Simulation of Implants and Fixators using Finite Element Method

D. Bubesh kumar, Aravindan Varadan, Praveen kumar, Hariharan, Rajaram

Abstract*: *Finite Element Analysis (FEA) of implants and fixators were carried out in this paper. Various implants and fixators were carried these fixators were used for various fractures occurring in the human bone. The implants and fixators were modeled and analysed using FEA software called ANYSWorkBench. These results were analysed, it is found titanium implants are more suitable for implants and fixators due to its rigidity and strength and young's modulus very near tom the young's modulus of the bone.

Keywords*: *FEM, WorkBench, Fixators, FEA

I. INTRODUCTION

Restricted component examination of Implants as well as Fixators restricted component process is the basis used for the FEA [1]. The implants and Fixators are Modeled and Analyzed by the ANSYS Work Bench software package [2-3].

A. FEA of HIP Implant

Restricted component study of artificial HIP joint be modeled by ANSYS Work Bench Software Package [4]. The load is applied on the ball and socket joint of artificial HIP joint. Various parameters of the femur are analyzed by the Finite Element Analysis [5-7].

B. Sample

Cadaveric Human femur bone, [8-9] The Poisson ratio of each material is given in the following table 1.

Revised Manuscript Received on January 15, 2020

* Correspondence Author

Dr.D.Bubesh Kumar, is working as Associate Professor in the Department of Mechanical Engineering. He has published more than 15 research papers, He has written a book in the area of Fatigue corrosion of Biomaterials He has completed his Ph.D in the area of Biomedical Implants and fixators.

Aravindan Varadan*, Assistant Professor Mechanical Department, Aarupadai Veedu Institute of Technology, Vinayaka Mission's Research Foundation, Deemed To Be University. aravindan.mech@avit.ac.in

Praveen Kumar, UG Student , Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation

Hariharan, UG Student , Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation

Rajaram , UG Student , Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation

Table: 1 Material Properties of metals used in FEA

Material	Young's modulus	Poisson Ratio
Titanium	110	0.31
SS	210	0.3
Magnesium	45	0.35
Aluminium	70	0.33

The above table: 1 is used as medical equipment for the (Femoral Stem) Implant. All the equipment is implicit to be Homogeneous and linear elastic Tool ANSYS Package Software for modeling and Analysis.

II. METHODS AND MATERIALS

A. Steps for using tools to analyze

A femur is used to representation designed for the mean of fake HIP joint stem. A FEA is old to predict pressure, strains, deformation generated in the femur head and neck using FEA stress, strains of femur. To prepare an FEA representation, the head of the femur is a fairly curved hyaline cartilage enclosed outside that is a little better than a accurate hemisphere. The head of the femur is measured to be round. The femoral head is permanent to the neck.

As well as the femoral neck is permanent to the shaft of the femur among the better trochanter as well as the lesser trochanter. The femoral neck is angulated. So that the femoral head is the majority frequently faces medially, superior as well as anteriorly. Angle of inclination of the femur is 1250 in the normal adult. The 3D model of the HIP is constructed using ANSYS Work Bench Software. Setting FEA conditions, Implant materials such as Titanium, stainless steel as well as Magnesium is modeled. An examination model is prepared using a restricted component pre-processor (meshing). The tetrahedral main component with dimension of 500µm is used 49170 numbers of basics and 592088 numbers of nodes. The loading situation be based on the process of the load on the femur is three times more than the body weight. The FEA is performed using nonlinear software. The computation is performed in an elastic section. The stress, strain, deformations are also performed and analyzed.

FEA of fixator is modelled by ANSYS WorkBench Software Package. The load is applied on the proximal end of the fixator. Various parameters of the fixator are analysed by the Finite Element Analysis.

B. FEA Fixator Samples

Illizarov fixator, cadaveric tibia model, Poisson ratio are 20 GPa, 0.27.

C. FEA Fixator Tools

ANSYS Package Software for modeling and Analysis. Steps for using tools to analyze the Illizarov fixator consists of threaded rods, fixed on the Illizarov rings, k-wires transfix the bone and the k-wires are fixed on the ring. The rings are placed on the proximal and distal end of the bone. A fracture gap of 3mm is artificial created by the surgeon. When load is applied of Illizarov apparatus, micromotion takes place at the fracture site results in bone growth at the fracture site. FEM was used to model the Illizarov fixator. This will help the surgeon to understand the behavior of the fixator before it is fixed on the human body. Design, Analysis of HIP Implants and Fixator.

