World Academy of Science, Engineering and Technology
International Journal of Biomedical and Biological Engineering
Vol:14, No:9, 2020

Design of Polyetheretherketone Fixation Plates for Fractured Distal Femur

Abhishek Soni, Bhagat Singh

Open Science Index, Biomedical and Biological Engineering Vol:14, No:9, 2020 publications.waset.org/10011465/pdf

***Abstract*—**In the present study, a methodology has been proposed to treat fracture in the distal part of the femur bone. Initially, bone model has been developed using the computed tomography scan data of the fractured bone. This information has been further used to create polyether ether ketone (PEEK) implant for this fractured bone. Damaged bone and implant models have been assembled. This assembled model has been further analyzed for stress distribution. Moreover, deformation developed was also measured. It has been observed that the stress and deformation developed was not so appreciable. Thus, it proves that the aforementioned procedure can be suitably adopted for the treatment of fractured distal femur bone.

***Keywords*—**Distal femur, fixation plates, PEEK, reverse engineering.

I. Introduction

THE bone in thigh region is termed as femur bone. Part of this thighbone just above the knee portion is technically termed as distal femur. Fractures in this bone are generally observed in aged persons. However, this kind of fracture can also occur in young person who have high impact injuries. Incidents such as motor vehicle accidents and falling from height results in high-energy trauma in femur bone [1]. Femur bone fractures are rare fractures, almost 0.4% of all fractures. Mostly, preferred treatment of distal femur fracture is pre-countered distal femoral locking plates [2], [3]. However, there are still some possibilities of complications with this treatment such as: malunion, non-union, leg length discrepancy, infection and other potential complications [4], [5].

In most of the cases, distal femur fracture non-union occurs after treatment [6], [7]. Thus, in order provide stable fixation, anatomic reduction must be achieved and this fixation must be able to resist high stresses across the fracture with motion, weight-bearing and muscle tone. Loosening of screws also increases the risk of unstable fixation [8].

Nowadays, technological advancement in the field of reverse engineering has motivated researchers and practitioners to utilize this concept in medical applications. There are various stages in RE such as: scanning of physical dimensions of an object, processing of scanned data, utilizing the data to create a 3D model of the object and finally this model is physically fabricated using additive manufacturing technique. Researchers have developed contact and non-contact type scanning techniques. In the field of medical applications generally non-contact type of imaging techniques are employed such as: computed tomography (CT), magnetic resonance imaging (MRI), ultrasonography (USG) etc. CT scan is generally performed for hard tissues like bone [9].

In this study, authors have used CT scan data of fractured distal femur bone to develop and analyze customized orthopedic implant. Limitation of using traditional orthopedic implant is that it requires modifications (bending and shaping) for better counter fit on the fractured/damaged region of the bone depending on the anatomy of a specific patient [10]. Aforementioned implant modification can be done only after incision and observing the fractured region, thereby increasing surgery time as well as the risk of infection. Moreover, this tailoring and alteration in the shape of implants affects its mechanical strength and ultimately results in unstable fixation [11]. These limitations have motivated the researchers to design a patient-specific counter fit implant. Proposed work is to simulate the behavior of bone and customized implant system under physiological loading conditions. Moreover, it is a non-invasive technique for analysis of stress distribution and deformation.

II. Methodology

Initially, a patient-specific CT scan data of fractured femur bone in Digital Imaging and Communications in Medicine (DICOM) file format are obtained. The data are invoked to create a 3D geometric model using RE approach in Materialise Mimics software, which is stored in the form of standard tessellation language (STL) format. Then 3D CAD Model is prepared using the STL model. Thereafter, fracture is reduced digitally and a counter fit implant is designed pertaining to the morphology of the fractured region. Then assembled 3D CAD model of the reduced fracture bone along with the implant is prepared (clinical setup). Finally, in order to ascertain the strength of the implants and intactness of the fractured bone, biomechanical analysis has been performed on the clinical setup.

A. Finite Element Model Generation

Initially, the patient's fractured femoral bone has been scanned using CT scan technique and stored in DICOM format. DICOM file format is generally used in medical imaging for storing, handling and transmitting anatomical information. Anatomical information stored in DICOM file format comprises of various images taken with reference to three mutually perpendicular planes viz. top view (axial), side view (sagittal) and front view (coronal).

