Simulation Driven Analysis of Complex Bio-mechanical Dynamic Processes

An OpenSim biomechanical model and FEA model to estimate socket-stump interface pressure

Ayush Srivastav

Technical Report
Department of Machine Design
Royal Institute of Technology
SE-100 44 Stockholm

TRITA-ITM-AVL XXXX:XX
ISSN XXXX-XXXX
ISRN/KTH/MMK/R-XX/XX-SE
ISBN XXXXXXXXXXX

TRITA-ITM-AVL XXXX:XX
ISSN XXXX-XXXX
ISRN/KTH/MMK/R-XX/XX-SE
ISBN XXXXXXXXXX

Data-Driven Analysis of Complex Bio-mechanical Dynamic Process - Modeling and Simulation
Ayush Srivastav
Technical Report

Technical Report.

Abstract

This deliverable describes the design and evaluation of a robotic test rig for advanced development of a dynamic FEA models to estimate the socket-stump interface pressure in a transfemoral amputee. The ultimate objective is to enable the replication of a variety of stump-socket interaction scenarios and thereby allow the generation of effective data for the identification and characterization of complex bio-mechanical properties. With this report, we present the results from the following tasks:

- Finite Element Analyis of transfemoral amputee
- Biomechanical Modelling of transfemoral amputee

Acknowledgement

This work was completed as part of the SocketSense project that has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 825429.

Keywords
finite element analysis, c3d, trc, mot, piston forces

Contents

Chapter 1

Introduction

1.1 Transfemoral amputation

Transfemoral (above knee) amputation is a surgical procedure performed to remove the lower limb above the knee joint (as shown in Figure 1.1) when that limb has been severely damaged via trauma, disease, or congenital defect.

Most transfemoral amputations are performed due to peripheral vascular disease or severe disease of circulation in the lower limb. Poor circulation limits healing and the immune response to injury, leading to life-threatening bone infections. Amputation is performed to remove the diseased tissue and prevent the further spread of infection.

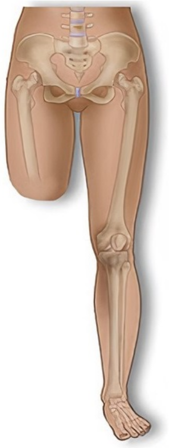


Figure 1.1: Transfemoral (above-knee) amputation of the right leg

1.2 Prosthetic solution for transfemoral amputee

Amputation of a limb is one of the most traumatic events in one's life. Apart from the obvious loss of functionality, the psychological consequences and economic loss of the amputee are immense. A prosthetic solution (as shown in Figure 1.2) can be envisaged when a person is amputated after an accident or a vascular disease. Essentially, the prosthetic device aims to restore: (1) the self-esteem of the patient by using the prosthesis like a complement of his complete body shape and (2) a normal and independent ambulation as much as possible.

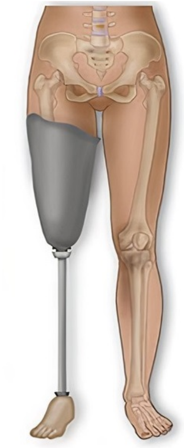


Figure 1.2: Transfemoral (above-knee) prosthetics for the right leg

1.2.1 Transfemoral prosthesis components

The major components of a transfemoral prosthesis are shown in Figure 1.3 and briefly described below:

1. **Socket**: The socket serves as the interface between the residual limb and the prosthesis. It must protect the residual limb and appropriately transmit the forces associated with standing and ambulation.

2. **Suspension system**: The suspension system keeps the prosthesis from falling off the residual limb and provide proprioceptive feedback.

3. **Knee joint**: The prosthetic knee joint must provide support during the stance phase of ambulation, produce smooth control during the swing phase and maintain unrestricted motion for sitting and kneeling.

4. **Pylon**: Pylon is a rigid, usually tubular structure between the socket (or knee joint) and the foot that provides a weight bearing shock-absorbing support shaft for the prosthesis.

5. **Prosthetic feet**: The prosthetic feet provide a stable weight-bearing surface, absorb shock and replace lost muscle function.

Figure 1.3: Transfemoral (above-knee) prosthetics for the right leg

1.3 Objective

The transfemoral prosthesis needs a socket to act as an interface between the stump (residual lower limb) and the prosthetic device. This design completely modifies the natural performance of the residual limb and exposes the soft tissues to physiological change by the transfer of body loads generated during gait. These new conditions can induce skin problems such as callosities, abrasions, blisters, and can also affect the vascular system.

The stress state in the soft tissues of a lower limb amputee can be measured by experimental procedures using force transducers. However, the sensors used in the experiment can produce stress concentrations over the soft tissues and modify the gait, producing results only valid at the points where the sensors are located. All these experimental difficulties favor the use of numerical methods like the Finite Element Analysis (FEA) to assess the stress–strain state in the stump.

Thus, the ultimate objective of this study is to re-design personalised prosthetic sockets for transfemoral amputees to improve comfort and reduce pain. The initial step to achieve this objective is the development of advanced dynamic FEA models to assess the stress–strain state of the soft tissue in the residual lower limb during different stages of gait.

Chapter 2

Finite Element Analysis

Finite Element Analysis (FEA) is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM). In modelling the mechanical response of soft tissue, the FEM has been employed to solve the established constitutive equations which describes the soft tissue behavior in continuum mechanics. FEA in transfemoral amputees is implemented to assess the stress–strain state in the stump and the pressure distribution at the socket–stump interface.

2.1 Geometry

2.1.1 Socket

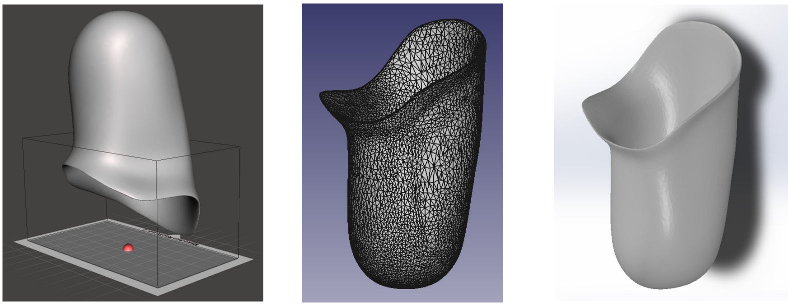
Open-source files of transfemoral quadrilateral prosthetic sockets is downloaded from Thingiverse[1]. From the open-source file package a quadrilateral socket design named "TFA_Quad_socket_d8" (as shown in Figure 2.1) is selected for the FEA due to the availability of a volunteer amputee with the same socket.

Steps to construct a CAD file of the transfemoral quadrilateral prosthetic 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 2.1a. 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 2.1b. 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 2.1c.



