

Modelling The Tibial Bone Using Reverse Engineering Technique and FGM Modelling of Simple Bar Using ANSYS.

Samad Khan, Rajnarasimha Sangam, Dinesh Chawde

Abstract— This work explores the area of biomedical imaging processing. Tibia is the largest bone after femur in the human body. Due to its complex geometry, developing an accurate and reliable model of Tibia bone is a challenging task for orthopedics. Therefore, to develop a 3D bone model Reverse Engineering Technique is used. This method minimizes the error in importing the models into the Finite element software to get better results. 3D slicer software is used to reconstruct the bone from Ct scan using DICOM files. A 3D model of human tibia bone has been modelled from CT scan images which is further segmented and modified to form an accurate 3D model by varying the segmentations parameters and methods. An open-source code 3D slicer is used to perform segmentation. Autodesk Fusion 360 software is used to convert it into a solid body and to reduce the facets and finally the model is imported to ANSYS software for meshing and analysis. This segmentation 3D CAD model can be used for analysis such as Model analysis, structural analysis and artificial bone manufacturing.

In-plane stress and displacement solutions are found using the elasticity approach for functionally graded rod having rectangular, hollow, tapered and tapered hollow cross-section and are validated with Finite Element Solution in Ansys. The analytical results show a good agreement with FE results.

Index Terms— ANSYS; Biomechanics; Finite element analysis; Functionally Graded Materials; Reverse Engineering Technique; Tibia bone; 3D slicer.

1 INTRODUCTION

Bone in biological terms is defined as connective tissue because it binds together and provides protection to various parts of the body. The lower limb consists of thigh, leg and foot. The leg bone consists of Patella, Tibia and fibula. The fibula bone is not involved in knee joint structure; the upper end is located further down and is located at the lateral side of the tibia bone. Tibia bone is the second largest and the strongest bone after femur bone, it is also called as shinbone, the proximal(upper) end of tibia bone articulates with the femur bone forming the knee joint and distal(lower) end of the tibia bone articulates with the talus forming the ankle joint. Tibia consist of 2 epiphyses and one diaphysis, epiphysis are the two rounder extremities of the bone, upper extremities (closet to femur) and lower extremities (closet to knee). Diaphysis is also known as the shaft is the mid-section of the tibia. It is very complicated to develop the computational model of tibia since it contains three layers of architecture. i) Cortical also known as compact bone.it is the dense outer surface of bone that protects the internal cavity by forming a layer. It is found in the shaft of tibia and femur bone and forms the outer shell around the trabecular bone at the ends of the joint and vertebrae. ii) Trabecular (also known as spongy bone or cancellous bone) from the articular surface it transfers the mechanical load to the cortical bone.It is an anisotropic, heterogeneous and highly porous material that is found at the epiphyses of the tibia and femur bone and in vertebral bodies is a highly iii) Bone marrow is a soft tissue stored in the central cavity of bone shafts known as medullary cavity. Objective of

this study is to model the femur bone by using reverse engineering technique and to compare the analytical and FE simulation results of functionally graded plane rod. Javad Malekani et.al provides the guidelines for the user to reduce the computational time and increase the accuracy of analysis in ANSYS. Various parameters like solution and geometric model, material model, mesh generation, number of analysis step, program control parameters, large deformations, operating system, etc. These parameters affect the accuracy and increase the computational time therefore it should be chosen carefully [1]. Martin Kubířek et.al used CT images of femur and tibia bone to create the geometric model and other model like cartilage and meniscus obtained with aid of literature. Material properties of models were created as isotropic and homogeneous.[2]. WanchalermTarapooma constructed the 3D model of Tibia bone from CT scan of by using MIMICS, GEOMAGICS and CATIA software. HyperMesh software used to generate the FE models than it is imported to ANSYS to study the stress distribution during stance phase running [5]. S. Karuppudaiyan et.al used CT scan images to develop the FE Bone model but for accurate model they required a better algorithm. In this paper A reverse engineering method is used to develop a finite element model of tibia bone to study the fracture risk and structural behavior of tibia bone under physiological loading conditions. This method can minimize errors in importing the models into the FEM software to obtain better results. This study helps doctors to have knowledge of implants during fracture. FE analysis was carried out and the results obtained were compared with literature. The maximum stress was obtained in the mid-

diaphysis of the tibia bone. This stress is 8% is higher than the previous literature. Therefore, reverse engineering approach is one of the best methods to develop complex geometric model [6]. Naveen Tippanagoudar et.al carried out the analysis of Tibia bone during static loading condition and during impact loading condition. From the analysis it is found that, for static load the stress induced is within the Yield point, and the stress induced for impact load is higher than the yield strength of the bone therefore the bone would fracture under impact loading during crash [7]. Özkan et.al the congruent geometry model obtained from CT images and non-bone structure (cartilage, tendon, etc.) is develop in GEOMAGIC the model is assembled in MIMICS and then it is transferred to ANSYS workbench for FE simulation. Boundary conditions are applied by fixing the lower side of Talus, and a rotational moment is applied on the upper surface of the tibia, which is a connection interface between a tibial plateau and meniscus; analysis was performed with both fixed and broken fibula. With fixed fibula the stress and displacement values decreased and have less value than broken fibula. it can be concluded from the paper that the fixation of fibula fracture affects positively more than no fixation operations [3]. Ehsan Taheri et.al obtained the bone geometry from CT images of a 32-years old healthy individual. MIMICS software was used to separate the tibia bone from femur, fibula and other muscles and then export point clouds text file, to have a perfect model of compact and spongy parts and of tibia it was imported to solid works where the volume construction & extension of nodes were done. For analysis it was imported to Abaqus software. Material properties selected were anisotropic material. Torsion stiffness and bending of the CAD model was calculated and the results were compared with previous literature studies [12]. .Kirthana et.al collected MRI or CT data to make a CAD model of bone. The bone is reconstructed using a 3D slicer and to obtain a smoother surface blender is used. Analysis is done to obtain the maximum stress and deformation. Weak zone areas are taken as regions of interest after an impact load is applied, prosthetic plates are modelled and assembled in the femur bone at the ROI. Different plates are used and assembled and analysis done and comparison made to find out the best material i.e., Titanium plate [14].

2 TIBIA BONE MODELLING

Developing a reliable and accurate CAD model of tibia model

is a challenging task for orthopedics due to its irregularity of shape of medullary cavity and outer part (internal and external). Therefore, to develop an accurate model Reverse engineering method is used.

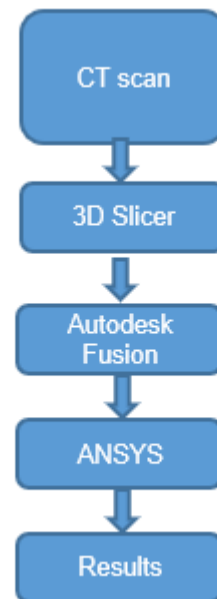


Fig1-Methodology and process flow of modelling and analysis

2.1 Medical Imaging Technique

X-ray, CT scan and MRI are the techniques used mostly to capture images of skeletal of humans. These techniques mainly differ in their capacity to penetrate tissues and bones and degree of resolution that can be achieved. CT (computerized Tomography) can process a series of X-ray images that have been taken around the body at different angles to create cross-sectional images of the bones(slices), soft tissues & blood vessels inside your body. computerized Tomography scan provide more detailed information than X-rays images. In this study the Tibia bone is reconstructed from CT scan. A CT scan of individuals suffering from degeneration is taken and reconstructed from DICOM images represented as .dcm. . Major advantage of .dcm format is that it does not provide the information about the source (anonymized data) and it cannot be misinterpreted

2.2 3D slicer

The CT scan DICOM files of lower limb is imported into 3D slicer software. 3D slicer software is used to construct a CAD model of tibia bone. It can read CT and MRI data.

