Dartmouth
Biomedical Engineering
Center *for Orthopaedics*

Optimal Surgical Plating of Mandibular Angle Fractures: A Validated Finite Element Model

Brett Seeley-Hacker
Thayer School of Engineering, Dartmouth College, Hanover, NH
brett.l.seeley-hacker.18@dartmouth.edu

THAYER SCHOOL OF
ENGINEERING
AT DARTMOUTH

Introduction

Internal fixation via a surgical mini-plate is the prevailing treatment employed by craniomaxillofacial (CMF) surgeons dealing with a mandibular angle fracture, which is a common type of jaw fracture with relatively high complication rates[1]. The plate serves to counter the zones of tension present in the jaw *in vivo*, and there are two commonly accepted locations for plating: the 'Champy' plate along the superior oblique ridge, and a lateral border plate, as shown in Fig. 1[2].

Medical device manufacturers produce a variety of mini-plates, but they offer only limited advice to surgeons concerning the optimal plate choice, so the decision is largely based on a physician's opinions and preferences[3]. At the same time, it has been shown that plating methods are linked to complication rates and patient outcomes[4], so there exists a need for a standardized method by which physicians may select the optimal plating technique given a specific patient's anatomy.

Modeling the jaw is challenging due to complex *in vivo* biomechanics, so finite element analysis (FEA) has become a well established tool for producing accurate results[1]. While several studies have used physical experimentation to validate finite element models of the mandible, there exists no physically validated model of a jaw fractured and plated along the mandibular angle.

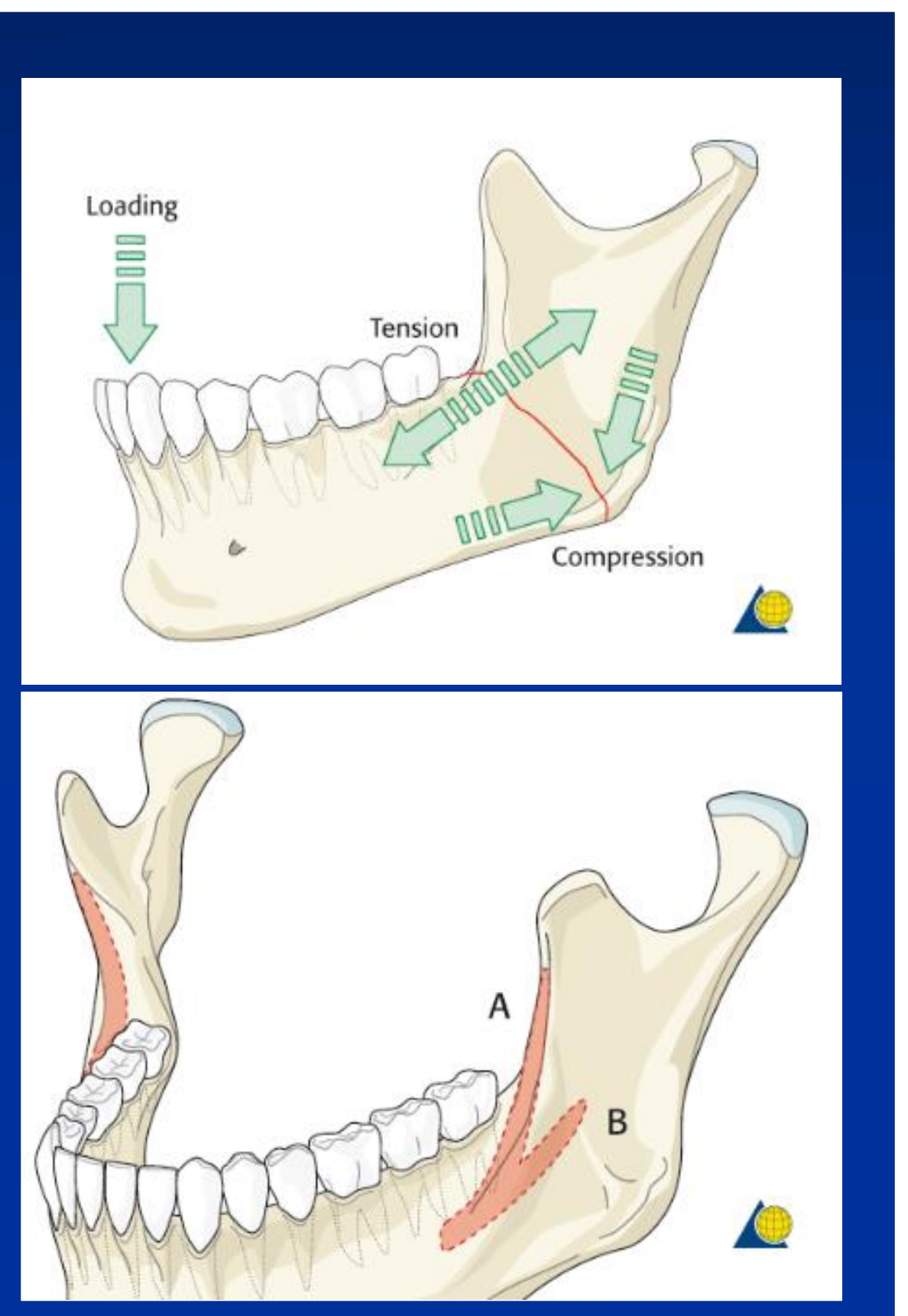


Figure 1: Top: Hemimandible showing the zones of tension and compression in the angle during in vivo loading Bottom: Ideal lines of osteosynthesis along which plates are placed; the superior oblique ridge (A) and the lateral border (B)[5]

Methods

Physical Benchtop Model

- Determined resultant muscle force using the Greaves' model and vector summation
- Cut standardized angle fractures into Sawbones cortical foam mandible models (SKU# 1338; See Fig. 2) and marked them for video tracking with scale bars and contrasting dots spanning the fracture
- Plated four jaw models, plating two for each line of osteosynthesis
- Constructed a rig for orienting the jaw, comprising three supports and a piston to simulate *in vivo* loading (See Fig. 3), and fastened rig to Instron 8501 servohydraulic machine
- Using the Instron WaveMatrix Dynamic Testing software, ran a test method ramping to loads of 200 and 400 N at a rate of 20 N/s
- Filmed tests from the side and the top under LED illumination using two Canon Vixia HD cameras mounted on the Instron
- Measured model deformation via video analysis using the open-source Tracker software

Figure 2: Sawbones model jaw used in experimental testing[6]

Finite Element Model

- Scanned Sawbones model into a mesh file using 3D Systems' Sense Handheld Scanner
- Refined mesh and inserted teeth and fracture using Geomagic Studio 2014
- Converted mesh to solid body step file using InStep (v2.3.11)
- Imported step file into SolidWorks 2017 and virtually plated fracture along each line of osteosynthesis
- Transferred assembly to ABAQUS CAE 2017 FEA software to run static, elastic body analyses of plated jaws using:
  - Ten node tetrahedral elements (C3D10), averaging 91,340 per model
  - Fully constrained boundary conditions at the teeth and condyles
  - Homogeneous isotropic solid material sections, with properties shown in Table 1
  - Tied constraint interactions between plate, screws, and bone
  - Loads along the same vector as physical testing

