International Journal of Cell Science and Biotechnology
E-ISSN 2320 –7574

Research Article

The Modeling and Effect of FEM on Prosthetic limb

Hasan Saad[†]*, Mohammad Qasim Abdullah[†] and Hatem R. Wasmi[†]

[†]Department of Mechanical Engineering, University of Baghdad, Iraq/Baghdad

Accepted 01 May 2017, Available online 05 May 2017, **Vol.7, No.3 (June 2017)**

Abstract

In this study, three dimensional Finite Element (FE) modeling of prosthetic lower limb was performed. A modified three dimensional finite element model of socket was developed in workbench of ANSYS 14.0 to find out the stress distribution and deformation pattern under functionally appropriate loading condition during normal gait cycle. This work tried to improve the deflection of the loading of Socket, foot, and pylon shank. The best designs results of the prosthetic above knee socket are (94.64%).In addition, a simplified stiffener has been added to optimize a moment of inertia of this main part thereby the deflection will be better, for this purpose a case study has been considered according to international standard for the Socket specification that investigated to illustrate the goal of this work . Moreover, the adopted design was selected carefully in order to match high strength and minimum weight criteria. Two types of FE analyses were performed; static analysis at standing position and dynamic analysis at jumping condition. The obtained results showed that the dynamic impact load is the most dominant since both materials were capable to stand with static load with minor Von Mises stresses acting on the pin connections. On the other hand, high Von Mises stresses were developed in the case of dynamic impact load which reached 90.19 MPa model. The results show that using stabilized method enables us to get stable and accurate numerical approximations consistent with the physical configuration of the problem over rough mesh by using ANSYS Workbench 14.0, AutoCAD and COMSOL 5.2. A high-density polyethylene (HDPE) was used for pylon.

Keywords: *Prosthetic; Safety factor; Finite element model; Socket; pylon.*

---

1. Introduction

Prosthetic limbs are incredibly valuable to amputees because prosthesis can help in restoring some of the lost capabilities with the amputated limb. Prosthetic lower limb consists of ankle joint and the foot. In order to make a highly efficient artificial limb, it should be provided that each and every component of the limb must function at high performance under several types of loading, particularly, impact loading. One of the important key factors in the design and fabrication of lower limb prosthesis is the type of material used in construction which plays dominant role in increasing the strength and lowering the overall weight of prosthesis.

Many studies showed that carbon fiber, titanium and steel are preferred for the fabrication of mono-limb, joint and the inner foot structure proved by (Jack E. Uellendahl, 1998). Finite element analysis for the evaluation of structural behavior of the Sure-flex prosthetic foot and other loadbearing components was conducted by (Omasta M., 2012). The Finite Element Method (FEM) has been used widely in biomechanics to obtain stress, strain and deformation in complicated systems and have been identified as an important tool in analyzing load transfer in prosthesis. The finite element analysis (FEA) models have been used to study the effects of the inertial loads and contact conditions on the interface between prosthetic socket and stump of an amputee during gait cycle (Radcliff, 1955) . Detailed approach for the finite element modeling which includes foot analysis, reverse engineering and material property testing was provided. The foot analysis incorporated ground reaction forces measurement, motion analysis and strain gauge analysis. For the material model determination, non-destructive laboratory testing accompanied with FE simulation was used. Based on the obtained results, it was concluded that the fatigue failure was the most common component failure in the lower limb prosthetics could be predicted (Falsig J., Hvid MD, and Jensen N., 1986).

The stress distributions and deformations were calculated through ANSYS Workbench, a finite element program. This page details the procedure and analysis that was used obtain results that were further discussed and compared to find an optimal pylon design. This paper will address finite element analysis of the socket prosthetic prescription. Specifically, the contribution of Mechanical properties of developed

---

*Corresponding author: **Hasan Saad**

model took into account the mechanical characteristics of socket, foot and pylon that include stress, total deformation and safety factor will be discussed. This study compared varying combinations of different parameters in order to better understand how different parameters affected the performance of the prosthetic pylon. The parameters that were evaluated were the length of the rib on the back side of the pylon and the outer diameter of the pylon. The methodology for this range of the study was to test a design with varying parameters to determine a combination of the varying parameters that had the lowest the overall von Mises stresses and had an optimal displacement throughout the pylon which could mimic a natural gait cycle. This study also looked at different elliptical by using several different height to width ratios.

