^3*	*mm*	*mm*
1	88.603	451.726	*6*	3
2	84.612	456.715	*6*	2.75
3	88.299	458.723	*4*	2.5
4	86.236	465.328	*4*	2.25
5	86.88	475.650	*4*	2
6	94.918	496.807	*4*	1.75
7	90.159	520.870	*4*	1.5
8	98.834	559.596	*4*	1.25
9	97.581	635.037	*4*	1
10	100.491	702.533	*4*	0.9
11	100.509	769.009	*4*	0.8
12	100.014	879.488	*4*	0.7
13	100.368	1071.500	*4*	0.6

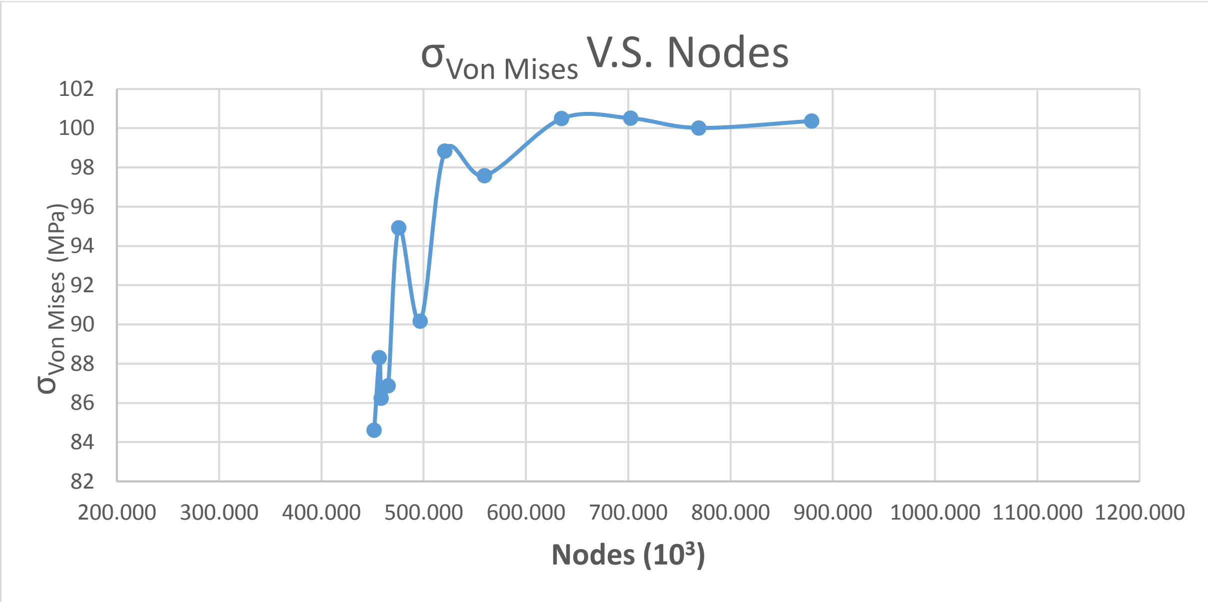


Figure 47: The value was converging to 100Mpa

Table 9: 200 N Force on the lower plate								
Case #	**$\sigma_{Von\ Mises}$**	**Displac.**	**Element**	**nodes**	**DOF**	**Global**	**Local**	**time**
	MPa	*mm*	*10^3*	*10^3*	*10^4*	*mm*	*mm*	*sec*
0	98.476	0.667				6	-	
1	80.73	0.675	127.926	167.593	63.022	6	0.675	12
2	84.75	0.667	103.028	167.688	50.296	6	0.667	13
3	83.29	0.667	103.038	167.77	50.320	6	0.667	13
4	88.78	0.667	103.132	167.862	50.348	6	0.667	11
5	87.35	0.667	103.104	167.875	50.373	6	0.667	11
6	92.45	0.668	103.125	167.905	50.351	6	0.668	11
7	89.95	0.667	103.131	167.926	50.352	6	0.667	12
8	90.23	0.668	103.226	168.052	50.405	6	0.668	12
9	90.04	0.667	103.266	168.156	50.436	6	0.667	12

