Three-dimensional Modelling and Mesh Convergence Analysis of Transtibial Prosthetic Socket using Finite Element Method

Muhammad Hanif Baharuddin
Bone Biomechanics Laboratory (BBL), Department of Biomedical Engineering and Health Sciences, Faculty of Electrical Engineering, Universiti Teknologi Malaysia

Mohammed Rafiq Abdul Kadir
Department of Biomedical Engineering, Faculty of Engineering, Universiti Malaya

Muhammad Hanif Ramlee
Department of Biomedical Engineering, Faculty of Engineering, Universiti Malaya

https://doi.org/10.5109/7395619

出版情報：Proceedings of International Exchange and Innovation Conference on Engineering & Sciences (IEICES). 11, pp.900-905, 2025-10-30. International Exchange and Innovation Conference on Engineering & Sciences
バージョン：

Three-dimensional Modelling and Mesh Convergence Analysis of Transtibial Prosthetic Socket using Finite Element Method

Muhammad Hanif Baharuddin[1], Mohammed Rafiq Abdul Kadir[2], Muhammad Hanif Ramlee[3*]
[1]Bone Biomechanics Laboratory (BBL), Department of Biomedical Engineering and Health Sciences, Faculty of Electrical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia, [2] Department of Biomedical Engineering, Faculty of Engineering, Universiti Malaya, 50403 Kuala Lumpur, Malaysia, [3] Department of Biomedical Engineering, Faculty of Engineering, Universiti Malaya, 50403 Kuala Lumpur, Malaysia.
*corresponding: muhammad.hanif.ramlee@biomedical.utm.my

Abstract: *Previous several researchers utilized finite element analysis (FEA) to evaluate the socket. However, there is restricted information regarding convergence study to identify the optimum mesh size. Thus, the study aimed to provide the method conducting the convergence study of prosthetic socket. The modelling of socket required the 3D model of residual leg. Hence, the residual leg of volunteer was 3D scanned to construct socket in 3-Matic (Materialise, Leuven, Belgium) software and underwent mesh refinement in between 2 and 8 mm. For loading and boundary conditions, a point load with 1545.1 N magnitude was applied at distal part of socket whereas proximal part of socket was fixed in Marc. Mentat (MSC. Software, Santa Ana, CA) software. From the results, it is found that 5 mm is the optimum mesh size which the percentage difference below 5 % compared to 4 mm model with higher computational time. In conclusion, the method used for convergence study is reliable and low time consuming, where the outcomes are still promising.*

Keywords: Three-dimensional modelling; Mesh convergence analysis; Finite element analysis; Prosthetic socket

1. INTRODUCTION

These All Amputees are people who have lost all or portions of their bodies, making it difficult for them to carry out daily duties. Amputation can happen because of an accident or sickness. In Malaysia, the Ministry of Health (MOH) revealed that the number of diabetic amputees is on the rise. Lower limb amputation can be classified as transfemoral (above knee) and transtibial (below knee) being the most common amputation cases being reported [1]. A soft tissue pad is formed over the excised tibia and fibula, which includes the soleus muscles and gastrocnemius, in below knee amputation (BKA) operation. A skewed or posterior skin flap closes the site after the anterior tibia bone or tibial periosteum is sutured with the pad [2]. Numerous individuals whose lower limb was amputated hope to return to or even surpass their pre-amputation levels of activity [3].

Amputees require a prosthetic device, which is an artificial appendage that substitutes a lost body part after limb removal operation [4]. The goal of prosthesis–human interface design is to enhance convenience and functionality for amputee to facilitate ambulatory recovery. This is usually accomplished through the employment of a prosthetic socket, which is constructed by rectification process, making geometric adjustments to the acquired shape of the stump to achieve the required load transfer pattern [5]. The socket connects the prosthesis to the residual limb and is intended to offer effective motor control, comfort, and proper load transfer. Conventional procedures for fabricating sockets with required features are labour and expense demanding, and they rely on the expertise of competent prosthetists, who are in short supply compared to the number of lower limb amputees [6]. Insufficient socket fit causes discomfort and may result in prosthesis refusal, limiting everyday activities.