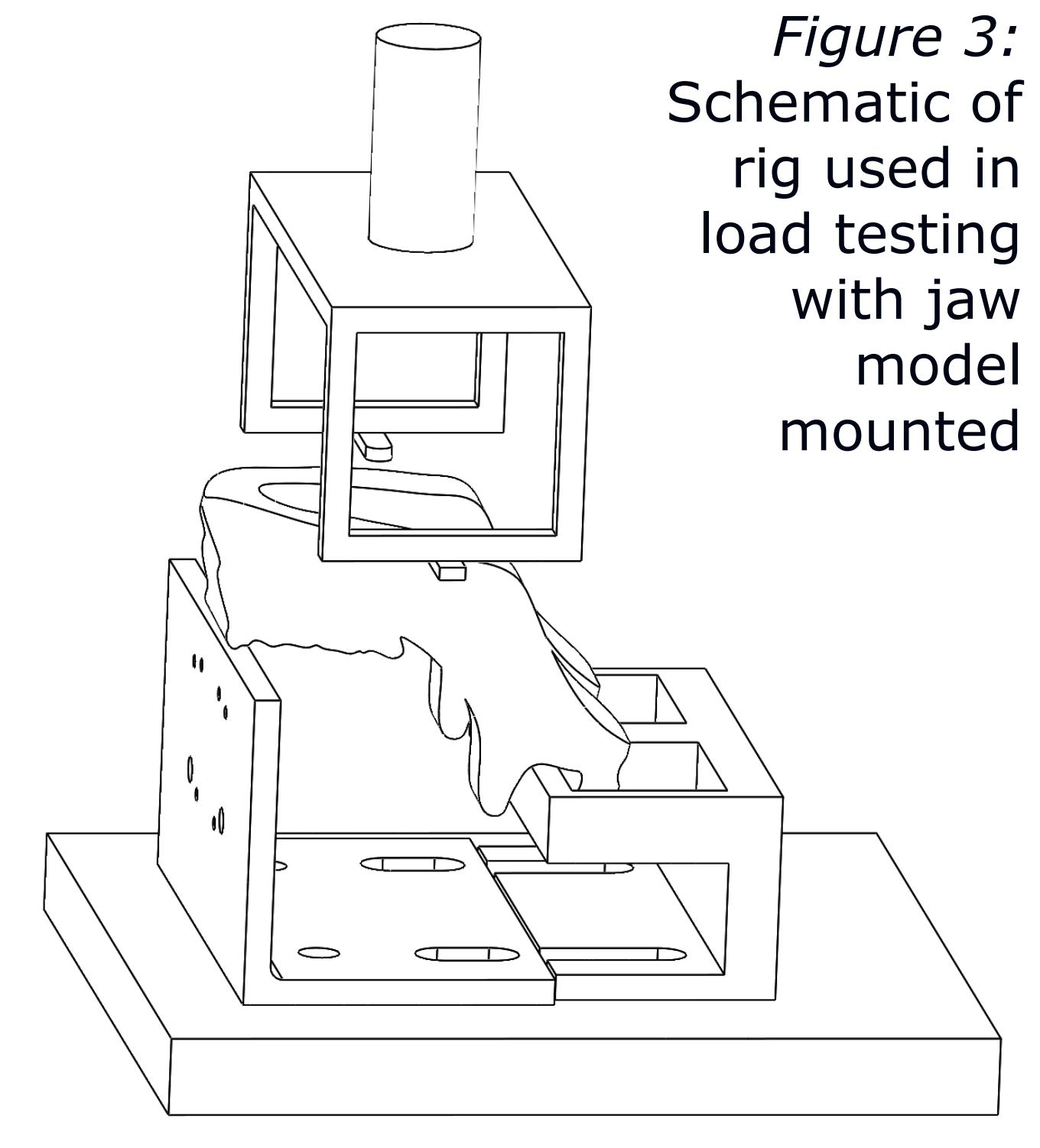


Figure 3: Schematic of rig used in load testing with jaw model mounted

Table 1: FEA Material Properties

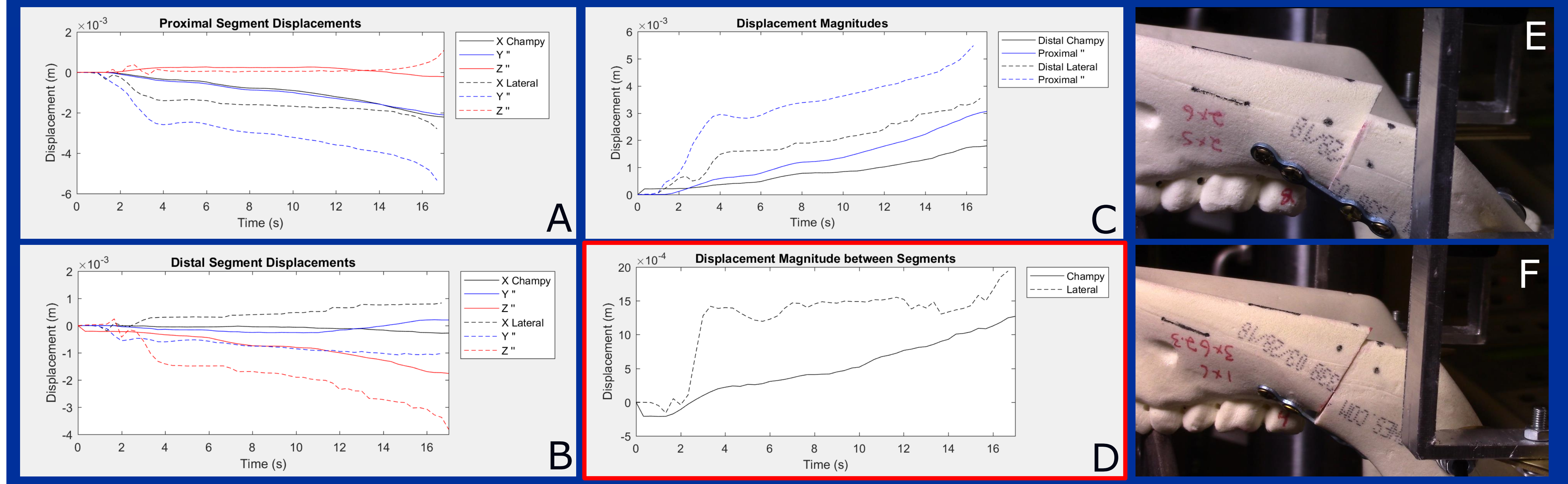
Material Property	Bone	Titanium
Elastic Modulus (GPa)	14.0	140.0
Poisson's Ratio	0.3	0.34

Research Goals

The long-term goal of this research is to produce a decision-making tool to guide physicians in the selection of the optimal plating method for a mandibular angle fracture. To that end, the current project seeks to develop the first physically validated, clinically relevant finite element model of angle fractures, so that plate performance may be accurately and efficiently investigated *in silico*.

Results

Physical Testing



The physical testing, summarized in Fig. 4, showed that lateral border plates result in larger displacement in every plane, and between fracture segments than do 'Champy' superior oblique ridge plates under the same load.

Figure 4: Plots showing various displacements of physical models under 400 N of load (A-D) and representative frames from video tracking footage showing both tracking markings and notable displacement under 400 N of load for a lateral plate (E) and a Champy plate (F)

Finite Element Analysis

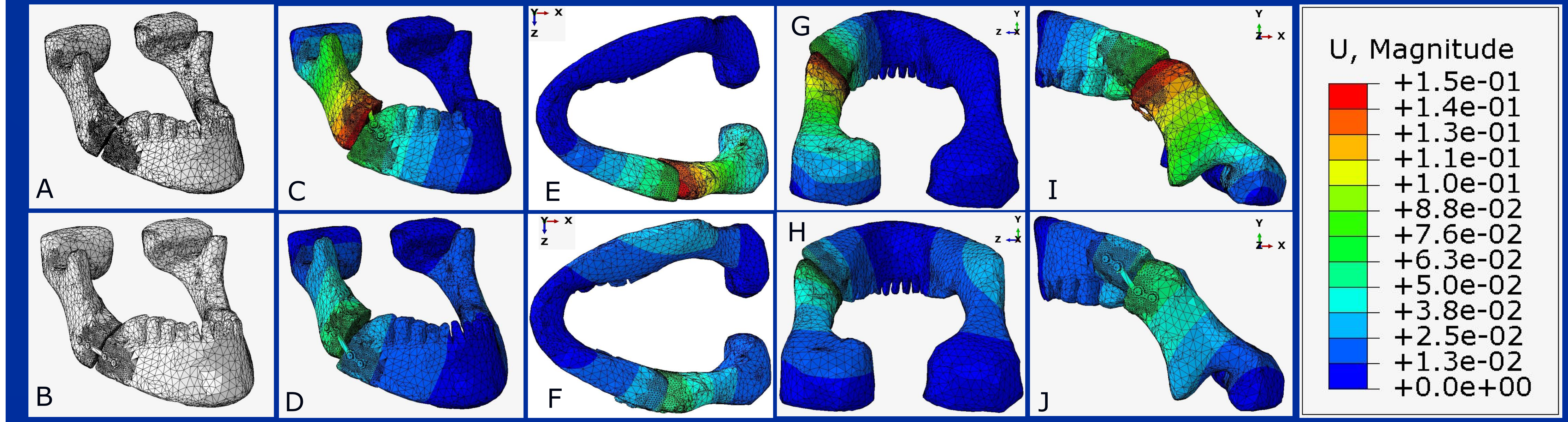


Figure 5: Pictures of mesh used in FEA with Champy plate (A) and lateral border plate (B), and contour plots showing relative displacement under 400 N of load (C-J) with associated scale in mm

The FEA resulted, qualitatively, in similar motion to physical testing, with the proximal segment moving down and outward relative to the distal segment. The Champy plate showed higher overall magnitude of displacement than the lateral plate at the fracture site. The Champy plate also better prevented displacement on the contralateral side of the mandible compared to the lateral plate, and prevented Mode I opening of the fracture line.

Discussion and Future Directions

Direct bone healing occurs when a fracture is reduced and stabilized to within 1 mm[7]. While both the physical and FEA models show clinically large displacements, this study neglects the stabilizing effect of the musculature *in vivo*, so these results should be interpreted as identifying and accentuating the differences between the two techniques. Such approaches to computational and *in vitro* simulation are widely used in industry to provide a "worst case scenario" wherein a patient might have severely damaged or missing soft tissue.

The present FEA model is limited by computational constraints that do not permit contact between segments under load. These, combined with simplifying material and boundary condition assumptions, result in overall displacement magnitudes that are smaller than the physical model. Nonetheless, the present results show that this finite element model holds promise because it replicates observed physical motion, and predicts motion vectors that would suggest a prevention of direct healing.

The immediate future direction of this research will be the refinement of the FEA to produce an accurate, and therefore truly physically validated model. Subsequently, the model will be used to identify key anatomical variables that correlate with plate performance, leading to the development of a decision-making tool for physicians, which may be further substantiated by a dataset of patient CT scans and clinical trials.

References

1. Kavanagh, E. P. *et al. Ir. J. Med. Sci.* **177,** 325–331 (2008).
2. Champy, M. *et al. J. Maxillofac. Surg.* **6,** 14–21 (1978).
3. Gear, A. J. L. *et al. J. Oral Maxillofac. Surg.* **63,** 655–663 (2005).
4. Al-Moraissi, E. A. & Ellis, E. *J. Oral Maxillofac. Surg.* **72,** 2197–2211 (2014).
5. Cienfuegos, R. *et al. AO Foundation Surgery Reference.* (2008).
6. Pacific Research Lab. *Sawbones Cranio and Maxillofacial Surgical Catalog*. (2016).
7. Marsell, R. & Einhorn, T. A. *Injury* **42,** 551–555 (2011).

Acknowledgements

I wish to thank Professor Douglas Van Citters for sharing his many resources and for his guidance, Dr. Eric Holmgren for sharing his time and medical expertise as a clinical partner, my family and Valentina for their constant support, and the generous donors who funded this research through the Kaminsky Undergraduate Research Award.