EXPERIMENTAL INVESTIGATION ANDSIMULATION OF AIR FLOW MOTION BY NOZZLE WITH EXTENDED SURFACES

Ravichandran V[1], Vinoth S[2], Vijayaraj T[3]

[1] *U G Student Department of Mechanical Engineering, PSVPEC, Chennai, India*

[2] *U G Student Department of Mechanical Engineering, PSVPEC, Chennai, India*

[3] *Assistant Professor Department of Mechanical Engineering, PSVPEC, Chennai, India*

Abstract

The aim is to enhance the air flow for better combustion. In this, the air flow is passed through the convergent nozzle. Here the extended surface were used to interrupt the air flow. It is attached at the end of the convergent nozzle, which creates turbulence in the air flow motion. The nominal jet core velocity of the air flow is reduced inside the combustion chamber. The air flow is simulated by using ANSYS 16.1 CFD software and also it will be experimentally investigated with the help of experimental setup. This may increase the mixing enhancement of air fuel mixture by decreasing the unburnt fuel at the exhaust.

Keywords – *Enhancement, Turbulence, Convergent nozzle, Jet core.*

I. INTRODUCTION

A nozzle is a tube of varying cross-sectional area (usually axisymmetric) aiming at increasing the speed of an outflow, and controlling its direction and shape. Nozzle flow always generates forces associated to the change in flow momentum, as we can feel by handholding a hose and opening the tap. In the simplest case of a rocket nozzle, relative motion is created by ejecting mass from a chamber backwards through the nozzle, with the reaction forces acting mainly on the opposite chamber wall, with a small contribution from nozzle walls. As important as the propeller is to shaft-engine propulsions, so it is the nozzle to jet propulsion, since it is in the nozzle that thermal energy (or any other kind of high-pressure energy source) transforms into kinetic energy of the exhaust, and its associated linear momentum producing thrust.

The flow in a nozzle is very rapid (and thus adiabatic to a first approximation), and with very little frictional loses (because the flow is nearly one-dimensional, with a favourable pressure gradient except if shock waves form, and nozzles are relatively short), so that the isentropic model all along the nozzle is good enough for preliminary design. The nozzle is said to begin where the chamber diameter begins to decrease (by the way, we assume the nozzle is axisymmetric, i.e. with circular cross-sections, in spite that rectangular cross-sections, said two-dimensional nozzles, are sometimes used, particularly for their ease of direction ability). The meridian nozzle shape is irrelevant with the 1D isentropic model; the flow is only dependent on cross-section area ratios.

II. RELATED WORK

Syed Ashfaq et al. (2014) [1]: suggested that the flow from converging nozzle to suddenly expanded circular duct of larger cross-sectional area than that of nozzle exit area, focusing attention on the base pressure and the flow development in the duct. Mach number and nozzle pressure ratio are considered as the flow parameters. The geometrical parameters considered are the area ratio of the sudden expansion duct cross-section area to the nozzle exit area and the length to diameter ratio of the duct. To find the effect of micro jets as an active control on base pressure as well as on the flow field developed in the duct, the micro jets of 1 mm orifice diameter are used. The geometrical parameter, the length to diameter ratio of the enlarged duct was varied from

10 to 1, and tests were conducted for L/D 10, 8, 6, 5, 4, 3, 2 and 1. When the micro jets were activated they are found to influence the base region, taking the base suction to considerably higher values compared to that for without control case for most of the cases.

R. Jagannath et al. (2007) [2]: suggested that pressure loss in suddenly expanded ducts is studied with the help of fuzzy logic as a tool. It is observed that minimum pressure loss takes place when the length to diameter ratio is one and it is seen that the results given by fuzzy logic formulation are very logical and it can be used for qualitative analysis of fluid flow in flow through nozzles in sudden expansion. The base pressure and the flow field downstream of the base are dictated by the vortex dynamics triggered by the sudden expansion of the flow in the enlarged duct. From this extensive experimental study in subsonic and supersonic flows, he concluded that the base pressure would be equal to the entrance pressure if the velocity was subsonic, but if the entrance flow was supersonic, the base pressure could be equal to or less than or greater than the entrance pressure. The base flow occurred because of the pressure difference across the shock wave originating where the jet strikes the wall. Korst investigated the problem of base pressure in transonic and supersonic flow, for the case in which the flow approaching the base is sonic or supersonic after the wake. They developed a theory to predict Mach number in a downstream location of sudden enlargement of known values of Mach number at the exit of the inlet tube, with compressible flow assumptions; they also assumed that the pressure across the face of the enlargement was equal to the static pressure in a small tube just before the enlargement. This study can be extended to different nozzles having different geometries with variations in Mach no, pressure ratio and area ratio. It is observed that L/D ratio is 1 for pressure loss for Mach 1.58, 1.74, 2.06 and 2.23 if the consideration is given only to pressure loss parameter.

