Original Research Paper

Volume - 10 | Issue - 9 | September - 2020 | PRINT ISSN No. 2249 - 555X | DOI : 10.36106/ijar

Physical Medicine

ANALYSIS OF INDIGENOUSLY DESIGNED EXTENDED KEEL CARBON FIBRE PROSTHETIC FOOT USING FEA

Sachin S Bhagat*	Lecturer in Prosthetics & Orthotics, GMC, Goa *Corresponding Author
Mr. A. G Indalkar	Lecturer and HOD, Dep. Of Prosthetics & Orthotics, AIIPMR, and Mumbai
Mr. Avinash Phirke	Professor, Watumull Institute of Engineering &Technology Mumbai.

ABSTRACT The biological foot designed by nature, which is propelled through a self dynamic mechanism is very intricate. No doubt, the design of this component is highly complex, it is perfectly tuned with response of highly vulnerable ground reaction force. In fact, it is very challenging task to design the foot, which is sophisticated, so far as the intrinsic and extrinsic activities are concerned. keeping this in mind we have developed an extended keel carbon fibre foot quasi indigenously. Various reasercher works towards the clinical trial and effectiveness of this designed foot. The current study aims to construct the finite element model of the present design of an extended keel prosthetic foot under loading condition, to analyse the finite element model, to characterised structural behaviour of present foot design under loading condition, thereby to predict the load distribution between the different section of the foot.

The study was focuses on Stress analysis and to know the failure prone areas of the foot. AutoCAD and Ansys 12.0 software were used to construct and analyses the model. Finite element models were constructed for three orientation of the gait i.e. heal strike, mid stance and push off phase. Finite element models were analyzed to know the failure prone areas and to observe the normal elemental stresses, shearing stresses and Von Mises stresses (combination stresses). During all the three stages, most of the stress variation and peak stress is found at shank spring (especially at ankle fillet). It was found that minimum stress is observed at foot spring during all the three stages of gait. The study alsoconclude that during Heel strike Peak stress of 49690 MPa is experienced at the lower ankle fillet region, therefore, additional care should be taken during actual designing of this part of the ankle fillet.

KEYWORDS : FEA, Finite element analysis, Ansys

INTRODUCTION:

The modified flex foot designed at All India Institute of Physical Medicine and Rehabilitation Mumbai, were under the stage of design optimization and further development. Various researchers' contributed towards this noble cause of design, material optimization and thus the further improvement in this foot. The current study aims to design the modified flexfoot (extended keel carbon Fibre foot) and then analyzed the foot using Finite Element Analysis (FEA). Finite element analysis is a computational tool that enables the complex structure to be broken down into smaller mechanically simpler regions then by assigning appropriate material properties and loading condition stress and strain can be determined, thus defining behaviour of the structure.

Aims & Objectives:

1. To construct the finite element model of the present design of an extended keel prosthetic foot under loading condition.
2. To analyse the finite element model.
3. To characterised structural behaviour of present foot design under loading condition.
4. To predict the load distribution between the different section of the foot.
5. To develope a multidisciplinary approach to prosthetic foot design and analytical testing.
6. To find out the failure prone areas in the present design of an extended keel foot

Design Materials & Methodology

1. **Research design** : Experimental.
2. **Parameters of study**
   a. AutoCAD foot modeling
   b. Stress analysis
   c. Finite element analysis

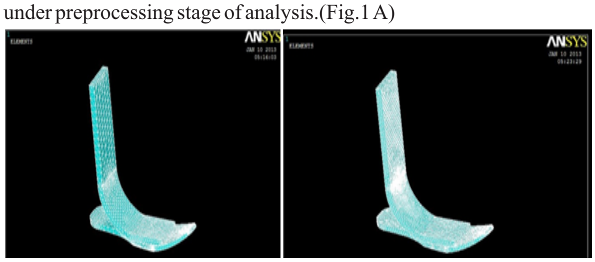
3. **Instrumentation**
   a. Ansys 12.0
   b. AutoCAD 2010

4. **Procedure:**

Study inhibits the following phases,

1. Fabrication of the actual prosthetic foot design.
2. Input parameters determination for finite element model.
3. Construction of 3D AutoCAD model of the foot. **(Preprocessing)**
4. Construction of finite element model (FEM). **(Processing or Solutions)**
5. Analysis of finite element model (FEA). **(Post processing)**

At the initial phase of the study the actual prosthetic foot which helps to generate the AutoCAD model of the foot was constructed. Input parameters like geometrical parameterand materialproperties, loading conditions & boundary conditions were determined. A 3D AutoCAD model is constructed using input parameters. Then the 3D AutoCAD model is imported into the Ansys 12.0 software, Mesh Model is created under preprocessing stage of analysis.(Fig.1 A)



(Fig.1 A: Coarse MESH Model) (Fig.1 B: Fine Mesh Model)

To increase the accuracy level of the solution, Mesh model is refined with smaller element size i.e. higher density mesh or Fine Mesh Model(Fig. 1 B) . Fine meshing is applied at the smaller dimensions and complicated points. With progression of study, failure prone areas are identified. Whereas To increase the accuracy of FEA model and to reduce the computational time, fine meshing is applied at those failure prone areas. Input parameters like loading conditions, loading orientation, boundary conditions and material properties are determined in the previous phase (Phase II) of the study. When the accurate fine meshing is finished the software required these input parameters. Maximum weights of 100 kg (981N) are applied at three different orientation of the gait. I.e. heel strike, mid stance and push off. Loading orientation taken as 30°, 0°, and 33º at heel strike, mid stance and push off respectively. Application of load is along the direction of flex spring and boundary conditions are different for each of the orientation.