Abhishek Soni is with the Jaypee University of Engineering and Technology, Guna, India (Corresponding author; e-mail: abhisonijuet@gmail.com)

Bhagat Singh is with the Jaypee University of Engineering and Technology, Guna, India.

Open Science Index, Biomedical and Biological Engineering Vol:14, No:9, 2020 publications.waset.org/10011465/pdf

The DICOM file of the fractured femur bone has been imported into medical image processing software Materialise Mimics software for converting the stacks of 2D images into a 3D model. In order to develop 3D CAD model for analysis, the 3D model created using the DICOM file has been saved in STL file format which is universally accepted and almost compatible file format for 3D printing. This STL file may contain some noise and errors like inverted normal, gaps etc. which needs to be identified and accordingly rectified before further proceedings. These identified errors have been rectified and further re-meshing of STL model has been done using MeshLab software. MeshLab is advanced mesh processing software with features of automatic as well as manual filtering, cleaning, editing, rendering and conversion of irregular meshed region. The STL file obtained from Materialise Mimics software has been loaded into MeshLab. Here, it has been checked for errors such as: duplicate facets, unreferenced vertices, null faces, small isolated pieces and non-manifold faces. These errors have been removed and small holes were filled up automatically as presented in Fig. 1.

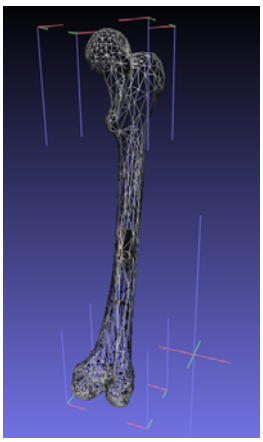


Fig. 1 Error free model of femoral shaft

After removal of errors in MeshLab software STL model has been imported into SolidWorks 2016 software. In this software 3D CAD model of the fractured femur bone has been created using the STL model as shown in Fig. 2. This 3D CAD model has been studied and further in consultation with an orthopedic surgeon customized implant has been modeled accordingly in SolidWorks 2016 software as shown in Fig. 3. This implant has been designed keeping in mind that it will keep the fractured bone intact and also provide stable support under static physiological loading conditions. Moreover, location of drilled hole in the implant for tightening the screws plays very important role. The location of holes should be neither very near nor very far from the fracture line. However, traditional implants have pre countered holes and thus positioning of implant along with the screws posed various hindrances. Moreover, traditional implants are available in standard sizes only. So, it has to be tailored before fixing it on the bone. These limitations can be overcome by designing a customized implant.

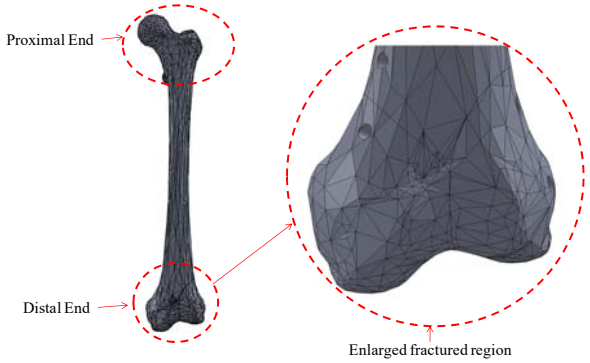


Fig. 2 3D CAD model of the fractured femur bone



Fig. 3 Customized implant designed for fractured bone

Further, implant screws are designed in the same software on the basis of bone and implant dimensions. In the present work, five implant screws are designed as shown in Fig. 4, for proper fixation on the fractured bone.

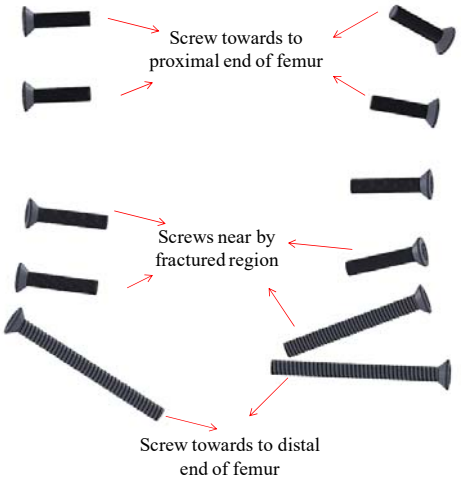


Fig. 4 Customized implant screws designed for fractured bone

Finally, clinical setup has been prepared by assembling the fractured bone along with the customized implant and screws in a proper way as shown in Fig. 5.



