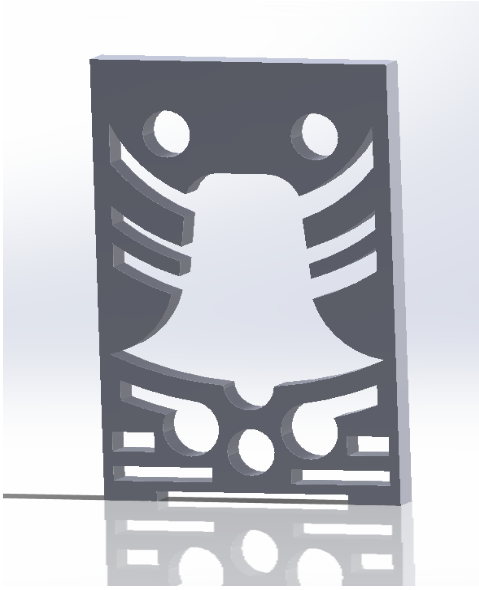
Dharmesh Patel (1214580198)



1) This is my Design, Designed in SOLIDWORKS, Initially I was considering material properties : Young modulus = 2500MPa (range is given 1800 to 3100 so I assumed this value) and Poison ratio = 0.35. By using linear triangular meshing with material strength approximately 55Mpa (for Plexiglass material). Theory used for Von Mises Stress and stress in Y normal.

2) After observing results from actual test in class, I assume that actual material property should be less than i took.so, for new test experiment Young modulus = 1800MPa , Poison ratio = 0.35

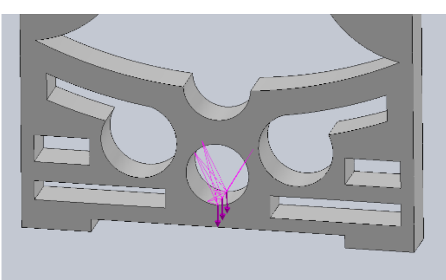
Load Conditions and Boundary conditions :

For support, I used fixed support. If this boundary condition will not satisfy in actual test it will reflect different result or error.

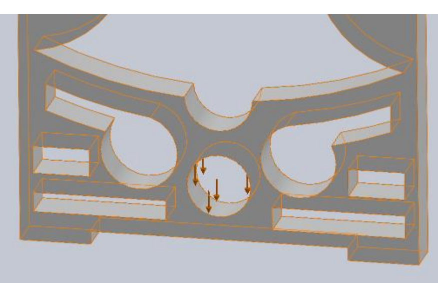
A) In first Experiment I applied load at 3 point on horizontal plane (bottom point of hole), but just on one line throughout the thickness. P1(z =0) = 75N, P1(z =3.175) = 75N,P1(z =6.35) = 75N (In downward direction) And fixed the both upper hole (which is 30 mm apart and vertically 60mm away from the bottom hole.

B) second time I applied load on whole surface of hole in downward direction with total magnitude P = 225N. Other boundary condition were same as above. and I got this.

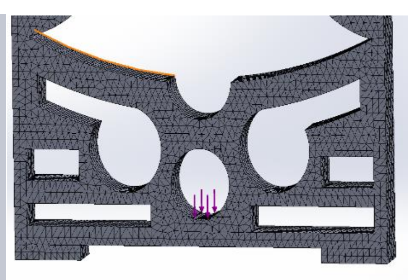
C) I made separate surface of 30 Degree at the bottom of hole. And applied load on whole that surface



1 Condition (A)



2 Condition(B)

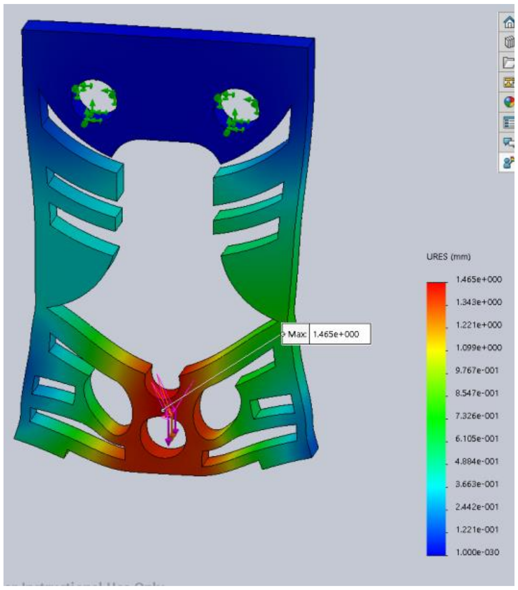


3Condition (C)

