2014

Predicting Pressure Distribution Between Transfemoral Prosthetic Socket and Residual Limb Using Finite Element Analysis

Rajesh Surapureddy
University of North Florida, rajesh_surapureddy@yahoo.co.in

Predicting Pressure Distribution between Transfemoral Prosthetic Socket and Residual Limb Using Finite Element Analysis

By

Rajesh Surapureddy

A thesis submitted to the School of Engineering in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering

University of North Florida

College of Computing, Engineering and Construction

December 19, 2014

The thesis "Predicting Pressure Distribution between Transfemoral Prosthetic Socket and Residual Limb Using Finite Element Analysis" submitted by Rajesh Surapureddy has been

Approved by the thesis committee Date

Alexandra Schonning, PhD
Thesis Advisor and Committee Chairperson

Alain Kassab, PhD, Committee Member

Stephen Stagon, PhD, Committee Member

Accepted for the School of Engineering

Murat Tiryakioglu, PhD, CQE
Director of the School of Engineering

Accepted for the College of Computing, Engineering and Construction

Mark A. Tumeo, PhD, JD, PE
Dean of the College of Computing, Engineering and Construction

Accepted for the University

John Kantner, PhD
Dean of the Graduate School

ACKNOWLEDGEMENTS

Foremost, I would like to express my deepest thanks to my advisor, Dr. Alexandra Schonning for her supervision, encouragement, and support throughout this work. Her timely supervision, guidance, thorough and excellent feedback from time to time allowed this research to be completed on time. She has always found the time to propose consistently excellent improvements and had patience to advise me in accomplishing this work in spite of her busy schedule. I am so deeply grateful for her help, professionalism and valuable guidance through my entire program of study, that I do not have enough words to express my deep and sincere appreciation. This thesis would not have been possible without her support. I express my gratitude to Dr. Stephen Stagon to serve in my thesis committee and for his valuable discussions, guidance and encouragement. I also express my sincere gratitude to Dr. Alain Kassab to serve in my thesis committee and for providing his valuable insights and invaluable support.

Also, I am thankful to Dr. Murat Tiryakioglu for his encouragement and guidance throughout my course of study. I would like to thank my friend, Michael Shelly for his guidance and valuable time in reviewing my thesis.

Finally, on a more personal note, I would like to thank my wife Harsha for her support, encouragement, quiet patience and unwavering love throughout my years of study and through the process of researching and writing my thesis. I thank my parents and parents-in-law for their faith in me and allowing me to be as ambitious as I wanted. This accomplishment would not have been possible without them.

Contents

Table of Figures

Abstract

In this study, a non-linear Finite Element (FE) model was created and analyzed to determine the pressure distribution between the residual limb and the prosthetic socket of a transfemoral amputee. This analysis was performed in an attempt to develop a process allowing healthcare providers and engineers to simulate the fit and comfort of transfemoral prosthetics to reduce the number of re-fittings needed for the amputees. The analysis considered the effects of interference due to insertion of the limb into the prosthesis, referred to as donning, and also the effects due to the body weight of the amputee. A non-linear finite element static implicit analysis method was utilized. This analysis implemented multiple finite element techniques, including geometric non-linearity due to large deflections, non-linear contacts due to friction between the contact surfaces of the residual limb and the socket, and non-linear hyper-elastic material properties for the residual limb's soft tissue. This non-linear static analysis was carried out in two time-steps. The first step involved solving the interference fit analysis to study the pre-stresses developed due to the effect of donning. The donning process results in soft tissue displacement to accommodate the internal geometry of the prosthesis. In the second load application time-step, an additional load of half the person's body weight was applied to the femur. The maximum normal stress (contact pressure) of 84 kPa was observed due to the combined effect of the donning procedure and body weight application, comparable to the studies performed by other researchers. The procedure developed through this work can be used by future researchers and prosthetic designers in understanding how to better design transfemoral prosthesis.

Chapter 1: Introduction

An amputation that occurs through the femur is known as transfemoral amputation or Above Knee (AK) amputation. A transfemoral prosthesis is used as an artificial limb to restore amputees' mobility functions for their daily life activities. The transfemoral prosthesis is in contact with part of the above-knee residual limb. The uppermost part of the prosthesis is called the prosthesis socket, which surrounds the residual limb and acts as a medium to transfer the load from the residual limb to the prosthesis [1]. The skin and the soft tissue of the residual limb experiences severe stress and excessive distortion during gait positions such as sitting, standing, taking steps, and walking [2]. Great care should therefore be taken in designing the prosthesis to ensure that the pressure between the socket and stump is minimized. The fitting of the prosthesis is an empirical process and varies for each patient due to the complexity of the residual limb geometry and the pressure tolerance level of the amputee [2]. Proper fitting of the prosthetic socket to the residual limb is one of the most important stages in the process of rehabilitation, as miss-fitting the prosthetic can cause an uncomfortable pressure distribution. Knowledge of the pressure distribution at the interface between the residual limb and prosthetic socket helps in understanding how to improve the design of the prosthetic socket for a better fit.

The primary objective of this study was to develop a procedure which will allow prosthetic designers to evaluate a patient's transfemoral prosthesis fit analytically and make scientifically based decisions on how to modify the prosthesis for enhanced fit. This is hoped to reduce the number of re-fittings needed for the patient. In an effort to achieve this objective, the pressure distribution between the prosthesis socket and residual limb were analyzed using the finite element analysis (FEA) method. The analysis employed

hyperelastic material properties to model the soft tissue of the limb. Additionally, a two-step load application procedure separated the donning process from the result of applying the weight of the person while standing. The finite element (FE) model was developed and solved using the nonlinear static standard implicit method of Abaqus 6.13 CAE. This software was also used in post-processing of the results. Figure 1 shows the separate volumes (prosthetic socket, soft tissue of the residual limb, and femoral bone) included in the FE model.

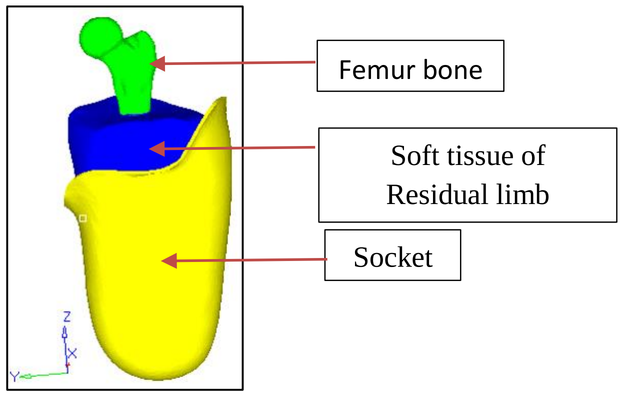


Figure 1: Transfemoral prosthesis and residual limb

Chapter 2: Literature Review

A thorough literature review was performed on topics related to the finite element analysis of transfemoral prosthetics in an effort to learn how others have modeled the prosthetic-limb interface to predict the pressure distribution between the limb and the socket. The literature search revealed that research on using finite element methods to model transfemoral prosthetic sockets was limited when compared to transtibial prosthetic sockets also known as Below Knee (B.K.) prosthetic sockets. Only a few sources were found on finite element modeling of the interface of Above Knee prosthetics [3-6].

2.1- Loads and Boundary Conditions

In the early 1990's, Zhang [7] developed a nonlinear finite element model to determine the pressure and shear stress distribution at the limb-socket interface in Transtibial (below-knee) amputees. The model considered friction and slip at the interface between the skin and socket liner. A static vertical load equivalent to the subject's full body weight was applied on top of the stump in the study. Zhang's study [7] concluded that the coefficient of friction is a very sensitive parameter in determining the interface pressures, shear stresses and slip. Later, in 1996, Silver-Thorn and Childress [8] performed a parametric analysis using FEA to investigate the prosthetic interface stresses for persons with transtibial amputations. This parametric analysis provided useful information in understanding the residual limb prosthetic-socket interface mechanics and the influence of various parameters such as frictional coefficients, external loads, socket stiffness, and effects of individual limb variations on the interface stresses. In 1997, Silver-Thorn and Childress developed a generic FE model to provide a quantitative estimation of prosthetic interface pressure. The load state in their study was limited to static stance with the load supported equally by both the prosthetic and physiologic legs [2]. Jia *et al.* [9] performed a FE study on the effects of inertial loads on the interface stresses between a transtibial residual limb and prosthetic socket using Abaqus. Jia's study focused on the pre-stress effect due to the donning process, but also included the application of weight forces while walking. The socket was modeled as rigid in the study and all materials were assumed linear. Lee *et al*. [10] performed a nonlinear contact analysis on transtibial prostheses to study the importance of considering the pre-stress in predicting the interface stresses at the loading stage for different prosthetic shank designs. A vertical force was applied at the

