ME3990 – Computational Solid Engineering

Final Project Report

December 5, 2025

Adaptive Mesh Convergence and Static Simulation of a Recurve Bow using Abaqus

---

Submitted by:

Jimmy Pare & Camden Piper

ABSTRACT

This study aimed to examine the stress and deflection experienced by the limbs of a recurve bow due to the load applied by the user via the bowstring. Abaqus was used to model the bow's reaction to forces applied to the bow's limbs, which mimicked the load that would realistically be applied by the drawstring. The secondary goal of this study was to explore mesh optimization in Abaqus for the purpose of effectively balancing model accuracy and computational cost for both this study and future studies. The model utilized a static, general step and employed Abaqus' adaptivity remeshing feature to mesh the assembly. Loads and boundary conditions applied to the bow resulted in moderate to high stresses on the limbs, which is expected for this application. Deflection of the limbs also aligned with expectations; noticeable deformation was observed, but no excessive results. Overall, this study gave valuable insight to the mechanics behind archery, as well as methods to produce more accurate finite element analysis simulations.

INTRODUCTION

Bows are used for hunting, competition, and hobby activities widely across the world. Different variants and designs are used for different purposes, but this study focuses on a recurve bow. A recurve bow was chosen over a compound bow due to the simplified mechanical function and geometry as well as reduced computational cost. The motivation behind this study is both personal interest as well as validating the safety of modern recurve bows. Draw weights range from 10 lbs all the way up to 70 lbs or more. With 70 lbs of tension in an elastic material, validation of safety is crucial due to risk of injury. If a bow limb were to snap while drawing, the user could be severely injured. The maximum displacement and material stress were of interest to this study to ensure user safety.

This study focuses on the displacements and stresses on a recurve bow during the ’drawing’ phase, when the user is pulling the bow string back. The specifics for how this is simulated are discussed in the Methodology section of the report. Appropriate assumptions and simplifications are also discussed in this section.

The displacement results are heavily dependent on the assumed modulus of elasticity for the limbs. The limbs are assumed to be isotropic to limit computational cost. This study uses a relatively low modulus of elasticity for the limbs to try to simulate the worst possible scenario. A low modulus of elasticity means that the simulated bow limbs are more flexible than a real world scenario.

Confidence is held in the mesh used to model the recurve bow. The mesh was refined to the model using an adaptivity process feature in Abaqus, discussed in the Methodology and Results sections.

STATEMENT OF OBJECTIVES

The specific objectives of our final project are to use the FEA software Abaqus to examine the stress and deflection for the limbs of a recurve bow when pulled to full-draw, and to explore methods to optimize the meshing of parts and assemblies effectively.

METHODOLOGY

The computational methods used within Abaqus highlight what makes this study unique. While the model itself is simple, the meshing of the assembly showcased the powerful programming working behind the scenes in the Abaqus software.

The bow model consisted of two parts, a riser and two identical limbs, that were imported from an external source. These parts were composed of different materials and material properties, shown below.

Part	**Material**	**Elastic Modulus**	**Poisson's Ratio**
Limb	Fiberglass	1.5e6 psi	0.35
Riser	6061 Aluminum	10e6 psi	0.33

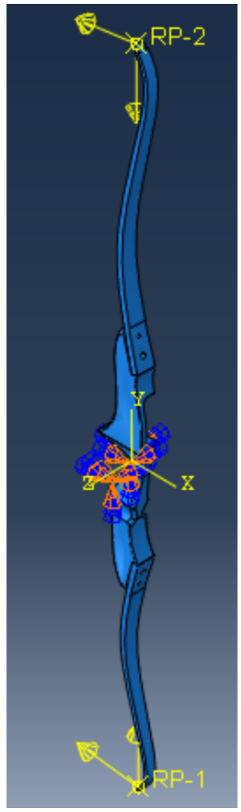

Figure 1: Bow assembly components shown with loads and boundary conditions.

A static, general step was used for this study as the goal was to examine the bow at full-draw, therefore it was sufficient. To build the assembly, a limb was fixed to both the top and bottom of the riser, constrained by a tie interaction with zero degrees of freedom. Loads consisting of horizontal and vertical (CF1 and CF2) concentrated forces were placed on the midpoints of the limb tips, and distributed across the surface of the limb tips with a kinematic coupling. CF1 and CF2 magnitudes were set to 39.9 lbs and 42.5 lbs, respectively, which corresponds to about 80 lbs of draw weight. Draw weight is the force the user actually pulls with, and 80 lbs is very heavy in practice. The riser was constrained in all degrees of freedom with an ENCASTRE boundary condition. Loads and boundary conditions on the assembly are shown in Figure 1. To mesh the part, Abaqus' adaptivity remeshing feature was utilized. Here the software meshes the assembly such that the mesh maintains desirable smoothness, skewness, and aspect ratios for all geometries in the model. Then, it iteratively refines the mesh and runs the job based on the previous iteration's results.