III. EXPERIMENTS AND RESULTS

Biomedical implants and fixators used in orthopedic implants and fixators were analyzed by the researcher. He used WorkBench software package to model and analyze the fixator system. Three Dimensional Model of Stainless steel HIP implant 3D solid analysis model is old for broad arrangement during 3D space so as to include neither a stable annoyed segment nor an axis of regularity creating a 3D solid analysis model by direction production methods which usually requires significant effort solid modeling which for all time makes the occupation simple.

FEA of the automatic properties of illizarov, hybrid1 as well as hybrid 2. The substance properties for all the components of fixators are unspecified to be elastic, isotropic, continuous, as well as uniform. The substance specification of 316L SS young's modulus of elasticity is 210 GP a, thickness 7800kg/m3, final pressure is 600MPa, give up Strength is 200MPa. Substance individuality of the packed in bone were: modules of regular elasticity is16.2Mpa; thickness 2000kg/m3 final strength 10MPa. To make simpler calculations, the Poisson's ratio used for all workings is taken to be 0.3. A three dimensional representation of the human tibia is shaped with ANSYS Work Bench 10 a gap of 20mm was generated in the center of the realistic representation to simulate a bone break of the tibial bone.

A. Coordinate system and loading

The coordinate system X-Y-Z of this hip Implant is based on the following definitions. The centre of the ball is chosen as the origin and the plane of the paper is chosen as the X-Y Plane with x-axis in the horizontal direction as well as y-axis in the vertical direction as well as the direction perpendicular to the plane of the paper is taken as Z Axis.

The loads on the hip joint include the resultant joint force which acts on the implant. The force is taken as positive if acting in the upward direction which includes compressive forces. A preselective load has been applied on the femoral head of the implant.

The plenty are practical in steps from 100N to 1800N which is three times the average body mass of the human body.

B. Implant Dimensions

The cross section of solid sphere head with diameter of 50mm and neck length is 6 mm with cylindrical cross section and the length of the stem is 135 mm with cross section as cylindrical.

C. Meshing

Meshing the model is the first step for FE analysis after the model has been generated. In order to analyse the model, it should be meshed with suitable element and element is the key factor. A proper selection of the element gives the result to the required accuracy. The model has been generated using ANSYS Work Bench 15 has been meshed using solid 72 tetrahedral element which is four node component with six degrees of autonomy.

(Translational in X, Y, as well as Z directions as well as rotations in X, Y as well as Z directions). The model that has been produced as well as meshed into 26,202 basics having 5430 nodes using solid 72 tetrahedral fundamentals. The element solid 92 also has pressure stiffening potential and 4 Node tetrahedral fundamentals without turning degrees of liberty solid 92. The solid 72 has been used in place of solid 92 fundamentals to reduce wave front as well as solution time. even though the component has extra degrees of liberty for node, it is not as accurate as the solid 92 fundamentals. Moreover the nodes of component, input data includes the Orthopaedic fabric properties, orthotropic substance technique is matching to the component organize directions.

D. Assumptions and restrictions

Solid 72 uses mixed method with steady cut off strains consequential in non border difference of the component hardness medium.

The component should have zero volume. Care must be taken while applying strength loads as well as dislocation constraints to the solid component with turning degrees of liberty, where appropriate 'h' method of convergence has been chosen for the analysis. The material is assumed as isotropic, no real constants have been chosen and the element dimensions are automatically done by ANSYS Work Bench 12. Static loading has been done and Influence of fatigue loading is taken into consideration.

Boundary conditions were applied on the model for analysis. Set of boundary conditions is chosen for analysis of the 3D model that was being generated. Applied force loads on a component face are mechanically changed to the corresponding force as well as second lots. Owing to the hypothetical restriction applying pressure loads to solid 72 fundamentals is avoided. For condensed examination turning degrees of autonomy, it be supposed to not be chosen as master degrees of liberty. Solid 72 basics though have advantage of fewer degrees of liberty; a minor signal front as well as smaller file size makes solid 72 tetrahedral the mass sensible for initial examination of models that are simply meshed with tetrahedrons.

IV. RESULT AND DISCUSSION

The performance of total deformations, titanium hip Implant at 1800N Load is shown in Figure 1.

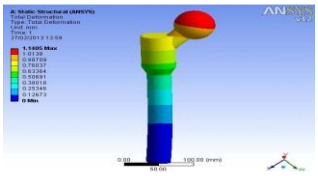

Fig. 1.Total Deformations, Titanium Hip Implant at 1800N Load

The table II below shows the mechanical parameters of the hip joint. The joint should not be more flexible or very stiff, Micro deformation of hip joint is needed that, with strains equivalent to the strains produced by the bone when load is applied due to normal activities. The maximum load applied to the Implant is taken as 1800N (equivalent to 3 times the body weight of the average human body). Steel Implants is structurally suited than other metals but the corrosion resistance of 316LSS is less than the titanium alloy and Aluminium alloy.