prosthetic foot for this simulation. In their work, at different walking phases, the effect of stiffness of transtibial limb shanks on the stress distribution of limb and socket were studied. In a different study, Patino *et al*. [11] focused on determining how the type of model used to represent the contact between bone and soft tissue affect the stresses. The loading condition in his study was a vertical load and assumed half of the weight of a person. He concluded that models including friction contacts resulted in higher stresses than those which used tie contacts [11]. The study performed by Linin Zhang *et al.* [4] focused on predicting the stress distribution between the socket and the residual limb. In his study, in order to determine the effect of pre-stress, a vertical load of 50 N was applied in the first time step at the top of the residual limb while constraining the lower surface of the socket. During the second time-step, the upper surface of the residual limb was constrained in all directions and the vertical load was applied on the bottom surface of the socket to simulate three different loading conditions such as foot flat, mid-stance and heel off. To summarize, most researchers analyzing transfemoral and transtibial prosthesis apply a load equivalent to half (400 N) or full body weight (800 N) at the femoral head or they apply forces equivalent to the reaction forces extracted from larger FE models [4, 7, 8, 10-14].

2.2- Material Modeling

Most previous FE studies of the pressure distribution between a prosthesis and transfemoral and transtibial limbs have used linear material properties for tissues, socket, and bone [7, 8, 10, 12, 13, 15]. It is observed that using linear elastic material properties instead of hyperelastic properties would alter the results of the pressure distribution on the limb. Simpson *et al.* [16] modeled the soft tissue of the transtibial residual limb as homogeneous, nonlinear, and hyperelastic in order to study what effect the material model of the soft tissue has on the pressure redistribution during the donning process. Tonuk and Silver-Thorn [17] performed research to estimate the nonlinear viscoelastic material coefficients of residual limb soft tissue using force-relaxation and creep data from vivo indentation studies conducted on individuals with transtibial amputation. A nonlinear viscoelastic material model can be used for large deformation analysis and for longer loading times such as creep, than are possible with a linear material model. Portnoy *et al.* [18] used hyperelastic constitutive equations for the soft tissue in their work on transtibial prosthesis. In 2011, Patino and Lacroix [6], developed five models from five different patients to study the effect of the donning procedure on the stress-strain state at the interaction between the socket and residual limb. Patino and Lacroix performed time dependent quasi-static explicit analysis with hyperelastic properties for the soft tissue as reported by Portnoy *et al*. [18].

2.3- FEA on Transfemoral Prosthesis

Limited research findings are reported related to finite element modeling of the transfemoral socket-limb interface [3-6]. In Chronological order, in 1996, Zhang and Mak

[3] performed 2D nonlinear static analysis to study the effect of variation of distal end loading on stress distribution between the transfemoral socket and residual limb. Zhang and Mak considered the donning effect and the weight bearing state during stance on transfemoral prosthesis. The donning process was simulated by adding or subtracting a radial displacement to the nodes on the external surface of the socket. Zhang and Mak's study was the first of its kind on transfemoral prosthetic sockets, though there were some limitations to their study such as 2D rather than 3D modeling. They also assumed a homogenous, isotropic, linear material property for the residual limb soft tissue. Researchers M.Tanaka *et al.* [5] performed a nonlinear analysis to identify the pressure distribution at the socket interface of transfemoral prosthesis. The major limitation in the study by M.Tanaka *et al*. was the application of linear properties for the soft tissue. The work performed by Patino and Lacroix focused on analyzing the donning procedure with enforced displacement applied on the socket [6]. Their work utilized hyperelastic material properties for soft tissue and gave insight into the pressure distribution between the socket and the prosthesis as well as regions of stress concentrations. However, the effect of the initial overclosure was not addressed. Linlin Zhang *et al*. [4] have attempted nonlinear FE modeling by applying a hyperelastic material for the soft tissue. Their study adopted a generalized assumption that the geometry shapes of the residual limb's outer surface and the socket's inner surface geometry are the same [19] for simulating the pre-stress condition. Vertical loads were applied at the bottom surface of the socket to simulate the loading condition. Table 1 summarizes the literature on FE research related to the pressure distribution between the residual limb and transfemoral prosthetic socket.

Table 1: FEA Research on transfemoral prosthetic socket

Researchers	**Load steps**	**Material property for residual limb soft tissue**	**Mesh methodology**	**Geometry**
Ming Zhang and Arthur F.T. Mak [3]	Interference step, vertical load step	Linear	2D	Generic, uniform overclosure between residual limb and the socket
M.Tanaka *et al.* [5]	Pressure load step	Linear	2D	Generic
Patino J.F and Lacroix D. [6]	Enforced displacement	Hyperelastic	3D	Patient specific
Linlin Zhang *et al.* [4]	Vertical load steps	Hyperelastic	3D	Patient specific, uniform overclosure between residual limb and the socket

Chapter 3: Current Study Overview

The current study focused on performing an analysis which combined the work performed by Patino *et al*. [6] with that conducted by Zhang and Mak [3]. That is, a nonlinear static analysis using a hyperelastic material model for the soft tissue was performed to determine the pre-stresses due to the donning procedure as well as the stresses developed due to body weight application. The interference between the transfemoral socket (red in color) and the residual limb (blue in color) due to the difference in geometric shapes, also known as overclosure, is shown in Figure 2. A simplification of the current model was that high density linear tetrahedral elements, rather than quadratic tetrahedral elements were used to reduce the computational cost. This simplification was based on the displacement results for quadrilateral tetrahedral elements (1.06 mm) vs linear tetrahedral elements (1.06 mm) presented in a comparative study of high density mesh linear tetrahedral and quadratic tetrahedral elements [20], which concluded that these modeling methodologies generate similar results. The model created for this study utilizes high density linear elements and the mesh was verified to converge. The results of this study are not specific to one particular individual, therefore applying the procedure developed

herein to a specific individual's anatomical geometry and prosthesis may lead to results which can guide the redesign of the prosthesis socket.

In the first time-step of load application, the interference fit analysis resolves the initial overclosure between the residual limb and the socket. An automatic shrink fit algorithm in Abaqus was used to remove slave node over-closure gradually during this load step. The second load application time-step involved applying an external pressure load of 0.4 MPa to the top of the femur. This pressure has a force magnitude of 400 N, equivalent to half of the body weight as experienced on the amputated limb while standing.

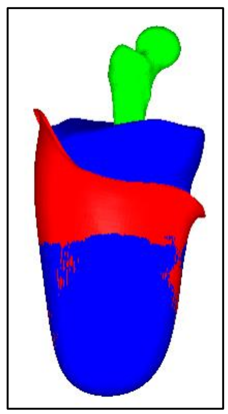


Figure 2: Initial over-closure of the residual limb and socket

In summary, the current study advances the state of the art of transfemoral socket FEA methodology and the work performed by Zhang and Mak [3], Patino and Lacroix [6]. Table 2 shows the details of load steps, material properties for residual limb soft tissue, mesh methodology and the geometry considerations in the current thesis work.

Table 2: FEA research methodology on current work

Researcher	Load step	Material property for residual limb soft tissue	Mesh methodology	Geometry
Rajesh	Interference step and Pressure load step	Hyperelastic	3D	Generic, non- uniform overclosure between residual limb and the socket

In comparing Table 2 to Table 1, it is clear that the work presented herein advances the current state of the art by improving the load application, material model, and by utilizing 3D elements.

Chapter 4: Finite Element Analysis Methodology

In developing the finite element model used herein, the model geometry was first obtained and refined, the pre-processing stage of the FE model was then completed by creating a mesh, applying load and boundary conditions, defining element types, contact and material models. Lastly, the analysis was run and the results were post-processed. This section is subdivided into sections describing each of these steps in more detail.

4.1- Data Acquisition and Geometric Smoothing

The femur bone data was imported in the form of an .iges file, which was developed by A. Schonning and her team [21]. The team generated the bone geometry from Computed Tomography (CT) data, available from the National Institutes of Health’s (NIH) Visible Human Project [22]. The residual limb geometry was developed by surface fitting point cloud data generated by scanning a physical model of a person’s transfemoral stump provided by the High-Performance Materials Institute of the FAMU-FSU College of Engineering. The socket model was developed by scanning a transfemoral prostheses

provided by Bremer Brace. The scanning was performed using a MicroScribe G2 Digitizer, a type of coordinate measuring machine.

