Paper ID: RT-44

A Finite Element Analysis to Design and Simulation of Effective Shank Piece in Ladies High Heel Shoe

Nil Ratan Mondal[1], Md. Minar Ahmed[2]*, Md. Samsul Arefin [3], Mahabub Hasan Rony [4]

[1,2] Undergraduate Student, Department of Leather Engineering, Khulna University of Engineering and Technology, Khulna 9203, Bangladesh
[3] Assistant Professor, Department of Leather Engineering, Khulna University of Engineering and Technology, Khulna 9203, Bangladesh
[4] Undergraduate Student, Department of Mechanical Engineering, Khulna University of Engineering and Technology, Khulna 9203, Bangladesh

E-mail: nilratan33@gmail.com, minarnahmed@yahoo.com, arefinkuet@gmail.com, mahabubrony16@yahoo.com*

Abstract

High heel shoes are a type of footwear typically worn by ladies. In ladies high heel shoe, shank is a very important component. It mainly made of steel, metal, wood, plastic but mostly steel shank piece are used. Before using a steel shank piece for high heel ladies shoe, it is important to know its effectiveness. For this purpose several prototype shank pieces are tested in biomechanics lab over and over. It is uneconomical and time consuming. To reduce the cost and time consuming trial and error cycle, a finite element based method can be used to simulate the performance test of the shank piece. In this study a shank piece with high heel was designed in Solid Works Software 2013. After that by assigning the material properties of cast carbon steel shank piece, the simulation was carried out to get the analyzed results. The simulation results reveal that in static analysis the maximum von messes stress is 914216704 N/m² and minimum von messes stress is 3.49019 N/m². The maximum strain is 0.00279003 and minimum strain is 2.77075e-011. The shank piece will be withstand up to maximum 10000 cycles and minimum 6000 cycles. Finally the simulated results were verified for the model correctness based on the comparative study with the standard requirements of the shank piece .The study represent that the Finite Element Analysis is an emerging approach in footwear industry for reducing time and cost in prototype making.

Keywords: High Heel Shoe, Shank Piece, Performance Test, Finite Element Analysis

1. Introduction

High heel shoes are a type of footwear typically worn by ladies where the heel of the foot is raised significantly higher than the ball of the foot and the toes [1]. It tend to give the aesthetic illusion of longer, more slender legs. High heel infuse the wearer with a sense of power, more importantly feminine power [2]. It come in a wide variety of styles, and the heels are found in many different shapes, including stiletto, pump (court shoe), block, tapered, blade, and wedge.

"High heels" covers heel height ranging from 2 to 5 inches (5.1 to 12.7 cm) or more. Extremely high-heeled shoes, such as those exceeding 6 inches (15 cm), strictly speaking, are no longer considered apparel but rather something akin to "jewelry for the feet". They are worn for display or the enjoyment of the wearer.

In ladies high heel shoe, shank is one of the vital component. The shoe's shank is the center section or bridge between the ball line sole and the front of the heel [3]. The shank bridges between the heel breast and the ball tread. The shank spring lies within the bridge or waist of the shoe, i.e. between heel and ball corresponding to the medial and lateral arches [4]. The shankpiece reinforces the waist of the shoe and prevents it from collapsing or distorting in wear. It provides essential support for the arch, maintains accurate alignment of forepart and heel, helps to retain the shoe shape, maintains structural stability in high heeled shoes and gives strength to the waist of footwear, absorbs shocks, resists stresses and strains, during weight bearing of the body [5]. It extending downwardly and forwardly from the first portion of the arch support, and a toe support region which extends at an inclination upwardly and forwardly from the shank where upon the first metatarsal of the wearer is buttressed by the phalanges of the wearer to prevent forward sliding of the foot of the wearer relative to the shoe, thereby preventing jamming of the human digits into the toe portion of the shoe [6].

The higher the heel height of the shoe the stronger the shank piece material must be. It mainly made of steel, wood, plastic, metal and mostly steel shank piece are used [7]. Before using a steel shank piece for high heel ladies shoe it is important to know its suitability, strength, flexibility, deformation scale, fatigue life, stiffness for the specific heel height of shoe .For this purpose several prototype shank pieces are tested in biomechanics lab over and over to obtain optimum properties for production [8]. The process is time consuming and costly. To reduce the cost and time consuming trial as well as error cycle a finite element analysis (FEA) based method could be used to analyze the performance test of the shank piece.

