ME EN 372 Lab 9 Part 1

and Jordan Hardy

Methods

Description of Boundary Conditions

The scissor jack pin experiences five primary forces: one from the crossbar and four from the diagonal members acting on either side of the pin—two from the upper bar and two from the lower bar. To model these interactions, the central hole through which the crossbar passes was fixed, as shown in Figure 1 (Force B). This constraint simulates the crossbar's role in balancing the horizontal components of the diagonal member forces, effectively maintaining equilibrium at the pin.

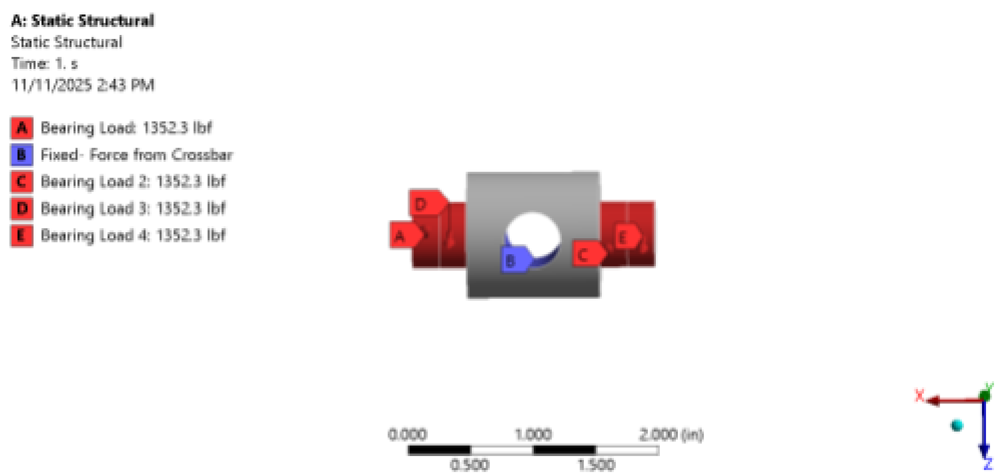

Figure 1. A depiction of the boundary conditions (fixed forces and bearing loads) applied to an ANSYS model of a scissor jack pin.

A minimum angle (θ) of 15° between the horizontal and the diagonal members was assumed. The maximum loading conditions occur at this smallest angle, so bearing loads were applied along directions of +15° and –15° relative to the horizontal. An applied load of 700 lb was assumed at the top of the scissor jack (though this value can vary without affecting the method).

Using a summation of forces approach, the vertical (z) and horizontal (y) components of the forces in the diagonal members acting on each side of the pin were determined as:

$$F_z = \frac{F_{applied}}{2} = \frac{700\ lbs}{2} = 350\ lbs$$

$$F_y = \frac{F_{applied} cot(\theta)}{2} = \frac{(700\ lbs) cot(15)}{2} = 1306.22\ lbs$$

The bearing forces from the upper diagonal members are applied at points A and C, while the lower diagonal members apply forces at points D and E, as illustrated in Figure 1. This configuration allows the assumption that the upper and lower diagonal members are identical in size and geometry, which simplifies analysis while maintaining realistic loading conditions.

The pin is modeled using Structural Steel as defined in ANSYS. Relevant material properties used in the simulation are summarized in Table 1.

Table 1. Selected Material Properties for Structural Steel

Property	Value (psi)
Tensile Ultimate Strength	66,717
Yield Strength	36,259
Elastic Modulus	2.9×10^7

Description of Meshing Methods Used

The meshing process for the scissor jack pin was performed in ANSYS Mechanical using the automatic surface meshing tool. The initial mesh used default settings, which generated a surface mesh suitable for preliminary analysis illustrated in Figure 2. Under these conditions, the model contained 3,027 nodes and 1,617 elements, as reported in the Mesh Statistics panel.