The designing, evaluation and fabrication of prosthetic socket can be optimized with incorporation of contemporary technologies [7]. The usage of computer-aided design and manufacturing (CAD/CAM) technology is rising, with the patellar tendon bearing (PTB) and total surface bearing (TSB) approaches often being used to construct sockets through digital customization of a 3D model scanned data. The CAD/CAM demands the same level of prosthetists skill and knowledge as same in plaster procedures [8]. This method typically entails digitizing the limb's surface contour with a surface scanner, executing patient-specific socket designing in a CAD software. In addition, Finite element analysis (FEA) has been recognized as an essential approach in evaluating load transmission in prosthesis due to its extensive employment in biomechanics to derive strain, deformation and stress in sophisticated systems. For parametric analysis of socket, FEA is an ideal approach to minimize cost and time with acceptable results as demonstrated by many researchers.

Previous researcher had implemented finite element analysis in prosthetic socket evaluation. As study by Fadzil et al., employed finite element method to evaluate the effect of infill density: 20%, 40%, 60%, 80% and 100 on the socket stress distribution and deformation. The distinct percentage was represented with distinct flexural and tensile modulus. Inside the socket, a load of 600N was applied and the socket was fixed at the bottom. From the findings, the 60 % infill density was selected as the optimal parameter the von Mises stress and deformation were low and aid in lowering the printing cost and time [9]. Even though there were several literature reporting about the use of FEA in evaluating prosthetic leg socket, however, limited information was found on the convergence study of socket to determine the best volume or mesh size prior to the parametric study. The mesh convergence study in FEA is important to determine optimum mesh size or element number that would not be affecting other parameters during the analysis. However, the convergence study on the prosthetic leg socket is very limited in the literature. Therefore, the aim of this study is not only to provide insight on the prosthetic leg socket modelling, but it is also for the mesh convergence study in FEA that would be useful in the future parametric study.

2. METHODOLOGY

The flow chart in Fig. 1 summarizes the process of mesh convergence study beginning from acquiring raw 3D model and finalized with finite element analysis.

