FEA Analysis of a Caliper Abutment Bracket

ME 404

Jesse Doty
March 17, 2008

Abstract

As braking forces are applied at the wheel of an automobile, those forces must be transmitted to the suspension of the vehicle. A caliper abutment bracket must take the frictional forces generated by the brake system and not yield or overly deflect. Using ABAQUS a finite element analysis was performed on a caliper abutment bracket. This analysis found the max stress of 45ksi and its location. The maximum deflection at the point of outer pad loading was found to be 0.04887in. in the direction of the load and was compared to hand calculations done using Castigliano's method to find deflection at the same point.

Introduction

Modern disc brake systems are composed of several components. The forces that actually stop the vehicle take place at the wheel. Here, hydraulic pressure pushes out a piston on the brake caliper. This action causes the outer portion of the caliper and the piston to "squeeze" the rotor between two brake pads. All the frictional force is conveyed at this pad interface. A caliper abutment bracket (see Figure 1) holds the pads (in the radial and tangential directions) as well as attaches the caliper to the spindle. Thus, the abutment bracket must withstand all the braking forces at the wheel, as well as the thermal loads generated under heavy braking.

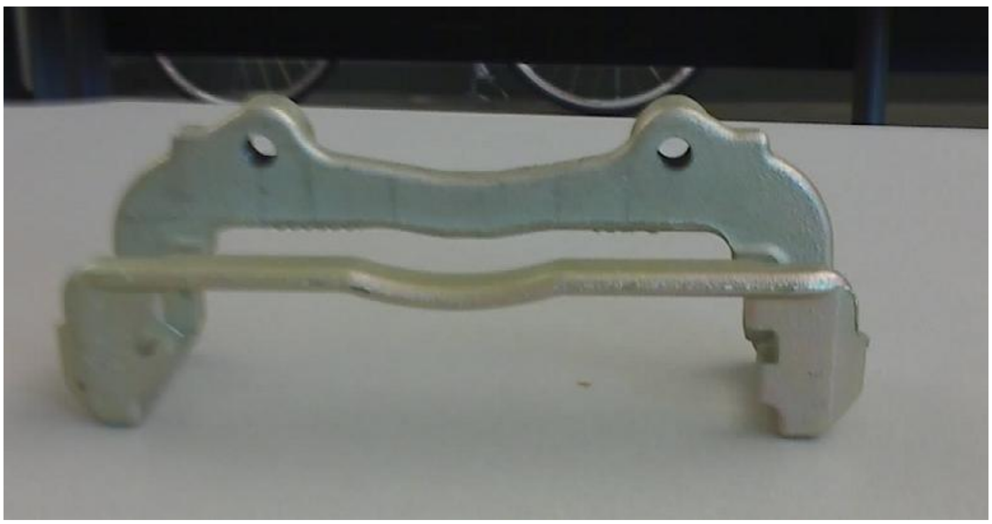


Figure 1. The caliper abutment bracket to be analyzed.

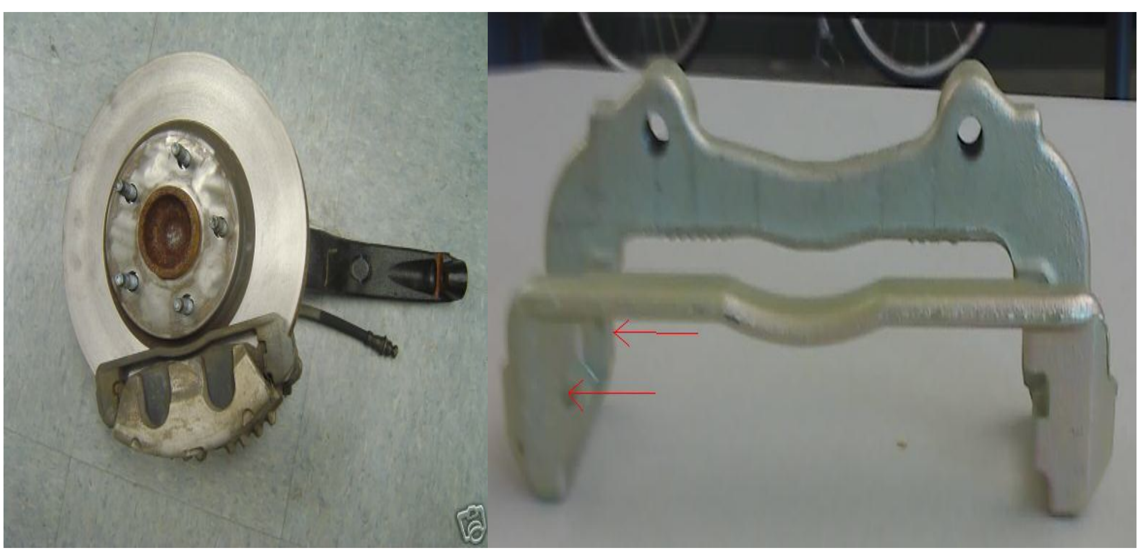


Figure 2. Left: Picture showing how the bracket positions the caliper on the spindle. Right: Bracket and location of applied brake pad force.

