



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

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 13, Issue 12, December 2024

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.524

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Fem of Human Mandible and Development of Implant of Human Mandible by Additive Manufacturing

Syed Abed Hashmi, M.E, Dr.T. Ramamohan Rao

Student, Vasavi College of Engineering, Hyderabad, India

HOD, Department of Mechanical Engineering, Vasavi College of Engineering, Hyderabad, India

ABSTRACT: A computational study to build the 3D model of the mandible from DICOM (Digital Imaging and Communications in Medicine) files. The files were imported to the Slicer 4D software and the model was built by thresholding each slice of the file in this software. Then the model was imported into the Siemens UG-NX 11.0 software in STL file extension, for the convergent modeling, to convert the facets into the single solid model. And Material optimization analysis is performed on the jaw, the model was imported in ANSYS-Workbench 2019R3 software with IGES file extension, with two different boundary conditions such as fixed supports and hinged supports on the condylar region of the jaw by using Bone, Titanium (Ti6Al4V), PMMA and PEEK as different materials. The behaviour of the implant was studied when the jaw is subjected mastication condition of 7.2MPa as the load as maximum bite force and 0.5MPa as static pressure as maximum pressure on the jaw, Suggesting the appropriate material for the implant from the analysis by the comparison of the results with the material properties of bone and the properties of different materials mentioned above and 3D model is fabricated using STL file in Cura software, and Ultimaker 3D as a fused deposited machine with material like PLA or ABS

KEYWORDS: DICOM images, convergent modeling, Finite Element Analysis, FDM.

I. INTRODUCTION

The mandible is the lower jaw connected to the skull. It is the largest and the strongest bone in the facial skeleton that supports the lower teeth. Mandibular defects are frequent in the dental practice which can severely affect the masticatory function. The need of developing the implants for the jaw, as the connectivity of the jaw is weakened due to fractures caused on the bone such that plates and screws were earlier used to preserve the connections of the jaw but these plates and screws fail to preserve the strength of the jaw. to overcome this, a new plating system was designed using CAD/CAM/CAE which would be used to replace the jaw the U-shaped plating was designed based upon the dimensions of the actual human jaw to save the time and efforts during surgical operations.[1] A 2D finite element model of the human temporomandibular joint was developed to investigate the stresses and the reaction forces during the normal sagittal closure. This was carried in two steps the first one was performed to evaluate the condylar displacements, and the other one was the finite element model was created to evaluate the mechanical parameters [2]. In this study, the procedure of obtaining a bio-CAD model of mandible from computed tomography (CT), 2D segmentation, calculating 3D object from scanned data, reverse engineering interface, point cloud data processing, surface reconstruction, solid model reconstruction, obtaining bio-CAD model, and fabricating the model using RP technology [8]. Reconstruction of the mandibular designs have been built up by many techniques, but those designs do not reflect the expectations of patients, as the mandible is the mobile bone which folds skull and articulated with it through teeth. A space frame model was approximated by the horizontal bars, curved part and two load cases have been examined one with symmetrical loading and the other case with the load on the left side with a value of 100N, with the material properties of the bone and also with PMMA [12]. This study aims to apply reverse engineering concepts and build a 3D model of the human mandible and to suggest the best alternative material for the implant using Finite Element Analysis, which serves the same function of the actual mandible.

II. METHODOLOGY

1. Problem Identification

From the research aspects, implants in the mandible are very specific in use, major due to detection of tumour in the bone, cracks, or fracture in the jaw they may be frequent due to the formation of cavities in the teeth root area in the jaw or due to accidents. [3]. Therefore, there needed to develop the quality of implants or the design of the implants,

recent developments in the implants like plates and screws fail to preserve the strength of the jaw with more than one fracture. To overcome the hustles in designing the complex structure, biomodelling of the jaw can be the input for the analysis and fabrication. The workflow and the process methodology is carried out to develop the prototype model of the jaw is as shown in fig 1

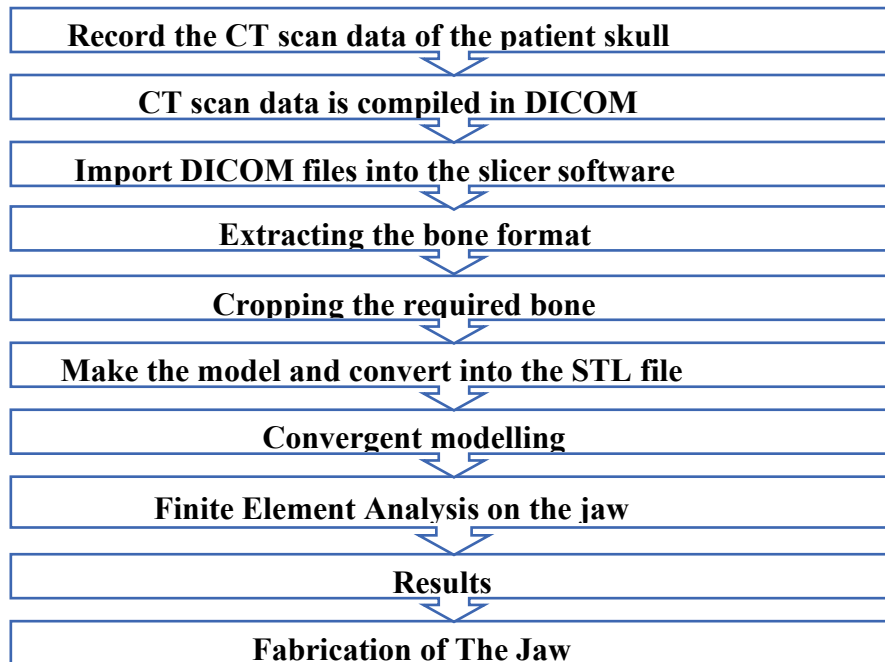


Fig 1 workflow of patient-specific jaw analysis.

Anatomical modeling represents the models of the patient's anatomical data, which were constructed based upon the Digital Imaging and Communication in Medicine (DICOM) from CT scans File. Creating the 3D model using medical files. The CT scan head section of a male patient of 65 years is received from the doctor, which will be in the DICOM format which is to be imported in the Slicer 3D software in volume module as shown in the Fig 2.

