JÓVENES EN LA CIENCIA
REVISTA DE DIVULGACIÓN CIENTÍFICA

Finite Element Analysis (FEA) of Swanson Flexible Hinge Toe Implant

Xiao (Cathy) Zhou (1), Agustín Vidal Lesso (2)

1 Candidate for Bachelors of Science in Mechanical Engineering, Massachusetts Institute of Technology | xiaozhou@mit.edu

2 Departamento de Ingeniería Mecánica, Biomecánica, DICIS, Universidad de Guanajuato | agustin.vidal@ugto.com

Abstract

Hallux Rigidus is an arthritis in the joint of the big toe that make it difficult to walk, run, and bend the toe. The condition can be temporarily relieved by surgery, but silicon elastomer implants can provide extra support in the toe after surgical procedures. The aim of this study was to conduct sensitivity analysis through finite element analysis on the Swanson Flexible Hinge Toe Implant through ANSYS software. ANSYS calculates von-Mises stress, strain, and total deformation on the implant for varying mesh size and applying an angle of 90º in one side of the implant. Through graphical analysis, a mesh size of 2.5 x 10^{-4} m was selected for optimal results. On the other hand, a structural analysis was performed using this mesh and one side of the implant was fixed and varied forces were applied to the other end to simulate conditions the toe may experience. Also, these forces let us compare the finite element results with the analytical solution for the geometry. Results showed a significant difference between FEM and analytical solution. This highlights the importance of using finite element method for irregular geometries.

Key Words 1. Hallux Rigidus, 2. Surgery, 3. Implant, 4. Finite Element Analysis (FEA), 5. Mesh Sensitivity.

INTRODUCTION

Osteoarthritis of the first metatarsophalangeal joint or hallux rigidus is characterized by pain and stiffness of the joint. Hallux rigidus is a common disease causing pain, restricted mobility, and reduced quality life. The goals of surgical intervention for hallux rigidus are to relieve pain, increase dorsiflexion and maintain stability of the first metatarsophalangeal joint. Replacing the joint by an implant is one of the surgical treatment modalities for the advanced stage of hallux rigidus. It is well known that the study of total joint replacement of the first metatarsophalangeal joint by means of finite element remain issues for attention, thus, in order to perform a biomechanics study of the surgical intervention for total joint replacement in hallux rigidus cases this work is proposed.

Hallux Rigidus

Hallux Rigidus is the arthritis of the big toe joint in which pain and stiffness in the joint makes it difficult to bend the toe.[1, 2] The condition is progressive in which sufferers' range of motion decreases from being limited, or "hallux limitus", until the foot becomes stiff or frozen, "rigidus". Overuse of the big toe can lead to this condition and those with the genetic predisposition, who have fallen arches or excessive pronation, and an elevated first foot bone (metatarsal) are among the causes of Hallux Rigidus. [1, 2]

There are several methods of surgery corresponding to various grades of pain. None, however, offer a perfect solution and implants are considered an advantageous addition to surgical removal of obtrusive bones in the toe. For example, dorsal chielectomy is recommended for patients with Grade 1 in which bone spurs are removed.[3] Chielectomy also shows some success in Grade 2, it fails to increase overall range of motion at the 1st MTPJ.[3,4] While the dorsiflexion of the first MTPJ increases, Proximal Phalanx pivots rather than glides and produces an abnormal motion. Arthrodesis is recommended to treat Grade 3, which produces a strong hallux in the long term and has a 90% success rate, but the joint loses its stability and the toe becomes useless.[4] Toe implants can be added to improve toe dynamics by relieving pain and the toe maintains stability and mobility.[5] Unfortunately, the implant can degrade over time due to reaction forces of bones.

Finite Element Analysis (FEA)

Conducting Finite Element Analysis (FEA) on implants can aid in improving functionality and design of implants. This mathematical model correlates accurately to experimental model of load displacement curves for boundary and loading conditions that are well-defined.[6] FEA can predict cracks and surfaces that develop after an extended period of time that flexion tests do not immediately show

Markers in structural analysis established a 3D line diagram of shin and foot to analyze gait for vector ground reaction forces.[7] Velocity and acceleration of heel rise can be calculated as well as calculating maximum angle of third rocker. It can also identify high-pressure areas as well as the center of pressure.[7] After performing FEA, results of information led to assumption that 1st MTPJ, surrounding muscles and ligaments were inactive until the start of heel rise. Matrix analysis can be used to calculate center of rotation based on two points on each surface of the metatarsal head and proximal phalanx.[7]

Understanding force components, stress, and material properties of bones will influence the decision for creating appropriate implants. Under normal physiological loads, FEA results show that articular cartilage underwent normal stress loads with maximum stress at 3.6 MPa.