Amongst various computer aided design (CAD) simulation techniques; Finite Element Method (FEM) is becoming more and more popular because of its versatile and controllable accuracy [9] in modeling irregular geometric structure, complex material properties and ease of simulating complicated boundary and loading conditions in both static and dynamic simulation [10].

Numerical simulation technique such as the finite element method (FEM) could allow the realistic simulations of foot as well as footwear interface which could offer in-depth biomechanical information including internal stress and strain distributions of modeled structure [11]. Finite element analysis is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects [12].

Simulation is a tool that extensively used in manufacturing system design and analysis for more than 40 years [13]. Computer aided engineering technique allow rapid change of input parameters to analyze their subsequent effects in virtual simulation environment without needing to conduct actual experiment.

In this study, an attempt has been taken to design a solid model of shank piece in order to evaluate the physical properties of a cast carbon steel shank piece such as von misses stress, strain, displacement, fatigue life using Finite Element Analysis (FEA). Moreover, most previous researches focused on the finite element analysis of biomechanics of the human foot, foot-footwear interaction, reduction of planter heel pain through insole but no research has been performed regarding the analysis of effective and durable shankpiece development which plays a vital role for the sustainability of a high heel shoe.

2. Methodology

2.1 Creating Solid Geometry of High Heel Shank Piece

An accurate solid geometry of a shank piece with high heel was designed by SolidWorks 2013 software (SolidWorks Corporation, Massachusetts) for analyzing the performance properties of a shank piece. In **Fig. 1**

shows the physical view of ladies high heel shoe and **Fig. 2** represents the solid geometry of shank piece.

Fig. 1. Ladies High heel shoe

Fig. 2. Shank Piece with heel shoe

2.2 Assigning Material Properties

The selected material properties were assigned to evaluate actual physical properties of cast carbon steel shank piece such as elastic modulus, Poisson's ratio, mass density, tensile strength, yield strength, thermal co-efficient. Materials properties often vary to some degree according to the direction of material in which they were measured. Material properties that relate two different physical phenomena often behave linearly in a given operating range, and may then be modeled as a constant for that range. The material properties of cast carbon steel are given below-

Table 1. Material Properties of Cast Carbon Steel Shank piece

Material Property	Value	Unit
Yield strength	2.48168e+008	N/m^2
Tensile strength	4.82549e+008	N/m^2
Elastic modulus	2e+011	N/m^2

Poisson's ratio	0.32	
Mass density	7800	kg/m^3
Shear modulus	7.6e+010	N/m^2
Thermal expansion coefficient	1.2e-005	/Kelvin

2.3 Loads and Boundary Condition

In case of finite element anlysis (FEA) an important step is the application of load and boundary condion. The force is applied on the top surface of the shank piece vertically depending on the body weight where the heel portion of the shank piece which is connected to the high heel was subjected to fix geometry. In this study 588 N force that was equal to the 60 kg weight of human body was applied on the shank surface (Fig. 3).

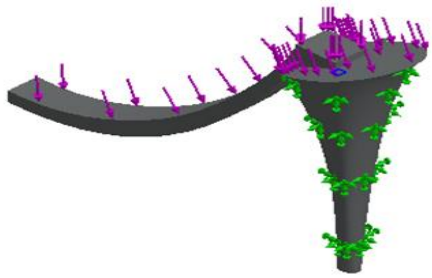

Fig. 3. Loads and Fix Geometry of the Shank Piece and High heel

2.4 Meshing the Solid Geometry of Shank Piece

Meshing is a crucial step in FEA. The automatic mesher in the software generates a mesh based on a global element size, tolerance and local mesh control specifications. A mesh consists of one type of elements unless the mixed mesh type is specified. The size of the generated mesh depends on the geometry and dimensions of the model, element size, mesh tolerance, mesh control, and contact specifications. The accuracy of the simulation mainly depends on the meshing. The solid model of the shank piece was finely meshed (Fig. 4).

Fig. 4. Meshed Model of Shank Piece

2.5 Running the Simulation Analysis

The simulation analysis of the shank piece was carried out for static (stress, displacement, equivalent strain), fatigue life analysis.

2.5.1 Von Misses Stress Analysis

The von misses yield criterion suggests that the yielding of materials begins when the second deviatory stress invariant reaches a critical value. For this reason, it is sometimes named as the -plasticity or flow theory. It is part of a plasticity theory that applies best to ductile materials, such as metals. Von misses stress is widely used by designers to check whether their design will withstand in a given load condition. It is considered to be a safe haven for design engineers. The force was applied on the top surface of the shank piece, then the von-misses stress was found, maximum and minimum values were noted.

