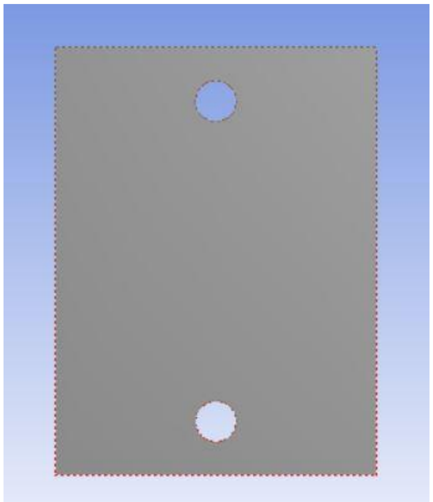
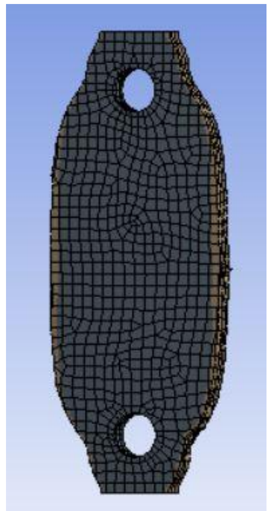
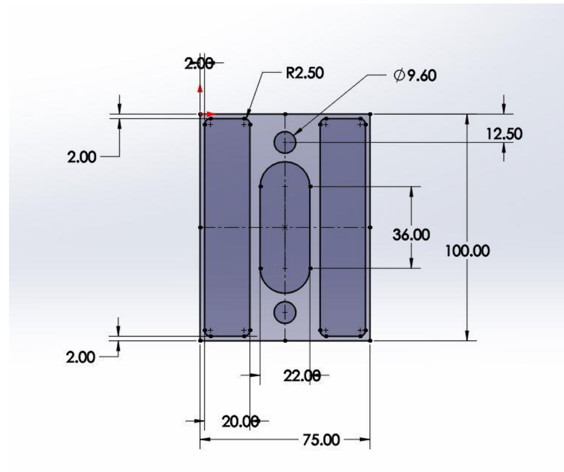
# INDIVIDUAL DESIGN ANALYSIS (PROJECT-3) – VARUN AGRAWAL (1215318065)








The figure on the left is the basic design of the plate without any optimization. The figure in the middle shows the result of topology optimization using Finite Element Tool 'ANSYS' which gave the idea of selecting a design showed in the right figure. A constant 2mm gap from the edges of the plate is taken for both the rectangles to work on different boundary conditions. The radius of the fillet (2.5mm) is constant at the corners of the rectangles. The width of the rectangle (w) is the design parameter. The slot at the center of the plate (with length in the direction of load application) is selected to decrease the stiffness of the plate. Its length (l) and width (b) are the design parameters. The distance of the hole from the top edge (h) (or bottom hole from the bottom edge) is also the design parameter.

## PARAMETER STUDY TO OPTIMIZE THE PLATE

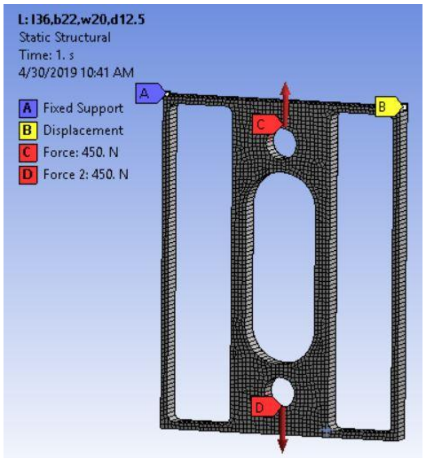
**Material Properties of Plexiglass** – Modulus of Elasticity = 1800 MPa, Poisson's ratio = 0.40, Density = $1180 kg/m^3$, Ultimate Tensile Strength = 80 MP. These properties are selected to get the minimum stiffness of the plate.
**Analysis type** – 2D (Plane Stress)
**Mesh size** = 1.25mm, QUAD Dominant (TRIA elements formed automatically due to the presence of curves in the plate), Element Order – Quadratic
**Boundary Conditions – 1.** Fixed support at the top left corner of the plate. **2.** The displacement boundary condition at the top right corner of the plate- Free in the X direction and 0 in the Y direction
**Force Applied on the plate –** Force is applied on the edge as the geometry is 2D surface. Downward ramped force (-ve Y direction) on the lower semicircle of the bottom hole and the equivalent and opposite ramped force in the upward direction (+ve Y direction) on the upper semicircle of the top hole.

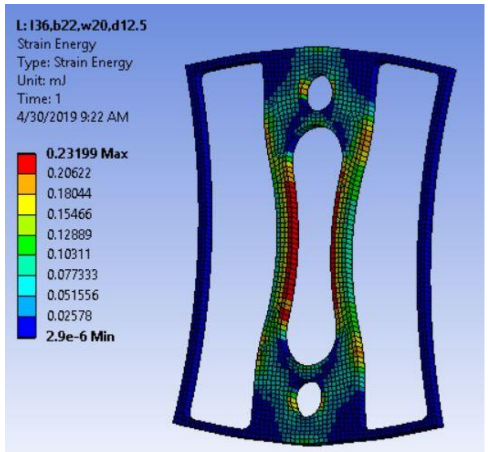
| Length of slot (l) (mm) | Width of slot (b) (mm) | Rectangle width (w) (mm) | Distance of hole from top edge (h) (mm) | Load (N) | Maximum Principal Stress (MPa) | Maximum Deformation (Xp) (mm) | Plate Stiffness (N/mm) | Mass of plate (gm) |
|---|---|---|---|---|---|---|---|---|
| 30 | 14 | 15 | 12.5 | 900 | 35.046 | 0.44406 | 2026.75314 | 29.313 |
| 30 | 14 | 15 | 14 | 900 | 33.673 | 0.4202 | 2141.83722 | 29.313 |
| 30 | 14 | 20 | 14 | 900 | 35.988 | 0.50279 | 1790.01173 | 22.12 |
| 30 | 14 | 20 | 12.5 | 900 | 37.884 | 0.53221 | 1691.0618 | 22.12 |
| 36 | 14 | 20 | 12.5 | 900 | 36.587 | 0.54262 | 1658.61929 | 21.49 |
| 36 | 22 | 20 | 12.5 | 900 | 28.268 | 0.85171 | 1056.6977 | 17.638 |