(a) MeshMixer (b) FreeCAD (c) SolidWorks

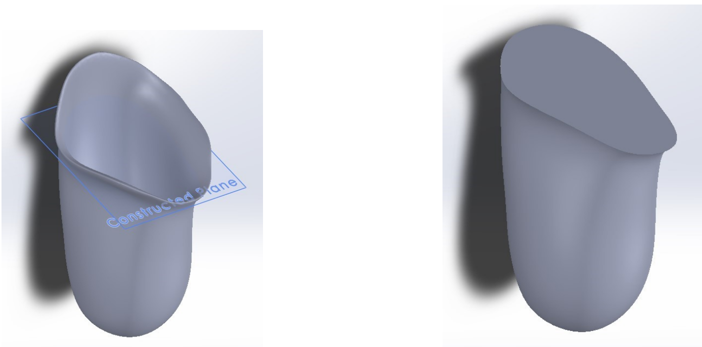
Figure 2.1: Steps to construct a CAD file of the socket

2.1.2 Stump

Instead to scanning the stump of a patient, a corresponding stump for TFA_Quad_socket_d8 is constructed using boolean intersection.

Steps to construct a CAD file of the stump:

1. STL file of the 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 boolean intersection between the socket and a constructed plane as shown in Figure 2.2a and Figure 2.2b. The Stump is exported as a STL file.



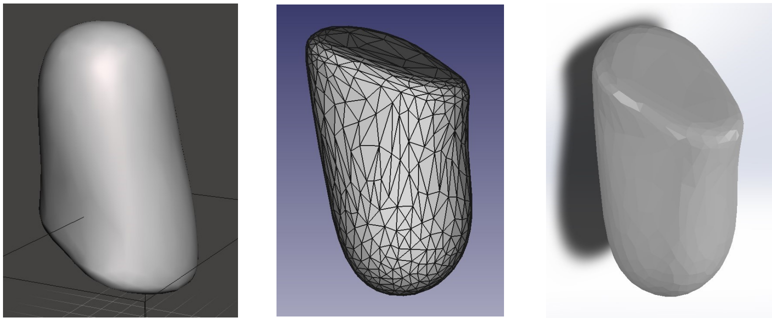
(a) Socket and constructed plane (b) Stump generated from boolean intersection

Figure 2.2: Steps to generate a corresponding Stump for the Socket

2. The STL file is imported into MeshMixer to smooth the model topography and reduce the mesh density as show in Figure 2.3a. 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 2.3b. 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 2.3c.



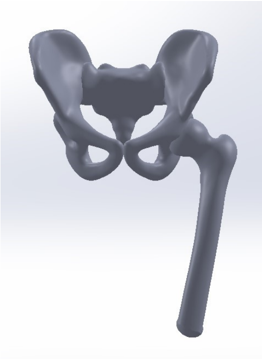
(a) MeshMixer (b) FreeCAD (c) SolidWorks

Figure 2.3: Steps to construct a CAD file of the stump

2.1.3 Femur

The Pelvis and Femur assembly – "Pelvis_femur" (as shown in Figure 2.2b) from Thingiverse[1] was selected and the amputated femur was extracted from the Pelvis_femur.stl using SOLIDWORKS as shown in Figure 2.2b. 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. Lower half of the skeleton – "HUMAN LEGS" was extracted from CGTrader's[2] open-source file package as shown in Figure 2.5a to construct the amputated Femur. Measurements of amputated femur from Thingiverse[1] were used as reference to replicate the same amputation on the healthy Femur extracted from CGTrader[2] to construct the amputated Femur as shown in Figure 2.5b.

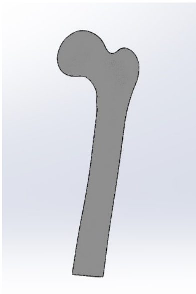


(a) Pelvis and Femur assembly

(b) Extracted Femur

Figure 2.4: Open-source files from Thingiverse

(a) Human Legs



(b) Amputated Femur

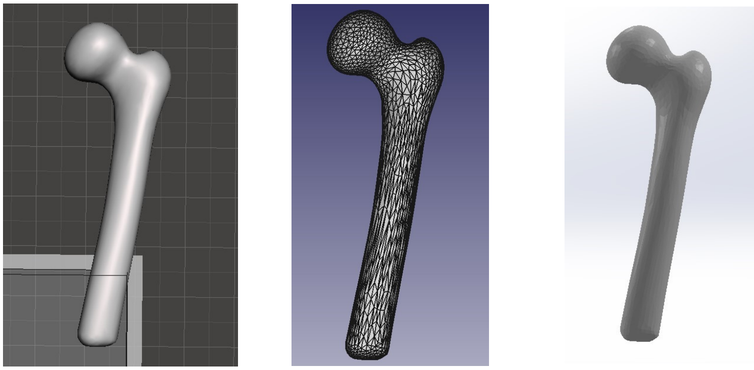
Figure 2.5: Open-source files from CGTrader

Steps to construct a CAD file of the amputated Femur:

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 2.6a. 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 2.6b. 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 2.6c.



(a) MeshMixer (b) FreeCAD (c) SolidWorks

Figure 2.6: Steps to construct a CAD file of the amputated Femur

2.1.4 Assembly

The CAD models of socket, stump and amputated Femur are arranged anatomically in SOLIDWORKS and boolean subtraction is used to assemble the amputated Femur and stump. Then, the amputated Femur and stump assembly is placed inside the socket to complete the assembly required for FEA. As the socket, stump and amputated Femur all have organic shapes, there is no possible way to define mates between them for a fixed assembly. Thus, a IGES file of the final assembly of socket, stump and amputated Femur is exported from SOLIDWORKS and contacts will be defined in ANSYS during FEA.

Rendered images of all the views of the final assembly of socket, stump and amputated Femur are shown in Figure 2.7.

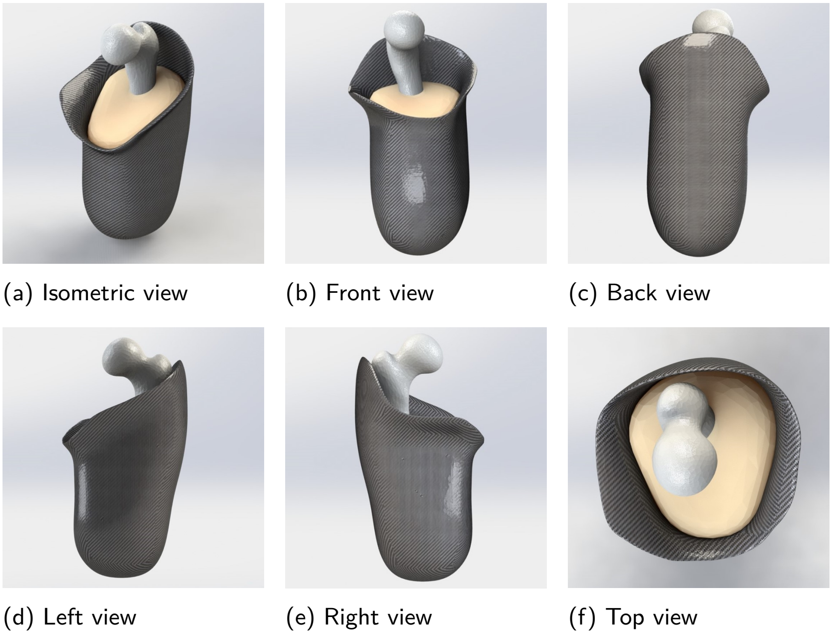


Figure 2.7: All views of the socket, stump and amputated Femur assembly

2.2 Material properties

2.2.1 Femur

- Young's modulus =15 GPa [3]
- Poisson's ratio = 0.3 [3]
- Density = 2000 $\mathrm{kg\,m^{-3}}$ [3]

2.2.2 Stump

Neo-Hookean hyperelastic model:
C_{10} = 11.6 kPa [3] and D_1 = 11.9 $\mathrm{MPa^{-1}}$ [3]

2.2.3 Socket

- Young's modulus =1.5 GPa [3]
- Poisson's ratio = 0.3 [3]
- Density = 800 $\mathrm{kg\,m^{-3}}$ [3]

2.3 Contacts

2.3.1 Between Femur and Stump

Bonded contact between the femur and the stump as shown in Figure 2.8

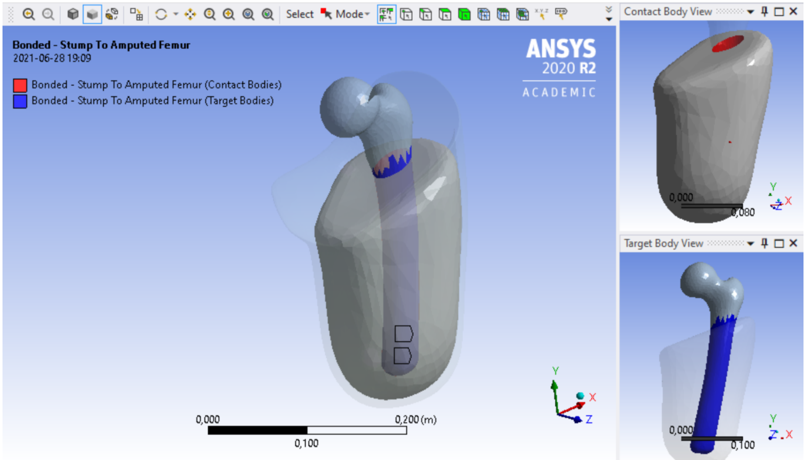


Figure 2.8: Bonded contact

2.3.2 Between Stump and Socket

Frictional contact between the stump and socket as shown in Figure 2.9.

- Friction coefficient: $\mu = 0.23$
- Behaviour: Asymmetric
- Formulation: Augmented Lagrange
- Detection Method: Nodal-Projected Normal from Contact
- Interface Treatment: Adjust to Touch

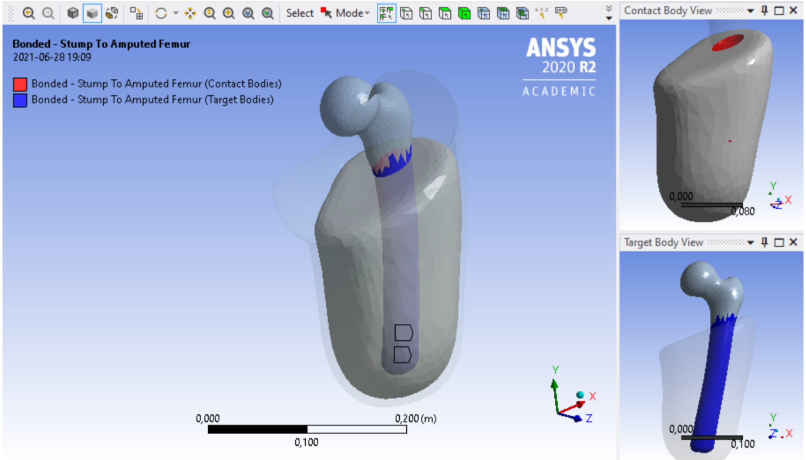


Figure 2.9: Frictional contact

2.4 Mesh

2.4.1 Defaults

- Physical Preference: Mechanical
- Element Order: Linear
- Element Size – Femur, Stump and Socket – Element Size: 0.01 m

2.4.2 Sizing

- Transition: Fast
- Span Angle Center: Coarse

2.4.3 Quality

- Error Limits: Aggressive Mechanical
- Smoothing: Medium

2.4.4 Statistics

- Nodes: 31417
- Elements: 120346

2.5 Load and boundary conditions

2.5.1 Analysis Setting - static structural

- Number of Steps: 26
- Auto Time Stepping: On
  - Define by: Substeps
  - Initial Substeps: 30
  - Minimum Substeps: 10
  - Maximum Substeps: 500
- Large Deflection: On

2.5.2 Donning Load

A load of 50 N [4] is applied at the top face of the stump as shown in Table 2.1 to simulate the donning process.

2.5.3 Standing Load

Assuming a man weighting 120 kg, a force of 600 N will act on a single leg (i.e., the stump in case of transfemoral amputees) while standing. Thus, a load of 600 N is applied on the bottom faces of the socket as shown in Table 2.2.

2.5.4 Fixed Support

A fixed support boundary condition is applied on the head of femur.

Steps	Time	Y[N]
1	0	0
1	1	-25
2	2	-50
3	3	-50
4	4	-50
5	5	-50
6	6	-50
7	7	-50
8	8	-50
9	9	-50
10	10	-50
11	11	-50
12	12	-50
13	13	-50
14	14	-50
15	15	-50
16	16	-50
17	17	-50
18	18	-50
19	19	-50
20	20	-50
21	21	-50
22	22	-50
23	23	-50
24	24	-50
25	25	-50
26	26	-50

Table 2.1: Donning load applied in 2 steps

Steps	Time	Y[N]
1	0	0
1	1	0
2	2	0
3	3	25
4	4	50
5	5	75
6	6	100
7	7	125
8	8	150
9	9	175
10	10	200
11	11	225
12	12	250
13	13	275
14	14	300
15	15	325
16	16	350
17	17	375
18	18	400
19	19	425
20	20	450
21	21	475
22	22	500
23	23	525
24	24	550
25	25	575
26	26	600

Table 2.2: Standing load applied in 24 steps

2.6 Solution

2.6.1 Solution Information – Force Convergence

The runtime of ANSYS Solver is 1 hour 37 mins, using AMD Ryzen Threadripper 2950X 16-Core Processor 3.50 GHz and 64 GB RAM, Windows 10 Education 64 bits OS.
The force convergence plot and the cumulative iteration vs time plot are shown in Figure 2.10 and Figure 2.11 respectively.