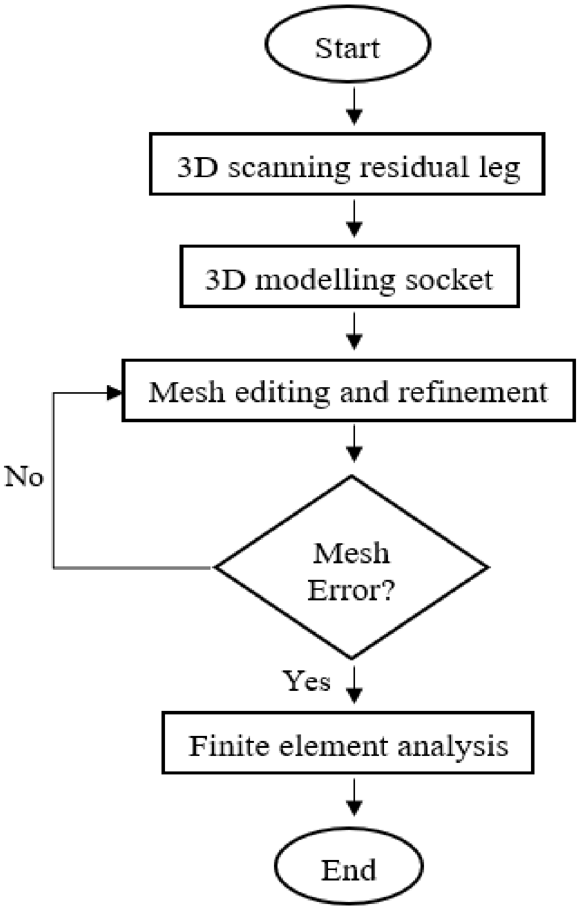


Fig. 1. Flow chart of the study

2.1 Three-dimensional Scanning of Residual Leg

A 50-year male amputee with 70 kg weight and 167 cm height was recruited and gave the consent to participate in the research. The selected volunteer fulfilled the criteria required such as above 18 years old, good cardiopulmonary condition, good cognitive condition, experienced prosthetic leg user, unilateral transtibial amputee and able to walk without aids. This protocol has been approved by UTM Research Ethical Committee with the approval number is UTMREC-2023-25. Prior to the scanning process, a tape was positioned on the surface of volunteer's skin, to ensure the next process of socket modeling could be done smoothly without compromising the quality and region of interest. The residual limb of the volunteer was 3D scanned using handheld optical Sense 3D scanner as shown in Fig. 2(a). During the data acquisition procedure, the volunteer was required to extend the residual leg and hold still until the scanning process was done. Then, the operator moved the scanner steadily and slowly around the whole residual leg to obtain the accurate 3D model. Later, the raw 3D model of the residual leg as shown in Fig. 2(b) underwent post processing via a computer-aided design (CAD) software, 3-Matic (Materialise, Leuven, Belgium). The 3D model of residual limb was smoothed, and the unwanted part was removed using the special tools in the software. The edited and final residual limb model as shown in Fig. 2(c) was finalized using the software for next stage which is three-dimensional modelling prosthetic socket. The data was saved in stereolithography (STL) format to be used in the next steps.

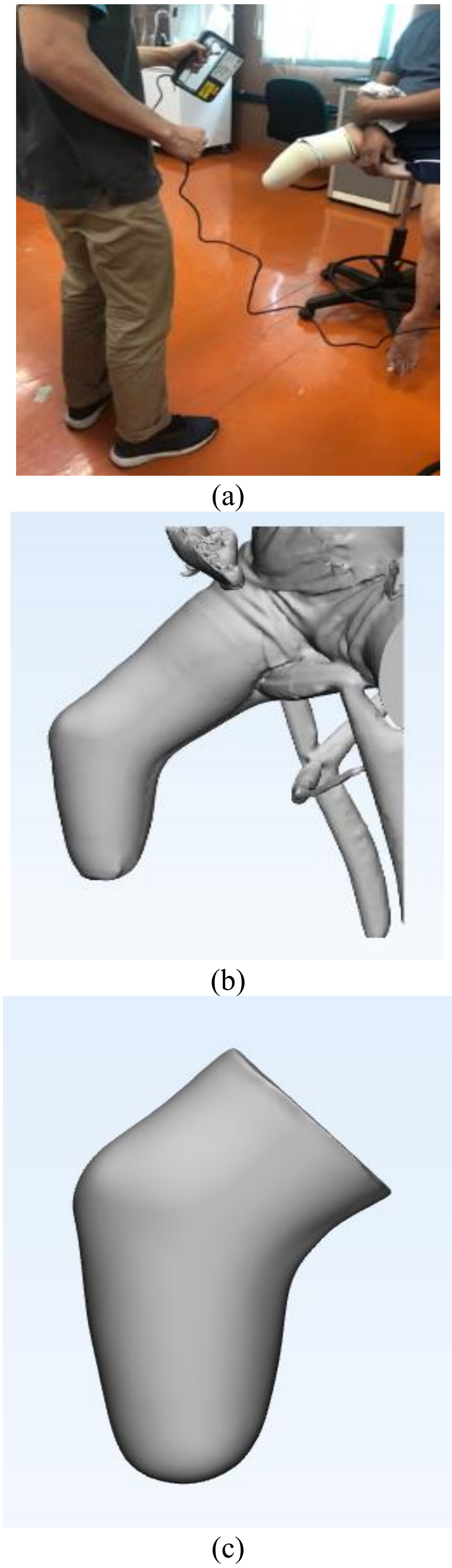
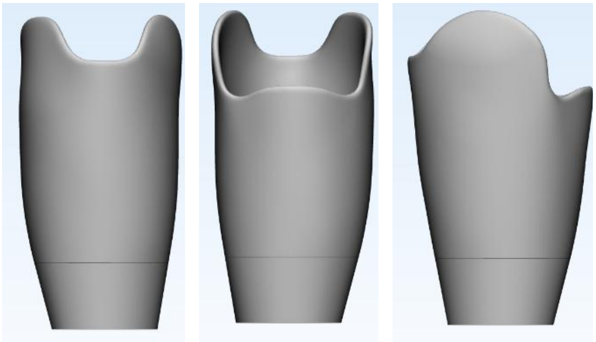


Fig. 2. (a) 3D scanning of volunteer's residual leg, (b) pre-processed scanned residual leg and (c) post-processed residual leg

2.2 Three-dimensional Modelling of Prosthetic Socket

From the STL format, a prosthetic socket was constructed via 3-Matic (Materialise, Leuven, Belgium). The region of interest of the socket (initially was done during the scanning process) was marked using mark brush tools in the software. Then, the marked area was separated to a new layer and thickened to 6 mm thickness using offsetting tools. The socket design resembled the conventional total surface bearing socket design where the pressure was transmitted on the residual limb uniformly as demonstrated by previous study [10]. For pylon attachment site, a specific design of cylinder base was placed at the bottom of the socket and were merged with the thickened entity. The sharp edge of the socket opening was smoothed using smooth edge tools. Finally, the fix wizard tool was applied to detect any problem on the model which can cause error during simulation. If errors found, the same steps of smoothing and eliminating surfaces or triangles were done afterwards. The final design of the prosthetic leg socket with different view were shown in Fig. 3(a), (b) and (c).