Desired Outcomes

To analyze the mechanical and thermal stresses the bracket endures under full braking from 60mph and to also observe the location of any stress concentrations. This will be done utilizing data from a 2001 Chevrolet Camaro. Also of interest, is the defection at the location where the outer brake pad contacts the bracket. This is marked in red in Figure 2.

Proposed Analysis

1. Mechanical – Analyze the stresses and strains on the bracket due to the braking forces. The inner pad will cause shear on inner portion of bracket and tension on portion containing the mounting holes to the spindle. The outer pad will cause shear and bending on the "saddle" portion of the bracket (the part that straddles the rotor). This will also cause tension on portion with the mounting holes.
2. Thermal - Braking transforms mechanical energy into heat energy. This heat is vented to the environment and to the brake components. The purpose of this will be to see the effects of the temperatures involved on the bracket stresses.

Approximate Solution

Mechanical

The first step in determining the stress/deflection of the bracket was to determine the loads applied to the bracket. Deceleration data for a 2001 Camaro was found from Motor Trend magazine, a well-known, reputable car testing publication. They recorded a stopping distance of 144ft from 60-0 mph. Constant deceleration was assumed and a weight of 3550lbs was used (again from the Motor Trend). From this, the average decelerating force was found. It is commonly acknowledged for a front engine, rear-drive vehicle, approximately 80% of the braking is done by the front wheels. Since all acceleration/deceleration forces created by the car must take place through the tire/ground interface, the resultant decelerating torque was found for the front wheels and then divided by 2 to get a per front wheel brake torque. Using the brake design equation (see Juvinall), assuming constant wear and two pad interfaces, the brake force per pad was found.

Hand calculations were performed using Castigliano's Method and breaking the bracket up into several simple shapes and then analyzing the brake pad load as creating either a bending or axial load on that section of the bracket.

For hand calculations, the pad load was treated as a point load acting at the center of the pad guide. Some difficulty was encountered in calculating the Area Moment of Inertia (AMOI) for the section that crosses over the rotor (referred to as a "saddle" in this report). Originally, a solution was attempted by breaking the complex shape up into smaller squares and rectangles this resulted in an AMOI of 0.006792in^4. This is the AMOI value used for the original hand calculations, which resulted in a total deflection of 0.086886in of which 91%occurs due to the bending of the saddle. With most of the deflection occurring in this area, it was important to make sure that the calculated section properties were accurate. In the process of developing the originally planned 3-D model, SolidWorks was used and a more accurate value was obtained using the section properties tool of the program. According to SolidWorks, the AMOI was 0.0203in^4, an increase of 199%. This value is used in the revised hand calculations and results in a total deflection of 0.03273in.

Thermal

Braking systems work by turning kinetic energy into thermal energy through friction. Knowing the vehicle's mass, starting velocity, and the distribution of the work done by the braking system front to rear, the energy dissipation necessary of the braking system could be calculated. This resulted in 427497ft-lb of energy being dissipated over 3.273s (assuming constant deceleration). Again assuming 80% of this going to the front wheels and being equally split between them, this resulted in 33.6 Btu/s of energy dissipation per pad. Given the measured area of the pads contact patch, this resulted in a heat flux of 3.584 Btu/(in^2-s) at the pad/rotor interface. Here is where issues with the thermal analysis began. Determination of how the flux was consumed between the rotor and pad would be difficult to determine. Also, obtaining thermal conductivity values for the pad material and the backing plate would be difficult. Another issue included the fact that the area where the pad contacts the abutment bracket, is only $0.055in^2$, definitely not much area for heat transfer to occur and over such a short amount of time (3.273s). At this point, a thermal analysis would be exploratory at best.

Both original and revised hand calculations are attached as Appendix A

Model Development

Originally, a 3-D solid model was going to be used for the analysis. Many measurements of the bracket were taken and used to draw a 3-D model in SolidWorks. This is shown in Figure 3. The model was then imported into ABAQUS where meshing became very difficult. Many partitions would be needed due to the complex geometry. Multiple attempts were made at using the "Bottom-Up" mesh feature in ABAQUS but only created bad quality meshes.

