Finite Element Analysis Application Verification

Tension Loaded Bar with Symmetrical Fillets

Maddie Goldberg
Roderick Landreth

June 6th, 2018
Ashok Ramasubramanian
MER-214, Strength of Materials

Contents

List of Figures

1 Introduction

Examination of the SolidWorks Finite Element Analysis (FEA) tool in comparison to mathematical static analysis reveals how reliable such a program is for computing maximum stresses in materials. An accurate simulation measuring internal stresses and strains increases safety and reduces material waste. Safety factors calculated by such a simulation eliminate excess material, and predict material failure. In a professional sense, this means saving millions of dollars and saving lives.

Finite Element Analysis is a way of avoiding the use of near-impossible functions which often involve several partial order derivatives or huge systems of equations. Instead of this, Finite Element Analysis uses geometric systems of equations to approximate values at specific points in a mesh. The smaller the mesh, the more accurate the result; this is analogous to increasing the resolution of an image. The more pixels, the clearer the image becomes.

2 Materials and Methods

In SolidWorks, we first created a straight bar of known dimensions with no fillets. This first bar serves as a control specimen. We chose dimensions for ease of use in calculations, as for our purposes the dimensions are arbitrary. Figure 1 shows the unaltered bar. The 1000N tensile force was directed axially, normal to the 10mm by 50mm face.

Then, to prove whether SolidWorks is an accurate tool for stress analysis, we tested a tension loaded bar with symmetric fillets, as seen in figure 2. We evaluated the resulting stress of a force on the bar in two ways: through static analysis and through SolidWorks FE Analysis.

Another way to analyze this second sample is to use its inherent symmetry, using one half of the filleted sample to deduce the stress on the whole sample. In practice, this method is useful for analyzing complex systems with axes of symmetry, but comparing it to the static analysis revealed whether or not this method measures static system stresses with acceptable accuracy for industrial use. For this test, the filleted sample (figure 3) is halved along the axis of symmetry parallel to the application of force. We used SolidWorks SimulationXpress to recreate it for the FEA, as shown in figure 4.

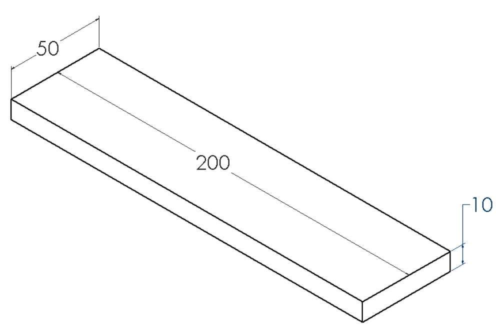

Figure 1: Baseline Test Bar (mm)

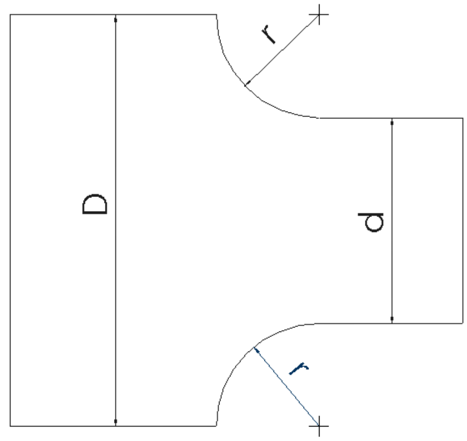

Figure 2: Symmetrically Filleted Bar Dimension Diagram

2.1 Static Analysis

Static analysis of the unaltered bar with a cross section of $5 * 10^{-4}m$ relied on using equation 1 (see Appendix A) to find the stress caused by the axially applied 1000N force. In this ideal case, the stress was assumed to be uniform, so the average stress was equivalent to the maximum stress.

In the case of a symmetrically filleted bar, with dimensions labeled as seen in figure 2, this assumption does not hold. Stress concentrations form in certain parts of the bar, so a stress concentration factor varying with the geometry of the bar revealed the maximum stress in these areas. This factor was multiplied by the average stress found by using equation 2 in the smallest cross section of the sample. The stress concentration factor, k, was found for the particular geometry using published data [1], relying on ratios of $\frac{r}{d}$ and $\frac{d}{D}$.