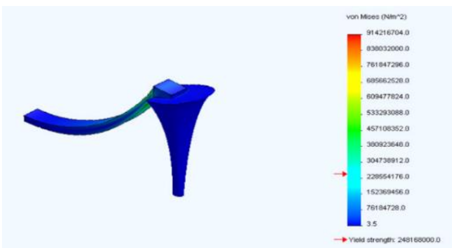

Fig. 5. Von Misses Stress Analysis

Table 2. Von Misses Stress Analysis

Name	Type	Min	Max
Static	Von Misses Stress	3.49019 N/m^2	9.14217e+008 N/m^2

2.5.3 Equivalent Strain Analysis

This analysis was carried out to know where the strain was occurred in shank piece in case of stress. It is the relative change in shape of an object due to applied force. It is an actual physical quantity related to the change in dimensions. In most cases strain is known as equivalent of strain.

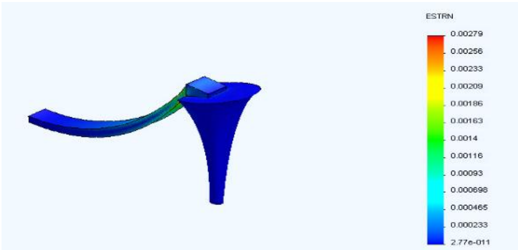

Fig. 6. Equivalent Strain Analysis

Table 3. Equivalent Strain Analysis

Name	Type	Min	Max
Static	ESTRN: Equivalent Strain	2.77075e-011	0.00279003

2.5.3 Displacement Analysis

This analysis was carried out to know how much the cast carbon steel shank piece is displaced by load from its original condition mainly in the fore part which is very essential portion to maintain the shape of the shoe and human foot**.**

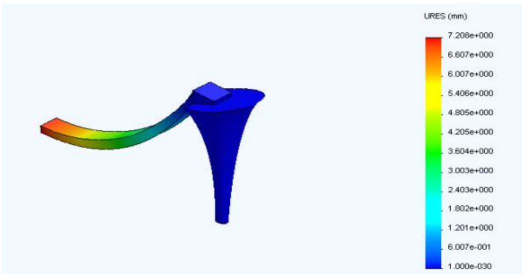

Fig. 7. Displacement Analysis

Table 4. Displacement Analysis

Name	Type	Min	Max
Static	URES: Resultant Displacement	0 mm	7.20802 mm

2.5.4 Fatigue Analysis

Fatigue is the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. This analysis was carried out to ensure how long the shank piece will be survived. Prior to fatigue analysis, the stress and strain analysis were performed. The fatigue analysis includes the damage plot (Fig. 8) and the S-N curve (Fig. 9).

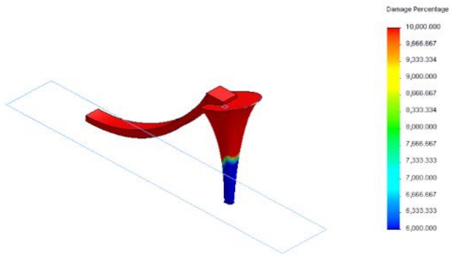

Fig. 8. Fatigue Life of Shank Piece

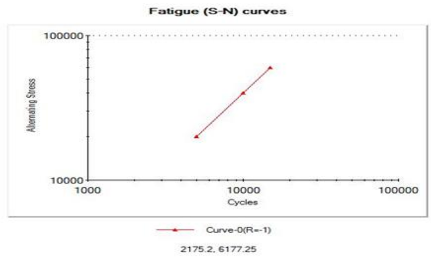

Fig. 9. S-N Curve

Table 5. Fatigue Life Analysis

Name	Type	Min	Max
Fatigue Life	Damage plot	6000	10000

3. Results and Discussion

The static simulation results reveal that in the back part of the shank piece (fig.5) the maximum von misses stress is 914216704 N/m^2 and minimum von misses stress is 3.49019 N/m^2. The red marked portion (fig.5) which is located in the back part of the shank piece where mainly maximum stress occur and the blue marked portion shows that there is minimum stress .Von misses stress is widely used by designers to check whether their design will withstand a given load condition. Bearing this information an engineer could state his/her design will fail, if the maximum value of von misses stress induced in the material is more than strength of the material [14]. Also in the back part of the shank piece (fig.6) the maximum strain is 0.00279003 and minimum strain is 2.77075e-011. Practical measurements of the strains on the shank have also shown that the major strain occur on the shank back part. From stress and strain analysis, it was found that the maximum stress and strain was occurred in the back part of the shank piece. Therefore, material density should be high in this portion to prevent crack development.

