Topology Optimization Of Femoral Shaft Implant: A Review

S. Kirthana[1]*, Dr. Sriram Venkatesh[2], Dr. T. Ramamohan Rao[3]

[1]*Phd Scholar, Department of Mechanical Engineering, University College of Engineering (A), Osmania University, Hyderabad.
[2]Senior Professor, Department of Mechanical Engineering, University College of Engineering (A), Osmania University, Hyderabad.
[3]Professor and HOD, Department of Mechanical Engineering, Vasavi College of Engineering, Hyderabad.

***Corresponding Author:** S. Kirthana
*****Phd Scholar, Department of Mechanical Engineering, University College of Engineering (A), Osmania University, Hyderabad, Email - kirthana6831@gmail.com

Abstract:
In last few decades, increase in the life expectancy has increased the number of surgical procedures involving the prosthesis implantations. As the human body ages, the load bearing capacity decreases. To meet the requirements of using implants, best material and optimized design model of implants are to be maintained regarding the manufacturability of implants. To reduce the manufacturability time, Additive manufacturing processes were introduced with the best finish and dimensional accuracy. Topology Optimization play a major role in obtaining the best possible optimized design model with the existing boundary conditions while maintaining necessary strength and rigidity of the implant.

The current paper presents a review of various journal papers on topology optimization and analysis on femoral shaft implants, analysis procedures followed, and loading conditions involved for various situations.

Keywords: Topology Optimization, femoral shaft implants

1. Introduction:

Femur bone is the longest and the strongest bone which almost covers 25% of the total height of a particular person. Fractures in the femur bone are caused by large force, impact loads, accidents, falls from high altitudes, and disease in the bone. Shaft fractures are classified into transverse, oblique, spiral, comminuted and open fractures. Out of the various types of implant plates that are available for the treatment of femoral shaft fractures, the most used one is a Limited contact dynamic compression plate (LC-DCP) as it can be conveniently used for both conventional screws and stable angular screws. Available materials for the femur implant plates are SS316L and Ti–6Al–4V.

Stress shielding in bone is a common problem induced by mild- to high-load-bearing medical implants and can be reduced by redesigning the medical implant. Topology optimization is gaining significant attention due to the ability to automatically generate optimal redesigns for a given design, considering different loading conditions and volume reduction constraints. Several authors demonstrated the feasibility of topology optimization for the redesign of orthopaedic medical implants to minimize stress shielding. In these cases, results showed improved load transfer to the bone in the case of optimized implants.

2. Modeling of bone, implant and screws:

Reconstruction of bone is a very difficult task as femur bone is not just a cylinder or any other geometrical surface. It comprises many complex surfaces. One of the best ways of creating a femur bone model is by using DICOM images obtained from a Computed Tomography scan (CT scan). The detailed procedure followed in creating the cad model of the femur bone is mentioned in the paper by Mohammed Khaja Nizamuddin et. Al. [5] .

CAD models of commercially available implant and screw models can be obtained either by directly modelling them in CAD software (in case the detailed dimensions are available) or by applying reverse engineering techniques using a 3D scanner. Assembling of bone, implant and screws can be done in any commercially available CAD software.

3. Load and boundary conditions:

Many authors have assumed different ways of loading conditions for the assembly. Abdulsalam Abdulaziz Al-Tamimi et. Al.[1] assumed that no fracture gap was imposed, and the plate was assumed to fixate a transverse fractured tibia bone. Only the cortical bone region was considered, and a hollow cylinder region with an external diameter of 24 mm and an internal diameter of 12 mm was assumed for simulation purposes. Loading and boundary conditions considered were as shown in figure1. Two equally distributed moments of 20 Nm were applied along the horizontal axis of the bone, simulating the moment load happening on the tibia during the swing phase (i.e. 10% of the body weight) in patients walking with crutches [6]. To prevent rigid body motion, the extremity faces of the bone were fully constrained.