2.2 Finite Element Analysis

We conducted a finite element analysis of the unaltered bar with a cross section of 50mm x 10mm as a baseline test. Each sample needed a material for testing in simulation, so although it did not alter the results, each tested sample was generated from 'Alloy Steel'. SolidWorks SimulationXpress 2-dimensional analysis calculated the maximum normal stress (in MPa). While setting the mesh resolution low and slowly increasing it, the maximum stress values eventually varied less and less. The used values of stress were averaged from three mesh values, with the uncertainties being the standard deviation of the stress values. The credibility of SolidWorks relied on the comparison of this stress value to the static analysis. If both methods concur, a more complicated FE analysis can be conducted.

After the baseline bar sample stress matched the static analysis, we tested the symmetrically filleted bar to further measure SolidWorks' accuracy. Maximum stress values converged to a specific value as mesh size decreased. A comparison of maximum stress values – found through static analysis versus through FEA – of this more complicated sample revealed how close SolidWorks' FEA is to a static analysis when dealing with more complicated models. This tells us whether SolidWorks can derive meaningful information from a complex real-world material application.

An FEA using symmetry required a new sample, the plane of symmetry specified in the simulation. After fixing one end static and adding an axial load to the other, the mesh also altered the calculated maximum stress on the sample. The maximum stress varied less and less within a certain range of mesh resolution, denoting a reliable value of maximum stress to use.

3 Results

3.1 Static Analysis

The baseline test of a bar with constant cross sectional area of $0.0005m^2$ and an axial load of 1000N yields an average, and maximum, stress of $2MPa$ utilizing equation 1 For a sample calculation, see appendix .

The second test, a filleted bar with dimensions shown in figure 3 (with 10mm in depth) has a stress concentration factor with $\frac{r}{d} = 0.25$ and $\frac{D}{d} = 1.5$, producing a k value equaling 1.63 [1]. Using equation 2 (in appendix a) on the smallest cross section of the filleted sample produced a maximum stress of $4.89MPa$.

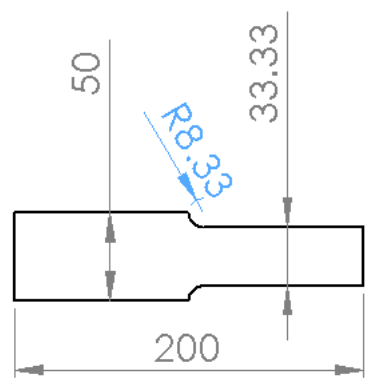

Figure 3: Symmetrically Filleted Bar with Tested Dimensions (mm)

3.2 Finite Element Analysis

The baseline 2-dimensional analysis with low mesh resolution displayed a maximum stress of $2MPa$. This is an ideal case, the strain being uniform throughout the bar, so the mesh size did not alter the results. In the more complicated second test with the dimensions given in figure 3, the calculated strain varied slightly with the mesh, presenting $4.89 \pm 0.01MPa$. Using symmetry in the SolidWorks FE analysis resulted in mesh resolution changing maximum strain found, as in the last test. The simulation provided a maximum value of stress equaling 4.91$\pm$ 0.1 MPa when using symmetry.

These values of simulated stress were taken as the strain converged within a range of mesh density. An analysis of the mesh was conducted on the sample using symmetry in

FEA, and figure 5 displays the percent variation from the static analysis as the mesh density changed. We found that the most accurate values for maximum stress were found at a high mesh resolution, between 2000 and 8000 elements for the symmetrically analyzed sample.

