Proceedings of the National Conference
On Undergraduate Research (NCUR) 2016
University of North Carolina Asheville
Ashville, North Carolina
April 7 – 9, 2016

Validation of a Radius Bone Model Using Finite Element Analysis Under Single Cycle Bending Load

Tulsi Patel
Department of Biomedical Engineering
Widener University
One University Place
Chester, Pennsylvania 19013 USA

Faculty Advisor: Dr. Anita Singh

Abstract

In this study we aimed to create and validate a computational model of the radius bone to characterize its behavior under mechanical loading. This enables the circumvention of predictive uncertainties in an experimental setting and potentially reduces cost and effort. First, a three – dimensional model of the human radial bone was developed using SolidWorks. Subsequently, the model was validated with Finite Element (FE) analysis (ABAQUS) where a flexural test with an implicit integration scheme under transient loads and constraints was replicated to simulate the experimental results. Successively, verified models were used to calculate the response under loading after the addition of fixation plate using screws interface. Currently this research is in its preliminary phase, although a working model under the forearm plate and screws interface will assist in testing optimized designs of the compression forearm plate commonly used during forearm fixation orthopedic surgeries.

Keywords: Computational model of Radius bone, Finite Element analysis, 4 – pt flexural test

1. Introduction

According to the National Hospital Amubulatory Medical Case Survey[1], distal radial fractures are the most common fracture types with over 0.5 million reported cases with an annual cost of $240 million for internal fixations. Forearm fractures include but not limited to open and compound fractures, dislocations, segmental fractures, and isolated[2]. Most of these fractures occurs when the forearm undergoes bending and twisting (torsion) motions requiring surgical assistance as part of the treatment[2]. Many different fixation methods exist for forearm fractures, including such as limited contact dynamic compression plates, locking compression plates, and point contact fixations. These systems are effective although, they can limit blood flow to the fracture site or even inhibit osseointegration, presenting a need to improve bone fracture fixation techniques[4].

Computational methods serve as a surrogate to *in vitro* laboratory experiments, providing an efficient and inexpensive technique of performing various biomechanical tests on long bones to obtain similar results. There are two aspects to using these techniques, modelling and simulating. Modeling involves creating an accurate representation of the design in question. The models are typically imported in software designated for simulations along with input parameters and boundary conditions or constraints. The simulation software solves the problem, typically requiring large amounts of numerical calculations using numerical analysis. Such methods can provide a deeper understanding and analysis of complex problems to reconstruct or deduce future behavior.

One of the most commonly used simulation method is the Finite Element method (FEM), which is a numerical approach to approximate solutions to boundary value or constraint problems. FEM initially breaks a problem into smaller elements to perform calculations and then reassembles the elements into one large system to simulate the behavior of the entire model. This allows for capturing the local effects of the constraints on complex geometries as well as a view of the overall solution. The use of computational methods permits one to completely understand the behavior of the radial bone under mechanical loads and assist in predicting its performance with plate and screws interface. However, before the radius bone's response is projected using a computational model, the model's accuracy must be supported by experimental results. In this study, we created a validated radius bone computational model that can predict the plate responses accurately and efficiently.

2. Methods

2.1. Modelling

The radius bone was modeled using SolidWorks (SW), a computer aided design (CAD) program. Dimensions of the bone were obtained from an anatomically equivalent radius surrogate sawbone[5]. Sawbones provide orthopaedic models of various bones including the radius bone based on specific material and size. In order to correctly model the radial tuberosity and styloid process, two orthogonal views were imported into SW and aligned at the center in a Cartesian plane manner as shown in Figure 1.

