Development of Cranium Using Mimics and Rapid Prototyping Using ANSYS

Sadhasivam.C[1], Sai Krishna Gunda[2]

[1]Assistant Professor, Department of Mechanical Engineering, Saveetha School of Engineering, Saveetha University, Chennai,India.

[2]B.E Final year, Department of Mechanical Engineering, Saveetha School of Engineering, Saveetha University, Chennai , India

Abstract: *The project is real case study, a patient who suffered from cranial or skull damage by accident. Here we discussed about the accurate three dimensional finite element analysis of cranium using mimics and rapid prototyping. By using CT scan images , which comes in multiple slices of DICOM (.dcm) file format recommended slice of this images varies from 0.5mm to 1mm minimum slice layer thickness will give accurate and perfect data. In mimics software all the CT scan images get converted from 2d to 3d surface and then go through into simulation. This 3d design template is remeshed. The designed template is reconstructed by using RP machine and then by keeping the result of accurate 3d analysis using mimics .In this project, the effects of bone quantity and bone quality on stress distributions in a cranium model with implant-supported complete skull are investigated using the finite element method. In implant cranium, predictable outcomes of implant treatment is most desired and many clinical studies have reported success rates higher than 90% for various implant systems. Hence, it is mandatory to study these effects on stress distribution in bone and implant with a three-dimensional model that closely mimics that in the clinical situation.*

Keywords: CT scan data and images,MimicsSoftware ,RPTin ANSYS.

1. Introduction

Head is the most vulnerable body part in a traffic accident, body contact sports, construction works etc. Number of fatal head injuries has been reduced substantially through the introduction of injury protection measures such as helmet, safety belt and airbag. The development of additional protection systems depends on better understanding of head injury mechanisms which requires a knowledge on how a global external head load is translated into local loads in the internal head structures (nerve cells, blood vessels), and beyond what critical load level these structures are damaged. The development of a computer model that allows the study of such phenomena requires the knowledge of the anatomy of the human head, the mechanical properties of the tissues involved, the representation of various forces acting on the tissues and the boundary conditions reflecting a realistic head impact scenario. Such a model can be used to determine the time dependent stress and strain distributions on various tissues comprising the skull complex. Head has an extremely complex geometry. It consists of eight numbers of layered bone segments of irregular thickness (6mm) joined by sutures. The base of the skull is a thick irregular plate (12mm) with several small foramina magnum mechanics under static loading is extensively required for subsequent development of an injury protective device .Using Mimics software and Rapid prototype machine; wedesigned Customized Implant for Mandible Reconstruction. In which the CT scan images are converted to 3D Surface then this surface is distracted and designed, With the help of Rapid Prototype Machine.

Designed part is taken out for undergoing various techniques like, Investment casting, Sand casting for Rapid Manufacturing. Designed part is analyzed by using ansys, to find out the stress distribution and material strength. Whether it is to suggest the surgeons for complicated surgeries. The objective of this Project is to investigate the influence of varying bone quantity and quality on stress. Distribution in the implants and the surrounding bone for a using three-dimensional finite element analysis.

Problem Identification

Over the decades, many studies have been conducted to evaluate the parameters which affect the functionality and longevity of different implant systems. In recent years, implant supported over skull is quickly replacing the conventional removable cranium and are proven to provide more stability and are well-taken in terms of aesthetic results. However, studies are lacking in a more complete analysis of the whole cranium with an implant-supported over skull. Hence, this project aims to provide some insights to the stress patterns encountered in such an implant system.

Work Done

In these different technologies, we selected Three Dimensional Printing Technology (3DP), for this project **"A FINITE ELEMENT ANALYSIS OF CRANIUM USING MIMICS SOFTWARE & RAPID PROTOTYPE TECHNIQUES"** Z Corporation, USA is the only Manufacturer of Three Dimensional Printing Technologies (3DP), This technology is developed by Massachusetts Institute of Technology (MIT), USA. They are the only Manufactures of Monochrome and Multicolor RP Machine. In this 3D Printing technology, We can built various parts with different materials like, Conceptual models, Investment Casting models, Sand Casting Models, Snap fit and Elastomers.

2. Methodology of Rapid Prototyping

The basic methodology for all Rapid Prototyping techniques can be summarized as follows:
A CAD model is constructed, and then converted to STL format. The resolution can be set to minimize stair stepping.

The RP machine processes the STL file by creating sliced layers of the model. The first layer of the physical model is created. The model is then lowered by the thickness of the next layer and the process is repeated until completion of the model .The model and any supports are removed. The surface of the model is the finished and cleaned.

2.1 Prototype & its Uses

A Prototype is one of the first manufactured units of a product, which is tested so that any changes can be made to the design if necessary, before the actual commercial manufacture of the product. Prototype is useful in figuring out any design flaws there may be in your invention and also to find out if your invention really works. Prototype is useful to find out if the invention is really the right size, shape and form. Prototype helps to present our ideas the best.

2.2 A Type of RP Technologies are Discussed As Below:

2.2.1 Three Dimensional Printing (3DP)

The 3-Dimensional Printing methods are very reminiscent of Selective Laser Sintering, except that the laser is replaced by an inkjet head. The figure 5.5 shows the construction and working of 3DP technology.
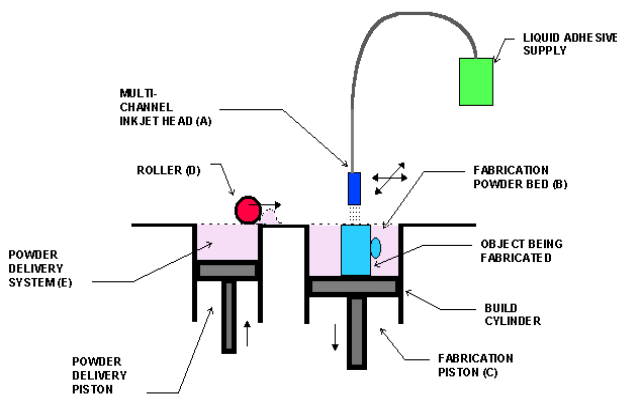

Fig. 7. Three dimensional printing.

Figure 2.2.1

2.3 Modeling of Cranium Implant

Firstly, computed tomography (CT) images of a patient's skull are obtained for this study. The CT images are segmented in image-processing software (MIMICS Version111.11, Materialize, Belgium) to obtain slides of cross-sectional views of the skull in the axial direction. The CT slides are reorganized to obtain the slide numbers which corresponds to the position of the cranium portion forthe three-dimensional reconstruction. Slide numbers 30 to 50 are chosen and the distance between each slide is 2mm.

The models are saved in IGES format to allow easy transfer of the models into ANSYS. The left fore head portion of the cranium is first created, followed by the thresholding Because of the limitations in the software in creating components which are perfectly bonded together at the curve surfaces, block volumes of the components are created and After the three-dimensional models are created, they must be imported into a finite element software for processing prior to analysis. The material properties of titanium implants are all obtained from the literature. The elements used for meshing the models are selected from the library of the finite element software. Finally, boundary conditions and loadings are applied for stress analysis.

2.4 Conversion of CAD Files to 3D Model Using RP Machine

Using Mimics, we Designed Customized Implant this implant got exported in STL+ and this CAD file is Imported to 3D Printer Rapid Prototype machine to make Prototype Model.

Figure 2.4: Process of RP Machine

3. MIMICS – 3D Visualization Software

Mimics interfaces between scanner data (CT, MRI, Technical scanner,) and Rapid Prototyping, STL file format, CAD and Finite Element analysis. The Mimics software is an image-processing package with 3D visualization functions that interfaces with all common scanner formats. Additional modules provide the interface towards Rapid Prototyping using STL or direct layer formats with support. Alternatively, an interface to CAD (design of custom made prosthesis and new product lines based on image data) or to Finite Element meshes is available.