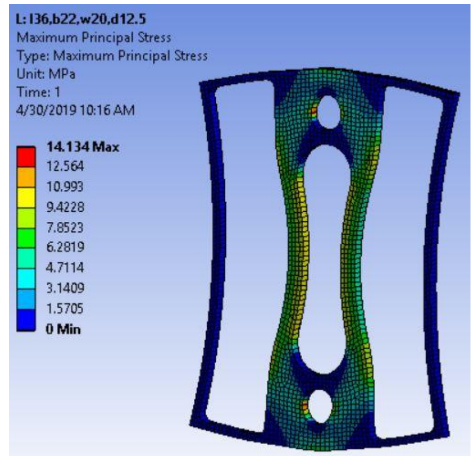
The highlighted row shows the parameters of the optimized design.

## ANALYSIS AT BREAKING LOAD IN OPTIMIZED DESIGN

It is given that the cord can withstand load up to 900 N before rupture. After in-class testing of the plate, it was observed that the cord remained safe when the mass of 4.54 kg was dropped from the maximum height. But, for the mass of 11.34 kg, plate remained safe and the cord ruptured. Considering the reduction of 50% in the load withstand capacity of the cord due to knots, the breaking load is 0.5*900 N = 450 N. Fig. in the middle shows Maximum Strain Energy in the plate at the breaking load.





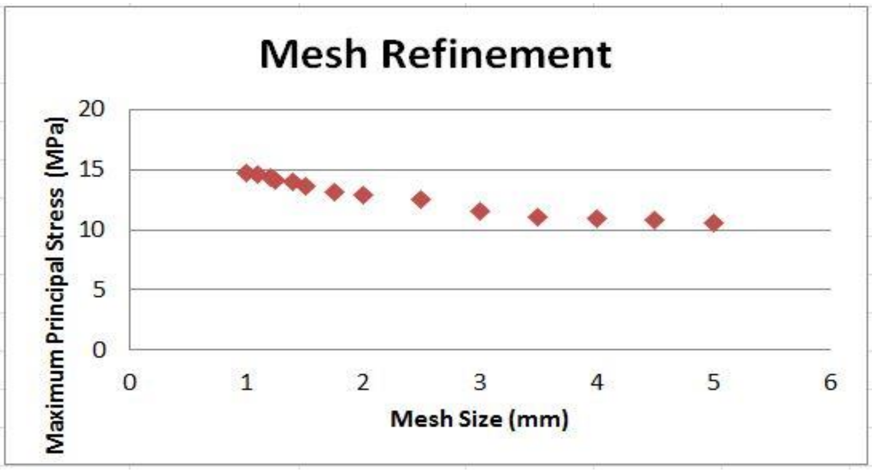



Total energy from ANSYS = 89.629 mJ
Deformation in plate = 0.42586 mm
Maximum Strain energy = $\frac{1}{2}K_p {x_p}^2 = \frac{1}{2} * \frac{450}{0.42586} * 0.42586^2$ = 95.8185 mJ
Maximum strain energy from the analytical solution and the one obtained from the ANSYS is almost same.

## MESH CONVERGENCE STUDY

To verify that the maximum principal stress is not sensitive to the element size, analyzing the optimal design at a load of 450 N for the different mesh sizes with quadratic element order.

| Mesh Size (mm) | Maximum Principal Stress (MPa) |
|---|---|
| 1 | 14.751 |
| 1.1 | 14.59 |
| 1.25 | 14.134 |
| 1.4 | 14.019 |
| 1.5 | 13.637 |
| 1.75 | 13.103 |
| 2 | 12.881 |
| 3 | 11.546 |
| 4 | 10.954 |
| 5 | 10.505 |




The maximum principal stress converges approximately at the mesh size of 1.4 mm.

**BOUNDARY CONDITION SENSITIVITY –** Load applied on the optimized body = 900 N, mesh size = 1.25 mm, QUAD dominant quadratic meshing.

| Boundary and loading conditions | Maximum Principal Stress (MPa) |
|---|---|
| Fixed support at the top left corner of the plate; Displacement constraint at the top right corner- Free in X and 0 in Y direction; Downward ramped force on a lower semicircle of the bottom hole and equal upward force on an upper semicircle of the top hole. | 28.268 |
| Fixed support (upper semicircle of the top hole) and downward ramped force on a lower semicircle of the bottom hole. | 31.903 |

The above results conclude that the maximum principal stress is sensitive to the different boundary conditions because different boundary conditions give the different value of maximum principal stress.

The point 1 boundary condition is selected for the simulation because it complies with the real-life situation. The top left corner restricts the translation motion in all the directions and the top right corner restricts the motion in the Y-direction allowing it to translate in X-direction. This condition makes the entire geometry symmetric about Y-axis because there will be no deformation in the Y-direction. It will only deform in X-direction. Also, it does not give any rigid body motion.