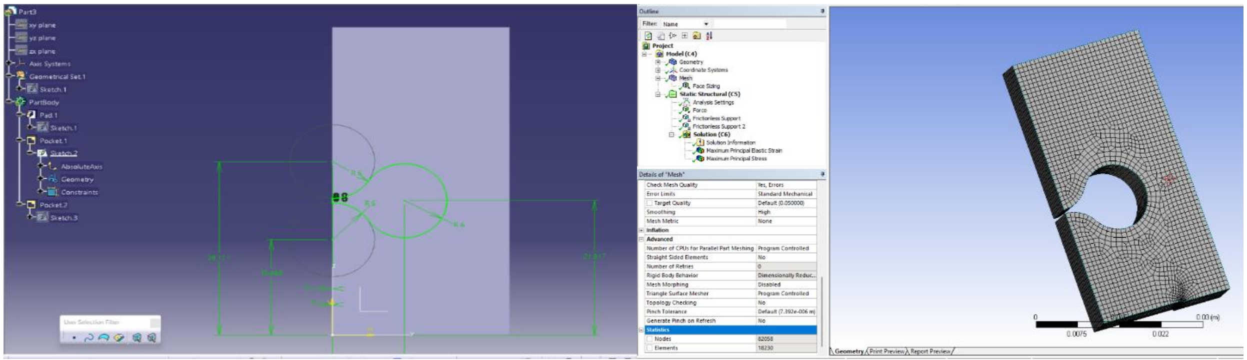
Name- Mangirish Kulkarni. ASU ID- 1223229852.

1. <u>Drawing of chosen design-</u>



2. <u>Materials used-</u>

PMMA- Plexiglass (Youngs Modulus-1800 MPa, Poisson Ratio- 0.35).

3. <u>Description of Boundary conditions applied</u>

Object modelled as 3D Plane stress problem and is simplified using symmetry. Half force applied at the top edge due to symmetry simplification.

4. <u>Description of Discretization-</u>

Element type- Quad Dominant

Mesh size- 0.5 mm

No of elements in Quarter model-18230 (Fine mesh)

No of nodes- 82058 (Fine mesh)

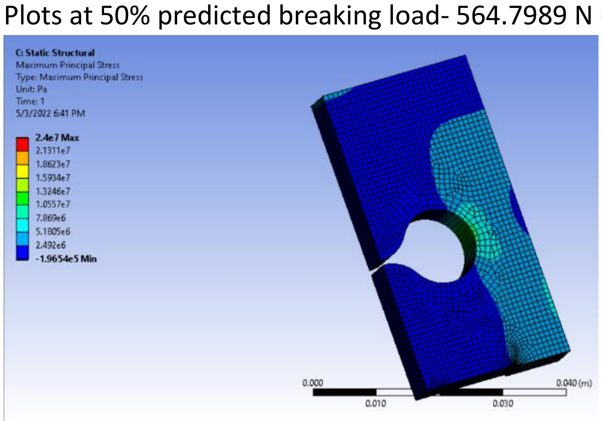
Mesh Metric-

Average element quality- 0.98688, Aspect ratio- 1.5867, Skewness- 8.39*e-002.

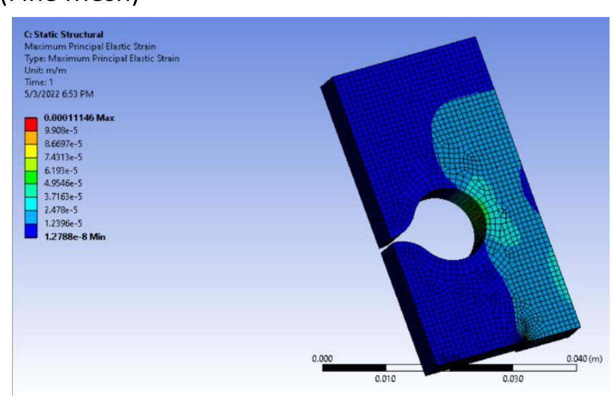
Average element quality maintained by ensuring curvature and proximity, edge sizing, smooth transition, span angle refined for best results.

5. <u>Contour Plot-</u>

Plots at 50% predicted breaking load- 564.7989 N (Fine Mesh)



Plot for Maximum Principal Stress



Plot for Maximum Principal Strain

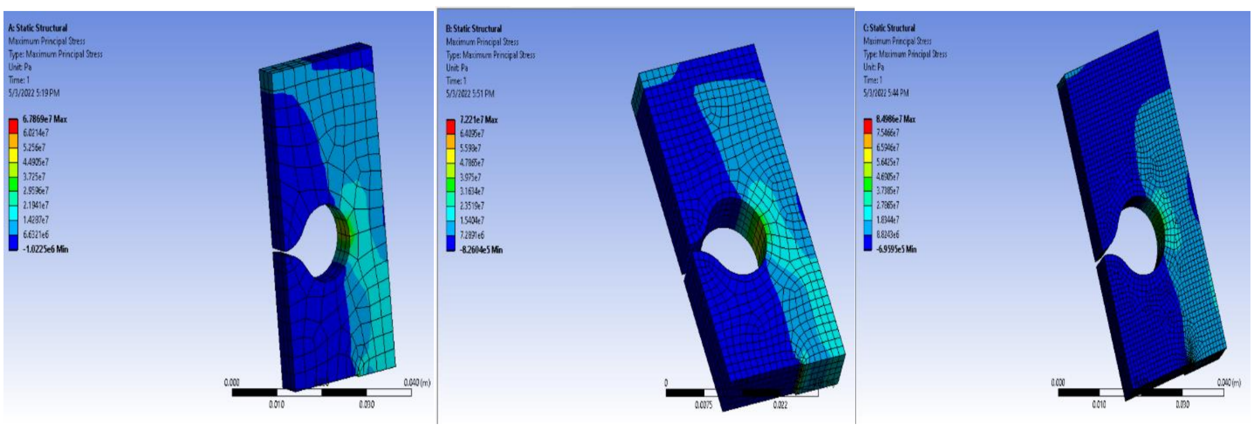
6. Prediction force for breaking-

The predicted force where model breaks is **1129.59 N**. Here this value is estimated using Youngs modulus 1800 MPa and Poisson's Ratio of 0.35 which gives us tensile strength of 48 MPa which will be our failure criteria. The value of stress achieved at this load is 48.88 MPa.

7. Verification of Solution-

A) **Mesh Refinement**-

Mesh Type	**Coarse**	**Medium**	**Fine**
No. of Nodes	2656	14212	82058
No. of elements	444	2810	18230
Max Principal Stress	**67.896 MPa**	**72.21 MPa**	**84.986 MPa**
Percent Change	**25.17 %**	**17.69 %**	**0**
Predicted Breaking Load-	**707.2448 N**	**664.7278 N**	**564.7989 N**

