

Final Project Report: Analysis of Modifications to a Notched Plate

MAE 598: Finite Elements in Engineering, Spring 2017

Group 4: Leighann Ngo, Kevin Espinoza, Adil Ansari, Michael Buller

Abstract

A notched Plexiglas plate experiences high stress concentrations at the peak of the notch, causing failure of the plate when under load. In this project, four modifications to the notched plate were analyzed using finite element methods (FEM). The modifications were designed to relieve the stress concentration to improve the structural integrity of the plate in an attempt to prolong the point of failure. The four designs were initially analyzed via FEM simulation software. The design that experienced the lowest maximum strain amongst the other designs was selected to be used in an experimental test. The results of the experimental test were then compared to the results of the simulation.

Introduction and Problem Statement

A notch in a structure causes stress to be concentrated at the notch where the discontinuity in material occurs. Due to an increased and concentrated stress at the sudden discontinuity, the probability of failure occurring due to crack propagation increases.

In this report, four unique plate designs were analyzed utilizing finite element methods (FEM). The four plates contained varying modifications to an originally notched plate in an effort to reduce the strain concentration generated at the notch and improve its structural integrity under a three-point bending test. Each modified plate was simulated in MATLAB or ANSYS to analyze the resulting mechanical properties under an applied load.

Based on the simulation analysis, a single design was selected to use in an actual physical experiment; the results would later be used to compare the differences between experimental and simulation results. To decrease variability in the process of selecting the optimal design, each model was characterized by the same material properties, load application, and boundary conditions. The optimal design was chosen based on the simulations' predicted maximum normal strain under load. The design with the lowest maximum strain value would have the lowest probability of failure amongst the other three designs.

Once the optimal design was selected, real world tests were performed to determine the accuracy of the model. The test utilized digital image correlation (DIC) software to generate strain data which was cross referenced and compared with the simulated data.

Member 1: Leighann Ngo

Introduction

In the analysis of this design, ANSYS Workbench was used to analyze two Plexiglas plates in a three-point bending test. One plate, the unmodified plate, was 127 mm long, 38.1 mm high with a vertical notch. The unmodified plate was analyzed to determine the location of the strain concentrations. The second plate, the modified plate, was the unmodified plate with additional holes made in an effort to reduce the maximum strain experienced by the plate by disrupting the strain concentrations occurring in the unmodified plate. Because the plates were planar and symmetric, the ANSYS analysis was done as a two-dimensional analysis with a single line of symmetry.

The project specifies the plate material to be Plexiglas. Due to the large range of Plexiglas material properties available, the average between the largest and smallest values were used (1.8-1.3 GPa Young's Modulus, 0.35-0.4 Poisson's ratio). The following table shows the material properties used during this analysis.

Table 1. Material Properties

Young's Modulus	2.45 GPa
Poisson's Ratio	0.375

Boundary conditions and loads

The project problem describes the three-point bending test with the plate held in a fixture at opposite ends of the plate with an applied load at the top-center of the plate (between the two fixed ends). After reviewing the resulting deformation plots of the plate with fixed support and roller support in ANSYS, it became clear that the roller support more realistically mimicked how the plate would behave in the actual test fixture.

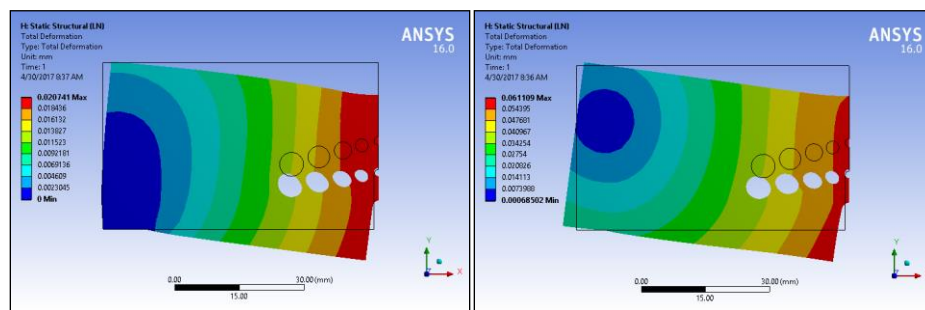


Figure 1. Supports: Fixed (left) & Roller (right)

Figure 1 shows the resulting total deformation using a fixed support (x-direction, y-direction, and moment reaction forces) and a roller support (y-direction reaction force) as well as an outline of the un-deformed model. The fixed support behaves as if the plate is glued or clamped at the supported area which does not accurately mimic the actual bending test. In the actual test, the plate should not experience displacement in the y-direction at the support point and should be allowed to rotate. The roller support more accurately modeled the actual bending test. Note, the roller support was applied in ANSYS as a “0” y-displacement support at a single point (9.5 mm from the edge of the plate).

The original project problem describes a point load of 70 lb-f being applied to the top of the plate. While the load theoretically is applied at a single point, the actual bending test would be performed with a load applied over a circular area with a 0.75 inch diameter due to the equipment used. Because this problem

was modeled as two-dimensional with a line of symmetry, the load was applied as a pressure of 2.59 MPa on a 0.375 inch line.

Mesh refinement and convergence

To insure the accuracy of the simulation, a fine mesh with adjusted sizing along curves and a high quality of smoothing was used. The elements were two-dimensional, mostly quadratic elements with the occasional triangular element. Note that quadratic elements would be the more conservative element type as it allows more bending (triangular elements are stiffer in bending, causing an under predicted result).