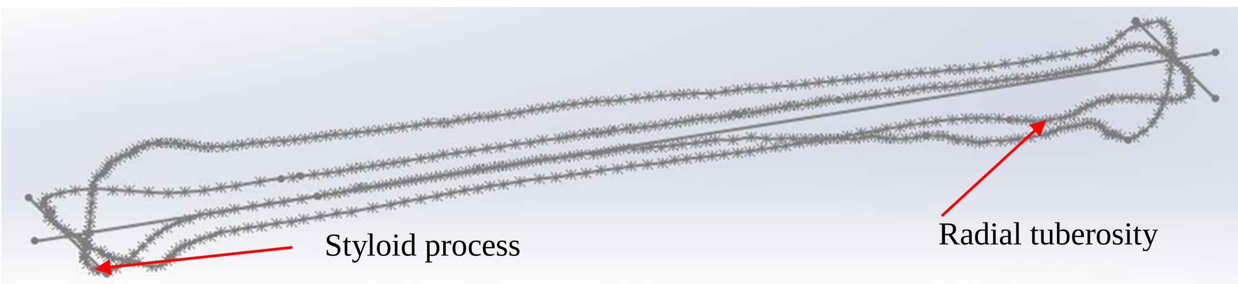

Figure 1. Orthogonally aligned wire frame of the radius bone

Following the wire frame construction as shown in Figure 2, elliptical rings were manually traced to fit the outer edges of both sketches ensuring that the model has a smooth and contoured outer surface. Given the skeletal drawing, the model was extruded through the rings and depicted in Figure 3.

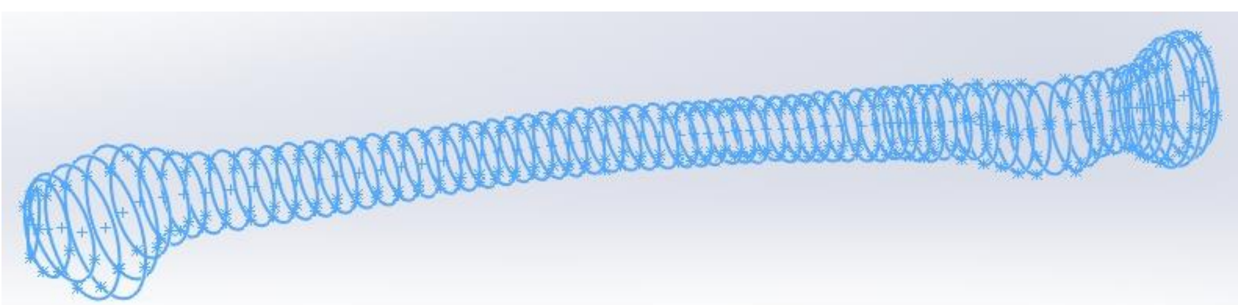

Figure 2. Ellipses created from the wire frame for defining the surface contour

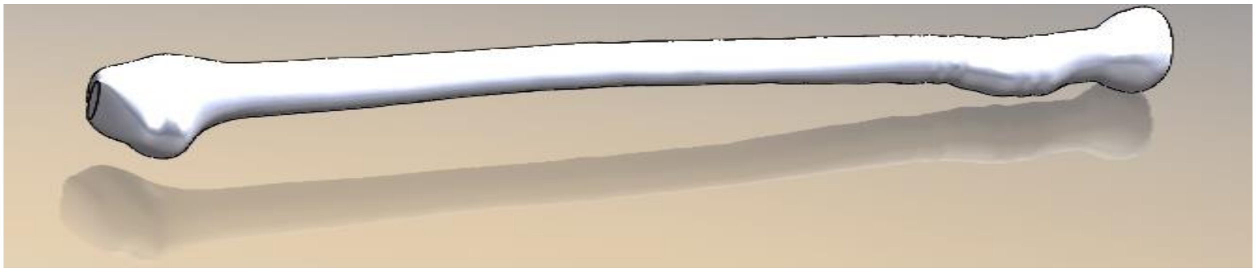

Figure 3. Complete SolidWorks model of the radius bone

2.2. Simulation

Once the model was developed, the next step was to create a simulation and performing the analysis. ABAQUS® is a software application that uses FE analysis techniques on pre–processed models to solve computer aided engineering (CAE) problems. In addition, ABAQUS offers a visual representation of the FE analysis results. We imported the radius bone model from SW into ABAQUS and entered the material properties. The model was defined with a Young's modulus of 350 MPa and a Poisson's ratio of 0.34[6]. The plastic yield stress used was set at 2.65 MPa, as obtained from the experimental data from Rowan University's Forearm Plate Designs Engineering Clinic. To incorporate non–linearity, contoured surface with irregular geometries, the model was meshed with 15,180 tetrahedral (Tet) elements (Figure 4) rather than a cubic elemental shape.

