Finite element analysis & topology optimization of biodegradable magnesium-alloy intramedullary rods

Abigail Park1, Elaine Lipkin1, Justin Unger2, Dr. James Guest2, Dr. Timothy Weihs1

1 Johns Hopkins Department of Materials Science and Engineering, 2 Johns Hopkins Department of Civil and Systems Engineering

Introduction

Intramedullary rods (IRs) are a common surgical fixation method for long bone fractures. Bone remodeling requires loading to develop appropriate bone density. Conventional IRs made from stainless steel or titanium alloys can cause stress shielding in the bone due to the significantly higher stiffness of the implant compared to bone. Magnesium alloy IRs present a solution as they are biodegradable and have mechanical properties near that of bone. These qualities will mitigate the effects of stress shielding and reduce the risk of secondary surgeries.

Objectives

1. Design a component with the aim of improving bone remodeling in long bone fractures
2. Explore IR geometries with improved bone compatibility
3. Leverage Mg-alloys and topology optimization (TO) to develop an innovative biodegradable IR design
4. Conduct finite element analysis (FEA) to benchmark design performance
5. Use TO to tailor implant stiffness while minimizing volume

Methods

Dimensions: ~8mm Ø x 200mm L
Mg alloy corrosion: CR = ΔV/At
Corrosion Rates [mm/year]: -0.2, -0.5
Considered time intervals: 0-, 2-, 4-, & 6-months
Stiffness calculated at each time interval: axial compressive, flexural, torsional. Compared with bone, stainless steel, titanium alloy (Ti-6Al-4V), Mg alloy corrosion: CR = ΔV/At. Elastic Properties: E = 45.2GPa, ν = 0.338
Optimization Objective: Minimize strain energy
Constraint: 30% volume fraction
Software: Abaqus, Abaqus TOSCA, MATLAB, SolidWorks

Results

Figure 1: a) Intramedullary rod placed in femur to stabilize fracture. b) Stress map of rod under loading during step.

Figure 2: a) IR geometries considered in FEA performance models. b) Loading conditions tested to mimic in vivo environment of femoral long bone fracture. Orange corresponds with a 500N compressive load, Blue with a 100N flexural load, and Green with a three-radian torsional deformation. c) FEA stress shielding model. Longitudinal cross-section of concentric cylinders, outer cylinder adopts elastic properties of bone and inner cylinder adopts those of stainless steel. d) Image of conventional IR design

Figure 3: a) Compressive stiffness comparison between experimental IR geometries, bone and conventional implants. Experimental geometries were modeled at 0-, 2-, 4-, & 6-month snapshots to simulate degradation of the implant within the body. b) Modeled 6-month degradation of Mg-alloy cylindrical IR. c) Modeled 6-month degradation of Mg-alloy 6-point star IR.

Figure 4: FEA stress map of hourglass-shaped IR (6mm center diameter) with 500N load applied in direction indicated by arrow

Figure 5: Plot of strain energy optimization process with a 30% volume fraction (constraint) using Abaqus optimizer. Cross-section of IR shown throughout optimization process at iterations 0, 12, and 45.

Figure 6: Topology optimization solution of an hourglass rod generated by ABAGUS. Resulting solution is a “stool” shape with four legs and a tapered center.

Figure 7: Optimization and corrosion of cylindrical IR generated by MATLAB. Colored dots represent models at various stages of degradation (Orange at 0 months, Yellow at 2 months, Green at 4 months, Blue at 6 months)

Future Work

- Incorporate surrounding bone environment and IR fixation points into FEA modeling to more accurately predict in vivo behavior
- Consider buckling in FEA and TO under compression
- Improve corrosion rate modeling through continuous FEA degradation models and in vitro testing of Mg-Zn-Ca
- Develop more complex TO models with degradation optimization capabilities using MATLAB

Acknowledgements

We would like to thank our mentors Justin Unger, Jamie Guest, Tim Weihs, and Orla Wilson for their support as we learned Abaqus and SolidWorks.

References