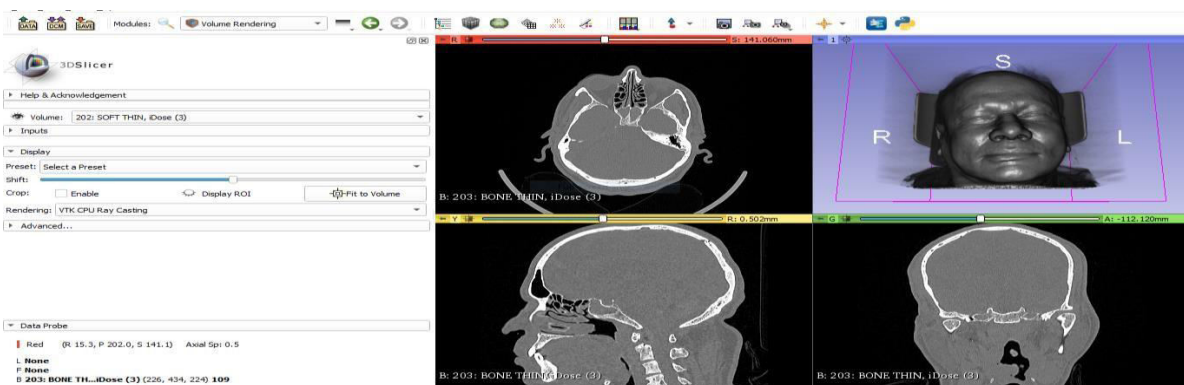


Fig 2 showing the DICOM file imported in the volume module of the slicer software

The second step is to extract the CT scan data in the bone formation in the Slicer software. This is done by changing the session to CT bone and the shifting to require form as shown in the Fig 3

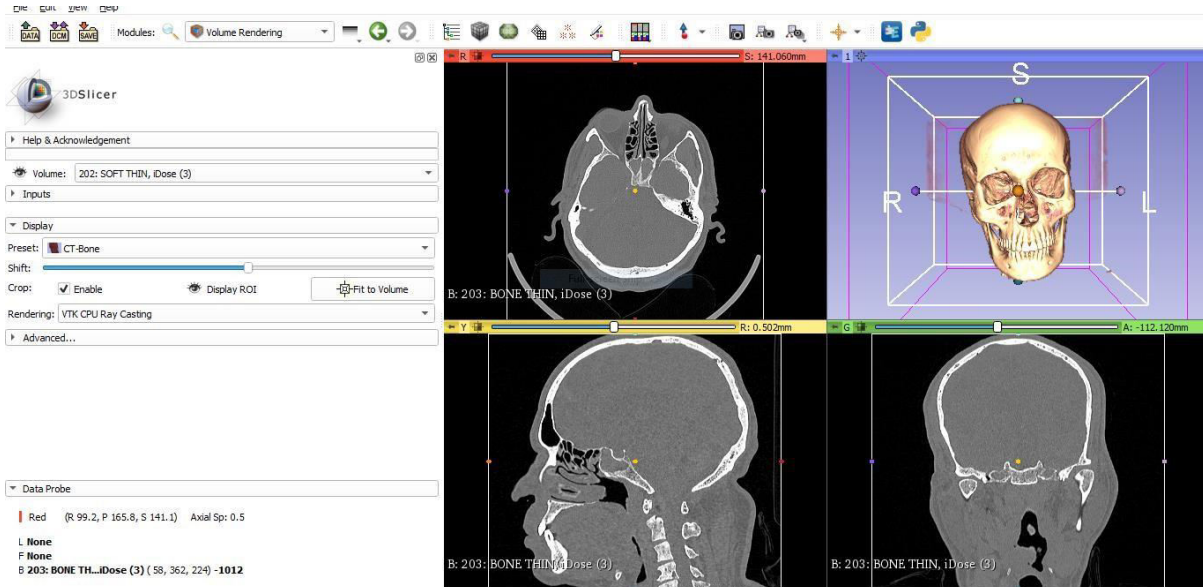


Fig3 extraction of the CT- Bone part from the DICOM file

As we only require the lower jaw, so we tried cropping the required part but we didn't get the only required part for this so we have to threshold the part of the mandible using the thresholded, Thresholding must be done carefully to get the good results in the analysis as it may lead to result in poor quality elements in solving the problem in analysis, After once the cropped and the only required session is thresholded then is modeled using model making option in the editing module and the model is made as shown in the Fig 4 (a) and Fig 4(b).

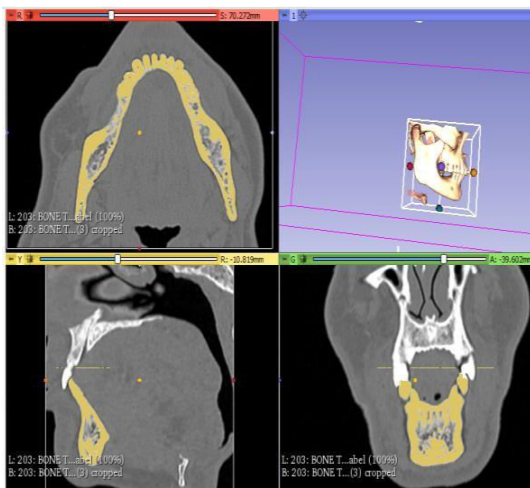


Fig 4 (a)



Fig 4(b)

Fig 4(a) thresholding using paint option to make the required model, Fig 4(b) different views of mandible model made in the slicer software

Once the model is made, it is saved in the STL (stereolithography) file format to proceed for further steps, which are convergent modeling, Finite element Analysis, and Fabrication. Convergent modeling is called such because it "converges" meshes and solids into a single model. Convergent modeling is designed to deliver three key benefits to designers:

1. Optimize part design for 3D printing
2. Speed up the overall design process
3. Make reverse engineering a more common and efficient practice in product design.

For the convergent modeling, we must import the model in the STL file format in the NX_11 software to converge the facets, meshes, and the solids into a single model, therefore, by selecting the menu option we can able to convert the files in the required format.