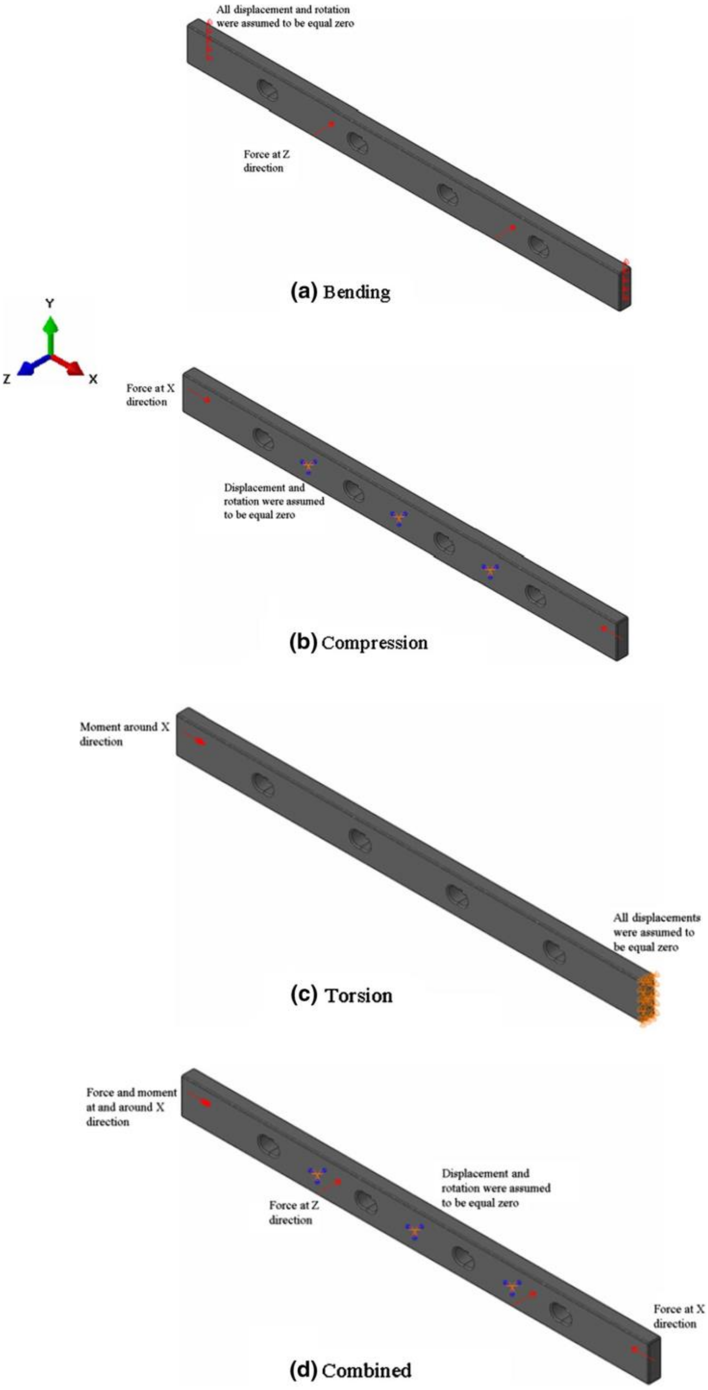

Fig1: Load and boundary conditions considered for the fixation plate optimisation of **a** four-point bending load, **b** uniaxial compression, **c** torsional and **d** combination of the bending, compression and torsion loads

Dan Leordean et. al [3] assumed that foot implant must withstand up to 1.5 times the weight of the body to sustain in walking time [7] and up to 8 times in running. Considering this, implant analysis will be done by loading it at 4000 N, which means 8 × 50 kg (patient's weight). 4000N is selected to observe the behavior of the femoral plate during a limiting event. To meet all requirements, the femoral plate must withstand all normal and unfavorable conditions [8], and to illustrate the results of the analysis, four studies with different parameters were performed.

Rana Idan Abed et. Al [2] have performed static structural analysis, in which, a fixed support is defined at the lower surface of the femur bone as shown in Fig.2 [4]. A compression force of 750N is marked at the femoral head surfaces based on the presumption that the person weight of 75 kg as shown in Fig.3.