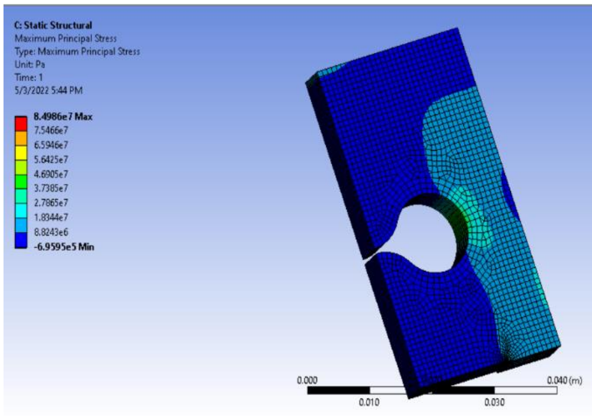


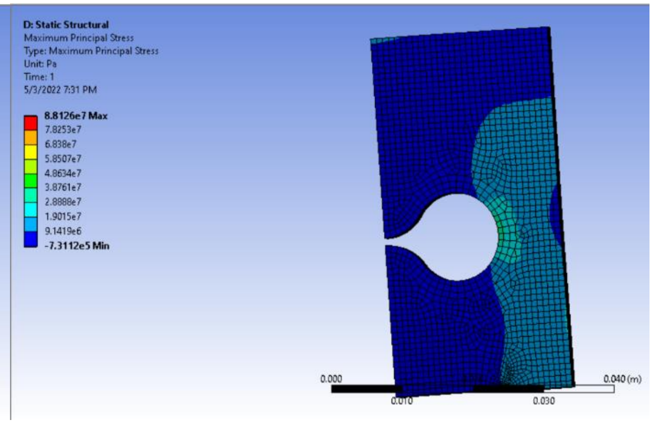
Max Principal Stress-Coarse mesh. Max Principal stress-Medium mesh. Max Principal Stress-Fine mesh.

As we can see from Mesh refinement study, The finer the mesh the more accurate the results obtained for stress concentration.

B) **Sensitivity to Boundary Conditions**-

Sensitivity Analysis	Fine Mesh (Parameters A)	Fine Mesh (Parameters B)
Youngs Modulus	1800 MPa	3100 MPa
Poisson Ratio	0.35	0.4
Max Principal Stress at Breaking load	**84.986 MPa**	**88.126 MPa**
Percent change	**3.69 %**	**0 %**





Max Principal Stress (Parameter A). Max Principal Stress (Parameter B).

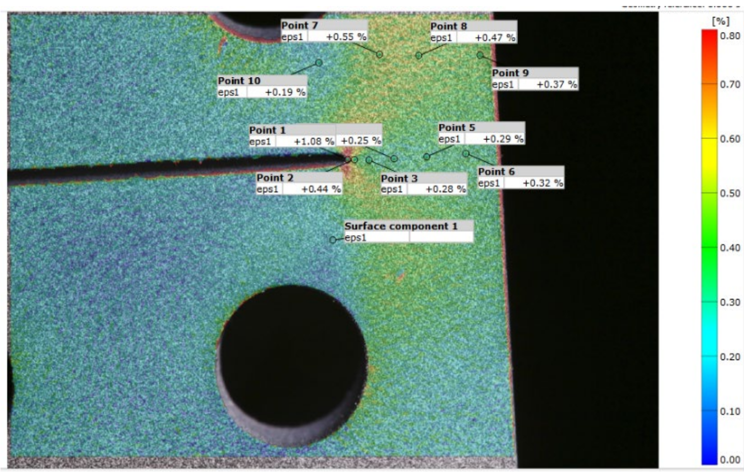
When we increase the Youngs modulus (Elasticity of material), The stress concentration value increases but not by much. As Youngs Modulus refers to relation between tension and deformation in elastic region. If the range of Tensile strength of material differs to initial value, the maximum stress value that the material can handle will also increase.

Comparison of predicted and measured values

Although our predicted breaking load for our model was 1100 N, it failed when a load of 1300 N was applied as found while testing the laser cut model. **But the location of predicted failure was correct, and the material did fail at the location we predicted it would**. To understand this difference, we again simulated our model by using the average of the bounds of material property as we believed this to be the main cause for the difference in results. Thus, the new values of material properties were:

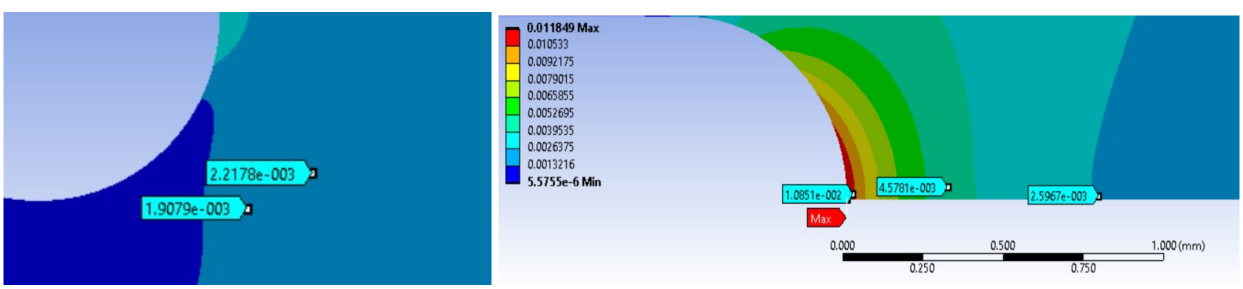
Poisson's ratio: 0.375
Youngs modulus: 2450 MPa
Tensile strength: 62 MPa

