iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL
FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 11 Issue: V Month of publication: May 2023

DOI: https://doi.org/10.22214/ijraset.2023.51649

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue V May 2023- Available at www.ijraset.com

A Review on Biomechanics and Development of Finite Element Models of Human Head Neck System

Lilia Dominic[1], Raiza Susan George[2]
[1]M.Tech Student, [2]Assistant Professor, Department of Civil Engineering, Mar Athanasius College of Engineering, Kothamangalam, Kerala, India

Abstract: The biomechanics of human head neck system could be better understood by creating finite element models of the same that accurately represents most of the components of head and neck. The initial geometry of the human head and neck was obtained using the computer tomography scans. The finite element models are created from these computed tomography scans and are further undergone through the various analysis procedures of finite element analysis. Thus, the developed finite element head neck models could be studied for various kinds of impacts acting on the head neck system during the time of any vehicle collisions, spaceflights etc. The development of these models and the various impact studies helps in the safety of the people. In this paper the concept of Finite element method, application of finite element method in biomechanics and various steps involved in this method are discussed in detail. Similarly, the finite element models of head and neck system developed by various researchers are also considered.

Keywords: Biomechanics, Finite Element Method, Computed Tomography Scans, Human Head Neck Models, Application, Impact

I. INTRODUCTION

The safety of human beings plays a vital role in various activities involved. Various responses that the human beings may subject to should be understood well in order to take of sufficient improvements for the safety of human beings. Limited by computational speed, earlier mathematical models tended to simplify the system under study so that a set of differential equations could be written and solved. Advances in computing technology and analysis software have enabled the development of many sophisticated models that have the potential to provide a more comprehensive understanding of human impact response, injury mechanisms, and tolerance thereby improving the safety of human beings. Finite element models using either implicit or explicit solvers where large displacement and nonlinear problems can be solved using the available libraries of elements, material laws, and contact algorithms. Many of these codes also provide capabilities in model meshing, model building, graphics, and animation. Due to limitations in computing power, early models tended to be simpler than the current ones. As computational capabilities increased, newer mathematical models became more sophisticated. The field of mathematical modeling will continue to grow at a rapid pace primarily due to developments of advanced numerical methods and improvement of computing power. More importantly, the cost of computing continues to fall, making it possible to exercise a very sophisticated model at a relatively low cost. This paper reviews the biomechanics of the head and the neck and the importance of the development of finite element models for developing the human safety.

II. FINITE ELEMENT METHOD

Finite Element Method (FEM) is an integrated product of many disciplines, including mechanics, mathematical physics, computational methods, computer technology, and so on. The finite-element method is an effective modern analytical tool. The analyses of various mechanical problems can be concluded to the following two forms, the analytical method and the numerical method. Although the basic equations and boundary conditions have been established, very few simple questions can be solved to obtain the analytical solution. For the questions with complex mathematical equations or irregular physical boundary, it is insurmountably difficult to solve the mathematical problems by analytical method.[1] After introducing the simplifying assumptions, the analytic method can be used to get the approximate solution which may sometimes lead to wrong answers. To meet the needs of the complex project problems, the numerical solution can be adopted to finish a variety of effective numerical calculation.

©IJRASET: All Rights are Reserved | SJ Impact Factor 7.538 | ISRA Journal Impact Factor 7.894 |

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue V May 2023- Available at www.ijraset.com

At present, numerical solution methods in engineering practical applications commonly include the limited element method, finite difference method, the boundary element method, weighted residual method, etc. With the rapid development and extensive application of computer, the finite element method, a relatively new and very effective numerical method, has gradually been used to solve the complex engineering problems. The essence of the finite-element method is that a complex continuum is divided into a limited number of simple cell body, which transforms the infinite freedom degrees to the limited freedom degree. The first step of a finite element method is to discrete the structure, that is, to divide the entire structure into a finite element based on different accuracy requirements, performance requirements and other factors ensure the convergence and stability of numerical solution.[2] The positions between elements or element and boundary are connected by nodes. Next is the element analysis, for the element, the nodal forces are the external forces on the elements through the nodes, which determines not only the displacement of this node, but also the effects of other nodes in this element. Afterwards, the overall analysis is done which include the combination of whole stiffness matrix and the establishment of balance equations. The introduction of support conditions and total load of nodes is the next step. All unknown nodal displacements can be solved for solving algebraic equations with the finite-element software. The element strain can be solved by the nodal displacements, and the element stress can be reversed through the physical equations.[3]

III. APPLICATION OF FINITE ELEMENT METHOD IN BIOMECHANICS

Biomechanics is the study of the structure and function of biological systems at any level from whole organisms to organs, cells by means of the methods of mechanics. Computational biomechanics is the application of engineering computational tools, such as the Finite element method to study the mechanics of biological systems.[5] Computational models and simulations are used to predict the relationship between parameters that are otherwise challenging to test experimentally, or used to design more relevant experiments reducing the time and costs of experiments. One of the main advantages of computational biomechanics lies in its ability to determine the endo-anatomical response of an anatomy, without being subject to ethical restrictions. This has led FE modelling to the point of becoming ubiquitous in several fields of Biomechanics.

The finite element analysis can improve the mechanical properties of medical devices and optimize the design of equipment. The mechanical properties of medical equipment often determine the effectiveness of the clinical application. Therefore, it is more important to evaluate the mechanical properties of appraisal instrument. In addition to the experimental methods, the simulative experiment to the mechanical equipment with the finite-element method has the advantages of short time, cost less, handling complex conditions, comprehensive mechanical properties and good repetitiveness, etc. [6] In addition, the finite-element method can also optimize the design, guide the design and improvement of medical devices, and obtain better clinical effects. The powerful modelling capabilities and interface tool if the finite element software can distinctly build the models of three-dimensional human bone, muscle, blood vessels and other organs, and be able to give their biomechanical properties. In the simulated experiments, the model simulation to experimental conditions and mechanical tests to simulate the stretching, bending, torsion, anti-fatigue can solve the deformation, stress, strain distribution, the internal energy change and the ultimate destruction situation on different experimental conditions. Some simple examples of biomechanics research include the investigation of the forces that act on limbs. Biomechanics is also applied to studying human musculoskeletal systems. It is also tied to the field of engineering, because it often uses traditional engineering sciences to analyse biological systems. Applied mechanics, most notably mechanical engineering disciplines such as continuum mechanics, mechanism analysis, structural analysis, kinematics and dynamics play prominent roles in the study of biomechanics. Usually, biological systems are much more complex than man-built systems. Numerical methods are hence applied in almost every biomechanical study.[2] Research is done in an iterative process of hypothesis and verification, including several steps of modelling, computer simulation and experimental measurements.

IV. HUMAN HEAD NECK SYSTEM

Skull as a complicated mechanical construction, consists from 28 different bones, connected by sutures of different structural kinds from smooth to teeth-like hardness. From a mechanical and structural point of view, the human neck is a very complex mechanism, containing vital neurologic, vascular, and respiratory structures as well as the cervical vertebrae and spinal cord. the vehicle injury priority rating data indicated that neck injuries became the fifth most important injury category (after head, face, chest and abdomen). The human spine supports the body’s weight, aids in movement, and plays an important r role in protecting the spinal cord. Among the vertebrae, the cervical spine is the most frequently injured part, and cervical spine injury can be life-threatening.[8] The intact cervical spine model consists of the cortical bone, cancellous bone, intervertebral disc (IVD), anterior longitudinal ligament (ALL), posterior longitudinal ligament (PLL), capsular ligament (CL), ligamentum flavum (LF), interspinous ligament (ISL), supraspinous ligament (SSL), and spinal cord.

From the clinical literature, about a dozen different types of neck fractures or fracture/dislocations have been described, the most frequent being of the flexion/compression or the extension/compression type. The number of fractures or dislocations of the cervical spine without cord injury would probably far exceed the number with spinal cord injury. [10]

The first finite element (FE) model of a human head was represented by an ellipsoid. [7] The skull and brain were represented by viscoelastic materials. He assumed the skull and brain to be linearly elastic and approximated the skull with a layered structure. A 3-D linear elastic model of the brain for frontal impact, which included representations for the dura folds, ventricles, and brain stem. Roller supported linear springs were used to simulate the tethering of the brain to the skull. studied various types of definitions for the sliding interface between the brain and skull to determine the best method to represent the movement of the CSF layer. [8]It was found that by adding a sliding interface between the brain and skull. Another detailed FE head-neck complex model. The aim of their study was not to reproduce bony fracture, but to simulate more moderate lesions, the vertebrae were modelled by rigid shell elements. The model responses were validated against volunteer and/or cadaveric impacts in frontal, lateral, oblique, and rear directions. studied available finite element human head-neck complex and concentrates on its modal and dynamic responses. Resonance frequencies and responses of the human head neck complex's finite element model in impact simulations have been analysed. The fundamental frequency of modal analysis of finite element model is 35.25 Hz which is reasonably close to existing literatures.

A three-dimensional linear elastic FE analysis of six adjacent cervical vertebrae (C2- C7) has been developed. The model is composed of vertebrae and intervertebral discs. The objective of this research is to address the new advances in the generation of finite element meshes of the human cervical spine. A procedure to create a finite element model based on available biomedical closed volumes obtained from CT scans was introduced. [11] The 3D biomedical parts were imported and meshed using the ABAQUS FE code. Continuous materials properties were assigned to the C2-C7 structure separating the vertebrae and discs in two different material groups. Boundary conditions were used to fix translation and rotation in every node located on the inferior surface of the C7 vertebra. To assess the overall behaviour of the C2-C7 spine segment in four different load modes (compression, lateral bending, flexion and extension), the model was statically loaded in four different surfaces created in the upward-facing long bony odontoid process (or dens) from the C2 vertebra. Another FE neck model, in which, each vertebra consisted of only 12 solid elements, and spring-damper elements were used to simulate the major ligaments. The model responses were compared to volunteer kinematics during frontal and lateral impacts. One limitation of this model was that muscles which could have a significant influence on model response during frontal and lateral impacts were not simulated. The study was carried out to illustrate the goodness of the finite element method on biomechanics research. [12]

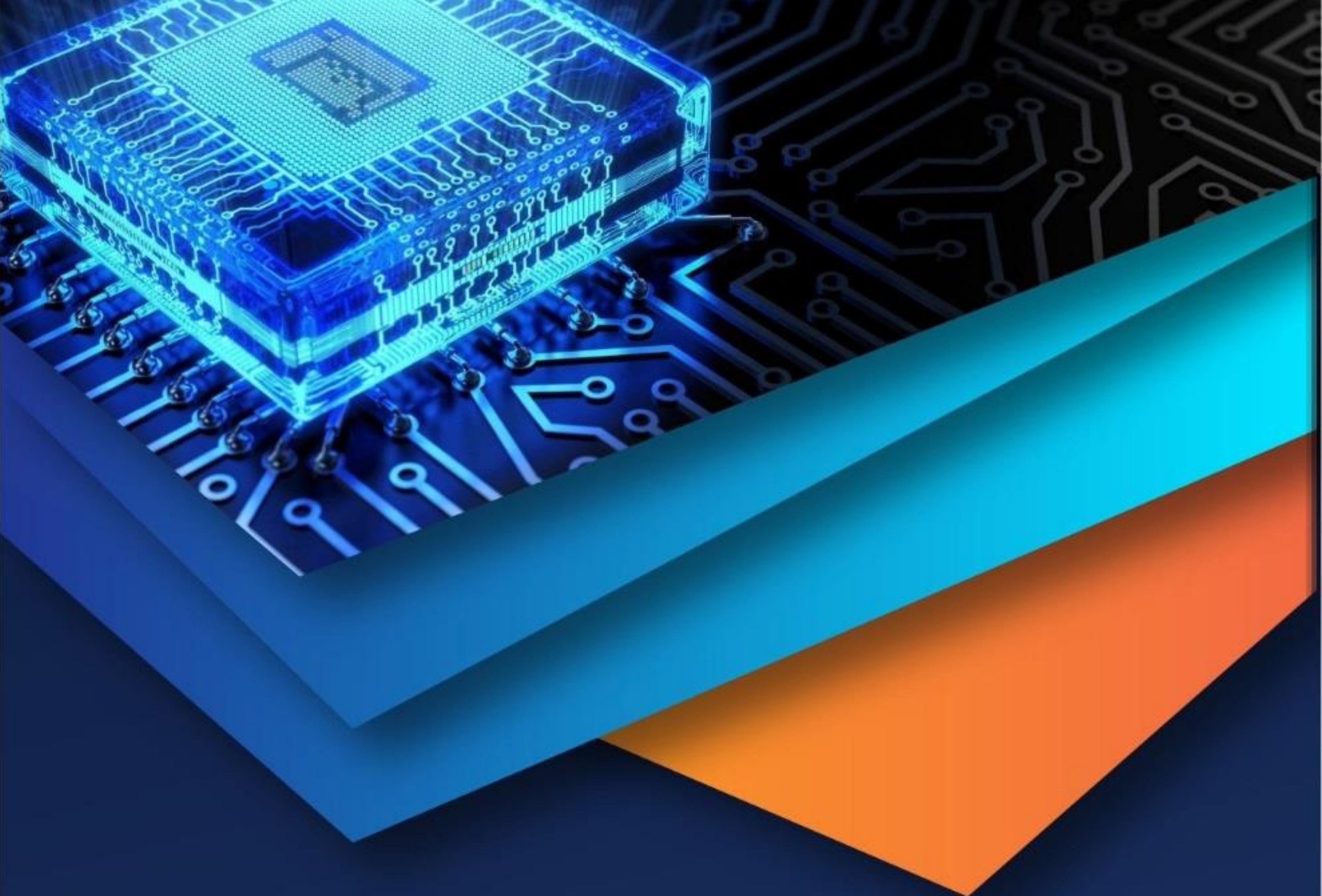
V. CONCLUSIONS

The biomechanics of human head neck system plays a very important role in understanding their responses to various sorts of impacts. Analyzing the head neck system as a whole according to the analytical methods were found to be tedious. Thus, such complex problems were solved using a new advanced technology called Finite element method. The development of Finite Element method proved to be a life saving factor for analyzing complex geometries. It includes various steps and procedures to solve the system. The developed finite element head neck models by various researchers were found effective in getting the responses. The head models containing skull and brain were studied and the neck model which depicts the cervical spine consist of seven vertebrae's and were modeled as solid elements and the ligaments between them as beam or spring elements. The head neck models proved to be effective in solving complex problems in a limited computational time.

REFERENCES

[1] Ward, Carley C. Finite element models of the head and their use in brain injury research. No. 821154. SAE Technical Paper, 1982.
[2] Ross, C.T.F. The finite element method of structural mechanics. 1st Eds. People Jiaotong Publisher, 1991. 1: 23-50
[3] Kaitai Li, Aixiang Huang, Qinghuai Huang. The finite element method and its application. 1st Eds, Xian'an: Xian'an Jiaotong University Publisher, 1992: 1-3
[4] Siyuan Cheng. The methodology of finite element method. Chongqing University Journal (Social science), 2001, 7(4): 61-63
[5] Yue X.F.，Wang L.，Zhou F. Finite element analysis on strains of viscoelastic human skull and duramater. Journal of Basic Science and Engineering，2008，16 (5): 686-694.
[6] Qiang X.， Jianuo Z. Impact biomechanics researches and finite element simulation for human head and neck. Journal of Clinical Rehabilitative Tissue Engineering Research，2008，12(48)：9557-9560.
[7] Yang, Bin, Kwong-Ming Tse, Ning Chen, Long-Bin Tan, Qing-Qian Zheng, Hui-Min Yang, Min Hu, Gang Pan, and Heow-Pueh Lee. Development of a finite element head model for the study of impact head injury. BioMed research international 2014 (2014).
[8] Tse, Kwong Ming, Long Bin Tan, Siak Piang Lim, and Heow Pueh Lee. Conventional and complex modal analyses of a finite element model of human head and neck. Computer methods in biomechanics and biomedical engineering 18, no. 9 (2015): 961-973.

[9] Yang, Bin, Li Bo Cao, Yong Tang, Ning Sun, Kwong-Ming Tse, and Heow-Pueh Lee. Modal and dynamic responses of the human head-neck complex for impact applications."Journal of Vibroengineering 18, no. 7 (2016): 4743-4755.
[10] Kim, Yoon Hyuk, Batbayar Khuyagbaatar, and Kyungsoo Kim. Recent advances in finite element modeling of the human cervical spine. Journal of Mechanical Science and Technology 32 (2018): 1-10.
[11] Bonifaz, E. A. An Elastic Finite Element Analysis of the C2-C7 Human Cervical Spine Segment. Biomedical Journal of Scientific & Technical Research 37, no. 1 (2021): 29041-29046.
[12] Silva, Afonso JC, Ricardo J. Alves de Sousa, Fábio AO Fernandes, Mariusz Ptak, and Marco PL Parente. Development of a Finite Element Model of the Cervical Spine and Validation of a Functional Spinal Unit. Applied Sciences 12, no. 21 (2022): 11295.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)