Three Dimensional Modeling and Finite Element Analysis of Conventional Type Miniplates Fixed at Mandibular Fractures

T. HIMA BINDU[1], L. SIVA RAMAKRISHNA[2], P. RAVINDER REDDY[3], M.TEJDEEP[4], B. KARTHIK ANAND[5]

[1]Dept. of Mechanical Engineering, University College of Engineering, Osmania university, Hyderabad, Telangana, INDIA, hima23382@gmail.com

[2]Dept. of Mechanical Engineering, University College of Engineering, Osmania university, Hyderabad, Telangana, INDIA, lsrkou@gmail.com

[3]Dept. of Mechanical Engineering, ChaitanyaBharathi Institute of Technology (CBIT), Gandipet, Hyderabad, Telangana, INDIA, reddy_prr@yahoo.com

[4]Dept.of Oral and Maxillofacial Surgery, SVS Institute of Dental Sciences, Mahabubnagar, Telangana, INDIA, tejdeep.muthuluri@gmail.com

[5]Dept. of Mechanical Engineering, Vignana Bharathi Institute of Technology (VBIT), Ghatkesar, Hyderabad, Telangana, INDIA, karthikanand.mech@gmail.com

*Abstract:-*Mandibular fractures mostly consist of the traumatic injuries occurring at the mandible, treating these fractures by Oral and Maxillofacial surgeons is one of the most challenging tasks for them. The aim of this paper is to do Finite Element Analysis (FEA) on the conventional type miniplate and screws which are fixed to join fracture occurring at the symphysis (mid-line) area of the mandibular bone subjected to uniform loading. The Computer Tomography (CT) scanned images of a patient are collected in Digital Imaging and Communications in Medicine (DICOM) format. These images are then imported into image processing software MIMICS to develop the 3D surface model of the mandible and importinto the 3-Matic software for modifying the surface model and then finally saved in .STL file format. The saved .STL file format is imported into Solidworks software to develop 3D CAD model consisting assembly of the mandible, conventional type of miniplate and screws.The 3D CAD model of the assembled file was imported into ANSYS workbench and analyzed by applying uniformly distributed load (UDL) on the teeth area of mandible. The input given for Finite Element Analysis was the material properties of the cortical bone of the mandible andfor miniplates and screws the material properties of titanium alloy Ti-6Al-4V ELI (Grade 5) of the ASTM standard F136. In this paper, five different cases were considered and analyzed under static structural loading conditions by applying UDL.