Fig. 5 Prepared clinical setup

B. Biomechanical Evaluation Using Finite Element Analysis

Clinical setup has been imported into ANSYS workbench software for biomechanical evaluation. In this analysis, stress variation and deformation of the fractured bone has been done considering Von–Mises and Rankine's (Maximum Principal Stress) stress criteria under static physiological loading condition. Moreover, aforementioned analysis has been done for implant of biocompatible material PEEK. The detailed analysis has been presented in the ensuing text.

Mesh Preparation: Meshing is an operation to divide the region of analysis into small size elements for evaluation. In the present work, mesh model for the fractured femur bone has been created using model wizard in ANSYS Workbench 14. Element considered for meshing purpose was tetrahedral. The number of tetrahedral elements and nodes used for the fractured distal femur model were 17,913 and 32,654 respectively as shown in Fig. 6.

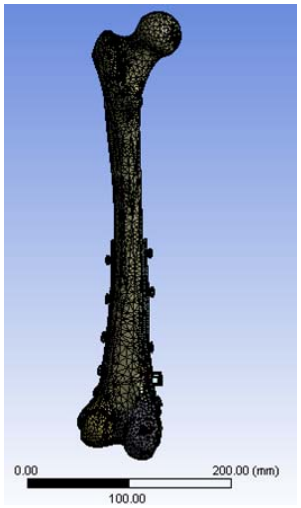


Fig. 6 Meshed clinical setup model

Boundary Conditions: Boundary conditions considered for finite element analysis of clinical setup are as follows:

a. Distal end of the femur has been fixed considering the human bone to be inflexible
b. A fixed loading condition has been applied on the proximal end of the femur.

Loading Conditions: In the present work, static load analysis has been done. It has been assumed that a person is standing straight and weight of the person is 1000N. This load is transmitted equally the through pelvic bone at the head of femur. Thus it has been assumed that 500 N load is acting vertically downward on each femur neck of the person. Biomechanical evaluation of clinical setup has been performed for biocompatible material PEEK. Material properties of this material have been mentioned in Table I.

TABLE I
IMPLANT MATERIAL PROPERTIES

Material	Young Modulus (GPa)	Ultimate Strength (MPa)	Poisson's Ratio	Density (g/cm^3)
PEEK	3.2	78.86	0.42	1.291

III. RESULTS AND DISCUSSIONS

A. Stress Analysis

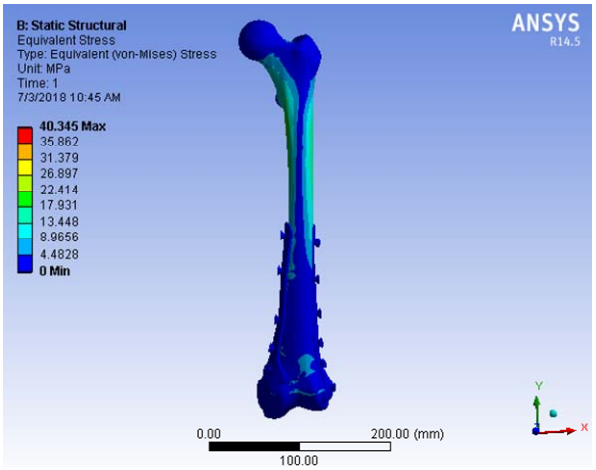


Fig. 7 (a) Von-Mises stress distribution in clinical setup

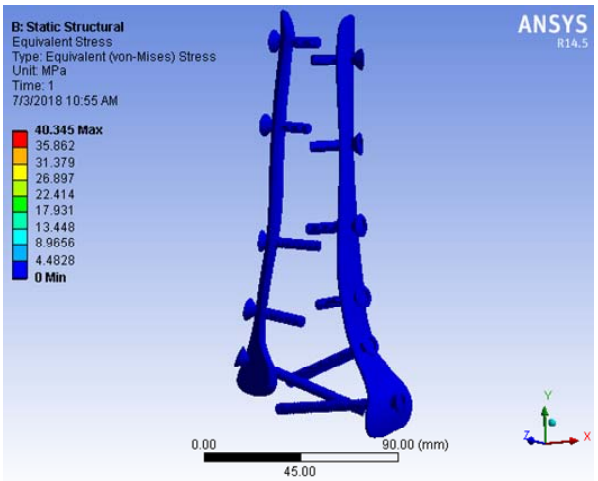


Fig. 7 (b) Von-Mises stress distribution in implants and screws

Open Science Index, Biomedical and Biological Engineering Vol:14, No:9, 2020 publications.waset.org/10011465/pdf