The displacement result reveals (Fig.7) that in the forepart of the shank piece, the maximum displacement occur which is 7.20802 mm and in the back part the minimum displacement occur which is 0 mm .The fatigue analysis (Fig.8) shows that the cast carbon steel shank piece will be sustained maximum 1000 cycles and minimum 6000 cycles. According to the fatigue life measurement done by SATRA, the shank piece is near to the grade 3 at the maximum cycle stage. The life cycle of the cast carbon steel can easily be extended by increasing the proportion of elastic modulus, mass density, tensile strength, yield strength.

The validation of the finite element analysis results mainly depends on the accurate materials properties selection. In this study, this factor was maintained properly. Knowing about the effectiveness of the shank piece is very essential before using the in high heel design construction which can easily be done by finite element analysis.

4. Conclusion

In this research paper, complete solution of virtual prototype making of shank piece and evaluation of its physical performance was presented. In industry, vast production of a product mainly depends on prototype model that has to pass through time consuming trial and error cycle for its performance testing. In this case finite element analysis can reduce time consuming trial and error cycle. The effectiveness of the shank piece under loading condition during walking can be easily predicted by finite element analysis. The simulated results which were found on finite element analysis facilitates to take decision on which portion more attention should be given for making stable shank piece for using as well as for protecting the foot on sudden failure.

Besides computer aided design (CAD) provides extra facility to footwear industry for product design and development and its sustainability during wearing life.

5. References

[1] E.Y.L. Au, R.S. Goonetilleke, “A qualitative study on the comfort and fit of Ladies dress shoes”, *Applied Ergonomics*, Vol. 38, pp.687-696. 2007

[2] http://serendip.brynmawr.edu/exchange/node/1923

[3] W. A Rossi , R. Tennant , FROFESSIONAL SHOE FITTING, *Pedorthic Footwear Association with acknowledgement to the National Shoe Retailers Association*, pp. 44, 2000

[4] Ö. H. ÖZKAN, "A RESEARCH ON FOOTWEAR AND FOOT INTERACTION THROUGH ANATOMY AND HUMAN ENGINEERING", *MSc Thesis, The Graduate School of Engineering and Sciences of Izmir Institute of Technology* ,2005

[5] B. VENKATAPPAIAH, Mrs. B.SITA, INTRODUCTION TO THE MODERN FOOTWEAR TECHNOLOGY,FIRST ED., *Central Leather Research Institute, Chennai*, pp. 51-52, 1997

[6] http://patents.justia.com/patents-by-us-classification/36/34R

[7] A. J. HARVEY, FOOTWEAR MATERIALS AND PROCESS TECHNOLOGY, *N. Z. Leather and Shoe Research Association*, pp. 209, 1992

[8] R.S. Goonetilleke, The Science of Footwear, *Taylor &Francis Group*, New York, pp. 322, 2012

[9] M. J. Rupérez, E. Giner, C. Monserrat, and E. Montiel, "Simulation of the behavior of the calfskin used as shoe upper material in footwear CAD", *Computer-Aided Design*, vol. 44, pp. 1205–1216, 2012.

[10] M. R. Shariatmadari, R. English and G. Rothwell, "Finite Element study into the Effect of Footwear Temperature on the Forces Transmitted to the Foot during Quasi-static Compression Loading", *IOP Conference Series Materials Science and Engineering*, vol. 10, 2010.

[11] J. T. –M. Cheung, J. Yu, D. W-C. Wong, and M. Zhang, "Current method in computer aided engineering for footwear design", *Footwear Science*, vol. 1, pp. 31-46, 2009.

[12] A. John, Dr. Y. V. K. S Rao, J. Sabu, and V. R Rajeev, "Stress Analysis of Polyoxymethylene which Leads to Wear in Pin on Disc Configuration using Finite Element Method", *International Journal of Research in Advent Technology*, vol. 2, no. 4, pp.305-310, 2014.

[13] S. S. Jeffrey, "Survey on the use of simulation for manufacturing system design and operation", *Journal of Manufacturing Systems*, vol. 22, pp. 157–171, 2003.

[14] M. S. Ganesh, G. P. Patil B, and N. A. Kharche, "Performance of the Structural Analysis of Ford Car Steering Rod", *International Journal of Research in Advent Technology*, vol. 2, no. 2, pp. 1-5. 2014.