Mesh sensitivity analysis creates numbers of elements that calculate forces, stress, strain and more at corresponding segments of a part. For

JÓVENES EN LA CIENCIA
REVISTA DE DIVULGACIÓN CIENTÍFICA

example, affine transformational matrices calculate center of rotation of first MTPJ based on two points on each surface of the metatarsal head and the proximal phalanx.

Flavin et al. calculated that bones had 204,006 tetrahedral elements whereas soft tissues had 291,980 nodal elements. The shape of the elements was determined by its material property as shown in Table 1.

The aim of this work is to model and simulate a flexible finger joint implant applied in cases of the hallux rigidus pathology by means of finite element. The structural analysis of the implant will be carried out in order to obtain a mesh sensitivity analysis. This model will be used in a total model of the first metatarsophalangeal joint.

Table 1. Material Properties Used to Calculate Number of Elements in Mesh

Type	Materials	ρ [kg/m^3]	E [MPa]	v
Bones	Metatarsal head, Proximal phalanx, etc.	1900	1600	0.28
Soft Tissue	Articular Cartilage	1100	2100	0.1
	Muscles	110	126	0.485
	Ligaments & Capsule	1100	260	0.4
	Tendons	1100	2700	0.47
	Plantar Fascia	1100	200	0.4

MATERIAL & METHODS

Swanson Flexible Hinge Toe Implant is a silicone elastomer double-stemmed flexible hinge implant that restores function to metatarsophalangeal joint. The mid-section is thicker and wider to meet anatomical and physiological requirements of MTP joint. It is designed as a load-distributing hinge that is not fixed to the bone, which encourages bone remodeling. In addition, the proximal and distal stems have a rectangular cross section that provides stability in intramedullary canals.

To conduct the Finite Element Analysis (FEA) of the implant, we used ANSYS simulation software. The anatomy of the foot provided necessary boundary conditions that were inserted into the software. For example, when the foot stands at a normal angle with respect to ground between the metatarsals and phalanges, it simulates a toe-off position. Thus, we fixed the phalanx side of the implant and applied a 90°-rotational displacement at the metatarsal side.[8]

Mesh size directly influence the number of elements and subsequently the maximum stress and strain that is calculated. We varied the mesh size to find the minimum size necessary to calculate stress and strain levels, shown in figures 1 and 2. It is expected that the stress/strain versus mesh size should follow a logarithmic curve.

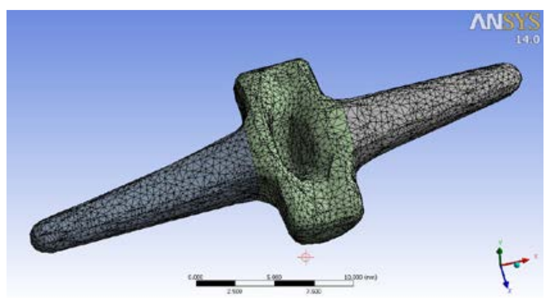


Fig. 1. ANSYS Simulation of Mesh Size 0.5 mm, close to default size.

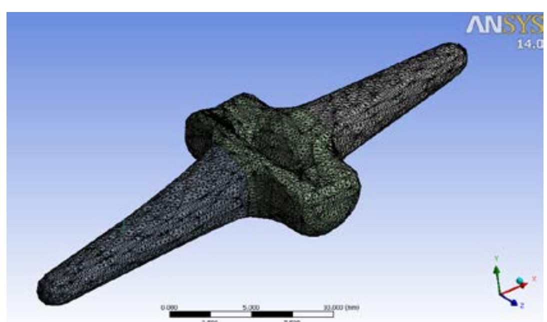


Fig. 2. ANSYS Simulation of Mesh Size 0.15 mm.