S. R. Balakrishnan et al. (2013) [3]: carried out the development of a new design algorithm for a modern gas turbine combustor. The algorithm includes a set of preliminary design procedures involving the use of perforated plate models. The perforated plate model captures complex processes such as, chemical reaction, jet mixing, and produces more turbulence. Due to the turbulence the fuel mixes with air thoroughly and gives better combustion efficiency. The preliminary design procedures are verified using the advanced numerical techniques of computational fluid dynamics (CFD). These techniques are used to solve the swirling flow field inside the pre-mixer, the reacting flow field inside the liner, and the rate of turbulence level inside a combustion chamber. The perforated plate produces a central recirculation by which Fuel and air must move slowly enough for the flame to propagate upstream and ignite fresh mixture. The point at which the flame can no longer propagate back through the flow is the stabilization point or anchor. Zones of flow reversal help stabilize the flame by creating localized regions of low velocity flow called flame holders. This hot gas helps stabilize the flame by providing a continual source of ignition to the incoming fuel. It also serves as a zone of intense mixing within the combustor by promoting turbulence through high levels of shear between the forward and reverse flows. It is found that 30° holed perforated plate is the best for producing appropriate recirculation zone with reasonable pressure drop. This perforated plate is reasonably performing with the results of Swirler.

Rotaru Constantin et al. (2015) [4]: observed that in an aircraft turbojet engine, for aviation fuels consisting of hydrocarbon mixtures. Recent achievements in chemical kinetic modelling of kerosene combustion are presented. Performance parameters investigated were combustion efficiency, unburned hydrocarbons as well as liner temperatures and smoke. The analysis is based on the theoretically and experimentally characteristics of a typical axial flow engine. The design of an aircraft or a gas turbine engine requires the knowledge of multiple academic disciplines including mathematics, aerodynamics, fluid mechanics, solid mechanics, thermodynamics, chemistry and material sciences. The thrust of the present aircraft engine is developed by compressing air in the inlet and compressor, mixing the air with fuel, burning the mixture in the combustor and expanding the gas stream through the turbine and nozzle. The expansion of gas through the turbine supplies the power to turn the compressor. The net thrust delivered by the engine is the result of converting internal energy to kinetic energy. The thermal energy of the air/fuel mixture flowing through an airbreathing engine is increased by the combustion process.

Marius Enache et al. (2017) [5]: developed the design of an annular combustion chamber for a micro gas turbine engine. The combustion chamber is designed for using biogas as fuel. It is designed based on the constant pressure, enthalpy addition process. The present methodology deals with the computation of the initial design parameters and arriving at optimized values. Then the dimensions of the combustor are calculated based on different empirical formulas. The air mass flow is then distributed across the zones of the combustor. The aerodynamic flow characteristics are numerically simulated by means of the ANSYS CFX software. The air-fuel mixture, combustion turbulence, the thermal and cooling analysis is carried out. The combustion chamber, or combustor, of a gas turbine, is the device that receives the pressurized air from the compressor and promotes its mixture with the fuel in order to release the heat energy through a combustion reaction. Gas turbines work with a high excess of air, usually out of the flammability limits, and so a flame tube, or liner, is used to improve the distribution of air through the reactor. Basically, the liner divides the combustion chamber into three zones: primary zone, secondary or intermediate zone and dilution zone. At the primary zone, a recirculation zone shall be developed to ensure the stability of the flame. The main function of the primary zone is to anchor the flame and provide sufficient time, temperature, and turbulence to achieve essentially complete combustion of the incoming fuel-air mixture.

III. OBJECTIVES

1. To enhance the air flow for better combustion.
2. To reduce the jet core velocity of air flow.
3. To increase the air fuel mixing than the nominal mixing.
4. To reduce the emissions of harmful gases at exhaust.
5. To increase the thermal efficiency.

