FINITE ELEMENT STRUCTURAL ANALYSIS OF PROSTHESIS SOCKETS FOR BELOW-THE-KNEE AMPUTEES MANUFACTURED USING SLS

Mario Faustini*‡, Mahendra Lokhande*, Richard Crawford*, William Rogers†, Andrew Gitter† and Gordon Bosker†

*The University of Texas at Austin
†The University of Texas Health Science Center in San Antonio
‡Supported by the National Council of Scientific and Technological Development (CNPq-Brazil)

Abstract

A very attractive application of Solid Freeform Fabrication is manufacture of prosthesis sockets for below-the-knee amputees. The custom geometric design required is very compatible with SFF techniques. The present work focuses on finite element analysis of sockets manufactured by Selective Laser Sintering using Duraform as the material. The objective is to ensure reliability of the sockets for their use by patients. This paper describes the finite element models developed for the sockets, as well as the derivation of realistic boundary conditions that may allow a simulation of the structure under regular workloads.

1) Introduction

The use of Selective Laser Sintering (SLS) in the manufacture of prosthesis sockets for below-the-knee amputees has been recognized as an appropriate potential application of Solid Freeform Fabrication [Rogers et al., 1991]. However, to ensure the reliability of these sockets and their safety for users requires a complete structural analysis of their mechanical behavior during operation. The focus of the current work is a finite element (FE) evaluation of the performance under standard conditions of one sort of socket, the single-wall socket.

This paper is structured as follows: first, the single wall socket design is presented, followed by the development and validation of its FE model. Next, the standard workloads during gait are defined as boundary conditions for the FE analysis. Finally, the results of the simulations of two sockets that were actually manufactured are exhibited.

2) The single wall socket

The single wall socket, as seen in Figure 1, consists of an upper part, where the residual limb fits, and a lower region, which is a hollow cylinder where a pylon is attached with a prosthetic foot at its other end. The first step in the design and manufacture of the socket is fabrication of a mold from the patient's residual limb, from which a copy of the limb is obtained. This copy is digitized with a laser scanning system. From this data, a perfectly fitting model of the socket is designed using computer-aided geometric operations on the data mesh. The thickness of the socket wall is decreased around specific points to provide compliance in those areas. The compliant regions relieve pressure on sensitive spots of the patient's limb, such as the fibular head and distal tibia regions, thereby increasing comfort. Thus, tessellated data file (STL) is generated from this model and transmitted to a Selective Laser Sintering workstation for fabrication. The material used is Duraform, the physical properties of which are given in Table 1.

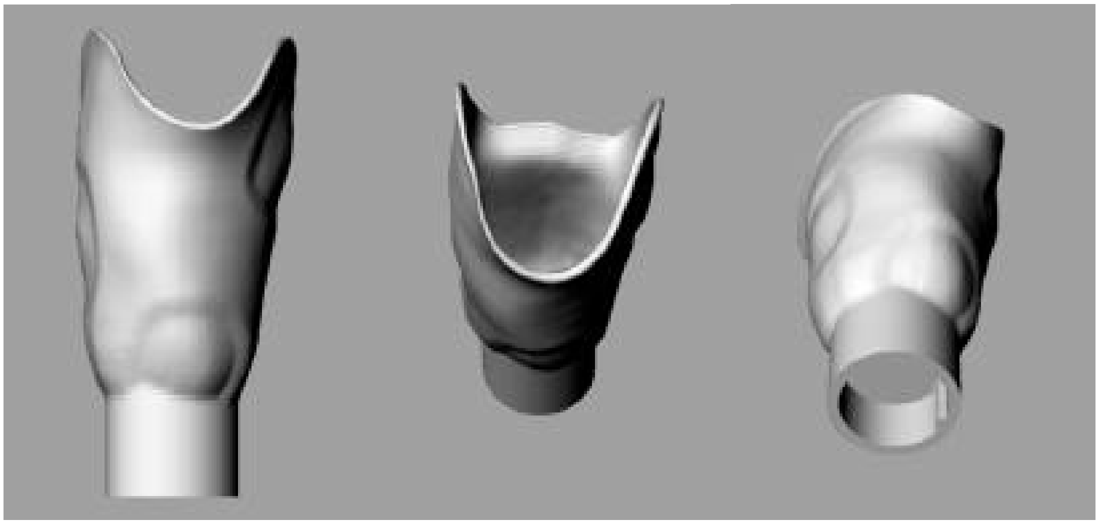

Figure 1. The single wall socket

Table 1. Duraform properties provided by Product Innovation & Development Centre, Liverpool, UK.

