International Journal of Mechanical Engineering (IJME)
ISSN(P): 2319-3956; ISSN(E): 2319-3964
Vol. 13, Issue 1, Jan – Jun 2024; 21–26

International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations

CFD ANALYSIS OF COMPRESSIBLE FLOWS IN A CONVERGENT- DIVERGENT NOZZLE

Sathish. V[1], Dr. M. Makesh[2] & D. Sundar[3]

[1]M E-Engineering Design, Paavai Engineering College, Namakkal

[2]Professor of Mechanical Department, Paavai Engineering College

ABSTRACT

A nozzle is a mechanical device that utilizes pressure energy and fluid enthalpy to increase the outflow velocity and control fluid flow direction. To obtain the nozzle duct's shock pattern, the flow inside the nozzle must be supersonic with a Mach number greater than one. Experimentally, the shock pattern is obtained for a nozzle with a Mach number of 2 and a nozzle pressure ratio (NPR) equivalent to 7 and below. For Mach M = 2, the necessary NPR is equal to 7.82 for correct expansion. When the NPR is greater than 7.82, flow from the nozzles is under-expanded. For NPR less than 7.72, the flow from the nozzle is over-expanded. In this paper, the computational fluid mechanics (CFD) technique was used to simulate the nozzle flow based on experimental investigation. A two-dimensional transient compressible flow of air through a supersonic nozzle was simulated using ANSYS Fluent software. A time-dependent flow using the density-based implicit solver was employed to analyzethe simulation results. The results illustrate that the CFD technique simulates fluid flows and the formation of shock in a duct, providing useful information about fluid dynamics analysis.

KEYWORDS: *CFD Analysis, Nozzle*

Article History

Received: 03 May 2024 | Revised: 08 May 2024 | Accepted: 10 May 2024
