**MESH CREATION AND OPTIMIZATION OF A FINITE ELEMENT KNEE MODEL**

Adam Veenis
University of Pittsburgh: Undergraduate Researcher at Human Movement and Research Lab

**INTRODUCTION**

Patellofemoral pain (PFP), defined as pain around or behind the patella, is the most common of all knee pain diagnoses at 25% of all knee pain diagnoses [1]. It is particularly prevalent in physically active populations and in women [1,2]. There is not a concrete consensus for the etiology of patellofemoral pain but one of the most commonly accepted causes is abnormal patella tracking and alignment. This has been shown to cause accelerated cartilage wear and increase the joint stress [3]. It has been demonstrated that increased joint stress at the cartilage-bone interface is linked to PFP, supporting the hypothesis that pain is caused by increased stresses on the bone that would normally be absorbed by cartilage [4].

Joint stresses of the patellofemoral joint have been measured previously using in-vitro cadaveric models; however, these models are limited by their non-physiological muscle loads [4]. Musculoskeletal models have also been created to predict in-vivo average joint stresses by dividing the joint contact force divided by the joint contact area [5]. Magnetic resonance images are used to determine the joint contact area. The downside to this approach is that peak stresses cannot be determined. Most recently finite element (FE) models have been used to look at joint stresses. By combining subject specific musculoskeletal parameters and magnetic resonance images of, computational model can be constructed to analyze the joint stress of complex biological geometries.

The purpose of this study is to create a procedure to create hexahedral meshes of the cartilage components of the patellofemoral joint and use 2D elements to create shells of the bones. The process should be mostly automated and it is necessary that the size of the elements used to create the mesh can be adjusted. It is important to be able to control the element size of the mesh as convergence studies will have to be conducted on the model.

The largest difficulty associated with creating finite element models of biological tissues is the geometry. Biological tissues such as cartilage are very thin and curved making it harder to create a mesh that can be used. The femoral cartilage, in particular, presents difficulties and most automated meshing processes struggle with its geometry. Manual creation of meshes is time consuming and avoided when possible.

A previous study by Farrokhi et al used tetrahedral elements to create finite element models of the knee, but hexahedral elements are preferred [4]. Tetrahedral elements conform better to complex geometries and have been used in previous stress analyses but they are generally stiff and take longer to converge [6]. Meaning that smaller elements must be used for the model converges which increases the computational time. Hexahedral meshes, while more accurate than tetrahedral meshes, are very time consuming to create manually. Automated meshing processes do exist but they do not always create geometrically accurate meshes.

**METHODS**

The MR image sequences were loaded into Materialise’s Mimics software for segmentation. 2D masks of the femur, femoral cartilage, patella, and patellar cartilage by highlighting the corresponding areas on the MR images. When the corresponding area has been selected in the sagittal and coronal planes the software automatically selects the area in the axial plane. The software then creates 3D objects from the masks. The effects of volume averaging are removed by smoothing the surface of the object using two functions built in to Mimics: Smooth and Wrap. A smoothing factor of 0.87-0.895 was used. After smoothing the objects they are exported as .stl files. This file format saves the surface geometry of the bones and cartilage as shells composed of 2D elements; a hollow 3D shape is constructed where the surface is comprised of individual 2D elements.

The .stl files are then imported into a modeling software. We used Altair’s Hypermesh to create the finite element models. When the .stl files of the bones and cartilage are imported into Hypermesh they appear in the same global coordinate system from the MR images. The 2D elements, however, are irregularly shaped and sized. All of the components must be remeshed so that they are more ordered. First, using the existing elements as a template for the geometry of the bone or cartilage, new elements are created. Using the 2D Automesh function we create a new mesh that is composed of both triangle and quadrilateral 2D elements with an average size of 2.0 mm. The new elements replace the original elements. Then the bone/cartilage is remeshed again using only quadrilateral 2D elements with an average size of 1.0 mm. This results in a finer, more ordered mesh. To further refine the mesh the elements are used to create surfaces. The surfaces contain the topography of the bone/cartilage and can be used to create an even smoother mesh. Once the surfaces have been created they must be visually inspected to ensure that there are no free edges or interpolation errors (abnormal curves in the surface topography where none exist). These can be fixed by deleting the problem surfaces and manually creating new surfaces using built in functions of Hypermesh. The final step for creating meshes of the bones is to use the ShrinkWrap function to create new 2D elements of the bones from the geometry of the surfaces. Select Tight Wrap and the desired element size. The cartilage components must be composed of 3D hexahedral elements. The ShrinkWrap function is once again used. Specify 3D hexahedral elements and Tight Wrap. The element size and jacobian can be controlled for the 3D hexahedral elements.
RESULTS
The bones, femur and patella, were meshed using 2D quadrilateral with an average size of 1.0 mm. The articular cartilage corresponding to the two bones were meshed using 3D hexahedral elements. The average size of the elements was also 1.0 mm and a jacobian of 0.1 was used. In Figure 1 an image of the patellofemoral joint finite element model can be seen. Figure 2 contains a close up image of the femoral cartilage.

DISCUSSION
The protocol developed in this study successfully created hexahedral meshes of the cartilage components of the patellofemoral joint. The surfaces of the mesh were smoother than any created using other functions of Hypermesh. The edges of the cartilage still contained some ragged spots but that is fine because they are not near the contact areas. The tight geometry option was specified when using the shrinkwrap function instead of the loose option because the loose mesh added volume to the geometry, although it did create a smoother mesh at the edges.

The protocol in Hypermesh was not completely automated but the actual creation of the mesh was. By smoothing the 3D objects in Mimics before exportation and then performing the manual manipulations to the surfaces and shell elements in Hypermesh, the geometry of the cartilage was cleaned up sufficiently enough for the program to create a mesh automatically using the shrinkwrap function. We tried using the shrinkwrap function at various other steps such as straight from the .stl file or before the creation of the surfaces but the resulting meshes were either inaccurate or failed to create. Therefore, the manual cleaning of the geometry was necessary.

The protocol allows the user to control the element size used in the final mesh for both the 2D shell elements and the hexahedral elements. This is necessary because meshes using several different element sizes will be needed to perform a convergence study, which will determine the optimal element size to minimize computational time. The minimum jacobian angle of the hexahedral elements can also be controlled. This allows the user to determine how much each element can deform on creation from a perfect cube. A smaller minimum jacobian allows the elements to conform closer to the geometry.

The next in this study will be to perform the convergence study to determine the optimal element size. Then the model will be validated using a cadaver knee.

ACKNOWLEDGMENTS
Thank you to my mentors Dr. Shawn Farrokhi and Richard Debski for the opportunity to work on this project and their help. Thank you to Dr. Carrie Rainis for helping me start the project. Thank you to Dr. Alejandro Almarza and Jingming Chen for their feedback and help developing the presentations and abstract.

REFERENCES