Open Science Index, Biomedical and Biological Engineering Vol:14, No:9, 2020 publications.waset.org/10011465/pdf

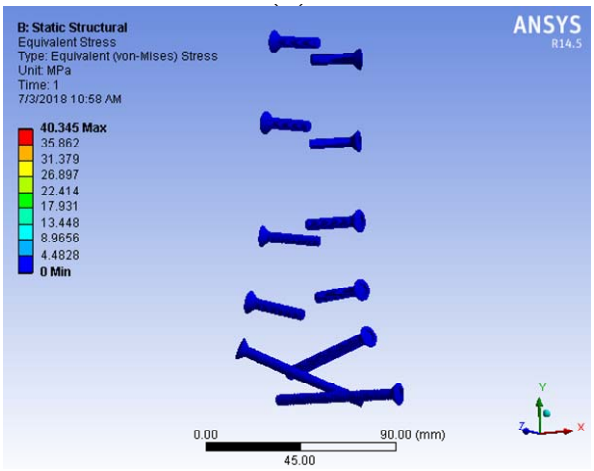


Fig. 7 (c) Von-Mises stress distribution in all screws

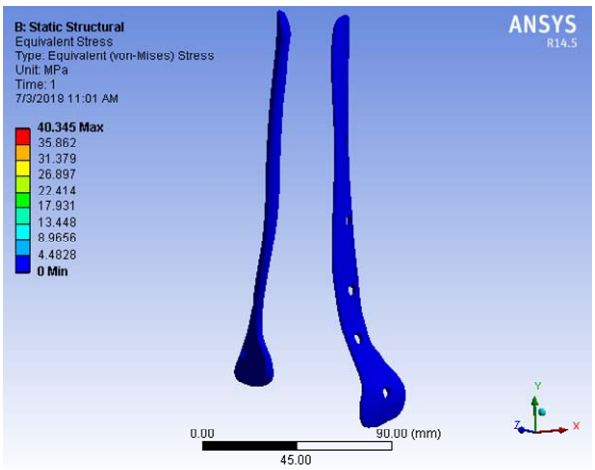


Fig. 7 (d) Von-Mises stress distribution in implants

Stress analysis has been done considering Von–Mises and Rankine's (Maximum Principal Stress) stress criteria. It has been observed that Von–Mises stress of magnitude 4.48 MPa is developed as shown in Fig. 7. However, on analyzing the stresses developed in the implant and screws, it has been noted that stress is raised up to 22.41 MPa. This stress was maximum in the screw thread of the nearest screw from the fractured region towards the distal side. This stress is higher in this region because of stress concentration. Moreover, this stress is also developed because of contact pressure between the bone and the screw threads. Similarly, stress distribution has been analyzed considering maximum principal stress criterion for PEEK as shown in Fig. 8. Overall stress distribution in the clinical setup was uniform and about 2.53 MPa. However, stress developed in the screw near the vicinity of fractured region was higher and observed to be 18.72 MPa.

