IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Vol. 8, Issue 8, August 2021

DOI: 10.17148/IARJSET.2021.88119

Nonlinear analysis of Human hard tissues using finite element code in MATLAB

Nikhil BN[1], Dr. Chandrashekhar Bendigeri[2]

Research Scholar, Department of Mechanical Engineering, University Visvesvaraya College of Engineering, Bangalore, Karnataka, India[1]

Professor, Department of Mechanical Engineering, University Visvesvaraya College of Engineering, Bangalore, Karnataka, India[2]

Abstract: The Development of patient-specific medical Implants is highly appreciated which requires prior determination of porous and mechanical characteristics. This is because the increase in porosity decreases the mechanical properties and increases the damping factor which leads to subsequent bone loss upon healing or remodeling. Bone is a living material that provides structural support to the body and therefore enabling locomotion and protection of the living being. On the contrary to isotropic materials from classical mechanics, this type of tissue can respond adaptively to its environment. Apart from the natural growth during aging and fracture healing, which are temporary, the internal structure is remodeled and repaired naturally based on the nature of the lifestyle. This work primarily aims at the formulation and development of a finite element program using MATLAB to study the non-linear behavior of hard tissues. The results from FEA software are validated with NAFEMS benchmark problems and the experimental results.

Keywords: Finite element analysis, MATLAB, Bone, Damage model, Biomechanics

I. INTRODUCTION

The trabecular bone exhibits a highly complicated non-linear material behaviour. For example, the three stages of failure mechanism of bone are, such as softening after the yield point, plateau, and densification phenomena, are observed during compression[1]. Soft tissues exhibit mechanical properties and behavior with higher complexity than most engineering materials and structures. For modeling purposes, it is crucial to select a material law as well as material parameters that well describe the mechanical properties of the soft tissue[2]. The current commercial finite element analysis software is not built for studying hard tissues and is not ideally suitable enough to solve highly non-linear geometry and material properties like human hard tissues. Even though current commercial software can be customized with user-defined functions the effort required and time consumed is very high. The complex geometry of causes huge variation of stiffness. And material properties vary in all three directions, the material properties depend upon the fluid density in the tissues and the density of pores. Because of all these issues, current solvers cannot be used as it is to solve the biomechanics problems efficiently. And Current constitutive models for trabecular bone are essentially limited to infinitesimal strains[3] [4].

This work is carried out by implementing the Material degradation method for rectangular plates of specified dimensions and studying its behaviour with human temporal bones. Temporal bones are mostly flat and can be hence can be studied in two-dimension FEA simulation with suitable assumptions. In this research, the finite element analysis is performed using the code developed with MATLAB routines and PDE toolbox.

II. LITERATURE REVIEW

Hadi S.Hosseinia et al., explains that vertebral fractures constitute a significant health problem and involve the progressive collapse of bone with large strains. Authors have extended the work of Garcia et al., for the simulation of higher magnitude strain values including the post-yield softening and densification phenomenon [5]. A fundamental model of bone based on both volume fraction and orientation is formulated and was implemented in a commercial finite element solver and was also validated with experimental results. The isotropic softening is controlled by the cumulative plastic strain and non-linear elastic spring is taken into account for densification. To avoid convergence problems during iterative solving due to softening of tissue a visco-plastic approach was considered [3].

Anita Fung et al., explains the methodology of validating a patient-specific finite element (FE) modeling program for bone strain prediction in the human metatarsal. Strain gauges were used for the measurement and were performed on 33 metatarsals from seven human cadaveric feet, and subject-specific FE models were generated from computed tomography

IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Vol. 8, Issue 8, August 2021

DOI: 10.17148/IARJSET.2021.88119

images. Material properties for the FE models were assigned based on the data from the published density-modulus relationship as well as density-modulus relationships developed from optimization techniques. With a training set of 17 Metatarsals, an optimized relationship was developed and was cross verified with a test set of 16 metatarsals [6].