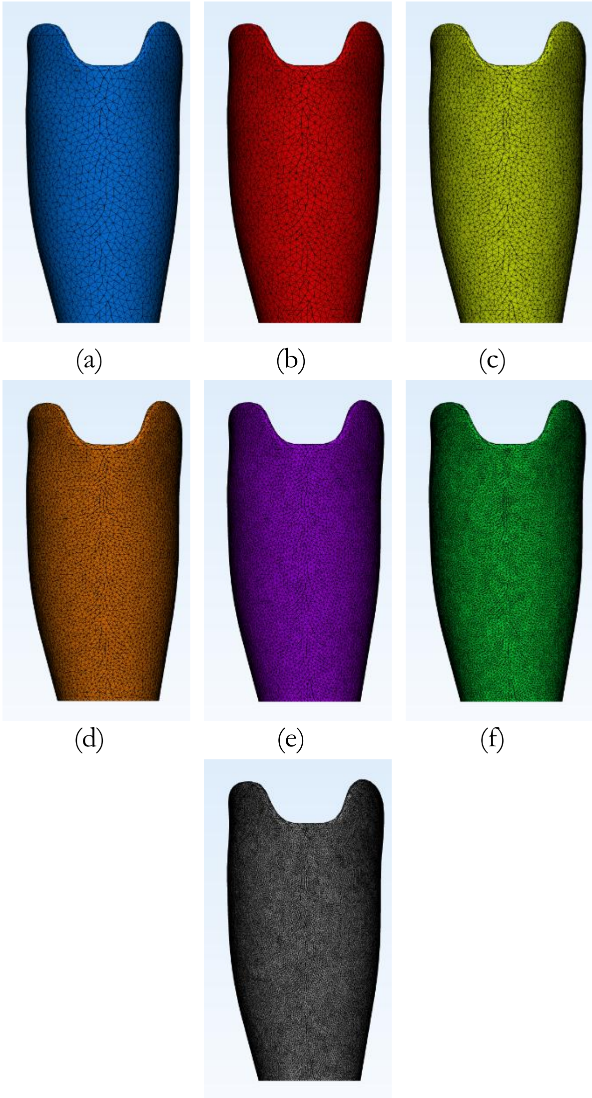


(a) (b) (c)

Fig. 3: (a) Front view, (b) back view and (c) side view of socket

2.3 Mesh Refinement

A mesh refinement was a crucial step in convergence study as the mesh size (maximum edge length) of finite element model can affect the final result of simulation although the boundary and loading condition were kept constant. In this study, H-refinement method was used where the elements were subdivided into several smaller elements and new nodes were generated automatically from the software. Thus, the surface mesh of the socket was meshed to several desired mesh size ranging from 2 mm to 8 mm in 3-Matic (Materialise, Leuven, Belgium). The first socket model was re-meshed to largest desired mesh size (8 mm) and subsequent socket model was refined to lower mesh size until 2 mm. The process was followed by solidifying each model into solid linear tetrahedral where the volume mesh size is equivalent to surface mesh size. Eventually, seven socket model with different mesh density were derived from the refinement process as shown in Fig. 4(a), (b), (c), (d), (e), (f) and (g). The number of elements and nodes for each socket are tabulated in Table 1. Since the software computed the magnitude of the stress at every node, time taken for the socket with the smallest mesh size or highest mesh density is the highest.