	SI units	DuraForm PA
General Properties		
Specific Density 20°	g/cm_	0.97
Water Absorption	%	0.41
Density	g/cm_	0.59
Average Particle Size	µm	58
Particle Size Range 90%	µm	25 92
Mechanical properties		
Tensile Strength Break	MPa	44
Tensile Strength Yield	MPa	37
Tensile Modulus	MPa	1600
Tensile Elongation at break	%	9
Flexural Modulus	MPa	1285
Impact Strength		
Notched Izod	J/m	216
Unnotched Izod	J/m	432
Surface Finish (up facing)		
As SLS processed, Ra	µm	8.5
After polishing, Ra	µm	0.13

3) The Finite Element Model

The STL data file of a socket consists of a collection of 48,000 triangular faces that define the socket surface in space. This level of detail would potentially result in unnecessary and time-consuming data preparation and analysis. Thus, the first step to generate the FE model is to apply a reliable mesh decimation algorithm to the STL file. We use Amira[1] to perform the decimation operation. As seen in Figure 2, the decimated version has 2000 faces, but the overall geometric details and features are well preserved (especially the thicknesses of the wall, which is fundamental for the structural analysis).

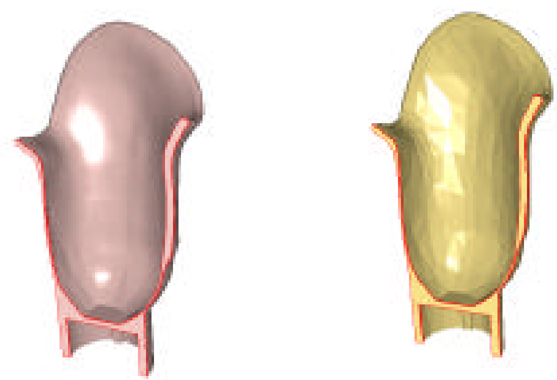

Figure 2. A section of the original STL model (48000+ faces) and the decimated version (2000 faces).

Once decimated, the STL file is loaded into Rhinoceros[2], where the triangular faces of the model are converted to parametric surfaces (NURBS) and exported as an IGES file. The IGES format is readable by SDRC I-DEAS[3] for FE analysis. In I-DEAS, the volume is meshed with solid elements (parabolic tetrahedra), assigned material properties and appropriate boundary conditions, and analyzed.

4) FE Model Validation

In order to validate the results of the FE structural analysis, an actual socket was tested to determine the vertical load that causes mechanical failure. The test is illustrated schematically in Figure 3.

[1] Amira is 3D Visualization package developed by Indeed-Visual Concepts GmbH, Berlin, Germany.
[2] Rhinoceros is a 3D modeling package offered by McNeel & Associates, Seattle, WA.
[3] I-DEAS is a CAD/CAE/CAM package provided by SDRC, Milford, OH.

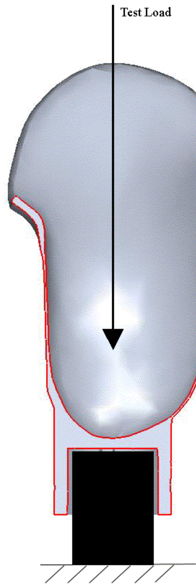

Figure 3. Experimental loading test to determine the maximum vertical load a socket supports.

The results were computed, as the load was gradually increased, in a force–strain graphic, shown in Figure 4. The figure indicates that the socket fails at a test load of 10 kN.

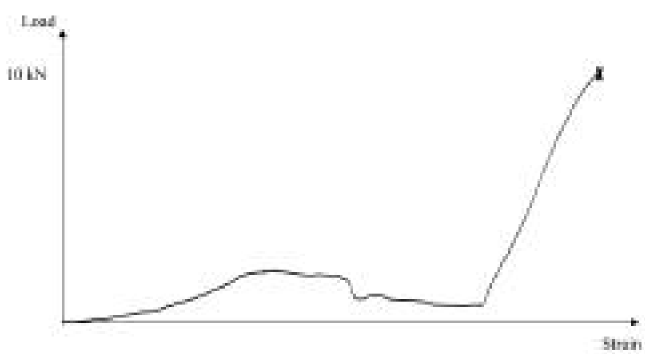

Figure 4. Results of the experimental loading test.

Similarly, a FE model equivalent to the physical model in Figure 3 was analyzed. Using the properties shown in Table 1, it was determined through repeated simulations that the socket reaches the ultimate tensile strength for Duraform (44 MPa) under a load of 6 kN. The simulation results for a 6 kN vertical load are shown in Figure 5.

