CRASH ANALYSIS OF FRONTAL PART OF CAR BODY USING ANSYS

M.Tech Dissertation

By

SOLANKI PREYANSHU BHANUKUMAR

180303206010

Under the supervision of

Prof. SwatiPrajapati

April 2020

DEPARTMENT OF MEHANICA ENGINEERING

PARUL INSTITUTE OF ENGINEERING & TECHNOLOGY

FACULTY OF ENGINEERING & TECHNOLOGY

PARUL UNIVERSITY

P.O. Limda – 391 760, GUJARAT, INDIA

ABSTRACT

With the headway of society, people have a regularly expanding number of solicitations for vehicle wellbeing, which requires the difference in vehicle structure and body material to satisfy the above vehicle security prerequisite during a car accident a few sections will have plastic distortion and ingest a ton of vitality. Henceforth it gets important to check the vehicle structure for its accident capacity with the goal that security is accomplished. A significant worry of both the car business and government is the progression of vehicles that would utilize less non-sustainable power source, alongside these essential focuses; the security is likewise a significant worry for every cutting edge vehicle. The idea of Finite Element Analysis of vehicle body has been featured right now. Presently In this task we are going to plan a vehicle Hatch back by utilizing configuration demonstrating apparatus Creo Parametric programming, Crash examination on the vehicle body will be act in Ansys workbench programming by Explicit Dynamic module by utilizing diverse material. On various velocities of vehicle body, we will contemplate the, stresses framed because of crash, twisting and regions of disfigurement of vehicle body and strain will be found after investigation.Right now will learn about various material utilized in vehicle body and their properties by which we will be thought about best material for the future vehicle. This idea is used for upgrading light weight vehicle which accomplish wellbeing highlight with economy.