Socket Sense
Progress Report – Week 2: CAD Model

Finite Element Analysis (FEA) requires an accurate assembly (using a CAD software) of the problem to be imported into ANSYS. Thus, an accurate CAD assembly is the first step to reliable FEA.

Open-source file packages:

1. Thingiverse.com (2021). *Transfemoral (above-knee) Quadrilateral Prosthetic Socket Pack 2 by MakisB*. [online] @thingiverse. Available at: https://www.thingiverse.com/thing:3233555 [Accessed 18 May 2021].

2. HUMAN LEGS | 3D model (2021). *BRO1977*. [online] CGTrader. Available at: https://www.cgtrader.com/free-3d-models/character/anatomy/human-legs--2 [Accessed 18 May 2021].

1. Socket

From the open-source file package [1] a quadrilateral socket design – "TFA_Quad_socket_d8" is selected for the study.

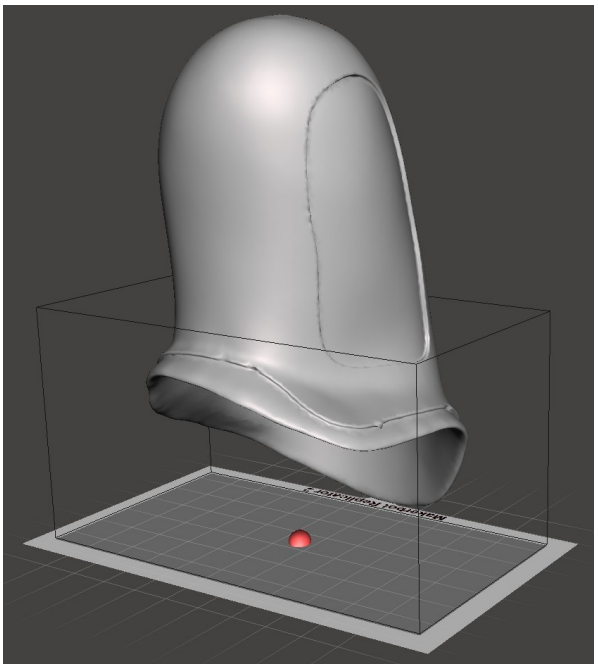

Fig 1. TFA_Quad_socket_d8

Steps to construct a CAD file of the socket:

1. MeshMixer is used to rectify the facet errors of TFA_Quad_socket_d8.stl. Further the mesh is reduced and model topography is smooth out to remove the unnecessary details for the FEA as shown in Figure 2. The file is exported from MeshMixer as a binary STL.
2. FreeCAD is used to convert this exported binary STL file to a Solid Part as shown in Figure 3. The constructed solid part is exported as a STEP file.
3. SOLIDWORKS is used to convert the exported STEP file to a SLDPRT file for assembly as shown in Figure 4.

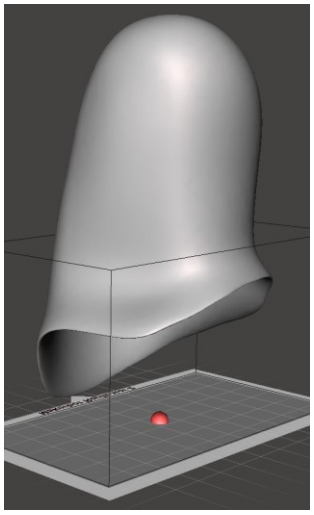

Fig 2. MeshMixer

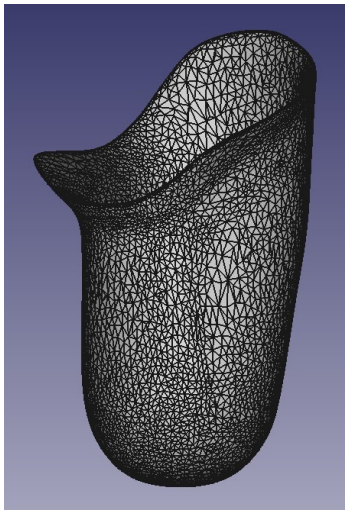

Fig 3. FreeCAD

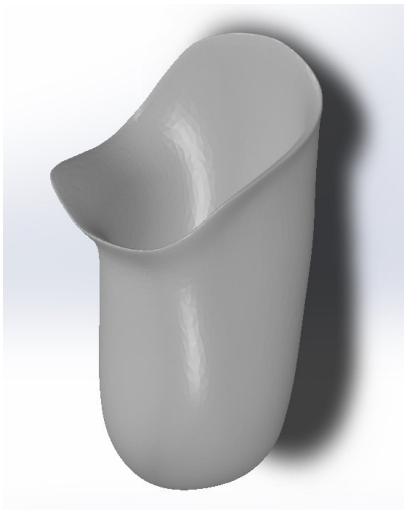

Fig 4. SOLIDWORKS

2. Stump

Steps to construct a CAD file of the stump:

1. The STL file of socket exported from MeshMixer is imported into SOLIDWORKS to generate a part which can be used to construct supplementary parts. The intersect feature of SOLIDWORKS is used to generate the stump i.e., an intersection between the socket and a constructed plane as shown in Figure 5 and Figure 6. The Stump is exported as a STL file.

