Marquette University
e-Publications@Marquette

Biomedical Engineering Faculty Research and Publications

Biomedical Engineering, Department of

8-1-2015

Validation of a Finite Element Model of the Humerus for Fracture Risk Assessment During Assisted Ambulation

Jessica M. Fritz
Marquette University, jessica.fritz@marquette.edu

Prateek Grover
Marquette University

Nicole M. Grosland
University of Iowa

Carolyne Albert
Marquette University, carolyne.albert@marquette.edu

Linda M. McGrady
Marquette University

See next page for additional authors

Published version. Published as part of the proceedings of the conference, *American Society of Biomechanics 39th Annual Meeting*, 2015. Publisher Link.

Authors
Jessica M. Fritz, Prateek Grover, Nicole M. Grosland, Carolyne Albert, Linda M. McGrady, Mei Wang, and Gerald F. Harris

Back to ToC

VALIDATION OF A FINITE ELEMENT MODEL OF THE HUMERUS FOR FRACTURE RISK ASSESSMENT DURING ASSISTED AMBULATION

[1,2] Jessica M. Fritz, [1,3] Prateek Grover, [4] Nicole M. Grosland,
[1,5] Carolyne Albert, [2] Linda McGrady, [1,2] Mei Wang, and [1,2,5] Gerald F. Harris

[1] Marquette University, Milwaukee, WI, USA
[2] The Medical College of Wisconsin, Milwaukee, WI, USA
[3] Washington University School of Medicine, St. Louis, MO, USA
[4] The University of Iowa, Iowa City, IA, USA
[5] Shriners Hospitals for Children, Chicago, IL, USA
email: jessica.fritz@marquette.edu

INTRODUCTION

Finite element (FE) modeling provides an established, useful tool for assessing biomechanics without invasive testing. One important and emerging application is the use of FE models to assess the risk of long bone fractures, which are prevalent in populations with bone fragility due to aging, osteoporosis, and various genetic and metabolic disorders. For example, FE modeling can be used to assess bone strength and fracture risk in individuals with osteogenesis imperfect (OI), a heritable bone fragility disorder [1, 2]. Children with OI have an especially high prevalence of long bone fractures. As the major load-bearing bones during ambulation, current models of OI have focused on lower extremity bones [1, 2]. However, the upper extremity bones, including the humerus, experience weight-bearing loads during assisted ambulation with crutches and walkers. Analysis of children walking with instrumented Loftstrand crutches showed they experienced shoulder forces up to nearly 35% of their body weight [3]. It has also been shown that assisted ambulation can cause a 24Nm bending moment at the shoulder joint [4]. However, to date, no study has examined how loads sustained during assisted mobility may put patients with OI at risk for upper extremity fractures.

This project focused on the development and validation of a humeral FE model subjected to physiologic bending loads such as those seen during assisted ambulation [3, 4]. This model is appropriate for application to various populations at increased risk for humeral fractures who use assistive devices for ambulation.

METHODS

Fourth-generation composite humeri (named H-VHP model; Sawbones Worldwide, Pacific Research Lab, Inc., Vashon, Washington, USA) were manufactured for this project based on the humerus geometry of the National Institutes of Health (NIH) Visible Human Project (VHP).

A composite humerus was instrumented with two stacked rectangular strain gage rosettes (Vishay Micro-Measurements, Raleigh, North Carolina, USA) and subjected to bending loads. The center gage of each rosette was aligned with the anatomic axis on the anterior and posterior side of the humerus. A four-point bending setup was used, in which the support rollers were on the anterior side, 92 mm distally and proximally from the gage rosettes. Loading was applied on the posterior side; each roller positioned 28 mm on either side of the gage rosette (Fig. 1a). The humerus was loaded with 200 N through a servo-hydraulic materials testing machine (MTS 809, Eden Prairie, Minnesota, USA). This load was selected in order to simulate a 24 Nm bending moment about the shoulder joint axis that an individual would experience during assisted ambulation with Loftstrand crutches [4]. Strains were recorded from each gage and the principal strains were calculated for each rosette.