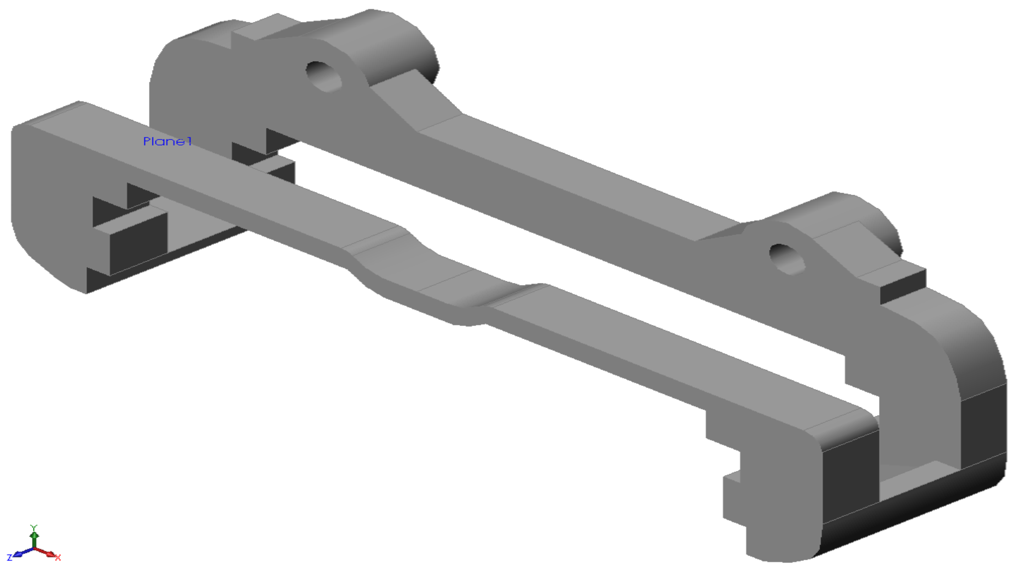
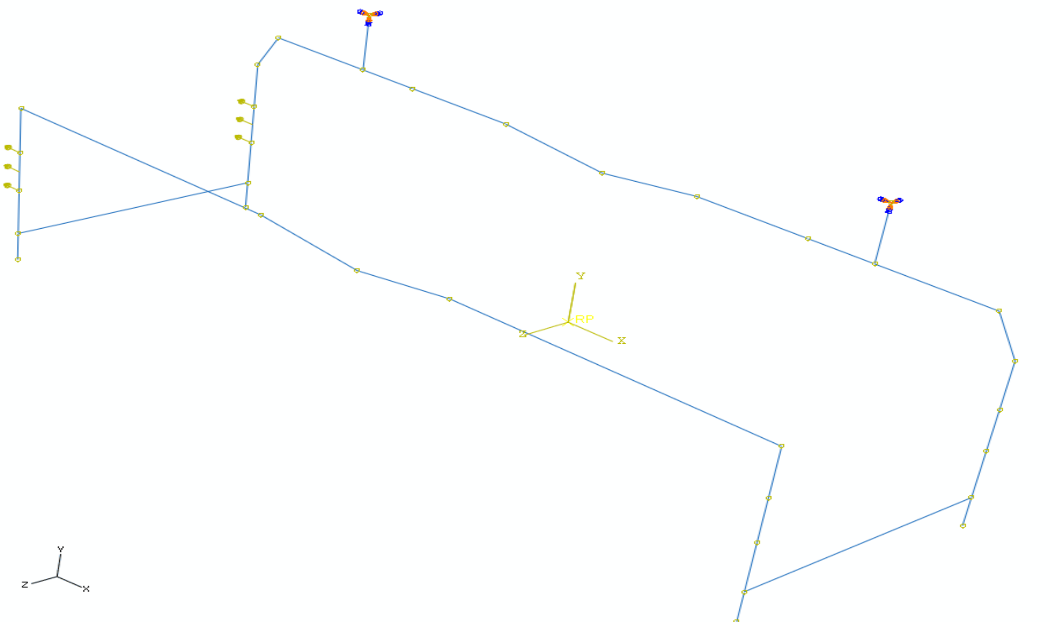


Figure 3. Original 3-D Model done in SolidWorks.

After these failed attempts and talking with Professor Schuster, a beam analysis was chosen. This had some benefits and drawbacks. The benefits would be much easier meshing but the drawbacks included using elements with multiple and axially varying cross-sections. Overcoming these would involve the same approach: breaking the bracket up into smaller beam lengths so that the change in cross-section over the length would not vary as much from one end of the beam to the other. This is analogous to using Riemann sums to approximate an integral. Measurements were taken from the centerline of the sections on the actual bracket. For the cross-section measurements, an average of the cross-section at the segment ends was used. The resulting model is shown in figures 4 and 5. To model the action of the bolt holes, a beam was drawn from the center of each bolt hole, down to the centerline of the beam that connects the two bolt holes. The points representing the center of the bolt holes were then fixed in all three directions and in all three rotations. The loads of the brake pads were then applied as line loads, distributed over the beam element containing the brake guides. The line load was calculated by taking the brake force and dividing it by 7/16in. which is the height of the pad guide. For the saddle, with it's complicated cross-section, a generalized beam was used and the values for AMOI and area entered from SolidWorks.