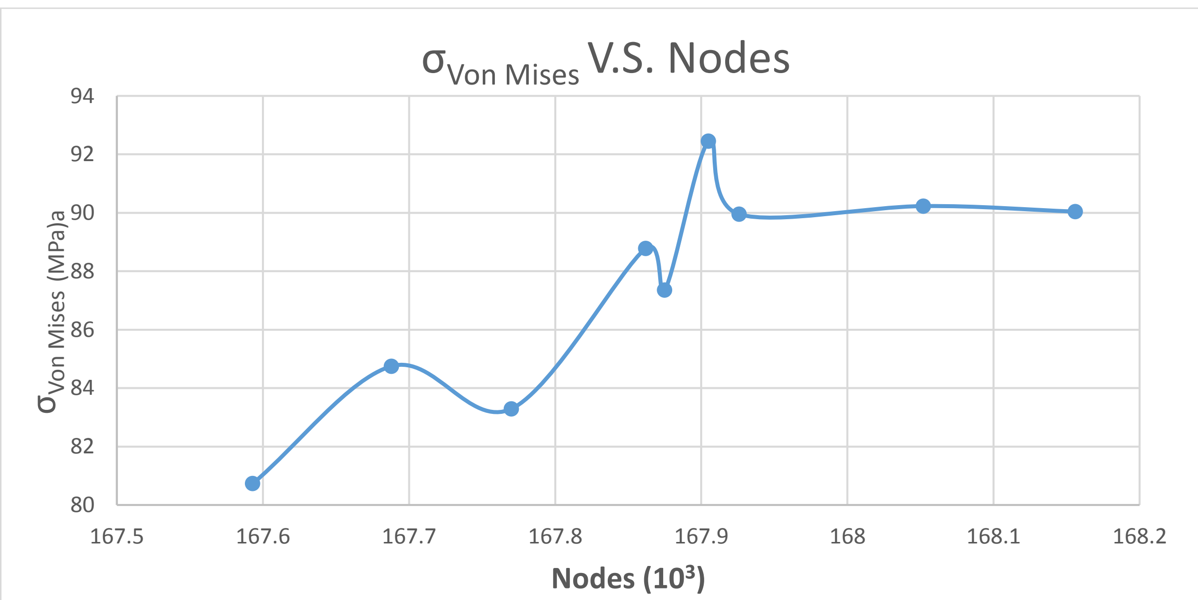


Figure 48: The value was converging to 90MPa

Table 10: Simulation result comparison							
		One point 200 N force in the middle of plate		Uniform 200 N force on the plate surface		Uniform 200 N force concentrated on the one side of the plate	
		$\sigma_{\text{Von Mises}}$	Displac.	$\sigma_{\text{Von Mises}}$	Displac.	$\sigma_{\text{Von Mises}}$	Displac.
		MPa	*mm*	*MPa*	*mm*	*MPa*	*mm*
200 N Force on the upper plate		116.9	1.579	113.8	1.535	100.368	1.678
200 N Force on the lower plate		104.9	1.737	100.45	1.675	90.04	1.378
100 N Force on both lower and upper plate		61.23	0.8426	60.56	0.8351	-	-
AISI 1020 Yield Strength		294.74 MPa					

As mentioned before, due to the complexity of the structure and deficiency of knowledge, there was no analytical solution which could be used to compare with. The results would be compared to the ASIS 1020 yield strength by using our engineering judgment. The AISI 1020 yield strength was 297.74 MPa and was higher than simulation results of all examining loading situation.