After analyzing the model using the above material properties, the value of stress at 1300 N comes out to be 58 MPa post 0.1% mesh convergence study. We shall now compare the plots for strain with that we got from simulation, for both 1100 N initial predicted model and updated model at 1300 N to the results we got from experiments using DIC.



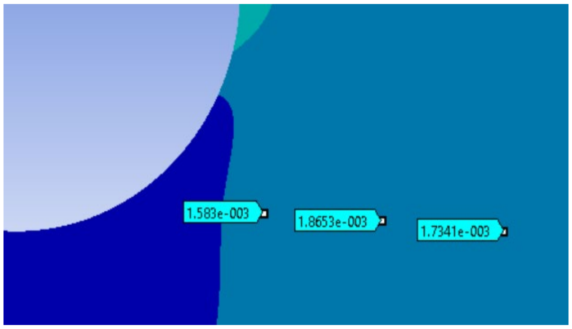
We notice that the pattern of the strain plot from DIC seems to closely resemble that from simulation with high values at location of failure, i.e., the end of the slot where the model fails at and some mid ranged values between the circle added to account for stress concentration and the slot. We shall now compare and talk about the values below:

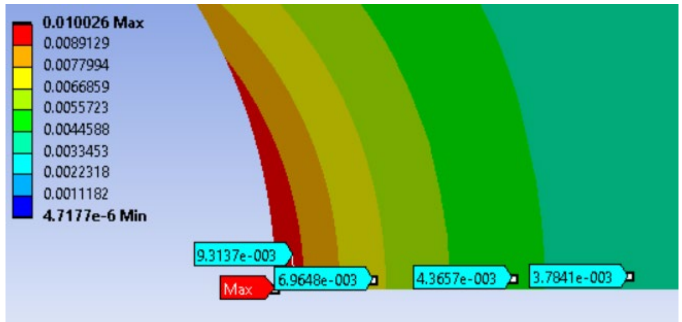
For 650 N breaking load (50%) (Updated model):



On comparing the values from DIC with that from the updated model, we notice the values to match more closely, with an error of +/- 0.05%. This again highlights the model was better approximated this time. Some values do seem to exactly match which came as a surprise to us and these were noticed at areas where mesh was very fine and of high quality according to the mesh quality metrics.

For 550 N breaking load (50%) (Original model):





We notice similar values near the region of failure and a very high value at the point of failure which naturally isn't captured in the DIC as the model did not fail yet at that point. We also notice that the value isn't that high near the edges or the length of slot in the simulation compared to the DIC result which goes to justify the comment made by the professor that those values are wrongly captured by the camera. We observe a significant difference in values at some locations which could be since this model simulated at the lower bounds would give slightly different results.

Comparing the plots of experiments and simulation.