2. Design

The developed design was modeled initially by three separate parts to create an FE model, first the geometry of the modelled objects needs to be obtained, including the shapes of the free residual limb, pylon, the socket, foot and the adapter if involved. Although some models were based on ideal and simplified geometry, an accurate description should be based on an actual geometry as shown in Fig.1. The main object of this study is the prosthetic socket. The simplified geometry of his residual limb was modeled in Pro-engineer and then it is being imported (from AutoCAD) and modified in ANSYS Workbench 14.0 (HANIE NADIA SHASMIN, 2012). To conform to ISO Test Standard 10328 the pylon length was fixed for all tests. The standard length for this evaluation is 420 mm. The outer diameter is 50 mm.

2.1 Interaction properties of the model

In this analysis, the only interaction used was ‘General Contact (Explicit)’ interaction. This interaction takes place at the time of impact, between the foot and the floor and the between the ankle and the foot.

This singular value was compared to the yield strength of the material to ensure that the design does not exceed the uniaxial stress limits of HDPE material. The FEA contour plot for the von-Mises stress distribution will be examined for each of the models. These contour plots were used determine the approximate location and value of the maximum von Mises stresses.

3. Defining the Analysis Type and Applying Load

The displacements were examined in component form to determine how the model has moved from its original position in each of the three principal directions x, y, and z. These were graphed and analyzed in the program and plotted in Excel to gain a better understanding of how the loadings affected the models. The load which is used in the ANSYS workbench version 14 software will be fixed support at the adapter of socket. While, the interface pressure is distributed according to particular positions Shown values with positions of pressure distributions of present experimental case study. In fatigue solution, the fatigue tool is used to find the equivalent stress, maximum shear stress, total deformation, safety factor, and life at particular loads.

To account for the two main stages of the gait cycle, the ‘heel strike’ and ‘toe off’ phases were both applied on the Pylon Loading Plane, which lies on base of the pylon. At the top of the pylon, a fixed constraint in x, y, and z directions has been applied to ensure that there is no movement at this end of the pylon while the loads are being applied. We double the loads for the worst case scenario and added an equivalent load in the Z direction where Loading condition:

Table 1 The loads that applied

Dimensions	Toe off-loading forces	Heel strike loading forces	Worst case
X	-607 N	-112 N	-1214 N
Y	3240 N	2997 N	6480 N
Z	0 N	0 N	1214 N

Sensitivity Analysis

The results of the aforementioned analyses may be plotted as a direct comparison of the local interface stress versus parameter value. An alternative method of presenting these results is via sensitivity analysis.
Sensitivity is defined as the ratio of the relative change in behavior to the relative change in parameter.

If we define the parameters of interest, P, as the respective socket design or the residual limb parameter, and the behavior variables, B, as the interface stress, then the sensitivity can be defined as:

$$\text{Sensitivity}=\frac{(B2-B1)/B0}{(P2-P1)/P0} \qquad (1)$$

Where the subscript o refers to the reference model, and the subscripts, and 2 refer to the model variations.

Finite Element Analysis of ANSYS

The model will be assumed as linearly elastic, isotropic, and homogeneous. The values in the table below will be applied as the material values being applied to the model are listed in Table 2 (Victoria S. Richardson, 2008).

Table 2 The properties of materials

Material Property	Values
Poisson’s Ratio	0.3
Young’s Modulus	$1.5 * 10^9\ N/m^2$
Density	830 kg/m^3
Tensile Yield Strength	$3.07\ 10^9\ N/m^2$
Compressive Yield Strength	$4.85\ 10^9\ N/m^2$
Tensile Ultimate Strength	$3.68\ 10^9\ N/m^2$

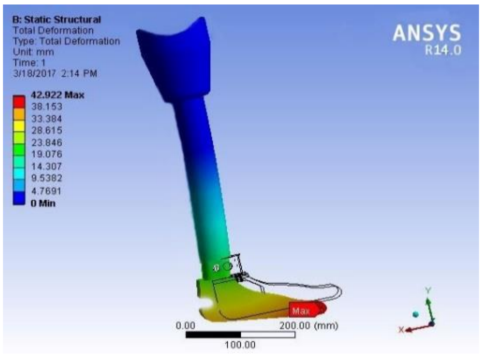

Fig.1 Deformation of prosthetic limb

In the above contour plots, Figure 1, the displacements of the 50 mm OD shank have been presented. In these plots, the blue portion represents the minimum displacement values and this area takes up nearly half of the shank.

For each of these models, the von Mises stresses are up to 70% higher than the Yield Strength. These high stresses are most likely occurring because the load location is 60 mm offset in the f direction from the origin of the model. With such a large offset, the bending stresses increase substantially, causing higher overall stresses throughout the shank. In figure 2 you can see that the maximum stress at the zone of adapter.