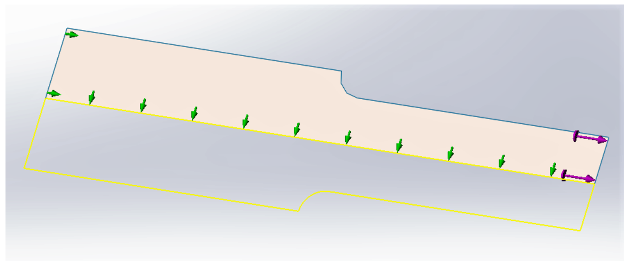

Figure 4: Symmetric 2-dimensional FEA with force in direction of red arrows

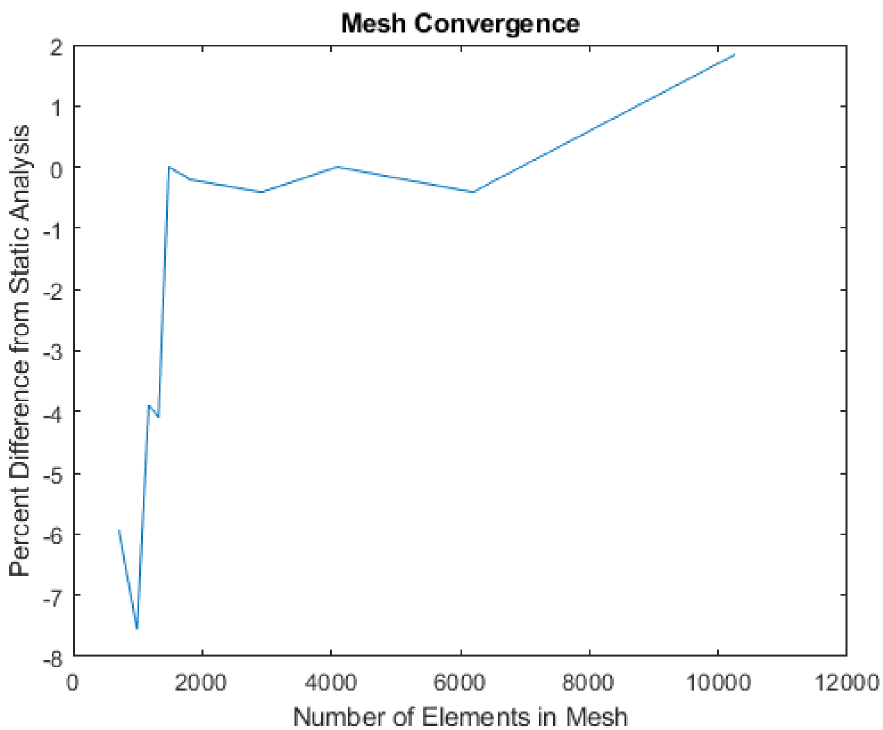

Figure 5: Mesh Convergence on Symmetrical Analysis of Filleted bar

3.3 Validation

Confirmation of the FEA maximum stress value accuracy lies in comparing the analytically calculated results to those of the FE analysis. Analytical methods based off the concepts in question detecting very similar results to simulated analysis establishes the accuracy of the simulation. From each experiment, both methods produced stress values within 0.1% of each other.

4 Discussion

The accuracy displayed by the SolidWorks FEA ratified that it is a reliable tool for quickly estimating material stresses. Rather than doing hours of calculations on complex shapes, using SimulationXpress offers a fast alternative. Factors of safety are always taken into account. Because of this, crucial structures are over-designed so that they are, in actuality, capable of bearing easily double or triple the allowable loads before yielding, so less than a percentile difference in analyzed stress will not matter. This simulation also provides additional information including what region will experience this maximum stress, and even the stress for any node in the mesh.

The symmetry fixture allows smaller pieces that can be more easily altered to represent the same sample, which reduces processing time but also reduces accuracy for analysis. While this is useful in either mechanisms with a safety factor or those that are not structurally imperative, if the structure is crucial this method will be less useful and less safe than analyzing the full design.

With an accuracy this high, comparing the simulated stresses and strains to the real life examples will vary more unpredictably with the deformations and defects within the material of the sample as well. The simulation uses set materials, but does not account for manufacturing accuracy, conditions of use, or (for example) discontinuities in crystallographic micro-constituents, or other factors which may change properties of the material.

5 References

[1] Young, W., Budynas, R., Sadegh, A. Stress Concentration. In: *Roark's Formulas for Stress and Strain. 8e.* McGraw Hill, 2012.

Appendix A Equations

$$\sigma = \frac{P}{A} \tag{1}$$

Where σ is the applied stress, P is the axially applied force, and A is the cross sectional area that the force is acting on.

$$\begin{gathered} \sigma_{max} = k\sigma = k\frac{P}{A} \\ where \;\; k = \frac{\sigma_{max}}{\sigma_{avg}} \end{gathered} \tag{2}$$

Appendix B Sample Static Analysis Stress Calculation Technique

$$P = 1000N$$

$$A = (10mm) * (50mm) = 5 * 10^{-4}m^2$$

(From Equation 1)

$$\sigma = \frac{P}{A} = \frac{1000N}{0.0005m^2} = 2.0MPa$$