Finite Element model is obtained for the applied conditions at the solution phase. In order to get the accurate results, the software solves the several equations during this phase.There were 64, 00,000 equations computer has solved to obtain the finite element model

Post processing stage, consists of obtaining pertinent information from the 'solved' model. After the mesh was successfully applied to the model and the program had finished running the model, the results were extracted to observe the **Von Mises stresses**, **shearing stresses**, and **elemental stresses** that occurred due to the applied loads.The results have been reported in the form of several contour plots, which are displayed through colours that correspond to different stress and displacement value levels (Fig 3).

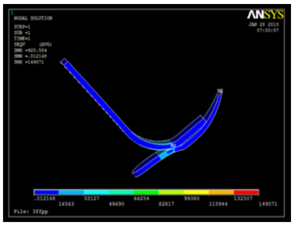


Fig 2 A:Von Mises stress solution duringHeel strike

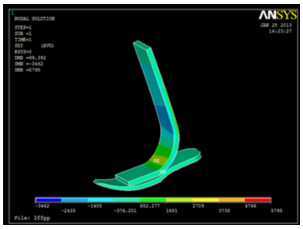


Fig 2 B:Nodal Solution for shear stresses

The colour scale of the contour plots was adjusted to ensure that the same values corresponded with the same stress values throughout all of the models.

Analysis and Results

Finite element models were analyzed to know the failure prone areas and to observe the normal elemental stresses, shearing stresses and Von Mises stresses (combination stresses). FEA result shows the stress distribution values experienced by the different parts of the foot during three different conditions of the gait. The stress distribution obtained from the analysis depicts the region of high stress &failure prone areas. Minimum stresses were observed at foot spring during all three stages of gait (Table 1) and thus foot spring has less chances of failure.

Table1: Stress distribution: at Heel strike

Stress distribution, During Heel strike.					
Parts of the foot			**Predicted FEA stress values, all are in N/mm2 (or MPa)**		
	Types of stress		**Normal, elemental stresses**	**Shear stresses**	***Von Mises* stresses**
Shank Spring	1. Proximal shank spring		0.011514	-991.669	0.012148
	2. Middle Shank spring		0.011514	-991.669	0.012148
	3. Ankle Fillet	upper	0.011514	-991.669	16563
		Middle	0.011514	-991.669	33127
		Lower	33665	**7805**	**49690**
	4. Base plate		0.011514	-991.669	0.012148
Foot Spring	5. Heel spring		0.011514	-991.669	0.012148
	6. Mid foot spring		16833	**7805**	33127
	7. Toe spring		0.011514	-991.669	0.012148

Table2: Stress distribution: at Push off

Stress distribution During Push off,					
Parts of the foot			**Predicted FEA stress values, all are in N/mm2 (or MPa)**		
	Types of stress		**Normal, elemental stresses**	**Shear stresses**	***Von Mises* stresses**
Shank Spring	1. Proximal shank spring		0.003379	-1082	0.003965
	2. Middle Shank spring		0.003379	-1082	0.003965
	3. Ankle Fillet	upper	2395	-1082	3416
		Middle	4790	368.819	3416
		Lower	7184	1820	5123
	4. Base plate		0.003379	-1082	0.003965
Foot Spring	5. Heel spring		0.003379	-1082	0.003965
	6. Mid foot spring		0.003379	-1082	0.003965
	7. Toe spring		0.003379	-1082	0.003965

The study concludes that, the present design of an extended keel modified flex foot will not fail under this loading condition because the model does not show any failure at specific region.

REFERENCES:

1. Marnie M. Saunders, Edwards P. Schwentker et. al(2003): Finite Element Analysis as a Tool for Parametric Prosthetic Foot Design and Evaluation. Technique Development in the Solid Ankle Cushioned Heel (SACH) Foot, Computer Methods in Biomechanics and Biomedical Engineering,
2. Ahmet Erdemir, Trent M. Guess, JasonHalloran, Srinivas C.Tadepalli et.al.Considerations for reporting finite element analysis studies in biomechanics, Journal of Biomechanics 45 (2012) 625–633.
3. Milan Omasta, David Paloušek, Tomas Návrat, Jiří Rosický, Finite element analysis for the evaluation of the structural behaviour, of a prosthesis for trans-tibial amputees, journal of Medical Engineering &Physics, Volume 34, Issue 1, January 2012, Pages 38–45.
4. Silverthorn MB, Childress DS. Parametric analysis using the finite elementmethod to investigate prosthetic interface stresses for persons with trans-tibial amputation. Journal of Rehabilitation Research and Development1996;33(3):227–38.
5. Quesada P, Skinner HB. Analysis of a below-knee patellar tendon-bearingProsthesis—a finite-element study. Journal of Rehabilitation Research andDevelopment 1991; 28(3):1–12.
6. Strang, Gilbert, Fix, George (1973). An Analysis of the Finite Element Method. Prentice Hall. ISBN)-13-032946-0