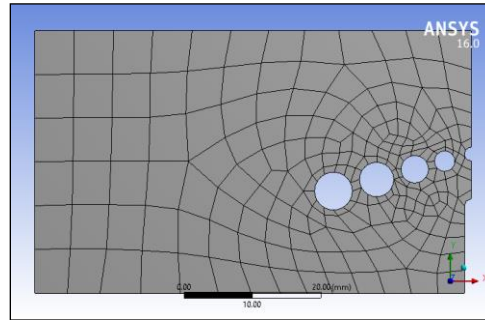


Figure 2. Modified Plate Mesh

A mesh refinement and convergence test was also performed to further insure all boundary conditions, loads, and mesh were appropriately specified. As a result, the analysis for the modified design quickly converged.

Table 2. Convergence Data of Modified Design

Solution No.	Normal Elastic Strain	Change (%)	Nodes	Elements
1	2.8298E-03		759	224
2	3.0908E-03	8.8165	1251	378
3	3.0828E-03	-0.26028	1830	563

Results

The resulting maximum normal plane strain along cross-sectional line “A-A” for the modified design was 0.0213 while the maximum normal plane strain for the unmodified design was 0.0190. The modified design experienced a larger maximum normal strain meaning it would likely fail at a lower applied load than the unmodified design.

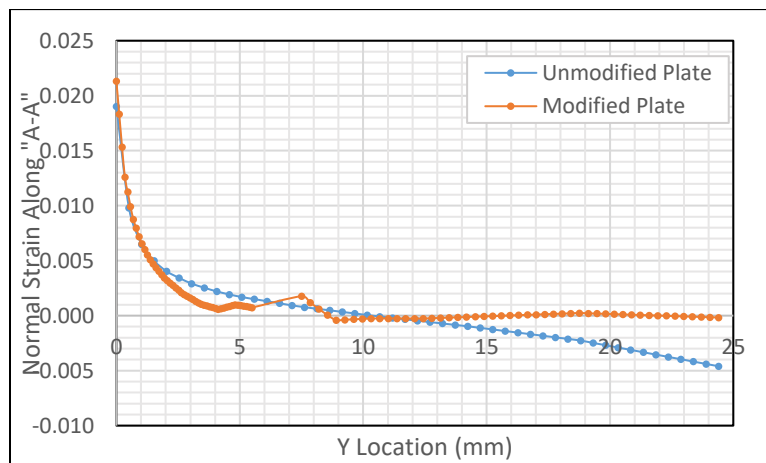


Figure 3. Resulting Normal Strain

Note that the “Y Location” in Figure 4 is the vertical location at which the strain was calculated, beginning at the notch made in the plate, in millimeters. The sudden discontinuity between 5 mm and 9 mm in the modified design is due to the hole made in the modified design between the notch and the edge of the plate. Also, the resulting strain of the modified design towards the edge of the plate where the load is applied should be negative because it is in compression under load, however round-off errors occurred in calculating the strain making it appear as though the strain is close to zero.

Member 2: Kevin Espinoza

The plate modification was designed with a unique shape in SolidWorks. For the design, the crumple zone design theory was used (the theory behind the design, Figure 1, was influenced by analyzing the crumple zones of cars). The intentions of the arrows was to direct the stress away from the notch and prevent it from opening. In cars, crumple zones are designed to absorb and redistribute the force of a collision, so when a force was applied on the top by the crumple zone would absorb and redirect the stress being created by the force on the plate.

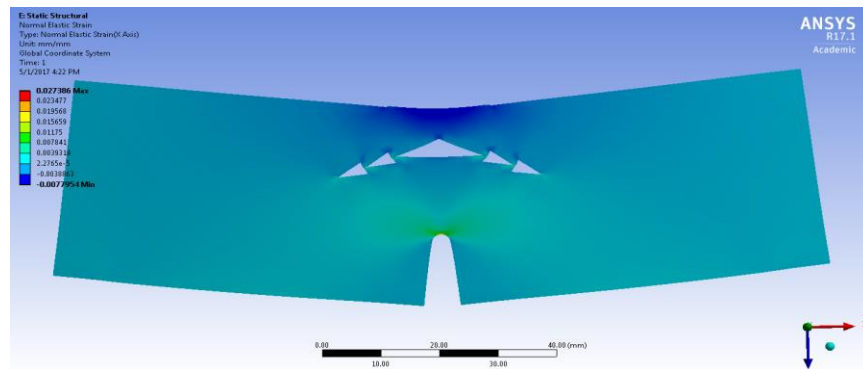


Figure 1: ANSYS Model Image of Normal Elastic Strain

To help with the boundary conditions, indentations were made on the bottom part of the plate for the length of 9.5 mm from the edge on both sides, which is for the fixture. In addition, a 19 mm ($\frac{3}{4}$ inch) extrusion was made to the center of the plate, with an infinitesimal height for the applied load. After the part was designed, it was extruded and saved to a STEP file.

A static structural module was inserted into ANSYS, and then the engineering data added a Plexiglas material. The Plexiglas material was adjusted to have a Young's Modulus of 2.75 GPa, and a Poisson's ratio of 0.375. After that the geometry was imported from SolidWorks.

Afterwards in design modeler a force was added to the top of the plate where it will act as a distributed load; the force applied is -300 N. The boundary conditions are also added. The boundary condition added at the bottom is a fixed support and it acts as a pin support. A fixed support was not used, as it would create artificial moments in the horizontal direction. Furthermore the strain used was maximum principal strain.

For the mesh, three difference things were done to improve the basic mesh given. One, the relevance center went from being coarse to fine. Second, the span angle was also changed from being a coarse mesh to a fine mesh. Lastly, the relevance went from 0 to 100, this created a hexahedral serendipity mesh. This mesh has 20 nodes per element and 3 degrees of freedom for every node. The results obtained from ANSYS were nonlinear as opposed to linear, but it is expected that both cases would give similar results, as the deformation is infinitesimal.