The femur CAD model was developed using Mimics [23] by first analyzing the CT data's density levels and developing 2D slices for the bone. Curve fitting algorithms were then employed in interpolating the geometry between the slices in generating a 3D model. The femoral bone was cut in a manner consistent with a femoral amputation [24]. The 3D models of the soft tissue and prosthetic socket were developed using curve fitting techniques available in Geomagics [25]. Smoothing algorithms were used on all three models (femur, socket, and soft tissue) to ensure the models were closed and to remove noise from the scanning and fitting processes. Non-Rational B-spline Surfaces (NURBS) were generated for each of the polygon models. This step consisted in defining NURBS patches based on geometry curvature. The NURBS surfaces were finally exported as .iges files to be used in the Finite Element Model. Images of the final CAD models used in the FEM model are depicted in Figure 3.

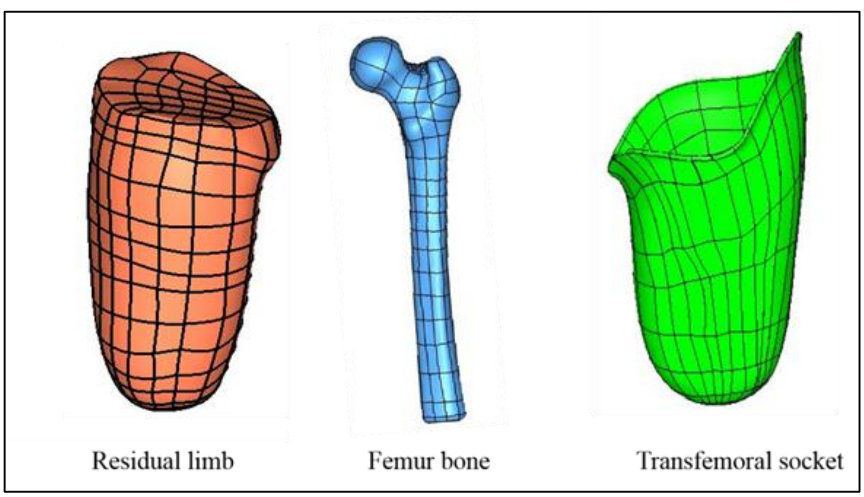


Figure 3: CAD geometries

4.2- Finite Element Mesh Details

In developing the FE mesh, the element types for each component were selected and applied, the material model was chosen, and a mesh convergence was performed. The following sections provide details on the element types, material model, mesh convergence, and the final meshed volumes.

4.2.1- Element Types

Tetrahedral meshes were generated on the three components (femur, tissue, and socket). Tetrahedral meshes are generally preferred over hexahedral meshes for free-formed complex geometries as the former are computationally more cost effective [26], and easier to apply. In the present study, generating hexahedral meshes would be problematic due to the complexity in the geometric shapes of the femur bone, residual limb and socket. Linear tetrahedral elements were used for the three volumes. Due to the large node count of 65000 nodes resulting in an average run time of ≈ 24 hours using linear tetrahedral elements, second order tetrahedral elements were not attempted as they would be computationally prohibitive. Comparative analysis of high density tetrahedral meshes and quadratic meshes performed by Tortworth Court support the use of linear elements as both meshes yield similar results [20]. A mesh convergence study was performed to select an appropriate element size as 3mm. The fine meshing was performed using C3D4 tetrahedral elements (C-continuum, 3D-three dimensional element, and 4- four noded element) for the socket and femoral bone, while C3D4H tetrahedral elements (C-continuum, 3D- three dimensional element, H-hybrid element and 4- four noded element) were used for the residual limb. The hybrid (H) element was used in modeling the residual

limb since a hyperelastic material model was applied to these elements. The use of the hybrid element is consistent with the recommendation by Simulia [27] as the hybrid elements in Abaqus/Standard are intended primarily for use in incompressible and nearly incompressible material models.

4.2.2- Material Models

The mechanical properties of the femur bone and socket were assumed to be linearly elastic, and as such obey Hooke's law in which strain varies linearly with the stresses developed in the elastic body. The femur and socket materials were modeled as isotropic, with all elastic properties uniform in all directions. Finally, these volumes were assumed to be homogenous with consistent materials properties throughout. The femur bone was modeled with a Young's modulus of 150,000 MPa and a poisons ratio of 0.3. These values are consistent with how bone was modeled by other researchers [7, 10, 12, 28, 29]. The prosthesis socket was modeled with a Young's modulus of 15,000 MPa and a poisons ratio of 0.3, also consistent with literature [3, 10, 30, 31].

In recent years, more advanced materials models such as hyperelastic models have been used in modeling biological soft tissues. Hyperelastic material is also known as Cauchy-elastic material, which means that the stress is determined by the current state of deformation, and not the path or history of the deformation [32]. Generally, the hyperelastic materials are independent of strain rate and are described in terms of a "strain energy potential". Strain energy potential is defined as the strain energy stored in the material per unit of reference volume as a function of the strain at that point in the material. Materials

like rubber, elastomers and biological tissues are often modeled using hyperelastic material models [18, 33, 34].

Hyperelastic material models are classified as phenomenological, in that they describe the observed material behavior, and they are also mechanistic meaning that the model characteristics are derived from the underlying structure of the material [35]. Marlow, Mooney-Revlin, Ogden, Polynomial, Saint Venant Kirchoff and Yeoh models are few examples of phenomenological hyperelastic models. Arruda-Boyce and Neo-Hookean fall into the category of mechanistic hyper elastic models. The Vanderwalls hyperelastic model is an example of a hybrid model which combine the properties of both phenomenological and mechanistic models [35].

To use any of these constitutive relations, it is required to determine values for the material constants. It is relatively simple to determine the material constant in Neo-Hookean material models as they only have one constant. It is a more laborious task to determine the material constants used in generalized polynomial equations, such as the Ogden and Mooney-Revlin material models. In general, tests are performed on a sample of material specimens. These tests include tension and shear or volumetric compression. Predicted stress-strain behavior is calculated for the specimen for each constitutive law. The material parameters are later estimated by curve fitting the results of the test data and choosing the material model which best fits the test results [36]. The simplest form of the hyperelastic material model is Neo-Hooken solid (fully incompressible), expressed below.

$$W = \frac{\mu_1}{2}(\, I_1 - 3)$$

Where,

W = the strain energy density; and

μ_1 = shear modulus, which can be determined by experiments.

I_1= first order deviatoric strain invariant

The Mooney-Rivlin strain energy potential [39] form is expressed as:

$$W = C_{10}(I_1 - 3) + C_{11}(I_1 - 3)(I_2 - 3) + \frac{(J-1)^2}{D_1}$$

Where,

W= strain energy per unit reference volume

I_1= first order deviatoric strain invariant, I_2= second order deviatoric strain invariant

$I_1=\lambda_1^2+ \lambda_2^2 +\lambda_3^2$ and $I_2=\lambda_1^{-2}+ \lambda_2^{-2} +\lambda_3^{-2}$

$\lambda_1, \lambda_2, \lambda_3$ are principal stretches

C_{10}, C_{11}, D_1 are constitutive material parameters

$J=\lambda_1 * \lambda_2 * \lambda_3$.

The Ogden material model [37], under the assumption of incompressibility, is more generalized and is expressed as:

$$W(\lambda_1, \lambda_2) = \sum_{p=1}^{N} \frac{\mu_p}{\alpha_p} \left(\lambda_1^{\alpha_p} + \lambda_2^{\alpha_p} + \lambda_1^{-\alpha_p} \lambda_2^{-\alpha_p} - 3 \right)$$

Where,

W = strain energy density;

N, μ_p, and α_p = material constants; and $\lambda_1, \lambda_2, \lambda_3$ are principal stretches

For particular values of material constants, the Ogden model will reduce to either a Neo-Hookean solid (N=1, α=2) or the Mooney-Rivlin material (N=2, α_1=2, α_2= -2) with the constraint condition $\lambda_1 \lambda_2 \lambda_3 = 1$.

In general, for moderate strains, the Neo-Hookean (one variant) model fits for the material behavior with sufficient accuracy. For higher strains, such as in biological tissues, Neo-Hooken is replaced by Mooney-Revlin, which has two invariants for an accurate fit of the soft tissue behavior [38]. In this analysis, the residual limb soft tissue was assumed to be a homogenous, isotropic, hyperelastic material and was modelled using Mooney-Revlin solid strain energy function [39], the same function used by Patino and Lacroix [11]. For the soft tissue at hand, the constants used in the Mooney-Revlin model were C_{10} = 4.25 kPa, C_{11} = 0 kPa, and D_1 = 2.36 MPa^{-1}. These values were established by Portnoy [18], who studied the average flaccid muscle. Table 3 provides the material properties that were applied for the femur bone, socket, and residual limb.