Maren Freutel et al., provided a summary of different modeling methods and strategies with associated material properties, contact interactions between articulating tissues, validation & sensitivity of soft tissues with a special focus on knee joint soft tissues, and intervertebral disk. Furthermore, it reviews and discusses some salient clinical findings of reported finite element simulations. The proposed model contributed to the understanding of functional biomechanics of soft tissues. Models can be effectively used to explain and visualize clinically relevant questions. However, end-users should be aware of the complexity of such tissues and of the capabilities and limitations of these approaches to adequately simulate a specific in vivo or in vitro phenomenon [2].

Anders Halldin et al, has explained a material model to study the time-dependent behaviour of cortical bones. The biomechanical response of the implant, when subjected to load, is affected by the visco-plastic behaviour. Also, creep, elastic behaviour, and the remodeling phenomenon affects the acceptability of an implant. Hence long term in-vivo effects were considered in the constitutive material model. An equivalent rheological model is presented and the simulation results with this constitutive model are in line with the published in-vivo and in-vitro experimental results. The constitutive model presented is based on the viscoelastic/plastic material model with remodeling term which describes the relationship between stress and strain in in-vitro. According to this research publication, the total strain "ε" is given by: ε = strain due to (elastic component + viscoelastic component + plastic component + remodeling) [7].

III. METHODOLOGY

To study the behaviour of cortical bones considering the material nonlinearity and suitable continuum damage mechanics model there are multiple programming languages available like C, C++, Fortran, MATLAB, Python, etc. Each language has there own advantages and disadvantages and MATLAB has one of the most powerful tool available for this type of study. The built in tools like PDE toolbox, and functions which simply the matrix operations etc. MATLAB (matrix laboratory) is a versatile and powerful numerical computing environment and fourth-generation programming language. A proprietary programming language developed by MathWorks, MATLAB allows matrix manipulations, plotting of functions and data, implementation of algorithms, creation of user interfaces, and interfacing with programs written in other languages, including C, C++, Java, Fortran and Python.

In the current research, the finite element solver development and validation are carried out as shown in the below flow chart fig 1.1. The work was started with a detailed literature review, studying the various continuum damage mechanics models available and also selecting a type of hard tissue for the study. The finite element code was developed with MATLAB routines and the PDE toolbox. The FE results are compared with available published experimental results.

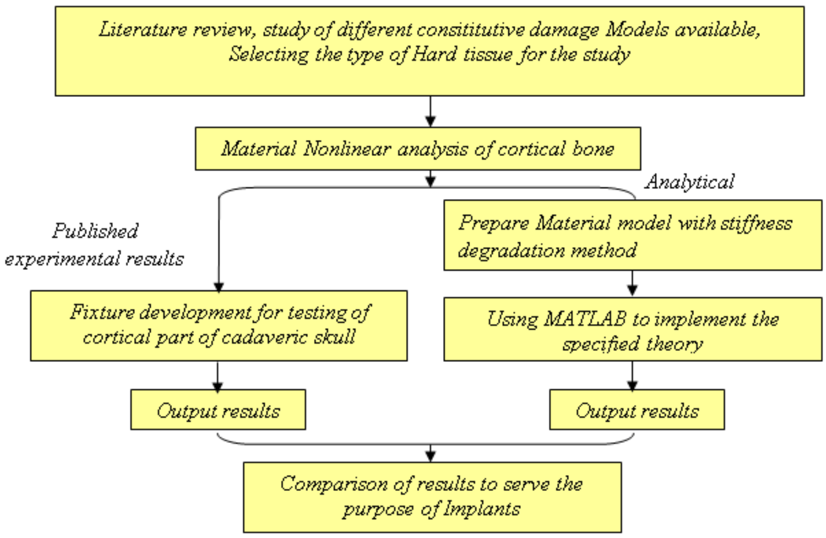


Fig: 1.1 Outline of the current work

IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Vol. 8, Issue 8, August 2021