Two paths were created because of the discontinuity along the half symmetry line and they plotted principal stresses. Elastic normal stress were computed along the paths of the shape. Maximum strain is 0.0174 default 0.0207 they are close enough that they are in the bounds of error. The result shows the strain was reduced by the geometric design.

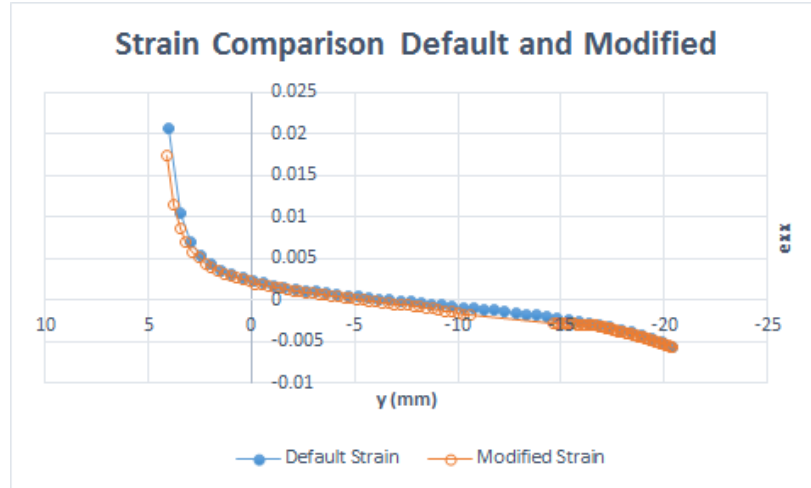


Figure 2. Strain Comparison of DIC Model and Modified Model

Maximum Stress concentrations are by the notch and that is expected to be the location of failure. The uncertainty seems to be higher along the notch. To determine the strain the principal stresses were divided by the Young's Modulus.

$$\varepsilon = \sigma / E$$

Where σ = Principal Stress (N/m²) E = Young's Modulus (N/m² (Pa)).

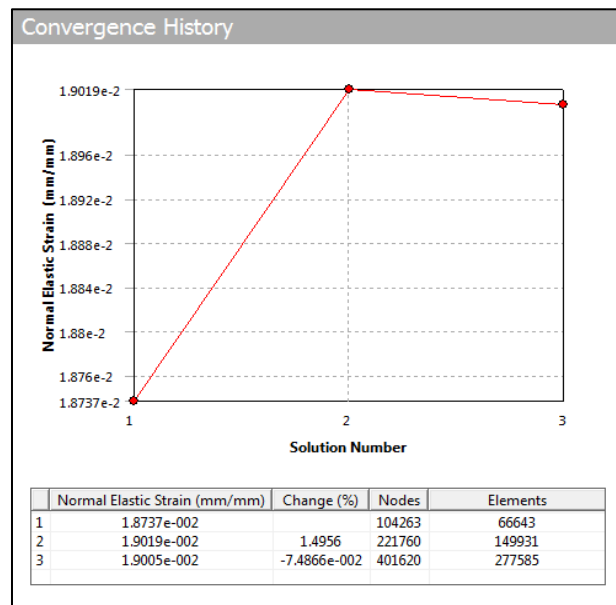


Figure 4: Strain Convergence Test

A brief convergence study was performed to verify that as the mesh was refined, the results did not vary by more than 2%, confirming the quality of the mesh. To get a better result, an improvement of the mesh could be done by refining it at critical locations in anticipations to have the stress raisers converge immediately and therefore reducing the number of elements needed to converge for the entire plate. In addition, a 2D analysis and/or symmetric analysis, can shape the inside of the plate. In addition, the geometry should be adjusted, so that there are no shapes along the geometric path.

Member 3: Adil Ansari

Modification and Meshing

The notch modification was an added semi-circular cut vertically offset from the existing notch. The idea behind this was that a larger radius would result in a lowered stress concentration. The semi-circle's radius was optimized for the lowest possible maximum strain. A radius of 7 mm was the chosen radius as any smaller would make it uncomfortably close to the 5mm distance from any boundary of the default design while increasing the radius would cause unwanted increase in strain. A triangular 2D mesh was used for the stress analysis. The mesh was generated using MATLAB's PDE toolkit. While this add-on is able to generate quadratic mesh it is not able to refine an existing mesh in an orderly fashion for quadratic 6-point triangle mesh. Hence a 3-point linear triangular mesh is used.