Table 3: Material properties for bone, socket and residual limb

	Young's modulus (MPa)	Poisson's ratio
Bone	150,000	0.3
Socket	15,000	0.3
Residual limb	C_{10} = 4.25 kPa, C_{11} = 0 kPa, D_1 = 2.36 MPa^{-1}	

4.2.3- Mesh Convergence

One of the important steps in FE modeling is to ensure that the mesh density is fine enough for the mesh to converge. A too coarse mesh utilizing too larger elements will yield inaccurate results while a mesh with unnecessarily small elements will increase the cost of the analysis in terms of time and usage of computer and software resources. Mesh convergence is an iterative process which applies consistent load application and boundary conditions while altering element size until the results of the analysis converge on a similar result for two mesh sizes. When further mesh refinement produces no change in the results, or only changes them below a determined threshold, the mesh is sufficiently converged. This process can be visualized by plotting a curve of element size vs. stress results. The optimum mesh density is defined at the point on the curve at which the percent difference from one analysis to the next is below a specified threshold.

The mesh convergence process was applied on the residual limb as it is the volume likely to observe highest deformation and is the volume of most interest. The element size for the other parts such as femur and socket was 5mm and was consistent with the research study completed by Patino and Lacroix [6]. The mesh convergence study was performed with a vertical load of 100 N at the top of the residual limb and by constraining the bottom of the limb. The analysis (iterative) was performed by maintaining constant applied load and boundary conditions while varying the mesh size. The maximum von-Mises stress observed near the bottom of the residual limb in the first iteration (with element size 6mm) was 61 kPa and for the other iterations, the von-Mises stresses were reported at the same location for consistent mesh convergence study. With a threshold error of 1%, the mesh analysis yielded that the mesh converged with 229340 elements. This mesh has an element

size of 3mm, which is consistent with converged meshes of other researchers [6, 11]. Table 4 shows the details of the mesh convergence iterations, including element size, number of elements and nodes, highest von Mises stress, and percent error. Figure 4 shows how the relationship between the stress and number of elements in a graphical manner.

Table 4: Mesh convergence check

Iteration	element size (mm)	number of elements	number of nodes	Von Mises stress (kPa)	Error calculation, % (Von Mises Stresses)
1	6	52021	11834	61	14
2	5	65398	14675	71	6.5
3	4	126141	27696	76	3
4	3	229340	50332	78.4	.75
5	2	268425	58427	79	-

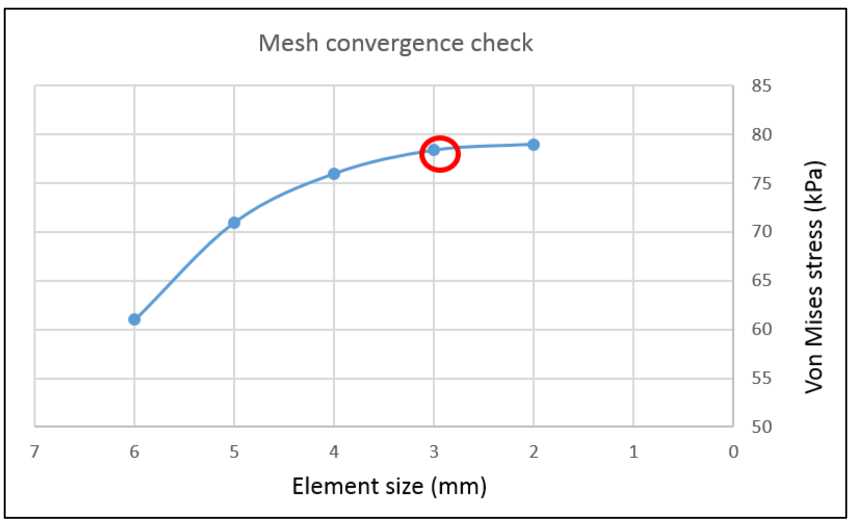


Figure 4: Mesh convergence plot: Element size vs Von Mises stress

4.2.4- FE Meshes

After determining the element size, element type and material model, the three volumes (femur bone, socket, and the residual limb) were meshed with tetrahedral elements in Abaqus, each shown in Figure 5. The nodal distance at the outer surface for all parts was uniform and consistent (3mm for residual limb, 5mm for femur and socket), whereas the element sizes in the inner volumes of the parts was non-uniform. The total number of nodes, elements, and element types for the volumes are specified in Table 5.

Table 5: Details of FE model

Part name	Number of nodes	Number of elements	Element type
Femur Bone	2943	11086	C3D4
Residual limb	50332	229340	C3D4H
Socket	11775	35126	C3D4
Total Assembly	65050	331136	

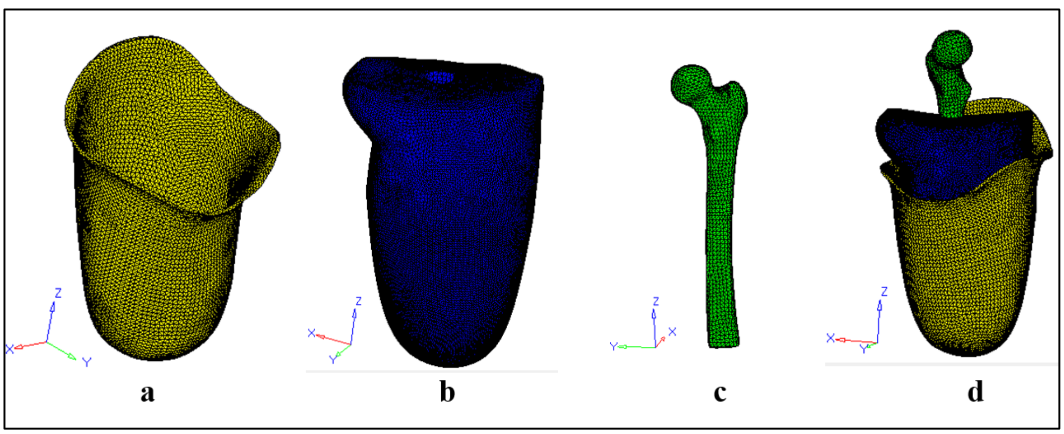


Figure 5: FE models – a) socket, b) residual limb, c) femur bone, and d) final FE model

4.2.5- Contact Conditions

The physical frictional behavior between the residual limb and the socket was represented within the FE model by using contact pair definitions in Abaqus. Generally, contact pairs are defined between the surfaces of two bodies that could possibly come in contact due to loading conditions. Abaqus/standard provides small sliding contacts and finite sliding contact definitions to define a relative motion between two surfaces. Small sliding contact assumes that a relatively small sliding occurs between two surfaces, while finite sliding contact allows large deformations between two surfaces. In this analysis, since large deformations are likely to occur on the residual limb, finite sliding contact was defined between these two components. In Abaqus, by default, surfaces interact with each other along the surface normal direction in order to resist penetration. A frictional property was defined, so that when surfaces are in contact, normal forces, as well as shear forces, will be transmitted between the two contact surfaces. This frictional coefficient makes it possible to determine the contact pressure as well as normal and tangential stresses.

Two contact conditions were defined in the current FE model to perform nonlinear analysis. The first contact definition was a surface-to-surface contact between the residual limb and the socket (as shown in Figure 6 a). Generally, the more rigid and stiffer surface of the contact pair is defined as the master surface (brown in color), while the deformable surface with softer material is selected as the slave surface (purple in color). Hence, the outer surface of the residual limb was defined as the slave surface and the socket's inner surface as the master surface. The contact definition requires that the slave surface conform to the master surface, therefore it is recommended that a finer mesh be applied over slave surface and coarser mesh over master surface. This modeling approach insures that the

slave surface restricts the penetration of the master nodes into slave surface [27]. A coefficient of friction of 0.5 was assigned as an interaction property for the contact surfaces, as was justified in Lee study [10].

The second contact definition applied a tie contact between bone and limb shown in Figure 6b. A tie contact is a surface based constraint using a master-slave formulation. Tie contact provides a simple way to bond surfaces together permanently, which prevents slave nodes from separating or sliding relative to the master surface. Here, the outer surface of the bone was defined as the master surface (brown in color) and the inner circumferential surface (purple in color) of the limb was the slave surface.