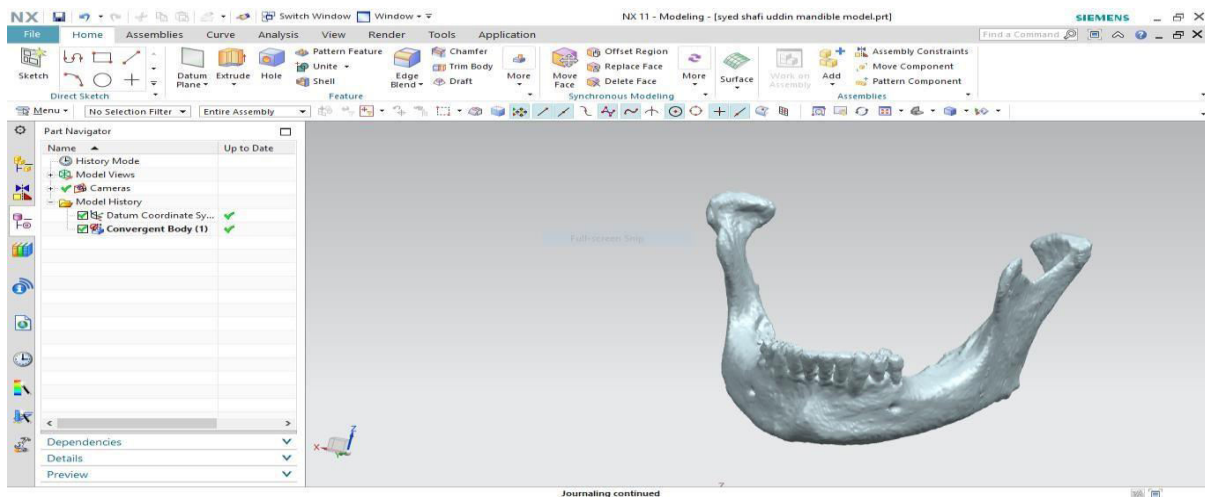


Fig 5 convergent modeling of the lower jaw in the NX software.

Once the convergent model is created in the NX 11.0 software shown in Fig 5, the model is ready to undergo the further process like analysis and 3D printing.

2. Materials Selected for The Implants in The Project

The materials selected for the implants are as follows:

Titanium Alloy (Ti6AL4V)

PMMA (Poly Methyl Methacrylate)

PEEK (Poly Etherether Ketone)

Bone

The table 1 shows the materials selected for the implants, with their material properties like Young's modulus, poisson's ratio, density, tensile yield strength and ultimate tensile strength.

Table 1 material properties for the mandible implants.

Properties	Titanium alloy	Polymethyl Methacrylate	Polyether ether ketone	Bone
	(Ti6Al4V)	(PMMA)	(PEEK)	
Young's Modulus (MPa)	110000	3000	4000	3000
Poisson's ratio	0.3	0.38	0.4	0.3

Ultimate tensile strength (MPa)	950	72	103	150
Yield strength (Mpa)	800	72	100	100
Density(kg/mm ³)	4430e-9	1180e-9	1360e-9	1850e-9

3. FEA on the convergent model of the jaw

The static structural analysis enables us to estimate the effects of steady loading conditions on the structure and to study the changes in the structure under the different boundary conditions and the loading. The interface for analysis for the static structural in the ANSYS WORKBENCH 2019 R3 Import the mandible file which is in IGES file format in the geometry module of the static structural analysis The imported mandible is meshed using tetrahedral meshing mesh using patch conforming method with the global element size of meshing of 2mm shown in Fig 7

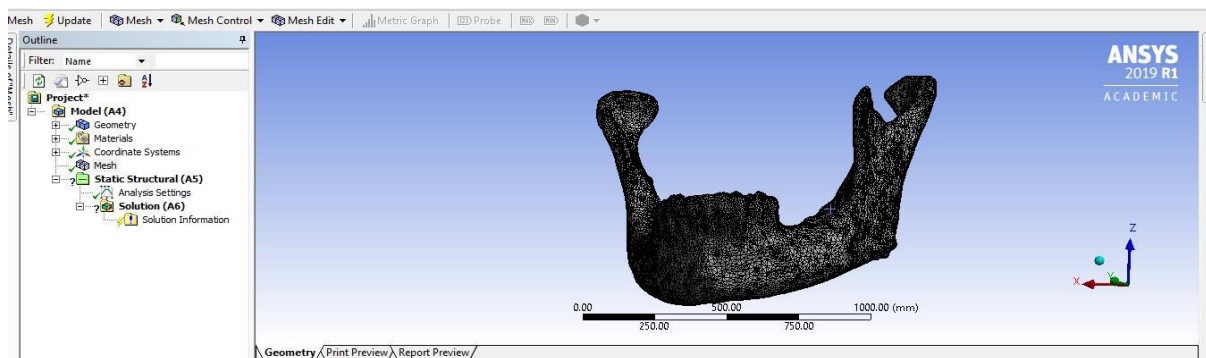


Fig 6 Tetrahedral meshing of the mandible with element size 2mm

The analysis comprises the 2 different boundary conditions such as the hinged boundary condition and the fixed boundary condition on the condylar region of the mandible. To apply the boundary conditions. In the preprocessor we must define loads under which-displacement is selected here it arises with 2 cases of analysis for the project

- Fixed end condition on the condylar part - (select All DOF)
- Hinged end condition on the condylar part – (select only F_x and F_z)

For the application of load on the jaw select the required nodes for two loading conditions they are as follows:
Point load as the maximum force required to bite i.e., 7.2MPa

Uniformly distributed pressure Load on the jaw i.e., 0.5MPa, as the muscle force on the jaw during mastication condition. The boundary conditions and load applications on the jaw required for mastication, for the static structural analysis in ANSYS- Workbench 2019R3 with fixed and hinged supports are shown in the Fig 5.5 and Fig 5.6 respectively.

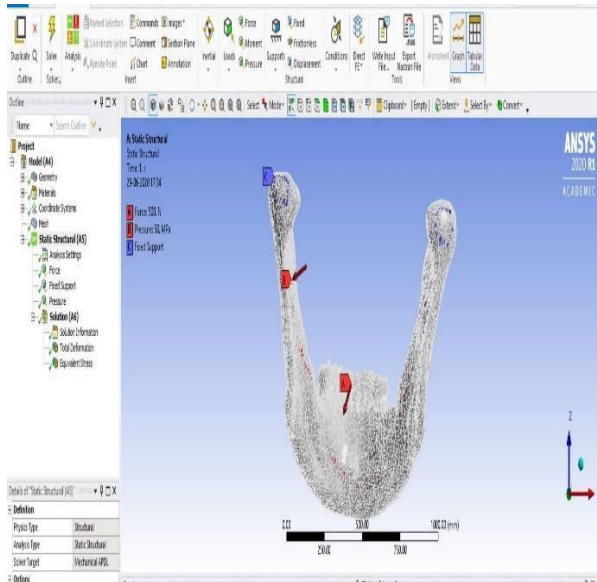


Fig 7(a)

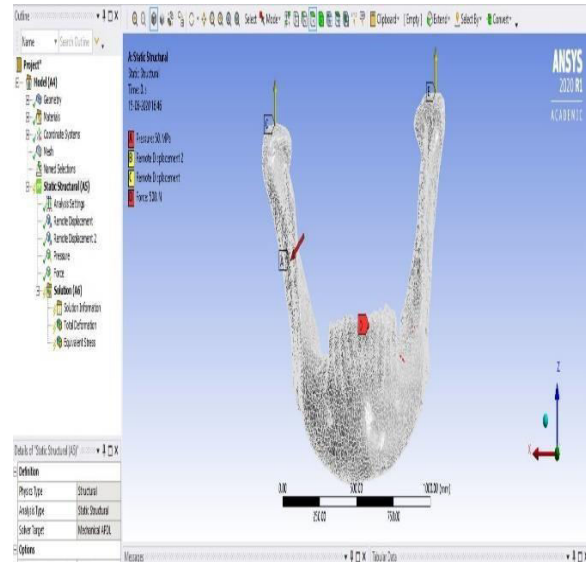


Fig7(b)

Fig 7(a) fixed end Boundary conditions and load application on the jaw during mastication and Fig 7(b) Hinged end Boundary conditions and load application on the jaw during mastication

III. RESULTS

As per work specified analysis and development of mandible implants, the implant materials have been taken into the study, and analysis was performed on the jaw with different implant material, which suits the best alternative for the jaw implants were Titanium Alloy, PMMA and PEEK. The analytical results are concluded by representing the various materials and their total deformation values and the Equivalent von Mises stress values. The static structural analysis was carried out on the lower jaw in Ansys Workbench 2019 R3 gives the results deformation values and maximum and minimum Equivalent stress of two cases solved in ANSYS- Workbench 2019R3. The results for Fixed end conditions the maximum deformation values are 912.41mm for bone, 904.91mm for PMMA, 676.6 mm for PEEK and 248.84 mm for Ti-alloy respectively. while the Equivalent Von Mises stresses, the minimum and maximum values are 0.010038& 2948.3 MPa for bone ,0.0084 &2733.8 MPa for PMMA 0.0080 &2675.5 MPa for PEEK and 0.010038& 2948.3 MPa for Titanium Alloy respectively. The variation of deformation and equivalent stress for different material with same load but with fixed supports is plotted in graph 1. For Hinged end conditions the minimum and maximum deformation values are 5.46& 13313mm for bone, 16.71& 96026mm for PMMA,14.19 &70223mm for PEEK and for 4.800 & 2787.9 mm Ti-alloy respectively. while the Equivalent Von Mises stresses, the minimum and maximum values are 0.041558&1.80e5 MPa for bone 0.04923 & 1.06e5 MPa for PMMA, 0.0543 & 1.05e5MPa for PEEK and 0.041558&1.80e5MPa for Titanium Alloy respectively. The variation of deformation and equivalent stress for different material with same load but with Hinged supports is plotted in graph2 summary of the results obtained is tabulated in the table 2

Table 2 static structural analysis results of the jaw with different materials

Material	Boundary condition	Total deformation (mm)		Equivalent stress (Von-Mises) (MPa)	
		Min	Max	Min	Max
BONE	FIXED END	0	8.7314	0.00070	30.18
PMMA	FIXED END	0	20.273	0.00070	28.15
PEEK	FIXED END	0	15.178	0.00068	27.59
Ti -ALLOY	FIXED END	0	5.5563	0.00070	30.18
BONE	HINGED END	0.00692	6.064	0.000181	15.97
PMMA	HINGED END	0.01693	14.243	0.00013	16.492
PEEK	HINGED END	0.01286	10.697	0.000189	16.554
Ti- ALLOY	HINGED END	0.000440	0.3859	0.000181	15.972

In this analysis the deformation and stress on the jaw helps us to estimate out the implant material suitable for the bone, following fig 8 (a) and 8(b) shows the total deformation and the equivalent von mises stress on the jaw with BONE material properties under fixed conditions on the condylar region respectively.

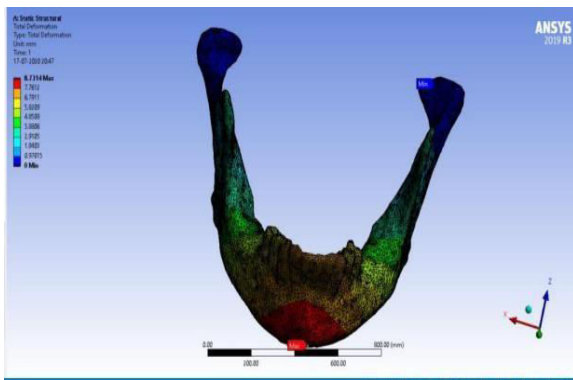


Fig 8(a)

Fig 8(a) total deformation of the jaw, fixed condition for bone

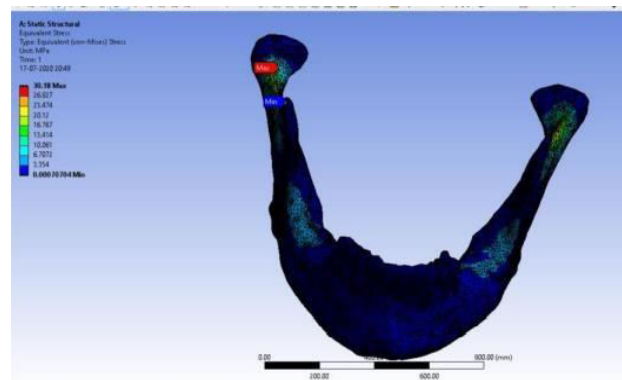


Fig 8(b)

Fig 8(b) element stress (Von-Mises) of the jaw, fixed condition for bone

The following fig 9(a) and 9(b) shows the total deformation and the equivalent von mises stress on the jaw with PMMA material properties under fixed conditions.

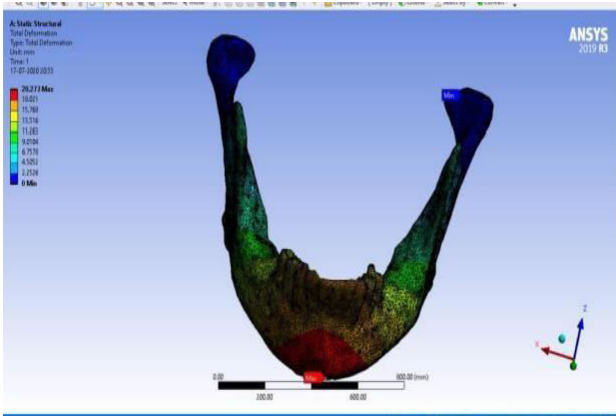


Fig 9(a)

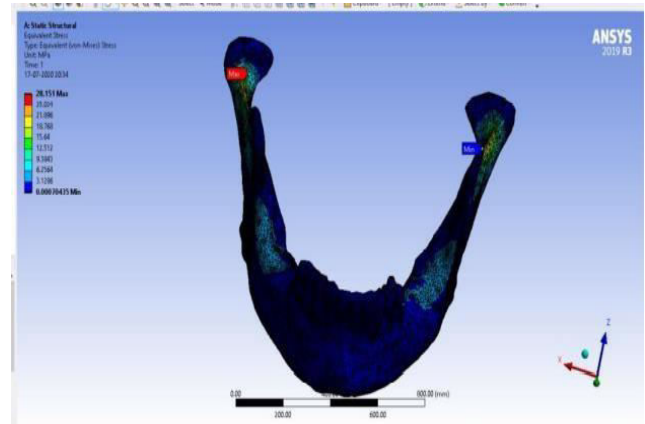


Fig 9(b)

Fig 9(a) total deformation of the jaw fixed condition for PMMA

Fig 9(b) element stress (Von-Mises) of the jaw, fixed condition for PMMA

The following fig 10(a) and 10 (b) shows the total deformation and the equivalent von mises stress on the jaw with PEEK material properties under fixed conditions.

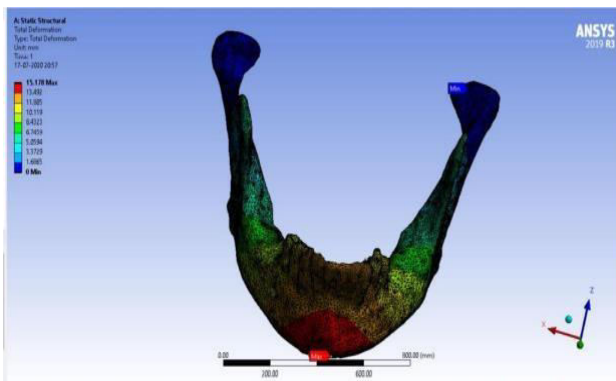


fig 10(a)

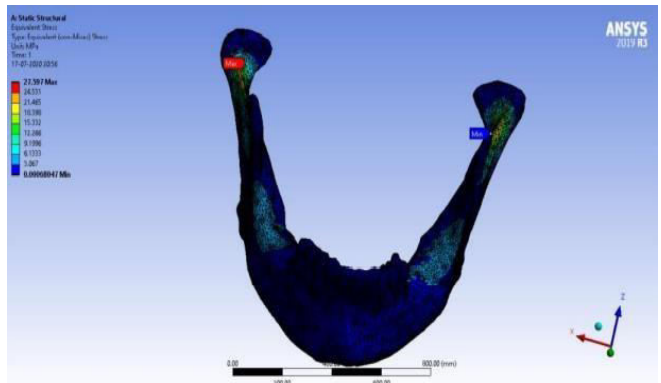


fig 10(b)

Fig 10(a) total deformation of the jaw, fixed condition for PEEK

Fig 10(b) element stress (Von-Mises) of the jaw, fixed condition for PEEK

The following fig 11(a) and 11(b) shows the total deformation and the equivalent von mises stress on the jaw with Titanium Alloy properties material properties under fixed conditions on the condylar region respectively.

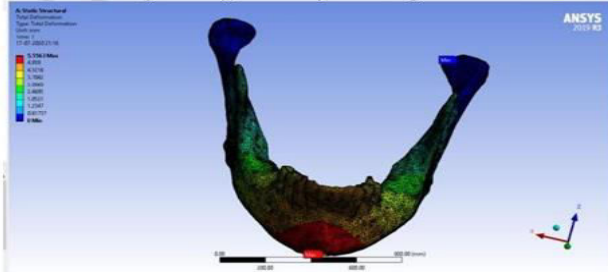


Fig 11(a)

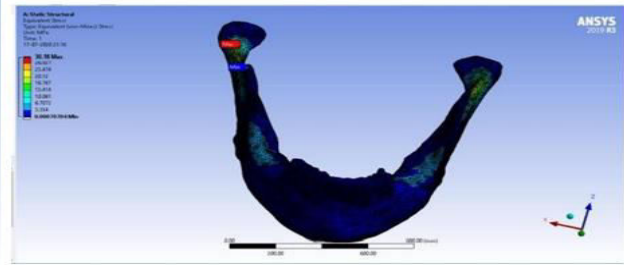


Fig 11(b)

Fig 11(a) total deformation of the jaw, fixed condition for Titanium-Alloy

Fig 11(b) element stress (Von-Mises) of the jaw fixed condition for Titanium-Alloy

The following fig 12(a) and 12(b) shows the total deformation and the equivalent von mises stress on the jaw with BONE material properties under Hinged conditions on the condylar region respectively.

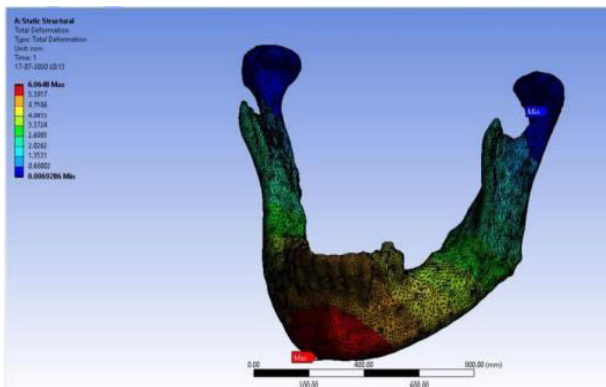


fig 12(a)

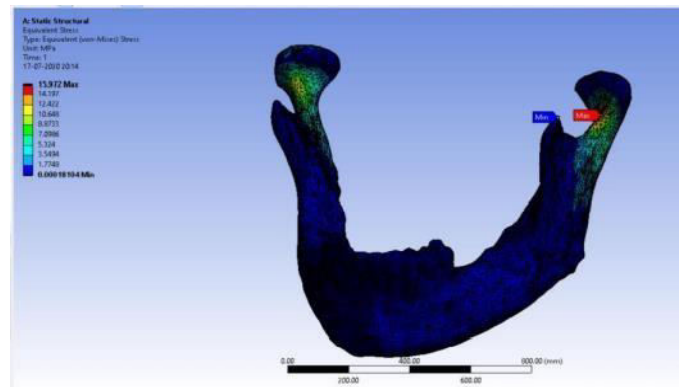


fig 12(b)

Fig 12(a) total deformation of the jaw Hinged condition for Bone

Fig 12 (b) element stress (Von-Mises) of the jaw, Hinged condition for Bone

The following fig 13(a) and 13(b) shows the total deformation and the equivalent von mises stress on the jaw with PMMA material properties under Hinged conditions on the condylar region respectively.