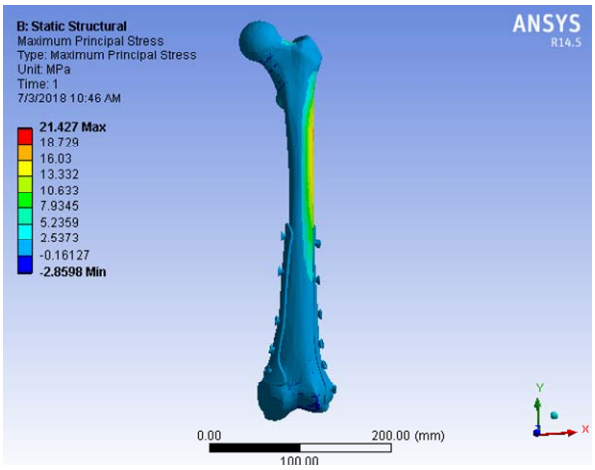


Fig. 8 (a) Maximum principal stress in clinical setup

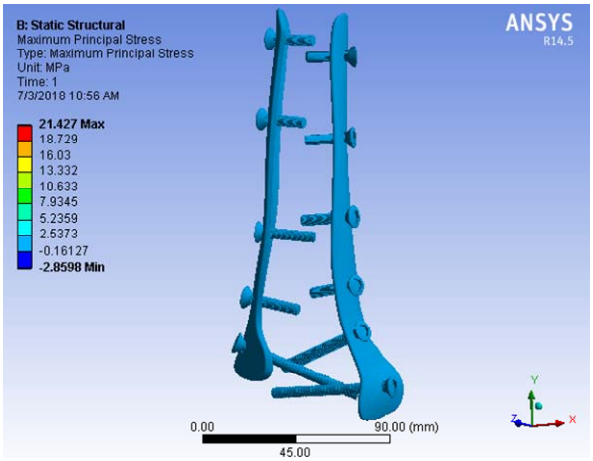


Fig. 8 (b) Maximum principal stress in implants and screws

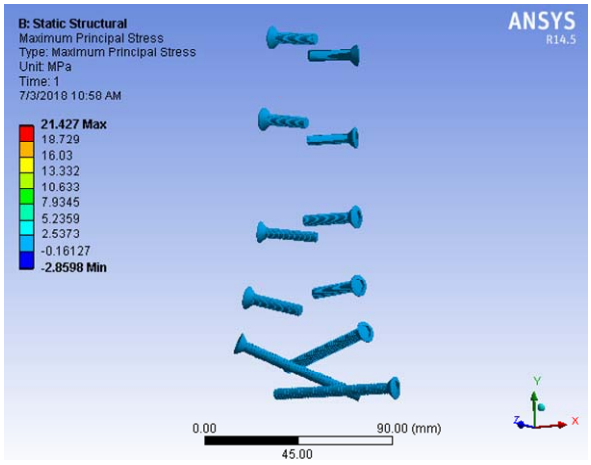


Fig. 8 (c) Maximum principal stress in all screws

Open Science Index, Biomedical and Biological Engineering Vol:14, No:9, 2020 publications.waset.org/10011465/pdf

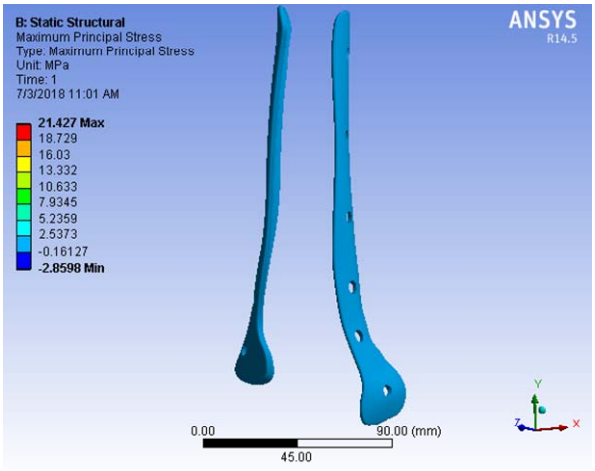


Fig. 8 (d) Maximum principal stress in implants

B. Deformation Analysis

Total deformation has been analyzed for biocompatible material PEEK. The variation of deformation on the clinical setup has been presented in Fig. 9. From this figure it is evident that the deformation is higher near the proximal end, where load is applied. Moreover, the proximal region is a spongy bone and so deformation is higher as compared to the rest portion which is comparatively more rigid. However, the maximum deformation observed in PEEK, was 0.22 mm. It is observed that the deformation is within the acceptable range. So, the customized design of the implant for this type of fracture in femur bone is quite apt.