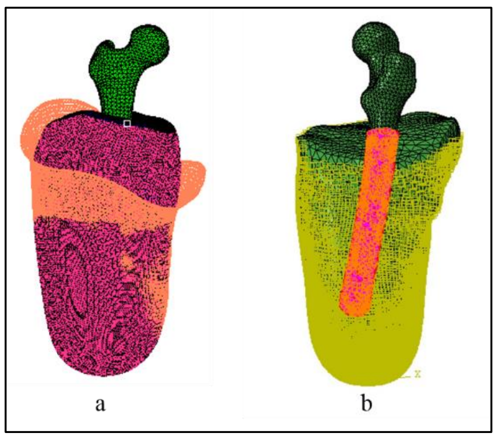


Figure 6: a) Frictional contact. b) Tie contact

4.2.6- Loads and Boundary Conditions

The analysis for the project at hand was carried out in two phases. The first phase was to perform an interference analysis to simulate the overclosure effect during the donning procedure and the second phase was to apply the body weight to the limb-prosthesis assembly. Both of the loads are described in the following subsections.

To solve this nonlinear static analysis, Abaqus standard uses the Newton-Raphson method which employs incremental and iterative procedures. In static analysis, the total load is applied incrementally in a series of steps while the solver performs iterations in an attempt to find the intermediate equilibrium solution for each of these incremental steps. The loads are defined as a function of time and Abaqus chooses suitable time increments automatically. In this study, we defined the initial increment size in Abaqus/Standard as 0.1. Abaqus uses empirical algorithms to control the size of increment during analysis. If Abaqus finds a solution for a given increment of applied load the increment size is increased by 1.5 times, if not, the next increment size is reduce by 25 percent [27].

Phase-1: Donning Procedure

This load step simulates the person putting on the prosthesis (donning) and helps determine how the soft tissue is deformed during this process to allow for fitting inside of the prosthesis. The load applied in this step was not a force, but rather a contact enforcement. Initially, there was some overclosure of 3.47mm, meaning that volume of the soft tissue crosses the boundary of the prosthesis socket. A shrink-fit method was used in which the overclosure was resolved gradually during the first phase of the analysis. Contact definition was defined between the interacting surfaces of the socket and the limb, so that there will not be any overlap or interference after the first load application phase. The interference fit analysis cause stresses and strains to develop in the model as the overclosure in the model was resolved. At the end of the analysis, the nodes of the slave surface (residual limb contact surface) were moved so that they precisely contact the master surface (socket inner surface). This analysis was performed by applying two boundary conditions with no external load. The bottom of the socket was constrained in all degrees

of freedom as shown in Figure 7 to resemble the physical scenario as the link at the bottom of the socket is fixed. The second boundary condition was applied in order to constrain the top of the femur bone in all directions as shown in below Figure 7. This constraint forces the soft tissue to deform, rather than allowing the limb to translate out of the prosthesis. The second boundary condition was removed in the second load phase of the analysis when the weight was applied. The interference phase was modeled under frictionless conditions. However, friction between the contact surfaces was introduced in the next load phase. At the end of the first phase, due to the overclosure affect, a contact pressure developed resulting in pre-stress on the limb due to the overclosure effect. The stresses developed and the deformation in the first phase was retained and propagated to the next analysis step.

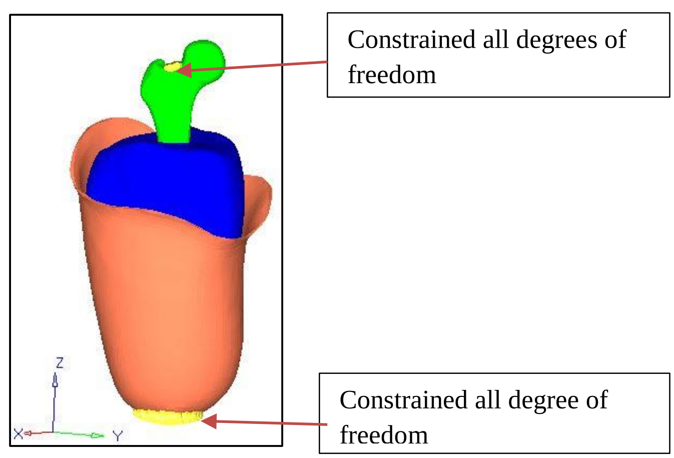


Figure 7: Boundary conditions for load step-1

Phase 2: Application of Body Weight

In the second phase of the analysis, a generalized loading condition similar to that used by most other researchers was assumed [2, 3, 6, 12, 21]. The fully constrained boundary condition applied to the bottom of the prosthesis during the first phase of the analysis (donning) was retained during this phase, whereas the boundary condition applied to the femoral head was removed in this phase. An external load was applied to the top of the femur in a vertical downward direction to simulate the person's body weight. It was assumed that 50% of the amputee's body weight (400N) was supported by the femur bone, with an equivalent pressure of 0.4 MPa applied on the top surface area of 100 mm^2 of the femur bone as shown in Figure 8. The interaction characteristics of the contact definition between the residual limb and the socket from the first load phase was modified for this second analysis step by introducing a coefficient of friction of 0.5, a value consistent with literature [10].

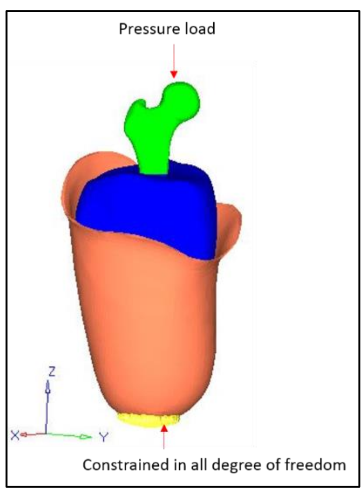


Figure 8: Boundary and loading condition for load step-2

Chapter 5: Results and Discussions

This section outlines the stresses obtained at the interference between the transfemoral prosthesis and limb due to the donning procedure and also due to the body weight application. Following the analysis results is a section outlining the process developed during this work.

5.1- Donning Procedure Results

The effects of the initial overclosure results are shown below. The actual overclosure value at the end of the step were reported using a variable called contact open (COPEN). A positive COPEN value indicates clearance (no interference) between the contact surfaces, while a negative value of COPEN signifies an overclosure or interference condition between the contact surfaces. At the start of the interference step, a maximum contact open value of -3.47 mm was observed indicating a geometric interference between the residual limb and the socket. This result is shown in Figure 9. At the conclusion of interference step, the COPEN value was reduced to nearly zero (-2.73 e-3 mm), this condition is shown in Figure 9. This result indicates that contact overclosure was resolved and the slave surface was moved, so that it contacts the master surface.

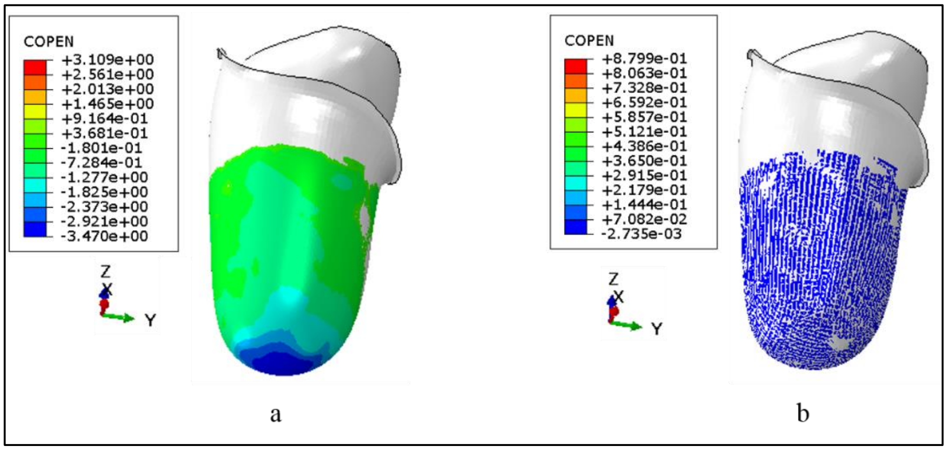


Figure 9: a) Contact Open at start of donning (mm); b) Contact Open at the end of donning (mm)

The residual limb deforms during the donning procedure and therefore stresses develop in the limb. The analysis predicted the maximum interference to occur near the bottom of the residual limb. Not surprisingly, this is also the location of highest contact pressure. Figure 10 shows the distribution of contact pressure on the residual limb at the end of donning, or at the end of the first load application phase. It was found that the highest contact pressure was 3.9kPa where the contact interference was maximum. We also observe other high contact pressure locations because of non-uniform interference between residual limb and the socket. The results of this study cannot be easily compared to that achieved by other researchers since there has been limited work performed on simulating the actual donning procedure for transfemoral prosthetics. Patino and Lacroix studied the donning procedure by performing a quasi-static analysis. However, they did not include the overclosure effect. The contact pressure observed for one of the patients in their study was 4.4 kPa [6], which is close to the value obtained in the current study. However, the stress locations predicted by Patino and Lacroix in their work were different due to differences in geometry as well as loading conditions. The maximum von Mises stresses

observed on the outer surface of the residual limb was 1.2 kPa, as shown in Figure 10b. It was observed that the dominant contact enforcement results in higher contacts pressure than the von Mises stresses. Since this step of the analysis was performed without including frictional effects, resulting shear stresses on the residual limb were not present. However, as seen in the next sections where the results from applying the body weight are presented, shear stresses are present when friction is included in the analysis.