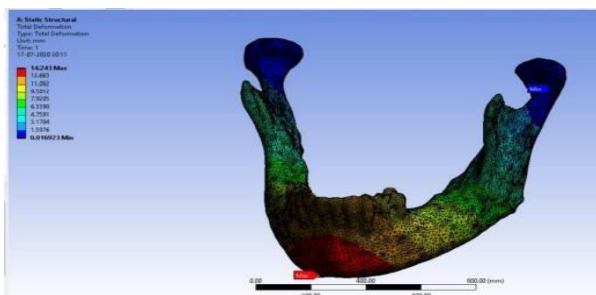


fig 13(a)

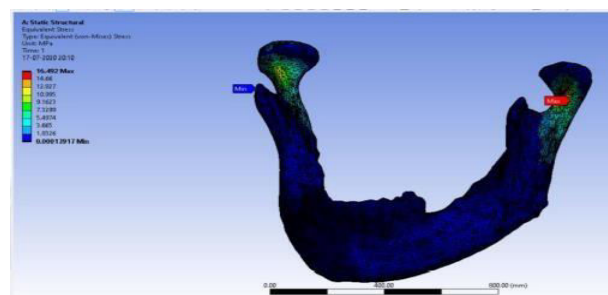


fig 13(b)

Fig 13(a) total deformation of the jaw, Hinged condition for PMMA

Fig 13(b) element stress (Von-Mises) of the jaw, Hinged condition for PMMA

The following fig 14(a) and 14(b) shows the total deformation and the equivalent von mises stress on the jaw with PEEK material properties under Hinged conditions on the condylar region respectively.

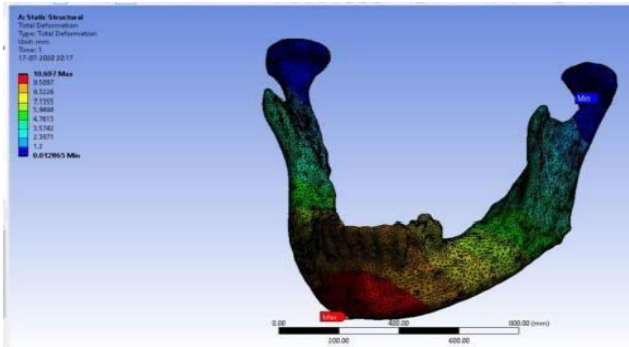


fig 14(a)

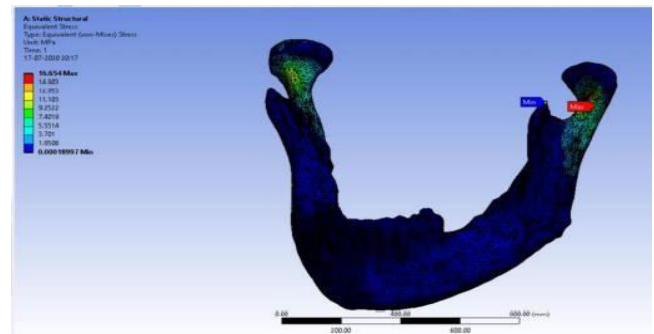


fig 14(b)

Fig 14(a) total deformation of the jaw, Hinged condition for PEEK

Fig 14(b) element stress (Von-Mises) of the jaw, Hinged condition for PEEK

The following fig 15(a) and 15(b) shows the total deformation and the equivalent von mises stress on the jaw with Titanium Alloy material properties under Hinged conditions on the condylar region respectively.

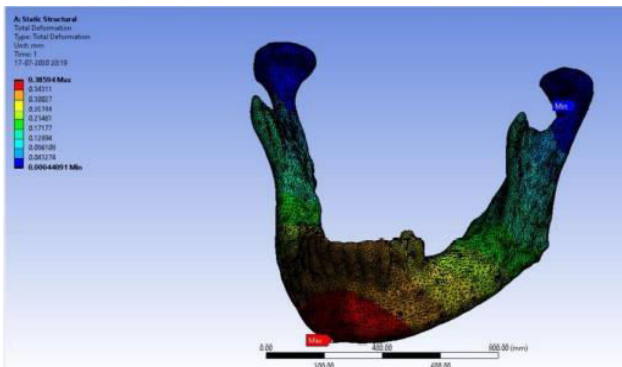


fig 15(a)

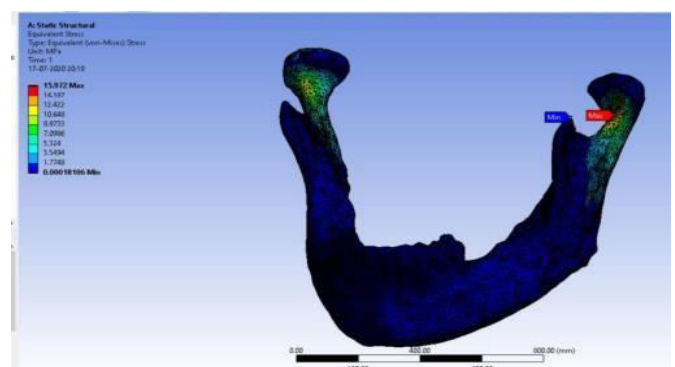
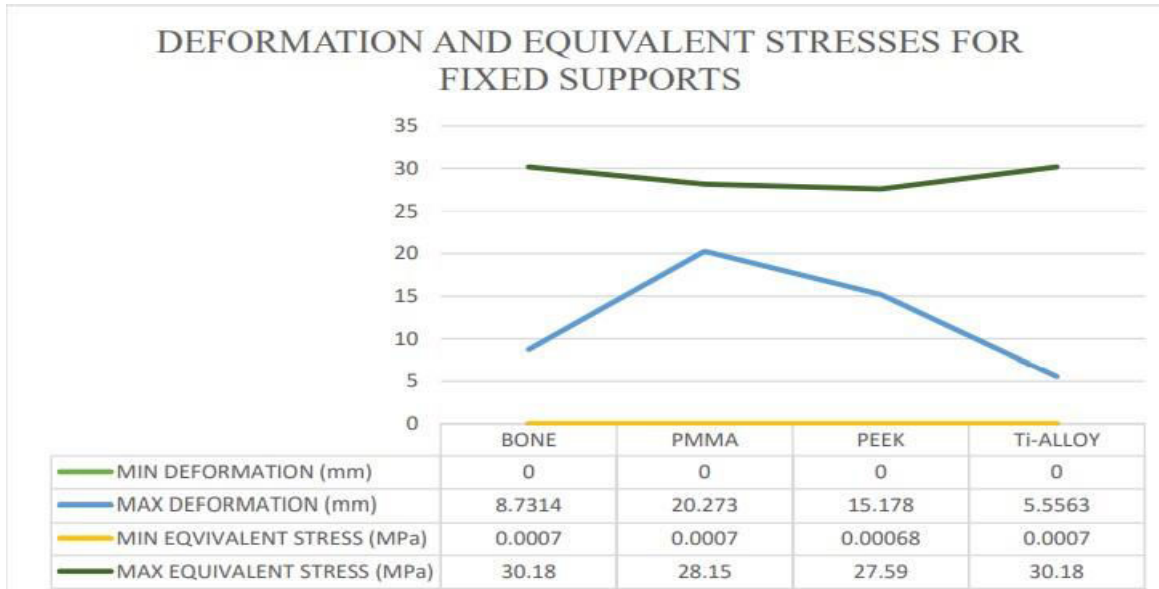