RESULTS

Simulation of the bow in Abaqus provided a useful perspective into the mechanics and possible failure mechanisms that may not be initially obvious.

To ensure a quality mesh at a reasonable computational cost, the adaptive remeshing feature was used to optimize the mesh. Five iterations of remeshing were performed, and the total strain energy (ALLSE) was plotted versus mesh element count, shown in Figure 2.

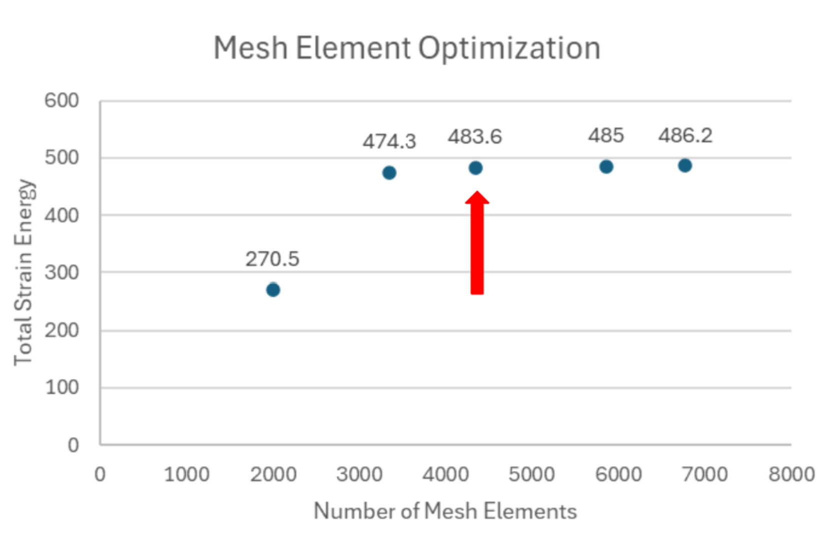

Figure 2: Convergence of total strain energy versus mesh count.

Total strain energy was considered to be converged when the change in ALLSE from one iteration to the next was less than one percent, which corresponded to a mesh containing 4349 elements. The initial iteration and final iteration meshed models are shown in Figure 3.

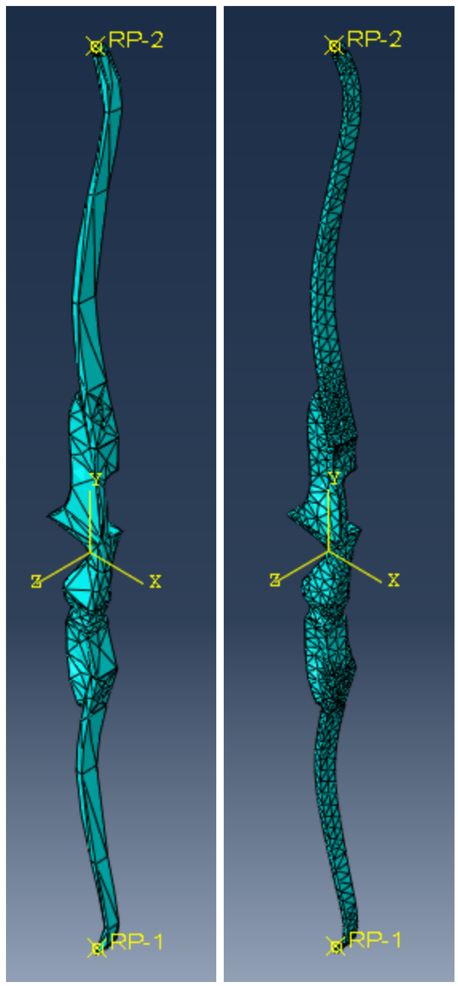

Figure 3: Initial and final meshed parts produced from adaptivity remeshing procedure.

Running the adaptivity remeshing process creates and performs a job for each mesh iteration, so the first converged iteration's results were used. The model yielded a maximum stress of 52.38 ksi, found at the location where the limbs are tied to the riser. Maximum deflection was found at the tip of the limbs, with a magnitude of 12.12 inches. Von Mises stress and deformation plots are shown in Figure 4.