IV. WORKING PRINCIPLE

The components used are air compressor, settling chamber with pressure gauge, nozzle, extended surfaces and experimental setup. The air from the atmosphere is compressed by the compressor and it is stored in the storage tank. The compressed air is stored in the tank. This is provided with the pressure gauge to monitor and regulate the flow of air. The gate valves are kept at both the inlet and outlet in order to control the air flow to the nozzle. The nozzle is provided with the extended surfaces at the end, the compressed air is fed through it form the settling chamber. The required air flow is obtained with the help of the pressure gauge and valve. The extended surfaces are provided at the end, which disturbs or steers the air which flow through the combustion chamber. It makes the air flow as turbulent flow. The pressure and velocity values are measured from the experimental setup with the help of data acquisition system. Similarly, other type of extended surfaces are placed and flow is fed through it and valves were measured from the experimental set up. The experiment is carried out for different mach numbers. Then, the table is made by the measured valves of different mach number and the graph is plotted. The simulation is done, by using the computational fluid dynamics (CFD) software and the results were taken for air flow with different mach number. The measured values from the experimental setup and the result from the simulation values are compared.

V. DIAGRAM

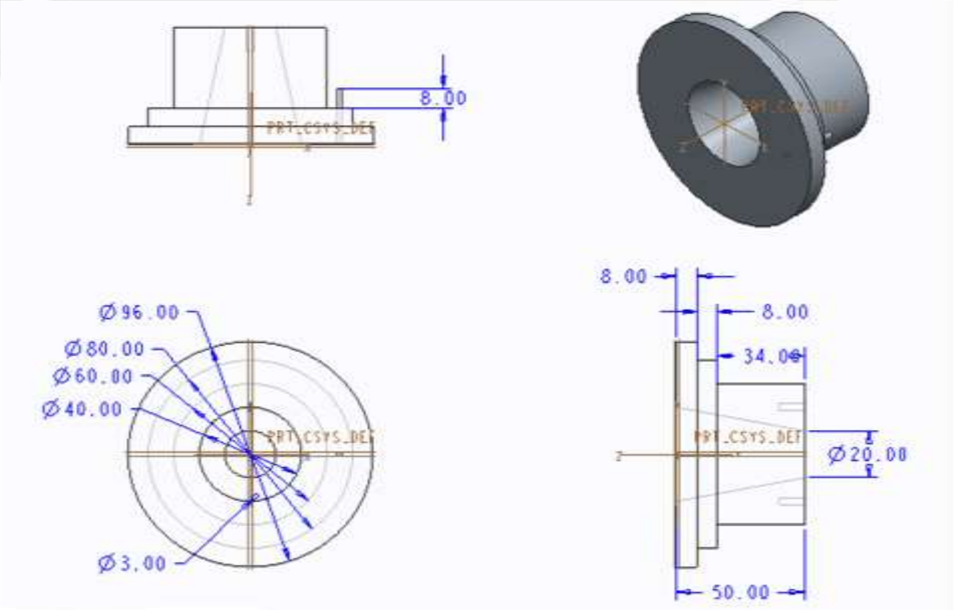

Figure 1. Nozzle

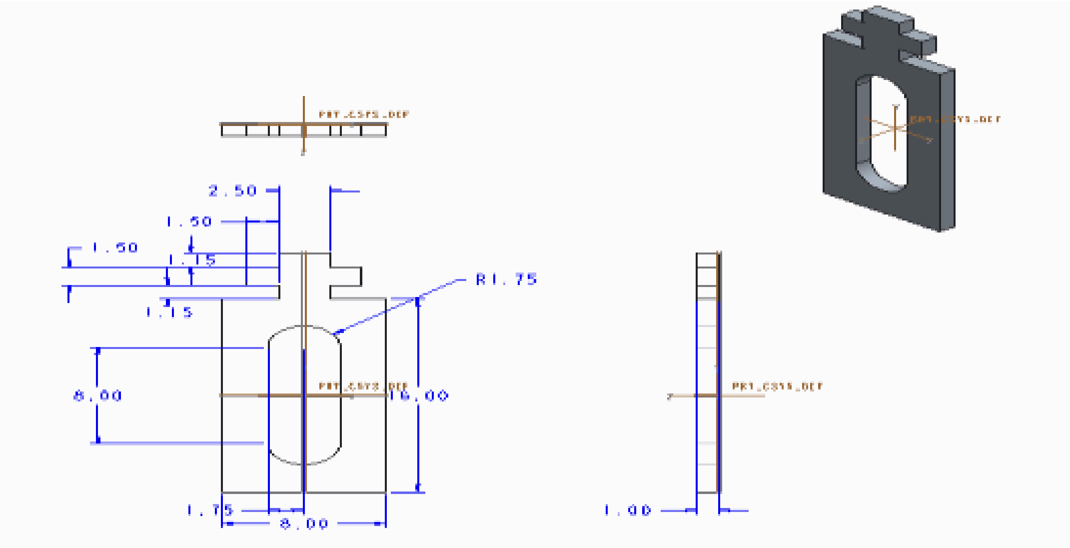

Figure 2. Square surface

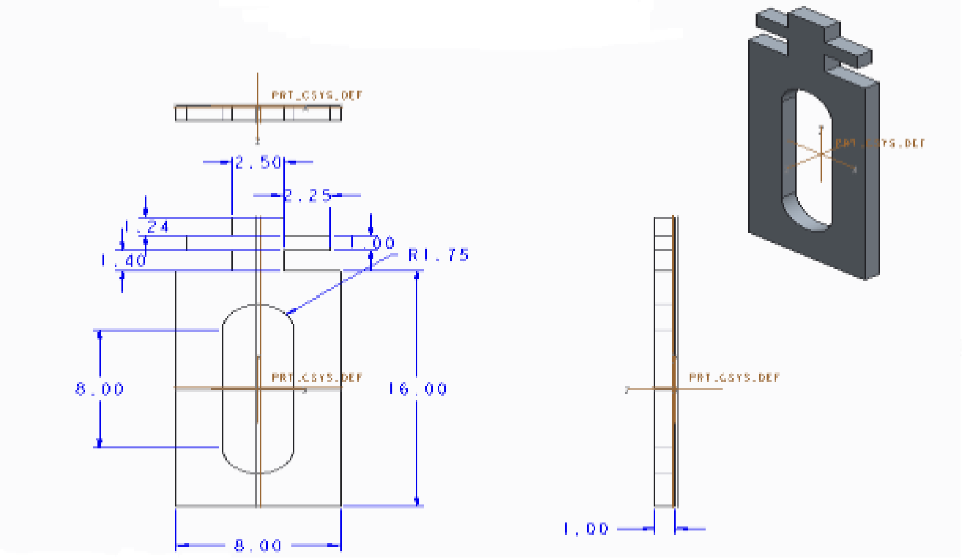

Figure 3. Rectangular surface

VI. DESIGN CALCULATION

Nozzle calculation:

Diameter $d_1 = 40$ mm
$d_2 = 20$ mm

Area $A_1 = \pi/4\ (d_1)^2 = \pi/4\ (40^2)$
$A_1 = 1256.63\ mm^2$
$A_2 = \pi/4\ (d_2)^2 = \pi/4\ (20^2)$
$A_2 = 314.16\ mm^2$

Angle $\tan\theta = 10/50$
$\theta = 11.3^\circ$

Rectangular Surface:

Area of side rectangle = 2.25×1
= 2.25mm²

Total area of side rectangle = 2.25×2
= 4.5mm²

Area of center rectangle = 3.8×2.5
= 9.5mm²

Total area of the extended surface = 4.5+9.5
= 14mm²

Total area of the two extended surface = 14×2
= 28mm²

Square Surface:

Area of side square = 1.5×1.5
= 2.25mm²

Total area of side square = 2.25×2
= 4.5mm²

Area of center rectangle = 3.8×2.5
= 9.5mm²

Total area of the extended surface = 4.5+9.5
= 14mm²

Total area of the two extended surface = 14×2
= 28mm²

VII. SIMULATION

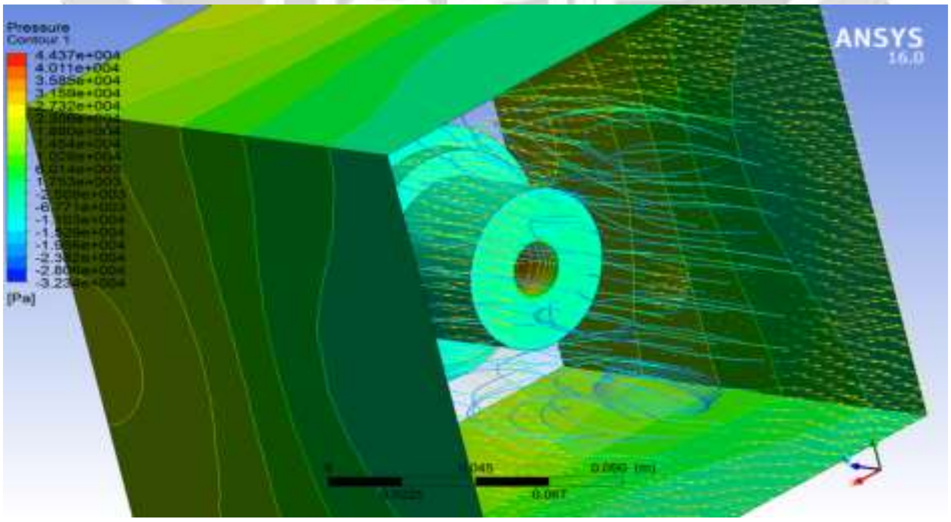

Figure 4. Simulation for pressure values

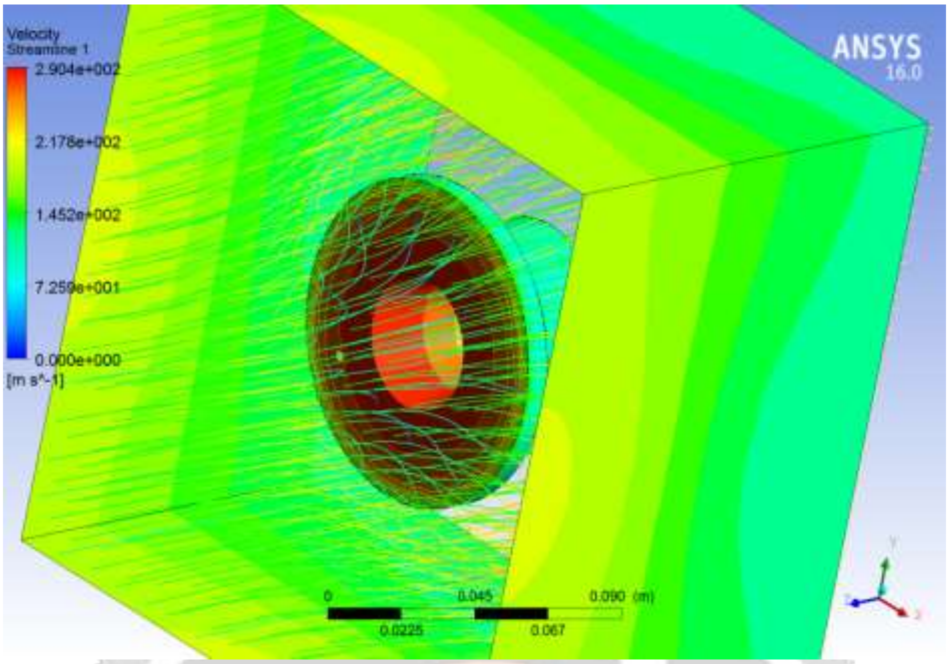

Figure 5. Simulation for velocity values

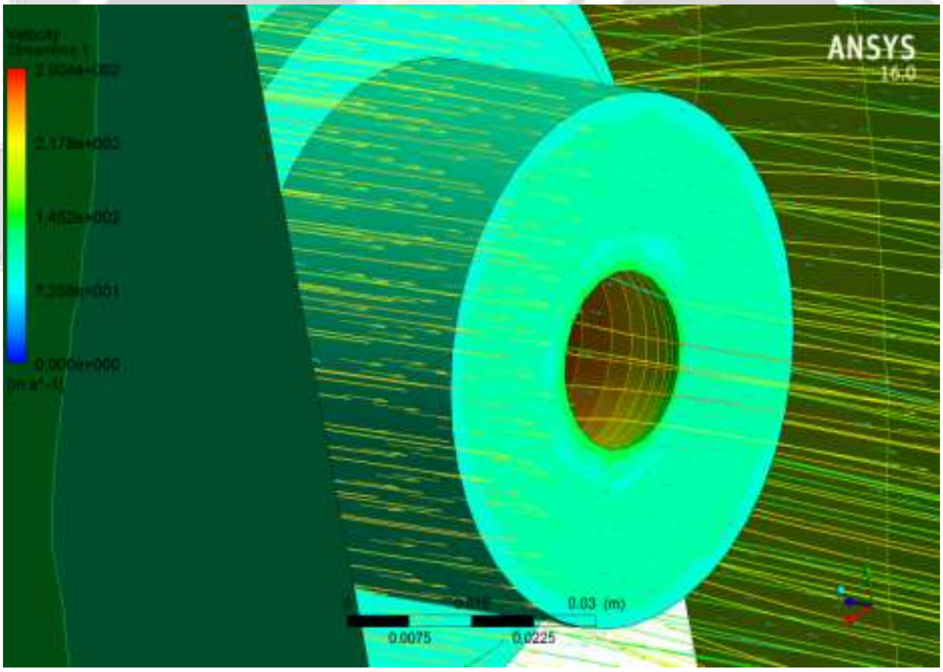