F

Figure 4. Wireframe model of caliper bracket with loads and boundary conditions.

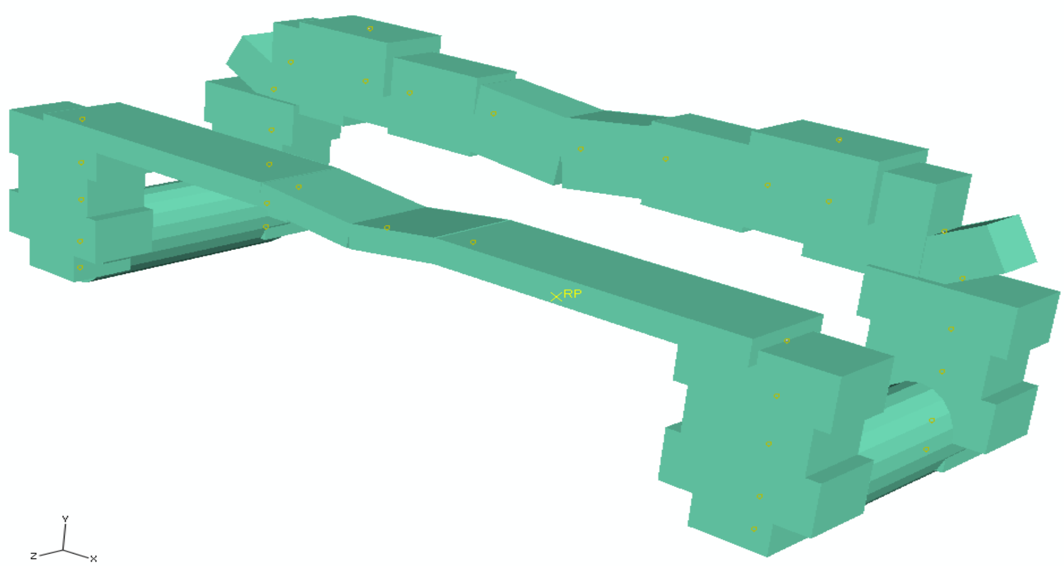


Figure 5. Model with beam profiles showing

Mesh Development

As stated earlier, the original proposal was for a 3-D solid model but meshing required using a bottom-up mesh which had many issues which can be seen in Figure 6 on the following page.

Doing hexahedral elements would have required significant partitioning of the model. A free tetrahedral mesh did work fairly well, however this is not the best for a 3-D model as it usually has issues with calculating stress. This was the case as it calculated a deflection that was within 25% of the beam element model and 5% of hand calculations yet calculated stress values that were 300% that of the beam model. Figure 7 shows these results

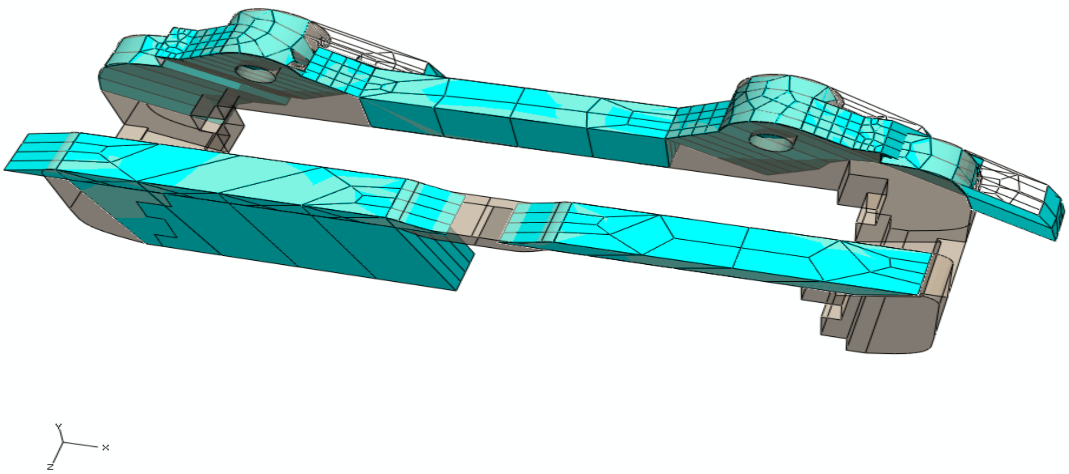


Figure 6. Attempt at bottom-up mesh of solid model.