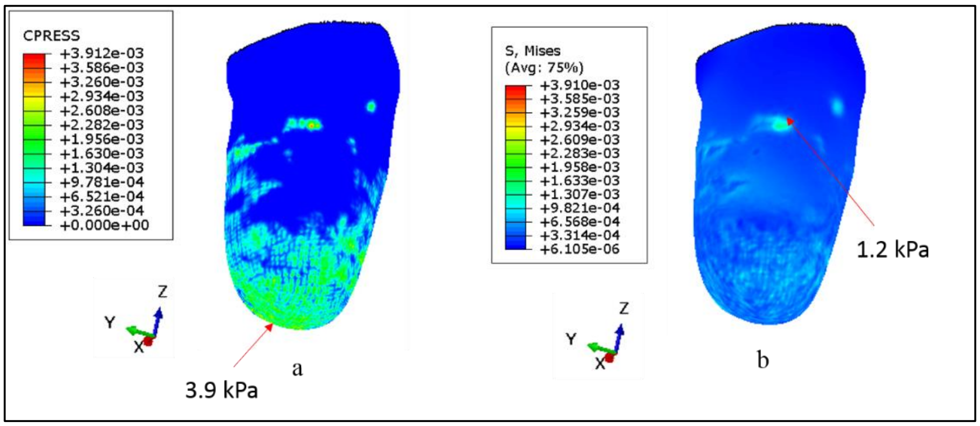


Figure 10: a) Contact pressure observed on the residual limb (kPa), b) Von Mises stress observed on the residual limb (kPa)

5.2- Body Weight Application Results

The stresses and strains produced due to the donning analysis step were retained and carried into the second phase of the analysis during which the body weight was applied. The maximum displacement observed on the residual limb was 45mm as shown in Figure 11. In this second phase of the analysis, the body weight on the femur resulted in increased contact pressure on the residual limb where it contacts the socket. The maximum contact pressure observed was 84 kPa at a location where the maximum contact interference region between residual limb and the socket was observed, as shown in Figure 12. The observed

maximum contact pressure (84 kPa) is significantly lower than the threshold pain observed by Lee and Zhang for a transtibial amputee which is 690 kPa [40].

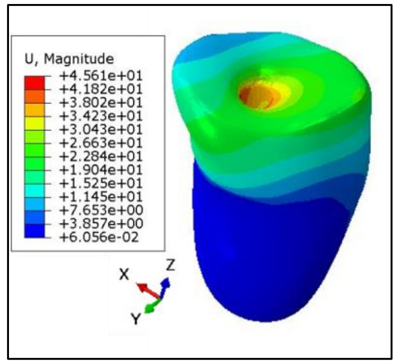


Figure 11: Maximum displacement observed on residual limb (mm)

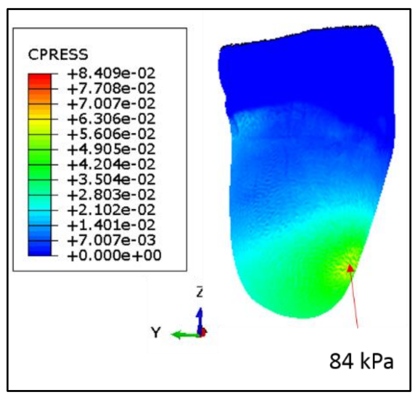


Figure 12: Contact Pressure observed due to donning and body weight (MPa)

The maximum von Mises stress developed on the outer surface of the residual limb was 14 kPa, as shown in Figure 13. In this load step, the contribution of the contact surfaces results in a higher normal stress than the von Mises stress.

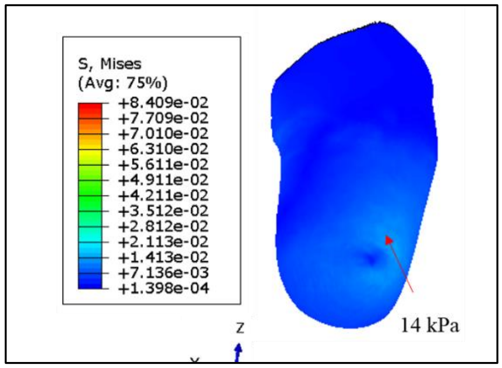


Figure 13: Von Mises stress distribution observed on the outer surface of the limb (MPa)

Unlike in the donning procedure, this analysis step included the application of friction between the residual limb and socket, which resulted in circumferential and longitudinal shear stresses on the residual limb. The maximum circumferential shear stress observed on the outer surface of the residual limb socket was 15.6 kPa as shown in Figure 14a and the maximum longitudinal shear stress was 14.9 kPa, as shown in Figure 14b.

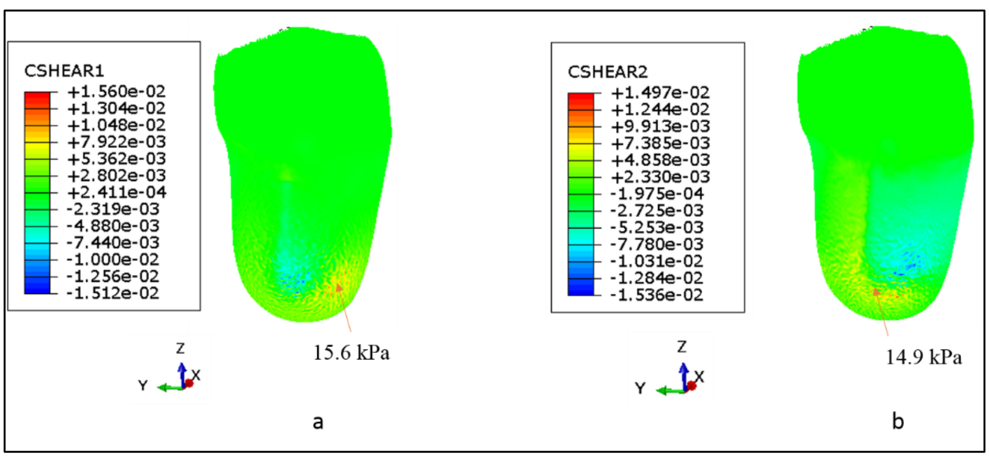


Figure 14: a) Circumferential shear stress observed on the residual limb (kPa), b) Longitudinal shear stress observed on the residual limb (kPa)

Table 6: FEA results generated on residual limb

Response over residual limb	Step-1	Step-2
	Interference effect during donning procedure	Application of body weight
Displacement (mm)	3.6	45
Contact pressure (kPa)	3.9	84
Von Mises stress (kPa)	1.2	14
Shear stress (kPa)	0	15.6

In summary, the stresses and displacement generated on the residual limb's surface during this nonlinear analysis are reported in Table 6. The maximum deflection found on the residual limb at the end of the analysis is 45 mm which was close to the values shown by Patino and Lacroix [6]. The large deflection value observed was due to the large deformation behavior of the soft tissue as well as the body weight application on the femur. However, the deflection pattern and the location are different from others work due to variation in loading and boundary conditions. The contact pressure developed on the residual limb during the donning process was 3.9 kPa and 84 kPa due to the half body weight application on the femur. These stress levels are significantly lower than the pain threshold limit 690 kPa [40]. The development of contact pressure on the residual limb over the course of the analysis is shown in Figure 15. The results observed during donning process as well as the application of body weight will be helpful to the prosthetic designers to enhance the prosthetic fit design by understanding the effect of pre-stress developed and the pressure distribution between the residual limb and the socket