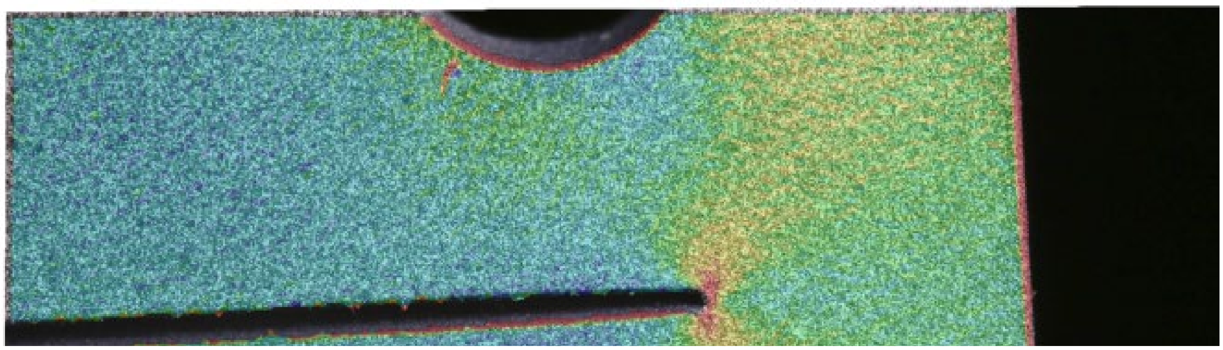


Image of strain captured from DIC at 50% breaking load

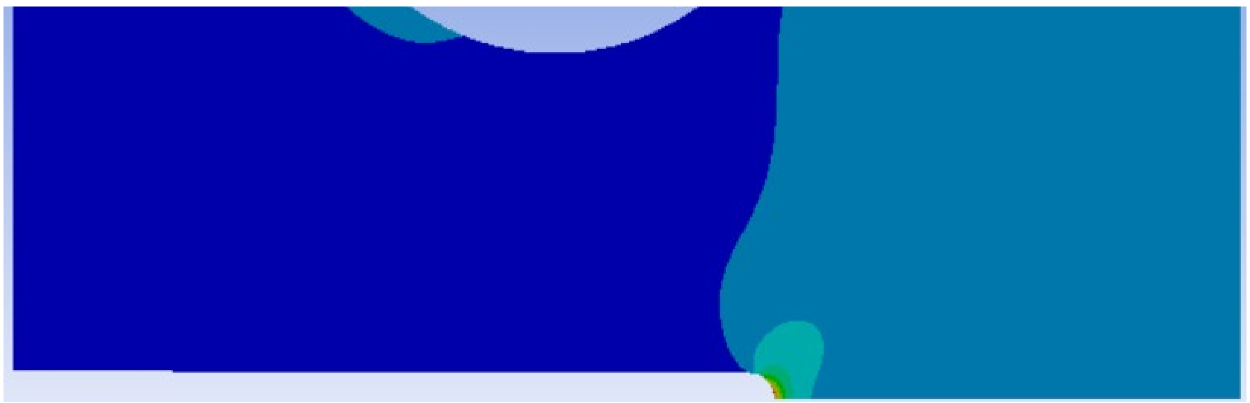


Image of strain captured from simulation at 50% breaking load

We notice from these two plots that the strain distribution seems very similar if you remove the error in strain as captured from the DIC near the edges and at the slot along its length.

Potential reasons for the difference in results could be as follows:

1. Since we designed the model to be safe considering the lower bounds of material properties, had we formed the material using average of the bounds, we get a stress of 58 MPa for a breaking load of 1300 N which is what we saw from the testing.
2. A major cause of differences in result could be the way loading is applied on the model. Since in our case, we did the analysis as 2D and applied the load on the top edge as a uniformly distributed load while in testing, it was gripped by a machine over a small area and then pulled.
3. PMMA is not an isotropic material, but the material created from this problem was assumed to be isotropic in nature due to lack of information regarding anisotropy of PMMA.
4. Inconsistency in dimensioning of manufactured model due to inaccuracy of laser cutting machine due to precision and repeatability error because of mass manufacturing of models.
5. The difference in strain values could also be possible due to problems because of focusing for DIC values as mentioned by the professor in his announcement.