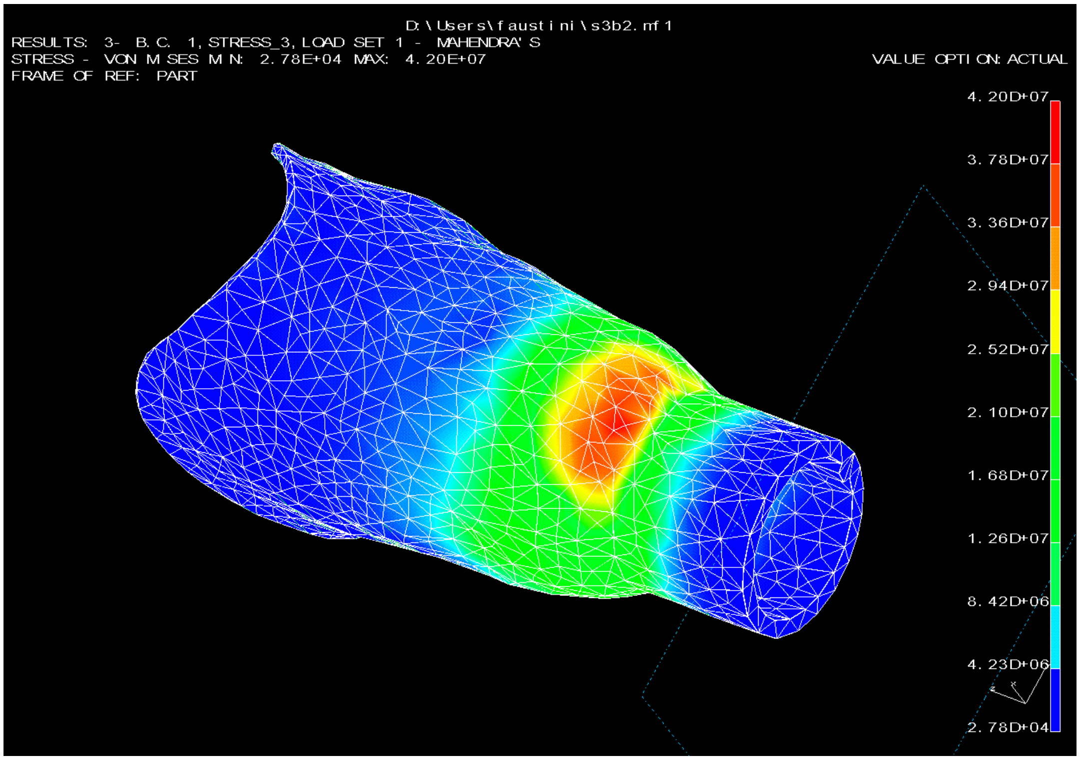

Figure 5. Results for the simulation of the loading test using FE analysis.

The difference between the ultimate load for the empirical test and the FE analysis may be explained by the fact that the socket used on the first one was infiltrated with cyanoacrylate, which increases the tensile strength of the part. Qualitatively, however, the location where the actual socket cracked matched exactly the region of largest stress predicted by the FE analysis.

5) Finite Element Analysis of Gait

In order to simulate the structural behavior of the FE model of the socket during gait, it is necessary to evaluate the loads applied by the residual limb. Figure 6, from Noguchi et al. [2000], shows the components of the ground reaction force of the foot in one stride (with z being the direction of the weight and x the direction of progress) in terms of percentage of body weight. Note that during the double supporting period (between heel contact of one foot and toe contact of the other ¾ when both feet are in contact with the ground), the forces have their greatest values, especially in positions A and B. Therefore, those two positions were analyzed in a worst case scenario, with all forces applied exclusively to the prosthetic foot.

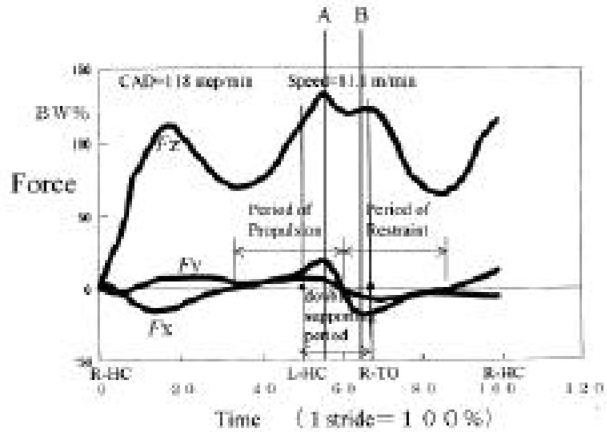

Figure 6. Components of ground reaction force of feet during gait [Noguchi et al., 2000].