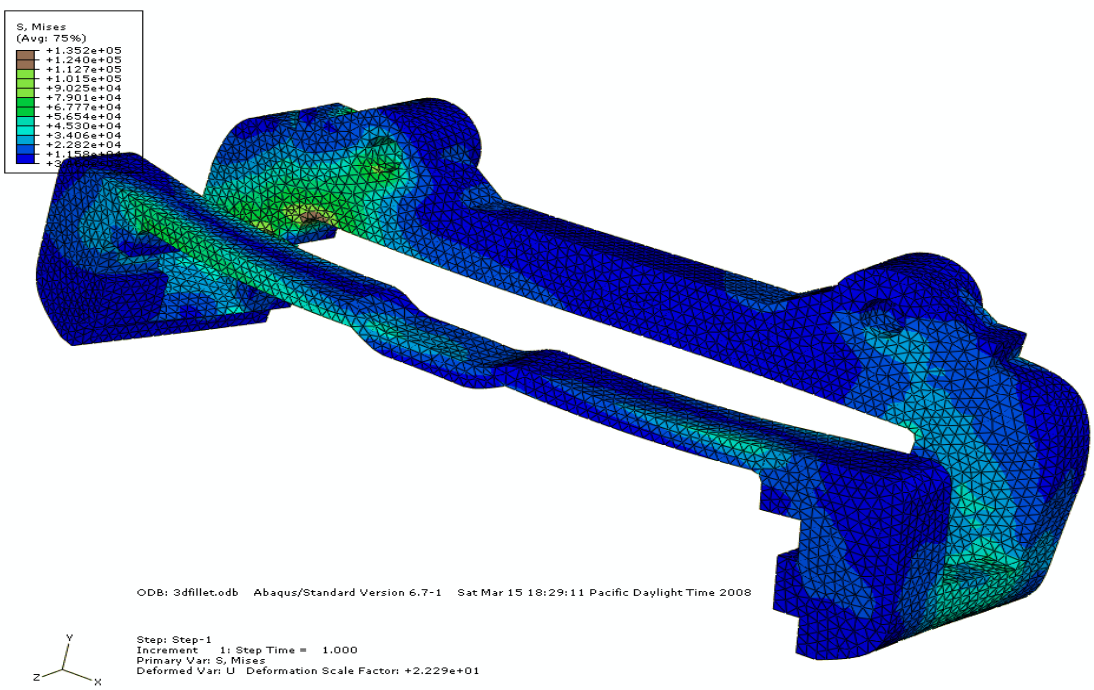


Figure 7. Mesh and contour plot of Von Mises stress for solid model using tetrahedral elements. The final result was achieved using a beam analysis, and varying the number of elements until convergence was reached.

Seed Number	Displacement at Point of Outer pad Force	Number of Elements	Number of Nodes	DOF
1	-0.0469918	34	102	204
2	-0.0484036	68	204	408
4	-0.0487563	136	408	816
8	-0.048845	272	816	1632
16	-0.0488665	544	1632	3264
32	-0.048872	1088	3264	6528
64	-0.0488734	2176	6528	13056
128	-0.0488737	4352	13056	26112
256	-0.0488738	8704	26112	52224
512	-0.0488738	17408	52224	104448
1024	-0.0488739	34816	104448	208896

Table 1. Results of convergence study

Analysis

The final analysis was done using 3-D beam elements and observing the stresses throughout the bracket but most importantly, measuring the deflection of the bracket at the point where the force of the outer brake pad is applied to the bracket.

Errors/Warnings (Mechanical Analysis)

Wireframe (Beam)

Errors

The most common errors encountered were due to section assignments and/or beam orientations not being assigned. This happened a few times due to how the model was created. Using 3-D points, and then connecting them, there is no way to simply edit the part geometry. In order to break down a section into two sections, a point must be inserted and the whole part recreated (i.e. the dots connected). So, every time the model was redone, every section had to be reassigned, beam reoriented, and loads/boundary conditions reapplied.

Warnings

- Node set has no members and will be ignored. This was in regards to the reference point used for model construction and did not affect the analysis.
- For 12 beam elements either the average curvature about the local 1-direction differs by more than 0.1 degrees per unit length…may want to verify that the beam normals are correct for your problem. All beam sections and orientations were double checked to ensure correct assignment.

Errors/Warnings (Thermal Analysis)

There were many errors when trying to run the thermal analysis. In fact, the analysis never went past the Input File Processor. Here is a list of some of the errors obtained.