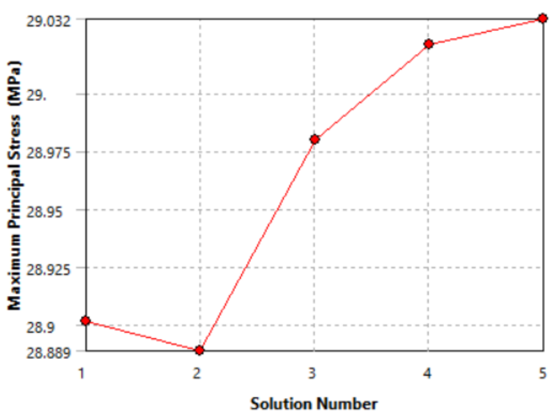
Mesh metrics for updated model:

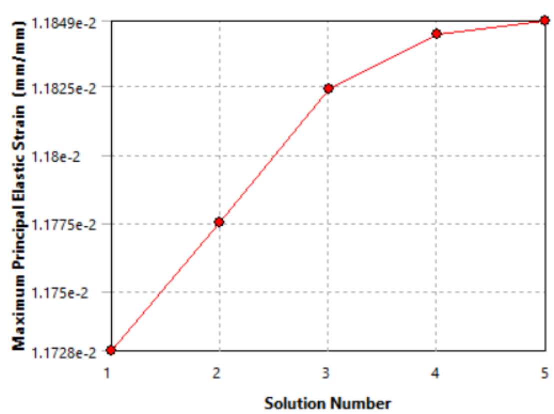
Quality	
Check Mesh Qua...	Yes, Errors
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Element Quality
Min	0.73582
Max	1.
Average	0.99366
Standard Devi...	2.1384e-002

Mesh controls was kept like one before to ensure enough elements to capture stress along the 0.5 mm radius of slot and circle.

Statistics	
Nodes	14607
Elements	4747

<u>Convergence plots for stress and strain to ensure results are not sensitive to mesh size</u>



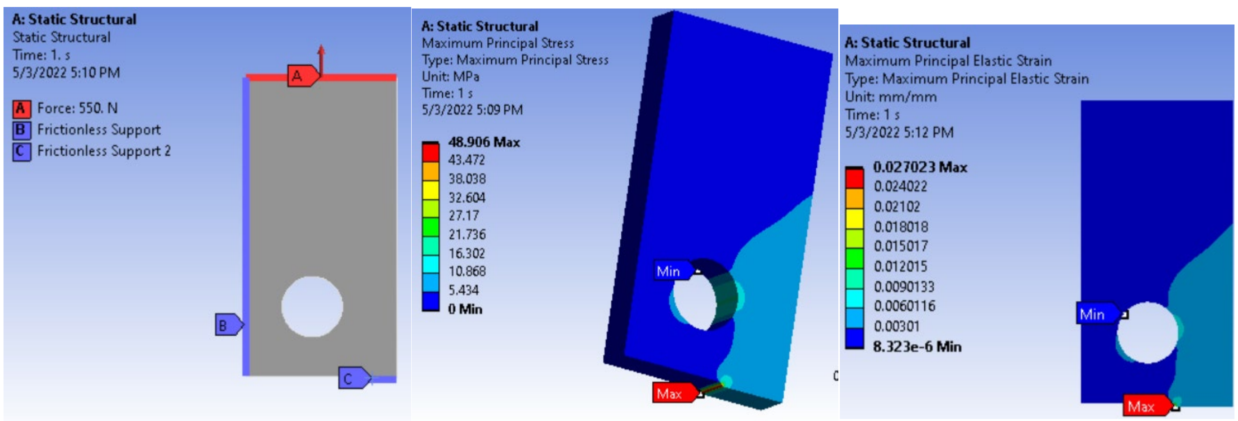


	Maximum Principal Stress (MPa)	Change (%)	Nodes	Elements
1	28.902		14607	4747
2	28.889	-4.5457e-002	14968	4864
3	28.98	0.31488	16786	5460
4	29.021	0.14269	20990	6842
5	29.032	3.7316e-002	25126	8198

	Maximum Principal Elastic Strain (mm/mm)	Change (%)	Nodes	Elements
1	1.1728e-002		14607	4747
2	1.1775e-002	0.40521	14968	4864
3	1.1824e-002	0.40969	16786	5460
4	1.1844e-002	0.17247	20990	6842
5	1.1849e-002	4.6445e-002	25126	8198