(a) (b) (c) (d) (e) (f) (g)

Fig. 4(a), (b), (c), (d), (e), (f) and (g): seven socket model with different mesh density

Table 1. The properties of the socket finite element model

Model	Mesh Sizes (mm)	Total Number of Elements	Total Number of Nodes	Time for FEA (s)
Socket 1	2	1959400	359600	1372.41
Socket 2	3	606686	121939	358.38
Socket 3	4	247774	51905	135.84
Socket 4	5	126455	29752	68.71
Socket 5	6	80609	19545	43.94
Socket 6	7	42251	10806	23.60
Socket 7	8	20083	5897	12.02

2.4 Finite Element Analysis

The next procedure in convergence study is finite element analysis for each socket model in Marc. Mentat (MSC. Software, Santa Ana, CA). The material assigned for the sockets are polylactic acid (PLA) where the

Young's modulus and Poisson's ratio were 3.75 and 0.3897 respectively [11]. The material properties of the sockets were set as linear isotropic and homogenous. A vertical point load with 1545.1 N magnitude was applied on the inferior of the socket with which correspond to 2.25 times the volunteer's body weight (70 kg). The magnitude of the load represented the loading during heel strike in gait cycle [12]. Meanwhile the proximal part of the socket was fixed in all direction as shown in Fig. 4. The steps were repeated for each of the socket model. Each model was simulated individually once. The von Mises stress distribution of each socket was analysed and compared consecutively between mesh sizes and the percentage differences was calculated.

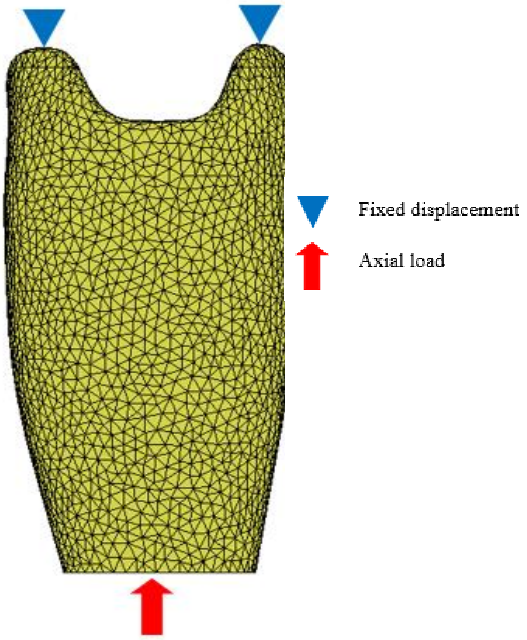


Fig. 5. The location of load and fixed region at the socket.

3. RESULT AND DISCUSSION

Percentage difference was calculated using the magnitude of VMS for five selected nodes at the same location on all models (Fig. 6). The percentage difference was obtained between two consecutive mesh size and tabulated in Table 2. The range of percentage difference between 8 mm & 7 mm mesh size is from 0% to 10%; between 7 mm & 6 mm mesh size is from 1% to 24%; between 6 mm & 5 mm mesh size is from 0% to 7%; between 5 mm & 4 mm mesh size is from 2% to 4%; between 4 mm & 3 mm mesh size is from 1% to 8%; between 3 mm & 2 mm mesh size is from 4% to 5%. The line graph plotted in Fig. 7 showed that the convergence was identified at 5 mm and 4 mm mesh size where the points of VMS magnitude for all five nodes almost stacked. The percentage differences were less than 5 % which justified the convergence at 5 mm and 4 mm [13]. Taking into account the computational time, mesh size, 5 mm was selected instead of 4 mm as it reduced the time for simulation.

This study consists of several limitations. First, the convergence study was conducted excluding other prosthetic components such as pylon, adapters, foot and liner. The exclusion of other components was to simplify the simulation and reduce overall computational time as well as to reduce the load and resources usage for low end computer that unable to simulate complex finite element body instantly. Furthermore, the loading condition did not imitate the loading from volunteer's body or residual limb because the residual leg model was absent during simulation process. The function of the residual limb was to exert load to overall prosthetic socket wall. The load applied and fixed region were set randomly on the socket model to evaluate the behaviour of the socket correspond to boundary and loading condition applied. Thus, the results of simulation only assist the researchers to understand the effect of mesh size on finite element analysis simulation instead of structural strength in prosthetic leg loading application. Another limitation of study is the number of participant where there was only one involved in this study. In the future, it is suggested to include more participants to ensure the data is more reliable and accurate.