Figure 6. Simulation for velocity values at exit

VIII. EXPERIMENTAL VALUES AND GRAPH

X/D	Pe/P0
0	0.985694
0.25	0.989599
0.5	1
0.75	1
1	1
1.25	1
1.5	1
1.75	1
2	1

Table 1. P_e/P_o and X/D values for Mach number

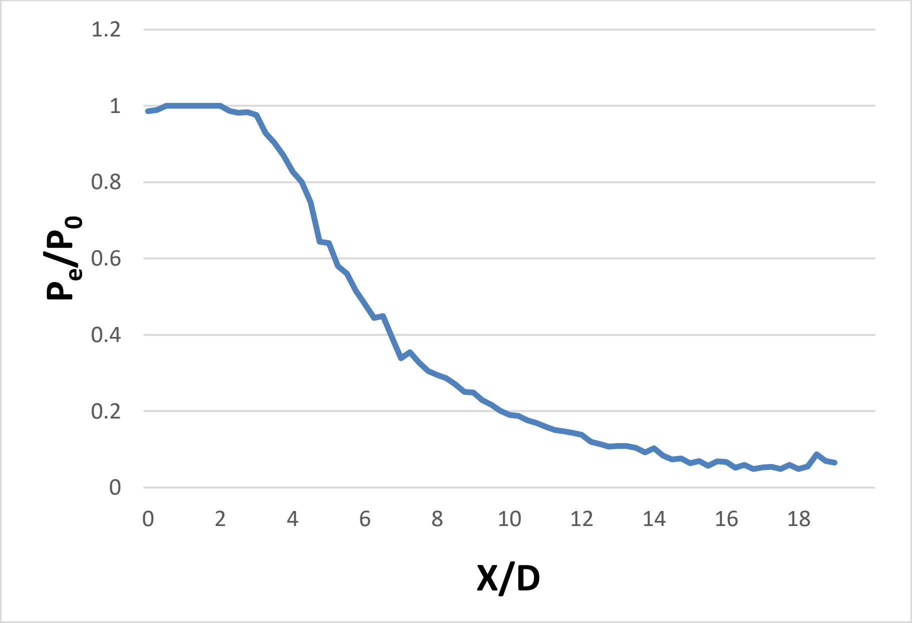

Figure 7. P_e/P_o and X/D values for Mach number 0.4

X/D	Pe/P0
0	1
0.25	0.992659
0.5	0.985112
0.75	0.995368
1	0.99533
1.25	0.980919
1.5	0.985337
1.75	0.987901
2	0.990465