The testing conditions were then replicated in a three-dimensional (3D) FE model of the humerus (Fig. 1b) and processed in Abaqus 6.10 (Dassault Systèmes Americas Corp.; Waltham, MA). The 3D model geometry was defined from a CT scan of the composite humerus and then imported into IA-

Back to ToC

FEMesh software [5]. The humerus was meshed with 59,852 eight-noded linear hexahedral elements (C3D8) with an average element size of 1.25 mm. Meshing of both the cortex and cancellous layers of the humerus was performed using the multi-block approach in IA-FEMesh [5]. Cortical bone was defined with a Young's modulus (E) of 10.6 GPa (based on experimental data of tensile testing) and a Poisson's ratio (ν) of 0.3 (from composite bone manufacturer). Cancellous bone regions were assigned material properties provided by the composite manufacturers: E = 0.160 GPa and ν = 0.26. A set of elements corresponding to the locations of each strain gage rosette was defined. These elements were used for the FE model analysis in order to correspond with the testing data since the rosette locations may not coincide with overall maximum and minimum principle strain locations.

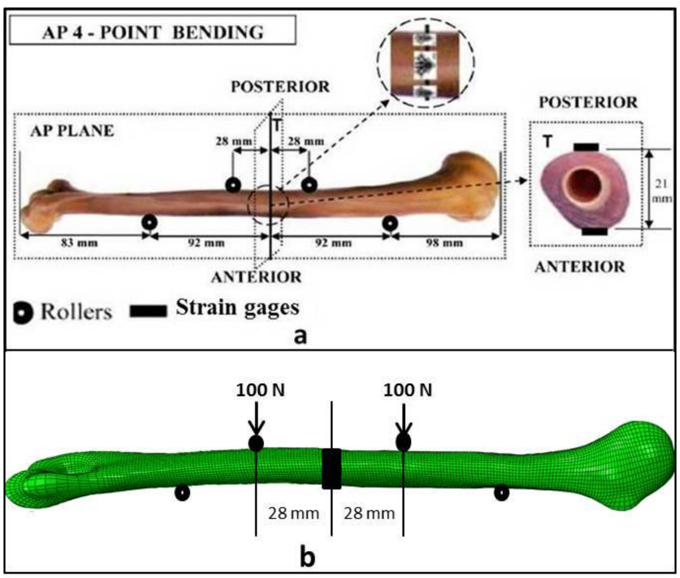

Figure 1: Four-point bending of humerus models during a) experimental bending test of the composite humerus in the anterior-posterior (AP) plane and b) finite element (FE) model analysis. The FE model also shows the location of the strain gages (SG) from experimental testing.

RESULTS AND DISCUSSION

The FE analysis showed excellent agreement with the experimental data during AP bending in maximum principal strain with a difference of 1.4% (Table 1). The difference in minimum principal strains between the experimental and FE results showed good agreement with a difference of 10.8% (Table 1). Limitations in this study include some variability between the three experimental trials and the inherent assumptions in FE modeling and analysis. Despite these limitations, there was good agreement between the FE model results and experimental data.

CONCLUSIONS

The high level of agreement between the two results validates the use of the FE model of the humerus to examine biomechanical responses to loading. This model has potential applications in populations such as the elderly or those with OI. These populations may use assistive devices for ambulation and would, therefore, be loading their upper extremity long bones, such as the humerus, with a percentage of their body weight.

REFERENCES

1. Fritz JM, et al. *Med Eng Phys* **31**(9), 1043-1048, 2009.
2. Caouette C, et al. *J Musculoskelt Neuronal Interact* **14**(2), 205-212, 2014.
3. Slavens BA, et al. *J Biomech* **44**, 2162-2167, 2011.
4. Grover P, et al. *Proc Inst Mech Eng H* **225**, 1169-1176, 2012.
5. Grosland NM, et al. *Comput Methods Programs Biomed* **94**, 96-107, 2009.

ACKNOWLEDGEMENTS

Funding for this work was provided by the following NIDRR sponsored grants: ARRT grants H133P080005 and H133P100008 and RERC grant H133E100007. Contents do not necessarily represent the policy of the Department of Education, and you should not assume endorsement by the federal government.

Table 1: Maximum and minimum principle strains for the finite element (FE) model and experimental testing.

Applied Load	Max Prin με FE Model	Max Prin με Experimental	Max % Difference	Min Prin με FE Model	Min Prin με Experimental	Min % Difference
200 N	1414	1394	**1.4**	-1134	-1271	**10.8**