Materialise's Interactive Medical Image Control System (MIMICS) is an interactive tool for the visualization and segmentation of CT images as well as MRI images and 3D rendering of objects. Therefore, in the medical field Mimics can be used for diagnostic, operation planning or rehearsal purposes. A very flexible interface to rapid prototyping systems is included for building distinctive segmentation objects.

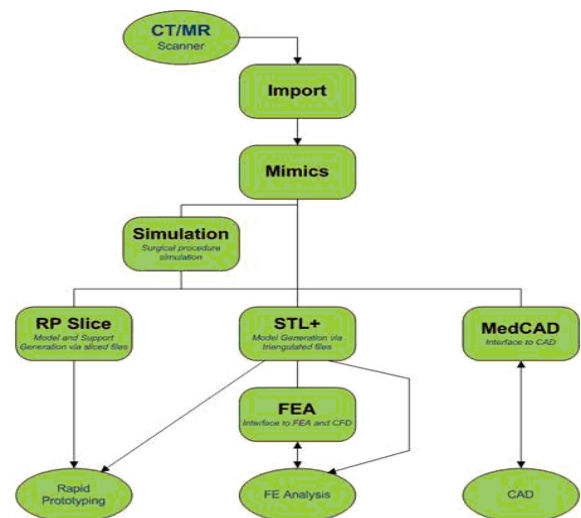## 3.1 Flow Diagram of MIMICS Software and its Working

Figure 3.1: Flow diagram of mimics

3.2 MIMICS based Module

Mimics interactively read CT/MRI data in the DICOM format. Segmentation and editing tools enable the user to manipulate the data to select bone, soft tissue, skin, etc. Once an area of interest is separated, it can be visualized in 3D .After this visualization, a file can be made to interface with STL+ or MedCAD. CAD data, imported as STL files, can be visualized in 2D and 3D for design validation based on the anatomical geometry

3.3Import Module

Import module imports CT and MRI data from a wide variety of scanner formats. The data can be accessed from CD, optical disk, DAT tapes, 4 mm tapes, etc.
- STL+ Module
- STL+ module provides interface options via triangulated formats.

3.4 Supported Formats

- STL (ASCII and Binary)
- DXF
- VRML
- PLY

3.5 Step 1: How to Import Dicom Image into MIMICS
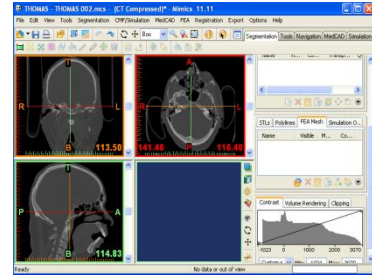
3.5.1 Dicom Images
The Dicom Images are known as Digital Imaging and Communications in Medicine. The CT/MRO Scan Images are taken as Dicom Images with 1mm (or User Selectable) slice as layer Thickness as shown in figure 5.6.
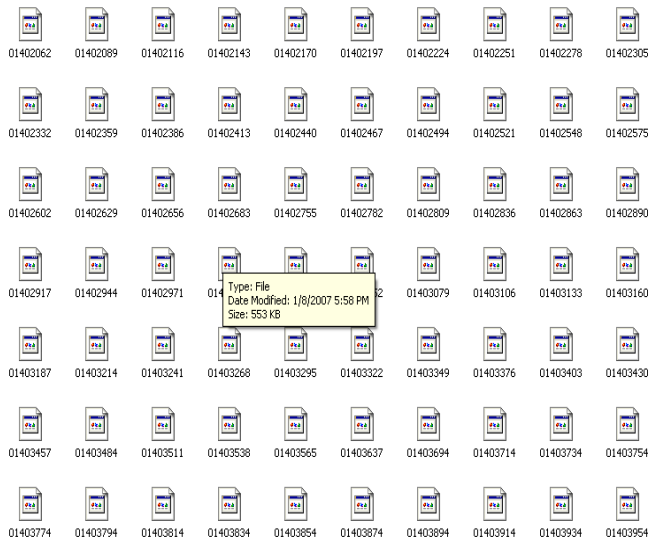
Figure 3.5.2: Picture of Dicom Images

