

FEA AND 3D PRINTING OF PELVIS BONE

A project report submitted In the partial fulfillment of the requirements for award of degree of

BACHELOR OF TECHNOLOGY IN MECHANICAL ENGINEERING

Submitted by

R.VENKU NAIDU	18815A0353
S.PRASAD	18815A0358
G.UDAY CHARAN	18815A0379
K.SAI RAM PRASAD	18815A0386

Under the guidance of

P.RAMANA BABU

Assistant professor

DEPARTMENT OF MECHANICAL ENGINEERING



AVANTHI INSTITUTION OF ENGINEERING AND TECHNOLOGY

(PERMENANTLY AFFILIATED TO JNTU-KAKINADA, ACCDREDITED BY NBA & NAAC, APPROVED BY AICTE, RECOGNISED BY UGC 12F & 2b)

(Affiliated to Jawaharl Nehru technological university Kakinada, A.P)
TAMARAM, MAKAVARAPALEM, NARASIPATNAM-531113

2018-2021

DEPARTMENT OF MECHANICAL ENGINEERING

AVANTHI INSTITUTION OF ENGINEERING AND TECHNOLOGY



CERTIFICATE

This is to certify that the project entitled **“FEA AND 3D PRINTING OF PELVIS BONE”** is this record of the work carried out **R.VENKU NAIDU (18815A0353)**, **S. PRASAD (18815A0358)**, **G.UDAY CHARAN (18815A0379)**, **K.SAI RAM PRASAD (18815A0386)** students of final year B.Tech in the department of Mechanical engineering . This work is done for the partial fulfillment for the award of BACHELOR OF TECHNOLOGY during year 2018-2021.

P. R. Baby

Project Guide

Head of the Department

External Examiner

ABSTRACT

In the musculoskeletal system, the pelvis is one of the most important components. The entire body rests on this construction. The role of the pelvis is to transfer gravitational and external load across the sacro-iliac joints and the hip joints. The purpose of this study is to observe the stress and strain distribution in the pelvic bone. Due to its complex geometry and structure, the biomechanics of pelvis is complicated. The Finite Element Method (FEM) can be used to analyze this type of very complex geometries. To get the realistic results, the finite element analysis have to be accomplished with a three-dimensional model similar to the shape and architecture of the Hip bone of Pelvis Joint. Model that was created using CT Scan data and available on internet was used. For proper understanding of geometry, the model is 3D Printed. Simulations were executed to study the response of hip bone subjected to various body weights. An analysis was executed to examine the failure load for this bone.