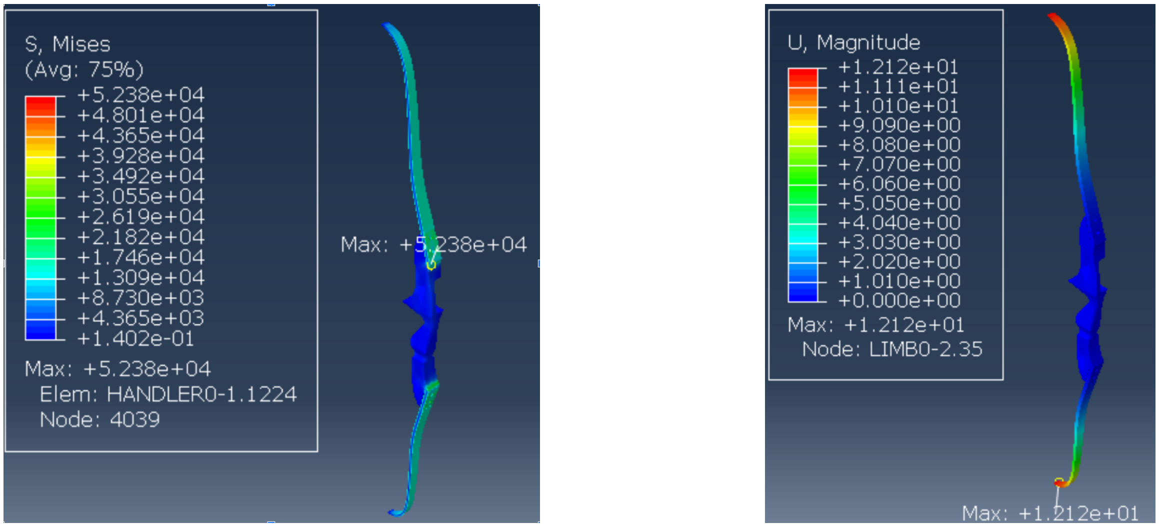

Figure 4: Simulation results showing stress distribution and deformation.

DISCUSSION

The results for stress in the bow limbs were close to expectations. For the majority of the limb's length, a stress of about 20 ksi was observed. This is well below the yield strength of about 150 ksi in tension assumed for fiberglass, which is a composite. It should be noted that due to fiberglass being a composite material, its strength varies drastically based on composition as well as direction. Results indicate that the failure point of the bow will occur where the limb is attached to the riser, which experienced a stress of 52.4 ksi.

Deflection of the bow limbs was large not but unexpected. Because fiberglass is a composite with varying properties, a low-end value for elastic modulus was used to account for the worst anticipated results. Even here, the part did not fail, but limb tip deflection of 12.12 inches is not very realistic for most recurve bows. Further simulations indicated that sweeping the range of elastic moduli could provide results closer to most bow deflections, but the elastic model was used to be conservative.

A unique but powerful characteristic of adaptivity remeshing method compared to the global or part meshing method is that the mesh is further refined at complex geometries and features. This results in total strain energy converging at a higher values, which indicates the model is more accurate. Also, convergence occurs with less mesh elements, indicating lower computational cost. These qualities are important for complex models, and the method should be applied wherever it is possible.

CONCLUSIONS

The major conclusions of this project are:

- A maximum von Mises stress of 52.4 ksi is present where the each limb attaches to the riser.
- The maximum displacement is heavily dependent on the assumed Young's modulus of the limbs. For this study, a maximum displacement of 12.1 inches is observed at the top and bottom points of the limbs.
- Adaptive mesh processing can be used to efficiently refine complex meshes in Abaqus.

Further research can be conducted to more accurately simulate the composite fiberglass material in the bow limbs, as well as modeling the bolt interaction between the riser and bow limbs. Work could also be done to simulate the force changing angle with the string, although this falls behind the previous items in order of priority.

The results of this study can help bow manufacturers by indicating the maximum displacement can be controlled by the fiberglass material. With a low modulus of elasticity, a very large displacement can be obtained.

BIBLIOGRAPHY

1. October Mountain Products, “Explorer CE Recurve Bow.” Available at: `https://www.octobermountainproducts.com`.

2. CenterPoint Archery, “Recurve Hunting Bows.” Available at: `https://www.centerpointarchery.com`.

3. Bowhunter Magazine, “How to Quick Tune Your Traditional Bow.” *Bowhunter*. Available at: `https://www.bowhunter.com`.

4. BowTech Archery, “Carbon One Bow.” Available at: `https://www.bowtecharchery.com`.

5. Pacific Institute for the Mathematical Sciences (PIMS), “Applied Algebraic and Geometric Structures – Past CRG.” Available at: `https://www.pims.math.ca`.

6. GrabCAD, “Takedown Bow – CAD Model.” GrabCAD Library. Available at: `https://www.grabcad.com`.