The boundary conditions for the FE model were calculated using the values for the forces in both positions A and B, as shown in Figure 7. In the figure, *Fr* is the horizontal resultant force due to F_x and F_y and M_r is the moment needed to balance the forces transferred from the ground to the middle of the socket.

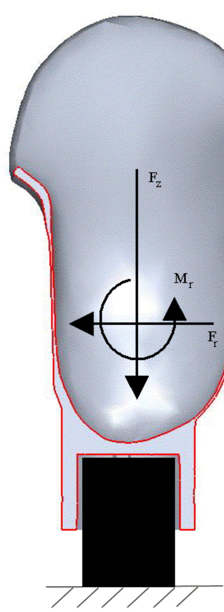

Figure 7. Boundary conditions used in the gait FE simulations.

6) Simulations

Two sockets were simulated for both positions A and B (Figure 6) using the FE analysis procedure described in Section 3. The patient was assumed to have a body mass of 75 kg.

The first socket analyzed had a notch in the junction between the base cylinder and the lower thinner part in the front of the socket, as seen in Figure 8. This socket was actually fabricated and broke while the patient was wearing it. The simulation results for this socket in position A are presented in Figure 9 (stresses are given in Pa). The simulation results for the same socket in position B are presented in Figures 10 (stresses in Pa) and 11 (displacement in meters).

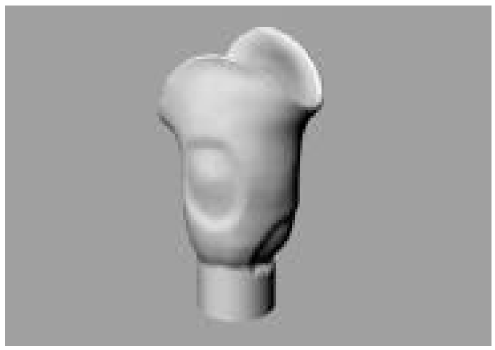

Figure 8. First socket simulated.

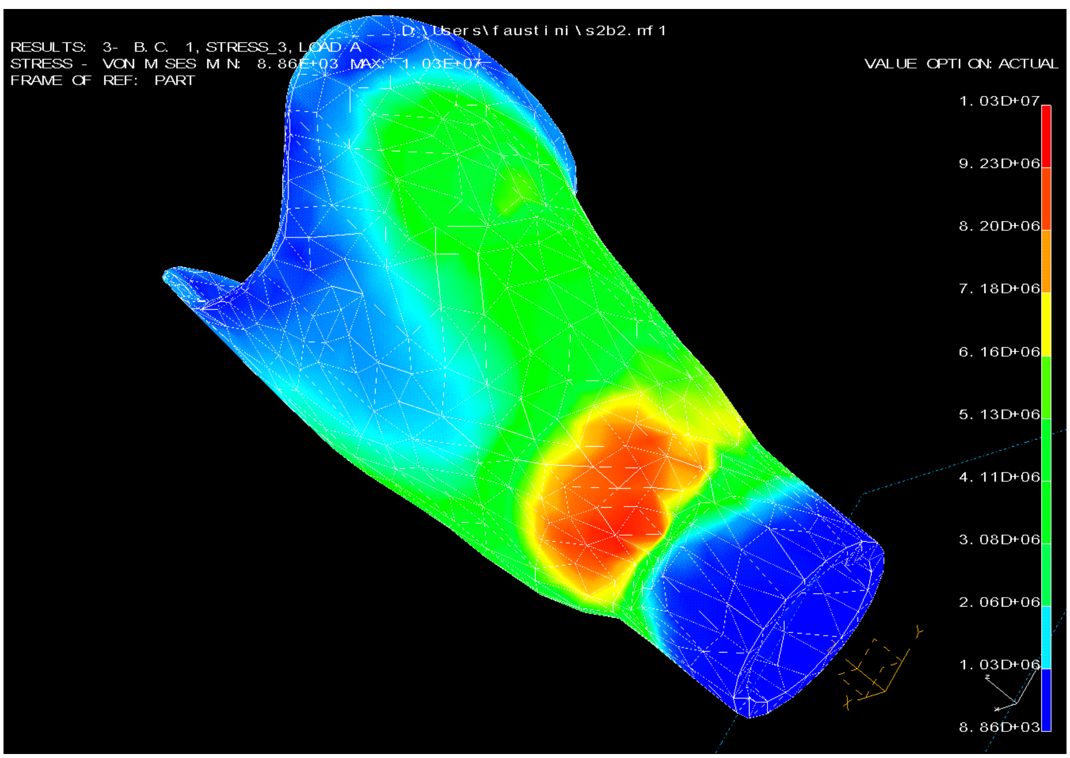

Figure 9. Predicted stresses for first socket in position A.