Table: II FEA of parameters of Hip Implants

Parameters	Magnesium	Titanium	316LSS
1.Directional Deformation	Very high	Very high	Very less
2. Elastic Strain Intensity	Very high	Very high	Some deviation
3.Equivalent Elastic Strain	Very high	Very high	Some deviation
4.Equivalent Stress	Similar	Similar	Similar
5.Maximum Elastic Strain	Very high	Very high	Very small deviation
6.Maximum Principle Stress	Less	Negative	More
7.Maximum Principle Elastic strain	More	More	Some deviation
8.Maximum Shear Elastic Strain	More	More	Some deviation
9.Maximum shear stress	Similar	Similar	Similar
10.Middle Principle Elastic Strain	Similar	Similar	Some deviation
11. Normal Elastic Strain	Similar	Similar	Small deviation
12.Normal Stress	More	Small	More
13.Shear Elastic Strain	Less	More	more
14. Shear Stress,	Similar	Similar	similar
15.Stress Intensity	Similar	Similar	similar
16. Structural Error	high change	high change	No change
17.Total Deformation	Very high	Medium	Small deformation

The various mechanical parameters are analysed. The total deformation is chosen as the important criteria because the deformation should not be large as well as small. Total deformation is medium for Titanium HIP Implants. Illizarov fixator Analysis results are shown in Table 3. The performance of Total deformation, Titanium fixator and steel K-Wire is shown in Figure 2.

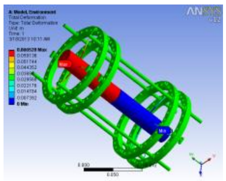

Fig. 2.Total deformation, Titanium fixator and steel K-Wire

Table: III Illizarov Fixator various Mechanical parameters

Parameters	Aluminum fixator and steelK-wire	Steel fixator and steel K- Wire	Titanium fixator and steel K-Wire
1.Diretional Deformation	0.00079802	0.00067593	0.00076193
2.Elastic Strain Intensity	0.43402	0.3623	0.38405
3.Equivalent Elastic Strain	0.27902	0.233	0.24153
4.Equivalent Stress	1.43E+10	1.23E+10	1.37E+10

5.Minimum Principle Elastic Strain	3.07E-05	1.50E-05	2.22E-05
6.Maximum Principle Stress	1.35E+10	1.14E+10	1.29E+10
7.Maximum Principle Elastic strain	0.25002	0.2209	0.2505
8.Maximum Shear Elastic Strain	0.43402	0.3623	0.38405
9.Maximum shear stress	7.18E+09	6.20E+09	6.93E+09
10.Middle Principle Elastic Strain	0.04491	0.041757	0.046908
11.Normal Elastic Strain	0.12638	0.11564	0.13111
12.Normal Stress	6.78E+09	5.71E+09	6.47E+09
13.Shear Elastic Strain	0.23548	0.20881	0.2358
14. Shear Stress,	1.18E+09	1.18E+09	1.35E+09
15.Stress Intensity	1.44E+10	1.24E+10	1.39E+10
16. Structural Error	0.031935	0.021619	0.028445
17.Total Deformation	0.06482	0.058122	0.066528
18. strain Energy	0.22607	0.16198	0.20635
19.Middle Principle stress	1.53E+09	1.57E+09	1.51E+09
20. Minimum Principle stress	8.29E+08	5.35E+08	7.20E+08

The Illizarov is modeled using ANSYS WORK BENCH Software. The load was applied on proximal end to the knee joint. The three types of fixator Ring materials are namely, Aluminium, Titanium, and 316LSS. The Illizarov fixator is analyzed using ANSYS WORK BENCH Software.

The from the above table 27 it is found that the total deformation is less in the 316LSS fixator and it is more in the results were given in the above table 29 titanium and Aluminium. According to Professor Illizarov the fracture gap should be maintained 2 to 3 mm. If the fixator is very stiff or the fixator is loose, micromotion at the fracture gap will not occur.The important criteria in this fixator are the total deformation, which occurs on the fixator, and has an influence on the fracture gap. The change in design based on the mechanical parameters which affects the fixator should not alter the gap size between the bone fragments. Figure 3, 4 and 5 shows Titanium fixator and steel K- Wire for 1800N Load. The Total Deformation of Titanium fixator and steel K-Wire is less when compared to the other fixators.

