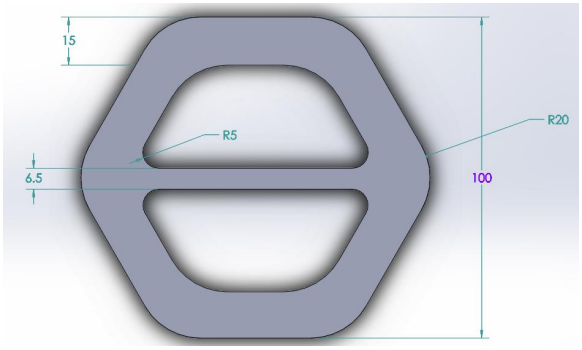
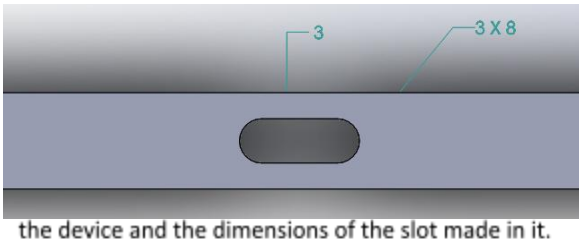
Individual Design Report: Gana Sai Kiran Avinash Raj Dwarampudi

MAE 404/503 Final Project

Arizona State University

Project Outline: The objective of the project is to design a device to demonstrate the stresses that are developed on application of compressive force with a vise within the gage section of the device. We design 2 models, with and without a stress concentration to see how the stress characteristics differ.





the device and the dimensions of the slot made in it.

The design parameters for the device are:

1. The device should not exceed 100mm.
2. Thickness of the device should be 6.35mm.

Solidworks is used to design the 2 device models. The first model is given a slot in the gage section to create a stress concentration. The models along with the dimensions are given in Fig 1 and 2:

Material Selection: The material used for the testing of this device is PMMA and the material properties:

Young's Modulus	1800 MPa
Poisson Ratio	0.35
Tensile Strength	50 MPa

Finite Element Analysis Parameters: Ansysis used to analyse the models for Maximum Principal Stresses. The designed models are imported into Static Structural Analysis and are given the loading in plane stress and boundary conditions for analysis after meshing the models. The parameters taken are:

Pressure Load	30 MPa on the top surface
Mesh Size	5mm
Mesh Type	3D Triangular
Boundary Conditions	u= 0 mm in all directions for bottom surface

Loading Conditions: The bottom support of the device is fixed and a pressure load is applied on the top surface to create a compressive force that travels from the top surface to the bottom. Then we can observe the tensile force produced in the gage section of the device. This is carried out for both the models with the same parameters and we can observe the stress concentrations produced in the device.

Mesh Convergence: To verify that the results are consistent with the parameters but not the element size, we make a Mesh Convergence study where we decrease the element size and compare the results.

Maximum Principal Stress: The Maximum Principal Stress observed in the devices are 138.39 MPa 252.18 MPa respectively. For the first model, the stresses are observed on the gage section but for the second model, the stresses are completely concentrated on the slot. They are given in Fig 5 and 6. The stress concentration at the slot for the second model can be clearly seen in Fig 4 below:

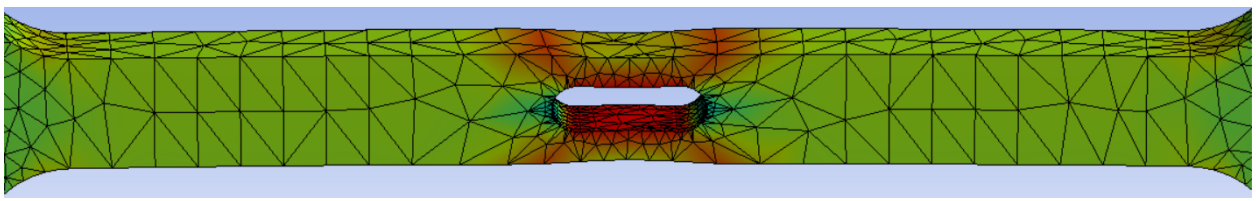


Fig 4: The Stress Concentrations at the slot under loading.

