



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 11, Issue 10, October 2022

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.118

 9940 572 462

 6381 907 438

 ijirset@gmail.com

 www.ijirset.com



Image Processing of Human Spine CT Scan Image with Different Stress Analysis Using ANSYS Software

V.K.Buvanesvari¹, C.Sivaprakasam², P.Suresh³, U.Saravanakumar⁴, E.Punarselvam⁵, T.Aravind⁶

¹Assistant Professor, Department of MCA, Muthayammal Engineering College (Autonomous), Namakkal, Tamilnadu, India

²Associate Professor, Department of MCA, Muthayammal Engineering College (Autonomous), Namakkal, Tamilnadu, India ³Professor, Department of Mechanical Engineering, Muthayammal Engineering College (Autonomous), Namakkal, Tamilnadu, India

⁴Professor, Department of ECE, Muthayammal Engineering College (Autonomous), Namakkal, Tamilnadu, India

⁵Professor, Department of IT, Muthayammal Engineering College (Autonomous), Namakkal, Tamilnadu, India

⁶Assistant Professor, Department of CSE, Muthayammal Engineering College (Autonomous), Namakkal, Tamilnadu, India

ABSTRACT: Spinal cord is the most important and complex structure between the human body and the brain. The brain and spinal cord together make up the Central Nervous System (CNS). These acts as information passers between the brain and other parts of the body. It control and coordinates the fundamental framework of the entire body and also resembles as protecting shield for the medulla spinalis. Ergonomics is the science of refining the design of products to optimize them for human use and also sometimes called as Human factor engineering. In this current scenario there are plenty of ways are there to study the Anatomy and Biomechanical Physiology of Human spine. Most of those technique are based on stiff or animal tested knowledge. To be more specific, those techniques are not considering the matter of variation between the patients. In this study, a Three Dimensional (3D) view model of the spine is demonstrated using Computed Tomography (CT) which is a latest digital biomedical technology equipment. With this technique mistreatments could be avoided. Moreover using this methodology, the exact location and precise information about the spine are possible. So that the Doctors can prescribe the accurate medication and remedy to patients who suffer from any spine diseases. This paper acts as a root for enormous applications like analyzing the biomechanical performance of the spine, testing the spinal performance to check spinal is healthy or unhealthy and also to validate the planning phase of spinal implantation. To analyze the Human spine, mainly focused on the Lumbar vertebrae (L1- L5) which is a largest region of spine. Using the Ergonomics analysis of human spine, the stress -strain relationship is calculated. The entire analysis accomplished by a real time simulation software ANSYS.

KEYWORDS: Spine Disc image, Finite Element Analysis, CT scan, Patch Conforming Algorithm, Patch Independent Algorithm, Discretization, ANSYS.

1. INTRODUCTION

Finite Element Analysis (FEA) is the simulating the products and systems in a virtual environment and mainly used for detecting and resolving potential structures. FEA is the real-world application of the finite element method (FEM), which is used to estimate and resolve very complex structural, fluid, and complicated problems. FEA simulation software like ANSYS can be utilized in a wide range of fields like biomechanical, automotive and aeronautical industries. Finite Element Analysis (FEM) is a numerical technique for estimating the approximate solutions to boundary value problems for Partial Differential Equations (PDE). It is also known as Finite Element Analysis (FEA). It splits a large problem into smaller, simpler parts that are called Finite Elements. The simple



calculations that model these finite elements are then collected into a larger system of equalities that helps to model the entire problem. FEM then uses variation methods from the calculative methods of variations to provide a precise solution by minimizing an associated error function. This methods are primarily employed for locating the solutions of mathematical problems of physical reality. The arrangement approach of finite analysis method is build either in light of wiping out the differential condition entirely or rendering the Partial Differential Equation into an approximating arrangement, which are then numerically incorporated and utilized for standard procedures. The example for the above approach is the Euler's technique. Limited Element Modeling is efficient amongst many modelling techniques because this model investigate the most strong and broad flaws and progressively update the issues associated with flaw. Of this reasons it termed as rule hard to witness modelling. In the light of accessible hypothesis examination of thick-walled barrels are completed with limited component demonstrating the standard results. The Limited Element Modeling follows a numerical strategy that separates a components into little discrete components consisting of interconnecting hubs. Limited component examination reveals the solution for the model by deciding the conduct of every component independently. Hence, the result obtained been consolidated with the respective individual impacts to forecast the conduct of the entire model. The interconnecting hubs of the components tend to form the arrangement of one component to another, denotes an exact arrangement. FEA diagnose every component a few times, perhaps a large number of times, to accomplish an approximate solution. The correctness of FEA approximation is based on the quantity of components. Supplementary elements will enlarge the model's accuracy. However if the quantity of elements increase then corresponding arrangement time raises accordingly. Finite component investigation can be implemented with either 2D or 3D models. Preferably 3-D models provides a more precise examination as they incorporate the three dimensional view of the physical world. A 3-D model consisting of more hubs and elements that positively expand the arrangement time. Subsequently, it is often desirable to employ a 2-D model to reduce the arrangement time. A 2-D exhibit measures the missing third category measurement that acquire either axisymmetric or planar geometry. Axisymmetric geometry creates a model consuming round and hollow segments in which the parameters x , r , and θ , where θ is taken as 360° .

The Planar geometry yields a model applying certain Cartesian components x , y , and z , where z indicates the steady esteem for the entire model. The iterative way of FEA creates the investigation of models irrational but it gives an ideal way for PCs. In electromagnetic models, the modelling includes plotting of flux thickness, and giving assurance to circuit attributes. For example; voltage drop, resistance, reactance, and inductance. A small number of electromagnetic FEA bundles exist which tend to extend it from

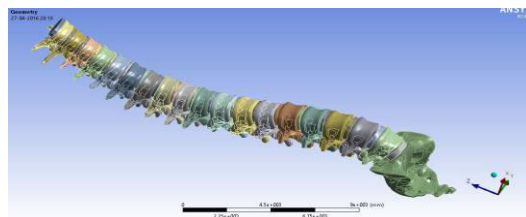


Fig.1: Diagrammatic illustration of Image Pre-processing

three dimensional bundles. For example some simulation software like ANSYS and Maxwell 3D are quite easier when compared to other 2-D package software like Maxwell 2D, Quick field, and FEM. Every FEA PC simulation process involve three sections namely; the pre-processing, examination, and post-processing. The initial section is pre-processing comprises of building the models from hubs, bends, and surfaces, establishing the limit conditions and square marks and fabricating the work. Examination section is the intermediate phase in FEA PC simulation process that computerized the model and consume the recommended conditions and computational methods. The final section in FEA PC simulation process is Post processing section that embraces the following tasks namely; perception, considering, and result analysis. The above described three stages are implanted in FEM model using simulation software ANSYS 14.5 device.



II. LITERATURE REVIEW

A basic knowledge of the spine's anatomy and its functions is vital to give solution for spinal disorders. Human spine has an extraordinary and complex anatomy. Spine does an extremely challenging functions on the human body. This study provides the thorough knowledge on specific anatomic structures such as vertebral structures and intervertebral discs of the spinal cord and additionally about nerve roots, joints, muscles and ligaments. The main physiology of human spine is to ensure the spinal line and exchange loads starting from the head and trunk to the pelvis. The three main functions of the spine are discussed below.

- Guard the spinal cord, where as nerve roots and several of the body's internal organs.
- Provide structural support and balance to maintain an upright posture.
- Enable flexible motion.

Finite Element Analysis (FEA) is a computerized digital method for forecasting how the a product reacts to real- World forces, vibrations, heat, fluid flow and other physical effects. FEM indicates whether a product would break, where out or work the way it was constructed. Finite Element Analysis has made an imperative commitment to the study and comprehension of the conduct of the spine.

This technique is the most appropriate way to analyze the spinal biomechanics due to the following benchmarks namely, 3D unpredictable geometry, non-homogenous material course of action, extensive complex stacking and development, and non-direct reaction including contact at aspect joints. Taking account all above constraints along with figuring forces and programming capabilities, resolved that FEM model is suitable to diagnose and examine the Spine related issues. Moreover, FEA has embedded as a part of spine research due to the following reasons:

- To examine the biomechanical behavior of the Human spine
- To gauge the spinal performance when it subjected to illness, degenerative changes, injury, maturing or surgery
- To investigate the impact of different spinal instrumentation on spine behavior
- To assist the strategy and development of new spinal attachments

FEM particularly analyze the region of human spine particularly lumbar spine. The same has proposed and reported in this paper. FEM model utilize. The reason behind to use this lumbar spine is pervasiveness of low back agony. The end goal is to inspect the influences of material property of spinal segments and vertebrae using FEM model.

III. METHODOLOGIES USED IN FEA

3.1 PREPROCESSING

The preliminary section is pre-processing stage. Preprocessing involves the preparations of data, such as nodal coordinates, connectivity, boundary conditions and loading and material information. The accuracy of the FE analysis is dependent on the work consumed, if higher request like cubic, quadratic and so on are consumed then the components are not utilized as expected. In this stage if all phases of preprocessing is done effectively then, then a better result will be produced instead of coarser result. At any instance if a state of consistent losses then it leads to the elongate thickness of the image and neglects to deliver a critical change in the results. If the expected state of consistence is achieved then, the work is said to be successful. This approach of refining the work and delivering the result is known as a "work joining" study or examination. Though numerous FE codes contain "fault assessments" of some sort, in that cases the work become more complicated so that the most solid method to be employed to find the accurate results. Because of these reasons it's necessary with a computer preprocessor which help with mesh plotting and boundary conditions plotting. Coarse networks often report anxieties in a model. It is better to have most extreme



reported weights to produce a precise results on a coarsely fit model. Thus, by implementing the above FE analysis phenomenon into the work then the feasible flawless outcome been produced as an end product.

3.1.1 MODELLING

The modelling stage of Finite Element Analysis is to define the association or connectivity. With reference to the initial step, the aggregate acquired an appropriate model. The diagram in fig 1 clearly shows the lumbar region is beneath the thoracic region of spine. The geometrical representation of vertebrae has been attained by arranging the images in a SliceOMatic fashion. In the Finite Element Analysis program design using simulation software ANSYS, few methodologies are newly created to enhance the reference quality and to get the required reference which similar to actual vertebrae. The volumes of each and every vertebra models are gathered for analyzing. The two vertebrae have been fixed using the plates in between them which is used to analyze the physiological ligaments. Moreover, as it is anticipated in most portrayed physiological cases, the spinal string is affixed into the medullar channel which is between the first and the second lumbar vertebra region of spine. The annulus filaments and the cartilaginous plates are independently performed. The vertebrae images are produced in accordance with the patterned pictures. The fitting lattice for the plate is directed the geometry region of the core and the annulus. Eventually, between first and the second lumbar vertebra are fitted using a spinal rope. Modular examination is the process of analyzing the dynamic properties of spine structures under vibration excitation technique. In basic mechanics, the primary aim of modular examination is to detect the distinguishing shapes and frequencies of different structure images among free vibration. A modular investigation records the characteristic methods of a framework work. It expand the request in accordance with recurrence and everything are numbered clearly. The examination results the characteristic methods of the discredited demonstration excluding the genuine constant framework. Anyhow, the discredited modes are nearer to the persistent ones and numbered modes boosts the components that are developed to establish the framework.

3.1.2 DISCRETISATION

Discretization is the process of isolating the model specimen into an equivalent system of many smaller bodies or units that are interconnected at point of common two or more nodes or boundary lines or surfaces. The precision in discretization performed from lower modes to increasing elevated modes constantly. Probably most elevated numbered modes are not practically possible subsequently they are in motions whose wavelengths are of blurry request from the varying length. In addition to this, the modular investigation have to be figured when the model is in static position since, it will provide methods of uncertain nature of that model. For a given mode, the mode shape encompasses the flawless and enhanced mean position in which a free degrees of opportunity have been incorporated into the modular investigation.

3.1.3 MESHING

Meshing has been furnished by using the fine components of Tetrahedron method. The Tetrahedron method is based on two algorithms namely; patch conforming algorithm and patch independent algorithm.

Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	200.0 mm
Initial Size Seed	Full Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	2.4180 mm



Statistics	
Nodes	31295
Elements	35881
Mesh Metric	None

Table 1. Tabular Description of Mesh Sizing and its statistical data

The following contents are the main essence of the algorithms for are explained in Tetrahedrons Methods

a) Patch Conforming Algorithm for Tetrahedrons Methods Control:

- It follows a bottom up approach in which meshing process performed from edge, face and then body
- Every faces and their corresponding boundaries are valued and meshed
- Provides Good outcomes to clean CAD geometries

b) Patch Independent Algorithm for Tetrahedrons Method Control:

- It follows a top down approach in which initially the volume mesh has been generated. Then, projected on to faces and edges to generate the surface mesh
- The faces and their boundaries (edges and vertices) are not necessarily valued. If they fall under an indicated tolerance, then they were subjected to load boundary condition or other object scoped to them, provided there is a named selection.
- Provides a Good results for gross de-featuring of dirty CAD geometries

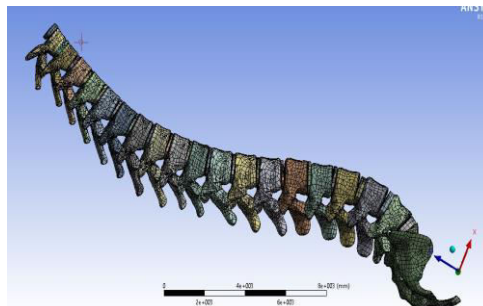


Fig 2: Pictorial Illustration of Spine Meshing

3.2 POST PROCESSING

The post processing stage deals with the demonstration of discovered results. In this section of finite element analysis the following thing are calculated namely; distorted configuration, mode shapes, temperature, and stress distribution. After successful computation of above specified units, they have to be visualized. For displaying enhanced post processing module of DTI information of the spinal rope, the custom based pipeline is utilized. Therefore, two vortex current approach and movement amendment approaches were analyzed using three-dimensional relative (3D-relative). Likewise, displayed another vigorous tensor-fitting strategy that reins for whole volume anomalies. Using vigorous tensor-fitting strategy the large 3D-relative enlistment enhances its information quality. At the same time, it can fail to record the processes because it’s a one-sided tensor evaluations. The proposed vigorous tensor strategy provides the appropriate minimal error recording predisposition and also produce more solid tensor assessments. Using the strong tensor strategy the better results are produced. Some of them are smart cutting or adjustments of spines and good medication for the vortex current. Strong tensor fitting yielded the best results. Also it expand the differentiation to-clamor proportion in finite analysis (FA) and diminished the intra-subject variety in finite element analysis (FEA). While expanding FA maps by around 30% and while demising FEA maps by around 18%. By this FEA maps higher nature and considered as superior qualification among dim and white matter devoid of expanding filter time and moreover maintains the picture-perfect quality with any multi-directional DTI information of spine rope.

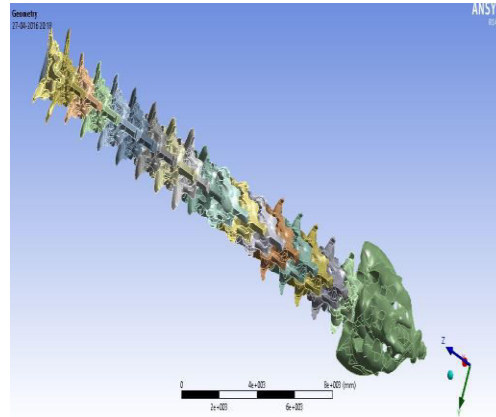


Fig 3: Diagrammatic Representation Image Pre-processing

For the practical implementation, the above specified strategy has one limitation, that is, the image of spine bone is broken down which reduces the natural quality of an image. To overcome this issue, an appropriate and effective common frequencies are to be examined in modular investigation phase.

3.3 BOUNDARY CONDITIONS

The Spine bone of Human beings has a capable to withstand 50 kilogram of weight regularly. Accordingly a sum of 490N acts from the vertical region of Cervical Vertebrae. Similarly the stomach weight of greatest 0.5 MP goes almost as the heap in the Lumbar Vertebrae. Spine bone is tightly bounded on the Sacrum and Coccyx territory region. It shows the joined stacking in the cumulative spine bone. The boundary conditions for the various cases use a settled base of the lumbar human unit (in the three headings) and provides the proper load analysis for each displayed case. For the illustration of pressure, a weight is fixed to the uppermost surface of the geometry to guarantee the homogeneity of the allied load. The weight is recorded by detaching every heap by the range situated on the primary lumbar vertebrate L1.

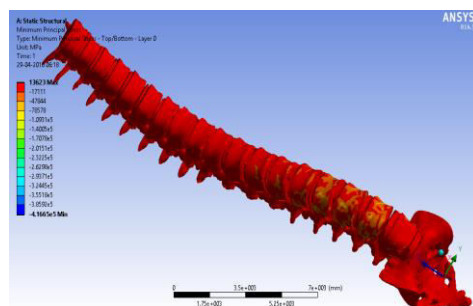


Fig 4: Diagrammatic Representation of Maximum Principal Stress Plot

IV. RESULT AND DISCUSSION

Finite Element Analysis (FEA) evaluates the overall strength and stiffness of the Human Spine bones and spine specimens. Apart from this, it additionally facilitates the investigation on mechanical behavior of bone and their mechano-biologic regulation of bone adaptation. Therefore, this paper creates a wide interest in Digital computational techniques for refining forecasts on fracture risks and also for pre-operative planning suggestions like vertebroplasty. The rapid growth of computing and image processing technique will provide the broader way to predict and estimate the boundary constraints and bone tissue material properties in advanced manner. The computational issues like



“failure load” and “bone strength” associated with bone remains controversial and require further validation. Table-2 depicts the results of different stress analysis of the spine lumbar vertebra when they subjected to different loads. Table-3 describe the different types of modal analysis modes when they subjected to the natural frequency of the human spine, under compression loading where the contours indicates the level of stress. From the tabular results it was found that the highest stress concentrations were at the adjacent lower posterior vertebral body. By this the Stress concentration will decrease the mechanical integrity of the bone and leads to creation of vulnerable fracture during trauma. In an entire spine, the vertebra region considered as critical area that tend to as axel point when additional load is loads on facet joints. Due to this action the effect of spinal occurs in human body. A larger distance between the facet joints and the vertebral body that tend to causes an increased bending effect and stress on human spine.

V. CONCLUSION

Based on the above results which are performed on bone spines, the study reveals that finite component strategy to be diagnosed at Lumbar region of L4 of a spine bone. The analysis to be performed on the lamina bone (L4 and L5) that originating from the pedicles curve. More specifically lumbar L4 bone diagnosing results greatly help to fix the problem associated with the spine. All the above discovered segment or regions of the lumbar vertebrae are considered as the potential break zones of the examined patient. The accuracy of the result depends on the observation of separation between the spinous procedures of the spine bones that rebuild using post-operation geometry. The results clearly exhibits that's the space into which the chamber was inserted in the pre-operation geometry reconstruction have a similar visualize prediction on the post-operation geometry phase. By using proposed finite component strategy, we can estimate the probability levels of bone removals and their stretching characteristics and also, provides a better clinicians knowledge about the conduct of the bones. The estimated outcomes of different stress analysis of the spine lumbar vertebra when they are subjected to different external loads or excitation on the human body delivers a severe consequences when the external excitation frequency matches with the natural frequency of the bone. From Table 3 it is discovered that the natural frequency for first six modes of Spine bone ranges from 0.822 Hz to 4.27.

REFERENCES

1. T.Rajesh Kumar, E.Vignesh, Dr.P.Suresh, Ergonomics Analysis of Functional Lumbar Spine during Manual Lifting of Loads Using ANSYS Software, International Journal of Innovative Research in Science, Engineering and Technology ISSN(Online): 2319-8753 Vol. 5, Special Issue 4, March 2016, pp(67-71)
2. Daniel Zewdneh, Seife Teferi Dellie, Tewodros Ayele, A Study of Knowledge & Awareness of Medical Doctors Towards Radiation Exposure Risk At Tikur Anbessa Specialized Referral And Teaching Hospi tal, Addis Ababa, Ethiopia Ethiopia,IOSR Journal of Pharmacy and Biological Sciences (IOSRJPBS) ISSN: 2278-3008 Volume 2, Issue 4 (July-August 2012), PP 01-05
3. Panjabi, M.M. (2003) Clinical spinal instability and lower back pain. Journal of Electromyography and Kinesiology, 13, 371-379. doi:10.1016/S1050-6411(03)00044-0
4. G. Floros and E. Dimopoulou, Investigating the enrichment of a 3D city model with various citygml modules, The International Archives of the Photogrammetry, Remote Sensing and Spatial Information Sciences, Volume XLII-2/W2, 2016 11th 3D Geoinfo Conference, 20–21 October 2016, Athens, Greece, pp(3-9).
5. Tomasz Lodygowski, Witold kakol, Marcin and Wierszycki, Three dimensional non linear finite element model of the human lumbar spine segment, Acta of Bio engineering and Biomechanics, Vol 7 ,No 2,2005 PP(17-28).
6. V Divya, M Anburajan, finite element analysis of human lumbar spine, 3rd International Conference on Electronics Computer Technology, (ICECT 2011) 978-1-4244-8679-3/\$26.00 C 2011 IEEE,PP(350-354)
7. Mary Ann McLaughlin, MD, MPH, Gretchen M. Orosz, MD, Jay Magaziner, PhD, Edward L. Hannan, PhD, Thomas McGinn, MD, R. Sean Morrison, MD, Tsivia Hochman, MS, Kenneth Koval, MD, Marvin Gilbert, MD, Albert L. Siu, MD, MSPH, Preoperative Status and Risk of Complications in Patients with Hip Fracture, JGIM, October 3, 2005, PP(219-225)
8. Arpan Gupta, Kwong Ming Tse, Finite Element Analysis On Vibration Modes Of Femur Bone, roc. of Int. Conf. on Advances in Mechanical Engineering, AETAME, Elsevier, 2013, PP(827-831)



9. M. A. Adams, D. S. McNally, P. Dolan, 'stress' distributions inside intervertebral discs, vol. 78-B, No. 6, November 1996, PP(965-972)
10. An Adaptive Finite Element Method for Minor Shape Modification in Optimization Algorithms, IEEE transaction on Magnetics, Volume: 44, Issue: 6, June 2008, PP(1202-1205)
11. Ankit Kumar, Mohit Kumar, Manisha Singh, Vijay Kumar Meena, Rajesh Kumar, Finite element analysis of customized bone plate for distal femur fracture, Recent Advances in Engineering & Computational Sciences (RAECS), 2015, IEEE, 2nd International Conference, INSPEC Accession Number: 15936099, DOI: 10.1109/RAECS.2015.7453381
12. J. Anish Jafrin Thilak, N. Subramani, T. Rajeshkumar and Dr. P. Suresh, Ergonomics Analysis of Functional Lumbar Spine during Manual Lifting of Loads Using ANSYS Software, ISSN(Online): 2319-8753 Vol. 5, Special Issue 4, March 2016, pp(67-71)



INNO SPACE
SJIF Scientific Journal Impact Factor
Impact Factor: 8.118



ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 **9940 572 462**  **6381 907 438**  **ijirset@gmail.com**



www.ijirset.com

Scan to save the contact details