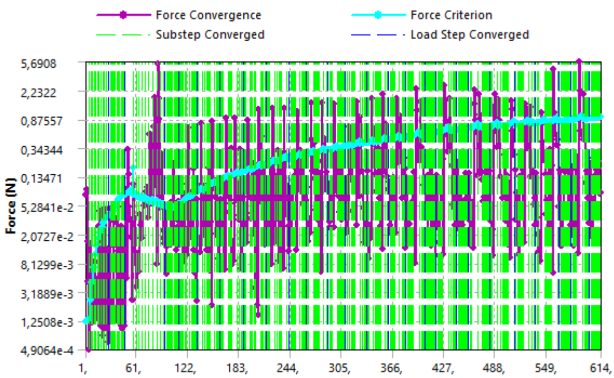


Figure 2.10: Frictional convergenece plot

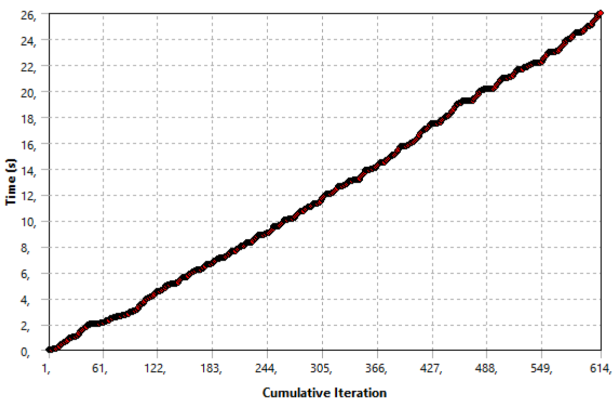
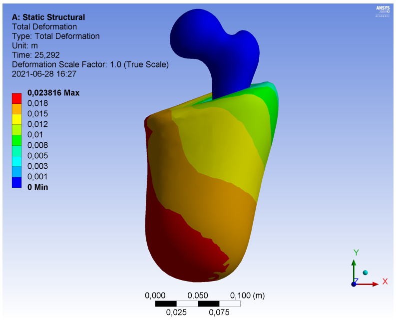


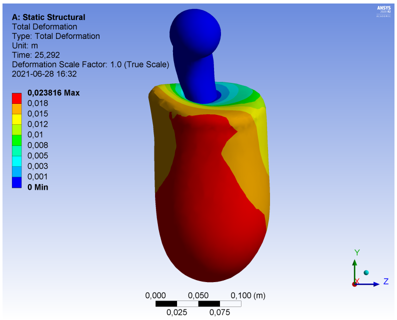
Figure 2.11: Cumulative iteration vs time plot

2.6.2 Total Deformation

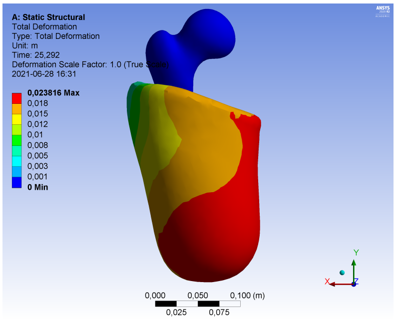
The resultant Total Deformation from FEA is shown in Figure 2.12.



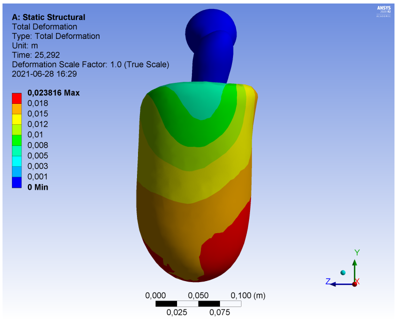
(a) Anterior view of Total Deformation



(b) Medial view of Total Deformation



(c) Posterior view of Total Deformation

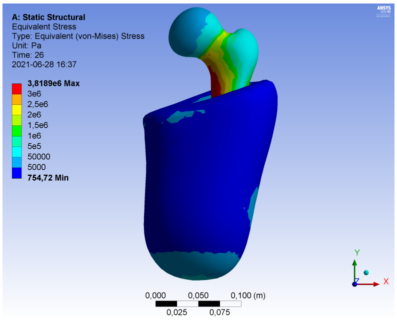


(d) Lateral view of Total Deformation

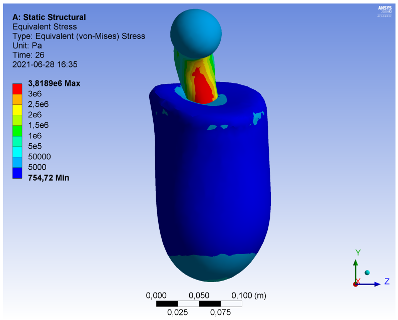
Figure 2.12: Total Deformation results from FEA

2.6.3 Equivalent Stress

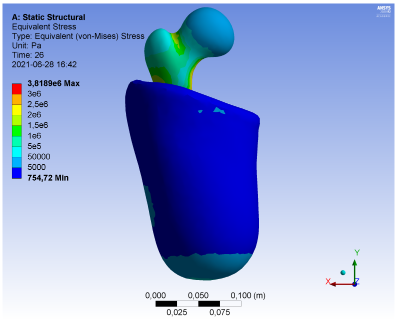
The resultant Equivalent Stress from FEA is shown in Figure 2.13.