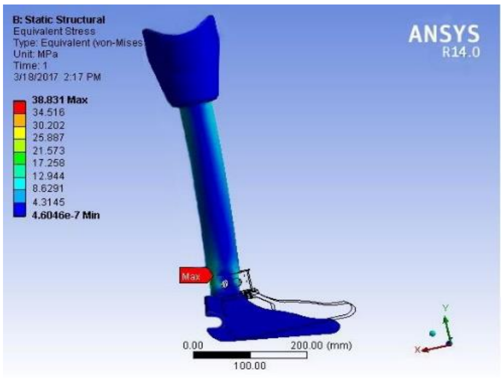

Fig.2 Load of von-Mises stress plot

Changing the load to another value by increasing, is shown in figure 3.

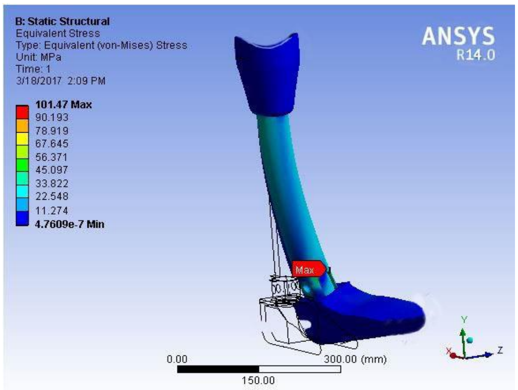

Fig.3 Represent the Worst case von-Mises stress plot.

The deformation caused by this loading can also be seen through this feature and it can be observed that there are significant levels of deformation present in this model. The deformation of prosthetic focused on the foot where it reached to more than 150 mm as shown in Figure 4.

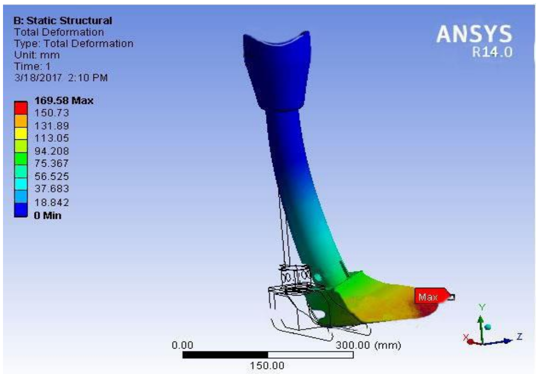

Fig.4 Represent the Worst case deformation plot.

The von Mises stresses were shown to decrease as the diameter increased. Also, the von Mises stresses decreased as the rib length increased, which demonstrated that the rib length is a positive addition to the overall design of the circular shank. The elliptical models that were tested measured the lowest von Mises stresses when compared to the stress values of circular shanks of the same diameter as the minor diameter of the elliptical shanks. The models were all compared to the Yield Strength of the material used to fabricate the limb in order to determine whether or not the models exceeded the stress limit under the ISO loading conditions.

Results

The results illustrate the optimal values of certain geometric parameters of a prosthetic leg, which allowed us to make a more informed decision as to what shape we wanted to use for our design were $2.791 *10^7$ N/ m^2 for 50 mm, $1.322 *10^7$ N/ m^2 for 55 mm and $1.1836*10^7$ N/ m^2 for 58 mm which is represent maximum von-Mises stress for elliptical cross sections with heel strike loading.

In addition, the maximum von-Mises stress results of elliptical cross sections with toe off-loading were $5.947 *10^7$ N/ m^2 for 50 mm, $2.283*10^7$ N/ m^2 for 55mm and $1.18*10^7$ N/ m^2 for 58mm.

The predicted resultant shear stresses were less than the experimental values at all measured sites. Best matches were achieved at postero-distal and anteroproximal sites. Consistent mismatches were seen (Zhang M., 1995), (Zhang M, *et al* 1998). measured the interfacial pressure up to 283 kPa, and shear stress up to 44 kPa, at the critical regions of BK sockets using triaxial transducers. The FE analytical pressures up to 226 kPa were, on average, 30% lower than those measured. The difference might result from the fact that the load applied to the prosthetic limb in the FE model was static and equal to the body weight, whereas the actual load during walking was dynamic and larger than the body weight. The comparison suggests that the dynamic analysis is necessary in future FE modelling studies. Owing to the application of the interface elements to simulate the slip/friction conditions at the stump/socket interface, the shear stresses have better agreement, difference in the magnitudes being 10%, on average, and the directions of the resultant shear stresses are identical to the experimental results.

Predicted normal stresses on a BK stump between 0 and 120 kPa, and the experimental measurements were between 0 and 128 kPa. Silver- Thorn predicted that the pressures on a BK socket varied from 0 to 275 kPa, whereas those measured were 0 to 205 kPa (Zhang M., 1995).