3-D Solid (Tetrahedral Mesh)

Errors

- Degree of freedom 11 and at least one degree of freedom 1 thru 6 must be active in the model for coupled temp-disp.
- Invalid load type on element

Warnings

- Degree of freedom 4, 5, and 6 are not active in this model and can not be restrained
- 3 elements are distorted

Wireframe (Beam)
Errors
- Degree of freedom 11 and at least one degree of freedom 1 thru 6 must be active in the model for coupled temp-disp.

Warnings
-Same as Mechanical Analysis

These errors could not be overcome for the thermal analysis for both the 3-D and wireframe models. After realizing the difficulty in determining the heat flux from pad to bracket, the relatively small contact area of the pad/bracket interface, and consulting with Professor Schuster it was determined that a thermal analysis would not produce very accurate results without testing for the conductivity properties of the parts and complex analysis of the problem to obtain a reasonable hand calculation for comparison purposes. Also complicating matters would be the considerable and complex airflow that takes place inside a spinning wheel with a vented brake rotor. Furthermore, with the temperatures involved, and depending on how well of an insulator the pad material is, radiation may have a larger impact than conduction. Thus it was not pursued further.

Post-Processing

Mesh Convergence
A mesh convergence study was performed by increasing the number of elements per edge until a steady deflection of a single point (node 23 always located at one of the datum points used for construction) was reached. This resulted in the values previously shown in Table 1 and the plot in Figure 8.

Comparison to Approximate Solution

	ABAQUS	
Hand Calculations	**Wireframe (Beam)**	**3-D (Tetrahedral)**
-0.03273	-0.04887	-0.03082

Table 2. Comparison of hand calculations and ABAQUS model results. These are deflection values, in inches, for the point where outer brake pad force is applied.

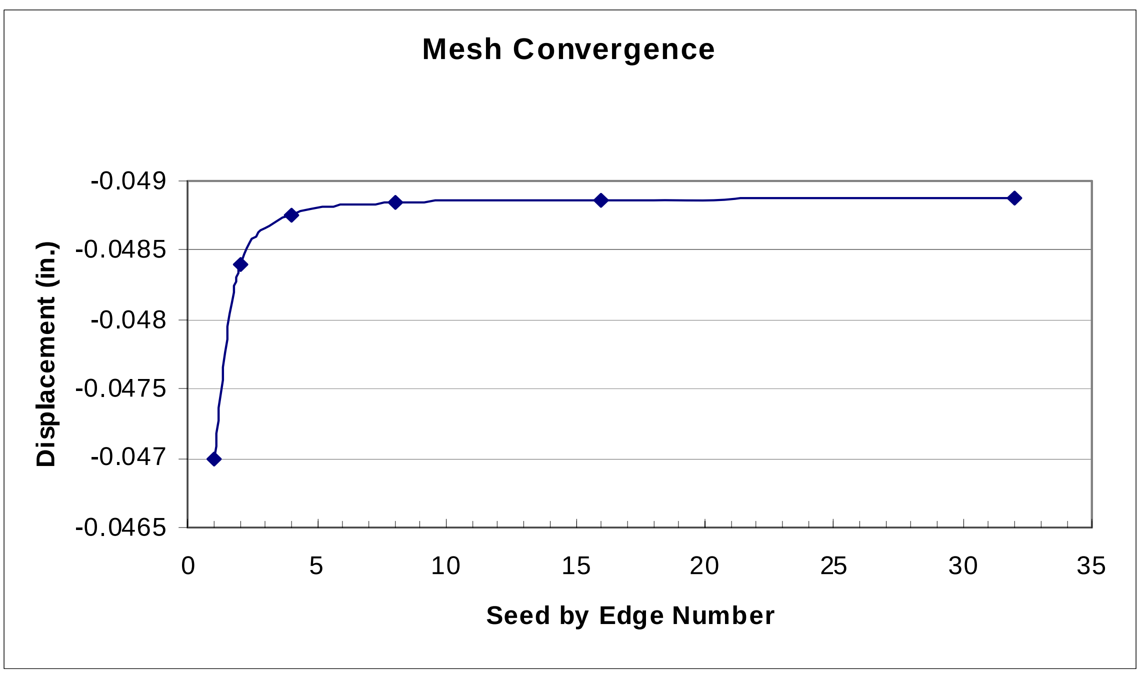


Figure 8. Convergence study results for wireframe model. Note: chart is truncated for readability, beyond 32 elements per edge, the change in displacement was less than 0.003%

Comparison to Test Results
No test data was available for comparison.

Results

As seen above in Table 2, the deflection results were relatively close, well within an order of magnitude of each other. Interestingly, the two models may have somewhat agreed in deflection but were off by 300% in terms of predicted Von Mises stress. They did however agree on the area of greatest stress which is in the area between the bolt hole and downward curve shown in Figure 9. The deflection in the direction that the brake force is applied to the bracket (U1) is shown in Figure 10.

