International Journal of Engineering Research
Volume No.4, Issue No.12, pp : 648-651

ISSN:2319-6890)(online),2347-5013(print)
01 Dec. 2015

Analysis of Stresses in Mandible and Skull under Angular Impact

Dr. Shobha E S[1], Dr. Suresh Nagesh[2], Dr. H P Raghuveer[1], Mr. Vinay K S[2]
[1] Dayanand Sagar college of Dental Sciences
[2] PES University, Cori

Abstract : Finite element analysis has gained a significant attention in the field of biomechanics. The versatility of finite element analysis is the major reason behind its implementation and adoption has made the study of biomechanics much easier and simpler. FEA, as a tool has helped in analyzing the various organs in human body in different loading conditions. Be it Orthopedic,Maxillofacial fractures,Orthodontics, Prosthodontics, hemodynamics, it is finding its applications in all areas of biomechanics. In this paper, one such study has been carried out to understand the behavior of skull and mandible under inclined impacts. Skull is one of the highly complicated structures made up of bones in the human body.A lot of research is being carried out throughout the world for studying the skull structure in multidisciplinary level. Research is also being carried out in treatments by effectively using FEA, developing new methodologies and materials for treatment in the field. Here, the model of the skull along with mandible is constructed using the CT scan data using which finite element model is created and impact analysis has been carried out to find out the stresses, strains and energy absorption of the skull material. Apart from the analysis, attempt has been made in this paper to suggest the optimum locations for placing implants to optimize treatment in trauma patients.

Keywords: Skull, implants, trauma, FES, LS-Dyna, impact, mandible

I. INTRODUCTION

Human skeleton is a perfect example of a mechanical structure. It acts as a frame for human body to support the different and highly sensitive organs such as the brain, eyes, nose, mouth, etc... It thus protects the various sensitive organs of the human body. Human Skull is one of the organs in the body which is highly vulnerable to injuries. Hence in most of the surgeries that are carried out around the globe, craniofacial surgeries form a significant percentage [1]. Continuous research has been going on for past couple of decades for preventing and treating of these injuries. Multidisciplinary research is also carried out in this regard. Implementing finite element study is one such part where significant amount of research is being carried out to understand and predict the behavior of the different organs associated with the skull under different loading conditions, Since Finite Element Analysis (FEA) is a robust numerical solution that can cater to varied complexity of geometry, material properties and loading conditions, FEA is being found to be a effective tool for prediction and solution. FEA is the most used technique in various fields such as automotive, aerospace etc for its versatility in solving different problems in structural mechanics, thermodynamics, acoustics, electromagnetics, flow and most recently biomechanics. Particularly FEA is finding its applications in prosthodontics where the stresses and forces on implants and its surroundings are calculated.

Since the biomechanical structures are complex in terms of their geometry and the material they are made of, it is a challenging task to develop a numerical model and to calculate responses like stress, and displacement. Finite Element Method (FEM) approach is the best way available to solve these kinds of problems. Sebastien Roth et al [2] studied the paediatric head impact. The authors in [2] considered a newborn head for their simulation of impact analysis. Impact analysis was carried out and intra cerebral response was analyzed. A new injury criteria was developed using these numerical simulations for minor skull fracture. The Finite Element (FE) model of the head in [2] included membranes, Cerebrospinal fluid (CSF), Scalp, Sutures and skull structure. Corresponding material properties were applied to model their physical behavior. Different mesh sizes such as 1.2mm, 2.5mm, 5 mm were compared for accuracy and cycle time for computation. Finally 5mm mesh was used, as the variation of results was only 7% and computational time was significantly lesser when compared to other two mesh sizes

Significant research has also been carried out by researchers [3] [4] in finding out the elastic modulus, density and other properties of the different bones and organs of the human body [3] [4]. This has helped to a great extent in the use of Finite Element Analysis (FEA) in analyzing the response of different bones for different loading conditions. Finite element simulations to mimic fetal head impact are gradually increasing over the years to study mechanical birth injuries [5] [6].

II. METHODOLOGY

Axial slice Computed tomography (CT) scan of a 25 year old male without any craniofacial abnormalities was taken at 0.6mm interval. The output of CT scan thus obtained will be in the format of .Dicom. This data is then processed using tool called Simpleware and converted to STL (Stereo Lithography) format or any other graphical formats which can be imported by Hypermesh 12.0. Hypermesh 12.0 is a software used for developing complex finite element models from computer aided design (CAD) or graphical data. In general, the STL model obtained will be having a lot of unnecessary data that needs to be cleaned up. So, surface extraction is again done on the STL data to obtain only relevant surfaces necessary for further processing and analysis. Geometry clean up is carried out in Hypermesh using clean up tools. Once the required data is ready, the geometry is meshed with triangular elements to obtain a triangular mesh. Triangular mesh is used to capture all the complexities of the geometry for the skull structure. After this step, since the skull model is of irregular geometry, the mesh needs to be cleaned up which is carried out to obtain good quality mesh so that accurate results can be obtained. Once the mesh with good quality is obtained, it is then meshed using tetra elements. The mesh data is then exported to LS-Dyna3D where material properties, loading and boundary conditions are updated to the model and solved to obtain the desired results. Non-linearities in terms of material, geometry and contact are then modeled using LS-Dyna3D and considered during the subsequent analysis.

The results are then analyzed to check the areas of failure and hence the locations for positioning the implants are suggested. Figure 1 below shows the methodology followed in this work.

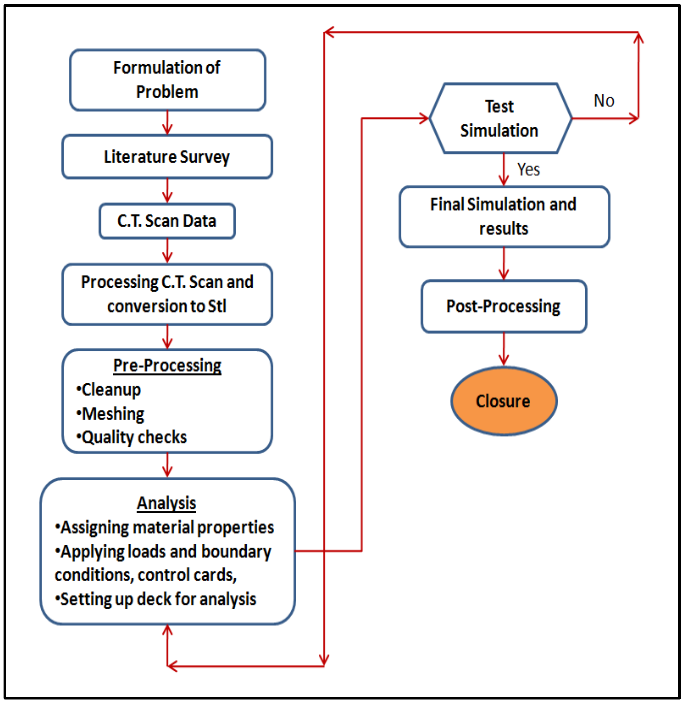

Figure 1: Flowchart of methodology followed

C T Scan

X-ray computed tomography uses computer processed combinations of X-rays taken from different angles to produce cross sectional images of the scanned area. These are called as slices. These slices are taken at 1mm interval for the study so that the model obtained will be of high fidelity. This CT scan data is surface data of the region that is scanned. This data is then processed using 3D slicer [7] / Simpleware [8] and converted to the .STL format. This is a stereo lithography file format which is used extensively for 3D printing, generating CAD data for computer aided manufacturing.
The STL file thus obtained is then imported into Hypermesh 12.0. The data from the .Stl files will be triangulated surface data which Hypermesh reads as triangular mesh. The triangular mesh imported usually will be of low quality and hence need to be cleaned. The surface is extracted from the data and geometry cleanup is carried out in Hypermesh. Then the surface mesh is generated using triangular elements. The below image shows the CAD data that is obtained from CT scan

Figure 2: CAD data

Figure 2 above shows the CAD data that is obtained from C.T scan data which can be used for the analysis

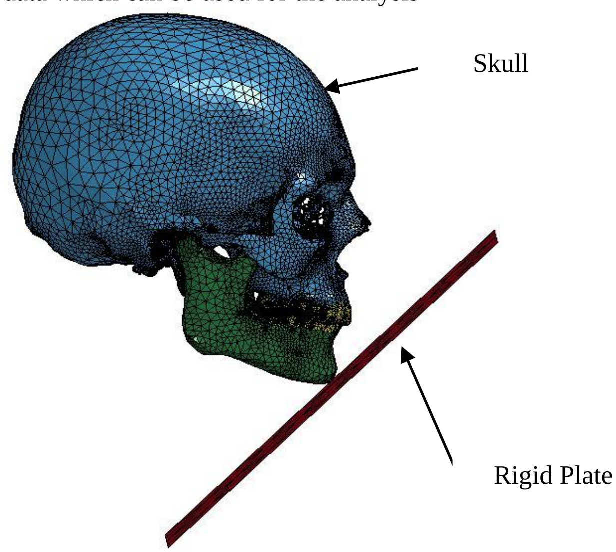

Figure 3: Finite element Model

Figure 3 shows the finite element model which is generated from the CAD data.

The number of elements generated directly from the CT scan data was around 19lakhs. This number is large as it takes a lot of computational time and disk space. It is then re-meshed and brought down to approximately 7.5Lakh elements without compromising the quality of the model. The finite element model shown in the above figure 3 consists of 7,61,978 elements and 1,91,662 nodes and weighs about 1.8Kgs.

Material properties

Once the finite element model is ready, suitable material properties have to be applied to obtain the desired results from the next step of simulation. Extensive literature survey [9] has been made to obtain the material properties for different regions such as maxilla, mandible etc. Maxilla, mandible and teeth portions are assigned with suitable material properties. The below table 1 shows the material properties considered for different regions of the skull

International Journal of Engineering Research
Volume No.4, Issue No.12, pp : 648-651
ISSN:2319-6890)(online),2347-5013(print)
01 Dec. 2015

Sl no	Component	Young's modulus (GPa)	Poisson's ratio	Density (tonnes/mm³)
1	Mandible	20.3	0.3	2e-9
2	Teeth	83	0.3	2.1e-9
3	Maxilla/Skull	20.3	0.3	2e-9

Table 1: Material properties different regions of skull

III. LOADS AND BOUNDARY CONDITIONS

To understand the response of the mandible and skull, an impact analysis is performed by applying an initial velocity of 1000mm/sec (3.6Kmph) to the skull model and made to impact with rigid plate which is inclined at an angle of 45 degrees to the mandibular symphysis region.

Figure 4 shows the constraints and velocity conditions that are applied to the model. The wall is made rigid and constrained in all directions and skull is given initial velocity to impact with the wall.

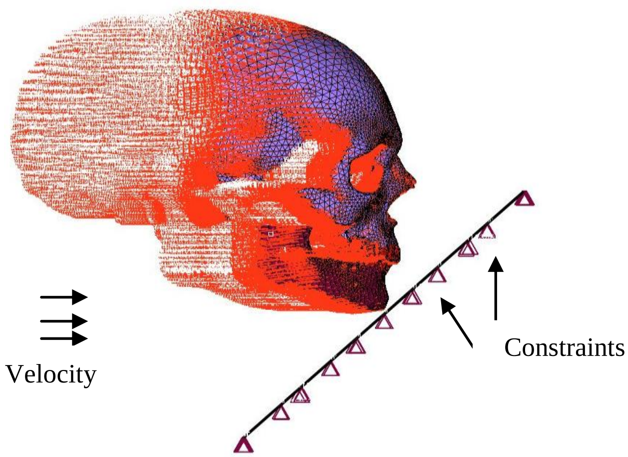

Figure 4: Load and boundary conditions

IV. RESULTS AND DISCUSSION

The 3-dimensional non-linear impact analysis is performed using LS-Dyna3D. Impact being an instantaneous event, and typically happens within 8milli seconds. Hence the simulation is performed for the same total time period and the output is obtained at 0.2milli seconds interval.

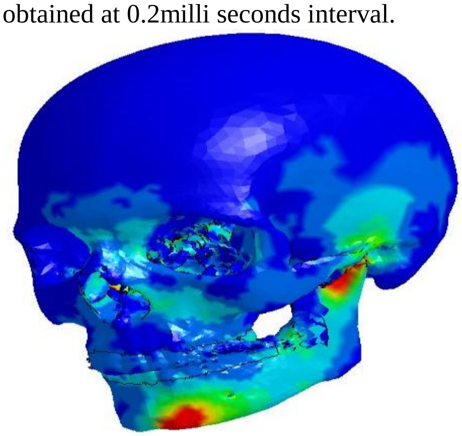

Figure 5: vonMises stresses at 3ms

The above figure 5 shows the contour of vonMises stresses that are developed in the different regions of the skull at 3ms.

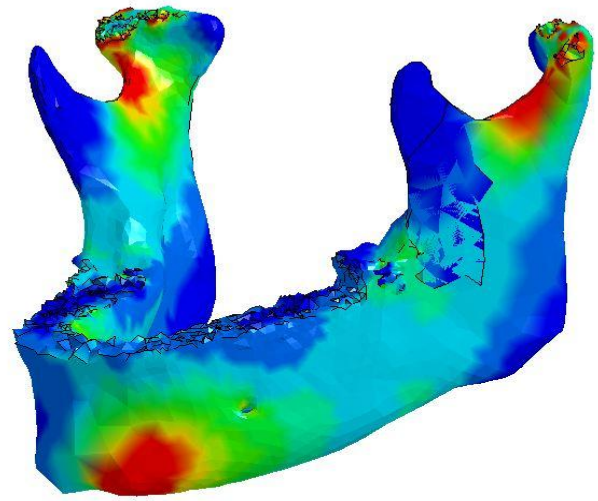

Figure 6: vonMises stresses in mandible at 3ms

The figure 6 above shows the stresses developed in mandible in the due course of impact at 3ms for an initial velocity of 1000mm/sec. Since it is a direct impact, the stresses are concentrated more at the parasymphysis region of the mandible. Also it can be seen that the stresses are more at the condylar region as well which is a weak zone for higher loads.

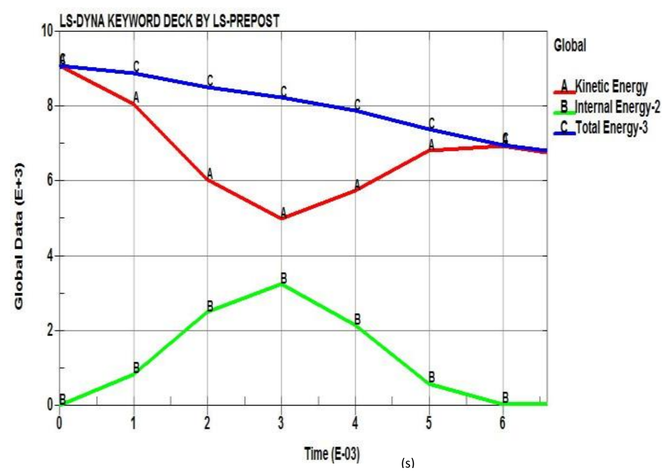

Figure 7: Energy plot with respect to time

Figure 7 shows the kinetic energy, internal energy and total energy plots for the model with respect to time for a velocity of 1000mm/sec. As expected the kinetic energy decreases and the internal energy increases which means that the skull starts taking the impact loading. Also around 3 milli seconds, the increase in kinetic energy is seen, because the skull rebounds back and gains some kinetic energy. Some energy is also lost in the fracture of the skull which is not shown in this plot. This will get added to the total energy plot and could be significant.

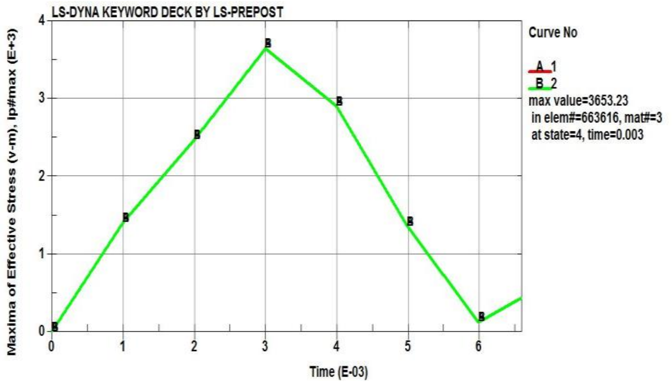

Figure 8: vonMises stress history plot

The above figure 8 shows the vonMises stress history plot for skull for the given velocity. As can be seen from figure 7, the maximum vonMises stress that has developed during the impact is of 3653MPa. This is much higher than the yield stress for the skull and mandible material which indicates that it has undergone fracture

V. CONCLUSION

The stresses are high in the para symphysis region and condylar regions of the mandible due to the direct impact forces suggesting fixation in these regions in case of fractures. Further work is ongoing to run the same simulation with implants to further optimize the size and location of the implants for early recovery and least post-operative complications.

VI. REFERENCES

i. *Horgan TJ, Gilchrist MD, Creation of 3 dimensional finite element models for simulating head impact biomechanics. IJCrash 2003,8(4):353-366*

ii. *Sebastien Roth, Jean sebastien-Raul, Remy Willinger-ELSEVIER journal. Finite element modeling of paediatric head impact: Global validation against experimental data. University de strasbourg, Strasvourg, France*

iii. *Jill Peterson, Qian Wang and Paul C. Dechow- Material properties of dentate Maxilla. Department of biomedical sciences, Baylor college of dentistry, Texas A&M university system health science center, Dallas, Tex*

iv. *Ervin K, Sabine G, Paula P, Bernd G, Anatomy based facial tissue modelling using the finite element method. San Francisco, CA 1996 OCT 27*

v. *Jean-Sebastien Raul, Daniel Baumgartner, Remy Willinger, Bertrand Ludes- Finite element modelling of injuries caused by a fall-Springer verlag 2005*

vi. *R.C. Van Staden, H. Guan, Y.C. Loo, School of engineering, Griffith University gold coast campus, Australia- Application of finite element method in dental implant research*

vii. *3D slicer software*

viii. *Simpleware software*

ix. *P. Maurer, S. Holweg, W-D Knoll, J. Schubert- Study by finite element method of the mechanical stress of selected biodegradable osteosynthesis screws in sagittal ramus osteotomy. Department of oral and maxio-facillary surgery, Martin luther university, Halle; Department of engineering, MArtinluther university, Merseburg, Germany*

x. *T.M.G.J van Eijden- Biomechanics of the mandible-Department of functional anatomy, academic center for dentistry amsterdam (ACTA), meibergdreef 15, 1105 AZ amsterdam, The netherlands*

xi. *Matthew.T.Davis, AndreM.Loyd, Han-yu Henry Shen, MauraH.Mulroy, RogerW.Nightingale n, Barry S.Myers, CameronDaleBass- The mechanical and morphological properties of 6 year old cranial bone, Department of Biomedical engineering, Duke university, Box 90281, Durham, NC 27708-0281, United states-Journal of biomechanics.*

xii. *P.K. Nayak, A K Mahapatra, Sanjay Gandhi post graduate institute of medical sciences, Lucknow -Primary reconstruction of depressed skull fracture - The changing scenario - Indian journal of neuro trauma*

xiii. *Kalyanaraman S, Ramamurthi B: An analysis of 3000 cases of head injury. Paper presented at the fifth Asian Federation Congress of the International College of Surgeons, 1973.*

xiv. *A.E. Abdulai, MI Iddrissu, TK Dakurah-Cranioplasty Using Polymethyl Methacrylate Implant Constructed from an Alginate Impression and Wax Elimination Technique- Ghana medical journal,Ghana Med J. 2006 Mar; 40(1): 18–21*

xv. *Horgan TJ, Gilchrist MD, Creation of 3 dimensional finite element models for simulating head impact biomechanics. IJCrash 2003,8(4):353-366*

xvi. *Richmond BG, Wright BW, Grosse I, Dechow PC, Ross CF, Spencer MA, Strait DS. Finite element analysis in functional morphology. Anat Rec A DiscovMol Cell Evol Biol. 2005 Apr;283(2):259-74*

xvii. *Van Essen NL, Anderson I A, Hunter P J, Carman J B, Clark R D. Anatomically based modelling of the human skull and jaw. Cells Tissues Organs 2005;180:44-53*

xviii. *Autuori B, Bruyère-Garnier K, Morestin F, Brunet M, Verriest JP. Finite element modelling of the head skeleton with a new local quantitative assessment approach. IEEE Trans Biomed Eng. 2006 Jul;53(7):1225-32.*

xix. *Pinnoji PK, Mahajan P. Finite element modelling of helmeted head impact under frontal loading. Sadhana Vol.32, Part 4, August 2007, pp445-458*

xx. *Rangan V, Raghuveer HP, Rayapati DK, Shobha ES, Prashanth NT, Sharma R. The influence of stress distribution in four different fixation systems used in treatment of mandibular angle fractures—A three-dimensional finite element analysis. Oral Surg 2013;6:186-192*

xxi. *ES Shobha, HP Raghuveer, Suresh Nagesh, Dilip Kumar Rayapati, NT Prashanth, Vinod Rangan 3D Finite Element Technology and Its use in Craniofacial Injuries, World Journal of Dentistry, October-December 2014;5(4):223-228*