In previous study, Ballit et al. conducted a mesh convergence analysis to verify the selected mesh size used for finite element simulation. They simulated the interaction between the stump and socket where they varied the mesh with four different sizes which the properties not stated by the author. At the selected points, the pressure for selected mesh size portrayed variation rate not more than 5% [13]. In other study conducted by Steer et al. they performed mesh convergence study for residuum-socket interface, varying mesh size ranging from 2 mm to 12 mm. The pressure at the location where related to socket comfort and fit on the liner: tibial tuberosity, head of fibula and residuum tip and strain of the soft tissue were used in the convergence study. The detail methods on how the optimum mesh size obtained from root mean square error calculation did not elaborated by the author. With a 5.0 mm length of element edge or a 2.5 mm node to node spacing, mesh convergence was achieved. This mesh size was chosen as a reasonable compromise between precision and computing resources [14]. Similar to our study, the converged mesh size was the same for socket structure despite using different method for computing the convergence. Thus, we conclude that our method for finding the convergence was reliable and acceptable.

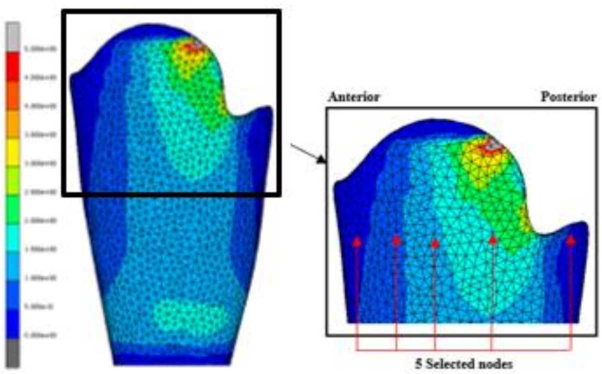


Fig. 6. The location of five selected nodes for obtaining von Mises stress.

Table 2. The properties of the socket finite element model.

Model	Percentage Difference [%]				
	Node 1	Node 2	Node 3	Node 4	Node 5
8 mm & 7 mm	0	1.1	2.88	2.49	10.1
7 mm & 6 mm	5.71	4.55	1.47	2.43	23.66
6 mm & 5 mm	0	2.35	0	1.9	7.06
5 mm & 4 mm	2.74	2.41	3.77	3.7	2.25
4 mm & 3 mm	7.79	1.23	1.55	5.74	2.2
3 mm & 2 mm	4.88	3.77	3.98	4.61	4.44

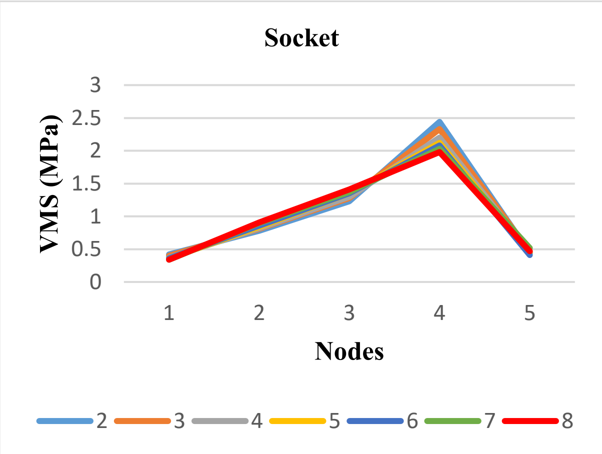


Fig. 7. Graph VMS versus location of nodes all mesh sizes

4. CONCLUSION

Mesh convergence analysis of prosthetic socket was successfully conducted in Marc Mentat (MSC. Software, Santa Ana, CA) software. Mesh refinement was done where existing nodes are retained, and new nodes are generated. It was discovered that the optimum mesh size (maximum element edge length) for the socket was 5 mm that produce 126455 elements and 29752 nodes. 5 mm is selected instead of 4 mm because time taken to complete the simulation is shorter [15-17]. The convergence study method implemented by the author was suitable and practical for low end computer and help to fasten the simulation process as only one body was simulated per simulation. Future improvement can be done on the study such as reducing the range of consecutive mesh size to improve the accuracy.

5. REFERENCES

[1] M. N. Azmi Ab Patar, M.S. Azlan, M.A. Mohd Hamim, J. Mahmud, H. Lee, Stress analysis of a patient-specific socket design during gait cycle, Int. J. Recent Technol. Eng. 8 (2019) 6972–6966.