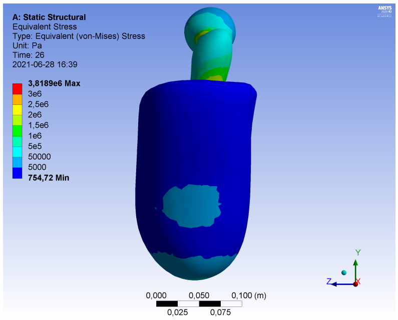
(a) Anterior view of Equivalent Stress



(b) Medial view of Equivalent Stress



(c) Posterior view of Equivalent Stress

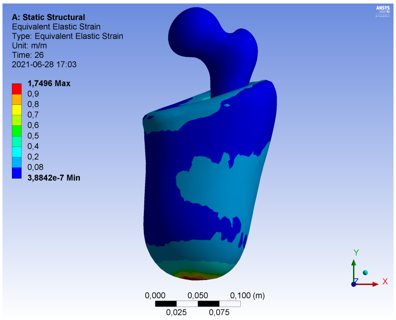


(d) Lateral view of Equivalent Stress

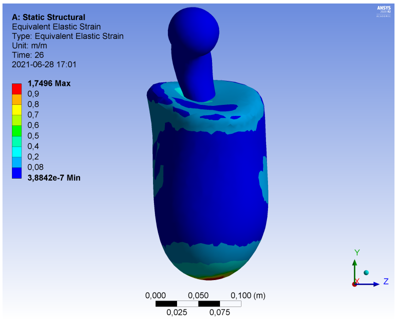
Figure 2.13: Equivalent Stress results from FEA

2.6.4 Equivalent Elastic Strain

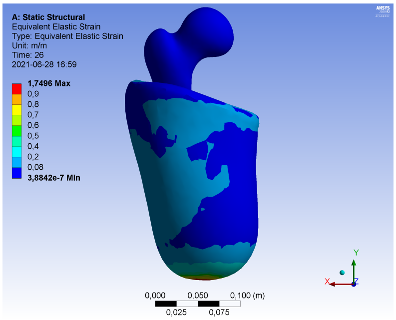
The resultant Equivalent Elastic Strain from FEA is shown in Figure 2.14.



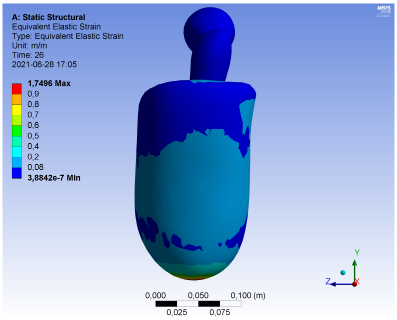
(a) Anterior view of Equivalent Elastic Strain



(b) Medial view of Equivalent Elastic Strain



(c) Posterior view of Equivalent Elastic Strain

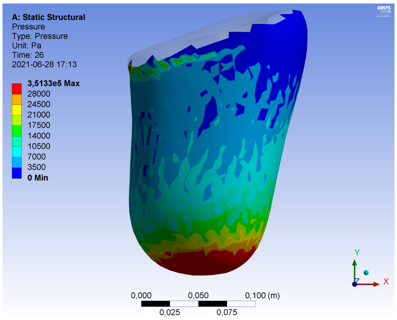


(d) Lateral view of Equivalent Elastic Strain

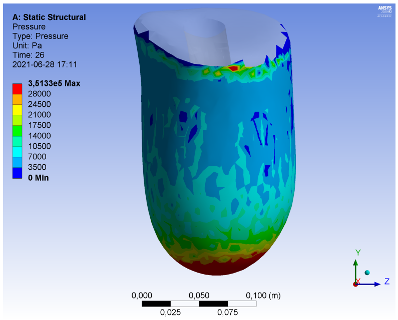
Figure 2.14: Equivalent Elastic Strain results from FEA

2.6.5 Pressure

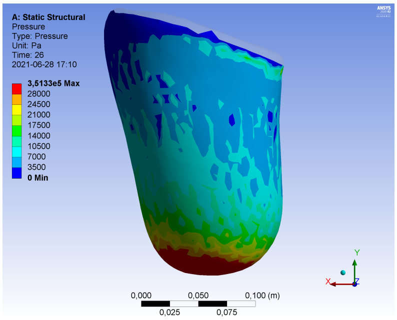
The resultant Pressure from FEA is shown in Figure 2.15.



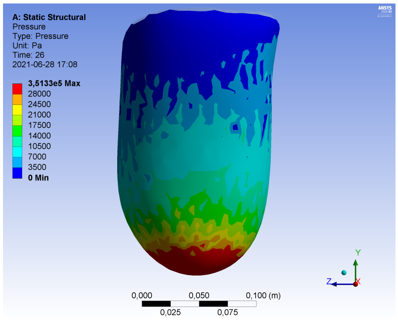
(a) Anterior view of Pressure



(b) Medial view of Pressure



(c) Posterior view of Pressure

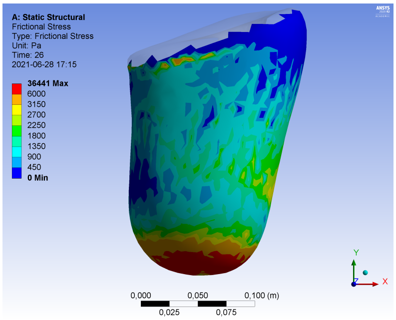


(d) Lateral view of Pressure

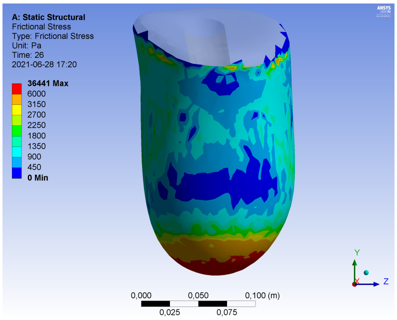
Figure 2.15: Pressure results from FEA

2.6.6 Frictional Stress

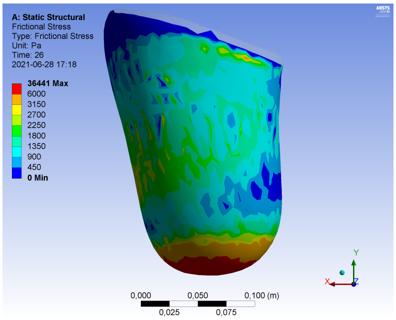
The resultant Frictional Stress from FEA is shown in Figure 2.16.