Once the mesh size was defined, a force needed to be applied to the metatarsal end of the implant, while the phalanx side remained fixed. Element size of 2.5 × 10^{-4}m was used as the optimal mesh size for the normal stress calculation. The ideal force was unknown, so we varied the force to see which produced a small stress on the implant. We initially chose 50 N, calculated the approximate stress we expected and compared it to the solution that the software computed. The force was then varied to compare amongst each other. The equation used to estimate stress is:

$$\sigma = \frac{My}{I} \qquad [1]$$

where M = moment ($length\ x\ Force$), y = distance from the center point of the wing to its top sections, I = Moment of inertia (obtained from Solidworks).

RESULTS & DISCUSSION

As shown in figures 3 and 4, the general trend of stress and strain curve increases with decreasing element size and behaves very similarly to each other. Because the stress oscillates around 3 × 10^{11} Pa and the strain oscillates around 1.5, the point that falls closest to both values is the element size 2.5 × 10^{-4}m. Figure 5 and 6 show the distribution of stress for the greatest mesh size and the smallest mesh size used in this study, respectively. In contrast, figures 7 and 8 show total deformation.

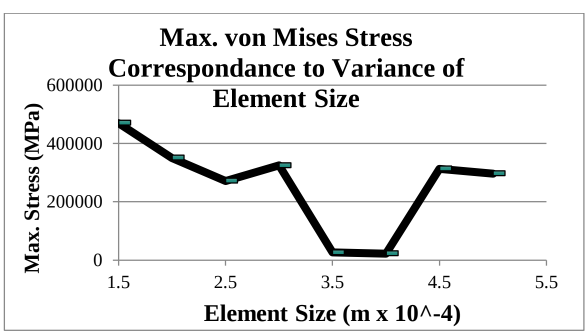


Fig. 3. Graph of Varying Element Size and its Correlation of Varying Maximum von Mises Stress.

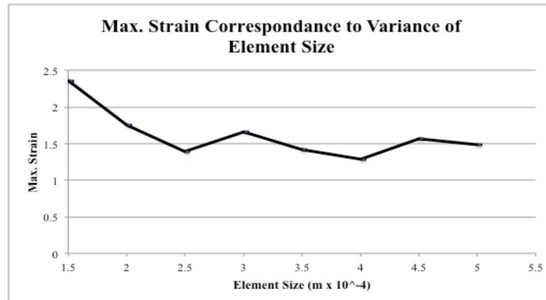


Fig. 4. Graph of Varying Element Size and its Correlation of Varying Maximum Strain.

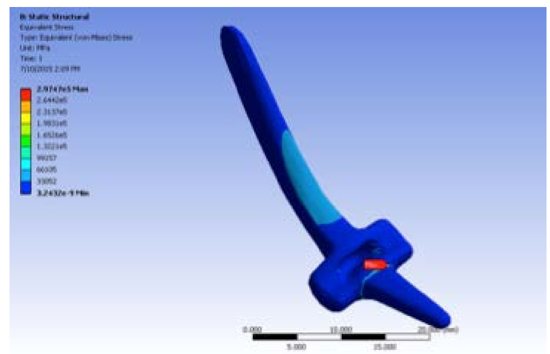


Fig. 5. von-Mises Equivalent Stress of Swanson Toe Implant of Mesh Size 5 × 10-4m.

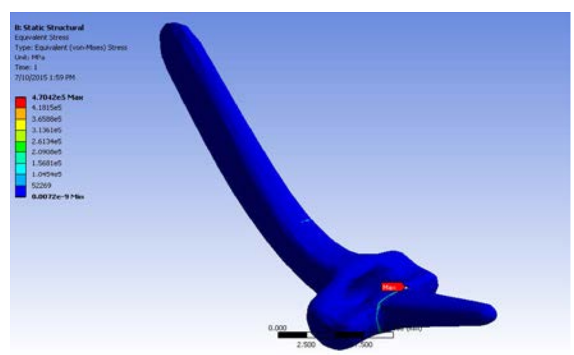


Fig. 6. von-Mises Equivalent Stress of Swanson Toe Implant of Mesh Size 1.5 × 10-4m.