fig 15(b)

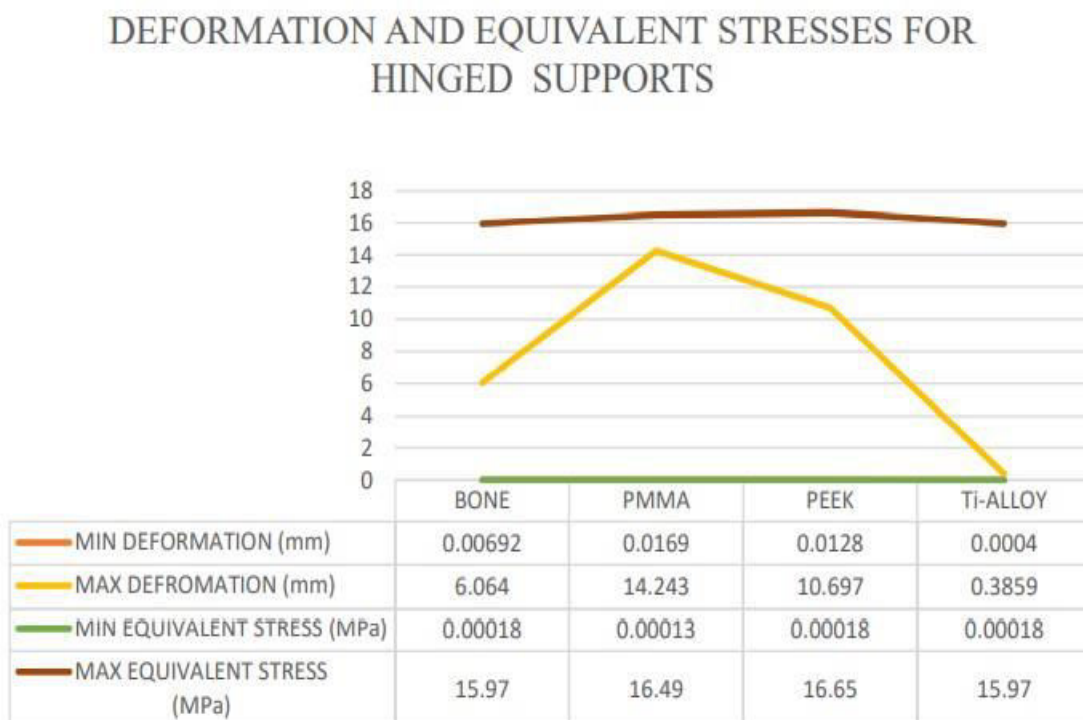
Fig 15(a) total deformation of the jaw, Hinged condition for Titanium-Alloy

Fig 15 (b) element stress (Von- Mises) of the jaw, with Hinged condition for Titanium-Alloy

The variation of deformation and equivalent stress for different material with same load but with fixed supports is plotted in graph 1. The variation of deformation and equivalent stress for different material with same load but with Hinged supports is plotted in graph 2.



Graph 1 Deformation and Equivalent stress for different materials under fixed end conditions.



Graph 2 Deformation and Equivalent stress for different materials under Hinged end conditions

IV. MANUFACTURING THE JAW USING FUSED DEPOSITED MACHINING

3D printing has given the various opportunities to develop intricate and complex parts either in the medical field or aerospace, specialized industries have dedicated to achieving the highest precision and accuracy while not every 3D printer can match the level of expectations, a few can. And one such machine is Ultimaker 3 Extended shown in Fig 17



Fig 16 Fig showing the machine Ultimaker 3 Extended

This infill pattern defines the strength of the model to be created. The pattern depends upon the infill density as it defines the air gap between the geometry. Higher density higher will be the strength of the model. For assigning the patterns and providing the support to manufacture the complex structure Ultimaker Cura software is used which enables us to import the geometry in the STL file format as shown in the fig 18.

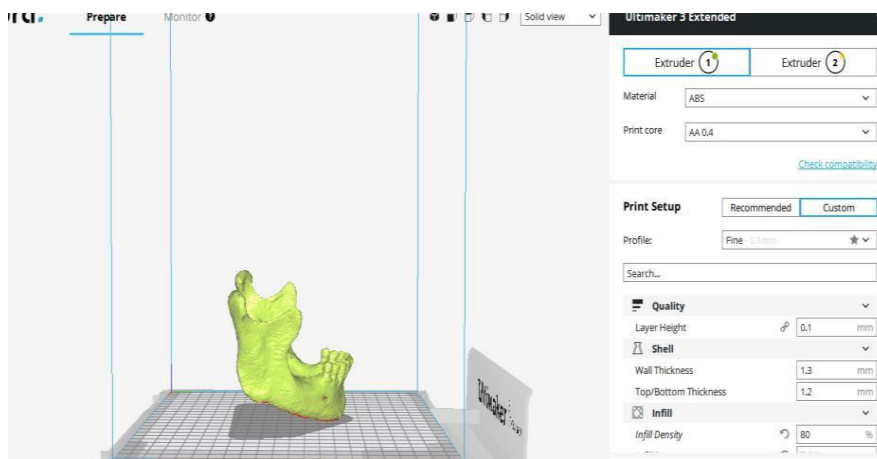


Fig17. showing the jaw imported in the CURA software.