### 3.5.2The Dicom Images are imported into the Software.
The Three Different views in Mimics Software. The View in the right Hand Top is the Images in the Top View. Left hand top is Front View and Left hand bottom is Side View. Right Hand bottom is for 3D Surface Visualization as shown in figure 5.7

**Figure 3.5.2:** The Dicom Images are imported into the Software

3.6 Step 2: Converting the Images into 3D Surface

The following steps are carried out to convert the dicom Image into a 3D Surface. At first a profile line is drawn to specify the area of interest to make 3D. The line will specify the threshold value of our area of interest and we can select the range of thresholding values as shown in figure 5.8
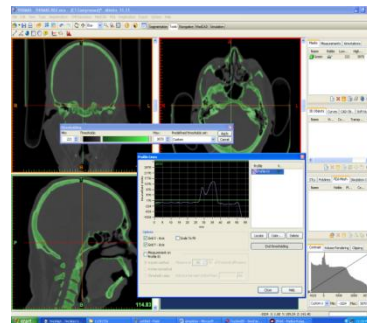

Figure 3.6: How to do Thresholding

After Thresholding of the part is done, Thresholding is the command which use to differentiate the Bone & soft Tissues, Using Region growing command again we have to select the Thresholding part to generate the 3D Surface. Then calculate the Selected Dicom Sliced images to view as 3D Surfaces as shown in figure 3.6.1
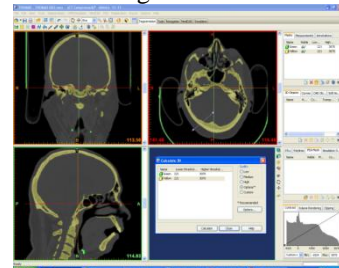
Figure 3.6.1: How to Calculate a 3D Surface

The 3D Surface of the Scanned and Sliced images are generated and visualized For Design as shown in figure 3.6.2
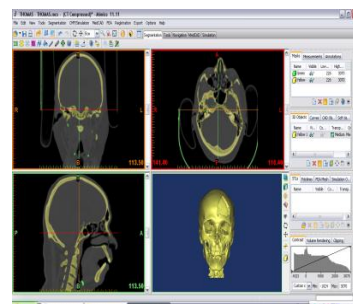
Figure 3.6.2: SURFACE FROM THE DICOM IMAGE

Paper ID: IJSER15294

4. Finite Element Analysis

4.1 Introduction

After the three-dimensional models are created, they must be imported into finite element software for processing prior to analysis. The material properties of cancellous, cortical, and titanium implants are all obtained from the literature. The elements used for meshing the models are selected from the library of the finite element software. Finally, boundary conditions and loadings are applied for stress analysis.
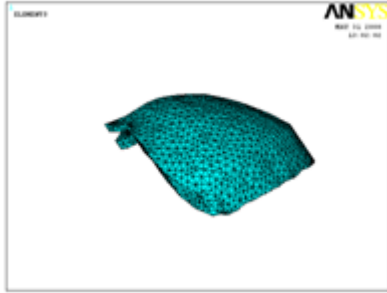

Figure 4.1: Meshing model

4.2 Materials Properties

In the absence of information about precise organic properties of the cancellous and cortical bone, they are assumed to be homogenous, isotropic and linearly elastic. Hence, all materials used in the models are considered to be homogenous, isotropic and linearly elastic, as frequently assumed by many authors. Elastic modulus of 108GPa and Poisson's ratio of 0.30 obtained from the values used by Lang et al (2003) is selected for the titanium implant and the elastic moduli and Poisson's ratio of cortical bone is same as that used by O'Mahony et al (2001) in studying the effect of anisotropic elasticity of cortical and cancellous bone on stress and strain in posterior cranium. Finally, the varying elastic moduli of the cancellous bone for the different bone types follow that of Tada et al (2003) in their findings of the influence of bone quality on stress and strain distribution in bone around implants. The material properties of all the entities are summarized as follows:

Table 4.2: Material properties used in analysis

Property	Young's Modulus	Poisson's Ratio
Titanium implant	108500	0.3
Cortical bone	14400	0.3
Cancellous bone	9500	0.3

4.3 Elements and Nodes

Finite element model required in the analysis is created by discretizing the three dimensional CAD model into smaller and simpler elements. This process is known as meshing. Because the model is symmetrical about the mesiodistal plane, only half of the model is meshed to shorten the processing and analysis time.

Differences in the mesh pattern may result in quantitative differences in the stress/strain values in the models and compromise the comparison between the values obtained for the different models. Therefore, it is important to derive the finite element models with a single mesh pattern. In this study, the element types chosen is shell93 in thelibrary ofANSYS, an eight-node tetrahedral element which is bettersuited for and more accurate in modeling structural solids with curved boundaries. The physical interactions between the surfaces of each entity are taken into account assuming that the different entities are perfectly bonded to each other. The meshing of the models results in a total of 2890 elements and 5870 nodes for Model.

4.4 Constraints and Loads

The models are constrained in all directions at the nodes on the top surface of the bone model. And since only half of the model is meshed, symmetry boundary conditions are prescribed at the nodes on the symmetry plane, which in this case, is the buccolingual plane. Forces of 100N are applied axially to different positions on the artificial skull ridge according to three cases (Figure23), as follows:
Case 1 (Loading position 1, 2, 3): Three vertical forces applied from the upper left incisor to canine
Case 2 (Loading position 4, 5, 6): Four vertical forces applied from the upper left first premolar to molar
Case 3 (Loading position 6, 7): Two vertical forces applied on the two upper left molars as shown in figure 4.2
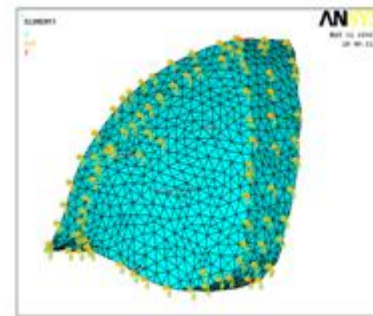

Figure 4.4: Applied boundary conditions and three loading conditions – Case 1, Case 2, and Case 3 (left to right)

Analysis with skull having a linear elastic property shows a uniform rise in shear stress and von-Misses stress value in cranium. But the locations of maxima are same, i.e. in the cranium stem region. In case of the level of von-Misses stress was much higher. The maximum vonMiss's stress obtained adjacent to the titanium was 3.605 MPa. 3-D static of analysis gives the following variation of maximum von-Misses stress and maximum displacement with time.

However, in high Stress values are observed in the thin cortical bone layer, resulting in undesirable stress concentrations. Maximum stress occurs in the bone-implant interface region at the base of the implant in the cancellous bone (Figure23) and also at the region around the neck of the implant in the cortical bone (Figure23) for both the models.

In the titanium implants, higher stress is observed with the exception of that for Case 1 loading condition. The stresses are well-distributed with the maximum stress located at the tissue area surrounding the implant. The stress is well-distributed in both models with the maximum value located at the region surrounding the implant (Figure23). In the

Paper ID: IJSER15294

cranial implant, high stress magnitude of 23494 MPa is observed for Case 2 loading condition. (Figure23).