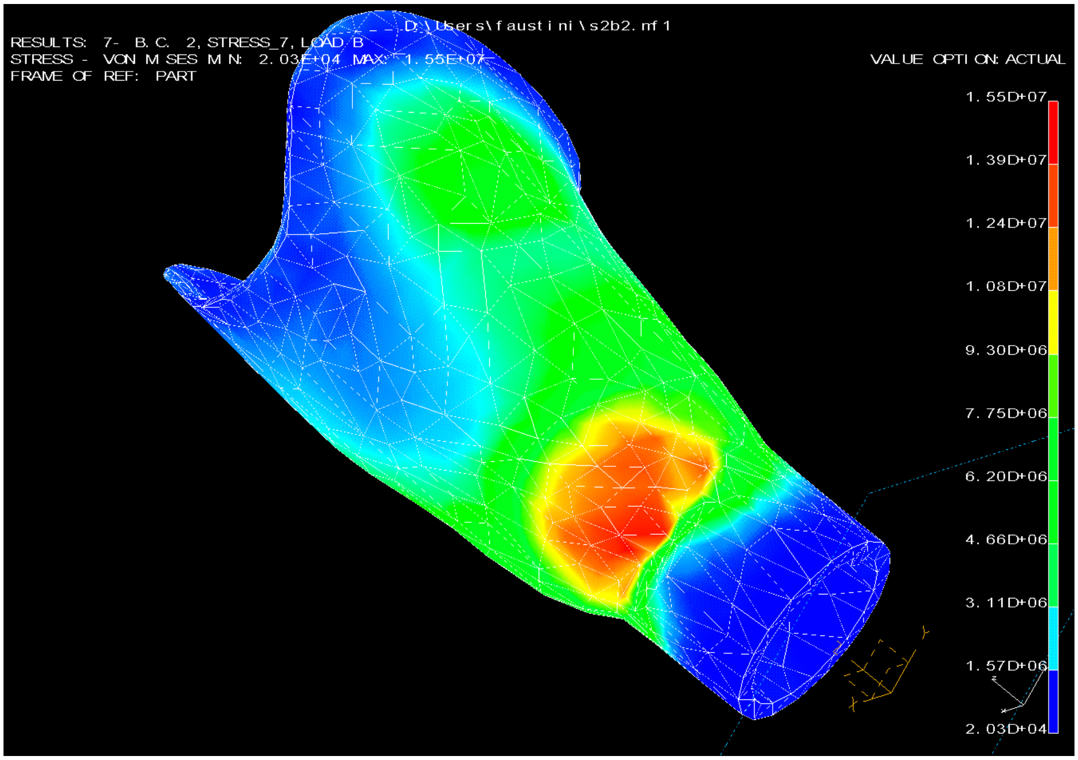

Figure 10. Predicted stresses for first socket in position A.