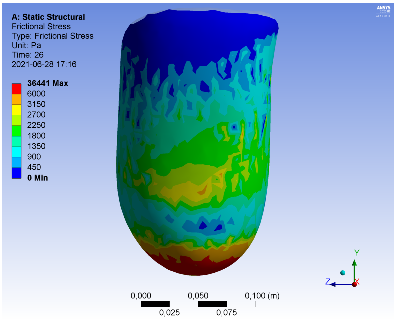
(a) Anterior view of Frictional Stress



(b) Medial view of Frictional Stress



(c) Posterior view of Frictional Stress



(d) Lateral view of Frictional Stress

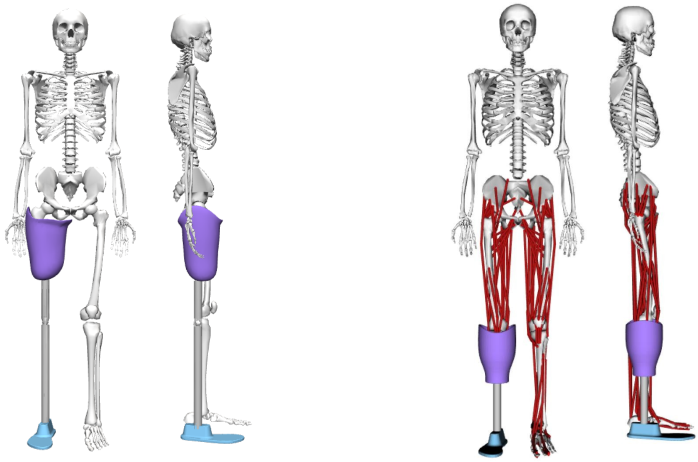
Figure 2.16: Frictional Stress results from FEA

Chapter 3

Biomechanical Modelling

3.1 OpenSim model

A biomechanical model of transfemoral amputee (shown in Figure 3.1a) is constructed in OpenSim by modifying an existing transtibial amputee model created by Andrea Willson[5] (shown in Figure 3.1b).



(a) Transfemoral amputee

(b) Transtibial amputee

Figure 3.1: OpenSim biomechanical models of amputees.

Moving forward, to compute the inverse kinematics and inverse dynamics analysis, OpenSim requires the trace (.trc) and motion (.mot) files which contain the marker and ground reaction force data respectively.

3.2 Experimental data

Experimental dataset published by Sarah Hood[6] is used for the biomechanical simulations. From the dataset of 20 amputees available, the data of a 42-year old man with a right-leg transfemoral amputation (Subject Code: TF08) is selected because of the subject's amputation similar to the biomechanical model built in OpenSim. The TF08 extracted data used here is uploaded to **Box**. TF08 data is stored in .c3d files in the dataset provided by Sarah Hood[6]. Thus, the .trc and .mot file data is to be generated from the .c3d files.

3.2.1 Convesrion of .c3d files to .trc files

The trace (.trc) files contain the marker data. The .trc files are generated from the .c3d files of TF08 dataset using a MATLAB script developed by Felipe Alvim[7]. A stable version of the MATLAB script has been uploaded to **Box** for future reference.

Steps to convert .c3d files to .trc files:

1. Download the c3d2OpenSim.rar package from SimTK[7].
2. Extract c3d2OpenSim.rar files to a folder with the .c3d files of TF08.
3. Open MATLAB in the same directory as the extracted files of c3d2OpenSim.rar and type "c3d2OS" in the command window as shown in Figure 3.2.

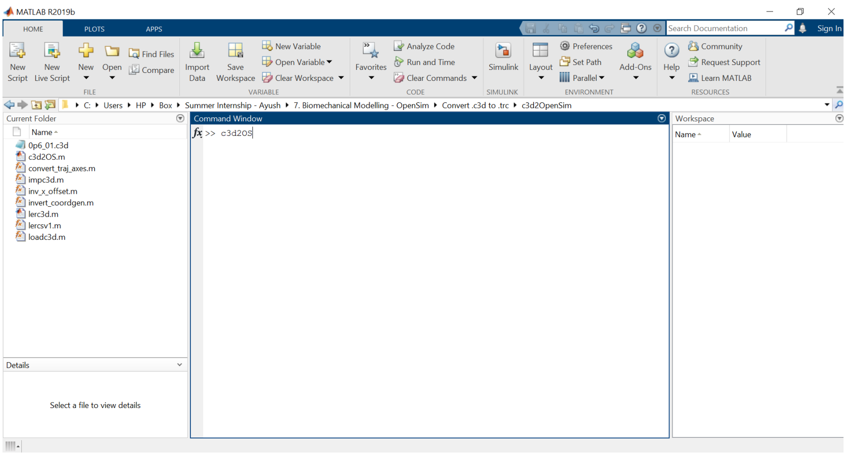


Figure 3.2: MATLAB directory and c3d2OS command

4. Select ".trc (marker trajectories)" from the pop-up as shown in Figure 3.3.

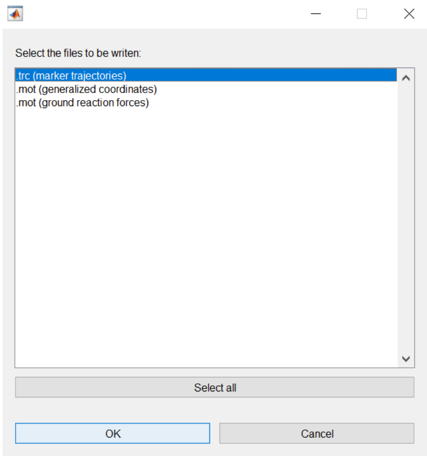


Figure 3.3: Output file format selection

5. Select a .c3d file to convert from the "Import C3D file" pop-up.
6. Enter a name for the .trc file (without the extension .trc).
7. Select the markers required in the converted .trc file or click "Select all" as shown in Figure 3.4

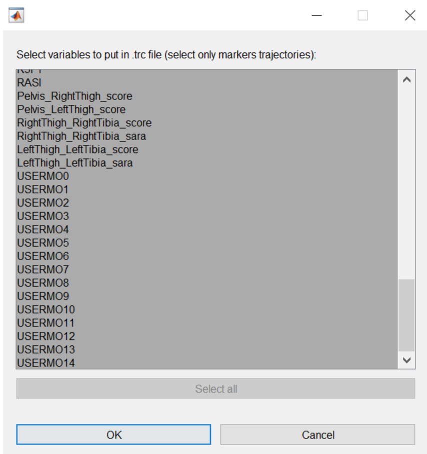


Figure 3.4: Selection of markers

8. Select "VICON" from the pop-up as shown in Figure 3.5.

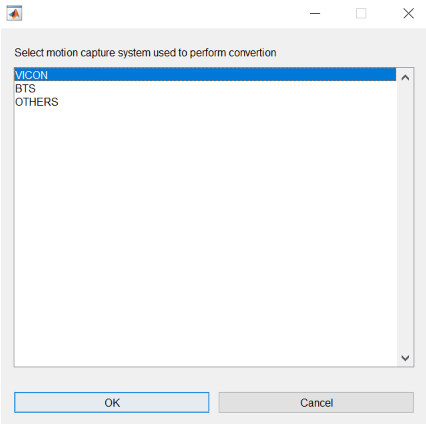


Figure 3.5: Selection of the motion capture system used for data collection

On successful completion of all the above steps a "Files recorded successfully!" message will pop-up and a .trc file will be generated in the same directory.

Repeat all the above steps for each .c3d file to generate a .trc file with the required marker data.