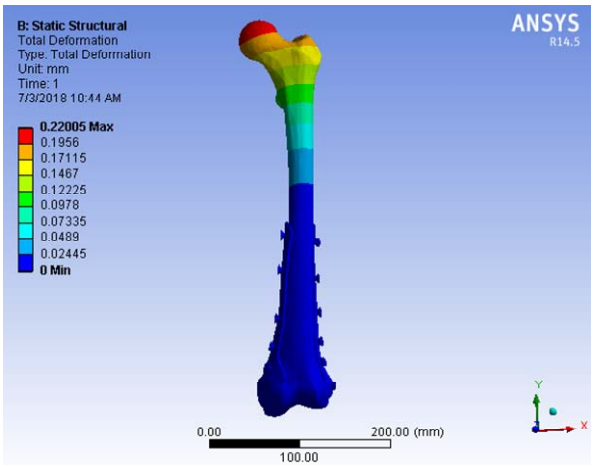


Fig. 9 (a) Total deformation variation in clinical setup

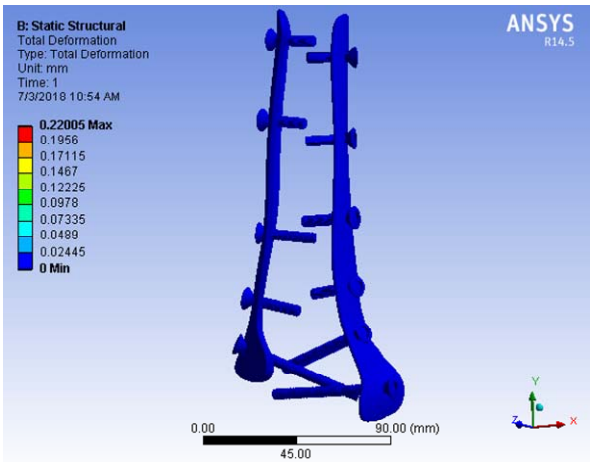


Fig. 9 (b) Total deformation variation in implant and screw

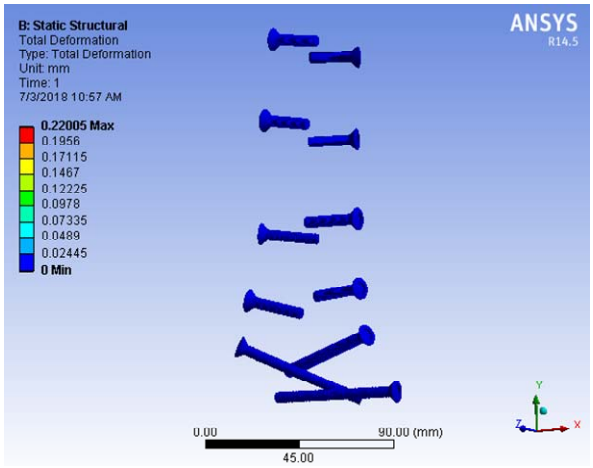


Fig. 9 (c) Total deformation variation in all screw

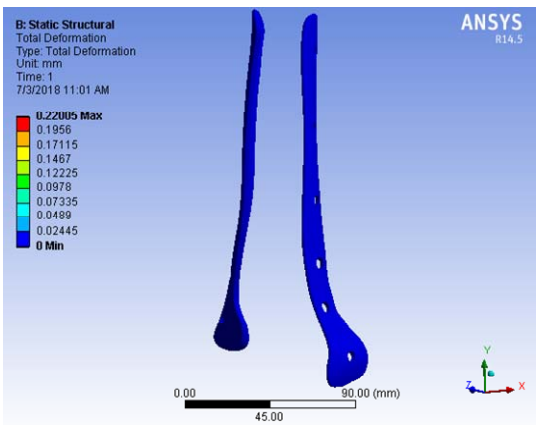


Fig. 9 (d) Total deformation variation in implants

IV. Conclusions

In the present work, a methodology has been suggested to create and analyze a clinical setup of a fractured femur bone considering CT image data. Moreover, in order to provide better fitting and overcome the stress shielding at the fractured region, a customized counter fit fixation plate has been created for the aforementioned clinical setup. PEEK has been

considered for the analysis. Further, static stress distribution and deformation analysis of the clinical setup has been performed for the aforementioned material. After analysis, following conclusions have been drawn:

- Stresses and stress shielding developed are found to be appreciably low as compared to its ultimate strength. Deformations are considerable in the aforementioned material. It shows that mechanical properties of implant are satisfactory for stable fixation.
- Created clinical setup can be helpful for pre-operative and intra-operative planning which reduces the surgery time as well as helps in understanding the nature and severity of fracture.
- Proper fixation of designed customized implant will help in reducing the post-surgical failures and residual pain.