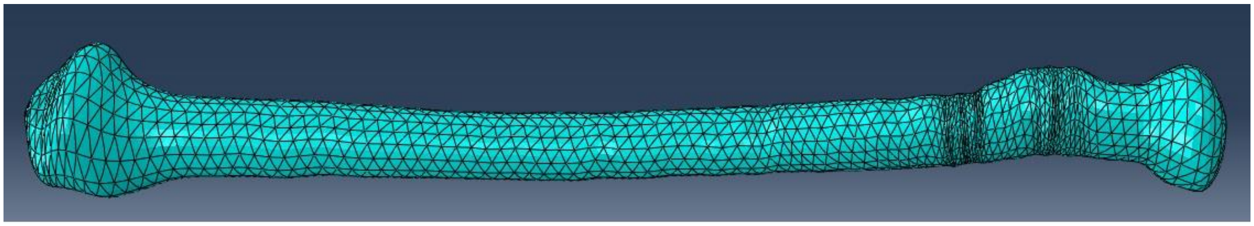

Figure 4. Tetrahedral meshed bone pre – loading

Furthermore, the boundary conditions were set to imitate the experimental 4–point bending loads and displacements. Figure 5 illustrates the simulated bend test had 6 steps, where each step had 2 frames increasing in displacement until the fracture point was reached. Table 1 shows the data corresponding to the simulated bend test.

Table 1. FE results of load and displacement for 4 point bending

Step	Displacement (m)	Load (N)
0	0.000169	35.6105
1	0.000335	70.5889
2	0.000671	141.388
3	0.001340	282.355
4	0.002680	564.711
5	0.005340	1125.21

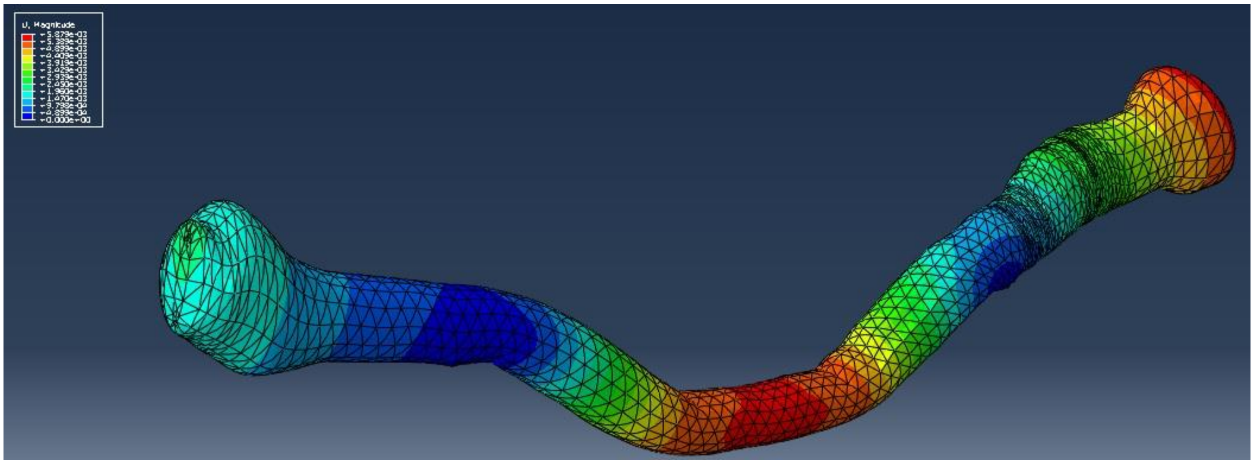

Figure 5. Displacements to 32 cm at the center under 4 point flexural test

3. Results and Discussion

To verify the model, the simulation results were compared to experimental data of 5 specimens provided. Figure 6 shows the elastic portion of the experimental data with an average slope of 215.2 kN/m.

