Experimental verification of flow through convergent nozzle using CFD simulation

R Jayakumar[1], Harish H.V[2], Santhosh N[3], NishantDesai[4], Srikanth H V[5], Shrilakshmi V Bentur[6], Sharmitha[7]

[1]*Professor & Head of Department of Aeronautical Engineering, NMIT, Bangalore*
[2,3,4,5] *Assistant Professor Department of Aeronautical Engineering, NMIT, Bangalore*
[6,7]*U G Student, Department of Aeronautical Engineering, NMIT, Bangalore*

Abstract— **Nozzle is a device where pressure energy is converted to kinetic energy. In this paper, the aim is to analyze the behavior of flow parameters through a convergent nozzle. The variation of pressure and velocity distribution profile inside the nozzle is experimentally obtained for an exit velocity. The resultobtained from experimental setup is validated through CFD simulation. For CFD simulation, ICEM-CFD was used for modelling& meshing, CFX-13 for solving and CFD POST for plotting the results.**

Key words: **Nozzle, CFD, Modeling, Meshing, Simulation.**

I. INTRODUCTION

A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. Nozzle is often a pipe or tube of varying cross sectional area and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them.A gas jet, fluid jet, or hydro jet is a nozzle intended to eject gas or fluid in a coherent stream into a surrounding medium. Gas jets are commonly found in gas stoves, ovens, or barbecues. Gas jets were commonly used for light before the development of electric light. Other types of fluid jets are found in carburetors, where smooth calibrated orifices are used to regulate the flow of fuel into an engine.

Nozzles can be described as *convergent* (narrowing down from a wide diameter to a smaller diameter in the direction of the flow)Convergent nozzles accelerate subsonic fluids. If the nozzle pressure ratio is high enough, then the flow will reach sonic velocity at the narrowest point (i.e. the *nozzle throat*). In this situation, the nozzle is said to be *choked*.The highest velocity that can be achieved by a converging nozzle is sonic velocity, which will occur at the exit of the nozzle.

In order to reach supersonic velocities, it is required to add a diverging section to the exit of a converging nozzle. If Mach number is less than unity,then it is a subsonic fluid flow, so subsonic velocity is possible with the help of convergent nozzles.

EXPERIMENTAL SETUP

Figure 1; Convergent Nozzle experimental Setup

The specifications of convergent nozzle used in the experimental setup are given in the table 1.

Table 1: Specifications of Experimental Setup

Parameters	Dimensions
Inlet diameter of nozzle	70mm
Exit diameter of nozzle	55 mm
Length of the nozzle	162mm
Blower	1 HP
Nozzle Type	Convergent
Maximum Velocity	45 m/s

II. EXPERIMENTAL AND CFD SIMULATION RESULTS

Experiment was conducted for a constant velocity of 40 m/s at the nozzle exit.

Table 2: Experimental Results Obtained

Sl no	H_w in mm	H_a in mm	Exit Velocity in m/s	Exit Pressure in Pascal
1	196	167.5	25.6	101868
2	198	169.2	26.37	101848
3	203	173.5	27.92	101799
4	206	176.0	28.81	101770
5	210	179.4	29.9	101732
6	215	183.7	31.32	101682
7	223	190.5	33.36	101604
8	233	199.1	35.81	101505
9	250	213.6	39.5	101339
10	252 (ambient)	215.3	40	101320