In general, comparison of the stresses predicted by FE analysis and those measured suggested that the results can be in the same range. However, one-to-one correspondence has not been achieved. The validation is limited to a number of the points where stresses can be measured.

Conclusions

This study establishes the development of a suitable modeling and analysis of prosthetic leg using ANSYS Workbench. In addition, the results summarized that assimilating local submissive properties within socket wall can be an effective methods to distribute maximum stress areas and also to relief contact pressure between the socket and stump. According to the study, von-Mises stresses for the 50 mm width elliptical pylon is more than double the Yield Strength, and also nearly three times the stress value when compared to the 55 mm width pylon. Also, stresses decreases as the minor diameter of the ellipse increases.

In both loading conditions, the 50 mm pylon has the highest displacement values and the 58 mm pylon has the lowest displacement values. The displacement increases as the diameter decreases. This trend shows that the rib length is important to the overall design. An increase in the diameter would cause a decrease in displacement. However, it would allow displacement throughout the pylon, which allows for flexibility for the user.

The most important aspect to displacement is that it should be present in the models because it contributes to the overall flexibility of the pylon, which mimics natural gait cycle. As the diameter and rib length increase, the displacements decrease; however, this makes the design more rigid and not natural to the gait movement. However, decrease in displacement is important and some displacement is still needed throughout the model to contribute to the flexibility of the pylon, which allows for natural lower body movement. We then ran our own FEA study on the whole assembly in ANSYS. The FEA implemented the same loading conditions in our previous study for both heel strike and toe off. Figures below illustrate both deflections and stresses experienced in different loading conditions such as toe off, heel strike, and worst case. For our worst case scenario, we doubled the loads in the toe off-loading condition and also introduced a 1000 N force in the Z direction. These values are used to simulate a fall where the loads introduce a twisting motion relative to the individual. The point which was probed on each of the models lies 180 degrees from the load application point. This plane lies 46.86° and 9.84° from the f-axis in the negative o-direction, which according to manual calculations, should be the plane upon which the highest von Mises stresses are located for loading conditions, respectively.

A big factor in used to compare our design to the natural gait cycle as well as comfort is the degree of dorsiflexion. For this measurement, we examined deflection characteristics during the heel strike phase. Looking at the Loading case 1 of figure 1 deformation plot, the contour plot shows a maximum deflection of 61.5 mm or 2.42 in. Using basic trigonometry, this corresponds to a dorsiflexion angle of 29.8 degrees. Although we ideally want the dorsiflexion angle to be 40 degrees, the dorsiflexion angle for this design is acceptable since it is a deflection based design. In order for our design to accomplish a greater range of dorsiflexion, we would need to translate this design into a mechanism based design with either linkages or gears to control the range of motion due to the applied forces. We wanted to avoid this design due to the significant costs associated with complex designs that contain mechanisms such as linkages and/or gears.

Examining the results below the von-Mises stress is well below the yield stress for all components in the assembly relative to their material. Moreover, the overall displacement in the prosthesis for both toe off and heel strike mimics the displacements seen in a natural gait cycle. Thus, our design proves to be a viable, comfortable, and safe solution for lower-limb amputees. Also, the results that obtained gave satisfy safety factor according to the composite materials that used.

References

J.E. Uellendahl, (1998), Prosthetic Primer: Materials Used in Prosthetics Part I, Amputee Coalition Organization, Volume 8, Issue 6.

O.M., Palousek, D., Navrat T. and Rosicky J., (2012), Finite element analysis for the evaluation of the structural behaviour, of a prosthesis for trans-tibial amputees, *Medical Engineering & Physics* 34, 38– 45.

Radcliff CW Functional considerations in the fitting of above-knee prostheses. Artificial Limb 2(1955) 35-60.

F.J., Hvid MD, and Jensen N., (1986), Finite element stress analysis of some ankle joint prostheses, *Clinical Biomechanics* 1,171–76.

H.N. Shasmin, (2012), Development of a new pylon material in transtibial prosthesis Dissertation, Faculty of Engineering University of Malaya Kuala Lumpur.

V.S. Richardson, (2008), Analysis of a Lower Limb Prosthesis Bachelor of Science, Faculty of Worcester Polytechnic Institute,

Z.M., (1995), Biomechanics of the residual limb and prosthetic socket interface in below-knee amputees, Ph.D. Dissertation, University of London.

Z.M, (1998), Turner-Smith AR, Roberts VC, Tanner A. Clinical investigation of the pressure and shear stress on the belowknee stump with a prosthesis, *Med Eng Phys* (in press).

Faustini MC, (2006), Neptune RR, Crawford RH. The quasistatic response of compliant prosthetic sockets for transtibial amputees using finite element methods. Medical Engineering & Physics 28, 114-121.s