However, there were few uncertainties lay behind these results. First of all, we didn't use the exact solid modeling. We add a fillet or r = 2 mm to predict the maximum stress. Also, we concluded that the reason for the maximum stress locating in the lower hole on the edge was because the geometry influence on stress calculation. We then ignored the stress in the hole and looked for the maximum stress outside of the hole. However, the maximum was still possibly located in the hole. Moreover, as mentioned in the boundary conditions section, the best way to simulate was the virtual wall. We use reference instead because we can't run the virtual wall simulation. These three factors might also influence the maximum stress value. Therefore, to ensure the correct engineering judgment, we wanted to apply a safety factor of 2.

$$F = 2$$

$$\sigma_{MAX.ALLOW} = \frac{\sigma_{YIELD}}{2} = \frac{294.74}{2} = 147.37\ MPa$$

The maximum allowable stress under the safety factor of 2 was still higher than the maximum stress value 116.9 of simulation. Therefore, we could conclude that the metal frame could be safely used when maximum loading was applied uniformly on both level, uniformly on one level, at a point in the middle, and at a spot on one end of the wood plate. However, the simulation results were different, but the difference was not large.

Note that this conclusion was valid only if the woods and the wall plug were not breaking, and the wall was strong enough during loading. As this result, the best method to examine the whole system was load testing on the real product.

Redesign/Improvement:

Based on the simulation results, there were few ways that we could think of to improve the safety of the whole system. According Prof. Benenson, the redesigns/ improvements of engineering designs follow two rules: First, all redesign/improvement should be addition of material instead of removal of material. Second, all redesigns/improvements should become redundancy if necessary. As a result of applying the two rules, we had the following solutions:

First, we needed to ensure the quality of the wood. Since the balsa was very light (density: 160 kg/m^3) and had a low elastic modulus (3000 MPa), even though we could not analyze the wood in SolidWorks, we assumed that balsa was not a good choice for the plate. We could replace balsa by oak. Oak had an elastic modulus of 11300 MPa, which was much stiffer than balsa. The density of oak was also greater (560 kg/m^3) than balsa. Moreover, the price for oak wood was also not expensive (but still more expensive than balsa). Therefore, the balsa could be replaced by oak wood.

Second, for the corner block of frame and the middle of wooden plate, we added diagonal support bracing as the Figure 49 shown. Also, for the bottom corner block, we would extend our frame post, so there could be enough space for the addition of diagonal support bracing. Moreover, the support bracing should be added in the back of frame, so the existence of support bracing would not interfere any items placed on wooden plate.

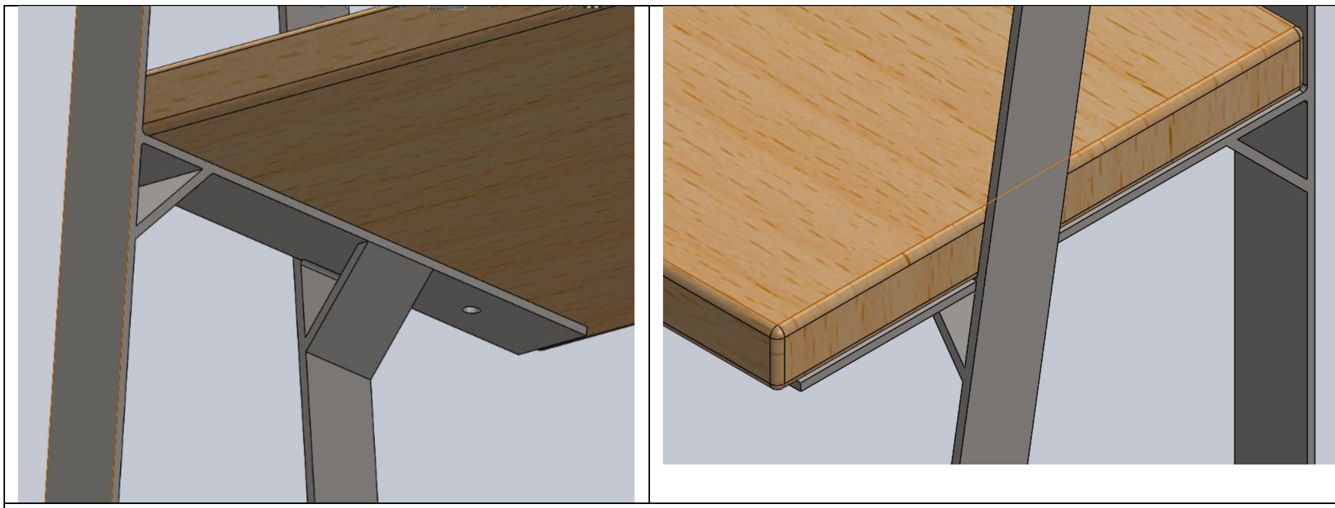


Figure 49: added diagonal support bracing

Last, for the frame hole, we added extra materials to the hole, so it would change from truncated oval shape to a circle shape with the same diameter as before as the Figure 50 shown.