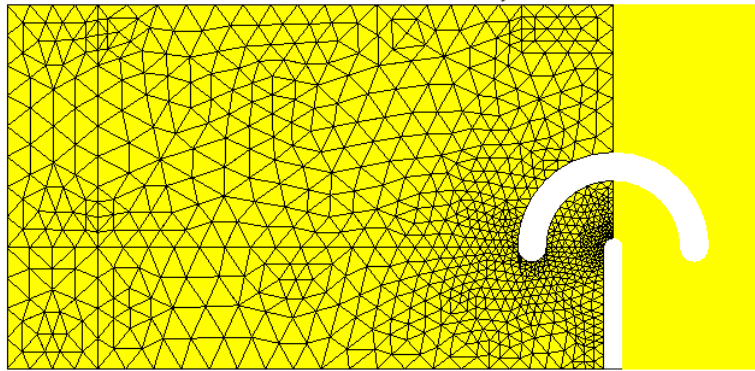


Figure 1. Generated Mesh

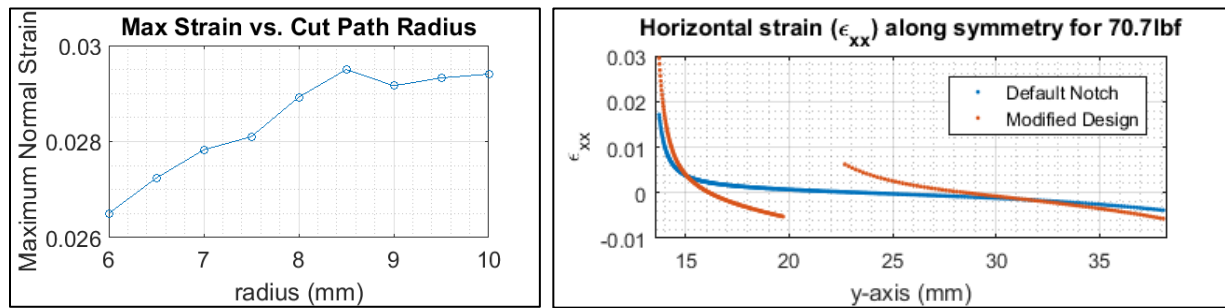


Figure 2. Max Strain vs. Cut Path Radius (left) and Horizontal Strain Along Symmetry for 70.7 lbf (right)

Stress Test

The most crucial parameter in assessing the structure's robustness is ϵ_{xx} along the axis of central symmetry. For the simulation, it was assumed Young's Modulus is 2.7 GPa and the Poisson's ratio is 0.35.

The resulting distribution of stress and its corresponding strain along the symmetry line suggests that the strain is increased in the modified design at the notch in comparison to the default unmodified design. Hence this design would have failed the linear stress testing. However, under a non-linear assumption for the relationship between deformation and force post fracture, one could imagine that the larger semi-circular cut could have prevented the crack from propagating.

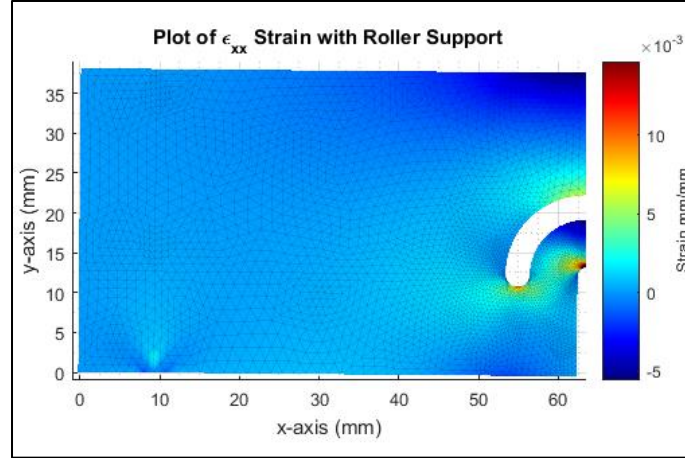


Figure 3. Plot of Strain with Roller Support

Convergence Test

In order to validate the simulation, the simulation was redone on successively refined mesh. The modified design was used for this test as validation required just that. Individual mesh elements were broken down into four equally sized, smaller elements while also taking into account the geometry of design especially curved edges. The rate of convergence was established to be 1.26 which is slightly better than the expected convergence rate of 1. The maximum strain in the default notch was extrapolated as $2.95e-2 \pm 0.02\%$.

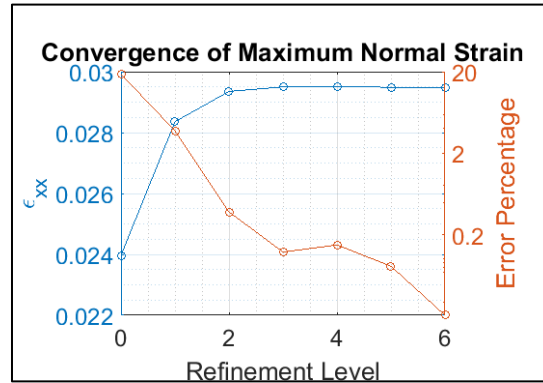


Figure 4. Convergence of Maximum Normal Strain Plot

Sensitivity to boundary Condition

Application of Boundary Conditions were crucial to the entire analysis. A symmetry boundary condition was applied at $x = 63.5\text{mm}$ as the design was symmetrical along that axis. This meant the nodes with this BC were only allowed freedom in the vertical direction. Since only half of the design was used, the computational cost decreased 8 fold. Only half the force was applied on top at node locations. This half force was divided by the thickness of the plate and force application length to compute the pressure applied. This pressure was applied along the force node locations. In order to arrest all freedom of movements, another pin was applied on the 9.5mm from the bottom left. However, on replacing the pin support with the roller support, it was seen that the deflection and strain parameters increased by a factor of 4. Later on the strain data was seen to match with the roller support better than fixed pin support and hence free support was chosen to be the choice. Also it can be noticed that fixed support causes support area to go below $y = 0$. The roller support raises the support area which is far more realistic. The trade-off is shifting of roller by an infinitesimal amount.