Proposed work will serve as a guideline for the medical practitioners to design and analyze suitable implants for the respective fractured bone. Moreover, it will enable the surgeons to select the most suitable material for these implants.

Compliance with Ethical Standards

Conflict of interest: The authors declare that they have no conflict of interest.

Ethical standard statement: This article does not contain any studies with human or animal subjects performed by any of the authors.

Informed consent: For this type of study informed consent is not required.

References

[1] OTS Canadian, "Nonunion following intramedullary nailing of the femur with and without reaming. Results of a multicenter randomized clinical trial," *The Journal of Bone and Joint Surgery American Volume*, vol. 85, pp. 2093-2096, 2003.

[2] M. J. Beltran, J. L. Gary and C. A. Collinge, "Management of distal femur fractures with modern plates and nails: state of the art," Journal of Orthopaedic Trauma, vol. 29, pp. 165-172, 2015.

[3] A. K. Gangavalli, C. O. Nwachuku, "Management of distal femur fractures in adults: an overview of options," *Orthopedic Clinic*s, vol. 47, pp. 85-96, 2016.

[4] R. Reina, F. E. Vilella, N. Ramírez, R. Valenzuela, G. Nieves and C. A. Foy, "Knee pain and leg-length discrepancy after retrograde femoral nailing," *American Journal of Orthopedics*, vol. 36, pp. 325, 2007.

[5] M. Malik, P. Harwood, P. Diggle and S. Khan, "Factors affecting rates of infection and nonunion in intramedullary nailing," *The Journal of Bone and Joint Surgery British Volume*, vol. 86, pp. 556-560, 2004.

[6] T. J. Lujan, C .E. Henderson, S. M. Madey, D. C. Fitzpatrick, J. L. Marsh and M. Bottlang, "Locked plating of distal femur fractures leads to inconsistent and asymmetric callus formation," *Journal of Orthopaedic Trauma*, vol. 24, pp. 156-162, 2010.

[7] W. M. Ricci, P. N. Streubel, S. Morshed, C. A. Collinge, S. E. Nork and M. J. Gardner, "Risk factors for failure of locked plate fixation of distal femur fractures: an analysis of 335 cases," *Journal of Orthopaedic Trauma*, vol. 28, pp. 83-89, 2014.

[8] A. Gefen, ":Optimizing the biomechanical compatibility of orthopedic screws for bone fracture fixation," *Medical Engineering & Physics*, vol. 24, pp. 337-347, 2002.

[9] Y. P. Lin, C. T. Wang and K. R. Dai, "Reverse engineering in CAD model reconstruction of customized artificial joint," *Medical Engineering & Physics*, vo. 27, pp. 189-193, 2005.

[10] N. Narra, J. Valášek, M. Hannula, P. Marcián, G. K. Sándor, J. Hyttinen and J. Wolff, "Finite element analysis of customized reconstruction plates for mandibular continuity defect therapy," *Journal of Biomechanics*, vol. 47, pp. 264-268, 2015.

[11] D. S. Shin, K. Lee and D. Kim, "Biomechanical study of lumbar spine with dynamic stabilization device using finite element method," *Computer-Aided Design*, vol. 39, pp. 559-567, 2007.

Abhishek Soni is a Ph.D. student at Jaypee University of Engineering and Technology, Guna (India). He has more than 5 years of teaching experience. He is pursuing his research under the guidance of Dr. Bhagat Singh in the area of "Reverse engineering approach for designing and fabrication of customized orthopedic implants. He has published three papers in peer reviewed international journals.

Dr. Bhagat Singh is currently working as an Assistant Professor (Senior Grade) at Jaypee University of Engineering and Technology. He has done his graduation in Mechanical Engineering from NIT, Kurukshetra (India). He obtained his Master of Engineering from NIT, Rourkela (India) in Mechanical Engineering. His specialization in M. Tech was Machine Design and Vibration Analysis. He was awarded PhD degree from NIT, Rourkela. The title of his Ph.D. dissertation was "Study of damping in layered and welded beams." Dr. Singh has 20 years of teaching and research experience. He has published 65 papers in peer reviewed international journals and conference proceedings.

Open Science Index, Biomedical and Biological Engineering Vol:14, No:9, 2020 publications.waset.org/10011465/pdf