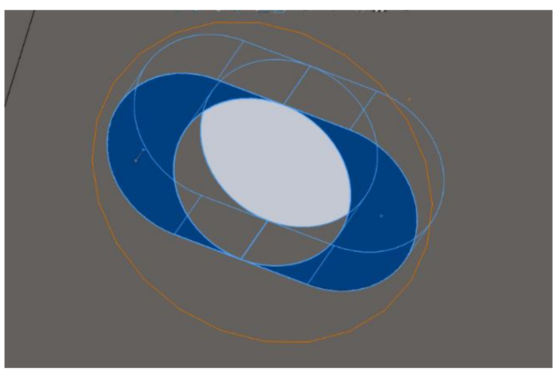


Figure 50: Remodeled frame hole

Conclusion:

On the aspect of technical results, the project could be considered successful. A set of practical boundary conditions had been set up. The special loading conditions of maximum load had been investigated using SolidWorks simulation and concluded to be safe. If the "Virtual Wall" boundary condition could ran, the simulation results could be more accurate. Even though the "safe" conclusion was granting under the assumption that the wood and wall were not breaking, the frame was examined to be safe. The safety of the whole system would still need to be tested in real loading situation. From the data table, we could see that different loading situations did make a difference in stress situation. It's very fortunate that the maximum loading (45 lbs) was safe for all the situations we showed. However, the difference between the stress situations were not as larger as we want. We originally wanted to use the big differences as a proof to show that the different loading situations could cause a very different result in stress and deformation and suggest detail information of using should be provided by the engineer. To further accomplish this goal, we might need to use another furniture, which had more complicated structures, or think about other special loading condition to investigate. However, based on the different values of maximum stress in each case, we still recommend that more detailed instruction on how to use the products such as the danger situation of usage, maximum loading value and its corresponding safe situation of applying and possible fail case could be provided into the instruction manual. These instructions could reduce the chance of product failing as well as causing injuries.

On the side of learning, this project was very interesting because not only we needed to keep thinking of modeling the real situation into simulation boundary conditions, but also it required us to examine out our own modeling. We learned a lot through deciding and choosing the boundary conditions. We learned to modify and evaluate the modeling by looking at the simulation results. We experienced that the initial boundary conditions were not correct and the deformation didn't look right so that we had to modify the boundary conditions and use the entire assembly, instead of the part, for simulation. When the boundary condition "virtual wall" could not run properly, we figured out to use "reference faces" and set displacement to 0. These experiences reminded us for how the structure engineer Jay developed the simulation method of reducing the stuffiness to examine the damaging building in earthquake when there was no appropriate software simulation applicable in the required reading "Mathematic Disposition". During the simulation, we learned how to examine the validity of the result. For example, we ignored the maximum stress in the lower hole on the frame to eliminate the geometry influence on calculation. When we were trying to conclude our data, we had to consider the factors which could influence the simulation results. The factors include the fillet, boundary condition of reference, and stress in the holes as we mentioned in data analyze section. In the improvement section, we also needed to consider the material properties of the wood; we learned that the price could be another factor in design. No one wants to purchase a $1,000 bookshelf which was made from ceramic matrix composites. These experiences were all related to making correct engineering judgments. Even though compared to commercial project, this product was still simple, we tasted about what was the real engineering project in person.

Reference:

1. http://ghentwoodproducts.com/price-lists-ghentwoodproducts/rough-cut-red-oak.php
2. https://www.fpl.fs.fed.us/documnts/fplgtr/fplgtr113/ch04.pdf