3.2.2 Convesrion of .c3d files to .mot files

The motion (.mot) files contain the ground reaction force data. The .mot files are generated from the .c3d files of TF08 dataset using "C3D tools.exe" developed by BSNlab[8]. A stable version of the executable file (V2.0) has been uploaded to **Box** for future reference.

Steps to convert .c3d files to .mot files:

1. Download the C3D TOOLS V2.0 executable file from BSNlab[8].

2. Install the C3D tools.exe file with all the default options and settings as shown in Figure 3.6.

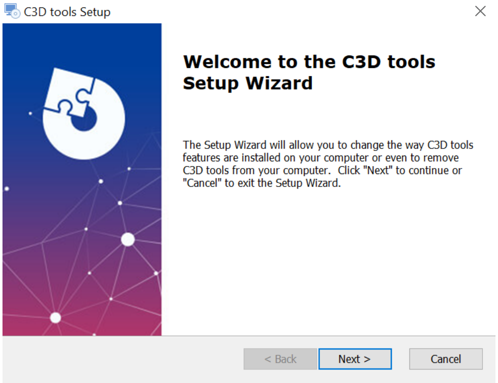


Figure 3.6: C3D tools Setup

3. After installtion, run the "C3D Tool 1.8.exe" file from the start menu.

4. Click the "Open C3D" button to select the .c3d file to be converted to .mot file and load its data into C3D Tools Figure 3.7

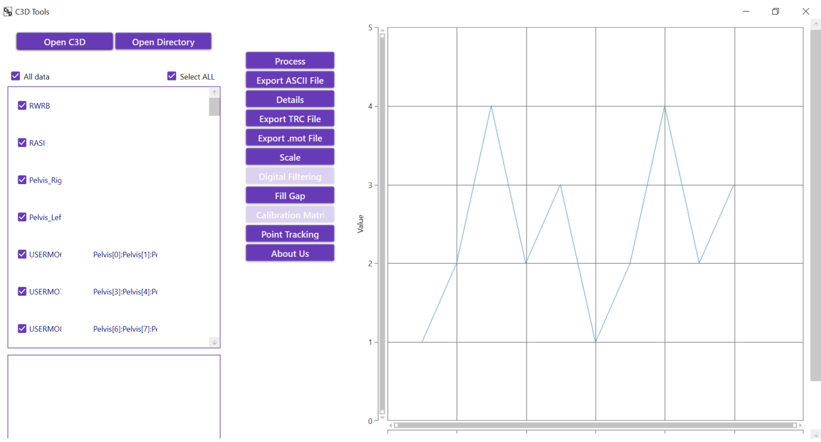


Figure 3.7: .c3d file loaded into C3D Tools

5. From all the data detected by C3D Tools only select the following data as shown in Figure 3.8.

   (a) Force.Fx1, Force.Fy1, Force.Fz1, Force.Fx2, Force.Fy2 and Force.Fz2

   (b) Moment.I

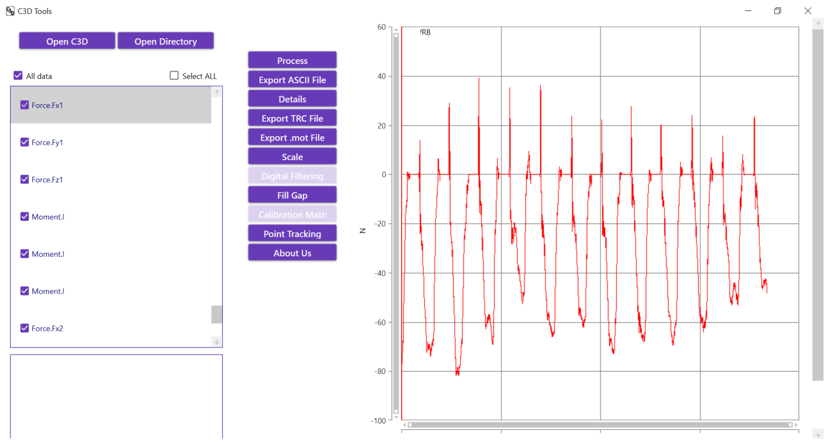


Figure 3.8: Selected force and moment data

6. Click "Process" and then click "Export .mot File" to generate the .mot file as shown in Figure 3.9.

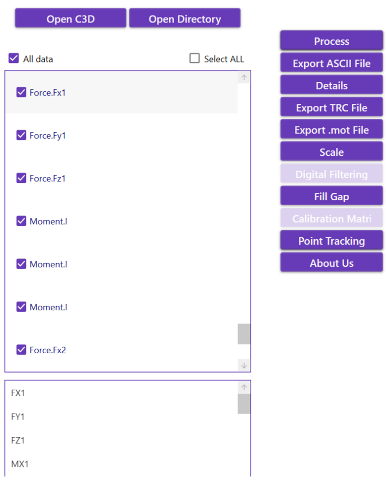


Figure 3.9: Process and Export .mot file

On successful completion of all the above steps a .mot file will be generated in the chosen directory.

Repeat all the above steps for each .c3d file to generate a .mot file with the required ground reaction force data.

3.3 Piston forces

FEA can produce realistic results if the inputs are accurate and realistic. Thus, the biomechanical simulations in OpenSim are used to extract the piston forces at the socket-stump interface. The piston forces can then be input as loads in the finite element models to study the pressure and frictional stress at the socket-stump interface.

3.3.1 Extracting piston forces

Steps to extract piston forces from biomechanical model:

1. Load the .osim file of transfemoral amputee into OpenSim.

2. Select Inverse Dynamics from the drop-down menu of Tools, select a .mot file with the marker data and a .xml file with the external load data. Set the filter coordinates as 12 Hz and run the Inverse Dynamics as shown in Figure 3.10.

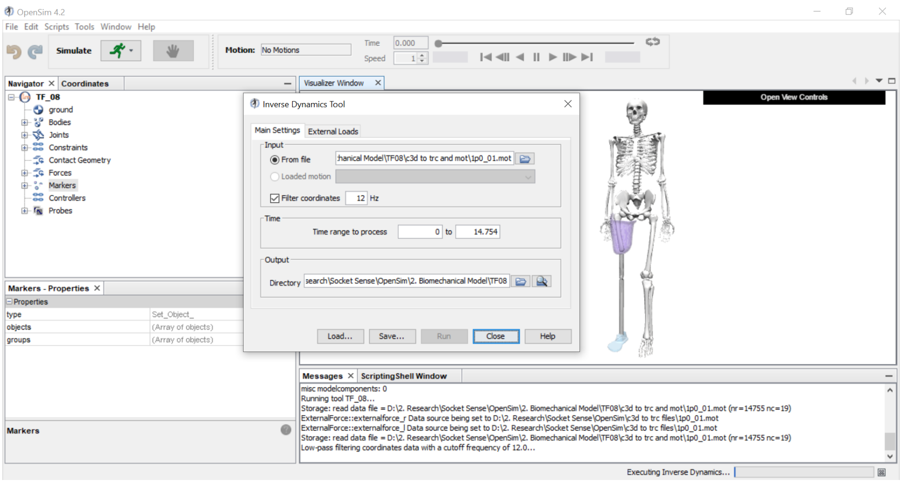


Figure 3.10: Running inverse dynamics

3. Select Plot from the drop-down menu of Tools and load the "inverse_dynamic.sto" file into the plotter. Select socket_piston_x_force, socket_piston_y_force, socket_piston_z_force in the Y-Quantity and select time in the X-Quantity as shown in Figure 3.11. Click Add to plot all the graphs.

