**CFD ANALYSIS ON ROCKET NOZZLE**

**ABSTRACT**

A nozzle is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube with variable cross-sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. The nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and low temperature. Our study is carried using software’s like SOLIDWORK for designing of the nozzle and fluent 6.3.2 for analyzing the flows in the nozzle. Numerical study has been conducted to understand the air flows in a conical nozzle at different divergence degrees of angle using two-dimensional axisymmetric models, which solves the governing equations by a control volume method. The nozzle geometry co-ordinates are taken by using of method of characteristics which usually designed for De-Laval nozzle. The present study is aimed at investigating the supersonic flow in conical nozzle for Mach number 3 at various divergence degree of angle. The throat diameter and inlet diameter is same for all nozzles with various divergence degree of angles. The flow is simulated using fluent software. The flow parameters like pressure, Area of nozzle at exit are defined prior to the simulation. The result shows the variation in the Mach number, pressure, temperature distribution and turbulence intensity.