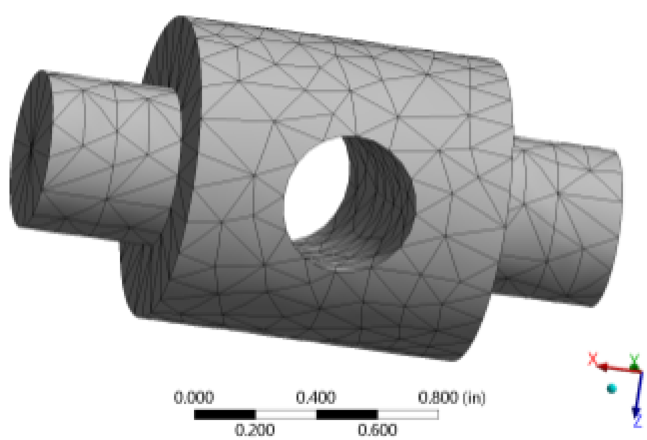

Figure 2. An illustration of the pin model mesh before the convergence tool was applied.

The element type was tetrahedral (solid) with a program-controlled element order, which selected linear elements. The global element size was automatically determined by ANSYS. An attempt was made to apply local mesh refinement on the smaller cylindrical regions that experience the highest stresses (near the bearing contact areas). However, these refinements led to inconsistencies when using the mesh convergence tool, preventing successful analysis. Therefore, the initial mesh was reverted to the default automatic sizing, which provided convergence and adequate resolution for the stress distribution.

Mesh Convergence

To ensure that the mesh was sufficient for accurate stress results, we utilized the mesh convergence tool in ANSYS applied to the Equivalent (von Mises) Stress results. Initially, the tool failed to complete due to localized stress concentrations at the sharp transition between the large and small cylindrical regions of the pin. To address this issue, a fillet was added back into the geometry at this transition, which allowed the convergence process to proceed successfully.

After implementing this geometric modification, the convergence tool was rerun with successive mesh refinements. The results from each iteration are summarized in both Table 2 and Figure 3. The table reports the maximum equivalent stress, the percent change between iterations, and the corresponding number of nodes and elements generated during each refinement step.

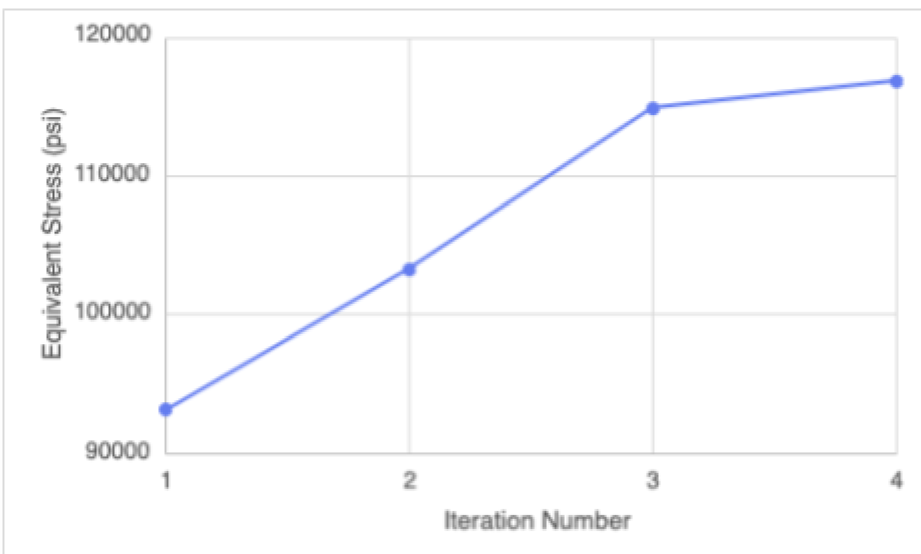

Figure 3. A graphical representation of the data obtained during the application of the convergence tool.

Table 2. Mesh convergence data for equivalent stress results.

	Equivalent Stress (psi)	Change (%)	Nodes	Elements
1	93072	0	3577	1970
2	1.0323e+005	10.346	23737	15187
3	1.1489e+005	10.695	105432	73242
4	1.1684e+005	1.683	207905	146733

After the fourth iteration, the percent change in maximum equivalent stress dropped to 1.68%, indicating that further refinement would result in negligible improvement relative to the increase in computational cost. Thus, this iteration was selected as the final mesh density for all subsequent analyses.