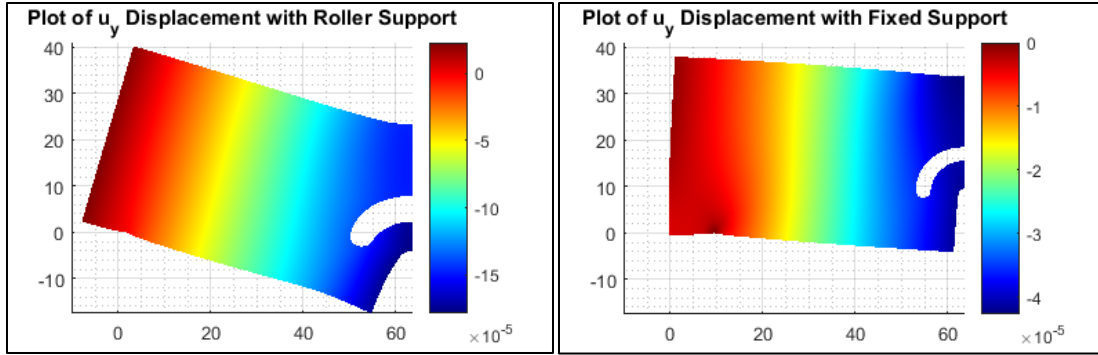


Figure 5. Plot of u_y Displacement with Roller Support (left) and Fixed Support (right)

Solver Description

Since this simulation was hardcoded top to bottom, many optimizations were done to improve performance of the code in terms of speed. The assembly was sped up by accumulating indices and value onto a 1D matrices and later assorted into a sparse matrix. Also the accumulation process was further sped up by converting assembly process into a machine-level C code and subroutine the assembly process. The solver uses LU decomposition. Iterative solving techniques weren't applied as it uses incomplete LU decomposition as a preconditioner whose memory footprint is of the same order of magnitude as that of LU decomposition and the RAM only allowed for 6-consecutive mesh subdivisions.

Member 4: Michael Buller

The predicted linear strain, x-direction, along section A-A increases from top to bottom. The prediction ignores the immediate region where the load is applied. In the presence of a notch the stress becomes concentrated at the notch location thus the strain increases. This occurrence is due to the immediate change in cross sectional area. To dissipate the concentration, the cross section near the region of concentration must decrease gradually leading to the notch. The modified plate is given can be found in the Appendix (A0). The placement of the holes was determined using the parametric feature in ANSYS. The analysis consisted of varying two of the three variables, radius and placement (horizontal and vertical, measured from base and centerline) of circular extrusion measured from the base and center line respectively. Strain measurements were taken per simulation, then compared.

Characterization of Design

The following description defines the model utilized in the analysis (ANSYS) of concentrated stresses located at the plate notch. The plate's symmetry was utilized in the analysis. Initially a pin support was chosen (9.0mm inward-left side). Due to the concentrated stresses at the support, ANSYS was unable to converge to an exact solution. The supports were replaced with a frictionless support over a 9.0X6.35 mm² region (measured from left edge - bottom surface). Additionally, displacement constraints (free displacement in x and y) were applied at the outer and inner points of the bottom left edge because stress raisers still existed at these locations. Transitioning to load conditions, the model includes conditions in which the load is applied uniformly across 19.05X6.35 mm². Finally, the material properties chosen included Young's Modulus and Poisson ratio of 2.7GPa and 0.35 respectively. The plate under observation was made from Plexiglas. The above was used in the analysis of a 3D model.

Mesh

Figure 1 shows that the majority of the elements (average = 0.66) are composed of tetrahedrons with ten nodes. The convergence rate of Tet10 elements are known to be approximately 4.0.

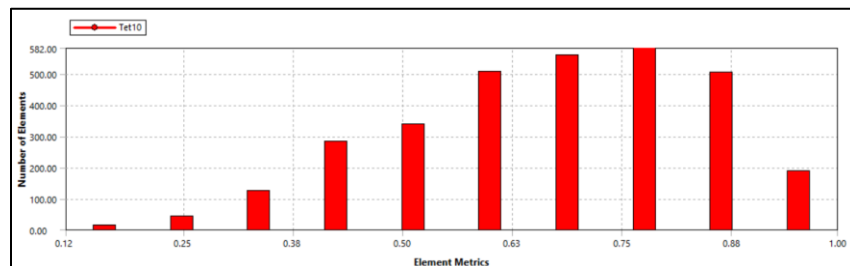


Figure 1: Perfect tetrahedral elements have an element metric of one

Simulated Results

The validity of the model was determined by comparing the strain results generated from the base model with an applied load of 311 N (70 lb-f) to the results given by the instructor. The graph produced gave a 7% difference between maximum strains. Observation was made once convergence was established and stress and total deformation contours were analyzed, Figure 2, and deemed “physically correct.” Additionally, the un-averaged strain color-band was analyzed in the region containing the notch to ensure no discontinuities existed within the mesh, Figure 2. The color band was continuous. Convergence was established at 0.44% for maximum principal stress within two mesh refinements, Tab. 1. Strain convergence was opted for stress convergence because stress raisers, mentioned in previous section, were the cause of diverging solutions. This indicates that the strain varies slightly between a difference of 5381 elements. In other words, the results are not sensitive to the element size for the given load. Given the above, the model was assumed to be “physically correct” and accurate.

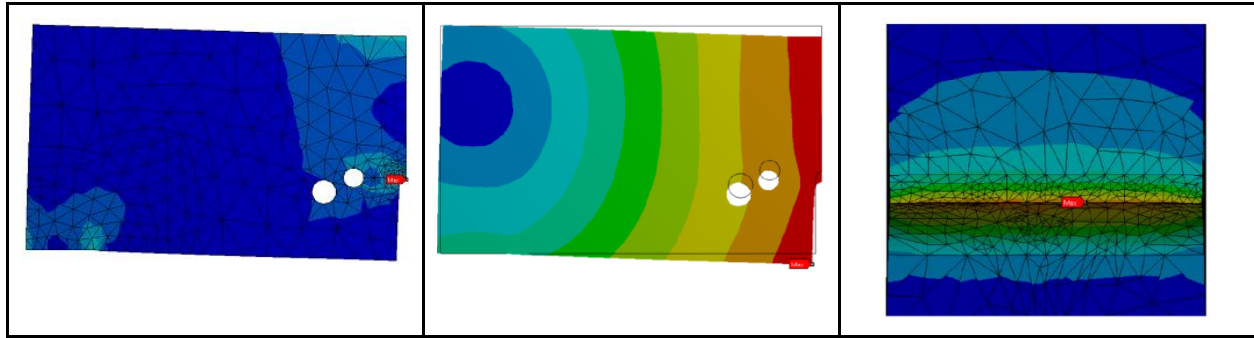


Figure 2: Stress/Total Deformation/Color Band

Table 1: Mesh Convergence

	Maximum Principal Stress (psi)	Change (%)	Nodes	Elements
1	5138.6		5515	3134
2	5161.	0.43403	13746	8515

Modification Assessment (Simulation)

The maximum strain produced from 320N (72lb) in the modified plate was 0.0091 [mm/mm], approximately 45% reduction of the concentrated strain.