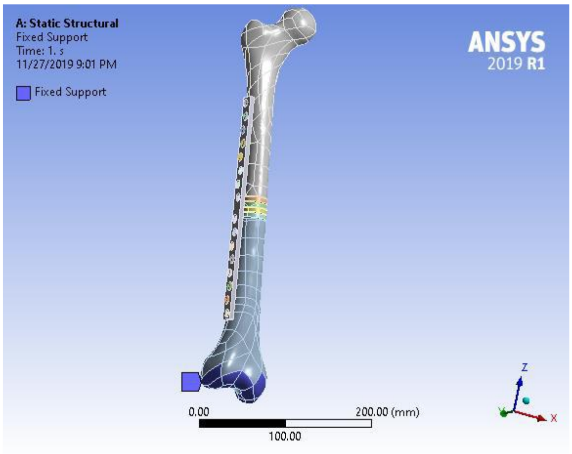

Fig 2 : Fixed support at femur lower end

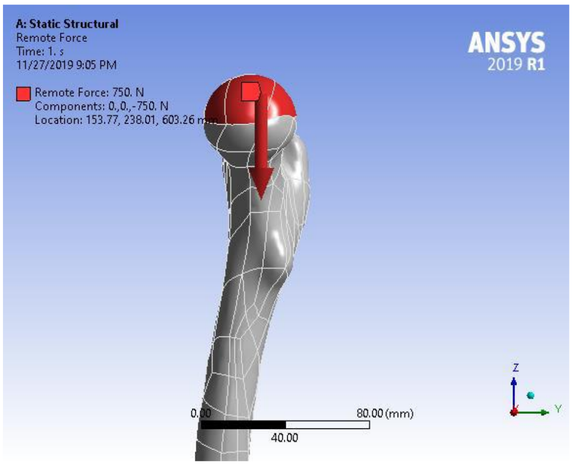

Fig 3: Load applied on femur head

Pravat Kumar Satapathy. Et. Al [4] applied both the compressive load of 700 N as in fig 4 and a torsional load of 50 N.m as in fig 5 are applied at the femoral head. The loading condition is based on the assumption of a weight of 70 kg person. A fixed boundary condition is assigned at the lower surface (lateral condyle, medial condyle and patellar surface) of the femur bone. Bonded contact is considered at the bone-to-bone interface. As the contact stress is generated at the interface between the bone plate and bone surface, frictional contact elements are used with a frictional coefficient of (μ=0.2) [9].

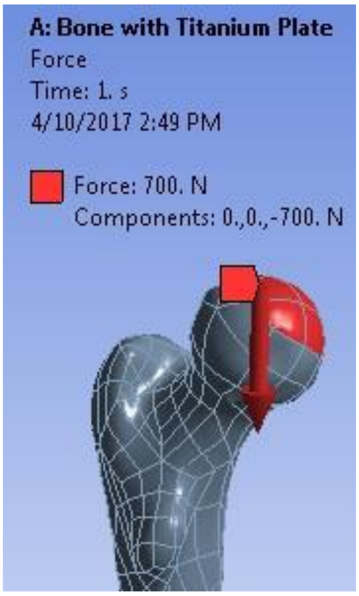

Fig 4 : Compression load of 700N on femur head

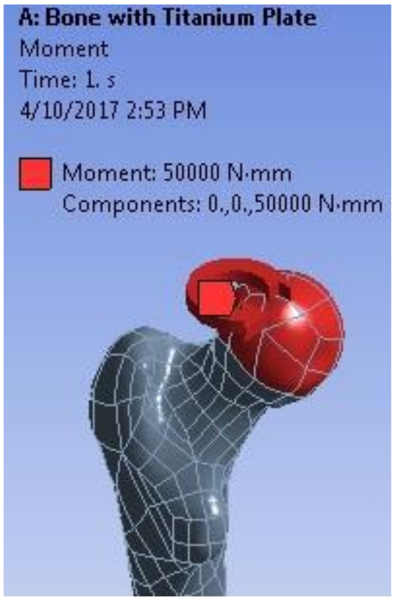

Fig5: Torsional load of 50 N.m on femur head

4. Stress analysis results of the assembly:

Abdulsalam Abdulaziz Al-Tamimi et. Al.[1] say that equivalent bending stiffness change increases as the volume reduction increases. For the same volume reduction and plates with different numbers of holes, there is no clear trend in terms of the equivalent stiffness change. Plates optimized considering bending loading conditions exhibit the least reduced equivalent stiffness change. The highest decreases were observed for the 75% volume reduction with combined load for the four-hole and eight-hole. the stresses in the bone increase when topology-optimized plates are used. The most substantial increase in stresses at the fracture plane was observed for the combined loading conditions and 75% volume reduction plates.

Dan Leordean et. al [3] identified overloaded areas , changed the geometry and material to improve the requirements that an implanted medical device must meet. The implant redesigning has many advantages. One of them is that working time decreases due to decreased processing volume. The complexity of geometry is not a disadvantage because the Additive Manufacturing processes can perform any kind of geometry. The final model (fig. 6, d) of the femoral implant has been chosen for an abnormal case that illustrates that the implant can fulfill its purpose even in unusual situations [10].