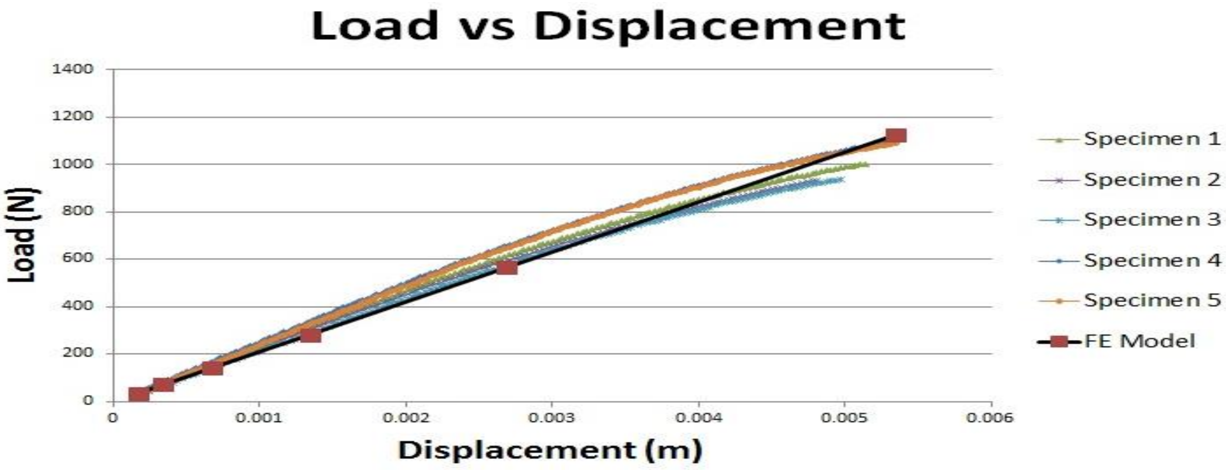

Figure 6. Comparison of experimental and FE results from 4 point. bending tests

The simulated FE results are in agreement with the experimental observations with slope of 210 kN/m as shown in Figure 6. At the given displacements over 30 seconds, the reaction forces matched the measured loads with 2.09 % accuracy.

4. Conclusion

Findings from the experimental studies and our model indicate that the model is able to simulate the 4–point bending responses with very high accuracy and offers the opportunity to further explore the model for simulating torsional load outcomes.

5. Future Work

With this model validated, the next step is to simulate torsional load and compare results with experimental data. Next, the fixation plate and screws can be modeled and tested using a similar approached to achieve the optimal design. The final phase would be to set up collaborations with surgical manufactures to consider further development of clinically applicable designs.

6. Acknowledgements

I would like to thank Dr. Anita Singh for allowing me to work on this project and guiding me through the course of the project. We would like to thank Euver Joaquin Castillo for collecting experimental data, Brian Wisner for helping with SolidWorks modelling and Dr. Xiaochao Tang's insights on the application ABAQUS. Finally, we would like to thank the Widener University science division for all their support and enthusiasm on this project.

7. References

1. "Ambulatory Health Care Data." *Centers for Disease Control and Prevention*. Centers for Disease Control and Prevention, 23 Sept. 2015. Web. 7 June 2016.

.2. Kakalala, Gopikrishna. "*Forearm Fractures Treatment & Management.*" Medscape, 29 Sept. 2015. Web. 29 Sept. 2016.

3. Chung KC, Spilson SV. The frequency and epidemiology of hand and forearm fractures in the United States. *J Hand Surg [Am]*. 2001 Sep. 26(5):908-15

4. Sod, G.A., Mitchell, C.F., Hubert, J.D. Martin, G.S., and Gill, M.S. In Vitro Biomechanical Comparison of Locking Compression Plate Fixation and Limited-Contact Dynamic Compression Plate Fixation of Osteotomized Equine Third Metacarpal Bones. (2008).

5. "Radius #1018" *Sawbones*. N.p., n.d. Web. 07 June 2016.

6. Trimed Inc. Sawbones and Composite Materials Properties.