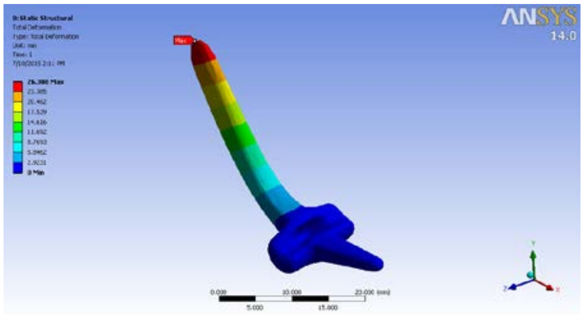


Fig. 7. Total Deformation for Mesh Size 5×10-4m.

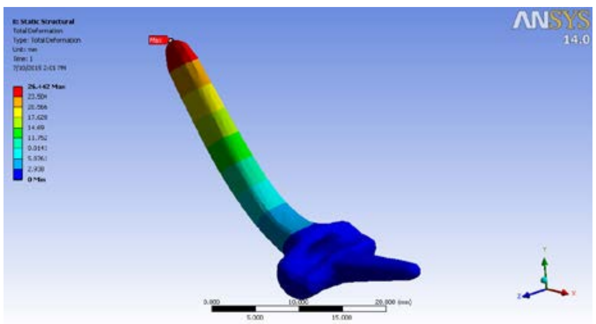
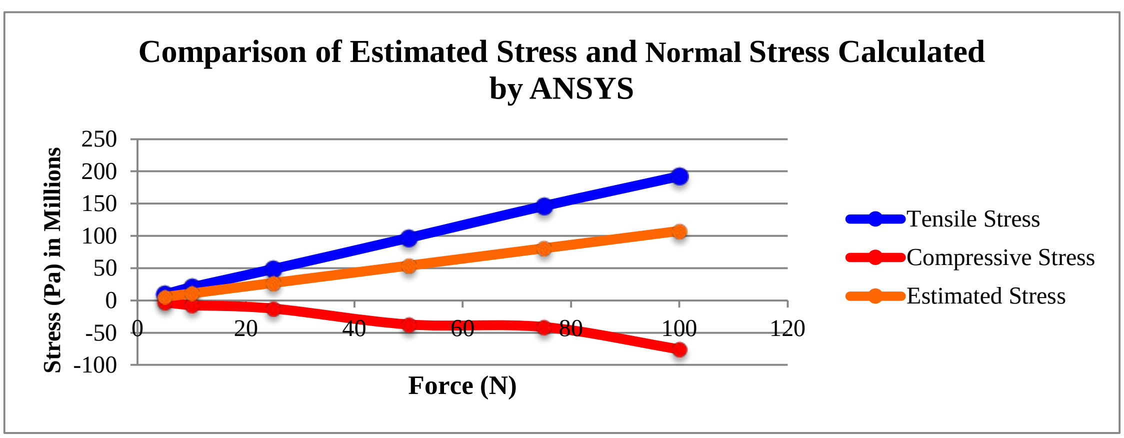


Figure 8. Total Deformation for Mesh Size 1.5× 10-4m.

According to figure 9, the smaller the force, the closer the calculated tensile relate to the estimated stress. The slope of the tensile stress is 1.78 times greater than the slope of the estimated tensile stress. When the estimated stress is negated, it also represents the estimate for compressive stress. In comparison, the slope of the compressive stress would be 0.71 times of its estimate. It may appear that stresses equal or less than 5N would be the ideal force to apply at the end of the implant. However, percent error between tensile and its estimated stress averaged 45.10% and 63.93% for compressive. The error may be contributed to the fact that the estimated stress equation does not take the irregular shape of the implant into account.

Figures 11 and 12 display the normal compressive stress and tensile stress distribution, respectively, after 5N was applied to the implant. Due to direction of force, the compressive stress points in the same direction (Figure 11).

Fig. 9. Compare Estimated Calculation of Normal Stress vs. Calculation by ANSYS Software.