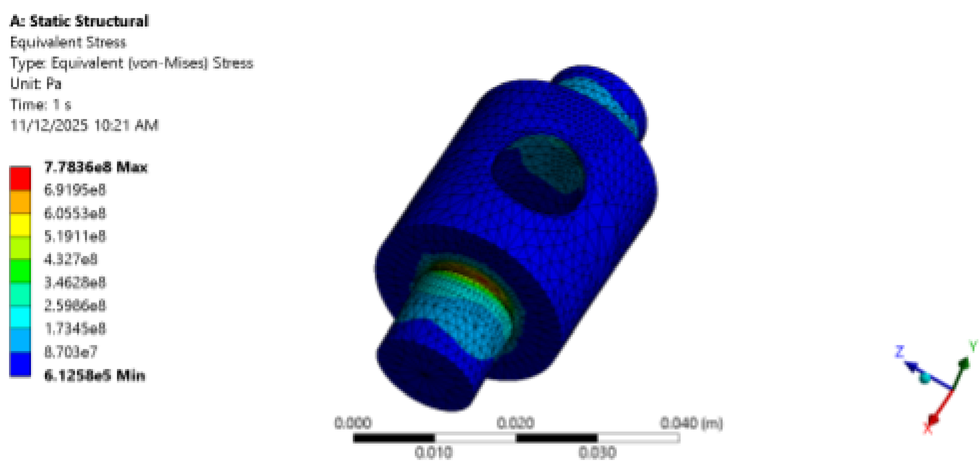

Figure 4. A plot showing the equivalent stress and final mesh of the pin after using the convergence tool.

The final converged mesh displayed smooth transitions, particularly around the filleted regions where stress gradients are highest. This final mesh can be seen in Figure 4. This consistency, combined with a low convergence error, confirms that the mesh was sufficiently fine to achieve reliable and stable stress results.

Model Validation

To validate the accuracy of the scissor jack pin model, a hand calculation of the maximum shear stress was performed and compared with the shear stress simulation results from ANSYS. This approach provides a simple check to ensure that the simulation produces reasonable stress magnitudes consistent with basic mechanics principles.

The maximum shear stress for a solid circular cross section under transverse shear loading can be estimated using the following relationship:

$$\tau_{max} = \frac{4V}{3A}$$

From the boundary condition analysis, the horizontal shear force on one diagonal member was found to be F_y = 1306.22 lb. Because the pin experiences shear from two diagonal members on one side, the total shear force considered in the hand calculation is:

$$V = 2F_y = 2(1306.22\ lb) = 2612.44\ lb$$

The cross-sectional area of the pin's smaller cylindrical section (diameter d=0.521) is given by:

$$A = \frac{\pi}{4}d^2 = \frac{\pi(0.521\ in)^2}{4} = 0.2132\ in^2$$

Substituting into the shear stress equation yields a maximum shear stress of 16,338.77 psi, or approximately 16.3 ksi. When the model was analyzed in ANSYS, the Maximum Shear Stress plot reported a peak value of 18,772 psi, while the Shear Stress plot gave a maximum value of 14,914 psi. Since the analytical value lies between these two simulation results, and both are within 20% of the hand calculation, the agreement is acceptable. This close correlation indicates that the boundary conditions and modeling approach were applied correctly and that the simulation accurately represents the physical behavior of the pin under load.

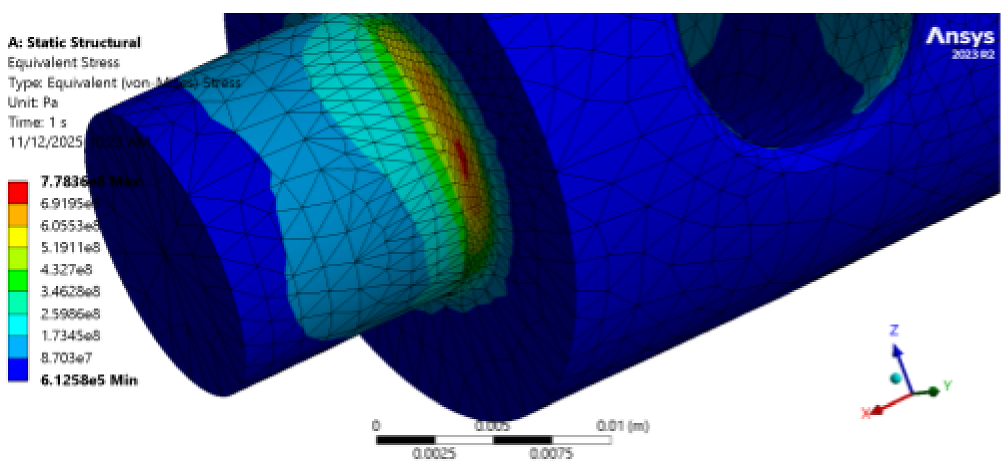

Figure 5. A plot depicting the increased stress in the fillet.