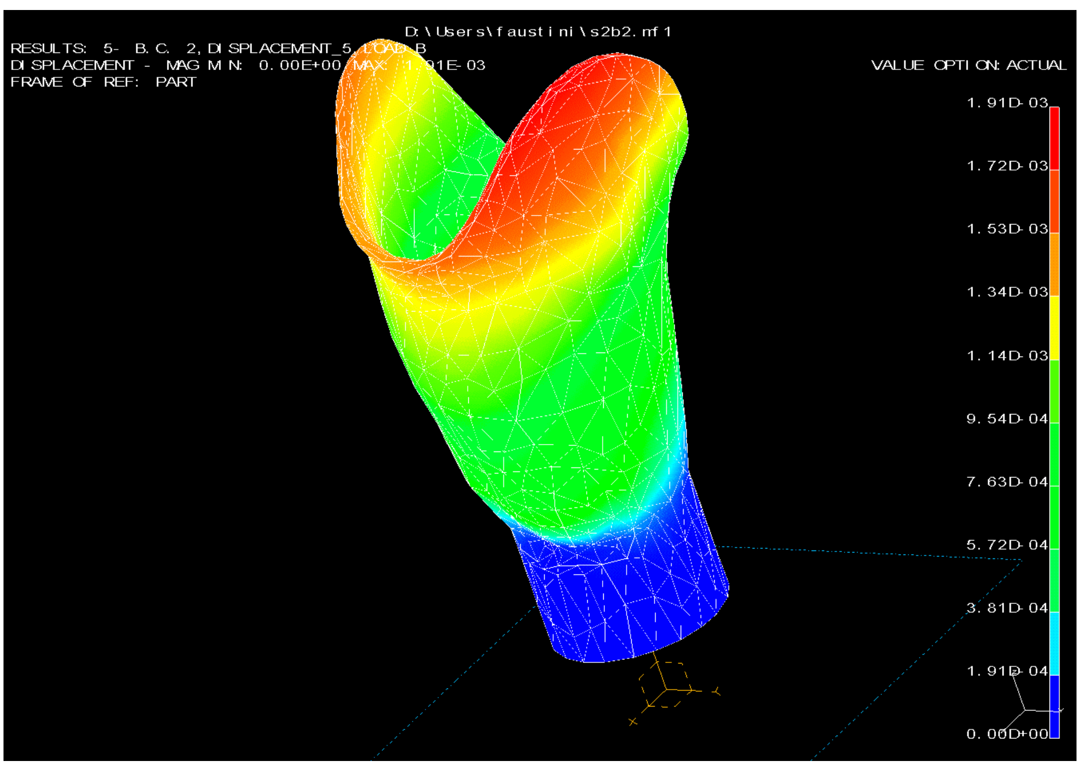

Figure 11. Predicted displacements for first socket in position B.

The second socket analyzed was designed with software with improved smoothing features, and is shown in Figure 12. This socket was also fabricated and has been used by the patient without failure. The simulation results for this second socket in position A are shown in Figure 13 (stresses in Pa). The simulation results for the same socket in position B are presented in Figures 14 (stresses in Pa) and 15 (displacement in meters).

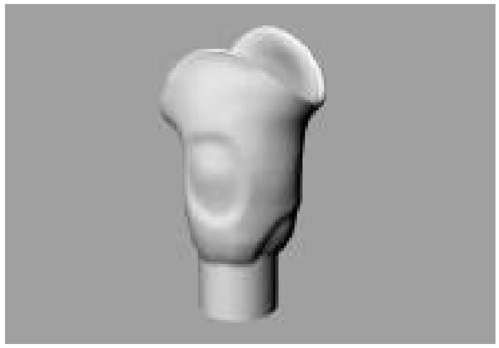

Figure 12. Second socket simulated.