[2] A.S. Dickinson, J.W. Steer, P.R. Worsley, Finite element analysis of the amputated lower limb: A systematic review and recommendations, Med. Eng. Phys. 43 (2017) 1–18.

[3] J.C. Cagle, P.G. Reinhall, K.J. Allyn1 J. McLean, P. Hinrichs, B.J. Hafner, J.E. Sanders, A finite element model to assess transtibial prosthetic sockets with elastomeric liners, Med. Biol. Eng. Comput. 56 (2018) 1227–1240.

[4] C. Nayak, A. Singh, H. Chaudhary, “Topology optimisation of transtibial prosthesis socket using finite element analysis, Int. J. Biomed. Eng. Technol. 24 (2017) 323–337.

[5] J.W. Steer, P.A. Grudniewski, M. Browne, P.R. Worsley, A.J. Sobey, A.S. Dickinson, Predictive prosthetic socket design: part 2—generating person-specific candidate designs using multi-objective genetic algorithms, Biomech. Model. Mechanobiol. 19 (2020) 1347–1360.

[6] S.G. Moulic, S. Singh, R. Hussain, G. Murthy, Y. Khawade, N. Bettaiah, Digital transformation and 3D printing of transtibial load-bearing prosthesis in India: recent advances, challenges and future perspectives, J. 3D Print. Med. 3 (2019) 185–193.

[7] J.W. Steer, P.R. Worsley, M. Browne, and AS. Dickinson, Predictive prosthetic socket design: part 1—population-based evaluation of transtibial prosthetic sockets by FEA-driven surrogate modelling, Biomech. Model. Mechanobiol. 19 (2020) 1331–1346.

[8] A. Dickinson, L. Diment, R. Morris, E. Pearson, D. Hannett, and J. Steer, Characterising residual limb morphology and prosthetic socket design based on expert clinician practice, Prosthesis, 3 (2021) 280–299.

[9] W.F.A.W. Fadzil, M.A. Mazlan, A.H. Abdullah, F.A. Hanapiah, A.I. Pangesty, Effects of infill density on 3d printed socket for transtibial prosthetic leg, J. Mech. Eng. 9 (2020) 229–238.

[10] M.R. Safari, M.R. Meier, Systematic review of effects of current transtibial prosthetic socket designs—Part 2: quantitative outcomes, J. Rehabil. Res. Dev. vol. 52 (2015) 509–526.

[11] H. Kamel, O. Harraz, K. Azab, T. Attia, Developing an optimized low-cost transtibial energy storage and release prosthetic foot using three-dimensional printing, J. Eng. Sci. Med. Diagnostics Ther. 3 (2020) 1–9.

[12] Y. Guo, X. Zhang, W. Chen, Three-Dimensional finite element simulation of total knee joint in gait cycle, Acta Mech. Solida Sin. 22 (2009) 347–351.

[13] A. Ballit, I. Mougharbel, H. Ghaziri, T.T. Dao, Computer-aided parametric prosthetic socket design based on real-time soft tissue deformation and an inverse approach, Vis. Comput. 38 (2022) 919–937.

[14] J.W. Steer, P.R. Worsley, M. Browne, A. Dickinson, Key considerations for finite element modelling of the residuum–prosthetic socket interface, Prosthet. Orthot. Int. (2020)1-9.

[15] C. Somtua, P. Aroonjarattham, N Saengpetch, N.

Rattanapan, K. Aroonjarattham, Biomechanical analysis of scoliosis adjusted by screw fixation system with finite element analysis, Eng. J. 28 (2024) 25-34.
[16] Z. F. M. Salaha, S. N. A. J. Efendie, M. I. Abd Samad, M. H. Baharuddin, A. H. Abdullah, M. H. Ramlee, Biomechanical evaluation of three-dimensional printed leg orthosis for ankle fixation: finite element and thermal analysis, Int. Exch. Innov. Conf. Eng. Sci. (2024) 564-570.
[17] S. A. Zulkefli, N. S. Ismail, M. I. Abd Samad, M. H. Baharuddin, A. H. Abdullah, M. H. Ramlee, Development and analysis of 3d-printed knee orthosis for post-anterior cruciate ligament injuries rehabilitation, Int. Exch. Innov. Conf. Eng. Sci. (2024) 583-589.