Modification Assessment (Real World)

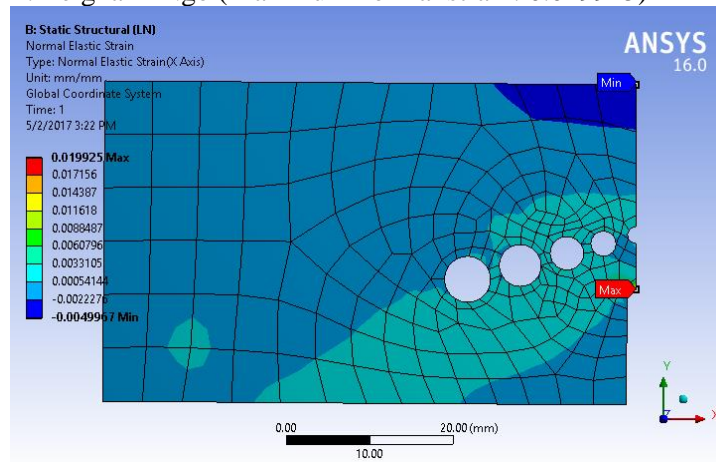
The plate given is rated with a yield strength of 55 MPa. In real world testing, the modified plate failed with an applied load of 560N (126lb). Simulations in ANSYS showed a maximum principal stress of 60.4 MPa from a 560N load. Simulations were iterated until a maximum stress ≤ 55 MPa was established. This occurred for an applied load of 500N.

Conclusion

Given the information above, the model defined has a factor of safety of 1.12. The difference between the simulation and real world are likely due to inconsistencies within the material such as surface cracks or measurements made by DIC software. Prior to introducing a theoretical model to the real world a factor of safety, minimum 2.0, would be established to take care of imperfections of the idealized model. With an established factor of safety of 1.2 and considering the sensitivity of the model (*Verification: Validity of Model Section*) and the stress, deformation, strain data generated (*Verification: Validity of Model Section*), this model is an accurate representation.

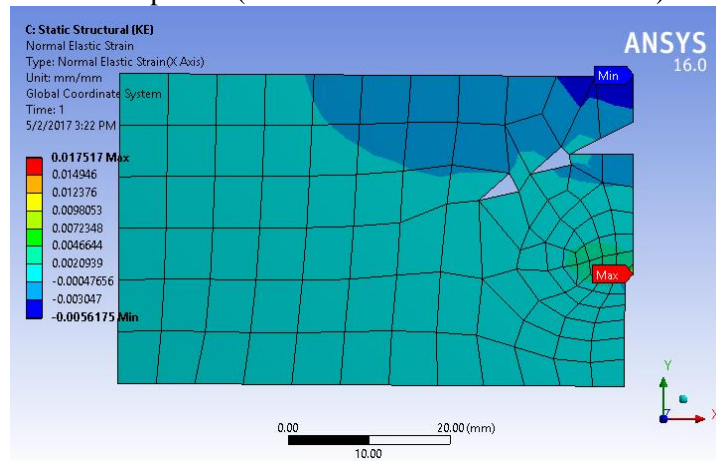
Team Evaluation and Selection of Designs

Member 1: Leighann Ngo (Maximum normal strain: 0.019925)



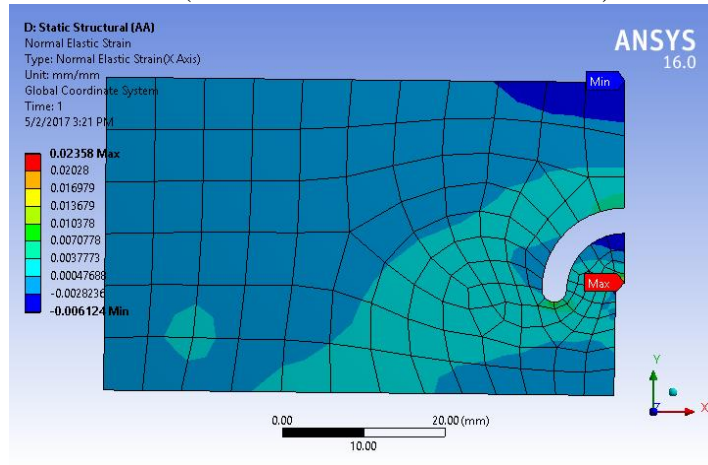
	Score
Report is written clearly and professionally	4
Description of analysis	3
Boundary conditions	4
Mesh convergence study	3
Design of geometry modification	2

Member 2: Kevin Espinoza (Maximum normal strain: 0.017517)