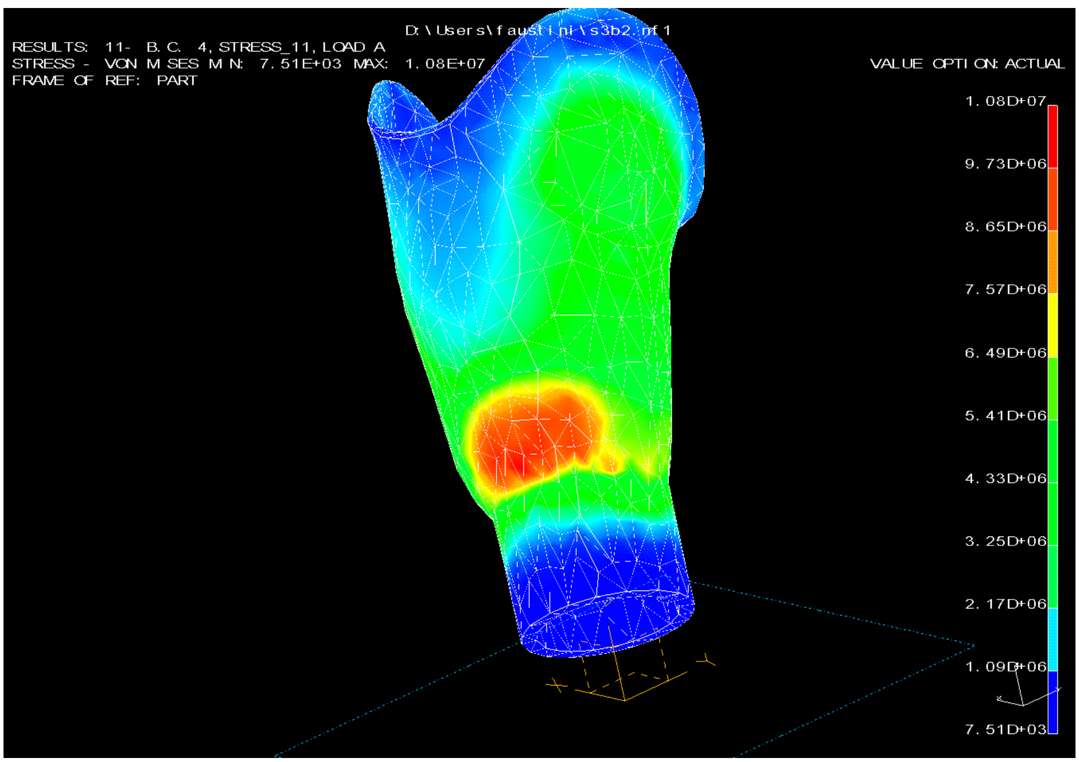

Figure 13. Predicted stresses for second socket in position A.

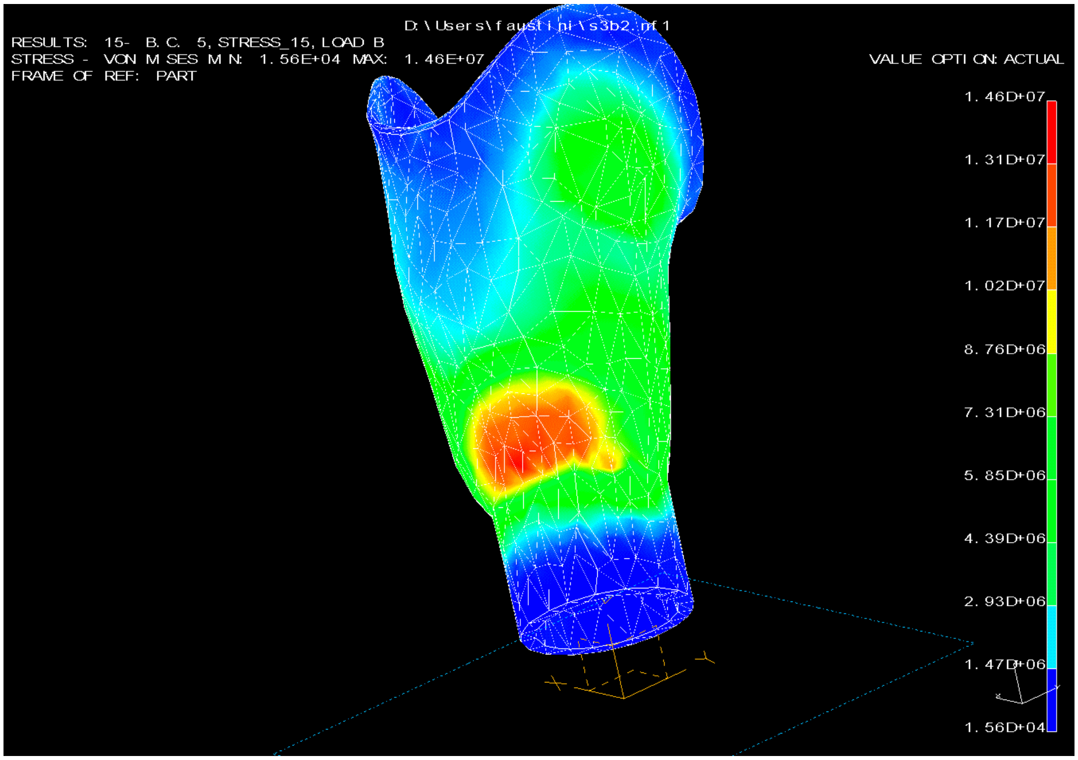

Figure 14. Predicted stresses for second socket in position B – bottom-left-front view.