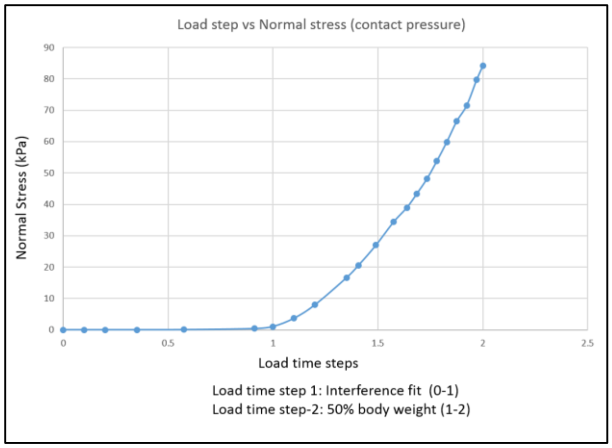
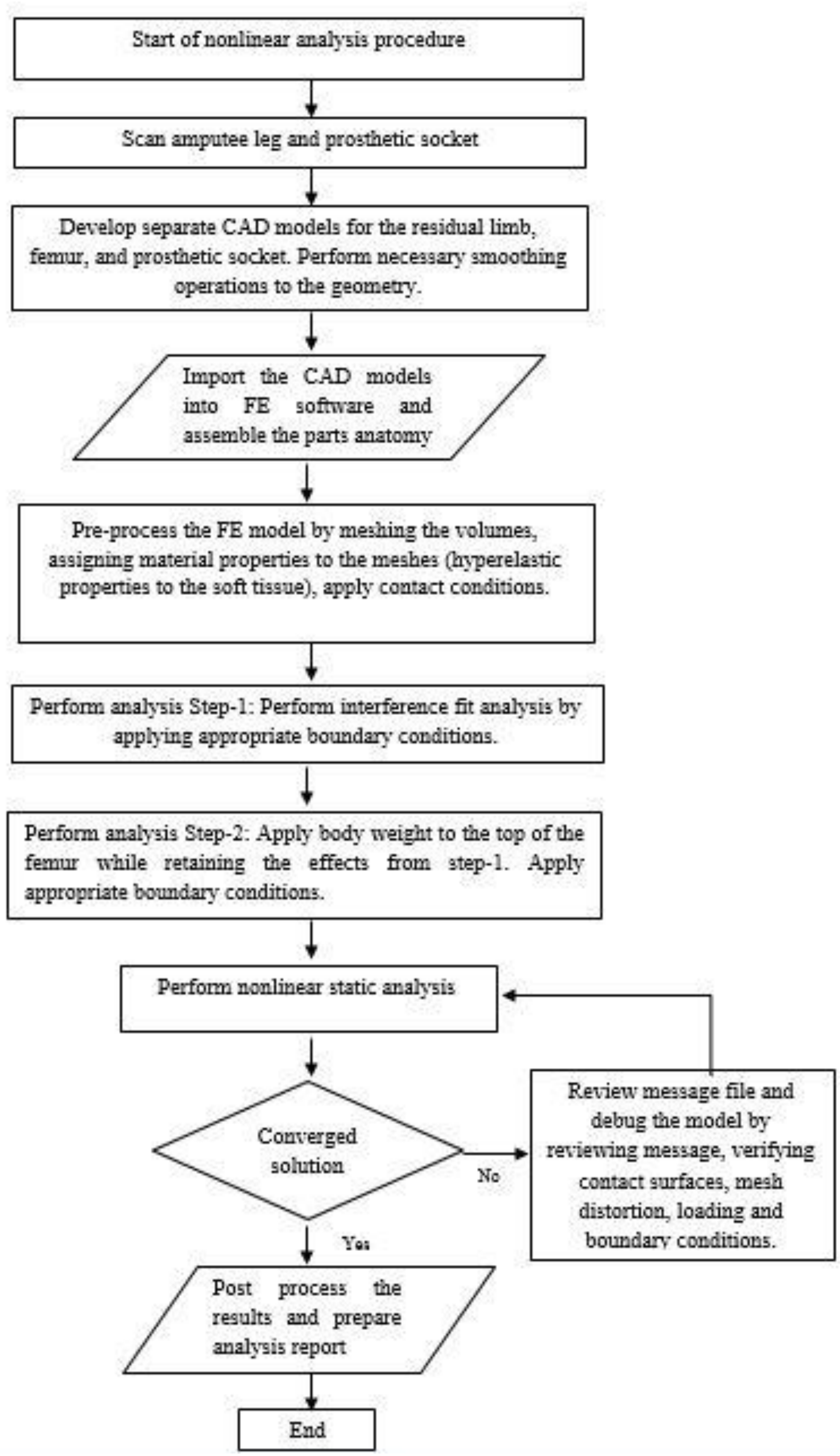


Figure 15: Load step vs Normal stress (kPa)

Chapter 6: Procedure Developed for Analyzing the Transfemoral Limb-Prosthesis Interface

The primary objective of this work was to develop a process allowing healthcare providers and engineers to simulate the fit and comfort of transfemoral prosthetics to reduce the number of re-fittings needed for the amputees. Through this work, a process has been developed which can be used by others in modeling and analyzing the transfemoral prosthetic fit. The process starts with scanning of the amputee's leg and socket followed by developing separate CAD models for the residual limb, femur, and prosthetic socket. The CAD models are then imported into FE software and assembled properly. Pre-processing operations are completed by meshing the volumes with appropriate element size, element type, assigning correct material properties, and by applying contact definitions where appropriate. The first analysis step is then completed by performing an interference fit analysis using appropriate boundary conditions. The results of the interference fit analysis are retained during the next analysis step in which half the body weight is applied to the top of the femur. The nonlinear static analysis is then performed. If the analysis solution converges, the results are analyzed. If the analysis does not converge, the FE model is updated by verifying and potentially modifying the contact

definitions, mesh, loading and boundary conditions. A summary of the steps included in this process are shown in the Flow chart-1 below.



Flow chart 1: Procedure Developed for Analyzing the Transfemoral Limb-Prosthesis Interface

Chapter 7: Conclusion

In this study, a nonlinear FE model of the transfemoral limb-socket interface was developed and analyzed. The study addressed the limitations of previous studies by performing a nonlinear analysis using hyperelastic material properties in modeling the nonlinear behavior of the residual limb's soft tissue, and by determining and including pre-stresses developed during the donning procedure in the overall load application. The analysis performed was highly nonlinear in terms of material, geometry, and contact definitions. Large deformations due to hyperelastic material properties were considered in the study of the soft tissue.

There is limited research data available to determine the effect of pre-stress due to interference which occurs when an amputee's transfemoral prosthetic socket is fitted to a residual limb. This study attempts to advance the state of prosthetic modeling by representing the donning process and also the nonlinear properties of human tissue using hyperelastic material models. This approach allows large deflections in the FE model, includes nonlinear contacts with friction, and predicts the pre-stress and pressure distribution resulting from interference between the residual limb and socket from the overclosure effect. This new modeling approach estimates the contact pressure (pre-stress) developed during the donning process due to the overclosure effect at 3.9 kPa, which is comparable to the results predicted by Patino and Lacroix [6] and Linlin [4]. However, the locations of the pre-stresses developed in the studies are different since these are highly dependent on variations in anatomical geometry and loading conditions. The methodology adopted in this study to simulate the donning process was quite different from the previous research efforts by Patino and Lacroix [6] and Linlin [4] where the donning process was

represented by boundary enforcement method [6] and vertical load on the top surface of the limb [4]. In this study, the maximum normal stress observed on the residual limb's outer surface due to application of half the body weight was 84 kPa, which is comparable to the results predicted by Zhang, where the maximum normal stress is 65 kPa [3] and Linlin, where the maximum normal stress determined was 80.57 kPa [4]. The maximum stresses occurred at bottom of the residual limb in this study which is similar to the locations observed for maximum normal stresses in the studies performed by Zhang [3] and Linlin [2]. The results of previous research studies, along with this current analysis, indicate that finite element modeling of prosthetics must be tailored to the specific individual for whom a prosthetic device is being developed.

Chapter 8: Future Work

In developing more advanced FE models of the transfemoral prosthetic-limb interface, it is recommended that additional research be performed on the hyperelastic material properties of soft tissues. Experimental studies on frictional coefficients can provide insight into how to better model the contact analytically. While this current study improved the understanding of the effect of pre-stress on the residual limb during the donning procedure, additional research is recommended in this area. Future work may also involve the evaluation of the fatigue conditions and enhanced modeling of the dynamic conditions experienced by the prosthetic devices. Additional research efforts in these areas will be helpful in understanding how prosthetic devices can be designed for better fit and improved comfort for patients.