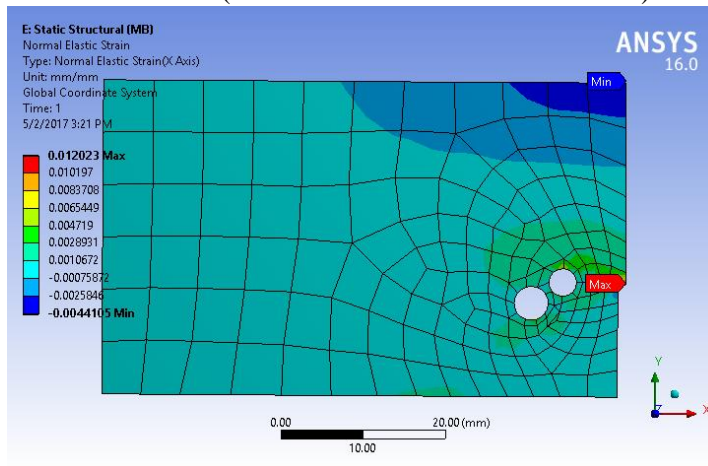
	Score
Report is written clearly and professionally	3
Description of analysis	4
Boundary conditions	3
Mesh convergence study	3
Design of geometry modification	3

Member 3: Adil Ansari (Maximum normal strain: 0.023580)



	Score
Report is written clearly and professionally	3
Description of analysis	4
Boundary conditions	4
Mesh convergence study	4
Design of geometry modification	3

Member 4: Michael Buller (Maximum normal strain: 0.012023)



	Score
Report is written clearly and professionally	4
Description of analysis	4
Boundary conditions	3
Mesh convergence study	3
Design of geometry modification	3

Comparison of Predicted and Measured Values

Experimental Test Setup

To monitor the strain and deformation of the modified plate under a three-point bending test, the plate was coated with a spectral ceramic coating specifically used for comparing deformations before and after the loading of 70.7 lb-f. Application of forces was successively increased and recorded until the point of failure.

As predicted by the FEM analysis, the plate's critical location and initiation of fracture occurred at the tip of the notch. The fracture propagated nearly instantaneously, resulting in a complete split of the plate and instant structural failure. The failure occurred at an applied load of 126 lb-f.

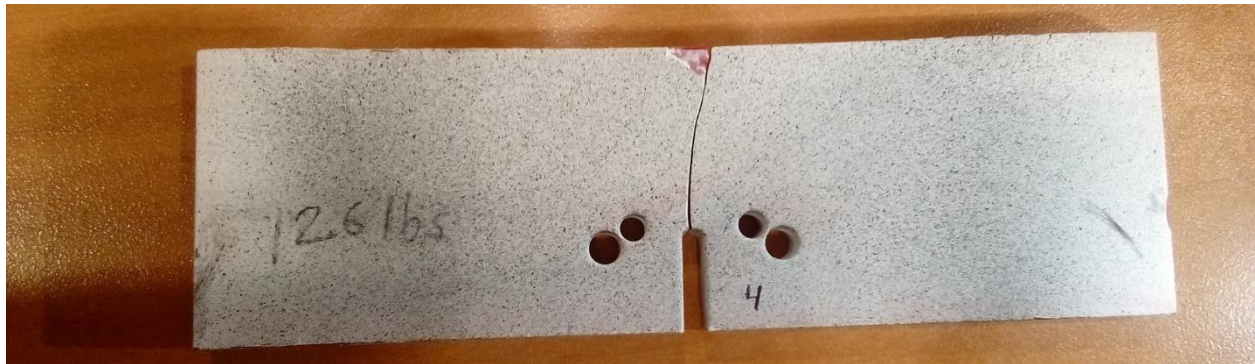


Figure 1. Photograph of Failed Modified Plate.

Digital Image Correlation Software and how it Works

DIC software detects the position of a speckle pattern based on a digital image. The spectral ceramic coating applied to the modified plate contained specks that were detected by the DIC and used to compare the position of the speckle pattern from before and after images of the Plexiglas plate. Using the comparison, the DIC calculates the change in the speckle pattern and the resulting strain data was obtained.

Note, while this ascertained displacement recorded by the DIC may not be entirely correct as the camera may have moved between the before and after shot of the plate, the displacements gradients (i.e. strain) should be independent of the camera angle perturbation.

Finite Element Method Assumptions

While performing the FEM analysis, it was assumed that the variability in strain parameters were negligible across the thickness of the plate. It was also assumed the supports were frictionless. These choices were made entirely based on the experimental data closely matching with the simulation data obtained from said boundary conditions and the realization that horizontal displacement is comparatively negligible despite the frictionless support.

Results

The following figure shows the data ascertained from a two-dimensional MATLAB simulation, three-dimensional ANSYS simulation, and the data gathered from the DIC displacement analysis of elastic normal strain.