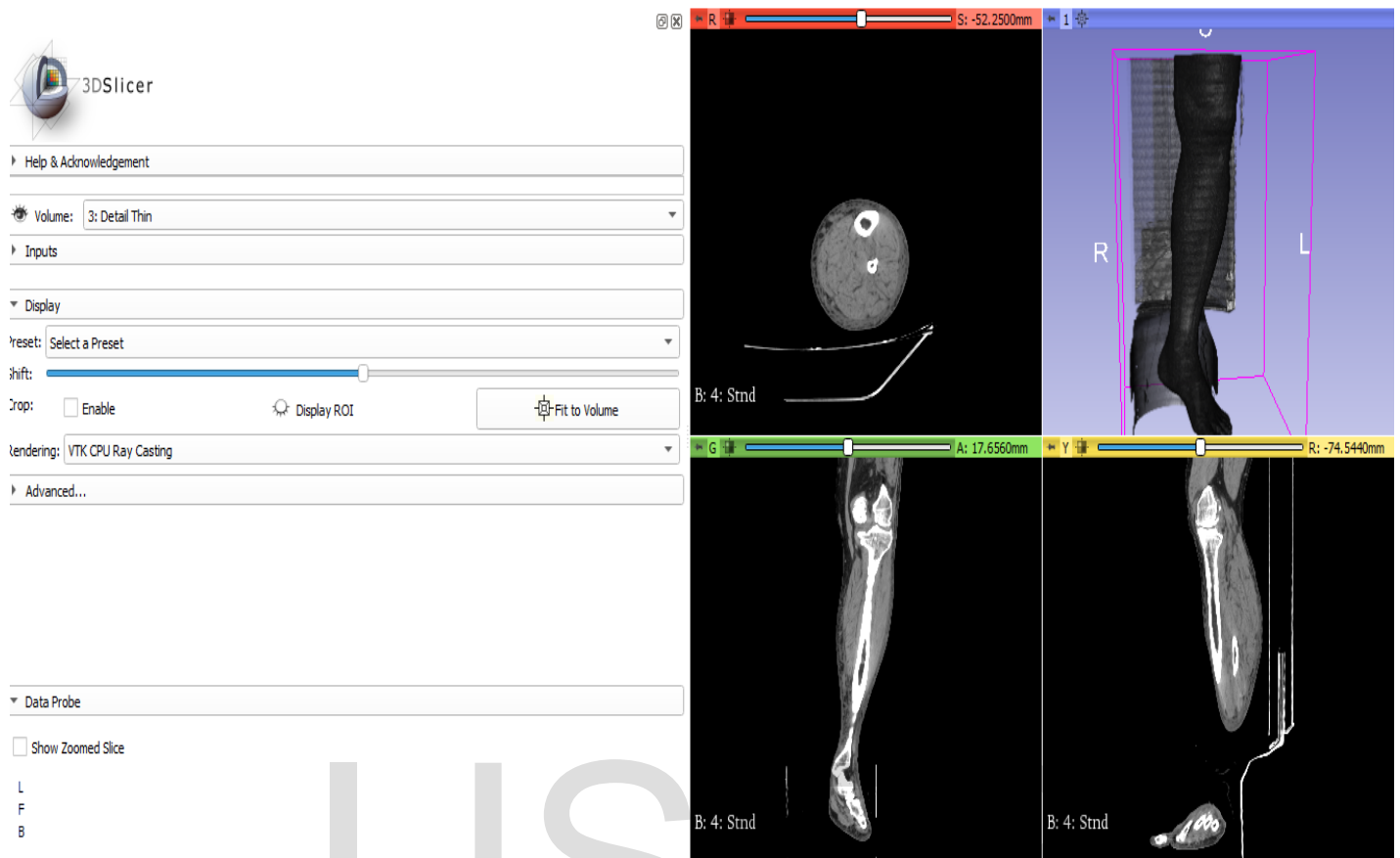


Fig 2 - CT scan of lower limb of 73 years old in 3D slicer

It is an open-source medical imaging software used for 3D visualization and for image processing. On the slicer window 4 views are shown, i.e., axial, Coronal, sagittal and 3-dimensional views as shown in the fig-2 then in the next step to select the bone type, a preset option is selected. In this study we took CT scan images therefore CT bones pre-set selection is selected and it can be observed in 3D view and by removing the unwanted material only the bones are visualized. Shift option is selected to view only bone and remove the unwanted material that got captured at the time of scanning. In the segment editor module, editing is done and then the threshold value for bone is adjusted from 90-3000 and since the Hounsfield Unit of soft tissue is less than 100 for bone is more than 200 so by adjusting the HU value only

the bone part is highlighted as shown in fig 3 in red color. After adjusting the threshold value there are lots of irregularities in the bone therefore to remove that, scissor option is selection and in that erase inside operation is ticked and the operation is carried out to erase the unwanted materials. The Ct scan of the lower limb is done. But for modelling we need only the tibia bone so in the next step the crop volume module is selected and cropping is done to get the only required bone. During cropping a rectangular box appears and we can adjust it to get the required part. The proximal portion of tibia articulates with the femur and the distal portion articulates with the ankle so after cropping little portion of femur and talus bone remains.

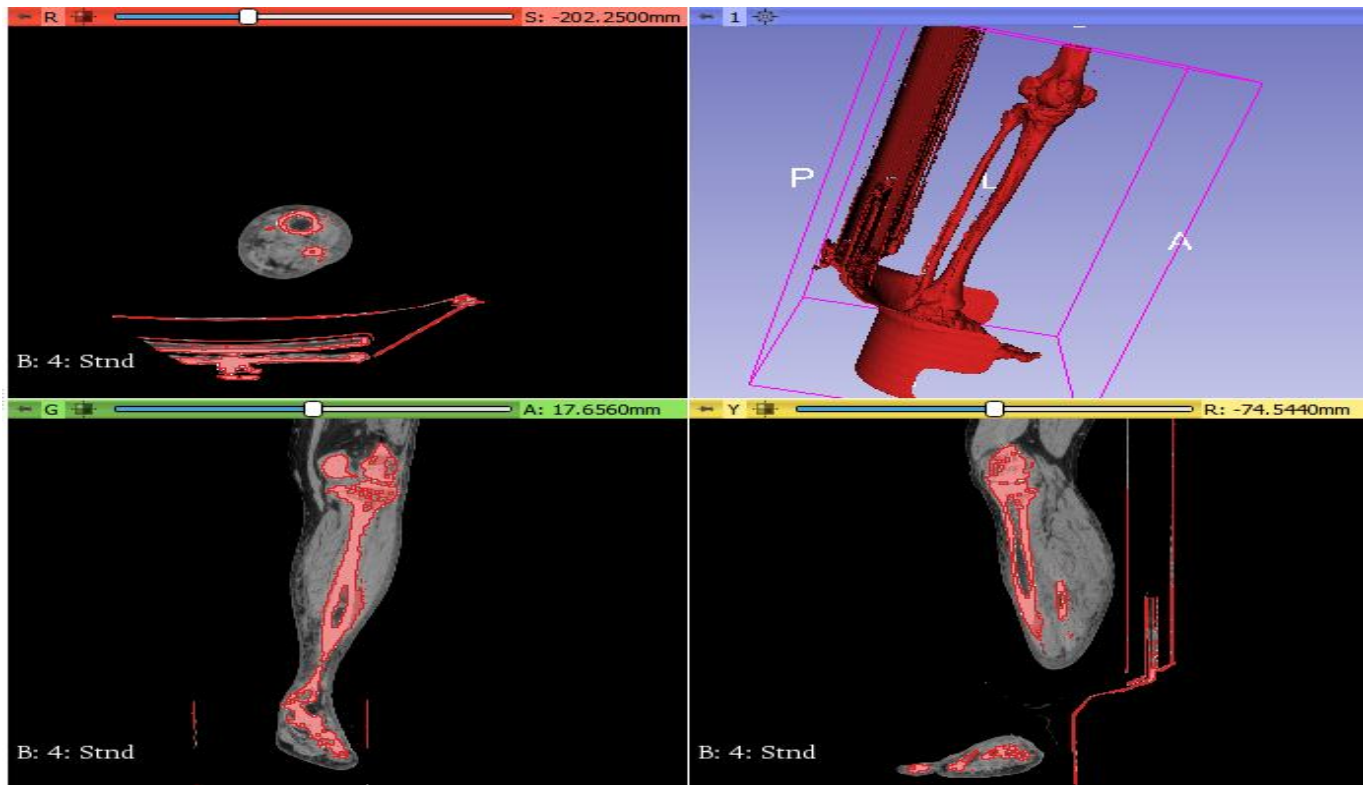


Fig 3- CT Bones pre-set selection in 3D slicer

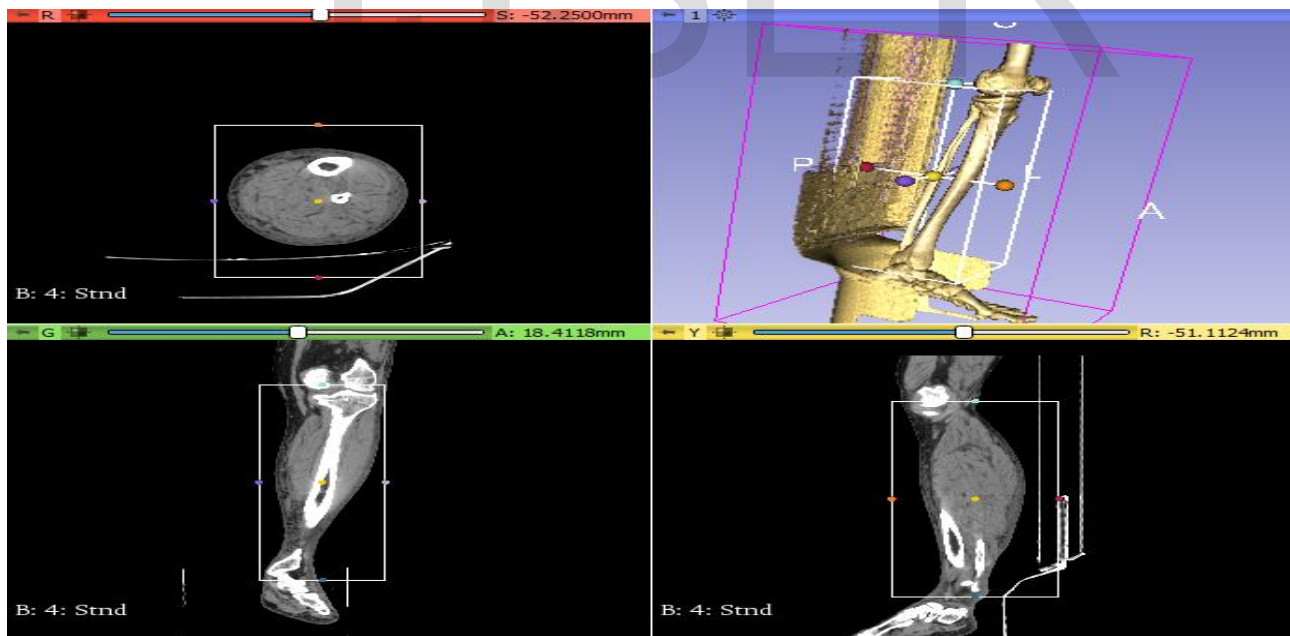


Fig 4- Volume rendering in 3D slicer

Segmentation- It is a process of locating the region of interest and separating it from the background (other

structure). Now to remove those unwanted portions of femur, fibula and the talus bone, segmentations are done. Various modes can be utilized in 3D slicers.

1. Manual segmentation
2. Automatic segmentation
3. Semi-automatic segmentation

Manual segmentation- In this segmentation the user manually outlines the region of interest (ROI) and manually the model is separated. It is a time-consuming process. For poor images data, Complex geometry or in cases where the model intensity is likely similar then manual segmentation is generally used. It is a tedious process and it is a refining method as it refines segmented. **Semi-automatic segmentation-** In this the computer

performs the segmentation process of the structure directed by the user. The user selects the parameters which guide active contour and this contour spreads all through the ROI and segments it.

For semiautomatic segmentation, in segment editor, add two layers: the first one is the structure that is required for study(tibia).and second layers that are not required (femur, fibula and talus bone) will work as a background. Green and blue color is given for the first and the second layers respectively. The first layer is selected and Paint effect is done on the tibia bone and blue paint is done for the structure that is not required which acts as a background. As shown in fig 5.

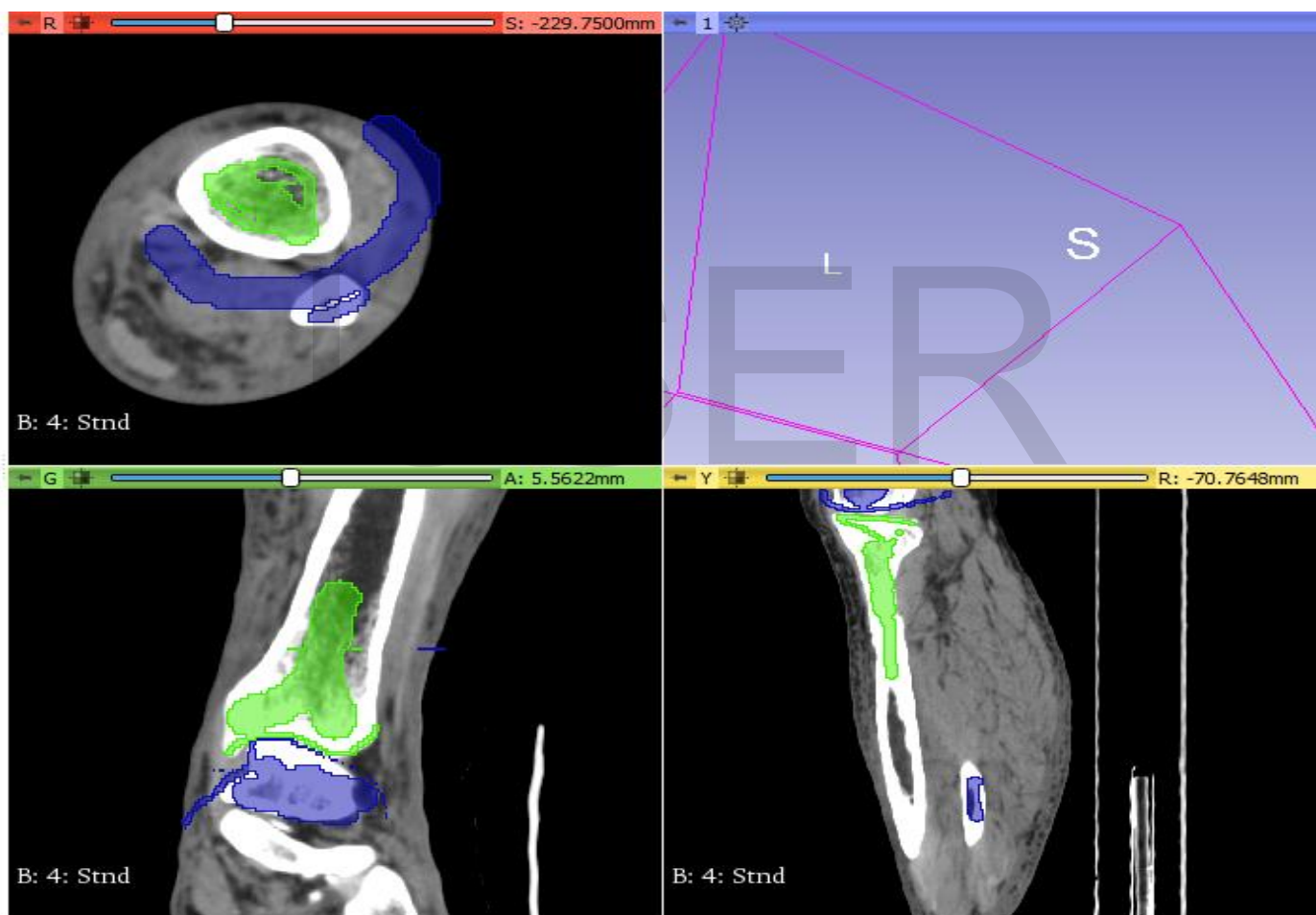


Fig 5- Manual segmentation in 3D slicer

In the segment editor module the Grow from seeds option is initialized and then the layer will expand and cover the region. The green color covers the ROI of the tibia bone and the remaining region will be covered in blue. In 3D

view the bone is now of two colors (tibia in green & the femur, fibula and talus in blue) now we go to the segment option and delete the background layer so automatically the blue region will get deleted. And the 3D view of the

tibia bone is created. The model obtained has a lot of irregularities and required refinement so smoothing operation is performed. In the segment editor module, a smoothing option is selected. Gaussian is a strong

algorithm and gives a perfect smoothing result so it is selected with a 3 mm kernel size. as shown in fig 6 we get the refined model of tibia and it is saved in STL format.



Fig 6- Tibia bone in 3D slicer

2.3 Autodesk Fusion 360

Autodesk Fusion 360 is a post-processor finite element software. It is the most effective tool for meshing the largest and most complex model. All the different shapes and surfaces that make up this model yet the object is still hollow on the inside as it's solely made up of the surface geometry. The STL file is an Unstructured Triangular Surfaces making up the outer surface of the model and it's not a completely solid object. BREP stands for Boundary

representation and can be seen opposite of our mesh file as it is a completely solid and a watertight model. The Brep file is made up of topological and geometric information. To convert it to a solid body the model is imported into Autodesk fusion software. Conversion from surface mesh to 3d volume is required otherwise the model will get open in CAD or Analysis software but it will show error during simulation.

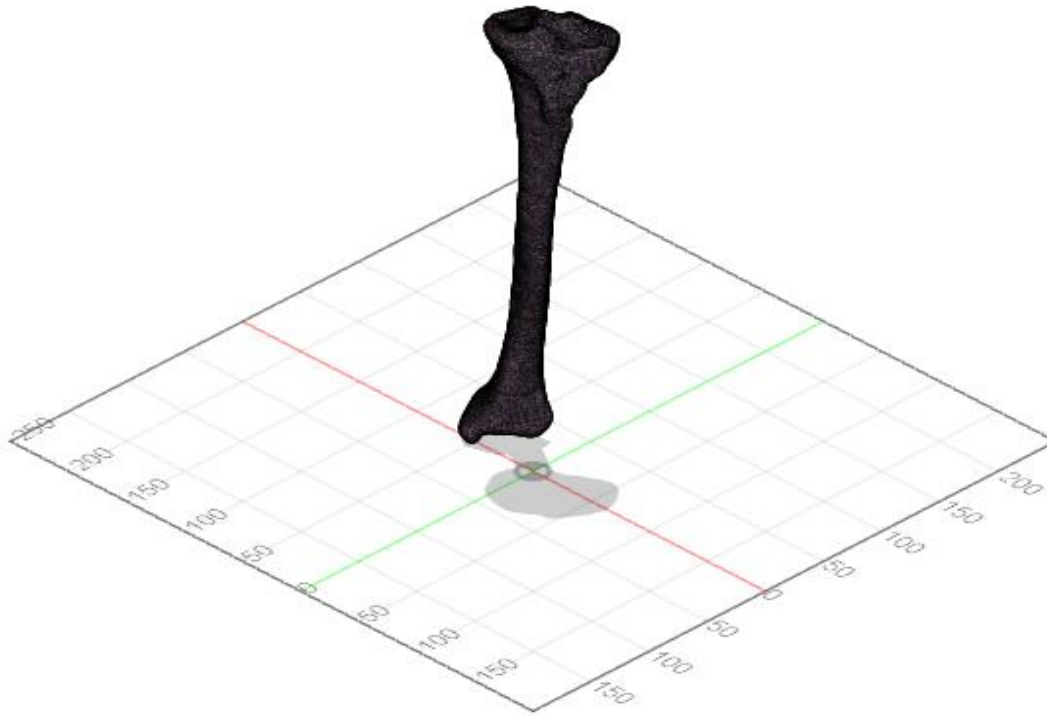


Fig 7-Volumetric meshing in Autodesk Fusion

2.4 ANSYS

The FE model of the tibia has been developed from the surface model generated by reverse engineering technique. It was meshed using Autodesk fusion 360

software and saved in IGES format. The volumetric meshed models imported in to Ansys software

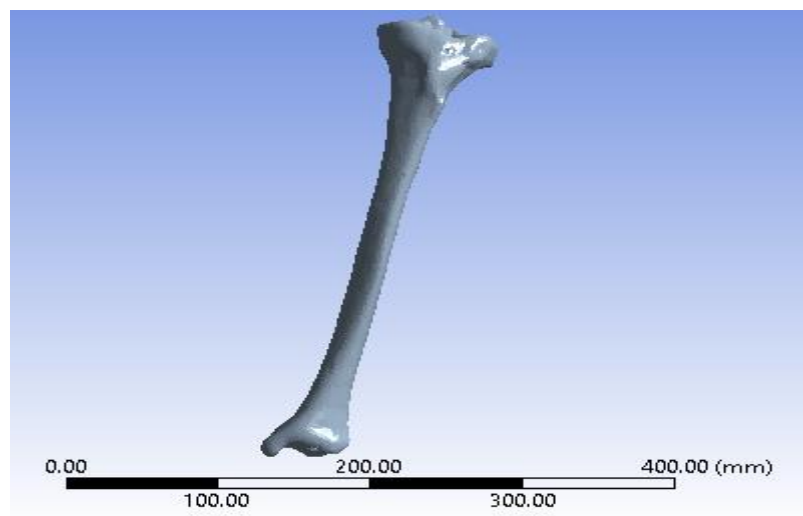


Fig 8 -Tibia bone in ANSYS software

3. RESULTS AND DISCUSSION

The geometry of the tibia bone is almost like a cylindrical tapered hollow bar and the young's modulus of bone varies in nonlinear manner. To achieve the accurate FE analysis results we compared the analytical solution with the FE simulation results by assuming linear variation of young's modulus w.r.t length. Starting with a simple rectangular bar to a Cylindrical tapered hollow bar,

analysis is done and the results are compared with an analytical solution. If both the results matched it can be concluded that the FE simulation that was performed on ANSYS software was correct. An analytical solution equation is required for Young's modulus varies linearly from $X=0$ to $X=L$. i.e., it varies linearly from E_{root} to E_{tip} along the length of the bar.

E at a distance x,

$$E_x = E_r + \frac{(E_t - E_r)x}{L} = E_r + Kx$$

Change in length with young's modulus varying linearly,

$$\Delta l = \frac{PL}{A(E_t - E_r)} \left[\ln \ln \left(\frac{E_t}{E_r} \right) \right]$$

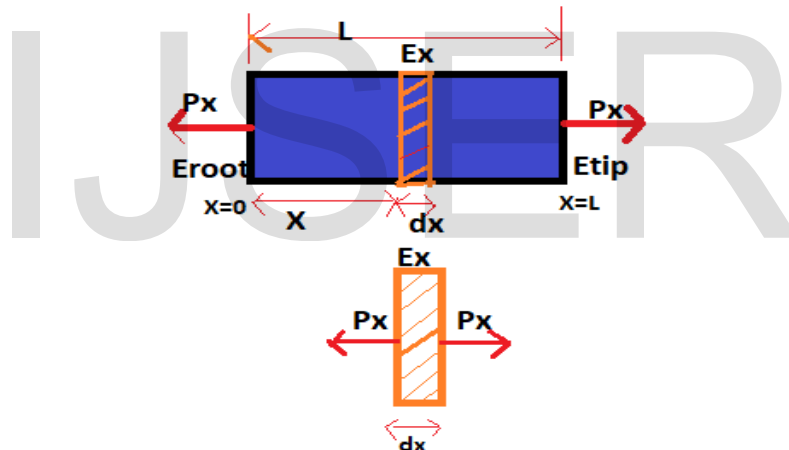


Fig 9 -Rectangular bar with varying young's modulus

3.1 FE simulation -ANSYS

FEA Steps in ANSYS

Step 1 – Define material properties as a function of temperature and do simple conductive thermal analysis to get the distribution of temperature through the longitudinal direction of the bar
Assign Coefficient of thermal expansion(α)=0 to neglect the effect of temperature in the solution
Step 2- Carry over structural analysis after mapping the thermal analysis results.

$$E_{\text{root}}=20000 \quad T_{\text{root}}=20000, E_{\text{tip}}=605000 \\ T_{\text{tip}}=605000(\text{Assume})$$

Where E = Young's Modulus and T =Temperature.
Any relation can be maintained between them like power law or exponential law i.e. (non-linear variation)
From fig-10 it can be seen that the temperature is varying linearly from $L=0$ (20000 Celsius) to $L=550\text{mm}$ (605000 Celsius) and since the temperature value is equal to young's modulus value, we get the linear variation of young's modulus w.r.t length.

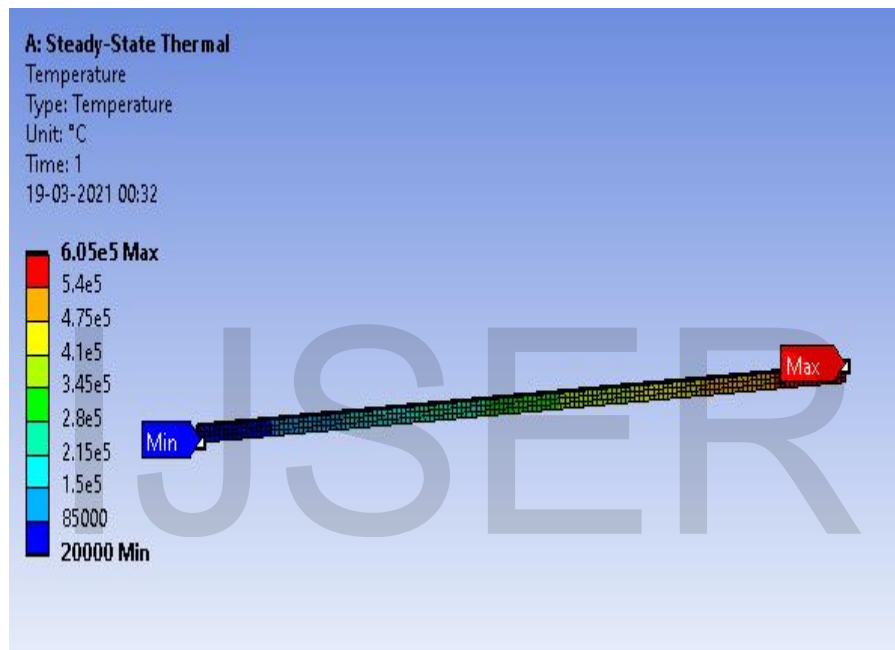


Fig 10- Linear spatial variation of temperature

3.2 Boundary Conditions:

For Finite element Analysis an important step required is the boundary condition for the model. Boundary conditions have a great impact on the analysis results. A fixed boundary condition is applied at $L=0$ surface and

a load of 500N is applied at the free end ($L=550$) in X-direction for all the model (rectangular, hollow cylinder, tapered and hollow cylindrical tapered bar).

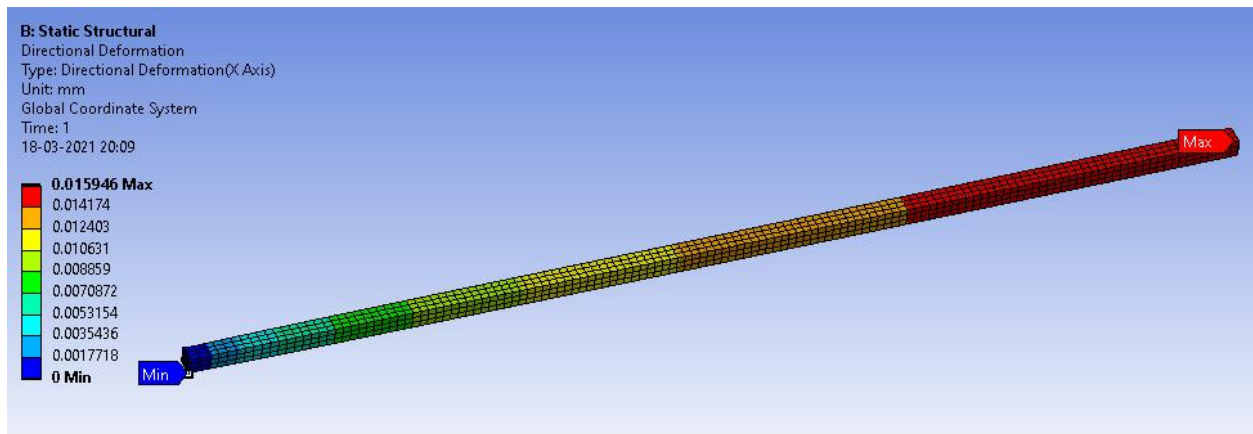


Fig 11-Deformation of rectangular bar in X-direction

Table 1-Comparison of analytical result with FE simulation result

Sr No	Model	Dimension	Deformation (Δl)(mm)		Stress (MPa)	
			Analytical	FE simulation result	Analytical	FE simulation result
1	Rectangular bar	Length-550mm, Width-10mm, Thickness-10mm	$\Delta l = \frac{PL}{A(Et - Er)} \ln \left(\frac{Et}{Er} \right)$ = 0.016	0.0159	$\sigma_{max} = \frac{P}{A}$ = 5	5
2	Hollow Cylinder	Length-550mm, Major diameter (d_1) =80mm, Minor diameter (d_2) =20mm	$\Delta l = \frac{4PL}{\pi(d_1^2 - d_2^2)(Et - Er)} \ln \ln \left(\frac{Et}{Er} \right)$ =0.00042	0.00041	$\sigma_{max} = \frac{4P}{\pi(d_1^2 - d_2^2)}$ = 0.1326	0.13194
3	Tapered Bar	Length-550mm, Major diameter (d_1) =80mm, Minor diameter (d_2) =20mm	$\Delta l = \frac{4PL}{\pi(d_1 d_2)(Et - Er)} \ln \left(\frac{Et}{Er} \right)$ = 0.0012	0.0012	$\sigma_{max} = \frac{4P}{\pi(d_2^2)}$ =1.591	1.6039
4	Hollow Tapered bar	Length-550mm, Major diameter (d_1) =80mm, Minor diameter (d_2) =40mm Hollow diameter(d_3) =10mm	$\Delta l = -\frac{2PL}{\pi D_3(D_1 - D_2)(Et - Er)} \left(\ln \left(\frac{D_1 + D_3}{D_1 - D_3} \right) - \ln \left(\frac{D_2 + D_3}{D_2 - D_3} \right) \right) \left[\ln \ln \left(\frac{Et}{Er} \right) \right]$ =0.00066	0.000654	$\sigma_{max} = \frac{4P}{\pi(D_2^2 - D_3^2)}$ =0.424	0.428

Table 2 -Cylindrical Tapered Hollow Bar maximum stress at various points varying from 0mm to 550mm.

Length (x)	Force	Young's Modulus $E_x = E1 - \left(\frac{E1 - E2}{L}\right)x$	Deformation	Tip Diameter $Dx = D1 - \left(\frac{D1 - D2}{L}\right)x$	Hollow Diameter D3	Stress $\sigma_{max}(\text{MPa})$ (FE simulation)	Analytical $\sigma_{max}(\text{MPa}) = \frac{4P}{\pi * (Dx^2 - D3^2)}$
0	500	E1=20000	0	D1=80	D3=10	0.009	0.1010
45.833	500	68750	1.82E-04	76.67		0.1138	0.1101
91.667	500	117500	3.22E-04	73.33		0.1186	0.1206
137.5	500	166250	4.36E-04	70		0.1316	0.1326
183.33	500	214996	5.36E-04	66.67		0.1459	0.1465
229.17	500	263753	6.28E-04	63.33		0.1624	0.1627
275	500	275940	7.15E-04	60		0.1817	0.1818
320.83	500	312500	7.99E-04	56.67		0.204	0.2046
366.67	500	361246	8.83E-04	53.33		0.2319	0.2319
412.5	500	458750	9.70E-04	50		0.2653	0.2652
458.33	500	507496	1.06E-03	46.67		0.3065	0.3063
504.17	500	556254	1.16E-03	43.33		0.3580	0.3581
550	500	E2=605000	1.27E-03	D2=40		0.4246	0.4244

Table 1 only give information about the deformation and stress value at the free end where the load is applied .Motive is to match the analytical and FE simulation results at all points from L=0 to L=550 to achieve this a path is created in ANSYS software to get the deformation and stress results at various length and for the analytical

solution, the Young's modulus will vary for rectangular and hollow cylindrical bar w.r.t. length, and both Young's modulus and Minor diameter(tip diameter) will vary for tapered and hollow cylindrical tapered bar at length varying from 0 to 550mm.

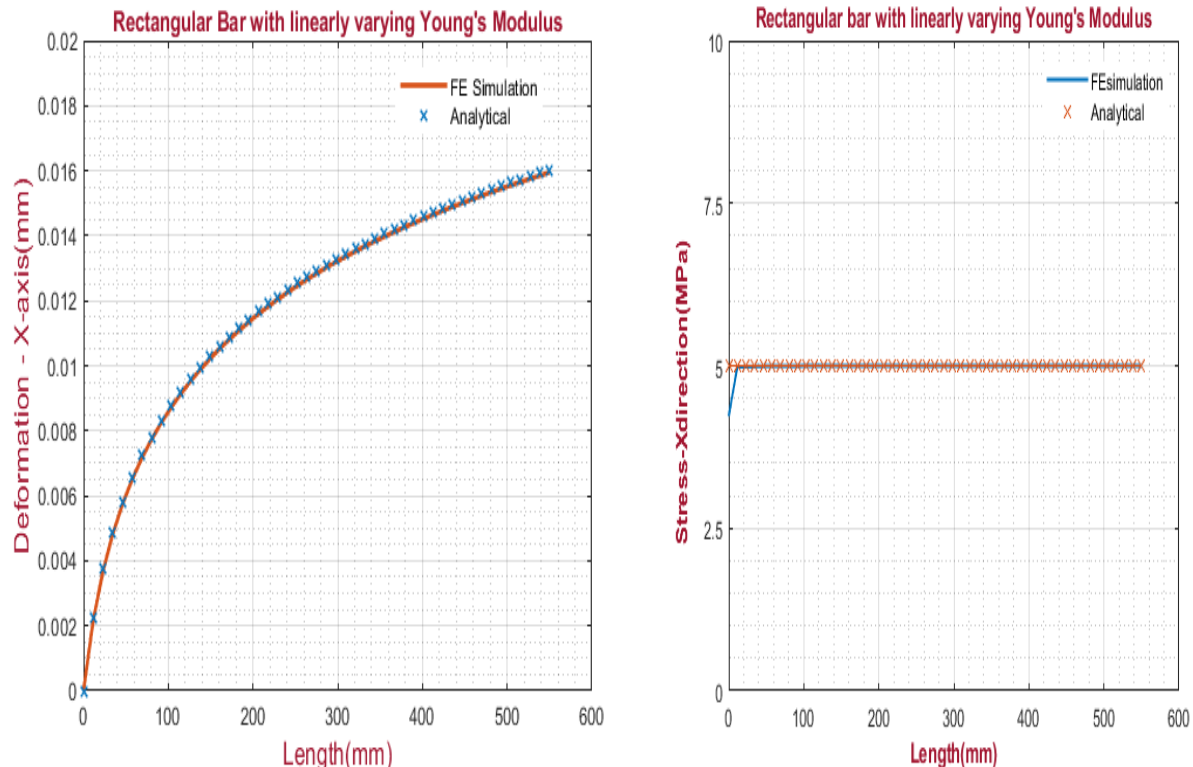


Fig 12- Comparison of Analytical and FE simulation result of Deformation and stress value w.r.t length for rectangular bar.

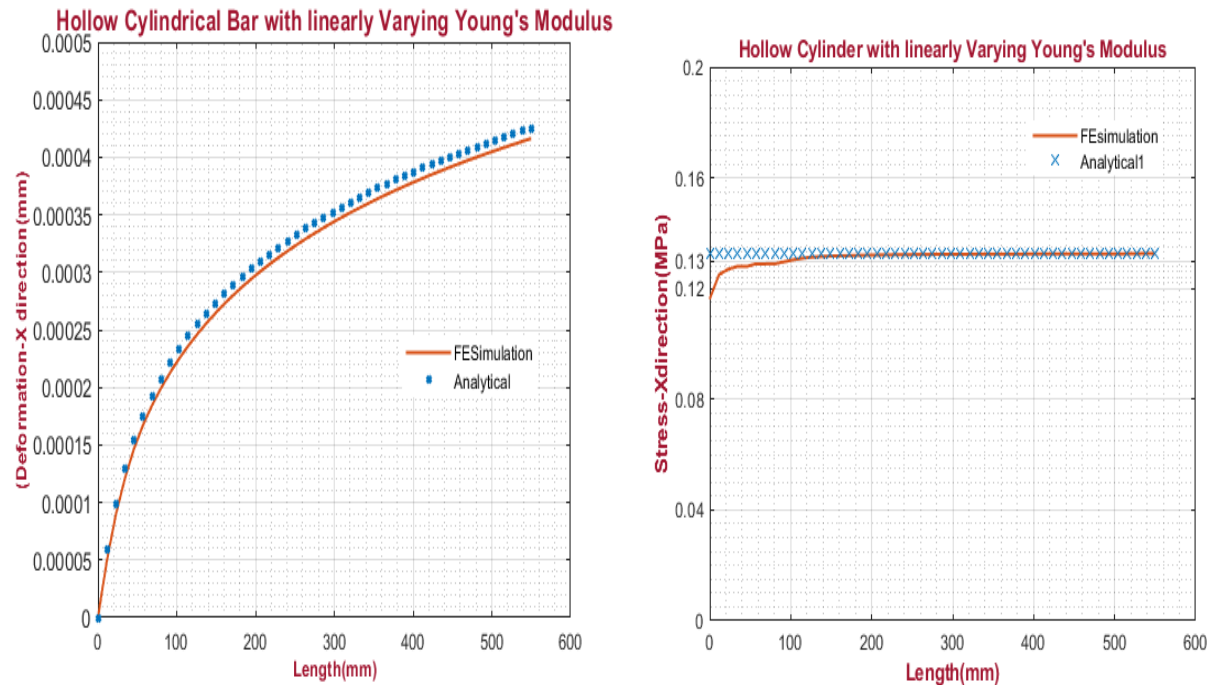


Fig 13- Comparison of Analytical and FE simulation result of Deformation and stress value w.r.t length for Hollow cylindrical bar.

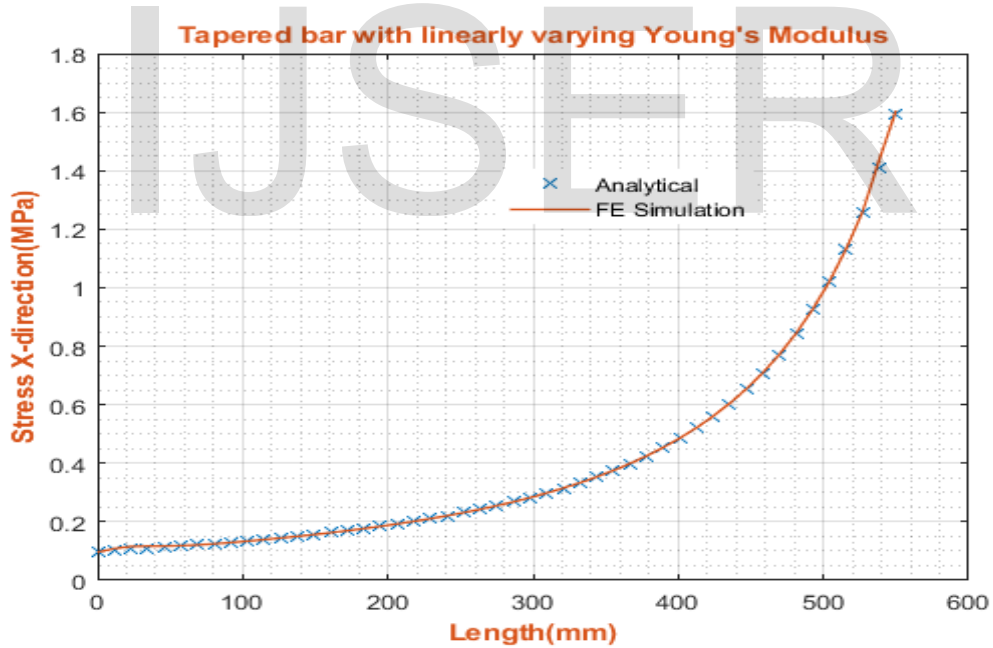


Fig 14- Comparison of Analytical and FE simulation result of Deformation and stress value at w.r.t. length for Tapered bar.

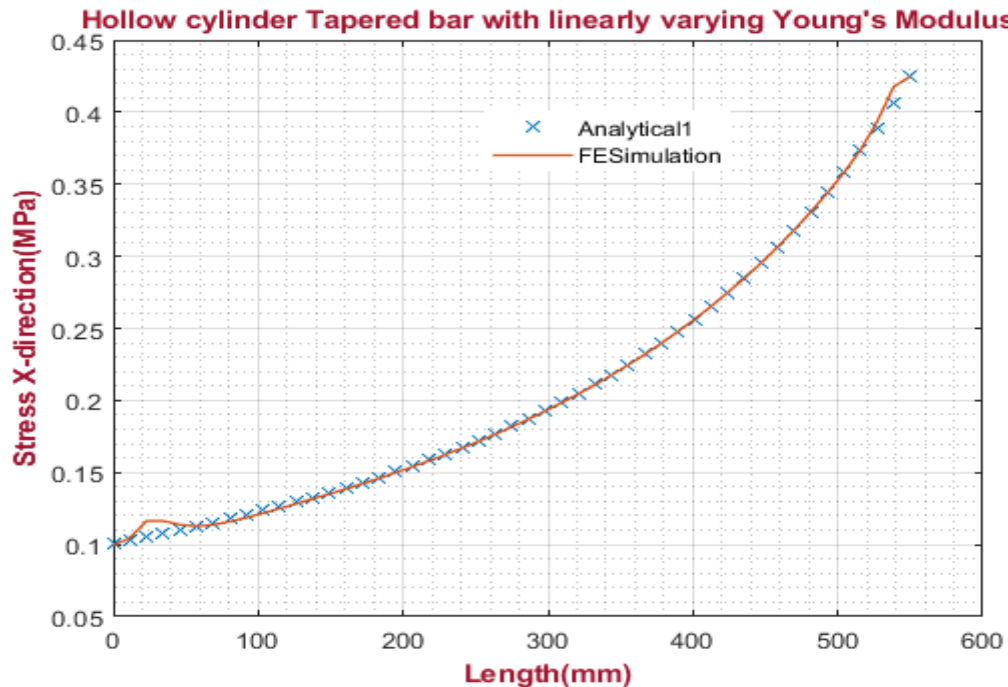


Fig 15- Comparison of Analytical and FE simulation result of stress value w.r.t length for Hollow cylindrical Tapered bar.

The normal stresses and the directional deformation obtained from the stress analysis of a simple bar (rectangular, hollow cylinder, tapered and taper hollow cylindrical bar) are shown in fig 12, fig 13, fig 14 and fig 15. The maximum stress and deformation occur at a region with least cross-sectional area. The results of stresses and deformation have been plotted w.r.t length from $L=0$ to $L=550$. Analytically 49 points taken by varying the length value from $L=0, 11.45, 22.917, 34.375, 45.833, 57.292, \dots, L$ element size of 6mm. For hollow cylinder bar 2269 nodes and 364 elements with 30mm elements size. For tapered bar, nodes and element are 3269 & 648 respectively with element size 22. For hollow tapered bar nodes are 2230 and elements are 435 and element size is 25mm. System response (stress, deformation) will converge to a repeatable solution with decreasing element size with minimum number of elements and nodes for the properties vary in linear manner.

=550. and the results are compared with the analytical solution, the analytical results validated with Finite Element Solution in Ansys. The analytical results show a good agreement with FE results.

Meshes in the ANSYS software are linear quadratic. Minimum number of elements required in a model to ensure that the analysis results are not get affected so that convergence is achieved, for rectangular bar minimum number of elements and nodes are 2781 and 368 with

4. CONCLUSIONS

- Reverse Engineering is one of the best methods to develop 3D bone models. The work done in this study is an initial step and could lead to the development of an expert system. The CAD modelling of the tibia bone took place without any experimental work. This study may help orthopedic surgeons to have a detailed knowledge about the tibia bone to choose appropriate implants for fractures which help in providing an efficient and fast fixture to a patient with a fracture. The contributions towards orthopedic biomechanics of both doctors and engineers could hence prove to be important and extremely beneficial to doctors and patients. Tibia bone material properties vary in nonlinear manner and in material properties of ANSYS

software it does not provide a direct way to model variation therefore a thermal-structural analysis is performed on simple bar by varying the young's modulus linearly from $x=0$ to $x=L$ and same value of temperature and young's modulus is taken and to neglect the expansion of bar due to thermal effect the coefficient of expansion value is taken as 0. FGM modelling of simple bar is with linearly varying young's modulus is calculated analytically and also solved in FE simulation software and their results are compared. As seen in the fig 12, fig 13, fig 14 and fig15 there is very small error and sometimes the error is zero. This method delivers high accuracy and it does

not require deep knowledge of FGM mathematical

models.

REFERENCES

- [1] Javad Malekani, Prasad KDV Yarlagaddab,, Beat Schmutz, YuanTongGud,Michael Schuetz 'How to increase the accuracy of analysis and reduce the computational time in ANSYS in the case of deformation study of orthopedic bone Plates' Advanced Materials Research Vols,2013.
- [2] MartinKub'ıcek,Zdeněk, 'STRESS STRAIN ANALYSIS OF KNEE JOINT', Engineering MECHANICS, Vol. 16, 2009, No. 5, p. 315–322
- [3] A.Özkan,Y.K.işioğlu, 'The rotational loading capacity and effects of fibula on lower extremity' ,Journal of Engineering Research and Applied Science, June 2013, pp 138-143.
- [4] Mihai-Constantin Balaşa, Simona Mihai, Viviana Filip, Alexis-Daniel Negrea, GheorghitaTomescu, 'MODELLING THE TIBIAL BONE USING CAD TECHNIQUES, STARTING FROM THE 3D SCAN MODEL' International Journal of Mechatronics and Applied Mechanics, 2018, Issue 3.
- [5] WanchalermTarapooma and TumrongPuttapitukpornb 'Stress Distribution in Human Tibia Bones using Finite Element Analysis', ENGINEERING JOURNAL Volume 20 Issue 3,2016.
- [6] S. Karuppudaiyan, J. Daniel Glad Stephen, V. Magesh,'FINITE ELEMENT ANALYSIS OF TIBIA BONE BY REVERSE ENGINEERING MODELLING APPROACH' , International Journal of Pure and Applied MathematicsVolume 118 No. 20 2018.
- [7] Naveen Tippanagoudar, Anantha Krishna G L,' Finite Element Analysis of Tibia Bone' International Journal of Engineering Science and Computing, December 2018.
- [8] G Narayanaswamy, Bindu A Thomas, 'Finite Element Analysis of Tibia Bone Model', International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-9 Issue-1, October 2019.
- [9] N. Suryarao, K. Sreedevi, MD. Touseef Ahamad, 'Analytical Study of the Tibia Bone under Static Load using Finite Element Method' International Journal for Research in Applied Science & Engineering Technology (IJRASET), Volume 5 Issue VIII, August 2017.
- [10] Mr. Amit Patil, Mr. V. R. Gambhire , Mr. P. J .Patil, 'EXPERIMENTAL AND ANSYS ANALYSIS OF STRESSES IN REAL PRESERVED INTACT HUMAN TIBIA BONE UNDER STATIC BODY LOAD TO ENHANCE THE DESIGN OF IMPLANT FOR ORTHOPEDIC SURGERY.', International Journal of Advance Engineering and Research Development Volume 5, Issue 07, July -2018
- [11] M. Ghosh, B. U. Chowdhury, M. S. Parvej, and A. M. Afsar, 'Modeling and analysis of elastic fields in tibia and fibula', AIP Conference Proceedings 1919, 28 December 2017
- [12] Ehsan Taheri1, Behrooz Sepehri2, Reza Ganji3, 'Mechanical Validation of Perfect Tibia 3D Model Using Computed Tomography Scan', Engineering, December-2012,
- [13] Fritz J. and Dolores H.Russ, '3-D Modeling And Finite Element Analysis of the Tibia'.
- [14] S. Kirthana, M. BindhuSupraja, A.S.N. Vishwa, N. Mahalakshmi,StaticStructural Analysis on Femur Bone Using Different Plate Material,Materials Today: Proceedings,Volume 22, Part 4,2020,Pages 2324-2333,ISSN 2214-7853,.