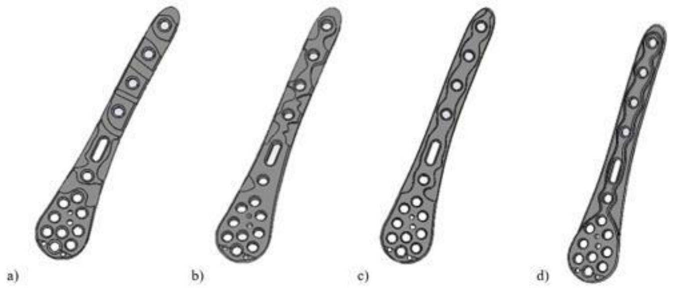

Fig 6: Different models for redesign

Pravat Kumar Satapathy. Et. Al [4] observed that the deformation of the bone plate system decreases significantly while using FG material as the bone plate. The least deformation is obtained when the power law exponent (k=0.2) is used as FG material. One important aspect of the present study is to investigate the stress distribution at the fracture site. It may be observed that the Von-Mises stresses increase when using the FG bone plate, in comparison with the titanium bone plate and again it is maximum at k=0.2. It could be established here that the stress at the fracture site increases when torsional loading condition is taken into consideration.

5. Conclusions:

Topology optimization permits the design of less stiff and lightweight fixation plates, reducing the stress shielding effect, promoting load transfer to the bone, and thus contributing to bone remodelling. Another advantage that comes with TO plate design is the distribution of equivalent stress. Due to the redesign and modification of the material, the implant has a higher resistance, and the stress graph indicates that the plate has a higher limit. The maximum stress increased with increasing the number of screws and the total deformation decreased by increasing the number of screws as exponential equation curves.

6. Scope for further study:

Screw threads were not considered in the simulation and their role in load transfer must be also considered. Although no critical failure was observed in the optimized designs, thin and sharp features have to be observed. Further analysis is still required, considering for example a fracture gap and measuring the gap strains to correlate the resulted strains (i.e. relative or absolute stability) with the healing process (i.e. secondary or primary healing).

References:

1. Abdulsalam Abdulaziz Al-Tamimi, Carlos Quental, Joao Folgado , Chris Peach, · Paulo Bartolo (2019), Stress analysis in a bone fracture fixed with topology-optimised plates, Journal of Biomechanics and Modeling in Mechanobiology, https://doi.org/10.1007/s10237-019-01240-3.
2. Rana Idan Abed, Sadiq Jaafer, Waleed A Alsaadan (2020), Mechanical Analysis of Bone-Plate Construct Regarding Strength and Stiffness, Al-Nahrain Journal for Engineering Sciences NJES 23(1)89-93, 2020. http://doi.org/10.29194/NJES.23010089.
3. Dan Leordean, Tatiana Ciobanu, Mircea Rusu (2019), Researches on the design of customized femoral implant , MATEC Web of Conferences 299, 03009 (2019) https://doi.org/10.1051/matecconf/201929903009 .
4. Pravat Kumar Satapathy, Bamadev Sahoo, Panda L N, Das S (2018) , Finite element analysis of functionally graded bone plate at femur bone fracture site, IOP Conf. Series: Materials Science and Engineering 330 (2018) 012027, doi:10.1088/1757-899X/330/1/012027.
5. Mohammed Khaja Nizamuddin, S. Kirthana (2018), Reconstruction of human femur bone from CT scan images using CAD technique IOP Conf. Series: Materials Science and Engineering 455 (2018) 012103, doi:10.1088/1757-899X/455/1/012103.

6. Ramakrishna K, Sridhar I, Sivashakner S, Khong K, Ghista D (2004) ,Design of fracture fixation plate for necessary and sufficient bone stress shielding. JSME Int J Ser C 47:1086–1094. https ://doi.org/10.1299/jsmec .47.1086.
7. G Bergmann , G Deuretzbacher, M Heller, F Graichen, A Rohlmann, J Strauss, G N Duda, Hip contact forces and gait patterns from routine activities, Journal of Biomechanics, 2001 Jul;34(7):859-71, doi: 10.1016/s0021-9290(01)00040-9.
8. Kandemir, Utku (2018). Distal femur: dynamization of plating. Injury, 49(), S44–S48. doi:10.1016/S0020-1383(18)30302-4.
9. Das S and Sarangi S K (2014), Finite element analysis of femur fracture fixation plates Int. J. of Basic and Appl. Biology ISSN: 2349-5839 Volume 1 No. 1.
10. Hoskins, Wayne; Bingham, Roger; Griffin, Xavier L. (2017). Distal femur fractures in adults. Orthopaedics and Trauma, 31(2), 93–101. doi:10.1016/j.mporth.2016.11.002.