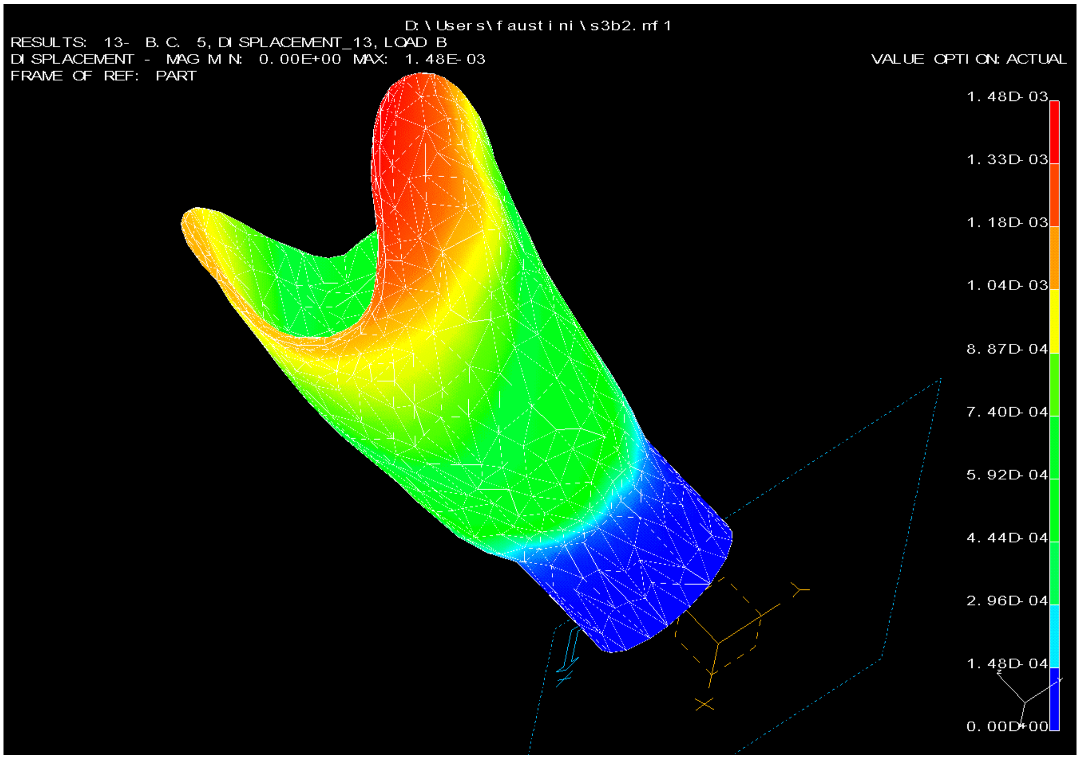

Figure 15. Predicted displacements for second socket in position B.

It should be pointed out that for both sockets the greatest stresses were computed for position B, even though the load values for position A were slightly larger (see Figure 6). Also, the analysis predicted, for position B, greater maximum stress for the first socket (15.5 MPa) than the second socket (14.6 MPa). However, this still does not explain the failure, since this maximum stress is considerably less than the yield strength for Duraform (37 MPa – Table 1). Additionally, the maximum displacement for each socket was less than 2 mm, which preserves the biomechanics of the gait [Stephens, 1999].

7) Conclusion

This paper presents a procedure for finite element based structural analysis for a below-the-knee prosthetic socket. Verification of the FE model and the simulation of two actual sockets are discussed. While the results agree qualitatively, the actual numerical results differ significantly. To address this disagreement, a more precise FE simulation algorithm will be tested (p-adaptive method). Also, more precise material properties for Duraform must be obtained. Both Thompson [1995, 1997] and Watson [1999] illustrated the orthotropic behavior of Duraform, which depends highly on the direction of the parallel layers of the material in the structure (defined during manufacture) with respect to the direction of the stresses applied. Finally, both impact and fatigue studies should be addressed in order to provide a more accurate structural analysis of the socket, as these modes of failure are likely to occur in real sockets.

8) References

1. Rogers, W. E., Crawford, R. H., Beaman, J. J., and Walsh, N. E., 1991, "Fabrication of Prosthetic Sockets by Selective Laser Sintering," *1991 Solid Freeform Fabrication Symposium Proceedings*, Marcus, H. L., Beaman, J. J., Barlow, J. W., Bourell, D. L., and Crawford, R. H., eds., Austin, TX, August 12-14, 1991, pp. 158-163.
2. Noguchi, T., Ochiai, H., Ukai, T., 2000, "Falls from slips in double supporting period gait," Proceedings, *12th Conference of the European Society of Biomechanics*, Dublin, pp. 427.
3. Stephens, S. D., 1999, *Design of a Compliant Socket Fabricated Using Selective Laser Sintering*, Master's thesis, The University of Texas at Austin.
4. Thompson, D. C., 1995, "The Optimization of Part Orientation for Solid Freeform Manufacture," Master's thesis, The University of Texas at Austin.
5. Thompson, D. C., and Crawford, R. H., 1997, "Computational Quality Measures for Evaluation of Part Orientation in Freeform Fabrication," *Journal of Manufacturing Systems*, vol. 16, no. 4, pp. 273-289.
6. Watson, D., 1999, "Process Optimization for Selective Laser Sintering Through the Use of Design Rules and Constraints," Master's thesis, The University of Texas at Austin.