The stress distribution shown in Figure 5 shows the highest stresses concentrated around the bearing contact regions, which are locations where one would expect stress to be highest due to loading. The agreement between analytical and simulated stresses, along with physically consistent stress patterns, validates our model.

ME EN 372 Lab 9 Part 2

and Jordan Hardy

Description of Initial Results

Our initial analysis revealed two primary stress concentrations in the part: the central through-hole and the diameter transitions at each end. As shown in Figure 1, the stress around the central hole remains relatively low. This is likely due to the fixed boundary condition applied in ANSYS as well as the larger diameter in that region, which helps distribute the load more effectively.

In contrast, the highest stresses occur at the locations where the cross-section changes. This behavior is expected, as these geometric transitions introduce sharp stress gradients and are subjected to significant bending loads. Because the diameter changes represent the dominant source of peak stresses, we chose to focus our improvement efforts on mitigating stress concentrations in these regions.

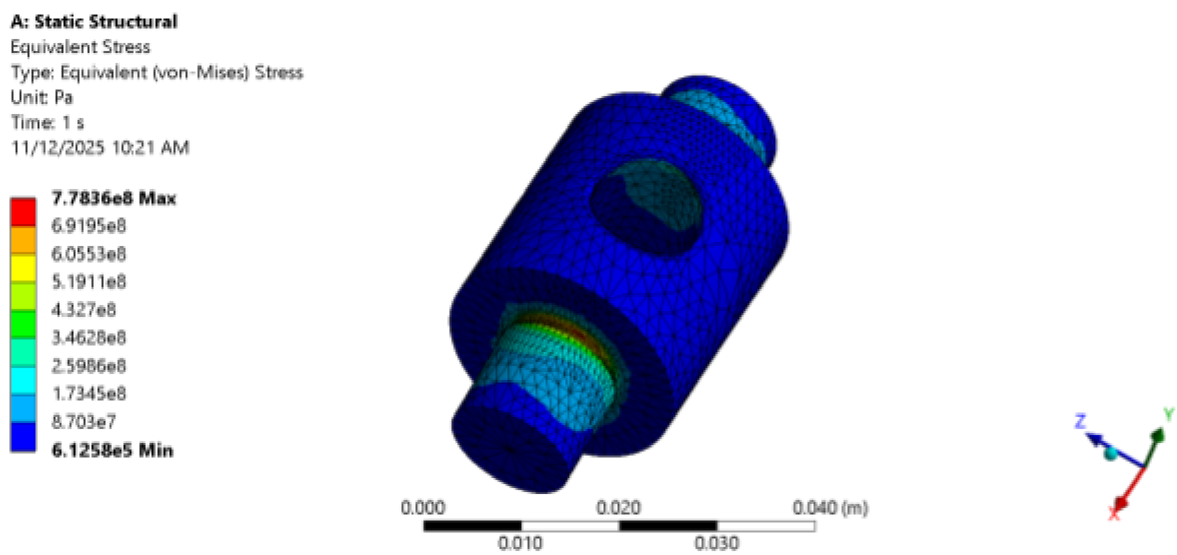

Figure 1. A plot illustrating the basic stress state of the pin before any geometry changes were made.

Changes for Improving the Part

To improve the part's safety factor, we aimed to reduce the maximum bending stress experienced by the pin. Since stress is inversely proportional to the cross-sectional area, we decided to increase the pin's cross-sectional area to lower the bending stress.

We focused on bending stress because it was identified as the largest contributing factor to potential failure. Two critical regions were identified as likely to experience the highest bending moments: the center of the part, and the location near the fillet, where there is a change in cross section.