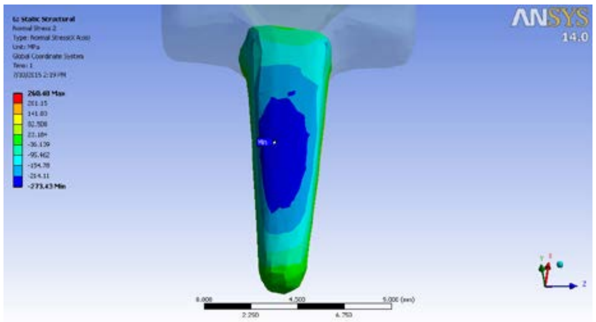


Fig. 11. Normal Compressive Stress Distribution After Application of 5N.

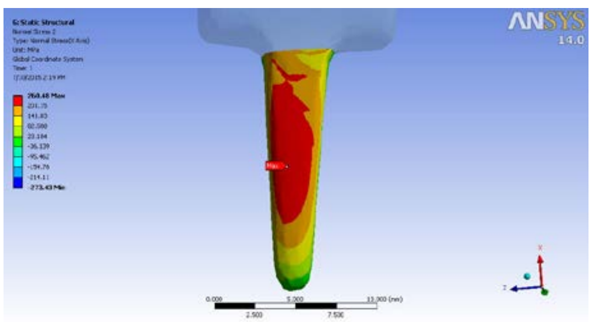


Fig. 12. Normal Tensile Stress Distribution After Application of 5N.

CONCLUSION

ANSYS allows detailed analysis of stresses, strains, and deformations felt on the implant through varying mesh size. Decreasing mesh size produced increasing number of elements, which led to more accurate finite element analysis and typically increasing stress calculation with decreasing mesh size. The graphs of maximum stress and strain versus mesh size did not follow a proper logarithmic curve, but its trend still hinted at the optimal mesh size to use to calculate normal stress. Tensile stress followed a regular slope that was consistently about 45% higher from the estimated stress at each mesh size, but compressive stress followed an irregular negative slope. This work let us obtain a mesh sensitivity analysis of the Swanson implant, also this work was useful for comparison of the finite element results with the analytical solution for the geometry. Results showed a significant difference between FEM and analytical solution. This highlights the importance of using finite element method for irregular geometries.

ACKNOWLEDGMENT

The authors of this paper would like to acknowledge the Mechanical Engineering Department at the University of Guanajuato for having the Summer Investigation program be open for internationals. Brady Enstad of MIT for finding out about the program and extending the invitation. Adrian Herrera, a Masters' student at University of Guanajuato, for connecting the authors together. Lastly, the staff at Veranos UG for making this program possible.

REFERENCES

[1] Foot Health Facts. c2015. Chicago (IL): American College of Foot and Ankle Surgeons (ACFAS); [accessed 2015 Jun 8]. http://www.foothealthfacts.org/footankleinfo/hallux-rigidus.htm
[2] FootCareMD. c2015. Rosemont (IL): American Orthopaedic Foot & Ankle Society (AOFAS); [accessed 2015 Jun 8]. https://www.aofas.org/footcaremd/conditions/ailments-of-the-big-toe/Pages/hallux-rigidus.aspx
[3] Horton, Greg A. 1999. Surgical Treatment Options in Hallux Rigidus. Operative Techniques in Sports Medicine(7): 32-35.
[4] Beeson, P. 2004. The Surgical Treatment of Hallux Limitus/Rigidus: A Critical Review of the Literature. The Foot(14): 6-22.
[5] Swanson. 2014. Swanson Flexible Hinge Toe Implant. Wright Medical Technology, Inc.: 1-16.
[6] Podnos, E., Becker, E., Klawitter, J., Strzepa, P. 2006. FEA Analysis of Silicon MCP Implant. Journal of Biomechanics(39): 1217-1226.
[7] Flavin, R., Halpin, T., O'Sullivan, R., FitzPatrick, D., Ivankovic, A., Stephens, M. M. 2008. A Finite-Element Analysis Study of the Metatarsopholangeal Joint of the Hallux Rigidus. J Bone Joint Surg [Br](90-B): 1334-40.
[8] Bocanegra, M.A. Lopez, J.B., Vidal-Lesso, A., de Bengoa Vallejo, R.B., Arroyo, R.L., Hernández, H.C. 2015. Biomechanics Aspects for Silastic Implant Arthroplasty Simulation of the First Metatarsophalangeal Joint. [Under Review.]