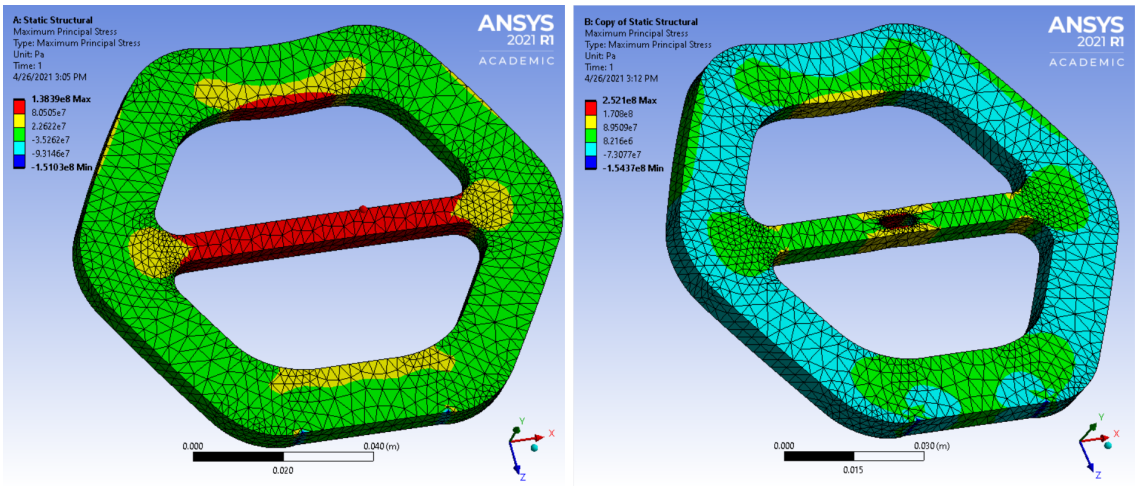


Fig 5 and 6: Maximum Principal Stress without and with stress concentration.

Mesh Convergence: Mesh Refinement is used for this process. The results obtained are 138.56 MPa and 252.74 MPa respectively where we have 0.1227% and 0.2216% variation from the previous analysis. Therefore the analysis is not sensitive to the element size. These results are given in Fig. 6 and 7 below:

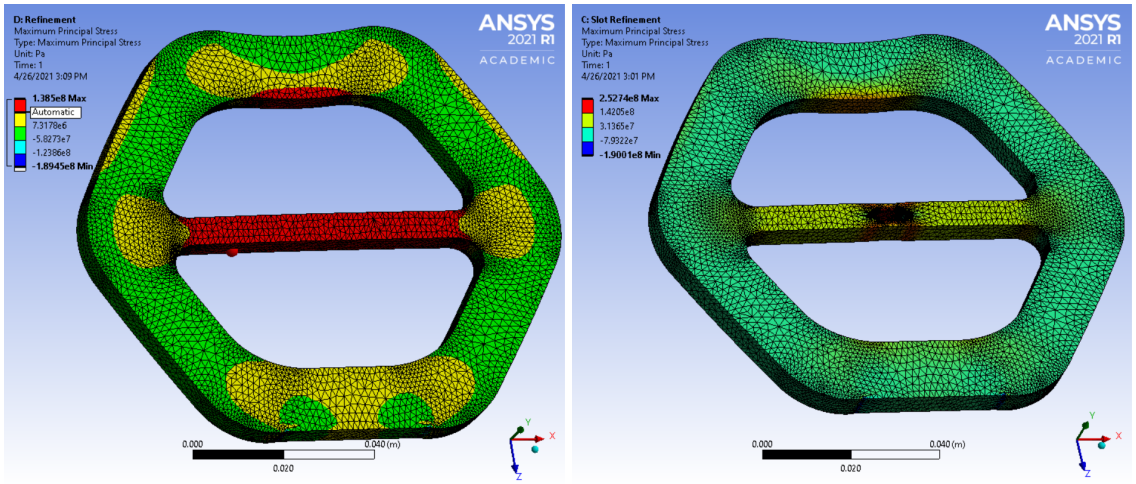


Fig 6 and 7: Mesh Refinement for the both models.

The observations from the above results is given in the table below:

	Mesh Size	Max. Principal Stress	Deformation
Without Stress Concentration	5 mm	138.39 MPa	6.39 mm
	1 mm	138.56 MPa	6.40 mm
With Stress Concentration (Slot)	5 mm	252.19 MPa	6.55 mm
	1 mm	252.74 MPa	6.57 mm

Optimization: After observing the analysis results, we can get a clear idea of where and what the problems are with the current designed model and we can then add changes to the model to get better results. This way we can create a design which is feasible for all conditions. The optimization of design using FEA is used in many industries because of its credibility.