DOI: 10.17148/IARJSET.2021.88119

The bilinear isotropic model with stiffness gradation method similar to the model proposed by Jacky Mazars et al [8] is considered in the study[2]. In the continuum damage mechanics formalism, a damage variable d represents the amount of deterioration due to crack growth. Damage plays an essential role in the weakening of the material's stiffness. Mazars model describes the fracture behaviour by anisotropic scalar damage variable d, which enters the constitutive stress-strain relationship:

$$\sigma = (1 - d)E\varepsilon \qquad \text{[i]}$$

Where:

E = The matrix of Hooke
d = The damage variable
ε = Elastic Strain

The value of d lies between and 1 and 0. 0 is healthy and 1 is broken. The damage is controlled by the equivalent deformation. The value of "d" is defined in the combination of two damaging modes i.e., one in tensile and other in compress loads. The relation of d would be

$$d = \alpha_t \times d_t + \alpha_c \times d_c \qquad \text{[ii]}$$

Where d_t and d_c are the damage variables in tension and compression and coefficient α is the weighing factor.

The tensile damage and compressive impairment constant are given by the relation:

$$d_t = 1 - \frac{k_0(1 - A_t)}{k} - \frac{A_t}{\exp\,[B_t(k - k_0)]}$$

$$d_c = 1 - \frac{k_0(1 - A_c)}{k} - \frac{A_c}{\exp\,[B_c(k - k_0)]}$$

The equivalent strain ε_q is updated in every iteration with the sum of principal strains in Macauley brackets $\langle . \rangle$. "k" is a state variable to memorize the value of equivalent strain:

$$\varepsilon_{eq} = \sqrt{\langle \varepsilon_1 \rangle^2 + \langle \varepsilon_2 \rangle^2 + \langle \varepsilon_3 \rangle^2} \qquad \text{[iii]}$$

The above algorithm is written in MATLAB routine. And a front-end graphical user interface is created. A screenshot of the GUI is shown below. The program has two variants:
1) standard biso solver
2) With material degradation and damage model

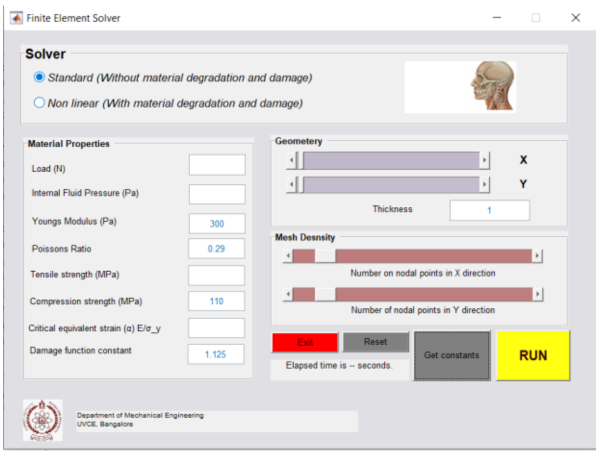


Fig 1.2: A screenshot of the GUI of the Program created in MATLAB

IARJSET

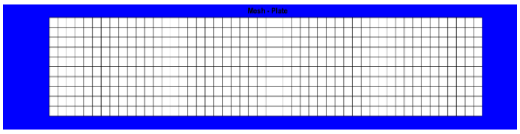
ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Vol. 8, Issue 8, August 2021

DOI: 10.17148/IARJSET.2021.88119

The finite element model with standard quadrilateral elements is created. The length and breadth of the rectangular plate is 38 mm and 11.82 mm respectively. The dimensional values are based on the samples used for experimental testing. So that the results can be directly correlated.



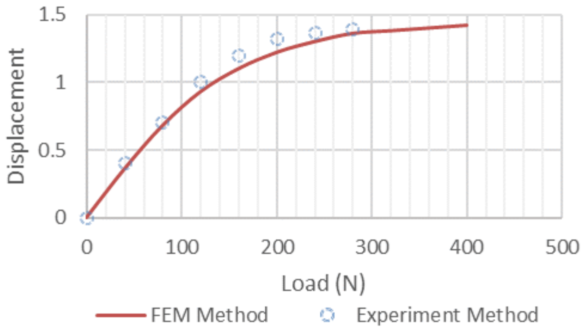


Fig 1.3: a) Meshed model. b) Load vs displacement compared for Experimental and FEM results.

From the theoretical formulations and analysis in MATLAB it observed the transverse displacement of a rectangular plate considering the damage model theory subjected to a transverse load of 360N has a transverse displacement of 1.44mm which almost matches to the experimental results. Refer fig 1.3.

IV. CONCLUSION

This study was focused on the formulation and development of non-linear theoretical methods using finite element analysis for predicting the nonlinear static response of temporal bone of human skull. A new approach in describing the continuum damage model for cortical bone has been proposed. This approach consists of considering material property degradation method or in other words the stiffness was being monitors and was used as parameter to model the damage.

The properties of these elements (Implants) can be derived from tensile test of cadaveric skull or from the journal references. The basic principles or refined finite element models including step loading and boundary condition have been simulated in MATLAB for the material nonlinearity. The results from finite element method was inline with the experimental simulation.

REFERENCES

[1] C. S. Lee *et al.*, "A new constitutive model for simulation of softening, plateau, and densification phenomena for trabecular bone under compression," *J. Mech. Behav. Biomed. Mater.*, vol. 65, pp. 213–223, Jan. 2017, doi: 10.1016/j.jmbbm.2016.08.028.

[2] M. Freutel, H. Schmidt, L. Dürselen, A. Ignatius, and F. Galbusera, "Finite element modeling of soft tissues: Material models, tissue interaction and challenges," *Clinical Biomechanics*, vol. 29, no. 4. Elsevier Ltd, pp. 363–372, 2014, doi: 10.1016/j.clinbiomech.2014.01.006.

[3] H. S. Hosseini, D. H. Pahr, and P. K. Zysset, "Modeling and experimental validation of trabecular bone damage, softening and densification under large compressive strains," *J. Mech. Behav. Biomed. Mater.*, vol. 15, pp. 93–102, 2012, doi: 10.1016/j.jmbbm.2012.06.005.

[4] B. Lethaus *et al.*, "A treatment algorithm for patients with large skull bone defects and first results," *J. Cranio-Maxillofacial Surg.*, vol. 39, no. 6, pp. 435–440, Sep. 2011, doi: 10.1016/j.jcms.2010.10.003.

[5] D. Garcia, P. K. Zysset, M. Charlebois, and A. Curnier, "A three-dimensional elastic plastic damage constitutive law for bone tissue," *Biomech. Model. Mechanobiol.*, vol. 8, no. 2, pp. 149–165, Apr. 2009, doi: 10.1007/s10237-008-0125-2.

[6] A. Fung, L. L. Loundagin, and W. B. Edwards, "Experimental validation of finite element predicted bone strain in the human metatarsal," *J. Biomech.*, vol. 60, pp. 22–29, Jul. 2017, doi: 10.1016/j.jbiomech.2017.06.010.

[7] A. Halldin, M. Ander, M. Jacobsson, and S. Hansson, "On a Constitutive Material Model to Capture Time Dependent Behavior of Cortical Bone," *World J. Mech.*, vol. 04, no. 11, pp. 348–361, 2014, doi: 10.4236/wjm.2014.411034.

[8] J. Mazars and J. Lemaitre, "Application of continuous damage mechanics to strain and fracture behavior of concrete.," in *NATO ASI Series, Series E: Applied Sciences*, 1985, no. 94, pp. 507–520, doi: 10.1007/978-94-009-5121-1_17.