A FLEXIBLE METHOD FOR PRODUCING F.E.M.
ANALYSIS OF BONE USING OPEN-SOURCE SOFTWARE

Abhishektha Boppana¹, Ryan Sefcik², Dr. Jerry G. Myers³, Dr. Beth Lewandowski³

¹Case Western Reserve University, Cleveland, OH 44106
²The Ohio State University, Columbus, OH, 43210
³NASA Glenn Research Center, Cleveland, OH, 44135

Background:
- Astronauts may lose up to 9% of load-bearing bone density per month in spaceflight¹
- Lower chance of fracture in space due to lower loads (0G)²
- Higher loads on Earth (1G) result in a higher potential for fracture due to lowered bone density when astronauts return to Earth²
- Computational bone strength model can be used to assess bone fracture risk for astronauts

Objective:
Develop and test an open-source computational bone strength model for acceptable performance in the assessment of pre-flight and post-flight astronaut bone strength studies.

Open Source Advantage:
- Publicly published with a community collaborative mindset, where others are encouraged to view and contribute to the code to advance development
- Allows for expanded future development and input from a large community of experts

Hypothesis:
- Combine existing open-source software with our own scripts (Python)

Image Segmentation:
- Python script imports CT scans with visualization toolkit (VTK)
  - Library allows for import of many popular medical image formats
  - Script translates pixel values to Hounsfield values using metadata in original CT scans
  - Script isolates bone from medical images with thresholding based on Hounsfield values
  - Final image is binary representation of bone regions

3D Model Construction:
- Python’s VTK toolkit includes a Marching Cubes algorithm
  - Creates a 2D surface mesh from binary segmented regions³
  - Also smooths mesh and removes unnecessary triangles
  - Blender⁴ used to repair mesh and isolate any areas of interest
  - 2D surface mesh was recreated into a 3D volume mesh with Gmsh⁵

Material Properties Assignment:
- Original CT scans’ Hounsfield values extrapolated into densities and Young’s modulus using Keneko et al.’s⁶ prior bone ash testing
  - Translates Hounsfield value to bone ash density
  - Extrapolates Young’s modulus from bone ash density
  - Python script writes material properties to an FEBio XML file for easy import

Finite Element Analysis (FEA):
- Finite Element Analysis performed through FEBio⁷ suite
  - Software allows for the graphical fixing of points, defining of loads and boundary conditions
  - Allows for graphical viewing of end results

Conclusions:
- No straightforward method to implement existing open-source software into desired product
- A combination of various open source software along with self-developed scripts was needed to complete the segmentation, 3D construction, and FEA analysis tasks

Future Work:
- Need to design and run a selection of test cases to validate our method, including a full end-to-end simulation
- Extend further aspects of tool into interface, allowing for full integration of method into a single location

Acknowledgments:
This project was supported by the 2015 NASA Glenn Space Academy

References: