BIOMECHANICS OF CRANIOFACIAL FRACTURES: A SIMULATION STUDY

Dr. Shobha E S[1], Prof. Dr. H.P. Raghuveer[2], Dr. Suresh Nagesh[3], Mr. Madhava M C[4]

1 Professor, Dept. of Oral and Maxillofacial Surgery, Dayananda Sagar College of Dental Sciences, Bengaluru, India.

2 Dean, Professor and Head, Dept.of Oral and Maxillofacial surgery, Dayananda Sagar College of Dental Sciences, Bengaluru, India.

3 Chair Professor in Computational Mechanics, CORI (Crucible of Research and Innovation), PES University, Bengaluru, India

4 Research Assistant,CORI (Crucible of Research and Innovation) PESUniversity, Bengaluru, India

ABSTRACT:

Objective of the current research is virtual simulation of Cranio facial fractures under impact to understand its Biomechanics. In particular the focus of this work is on the complex part of a human body, the human skull. One of the emerging areas of applications of computational bio-mechanics is to understand the behavior of the skull during a traumatic injury, such as head impact during accidents.

In this study a 3D finite element model of the skull is created using CT scan data. All complexities of the skull geometry are simulated using ABAQUS. This numerical model is then subjected to frontal, lateral. vertical, occlusal and angulated impact conditions. Impact analysis is done and weak areas susceptible to fracture and hence failure are identified. Further fracture fixations of various designs and materials used in craniofacial fracture fixations are placed in different fracture situations in the virtual model and subjected to different impact conditions. This will enable the study of fracture and stability of the fracture of the skull under cranio facial fracture conditions.

Results from the analysis can be used to come up with optimum locations of implants for different type of impact situations. This is expected to complement the existing treatment methodologies used by surgeons. Further, appropriate knowledge of fracture biomechanics can be used to develop safety measures in automobiles to prevent and reduce incidence of facial injuries.

Keywords: Skull, Craniofacial fractures, trauma, FEA, Abaqus.

1. INTRODUCTION:

The surge to understand the biomechanical behavior of the human body structures especially the complex human skull has led to the development of high fidelity human skull models.. Research is being carried out towards developing high fidelity human models, especially of the human skull and related organs, to understand the bio-mechanical behavior of such structures. In our research study, attempt is made to understand and numerically predict the resultant stress and energy fields in representative cases of craniofacial injuries. Different fixation systems have been used including the stress evaluation and stress shielding effect of titanium miniplates and monocortical screws used for the treatment of fractures resulting from such injuries. Experimental studies have been performed on animals, cadavers and crash dummies to calculate the forces, accelerations and displacements (Nahum et al. 1977). Cadavers have many advantages as they have the same geometry and mass but differ in texture .On the other hand, they are not widely available due to their costs and ethical reasons. Development of the Finite Element Method (FEM) brought many advantages to biomechanical researches. FEM is a numerical technique for finding approximate solutions to boundary value problems. It is also referred to as Finite Element Analysis (FEA) [14]. FEA methodology involves discretizing a large domain into smaller, simpler sub-domains that are called finite elements. The governing equations that model these finite elements are then assembled into a larger system of equations that models the entire problem along with appropriate loads and boundary conditions (constraints). Introduction of FEM has encouraged researchers to develop more accurate 3D models. The early models were only simple geometries simulating head and brain, but as the technology improved more accurate models are being created.

In the current study, a high fidelity finite element model of a 25 year old male skull is developed. The particular study involves understanding the behavior of this human skull under different impact conditions. Frontal impacts are quite common during the crash of a vehicle with another vehicle. Typically the head hits the steering or whips on to the side windows, causing significant damage to the skull. Such injuries can also happen when a person on a two wheeler, encounters a fall during an accident etc. The idea of a virtual simulation is to understand the stress and energy distribution for various types of such events to identify the possible areas of fracture and critical failure. This will provide information ahead of time to the medical practitioner to focus on areas of prime concern for prognosis rather than go by mere judgment and experience. The study involves considering the impact of virgin skull without any fixations. Then, critical areas of failure are identified. The simulation is done next, with fixations in areas of fractures created in the craniofacial region to understand the effect of the fixations. Such a virtual simulation process developed would help understand the injuries under many scenarios of impact and come up with the minimum number of fixations and their locations. Thus Impact simulations both on the virgin skull and the one with fixations is performed. This will also help further optimize such a treatment under multiple impact conditions.

The study includes validation of the numerical results obtained from the current study with any available data. A biomechanical model of human skull is presented in this study. The model allows us to understand the response of the skull to the external impact conditions and hence helping in better treatment in craniofacial surgery . A realistic model of the skull can be used to increase the accuracy of the results significantly

Three-dimensional radiographic imaging with Computer Tomography (CT) has become the most important source of input for medical modeling. Imaging in combination with quantification of the mechanical properties of bone with computerized solutions such as Computer-Aided Design (CAD) and Finite Element Method (FEM) has contributed to novel approaches which have improved their understanding.

2. MATERIALS AND METHODS:

A 25 year old patient healthy male patient who has no genetic, endocrinal or any systemic disease with full complement of dentition and who has not undergone any previous jaw surgeries and no previous history of trauma was taken up for the study. Spiral CT scan of the skull was taken with an X force/SH spiral CT scan machine. CT images of skull/craniofacial skeleton in Dicom format was taken at 0.6mm interval.

Sections were in Dicom format. Dicom, digital imaging and communication of medicine are a neutral image format basically for medical imaging purposes (figure 1). A three dimensional analytical model of the mandible and maxilla for analysis was developed from the scanned images recorded from the patient selected for the study.

Construction of 3D model - The dicom format CT scan images were then imported into medical modelling freeware software called 3D slicer (figure 2). This software is an interactive tool for the visualization and segmentation of CT images and 3D rendering of objects. The output from the 3D slices is then converted into a 3D model that can be used for creation of finite element or numerical models of the actual structure (human skull and associated components). Therefore to start with CAD models (figure 3) in steriolithographic (stl) format has been created from the 3D slices. This in turn is converted to an IGES format (International Graphic Exchange System), a mutual format form. The IGES format is imported into a finite element pre-processing software Hypermesh to further create a high fidelity model of the skull to be used for stress analysis. The mesh data is then exported to Abaqus where material properties, loading and boundary conditions are updated to the model. Nonlinearities in terms of material, geometry and contact are also modeled using Abaqus and considered during the subsequent analysis

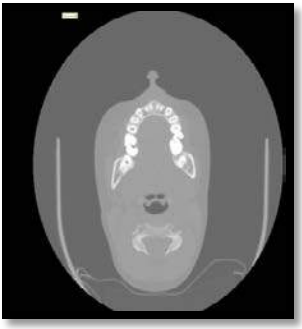

Figure 1: CT image

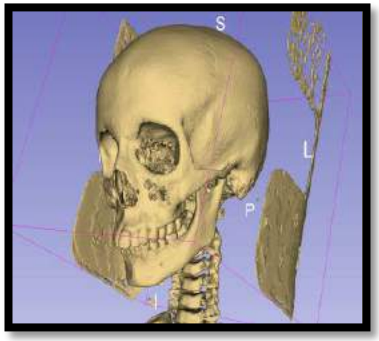

Figure 2: 3D slicer model

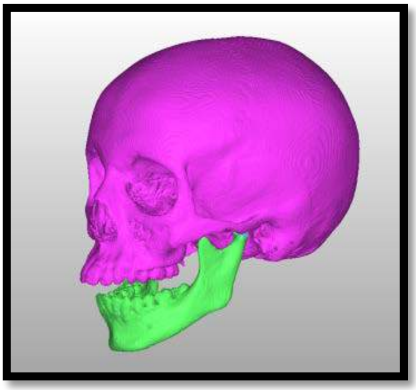

Figure 3: CAD model

Figure 4: CAD model from CT scan

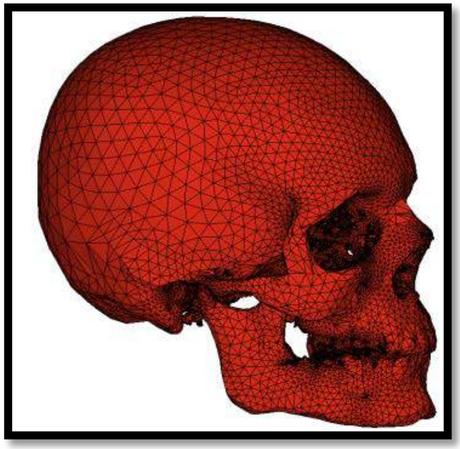

Figure 5: Finite Element Model generated from the CAD model

Finite element model of the skull with all its complexities has been created. The skull is modeled with shell elements. Actual material properties of different parts of the skull are incorporated into this model. The number of elements obtained from the CT scan data was around 1.9million. 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 0. 75 million elements without compromising the quality of the model. The final finite element model consists of 7,61,978 elements and 1,91,662 nodes and weighs about 1.8Kg.

Standard quality checks are then performed on this finite element model to ensure high fidelity of the numerical model. Typical checks done are on the finite element model w.r.t. internal angle, Jacobian, skewness and any initial penetrations. Once the finite element model passes the quality checks, it is then made ready for further simulation. Figure 6 shows the model of skull impacting rigid surface

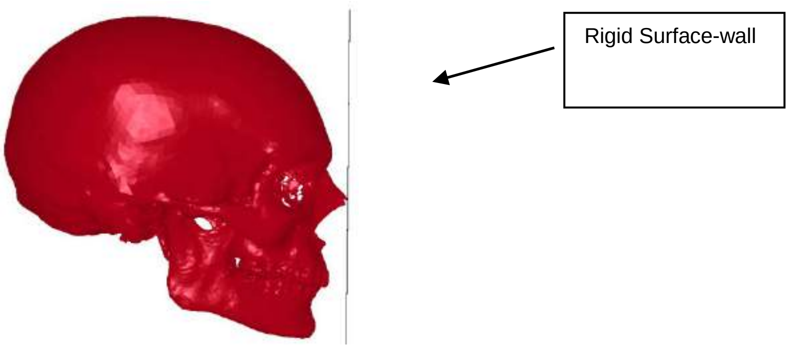

Figure 6: Model of a Skull Impacting a Rigid Surface

Table 1: Material properties used

SL NO	MATERIAL	YOUNG'S MODULUS (GPa)	POISSON'S RATIO	DENSITY (TONNES/mm^3)	YIELD STRESS (MPa)
1	SKULL	20	0.33	2E-9	100
2	CORTICAL BONE	13.7	0.33	2E-9	100
3	TEETH	84	0.33	2.2E-9	150
4	TRABECULAR BONE	1.37	0.22	0.99E-9	8
5	STEEL	210	0.33	7.83E-9	250
6	TITANIUM	110	0.33	4.8E-9	225

The figure 7 below shows the inputs that are to be given in this material model in ABAQUS.

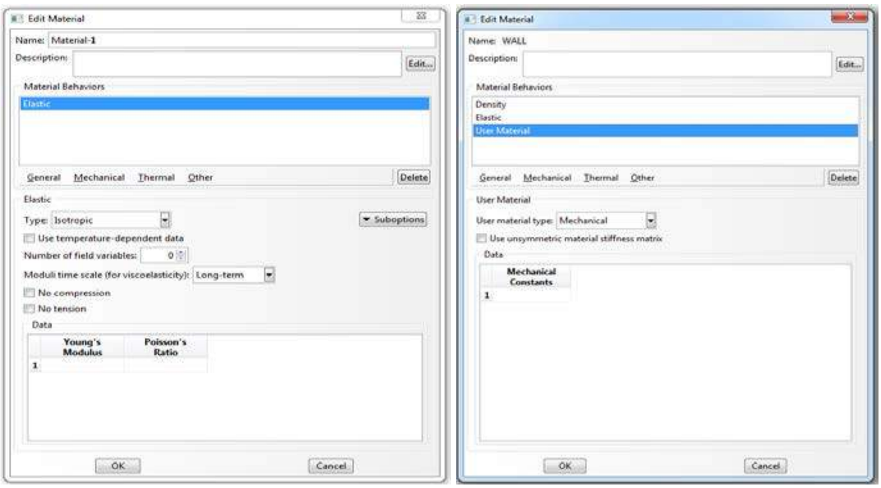

Figure 7: Elastic material model in ABAQUS Rigid material model in ABAQUS

Contact conditions:

In Abaqus general contact (explicit) was used to establish the contact between skull and wall during impact.

Loads and boundary conditions:

Once the model is updated with suitable material properties and contacts, the load and boundary conditions are applied to the model to understand the behavior of the components for various impact situations.

Initial velocity varying from 100mm/sec to 6000mm/sec is applied to the skull and made to impact with the rigid wall. The wall is constrained in all directions and is made rigid since the field outputs in the wall are not of interest. The stresses were studied within 10ms duration of the impact. Output of various responses are obtained at every 1.2ms interval.

The Figure 8 below shows the loads and boundary conditions that are applied to the model

Figure 8: Load and boundary conditions

.

In addition typical fractures for various scenarios of impact were created in the craniofacial region. These finite element models along with the fixations are being studied to understand the stress and energy fields under various conditions. In clinical situations as well ,craniofacial fractures are fixed with different types of fixations during surgery in the region of fracture. In this work similar type of fixations were chosen for the analysis purpose. These fixations or plates were fixed at the different regions of the skull with respect to the type of fracture. The following figure 10 gives the details about fixations which were used**.**

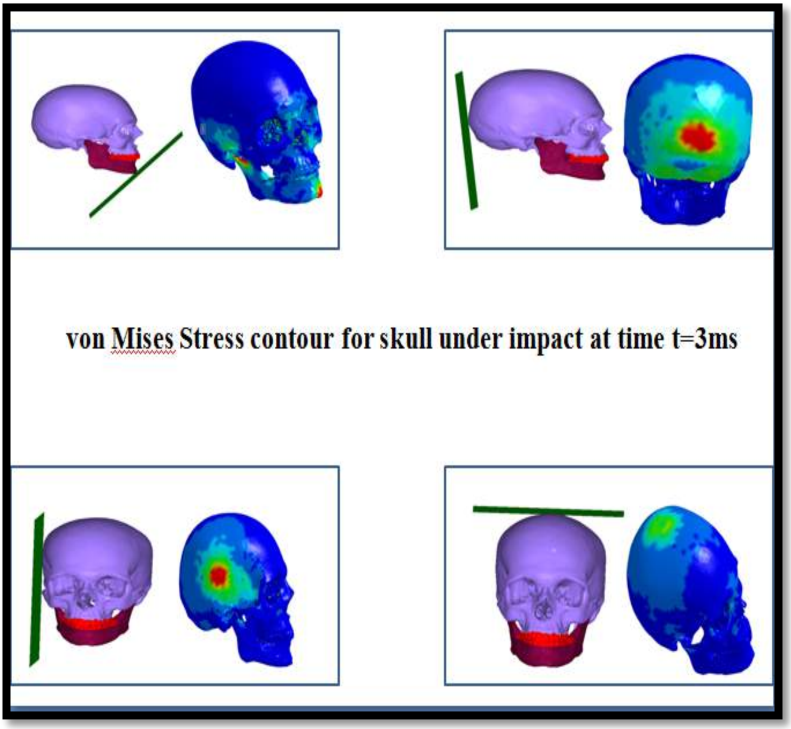

Figure 9: Different Impact conditions

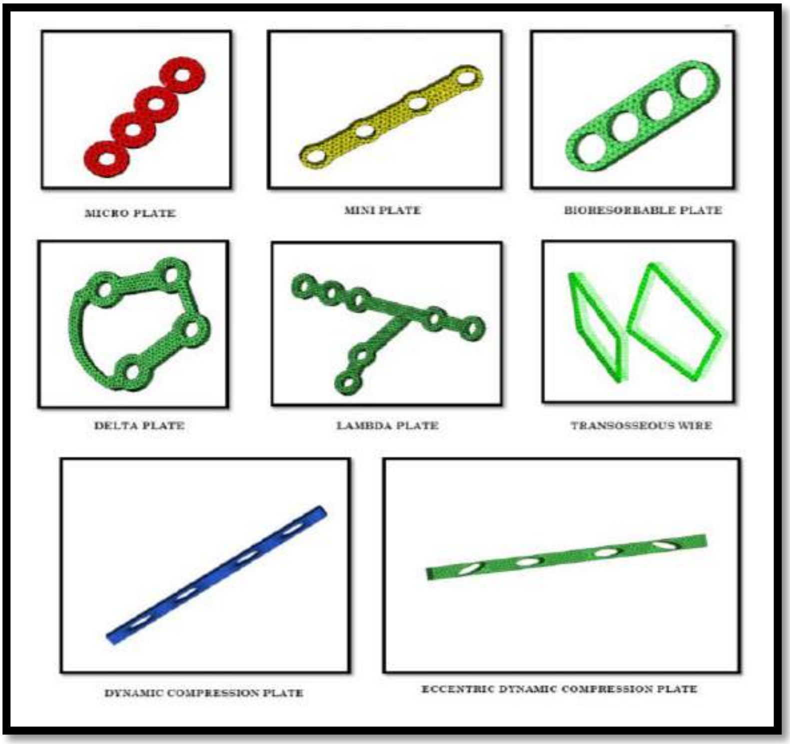

Figure 10: Types of Fixations

In particular output from a few representative simulations are shown below:

- **FRONTAL BONE FRACTURE WITH SS PLATES UNDER FRONTAL IMPACT**

This is the case where there is a fracture in the frontal bone subjected to frontal impact of the skull as shown in figure 11 below. A steel fixation (1.2 mm micro plate) is also modeled and placed at the fracture location. Figure below shows the stress distribution for this particular condition

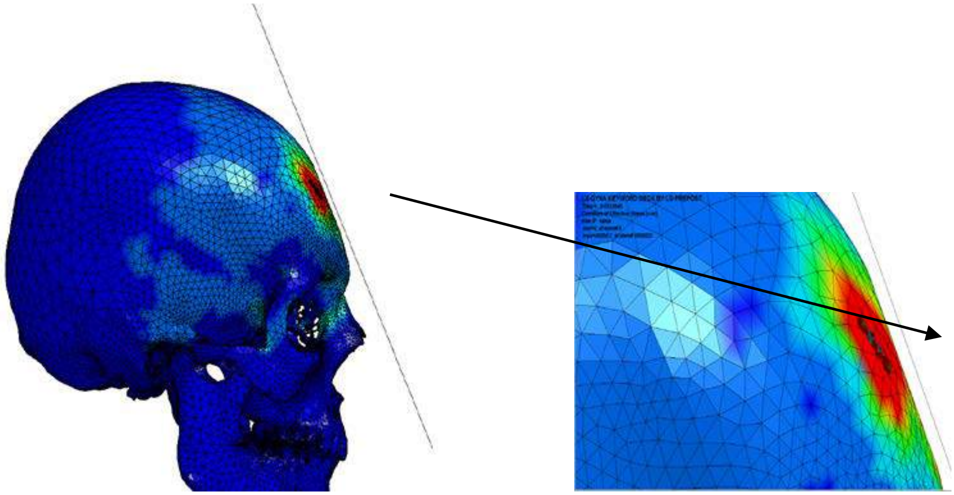

Figure 11: Frontal Bone Fracture With SS Plates Under Frontal Impact-Max Von mises stresses seen-118

- **FRONTAL BONE FRACTURE WITH SS PLATES UNDER A LATERAL IMPACT**

Figure 12 below shows the location of the fixation (1.2mm micro plate) and the location of the rigid wall. The figure also shows the von Mises stress distribution for this case.

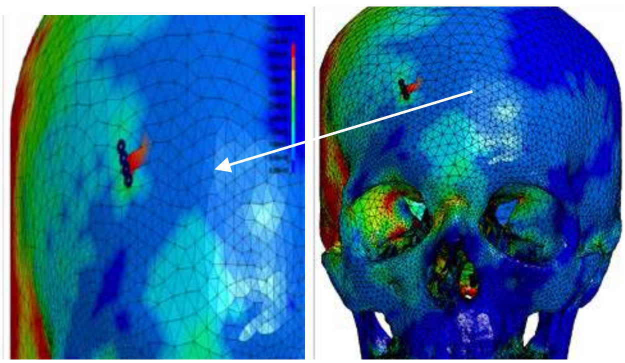

Figure 12: Frontal Bone Fracture With SS Plates Under a Lateral Impact-Max Von mises stresses seen-262

➢ NASO-ETHMOIDAL FRACTURE PLATED WITH SS PLATES UNDER LATERAL IMPACT

Figure 13 below shows the location of the fixation (1.2mm micro plate) and the location of the rigid wall. The figure also shows the von Mises stress distribution for this case.

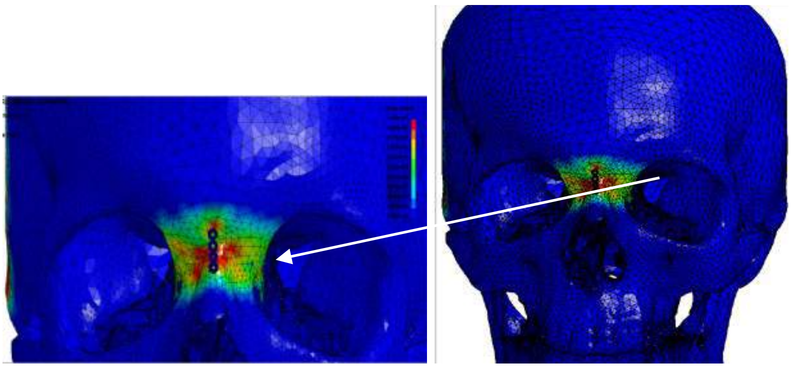

Figure 13: Naso-Ethmoidal Fracture Plated With SS Plates Under Lateral Impact-Max Von mises stresses seen-59

➢ MANDIBULAR PARASYMPHYSIS FRACTURE WITH TITANIUM PLATES UNDER 45^0 ANGULAR IMPACT

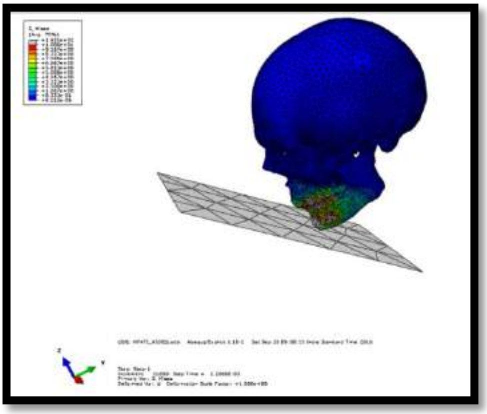

Figure 14: Mandibular Parasymphysis Fracture With Titanium Plates Under 45 degree_angular impact-Max Von mises stresses seen-193

Below figures 15 shows energy plots of skull under 45^0 loading condition it was observed that there is increase in internal energy and decrease in kinetic energy with respect to time.

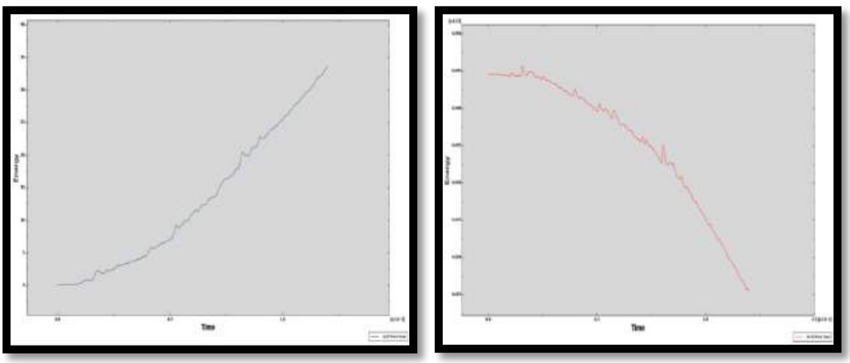

Figure 15: Internal energy and kinetic energy plot- Mandibular parasymphysis fracture with Titanium plates under 45 degree, Angular impact

➢ MANDIBULAR CONDYLAR FRACTURE WITH TITANIUM DELTA PLATE UNDER FRONTAL IMPACT

Figure16 shows stress distribution of skull under frontal impact condition and it was observed that maximum von mises stress is equal to 469 MPa

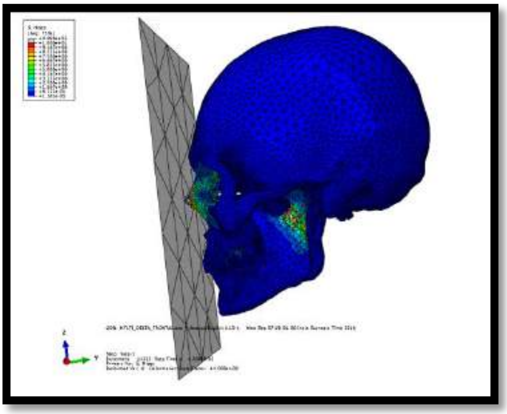

Figure 16: Mandible Condylar fracture with Titanium Delta plate under Frontal impact

Below figure 17 shows energy plots of skull under frontal impact condition it was observed that there is increase in internal energy and decrease in kinetic energy with respect to time.

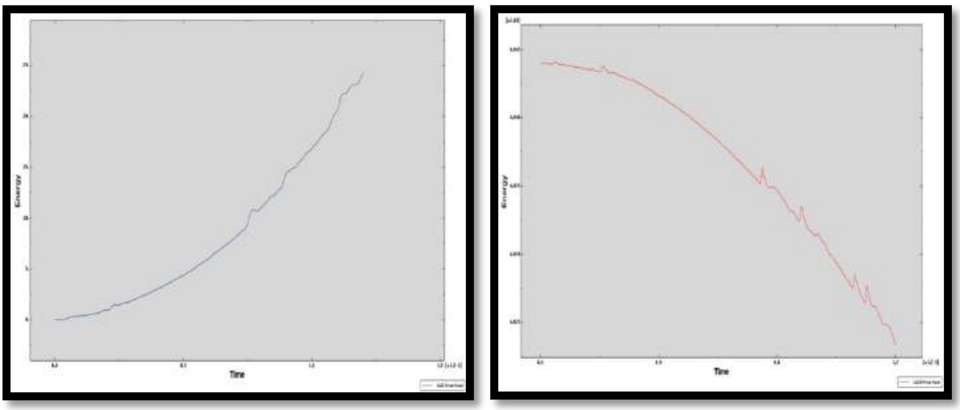

Figure 17: Internal energy and kinetic energy plot- Mandible Condylar fracture with Titanium Delta plate under Frontal impact

➢ LEFORT 3 FRACTURE WITH TITANIUM PLATES AND UNDER FRONTAL IMPACT

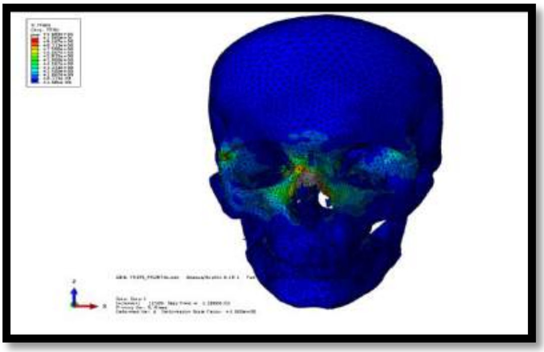

Figure 18: Max.Von Mises stress: 468 MPa

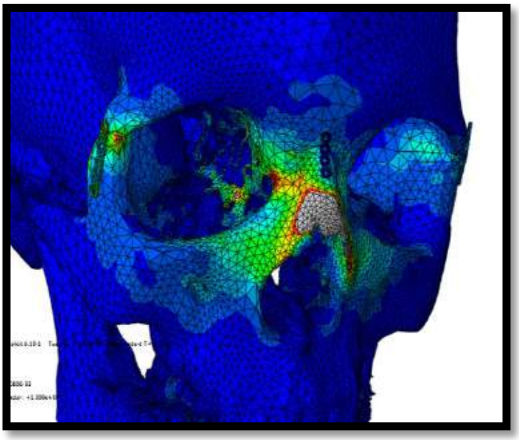

Figure 19: fracture with titanium plates and under frontal impact

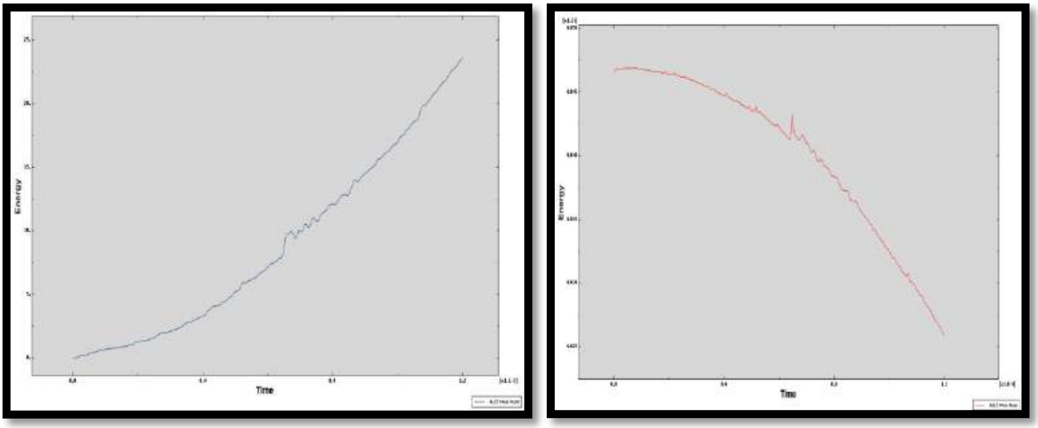

figure 20: internal energy plot and kinetic energy plot

3. RESULTS:

Impact analysis was carried out to understand the response of skull for different orientations and impacts. In the initial simulation, skull is made to impact the rigid wall with an initial velocity of 100m/s. The region of maximum von Mises stress distribution was observed. This was done for different conditions.which helps to analyze the stress and energy fields in a virgin skull and a skull with fixations during the event of an impact or crash of a skull to a rigid surface.further helps to evaluate the stability of the fractured fragments as well as to assess the stress- shielding effect of the fixations used

4. CONCLUSION:

The surgeries carried out are purely based on the past experience and clinical knowledge of the surgeon on similar type of treatment. There's an urgent need for development of effective methodologies to help medical practitioners with much better diagnosis and prognosis of such problems. Virtual simulation of different scenarios using techniques such as FEM (Finite Element Method) is very effective and efficient in bridging this gap.

Results from the analysis can be used to come up with optimum locations of implants for different type of impact situations. This is expected to complement the existing treatment methodologies used by surgeons. Further, appropriate knowledge of fracture biomechanics can be used to develop safety measures in automobiles to prevent and reduce incidence of facial injuries.

5. REFERENCES:

1. Shobha E S, Suresh Nagesh, H P Raghuveer, Vinay K S, International Journal of Engineering Research , Volume No.4, Issue No.12, pp : 648-651.
2. Finite element modeling of the head skeleton with a new local quantitative assessment approach. . Autuori B1, Bruyère-Garnier K, Morestin F, Brunet M, Verriest JP. :IEEE Trans Biomed Eng 2006 Jul;53(7):1225-32
3. TAshman RB, Van Buskirk WChe elastic properties of a human mandible.:.:Adv Dent Res. 1987 Oct;1(1):64-7.
4. Van Eijden TM,, Biomechanics of the mandible:Crit Rev Oral Biol Med. 2000;11(1):123-36
5. The creation of three-dimensional finite element models for simulating head impact biomechanics:T J Horgan and M D Gilchrist m IJCrash 2003 :Vol. 8 No. 4 : 353–366.
6. Seong WJ1, Kim UK, Swift JQ, Heo YC, Hodges JS, Ko, Elastic properties and apparent density of human edentulous maxilla and mandible: Int J Oral Maxillofac Surg. 2009 Oct;38(10):1088-93.
7. Feller KU, Schneider M, Hlawitschka M: ., Pfeifer G, Lauer G, Eckelt, Analysis of complications in fractures of the mandibular angle--a study with finite element computation and evaluation of data of 277 patients:JCraniomaxillofacSurg.2003 Oct;31(5):2905.
8. Tate GS, Ellis E 3rd, Throckmorton G, Bite forces in patients treated for mandibular angle fractures: implications for fixation recommendations,J Oral Maxillofac Surg. 1994 Jul;52(7):734-6.
9. Rozema, F.R,Optimization Of Boneplate Positioning And Device Choice In Mandibular Fractures By A Computer program,University Hospital Groningen ;
10. Otten, E. ; van Willigen, J.D. ; Bos, R.R.M. : Engineering in Medicine and Biology Society, 1991. Vol.13: 1991:1970 – 1971
11. Erkan Erkmen,Barış Şimşek, Ergun Yücel, Ahmet Kurt, Three-dimensional finite element analysis used to compare methods of fixation after sagittal split ramus osteotomy: setback surgery-posterior loading, :British Journal of Oral and Maxillofacial Surgery:Vol 43, Issue 2, April 2005, Pages 97–104.
12. A. Ziębowicz, J. MarciniakThe use of miniplates in mandibular fractures—biomechanical analysis: Journal of Materials Processing Technology:Volume 175, Issues 1–3, 1 June 2006, Pages 452–456
13. Rangan V, Raghuveer HP, Rayapati DK, Shobha ES, et al, ’’The influence of stress distribution in four different fixation systems used in treatment of mandibular angle fractures—A 3D finite element analysis’’, Oral Surgery, 2013;6:186-192
14. 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.