Analysis type and boundary conditions:

The analysis was performed in **ANSYS** in **2-D with plane-stress condition** since the thickness is by-far the smallest dimension and the force is applied on the bulk of the top face. **A distributed load in the y-direction, F (tensile), was applied to the top and bottom of the model.** Moreover, the ANSYS Symmetry Tool was used to model ¼ of the device since the device is symmetric with respect to the **ZX and YZ planes**. The only modification necessary to simulate the ¼ symmetric model is halving **the distributed load on the top edge ($F/2$).** No boundary conditions at the bottom of the symmetric model are needed since the ANSYS tool was used and it applies a <u>frictionless support by default along the planes it cuts.</u> The mesh generated were ensured to be mostly quadrilateral elements by using mesh control to set it to **Quad dominant elements of higher order**. To account for the small curvature of the slot, mesh sizing was used to ensure **5 elements across the half of the slot**, and another **mesh sizing to ensure 60 elements (A standard accepted thumb rule for meshing curved surfaces) across the circumference of the circle feature added.**



Parametric Study was used to optimize the design. The design points obtained apart from the chosen one is as follows:

Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - Height	P2 - Radius	P7 - X_dist	P6 - Maximum Principal Stress Maximum	Ret...	Retained Data
2	Units	mm	mm	mm	MPa		
3	DP 0 (Current)	12	10	11	49.131	☑	✓
4	DP 5	11	10	15	62.039	☐	
5	DP 6	11.5	10	15	62.375	☐	
6	DP 7	12	10	15	63.246	☐	
7	DP 8	12.5	10	15	64.787	☐	
8	DP 9	13	10	15	64.926	☐	
9	DP 18	12	10	15	63.246	☐	
10	DP 19	12	10	10	45.283	☐	
11	DP 20	12	10	11	44.542	☐	
12	DP 21	12	10	12	44.594	☐	
13	DP 22	12	10	13	45.611	☐	
14	DP 23	12	10	14	48.514	☐	
15	DP 24	12	10	15	52.705	☐	
16	DP 25	12	10	16	58.099	☐	
17	DP 26	12	10	17	66.394	☐	
18	DP 27	12	10	18	75.494	☐	
19	DP 28	12	10	19	88.724	☐	
20	DP 29	12	10	11	48.996	☐	
21	DP 30	12	10	11	53.45	☐	
22	DP 31	12	10	11	57.905	☐	

More points were taken into consideration but omitted to show due to lack of space. The pictures shown above are the simulation results for only the final chosen design model. To summarize the values, post convergence (0.1% criteria was chosen) is:

For 50% load: Stress = 24.44 MPa
For 100% load: Stress = 49.131 MPa

Team Peer evaluation:

The team score is graded out of 4 according to the rubric given in the project report guidelines.

	Score (Out of 4)						
		Report	Description of analysis	Boundary Cond.	Convergence	Geometry modification	Effort
Name	* Rohit Iyer	4	4	4	4	4	100%
	Ujjawal Jha	4	4	4	4	4	100%
	Harsh Verma	4	4	4	4	4	100%
	Maulik Kamdar	4	4	4	4	4	100%
	Onkar chavan	4	4	4	4	4	100%
	Mangrish Kulkarni	4	4	4	4	4	100%
	Kaustubh Nalawade	4	4	4	4	4	100%

* Selection of design:

- The Asterix indicates the design selected. The reason due to which this design was chosen was due to the high accuracy of meshing of structure where it was ensured that at all curved regions, mesh was fine enough to capture both the geometry and stress/strain values accurately which was verified again by mesh refinement study.
- For this design, it was properly analyzed using parametric study while tuning all design parameters to ensure that both the location (x and y coordinates) and the radius are optimized for the value of stress.
- This design also ensured a high value of load while ensuring the value of stress is less than 48 MPa, which is good since it provides a good model when using lower bound values for parameters.

The details about this design’s simulations are mentioned at the end of this report.