Fig 3 Illizarov fixator with rachose cube

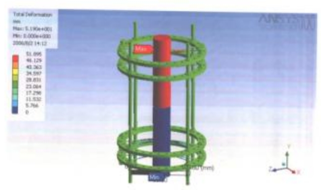

Fig 4 Illizarov fixator 300/300 K-Wire

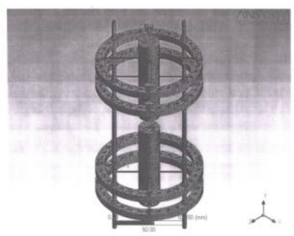

Fig 5 K-Wire Illizarov ring fixator mesh

Table: IV Material Properties of stainless and the four pre-tension cases

Material Properties E_1=193 GPa, E_2= 96.5 GPa,σ_1=520MPa,σ_2=1300MPa,Transverse bending. P_{mIN}=0N, P_{max}=250N		
	Initial extension (mm)	Pre-tension (N)
Case- 1	0	0
Case -2	0.1222	400
Case-3	0.2443	800
Case-4	0.3665	1200

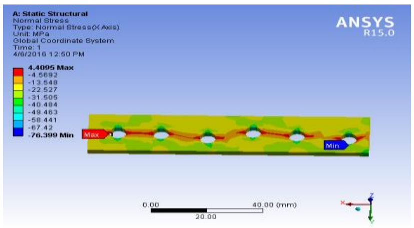

Fig 6 Normal stress stainless steel plate

V. CONCLUSION

The various implants and fixators were designed and analyzed for various types of bone fractures. The illizarov fixator with rachose cube is more stiff than other type of fixator. These types of fixators can take on more load. The illizarov fixator with 300/300 K-Wire is dynamically best design for bone lengthening.

REFERENCES

1. E. Yilmaz, O. Belhan, L. Karakurt, N. Arslan and E. Serin, “Mechanical performance of hybrid Ilizarov external fixator in comparison with Ilizarov circular external fixator”, Clinical Biomechanics, 18(6), Jul.2003, pp.518-522.
2. H. Stein, R. Mosheiff, F. Baumgart, R. Frigg, S. M. Perren, J. Cordey, “The hybrid ring tubular externalfixator: a biomechanical study”, Clinical Biomechanics, 12(4), Jun.1997, pp.259-266.
3. M. Pitkin, Y. Shukeylo and A. Gritsanov, “Mathematical modeling of fixation of a bone fragment in a new Double-needle external Fixator compared to hoffmann ii fixator”, Series on biomechanics, 23(1), Dec.2007, p.96.
4. L. Yang, M. Saleh and S. Nayagam, S, “The effects of different wire and screw combinations on the stiffness of a hybrid external fixator”, Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine, 214(6), Jun.2000, pp.669-676
5. G. Zhang, “Geometric and material nonlinearity in tensioned wires of an external fixator”, Clinical Biomechanics, 19(5), Jun.2004, pp.513-518.
6. B. Gasser, D. Wyder and E. Schneider, “Comparative investigation on the biomechanical properties of the circular and other three-dimensional external fixators”, In Biomechanics: Basic and Applied Research, Springer, Dordrecht, Research ,1987, pp. 429-434.
7. G. Bagnoli, G. Penna, A. Landini, N. Confalonieri, G. Torri, and V. Pietrogrande, “Experimental study on bone formation under straining forces: clinical and radiologic features”, Policlinico, 91(4), 1984, pp.908-13.
8. J. Goh, A. Thambyah, A. Noor Ghani and K. Bose. “Evaluation of a simple and low-cost external fixator”, Injury, 28(1), Jan.1997, pp.29-34
9. L. E. Lanyon and C. T. Rubin,”Static vs dynamic loads as an influence on bone remodeling”,Journal of biomechanics, 17(12), jan.1984, pp.897-905

AUTHORS PROFILE

Dr. D. Bubesh Kumar, is working as Associate Professor in the Department of Mechanical Engineering. He has published more than 15 research papers, He has written a book in the area of Fatigue corrosion of Biomaterials He has completed his Ph.D in the area of Biomedical Implants and fixators

Aravindan Varadan*, Assistant Professor Mechanical Department, Aarupadai Veedu Institute of Technology, Vinayaka Mission’s Research Foundation, Deemed To Be University. aravindan.mech@avit.ac.in

Praveen Kumar, UG Student, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation.

Hariharan, UG Student, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology**,** Vinayaka Mission Research Foundation.

Rajaram , UG Student, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation