Application of Finite Element Analysis to Assess Bone Strength of Patient Specific Model

Mithun Goyal[1], Parveen Kalra[2]

[1]Research scholar, Department of Production Engineering, PEC University of Technology, Chandigarh-160012, India

[2]Professor, Department of Production Engineering, PEC University of Technology, Chandigarh-160012, India

Abstract: ***Speculating pathologic fractures caused by metastatic disease like osteoporosis, cancer, infection due to weakness of the bone continues to be a challenging clinical problem. We assessed the ability of non-invasive imaging and Finite Element Analysis to predict the stress on the bone with osteosarcoma. A human bone model of influenced osteosarcoma to the femur bone was used as a model. Computed Tomography (CT) scans were obtained and Finite Element Analysis is done by applying loads on the bone to predict stress in tumor area of the bone. The relative success of this approach to evaluate stress in human osteosarcoma model indicates this with further development and attestation could serve a way to non-invasive stress calculation technique.***

Keywords: Finite element analysis (FEA), bone, strength, tumor, mesh.

1. Introduction

Pathologic fractures in patients with metastases are major cause of misery. Currently the most effective methodology adapted to prevent pathologic fractures is precautionary internal fixation. Predictive tools have been developed to differentiate between metastatic lesions. The rating scheme published by Mirels has been most widely adopted and uses information on radiographic evidence of bone destruction, anatomic location, and pain [1]. However Damron et al. concluded that Mirels's rating system is the best at this time but it is not the perfect solution to pathologic fracture risk prediction [2].

In effort to improve the technique toward getting better and accurate results regarding the stress on the bone & prediction of bone strength advance structural based methods like beam theory methods [3], dual energy X-ray absorptiometry (DXA), computed tomography (CT based section analysis [4], CT based finite element methods [5,6]. These approaches show promise of improving our ability to know the stress and bone strength with metastatic tumor.

One limitation of previous research studies is that predictive models are usually tested against experimentally created defects that probably do not accurately represent clinical tumor osteolysis [3,7]. Many tumors are associated with considerable trabecular bone loss before cortical bone loss; these situations are difficult to simulate experimentally. Another limitation of this CT based FE Analysis is that arbitrary loads are applied on the bone so we cannot get the actual stress acting on the bone & strength of the bone.

In this research paper we are focusing on applying approximate boundary and loading conditions on the bone to analyze stress on the bone and strength of the bone.

2. Methodology

The Methodology adopted for FEA of a patient specific model of tumor femur in which the stress on various areas of the femur bone due to body weight and muscle loadings is explained as in Figure1

2.1 3D Model Generation

A patient specific CT data in DICOM (Digital Imaging and Communications in Medicine) format was used to generate the 3D model in MIMICS. Mask was created by defining threshold value which means segmentation of the object that will contain only those pixels of the image with a higher than or equal to the threshold value. Sometimes an upper and lower threshold is needed; segmentation mask contains all pixels between these two values. The generated region mask was used to develop 3D model for the bone. The 3D reconstruction is based on 3D interpolation techniques that transform the 2D images in a 3D model. The 3D model generated was converted to STL (Stereo Lithography) format for mesh generation & analysis of the femur bone.

2.2 Mesh Preparation

3-matic software is able to combine CAD tools with preprocessing (meshing) capabilities. Surface mesh of the femur bone generated. While meshing this femur bone quality of the triangles are improved, extra shells are removed, intersecting triangles are eliminated and all parameters are optimized by defining the height to base ratio of the triangular elements. Volume mesh is generated in Abaqus by converting tri components to tet components.

2.3 Material Properties

Human bone is highly heterogeneous and nonlinear in nature, so it is difficult to assign material properties along each direction of bone model. So, in biomechanics material properties can be assigned to the bone either in MIMICS or in any Finite element module. The material properties to femur bone were assigned in Abaqus Finite element module. The properties that were assigned to the bone were density 1900kg/m^3, modulus of elasticity 21 GPa and Poisson ratio as 0.3 for FE analysis.

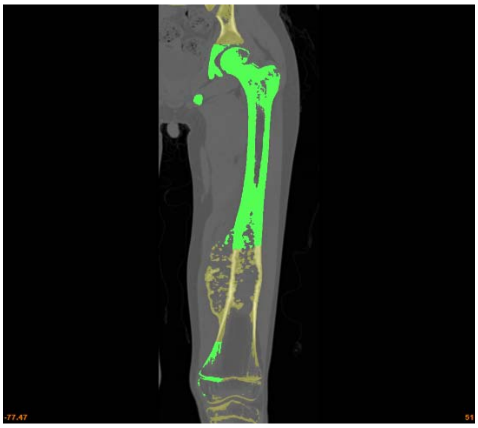

Figure 1: CT data in DICOM format

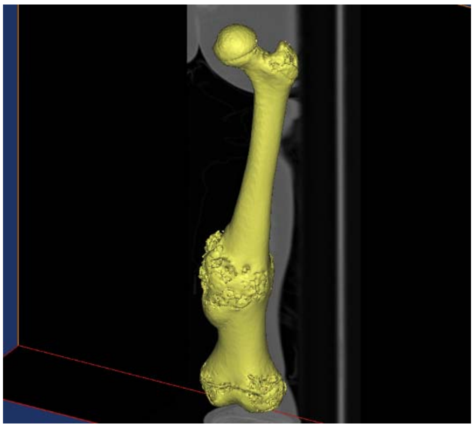

Figure 2: 3D Model Generation

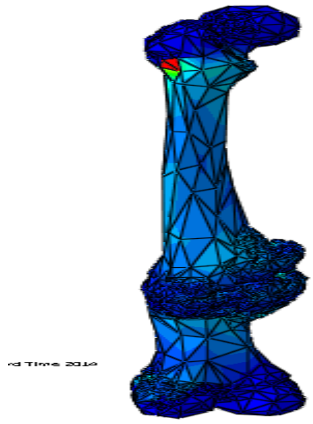

Figure 3: FEA Analysis

Ability of FEA to handle complex geometries and distributions of material properties, the FE method has been used frequently to estimate the strength and stiffness of whole bones and of trabecular bone, as well as to compute the distributions of stress and strain. Abaqus FEA is used to analyze the stress strain on the bone. Material properties are sectioned on the tumor bone i.e. Elastic modulus 21 GPa, density 1.9 gm/cm^3. Analysis is run in Abaqus because boundary conditions.

2.4 Boundary Conditions

Femur bone is solid and inflexible. The three dimensional Finite Element model of femur with volumetric mesh was imported in Abaqus and boundary conditions are applied to the bone and load of 400 N was applied with femur head and patella surface as a fixed support.

3. Results and Discussions

3D model generated from MIMCS in STL format was analyzed in Abaqus software by applying material properties of the bone, boundary conditions & loads to know that whether the bone would be able to withstand load or not and how much stress is coming on the tumor bone. It is clear from the analysis that the bone failed at the applied load.

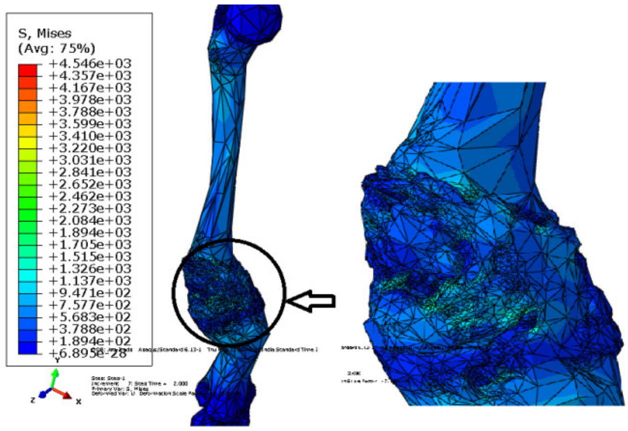

Figure 4: Analysis of bone

It was found that bone failed from the portion which was affected by tumor and the same portion was highlighted in figure 4.

4. Conclusion

The protocol is standardized for the analysis of strength of the patient specific bone from the CT scan images of the patient. With the help of this model we can achieve a certain level of pre surgical planning and design of bone plates for patient specific diseases. Further the model generated can be 3D printed for teaching aids.

5. Acknowledgement

The author wishes to express their sincerest thanks to the CSIR, Chandigarh staff for their guidance in carrying out my work. I would like to thank Dr. Parveen Kalra for their guidance in my research work.

References

[1] Arrington SA, Schoonmaker JE, Damron TA,Mann KA, Allen MJ, " Temporal changes in bone mass and mechanical properties in a murine model of tumor osteolysis", Bone. 2006, pp 359–367.

[2] Damron TA, Morgan H, Prakash D, Grant W, Aronowitz J, Heiner J, " Critical evaluation of Mirels' rating system for impending pathologic fractures", Clin Orthop, RelatRes 2003,415 (Suppl): S201–S207.

[3] Hong J, Cabe GD, Tedrow JR, Hipp JA, Snyder BD, "Failure of trabecular bone with simulated lytic defects can be predicted noninvasively by structural analysis", J Orthop Res, 2004;22: pp479–486.

[4] Keyak JH, Rossi SA, Jones KA, Skinner HB, "Prediction of femoral fracture load using automated finite element modeling", J Biomech, 1997;31: pp 125–133.

[5] Keyak JH, Kaneko TS, Rossi SA, Pejcic MR, Tehranzadeh J, "Skinner HB. Predicting the strength of femoral shafts with and without metastatic lesions", Clin Orthop Relat Res. 2005;439: pp 161–170.
[6] Keyak JH, Kaneko TS, Tehranzadeh J, Skinner HB, " Predicting proximal femoral strength using structural engineering models", Clin Orthop Relat Res, 2005;437: pp 219–228.
[7] Lee T, "Predicting failure load of the femur with simulated osteolytic defects using noninvasive imaging technique in a simplified load case", Ann Biomed Eng, 2007; 35: pp 642–650.

Author Profile

Mithun Goyal has completed his B.Tech from Giani Zail Singh College of Engineering and Technology. Bathinda. He is currently pursuing M.E in Production and industrial engineering from P.E.C University of technology, Chandigarh. His research work is in the area of application of finite element analysis to assess bone strength of patient specific model.

Dr. Parveen Kalra is a professor in P.E.C University of technology. He has vast experience in research related to ergonomics, robotics, finite element analysis, biomedical area, etc.