To compare these regions, we analyzed the bending moments at both locations using the force F_z from Part 1 of Lab 9, applied approximately halfway along the end peg (0.22 in).

Using a moment diagram, we calculated the moment at the fillet (M_f) and the moment at the center (M_c) as follows:

$$M_f = F_z(.22\ in) + 2F_z(.22in) = 862.1\ lb \cdot in$$

$$M_c = F_z(.22) + 2F_z(\frac{1.94}{2} - .22) = 2246.7\ lb \cdot in$$

Using these moments, a large diameter of 1 inch, and a small diameter of 0.521 inches, we calculated the stress at both locations using the bending stress formula:

$$\sigma = \frac{Mc}{I}$$

$$\sigma_f = \frac{(862.1\ lb{\cdot}in) \times (0.521\ in\ /\ 2)}{\pi\ (0.521\ in)^4\ /\ 64} = 62093.65\ psi$$

$$\sigma_c = \frac{(2246.7\ lb{\cdot}in) \times (1\ in\ /\ 2)}{\pi\ (1\ in)^4\ /\ 64} = 22884.68\ psi$$

These calculations confirmed that the fillet experienced the higher stress, so we focused our redesign there. The initial safety factor was:

$$n_{initial} = \frac{Yield\ Strength}{Max\ Stress} = \frac{S_y}{\sigma} = \frac{36{,}259\ psi}{62093.65\ psi} = 0.58$$

This was far below a safe range, so we targeted a safety factor of 2. Combining the bending stress formula with the desired safety factor, we solved for the required small diameter d:

$$d^3 = \frac{32Mn}{\pi S_y}$$

$$d = (\frac{32 \times (862.1\ lb{\cdot}in) \times 2}{\pi \times (36{,}259\ psi)})^{1/3} = 0.785\ in$$

We applied this new diameter to our CAD model to increase the safety factor to 2. The revised geometry is shown in Figure 2.

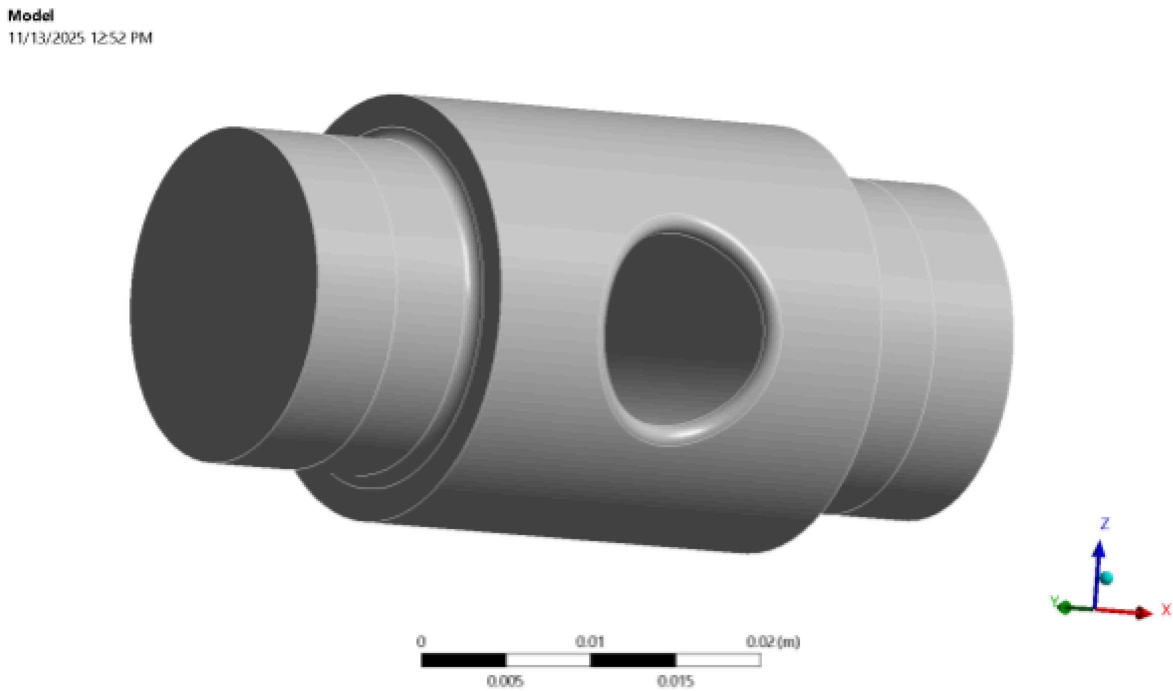

Figure 2. A depiction of the updated geometry with a larger end diameter.

Results of Changing the Geometry

Based on the FEA results of the updated geometry shown in Figure 3, we confirmed that the design modifications successfully achieved the intended improvements. In the original part, the fillet at the diameter transition experienced the highest bending stress and exceeded the material's yield strength, resulting in an unsafe design. After increasing the smaller diameter to reduce the stress concentration, the new results show that the maximum stress in this critical region is now comfortably below the pin's yield strength. This indicates that the modification effectively reduced the localized stresses. Furthermore, the overall stress distribution across the part is more uniform, and no unexpected stress concentrations were introduced by the design change.

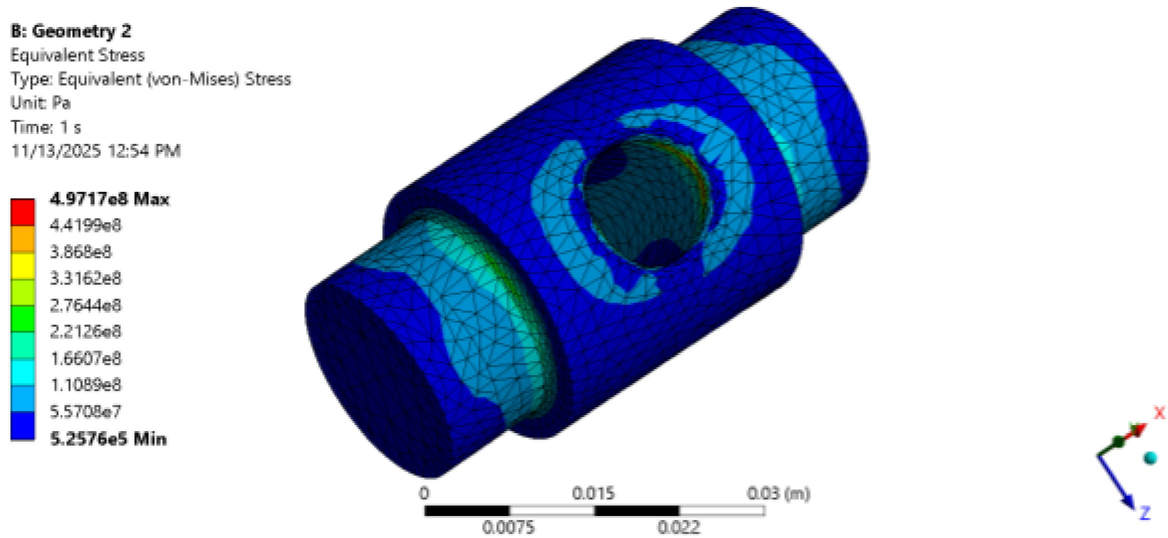

Figure 3. Equivalent stress results with the updated pin geometry.

Recommended Next Steps

We would recommend an operating load of approximately 700 lb, which is the load used in our analysis with a factor of safety of 2. At this load level, all of the calculated stresses—including those on the pin—remain below the allowable limits when the safety factor is applied. Because stresses scale linearly with load, increasing the operating load much beyond 700 lb would reduce the factor of safety below 2 and push the pin closer to yielding.

Some localized yielding may be acceptable, but our current design identifies the pin as the limiting component. Exceeding the 700 lb operating load would risk permanent deformation, so the jack is not suitable for lifting a full-sized car. However, it would be appropriate for lighter loads such as small vehicles, equipment, or lifting platforms.

If we were to make additional design changes, we would focus on strengthening the pin, since it is the component that limits the allowable load. We could increase its diameter even more or use a higher-strength material to raise the safe operating load. We would also aim to reduce stress concentrations by adding larger fillets, which would help lower peak stresses in critical areas. These changes together would allow the jack to safely support higher loads while maintaining our factor of safety of 2.