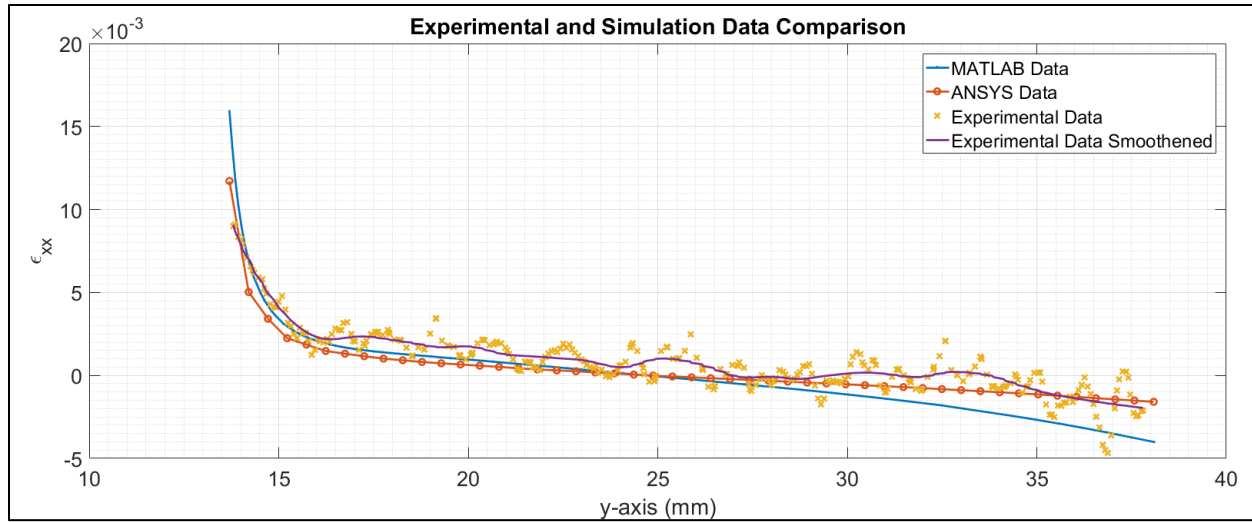


Figure 2. Experimental and Simulation Strain Data Comparison

The following figures show the strain field across the plate under load in MATLAB, ANSYS, and the DIC. The overall displacement gradients are very similar with the key maximum strains in the same location across all three figures.

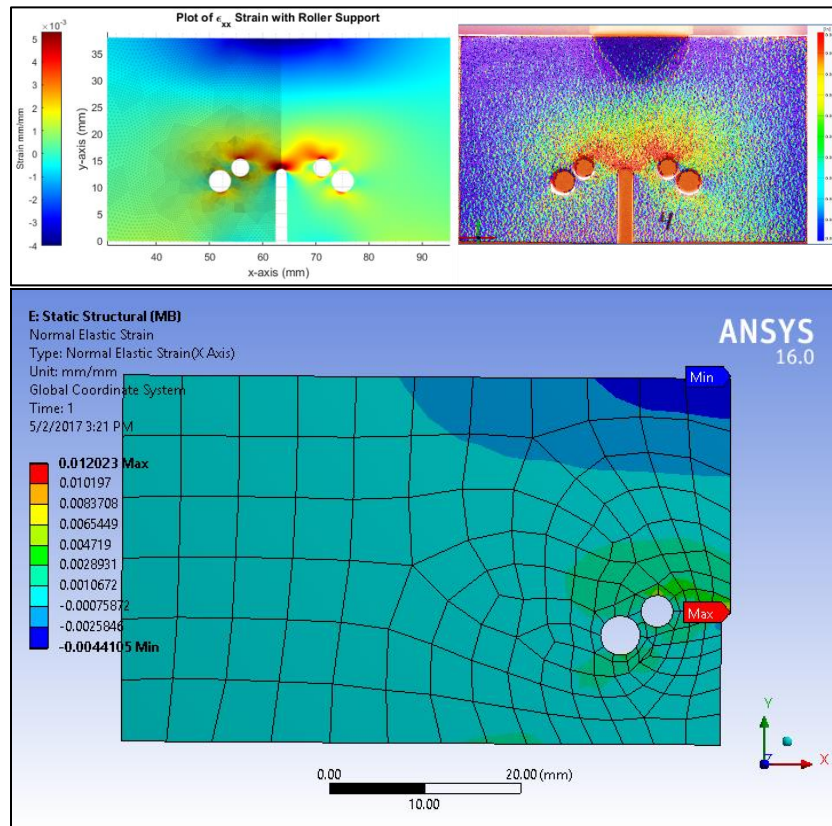


Figure 3. Strain Gradients Under Load from Matlab, DIC, and ANSYS (from left to right, top to bottom)

Upon comparison of the experimental strain data against the simulated strain data, the experimental data has noticeable fluctuation in strain which is visible in Figure 1. This fluctuation was likely due to the DIC's ability to collect data based on the spectral ceramic coating. Note, in Figure 2, the spectral coating generated a pixilated image that caused an overall lack of smoothness in the strain plot compared to the smoothness generated by both simulation software. This pixilation was likely due to the coating containing varying sized speckles that could have conglomerated in certain areas. In addition, in areas where the speckle pattern was not continuous could have caused the prediction of strain smaller than the speckle pattern itself not to be resolved experimentally by the DIC software. As a result, resolving strain at critical locations, especially the tip of the notch, was particularly challenging. Due to this inability to resolve strain at the critical locations, the simulation strain data exceeds the experimental strain data.

A second issue that could have contributed to the differences between the experimental and simulation strain data was variability in the Young's modulus of the plate material. The exact value of the Young's modulus of the experimental plate was unknown. For this reason, the simulations were performed with estimated values, causing one form of discrepancy between the experimental and simulation data. Also, there was potential for the actual Young's modulus value in the experimental plate to be inconsistent within the plate itself; there could have been material discontinuities in the plate that were not noticeable nor accounted for.

Another source of error between the experimental and simulation data could have been the lack of symmetry in the experimental setup. In both the MATLAB and ANSYS simulations, the models assumed a symmetric and equal force application at the center of the plate. In Figure 3, the digital image of the experimental plate's strain clearly shows the force application was not centered. The offset could be a reasonable explanation for the smoothed experimental data of normal strain not matching the simulation values spatially in Figure 2.

Conclusion

In this project report, an investigation was made to analyze and validate simulation results of modifications of a notched plate where the modifications were made to increase the factor of safety of a design. Various modification designs were tested to see which design was optimal. The optimal design was further tested experimentally and compared against two software simulation results, one being MATLAB and the other ANSYS. The simulations utilized different software and used distinct boundary conditions. As a result, the design that removed the least material along the path between the load and the notch was the optimal design. The variance between the experimental and simulation data was potentially due to discrepancies in the material properties, the experimental setup, and the simulated boundary conditions.

Appendix

A1. Modified Plate Dimensions.

All dimensions in (mm). Plate thickness is 6.35 mm.