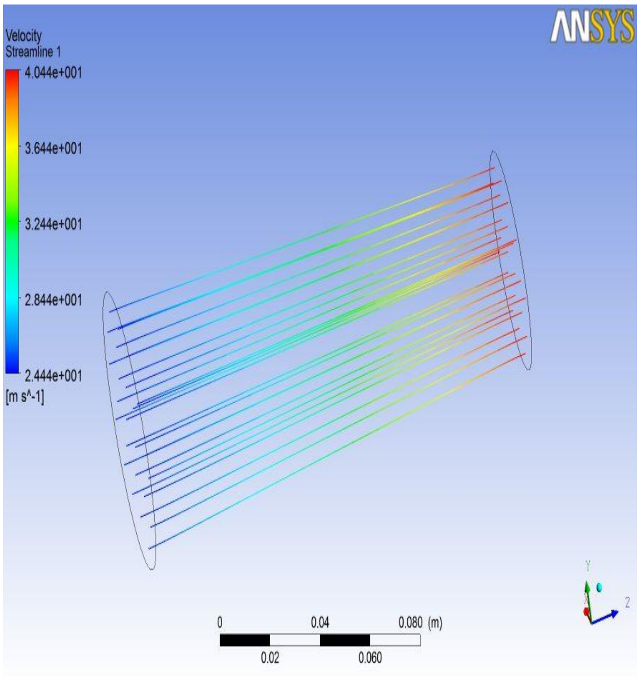

Figure 3; Velocity streamlines of convergent nozzle

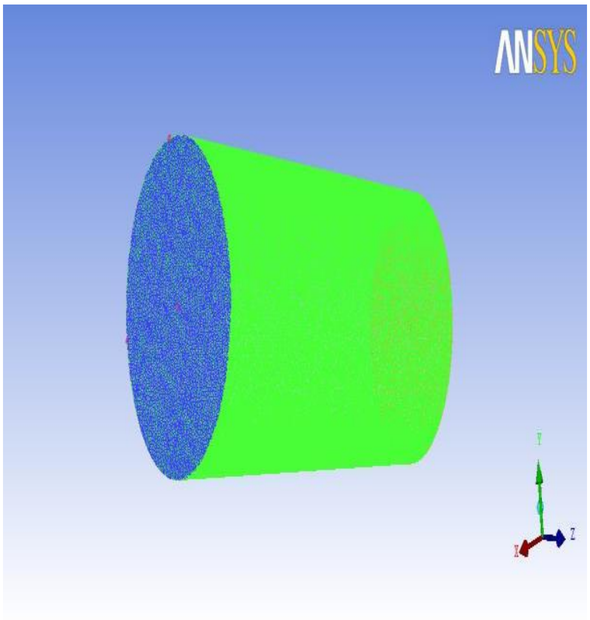

Figure 2; Convergent nozzle model meshed

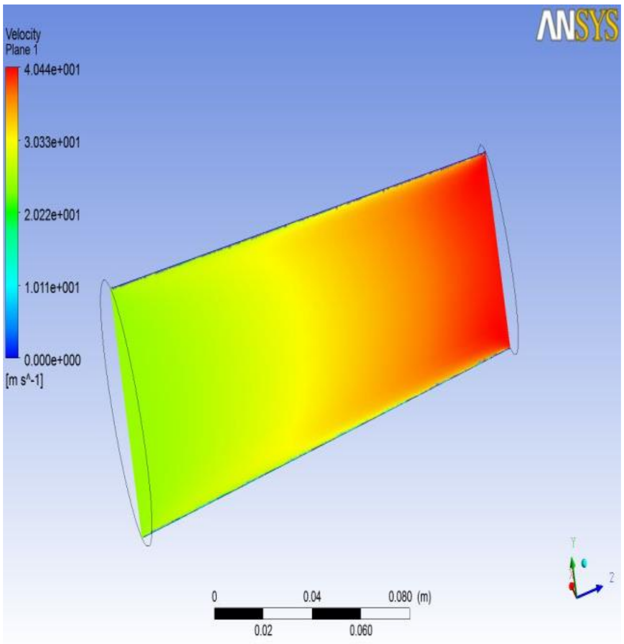

Figure 4; Velocity plane of convergent nozzle

III. CONCLUSIONS

From the above results, experimental results have been validated withCFD simulation, where velocity and pressure distribution profiles were clearly matching.There is a good agreement between the experimental and simulated results for velocity and pressure distribution profiles. Although there are some small discrepancies (less than 5%) due to some experimental imperfectness, we still have a good confidence in the CFD simulation program that can be used in the future.