The largest values for stress appear in the section as the bracket transitions from the bolt holes to the downward curved area. This area would be subject to a bending about the z-axis in Figure 9 and torsion in the xz-plane due to the outer pad load as well as torsion in the xy-plane due to the inner pad force. These combine to produce a torsional stress of 9235psi in the curved section. This was interesting, beforehand it was thought that the max stress would be occur in the saddle due to the bending caused by the outer pad force. However after performing the analysis, it appears that the inner pad force pushes the inboard side of the saddle out such that the saddle itself does not have to take the load. Rather, the bending cause by the these two forces are then taken up by the curved section (mainly) and the outer thin beam that connects the outboard portion of the saddles. This connection spreads the bending out to the other saddle and the curved beam on the right side.

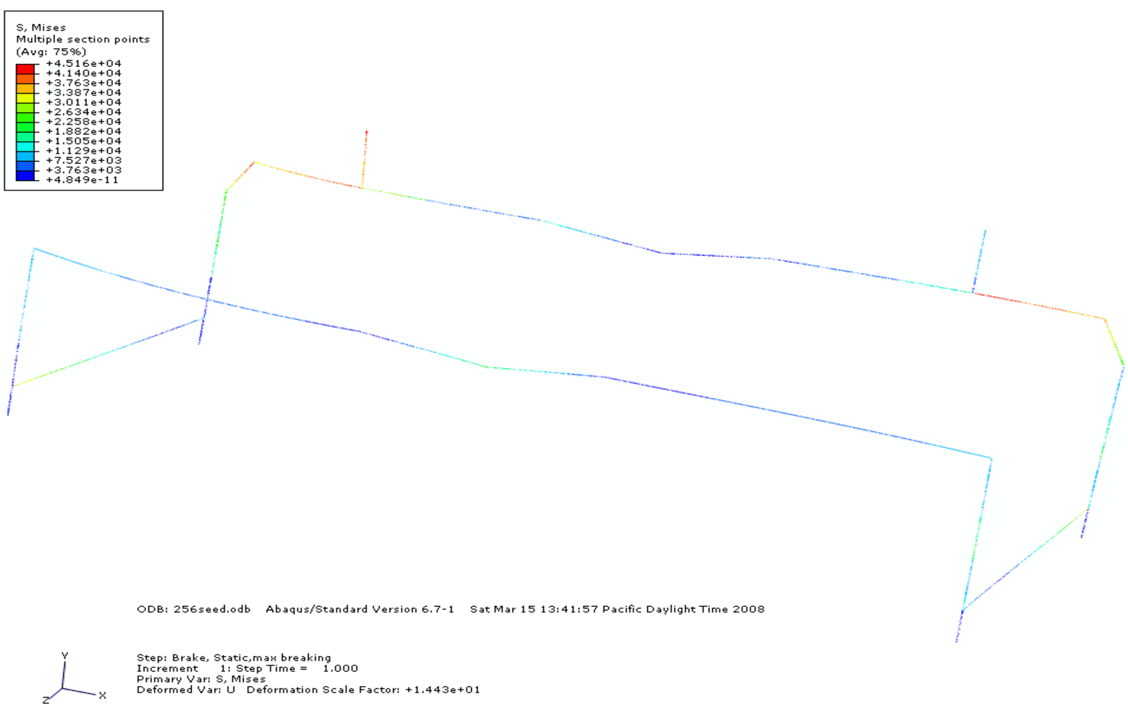


Figure 9. Stress results for wireframe model

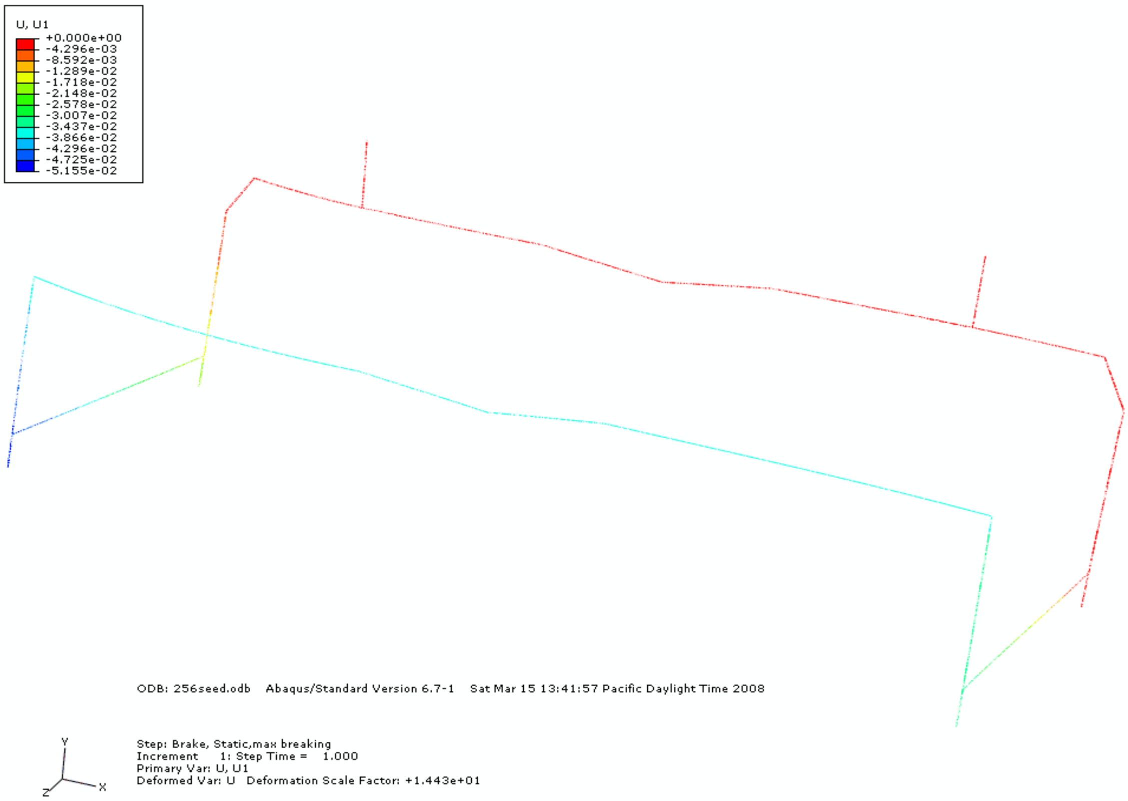


Figure 10. Deflection results in the direction of brake loading (U1).

Discussion

This was an interesting yet challenging project. There are a few things that I would do differently next time. I would have liked to get either of the thermal models to work although I don't know how valid the results would be for reasons already mentioned. Knowing more about partitioning and meshing now, I would like to get the 3-D solid model to properly mesh using hexahedrals. Also I would change the model slightly were it was showing max stress (Figure 9). The SolidWorks model had a small fillet in that area but upon further investigation, the actual bracket is slightly curved as it transitions to the drop-down section. Also, in that portion of the bracket, there is a change in section thickness. These could be part of the reason why the tetrahedral model predicted very large stress values in that area. For the current wireframe model an increase in the number of beam sections would be helpful. Breaking down the current sections into small individual beams would help to decrease the error due to the change in cross-sectional area along the length of each segment.