When comparing the different bone types, the results obtained showed similarities in all the three types of bone. Only a slight increase in maximum stress values is observed in the cortical bone layer when the Young's modulus of the cancellous bone is decreased.
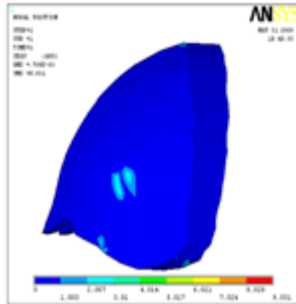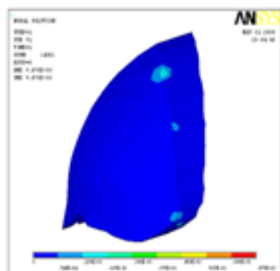

Figure 4.4.1: Stress distribution in cortical bone; case 1 loading condition

5. Conclusion

Both the Defected portion and the Designed portion of the left forehead of the Physical part are displayed for your reference. The main thesis, Using Rapid prototype part how we can go for manufacturing a customized Implant by various techniques in Castings like, Investment casting. This Customized Design Implant is now ready for Rapid Tooling for Rapid Manufacturing and for Direct Implant to the Patient. By Using this Techniques We can reduce the Lead-time of the Design and Manufacture the Implant.

The model consisted of 2890elements through 5870nodes. It was observed that the maximum von-Misses stress was around 9.031MPa and maximum shear stress was 20.04 MPa that occurred in cranial stem. Also observed that stem is most vulnerable to shear. Total stress contour map revealed that the average Von-Misses stress generated in most of the areas was below 0.490 MPa. In case of the skull, level of von-Misses stress was much higher
1. A thinner cortical bone layer and less cancellous volume results in higher stress concentrations in the cortical bone.
2. Stress concentrations are found mainly in areas surrounding the plant.
3. Bone qualities do not contribute to changes in stress distribution or stress values in the cranial implant supported model.

Figure 4.4.2: Stress distribution in cancellousbone ;case 2 loading condition
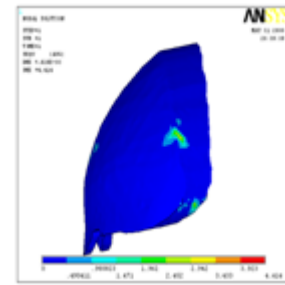
Figure 4.3: Stress distribution in titanium implant; case3 loading condition.

References

[1] Khalil, T. B. and Hubbard, R. P., "Parametric Study of Head Response by Finite Element Modeling", Journal of Biomechanics, vol. 10, 1977, pp. 119-132.
[2] Ruan, J. S., Khalil, T. B. and King, A. I., "Human Head Dynamics Response to Side Impact by Finite Element Modeling", Journal of Biomechanical Engineering, vo 113, 1991, pp. 276-283.
[3] Zhou, C., Khalil, T. B. and King, A. I., "A Human Head Finite Element model for Impact Injury Analysis", Proceedings of 5th Injury Prevention through Biomechanics Symposium, 1995, p. 137-147.
[4] Ayub K. Ommaya, "Mechanical Properties of Tissues of the Human Head" Journal of Biomechanics, 1, pp. 127-138, 1967.
[5] Ruan, J. S., Khalil, T., and King, A. I., "Dynamic Response of the Human Head to Impact by Three dimensional Finite Element analysis," Journal of Biomechanical Engineering, (February 1994) vol.116, no. 2, pp. 44-50.
[6] King, A. I., "Progress of Research on Impact Biomechanics," Journal of Biomechanical Engineering,(November 1993) vol. 115, no. 4b, pp. 582-587.
[7] Hallquist, John O., "LS-DYNA3D Users' Manual, version 936" Live more Software Technology Corporation.
[8] Ansys Operational Guide.
[9] Shuck, Z. Lowell, "Determination of the Dynamic Shear Modulus of Human Brain Tissue" Ph.D. Dissertation, West Virginia University, Morgantown, WV, 1970.
[10] Gross, Arthur G., "A New Theory on the Dynamics of Brain, Journal of Neurosurgery, 15, pp. 548561, 1958.

Paper ID: IJSER15294