Table 2. P_e/P_o and X/D values for Mach number 0.6

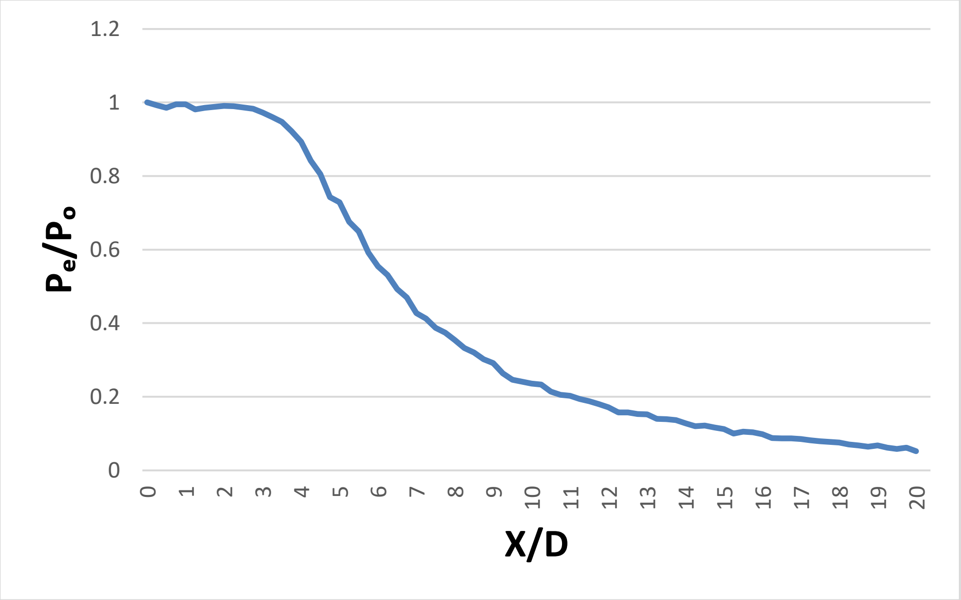

Fig 8. P_e/P_o and X/D values for Mach number 0.6

IX. CONCLUSION

In this paper “Experimental Investigation and Simulation of Air Flow Motion by Nozzle” was successfully designed and the air flow is passed through the convergent nozzle. The extended surfaces is attached at the end of the convergent nozzle, which creates turbulence in the air flow motion. The nominal jet core velocity of the air flow is reduced inside the combustion chamber. The air flow was simulated by using ANSYS 16.1 CFD software and also it was experimentally investigated with the help of experimental setup. This increase the mixing enhancement of air fuel mixture by decreasing the unburnt fuel at the exhaust.

REFERENCES

i. Shafiqur Rehman, S.A.Khan (2008), "Control of base pressure with micro-jets", Aircraft Engineering and Aerospace Technology, pp 158-164.

ii. Korst.H (2015), "Comments on the effects of boundary layer on sonic flow through an abrupt cross sectional area change", Journal of the Aeronautical Science, Vol.21, pp 568-579.

iii. Durst.F, Melling, et al (1974), "Low Reynolds number flow over a plane symmetric sudden expansion", Journal of Fluid Mechanics, Vol.6, pp 724-739.

iv. Yang, B.T and Yu, et al (1983), "The flow field in a suddenly enlarged combustion chamber", AIAA Journal, Vol.16, pp 139-146.

v. Anderson, J. S and Williams, T. T. (1968), "Base pressure and noise produced by the abrupt expansion of air in a cylindrical duct", Journal of Mechanical Engineering Science. Vol. 10, pp 262-268.

vi. Brady, J.F. and Acrivos, A. (1982), "Closed cavity laminar flows at a moderate Reynolds numbers", Journal of Fluid Mechanics. Vol. 115, pp 427.

vii. Raghunathan, S. (1987), "Pressure fluctuation measurements with passive shock/boundary layer control", AIAA journal. Vol. 25, pp 626-628.

viii. Pandey, K.M. (1996), "Study on flow through nozzle in sudden expansion with passive control", Advances in Mechanical Engineering.

ix. Scull, W.E., & Mickelsen, W.R. 1957. Flow and Mixing Processes in Combustion Chambers. Chap. II, pages 226–276 of: Basic Considerations in the Combustion of Hydrocarbon Fuels with Air. National Advisory Committee for Aeronautics. Report 1300.

x. Pandey, K.M. (1997), "Pressure loss in flow through nozzles with sudden expansion: A comparison between supersonic and subsonic flow regime". International conference on advances in Mechanical and Industrial Engineering.

xi. Viswanath, P. R. (1988), "Passive devices for axisymmetric base drag reduction at transonic speeds", Journal of Aircraft, Vol. 25, No. 3, pp. 258-262.

xii. Syed Ashfaq, S. A. Khan and E. Rathakrishnan (2013), "Active Control of Flow through the Nozzles at Sonic Mach Number", International Journal of Emerging Trends in Engineering and Development, Vol. 2, Issue-3, pp. 73–82.

xiii. S. Thanigaiarasu, S. Elangovan, E. Rathakrishnan (February 2010), "Effect of Arc-Tabs on the Mixing Characteristics of Subsonic and Sonic Jets", International Review of Aerospace Engineering, Vol. 3 pp. 1-8.

xiv. Shafiqur Rehman, S.A. Khan (2008), "Control of base pressure with micro-jets: part I", Aircraft Engineering and Aerospace Technology: An International Journal, ISSN: 1748-8842, 80/2, pp. 158-164.