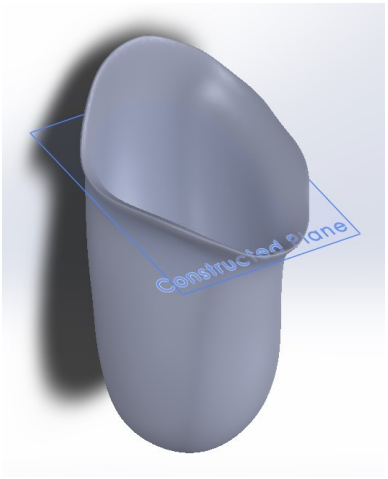

Fig 5. Socket and Constructed Plane

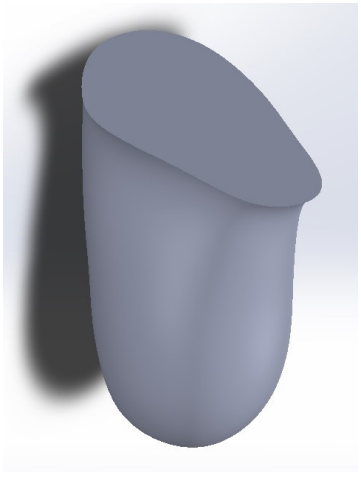

Fig 6. Stump generated from intersection

2. The STL file is imported into MeshMixer to smooth the model topography and reduce the mesh density as show in Figure 7. The file is exported from MeshMixer as a binary STL.
3. FreeCAD is used to convert this exported binary STL file to a Solid Part as shown in Figure 8. The constructed solid part is exported as a STEP file.
4. SOLIDWORKS is used to convert the exported STEP file to a SLDPRT file for assembly as shown in Figure 9.

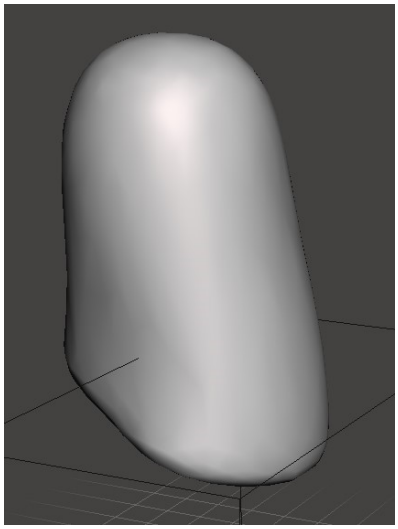

Fig 7. MeshMixer

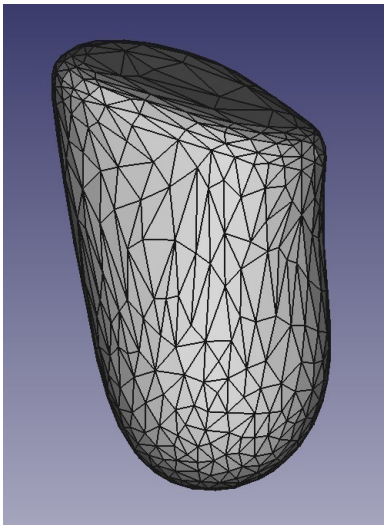

Fig 8. FreeCAD

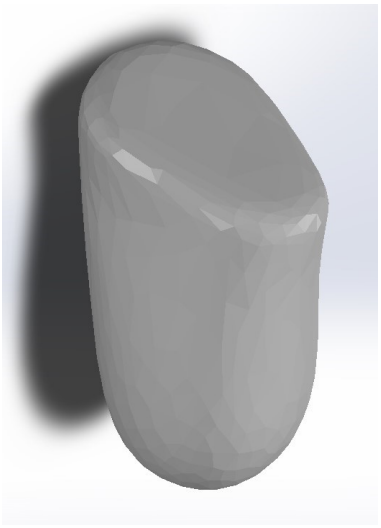

Fig 9. SOLIDWORKS

3. Femur

From the open-source file package [1] the Pelvis and Femur assembly – "Pelvis_femur" was selected for the study and the amputated femur was extracted from the Pelvis_femur.stl using SOLIDWORKS as shown in Figure 10 and Figure 11. The amputated Femur extracted by deleting the Pelvis, resulted in many facet errors and FreeCAD was unable to convert it into a solid part, even after corrections using MeshMixer. Thus, to bypass these errors, it was decided to cut the model of a healthy Femur in SOLIDWORKS to generate the amputated Femur.

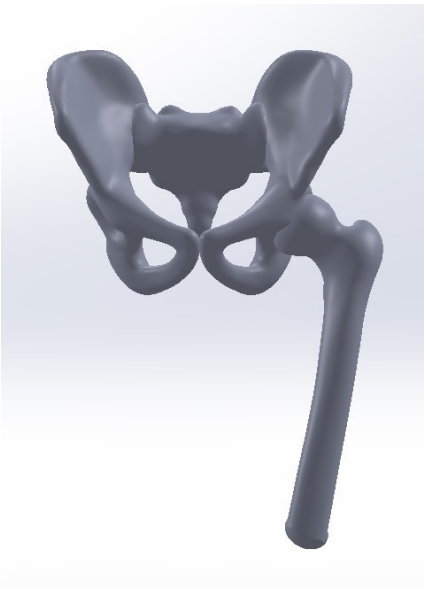

Figure 10. Pelvis & Femur assembly

Figure 11. Extracted Femur

From the open-source file package [2] the lower half of skeleton – "HUMAN LEGS" was selected as the base model to construct the amputated Femur. Measurements of amputated femur from [1] were used as reference to replicate the same amputation on the healthy Femur extracted from [2] as shown in Figure 12 and Figure 13.

Figure 12. Human Legs

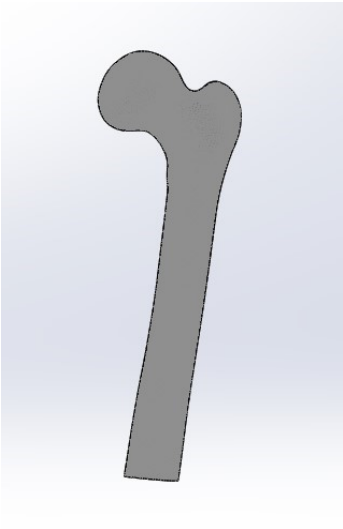

Figure 13. Amputated Femur

Steps to construct a CAD file of the stump:

1. STL file of amputated Femur is exported from SOLIDWORKS. This file is imported into MeshMixer to smooth the model topography and reduce the mesh density as shown in Figure 14. The file is exported from MeshMixer as a binary STL.
2. FreeCAD is used to convert this exported binary STL file to a Solid Part as shown in Figure 15. The constructed solid part is exported as a STEP file.
3. SOLIDWORKS is used to convert the exported STEP file to a SLDPRT file for assembly as shown in Figure 16.

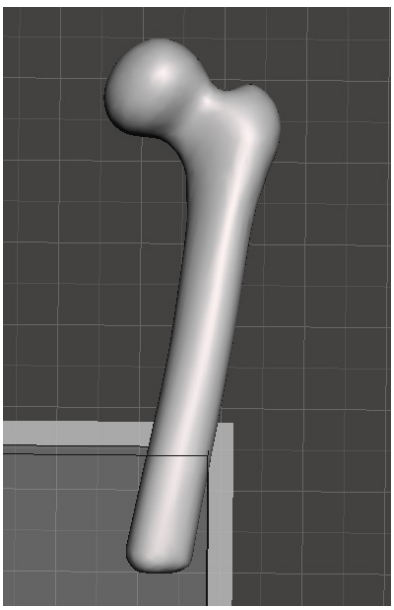

Fig 14. MeshMixer

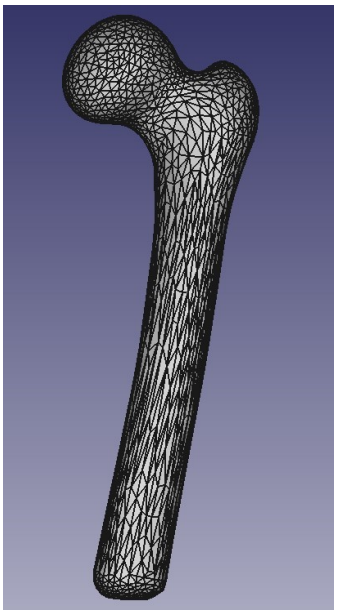

Fig 15. FreeCAD

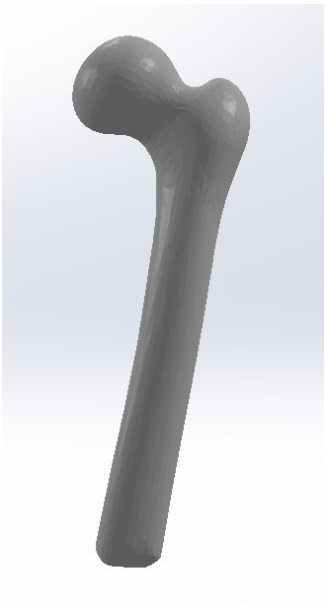

Fig 16. SOLIDWORKS

4. Assembly

The CAD models of Socket, Stump and Femur required are ready for assembly but as all the models have organic shapes, there is no possible way to define mates between them for a fixed assembly. Thus, the models are only arranged anatomically and contacts will be defined in ANSYS during FEA.

SOLIDWORKS is used to create a cavity in the Stump in the exact shape of the amputated Femur for their assembly. Finally, the assembled Femur and Stump is placed inside the Socket. Rendered images of all the views of the final assembly of Socket, Stump and Femur are shown in the figures below.

Figure 17. Isometric View

Figure 18. Front View

Figure 19. Back View

Figure 20. Left-Side View

Figure 21. Right-Side View

Figure 22. Top View

5. Conclusion

IGES file of the final assembly of Socket, Stump and Femur is ready and I will procced to the Finite Element Analysis (FEA) after KTH IT support resolves the TeamViewer connectivity issues and I have the access to ANSYS on the KTH Workstation again.