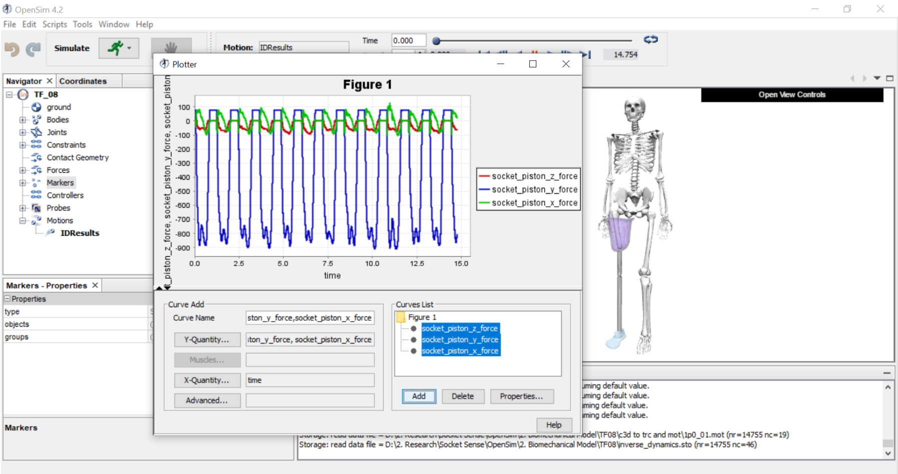


Figure 3.11: Plotting piston forces

3.3.2 Piston force results

The piston forces extracted from biomechancial simulations in OpenSim are plotted in Figure 3.12. Similarly, the piston forces for different cases can be generated to input in finite element models.

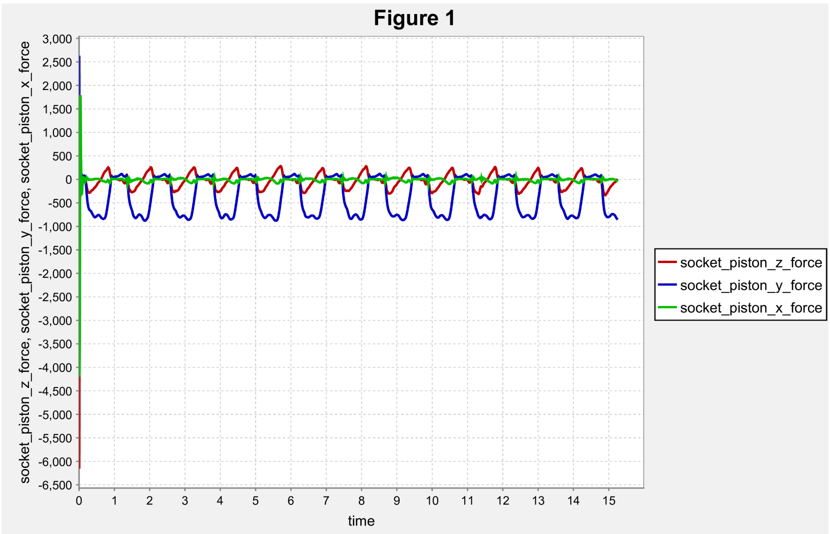


Figure 3.12: Piston forces at the socket-stump interface

Chapter 4

Conclusions and Future Work

4.1 Finite Element Analysis

CAD files of the amputated Femur, stump and socket have been uploaded to **Box** for future reference.
The archive file of FEA in ANSYS has also been uploaded to **Box** for future reference.

The following conclusions can be made about the FEA procedure presented in Section 2.2, Section 2.3 and Section 2.5:

- In future work, more complex hyperelastic material models can be used to define the material properties of the residual limb.
- In future work, different values of the friction coefficient can be implemented in the FEA.
- In future work, the finite element models can be simulated for different loads acting on the stump, during different stages of gait.

4.2 Biomechanical Modelling

All .c3d files and the converted .trc and .mot files of TF08 have been uploaded to **Box**.

The following conclusion can be made about the method of converting .c3d file to .trc file presented in Section 3.2.1:

- In future work, the MATLAB scripts can be automated to work without manual input.

The following conclusion can be made about the method of converting .c3d file to .mot file presented in Section 3.2.2:

- The use of .exe file will make the automation of the task comparatively difficult in future work.

Bibliography

[1] B. Makis, *Transfemoral (above-knee) quadrilateral prosthetic socket pack 2*, 2021. [Online]. Available: https://www.thingiverse.com/thing:3233555.

[2] *Human legs — 3d model*, 2016. [Online]. Available: https://www.cgtrader.com/free-3d-models/character/anatomy/human-legs--2.

[3] S. Henao, C. Orozco, and J. Ramirez, "Influence of gait cycle loads on stress distribution at the residual limb/socket interface of transfemoral amputees: A finite element analysis," *Scientific reports*, vol. 10, no. 1, pp. 1–11, 2020.

[4] L. Zhang, M. Zhu, L. Shen, and F. Zheng, "Finite element analysis of the contact interface between trans-femoral stump and prosthetic socket," in *2013 35th annual international conference of the IEEE engineering in medicine and biology society (Embc)*, IEEE, 2013, pp. 1270–1273.

[5] A. M. Willson, "A quasi-passive biarticular prosthesis and novel musculoskeletal model for transtibial amputees," Ph.D. dissertation, 2017.

[6] S. Hood, M. K. Ishmael, A. Gunnell, K. Foreman, and T. Lenzi, "A kinematic and kinetic dataset of 18 above-knee amputees walking at various speeds," *Scientific Data*, vol. 7, no. 1, pp. 1–8, 2020.

[7] F. Alvim, *Convert .c3d and .csv files to opensim files without any program but matlab*, 2015. [Online]. Available: https://simtk.org/frs/?group_id=1019.

[8] *C3d tools / extract experimental data that compatible with opensim*, 2016. [Online]. Available: http://bsnlab.com/c3d-tools-2/.