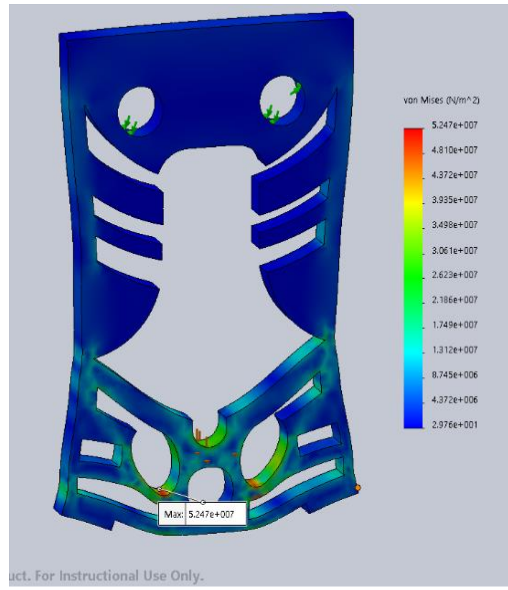
FEA – SOLIDWORKS Analysis

Following plots are the results of Analysis done on the given specimen. First, the analysis was done for Young's Modulus (1800 MPa) and Poisson's ratio 0.35 and then for 2500 MPa and 0.35 respectively.

Results for E = 1800 MPa and Poisson's ratio = 0.35

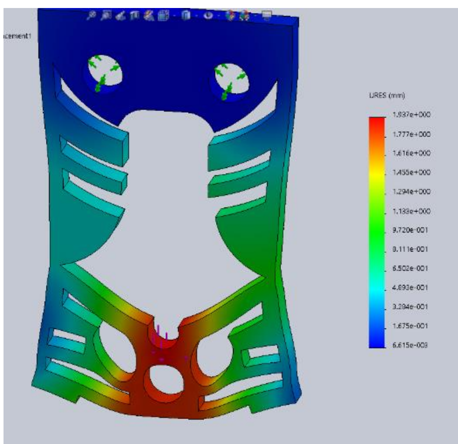


Displacement for condition (a)



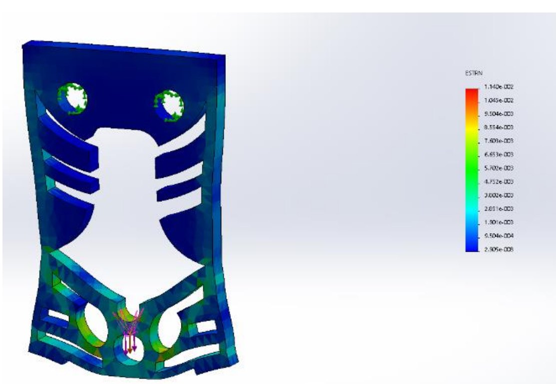
Von Mises for condition (B)

Results for E = 2500 MPa and Poisson's ratio = 0.35

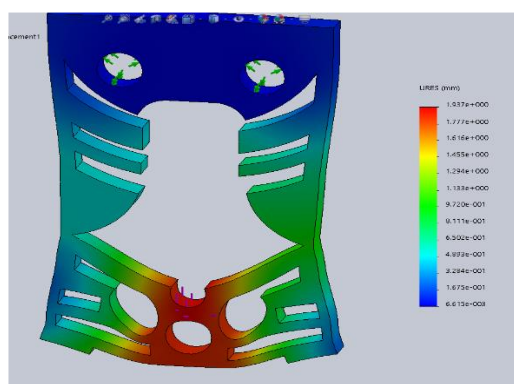


Displacement for Con.(B)

Displacement for con.(A)



Strain for Con. (A)



Principal Stress for con.(B)

Mesh Refinement :

Expected Result for Young modulus E = 2500MPa

Load (N)	E N/mm^2	Condition	Mesh (mm)	Displacement (mm)	Stress von N/mm^2	Principal stress N/mm^2	strain (10^-2) N/mm^2	Number of Nodes	Number of Element
225	2500	(A)	2.6575	0.8995	4.128	4.371	1.14	14534	7954
			2	0.9242	4.638	5.229	1.24	28091	16150
			1.5	0.9416	4.898	5.732	1.40	64412	39671
			1	0.9538	5.377	6.399	1.66	187499	121997
225	2500	(C)	2	0.9242	4.638	5.229	1.24	28091	16150
			1.5	0.9416	4.898	5.732	1.40	64412	39671
			1	0.9538	5.377	6.399	1.66	187499	121997

Expected Result for Half Load (112.5N) at Young modulus E = 2500MPa

Load (N)	E N/mm^2	Condition	Mesh (mm)	Displacement (mm)	Stress von N/mm^2	Principal stress N/mm^2	strain (10^-2) N/mm^2	Number of Nodes	Number of Element
112.5	2500	C	2	0.4621	2.319	2.614	6.198	28091	16150
			1.5	0.4708	2.449	2.866	7.044	64412	39671
			1	0.4769	2.688	3.199	8.309	187499	121997
			0.5	0.4827	3.210	4.367	10.10	1306393	905452

Expected Result for Young modulus E = 1800MPa

Load (N)	E N/mm^2	Condition	Mesh (mm)	Displacement (mm)	Stress von N/mm^2	Principal stress N/mm^2	strain (10^-2) N/mm^2	Number of Nodes	Number of Element
240	1800	A	2.65765	1.573	5.168	5.528	1.794	14534	7954
			2	1.620	5.595	6.313	2.238	28091	16150
			1.5	1.655	5.979	2.324	2.324	64412	39671
			1	1.678	6.065	7.073	2.697	187499	121997
			0.5	1.701	7.188	9.556	3.626	1306393	905452

Expected Result at Half Load for Young modulus E = 1800MPa

Load (N)	E N/mm^2	Condition	Mesh (mm)	Displacement (mm)	Stress von N/mm^2	Principal stress N/mm^2	strain (10^-2) N/mm^2	Number of Nodes	Number of Element
120	1800	A	2.65765	0.7863	2.584	2.764	0.8969	14534	7954
			2	0.8099	2.798	3.157	1.119	28091	16150
			1.5	0.827	2.990	3.259	1.162	64412	39671
			1	0.839	3.032	3.537	1.349	187499	121997
			0.5	0.855	3.594	4.778	1.813	1306393	905452

Load	E	Condition	Mesh	Displacement	Stress Von	Principal Stress	Strain(10^-2)
111.5	1800	Line load	2.65765	1.160	3.892	5.150	1.962

From table.1 and 2, we can see the stress are not changing for different application of load (Line load on bottom surface of hole and 30degree surface).

Observation :

So, it is depicted that for young modulus E = 1800 MPa my design will not work. it exceed the tensile strength , von Mises and Y normal stress at the sharpen edge. Stress concentration at sharp edge is very high and that is the main reason for failure of this design. There was 51.50 MPa, which is nearly 55.38 MPa limit for Principal stress. So, this error may occur due to my application load or due to lack of software performance (of my Computer).

Meshing : I used linear triangular elements with different sizes.

We can find out results are very sensitive to the element size, that's the biggest reason for our wrong chosen of design. At the time of design selection I only tried for 1mm element length. but the coarse mesh probably will not very well converged, this will lead the solution is inaccurate. And the distributed load over top hole have a very small influence for the solution. (Line concentrated load and distributed load on whole surface). The error is about 2.5%.I applied thin linear concentrated load because i assumed that string which hold the load is very small compare to hole.

Maximum Load for Design Failure :

My design already failure at 112.5 N in actual test. so by considering my properties, E = 2500MPa

Maximum load for failure is = 235 N

Design Optimization : in my design so many sharped edges and thin face are there. I used fillets at those edges which reduced the stress concentration at that edges and reduce the deflection. But, my design was tested already so that I can't change my design geometry. But I tried it see what will happen if I modified it. I didn't get desired deflection. Breaking Load for this design is

Load(N)	E (N/mm^2)	Mesh(mm)	Displacement (mm)	Stress Von *10^7 (N/mm2)	Principal Stress *10^7 (N/mm2)	Strain (10^-2)
150	1800	2.65765	0.6142	2.389	2.781	0.0965
		1	0.0391	2.556	3.084	1.099
		0.5	0.6422	2.726	3.108	1.272
270		1	0.9587	3.834	4.626	1.865

Modified Geometry

Breaking Load = (55.138 MPa / Max. Obtained Stress) * Applied Load

= 290 N

For 220N

Young modulus	Mesh	Nodes	Elements	Von Mises Stress max	Principal stress	Deformation	Strain Max	Error Percentage
2500	2	28091	16150	4.638	5.229	0.9242	1.24	
	1.5	64412	39671	4.898	5.732	0.9416	1.40	1.09
	1	187499	121997	5.377	6.399	0.9538	1.66	1.11
	0.5	1306393	905452	6.420	8.734	0.9653	2.20	1.36
1800	2	28091	16150	5.595	6.022	1.620	2.238	
	1.5	64412	39671	5.979	6.324	1.655	2.324	1.05
	1	187499	121997	6.065	7.073	1.678	2.697	1.11
	0.5	1306393	905452	7.188	9.556	1.701	3.626	1.35