Overall, I am pretty pleased with the results. Despite some simplification of the beam profiles which make the cross-sections smaller in some critical areas, the stress values (~45 ksi) were extremely workable with non-expensive steels. The deflection was within reason and close to hand calculations. If I were to design a part like this, I would think that decreasing the deflections would help to improve responsiveness and brake feel. Decreasing the deflection in the direction of applied braking forces would allow full braking force to develop faster as less frictional force would go into deflecting the bracket and more into actually slowing down the rotor.

From this project I have learned that doing a solid model analysis can be VERY difficult to get to run properly. Originally I thought that the most painstaking part would be creating the model. After this project I feel that meshing is probably the hardest part of getting any solid model to run. This is especially true after learning how easy it is to import SolidWorks models into ABAQUS.

Using beam elements to represent a solid part with somewhat complex geometry can work well while saving both physical time, in terms of creating the part and meshing, as well as computational time by using simpler elements. This kind of simplification can be extremely useful on more complex parts/models to help reduce set-up and run time.

Finite Element Analysis is a powerful tool that can help analyses of very complex parts but it is not a miracle cure for stress analysis problems. It still requires a great deal of work to set-up and make run properly.

Conclusions

The hand calculations and ABAQUS analyses agreed within 33% of each other. The largest deflection occurs at the point of loading for the outer brake pad which deflects 0.04887 inches in the direction of the pad force. The largest stress is found in the curved portion of the bracket as it transitions from the bolt hole to the vertical section. A wireframe beam element model can represent a solid part very well and be much easier to run. Thermal analyses are difficult to set-up and would not have had much value for this project and thus were not performed. Improvements to the stiffness of the bracket can be had by increasing the torsional rigidity and bending stiffness in the downward curved portion of the beam. Brake feel and responsiveness should slightly improve from these changes.

References

1. Fundamental of Machine Component Design, Juvinall and Marshek, 3rd Edition, Wiley & sons.