Bibliography

[1] M. T. Mark and B. John, "Transfemoral Manual," Advanced Prosthetic center.

[2] S. Barbara and S. C. Dudley, "Generic geometric finite element analysis of the transtibial residual limb and the prosthetic socket," *Journal of Rehabilitation Research and Development,* vol 34 (2), pp. 171-186, 1997.

[3] M. Zhang and A. F. Mak, "A finite element analysis of the load transfer between an above-knee residual limb and its prosthetic socket-roles of interface friction and distal-end boundary conditions," *IEEE Transcations on Rehabilitation Engineering,* vol. 4, pp. 337-346, 1996.

[4] Linlin Zhang; Ming Zhu; Ling Shen; Feng Zheng, "Finite element analysis of the contact interface between trans-femoral stump and prosthetic socket," *Engineering in Medicine and Biology Society (EMBC), 2013 35th Annual International Conference of the IEEE* , pp.1270,1273, 3-7 July 2013.

[5] M. Tanaka, Y. Akazawa, I. Kitayama and A. Nakazawa, "Identification of pressure distribution at the socket interface of an above-knee prosthesis," *Advances in engineering software,* vol. 28, pp.379-384, 1996.

[6] F. R. J. Patino and L. Damien, "Finite Element Analysis of Donning Procedure of a Prosthetic Transfemoral Socket," *Annals of Biomedical Engineering,* vol. 39, pp. 2972-2983, 2011.

[7] M. Zhang, M. Lord, V. Roberts and S. A. R. Turner, "Development of a non-linear finite element modelling of the below-knee prosthetic socket interface," *Medical Engineering and Physics,* vol. 17(8), pp. 559-566, 1995.

[8] S. C. Dudley and B. S.-T. M, "Parametric analysis using the finite element method to investigate prosthetic interface stresses for persons with trans-tibial amputation," *Journal of Rehabilation research and development,* vol. 33(3), pp. 227-238, 1996.

[9] J. Xiaohong , Z. Ming and C. L. Winson, "Load transfer mechanics between transtibial prosthetic socket and residual limb-dynamic effects," *Journal of Biomechanics,* vol. 37, pp. 1371-1377, 2003.

[10] C. L. Winson, Z. Ming, A. B. David and C. Bill, "Finite-element analysis to determine effect of monolimb flexibility on structural strength and interaction

between residual limb and prosthetic socket," *Journal of Rehabilitation Research and Development*, vol. 41, pp. 775-786, 2004.

[11] A. V. Jaime and F. R. J. Patino, "Incidence of the boundary condition between bone and soft tissue in a finite element model of a transfemoral amputee," *Annals of Biomedical Engineering,* vol. 36(4), pp. 405-414, 2012.

[12] Z. Ming, F. M. Arthur and R. V.C, "Finite element modelling of a residual lower-limb in a prosthetic socket: a survey of the development in the first decade," *Medical Engineering and Physics,"* vol. 20(5), pp. 360-373, 1998.

[13] X. Jia, "Load transfer mechanics between transtibial prosthetic socket and residual limb-dynamic effects," *Journal of biomechanics*, vol. 37, pp. 1371-1377, 2003.

[14] C. L. Winson, F. F. Laurent, H. Kerstin, A. F. Laurent, H. Eva, L. G. David, G. Steven and B. Rickard, "Magnitude and variablity of loading on the osseointegrated implant of transfemoral amputees during walking," *Medical engineering and physics,* vol. 30, pp. 825-833, 2007.

[15] M. Zhang, A. B. David and F. T. Mak, "State-of-the-art research in lower-limb prosthetic biomechanics socket interface: A review," *Journal of Rehabilitation Research and Development,* vol. 38(2), pp. 161-174, 2001.

[16] G. Simpson, D. K. Wright and C. Fisher, "Modelling the interactions between a prosthetic socket , polyurethane liners and the residual limb in tanstibial amputees using a non linear finite element analysis," *Journal of Biomedical Science Instrumentation*, vol. 37, pp. 343-347, 2001.

[17] M. B. Silver-Thorn and T. Ergin, "Nonlinear Viscoelastic Material Property Estimation of Lower Extremity Residual Limb Tissues," *Journal of Biomechanical Engineering,* vol. 126, pp. 289-3000, 2004.

[18] S. Portnoy, A. Kristal, N. Shahbsin, A. Gefen and Z. Y. Siev-Ner, "Surgical and morphological factors that affect internal mechanical loads in soft tissues of the transtibial residuum.," *Journalof Biomechanical Engineering,* pp. 2583-2605, 2009.

[19] C. L. Winson, Z. Ming, J. Xiaohong and T. C. Jason, "Finite element modeling of the contact interface between trans-tibial residual limb and prosthetic socket," *Journal of Medical Engineering and Physics,* vol. 26, no. 8, pp. 655-662, 2004.

[20] http://www.simulia.com/download/rum11/UK/Feedback-Presentation-1.pdf, "Abaqus Manual," 2011.

[21] S. Alexandra, O. Binu and L. Irina, "Hexahedral mesh development of free-formed geometry: The human femur exemplified," *Journal of Computer-Aided Design,* vol. 41, Issue 8, pp. 566-572, 2009.

[22] "http://www.nlm.nih.gov/research/visible/ visible human.html," [Accessed online].

[23] "http://biomedical.materialise.com/mimics," [Accessed online].

[24] S. Chadwick and J. Lewis, "Above-Knee amputation," *Annals of Royal college os Surgeons of England,* vol. 73, no. 1991, pp. 152-154.

[25] "http://www.geomagic.com/en/," [Accessed online].

[26] J. A.Ramos, "Tetrahedral versus hexahedral finite elements in numerical modelling of the proximal femur," *Journal of Medical Engineering and Physics*, vol. 28, pp. 916-924, 2006.

[27] "www.simulia.com," [Accessed Online].

[28] S. Portnoy, G. Yarnitzky, Z. Yizhar, A. Kristal, U. Oppenheim, I. Siev-Ner and A. Gefen, "Real-time patient-specific finite element analysis of internal stresses in the soft tissues of a residual limb: a new tool for prosthetic fitting," *Annals of Biomedical Engineering,* vol. 35, pp. 120-135, 2007.

[29] L. Duchemin, V. Bousson, C. Raossanaly, C. Bergot, D. Laredo, W. Skalli and D. Mitton, "Prediction of mechanical properties of cortical bone by quantitative computed tomography," *Journal of Medical Engineering and Physics*, vol. 30, pp. 321-328, 2008.

[30] M. Zhang, X. Jia, J. Cheung and L. C. C. Winson, "FE modelling of load transfer between transitibial residual limb and prosthetic socket," *Journal of Biomechanics,* pp. 655-662, 2004.

[31] S. Joan E, G. Joan M , M. Stuart B and Z. Santosh G, "Material properties of commonly used interface material and thie static coefficients of friction with skin and socks," *Journal of Rehabilitation Research and Development Vol. 35 No-2,* vol. 35, pp. 161-176, June 1998.

[32] D. R. J. -ink, Presentation for the 2nd SAXSIM, "Analysis of Hyperelastic Materials with MECHANICA," PTC, 2010.

[33] D. l. Juan Fernando Ramirez Patino, "Finite element analysis on Donning procedure of a Prosthetic Transfemoral socket," *Journal of Biomedical Engineering*, vol. 39 (12), pp. 2972-2983, 2011.

[34] A. H. Muhr, "Modelling the stress-strain behaviour of rubber," *Rubber Chemistry and technology*, pp. 391-425, 2005.

[35] "http://en.wikipedia.org/wiki/Hyperelastic_material," [Accessed online].

[36] A F Bower, Solid Mechanics, 2008.

[37] R. W. Ogden, "Large Deformation Isotropic Elasticity- On the Correlation of Theory and Experiment for Incompressible Rubber like Solids," *Series A, Mathematical and Physical Sciences*, vol. 326, pp. 565-584, 1972.

[38] "http://en.wikipedia.org/wiki/Ogden_(hyperelastic_model)," [Accessed online].

[39] M. Mooney, "A theory of large elastic deformation," *Applied Physics*, pp. 582-592, 1940.

[40] M. Zhang, A. Mak and W. C. Lee, "Regional differences in pain threshold and tolerance of the residual limb: including the effects of age and interface materail," *Archives of Physical Medicine Rehabilitation.*, vol. 86, pp. 641-649, 2005.