The G-code program is generated in Cura software to manufacture the jaw and this code are stored in the drive and setup is open in a machine that estimates the time required for building the model in the machine and printing is started as shown Fig 19

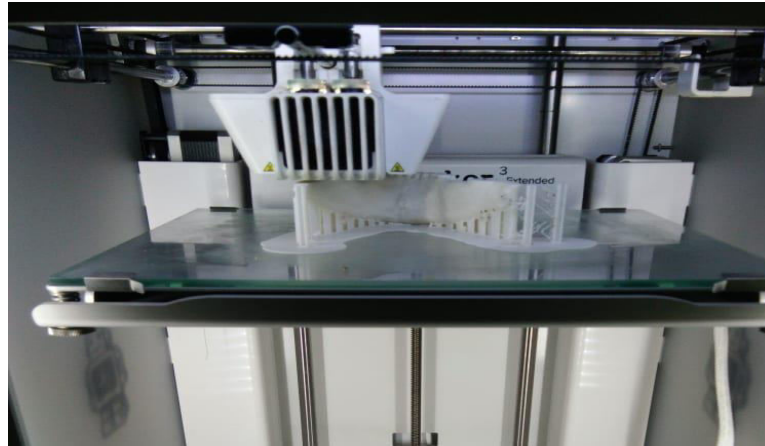


Fig 18 Printing the jaw prototype in Ultimaker 3D extended

This took about 1day,22hours of time to create the jaw, and the fine model of the jaw as the prototypic implant is manufactured using the STL file model created in the medical software slicer. Once manufacturing the model completely, the created model is shown in the Fig 20

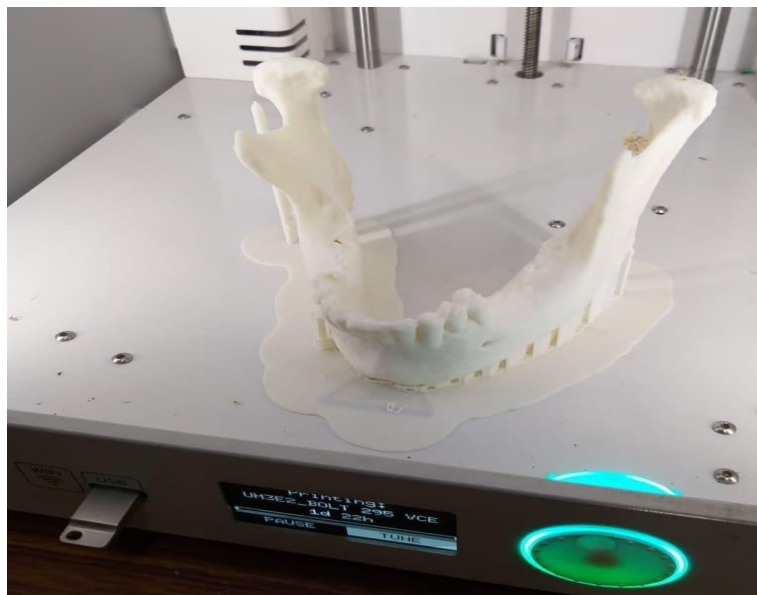


Fig 19 (3D printed prototypic jaw)

V. CONCLUSION AND DISCUSSION

The following conclusions can be derived from the detailed thesis report :

1. With the involvement of 3D printing technology, procedures can be successfully accomplished. MRP (medical rapid prototype) models' productions with 3d printing provides the low-cost anatomical models which are helpful in surgical planning. This fabrication reduces the time consumption for surgical procedures to rectify the problem caused and this would be great help for the dentists to understand the anatomy of the mandibular defects.
2. 3D printing gives the real time experiences to practise the surgery on the prototype which nearly replicate the actual body part structure. The static structural analysis was carried on the mandible with input data of different materials suitable for implant under two boundary conditions using Ansys Workbench2019R3. The deformation on the jaw with properties of bone found to be 912.4 mm as maximum under fixed end condition and 96026mm

maximum under hinged condition and Equivalent stress (Von-Mises) were 0.01003 MPa min and 2948.3MPa max for fixed end and 0.04155MPa min and 1.086e5 MPa max values for Hinged conditions.

3. Titanium alloy has the suitable properties to be replaced by the bone but due to its high cost it would not be affordable. The next alternate material for the bone implants, are of PEEK has the lesser cost and nearly matches the values as that of bone.

REFERENCES

1. Reconstruction skeleton for the lower human jaw using CAD/CAM/CAE. A.T. Abbas. (2011)
2. Fabricating CAD/CAM implant-retained mandibular bar overdentures: a clinical and technical overview. Chui Ling Gool and Keson Beng Choon Tan² (2017)
3. Biomechanical stresses and the strain analysis of mandibular human region from computer tomography to custom implant development. Rafael Ferreira Gregolin, Cecília Amelia de Carvalho Zavaglia, Ruís Camargo Tokimatsu, and João A. Pereira (2017)
4. Design and analysis of 3D customized models of the human mandible.
5. Massimo Martorelli, Saverio Maietta, Antonio Gloria, Roberto De Santis, Eujjin Pei (2015)
6. Geometrical parameters of the mandible in 3d CBCT imaging Wojciech Ryniewicz, Anna Ryniewicz, Lukasz Bojko
7. Finite element analysis and additive manufacturing of patient-specific cranial implants. G. Pradeep Vasavi College of Engineering. (2018)
8. Failure location prediction by finite element analysis for an additive manufactured mandible implant. Jinxing Huo, Per Derand, Lars- Eril Rannar, Jan Micheal, E.Kristofer-Gamstedt.(2015).
9. 3D bio-cad modeling of human mandible and fabrication by rapid prototyping technology Mustafa Aya, Tinaz Kubatb, Cagri Delilbasic, Bulent Ekicid, Huseyin Emir Yuzbasioglu, Selim Hartomacioglu. (2013).
10. An Investigation of Two Finite Element Modeling Solutions for biomechanical simulation using a case study of a mandibular bone Journal of Biomechanical Engineering DECEMBER 2017, Vol. 139 / 12100
11. Modeling and fabrication of 3d printed prototype model of the lumbar spine for study and training of endoscopic surgery thesis work Vidyasagar, Dr.L. Siva Rama Krishna Associate Professor Osmania University.
12. Parametric modeling of the human mandible lateral body for the shape optimization of its section in the case of the bone augmentation. Suci Mircea, Cazacu Razvan, Bica Cristina Ioana. (2014).



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