Design Optimization of Fuel Injector Nozzle Used In Diesel Engine

Basawaraj S Hasu[1], B Ravi Kiran[2]
[1, 2] Dept of Mechanical Engineering
[1, 2] AVN Institute of Engineering & Technology, Hyderabad, India

Abstract- *Fuel injection system is used for supplying fuel into the combustion chamber for different engines used for the purpose of automotive and also for industrial purpose. In diesel engines mainly theses type of fuel injectors are used and coming to petrol engines it may or may not as it can be alternative.*

The fuel is injected into the combustion chamber with the help of nozzle which is an integral part of the fuel injection system. The nozzle is used to convert the potential energy of the gas into kinetic energy in turn chemical and thermal energy generated in the combustion chamber. The nozzle is major part which supplies the gas with high velocity at low pressure and temperature. Nozzle is used to control the rate of flow, speed and direction of the stream According to the consideration of engines there are different types of Nozzles. Convergent nozzle is the most commonly used nozzle in diesel engines. In this project CFD analysis of the convergent nozzle changing with the different diameters and with different mass flow rates is to be performed. Later the results involving the pressure drop, velocity, and mass flow rate for the fluid are determined and compared for the different diameters. The design required for the CFD analysis is done is NX-CAD and the CFD analysis is performed in Ansys software.

Keywords- Design, CFD Analysis

I. INTRODUCTION

As the technology improved a lot in these years automobile sector is meant to be meeting ecological and economical conditions and requirements for modern engines which are manufacturing with advance technology. The parameters like performance-fuel consumption ratio, low maintenance costs should enable at operations under prescribed emission regulations. As this process can be followed for reducing of combustion pollutants and noise emissions will mainly controlled by the process of fuel injection, so a lot of effort must put into the development of new and improvement of existing diesel fuel injection systems.

In in-line fuel injection system the pump characteristics are mainly affected due to the change of cam profile and the geometry of plunger. The above mentioned pump characteristics and the geometry of the injection nozzle influence the injection characteristics and further the fuel spray characteristics. Even though in recent years many researchers have been made considering the fuel injection systems, as well as their application in variety of the motor vehicles, the fuel injection nozzle still presents quite an un researched area.

The part of the nozzle downstream of needle seat is almost the same whether the nozzle is made for a common rail injection system or unit injector system or even for the in-line FIS.

Therefore the results of the researches on one type of the fuel injection system can be applied on almost every system. In manufacturing a nozzle which is used in fuel injector the nozzle parameters can be improved by conducting CFD analysis, the different characteristics of the nozzle can be examined.

If the value of flow coefficient is higher, also it makes larger fuel quantity is injected per time, which may also vary the higher outflow velocity (suppose no changes in outflow cross-section) and by better atomization.

II. METHODOLOGY

An injector nozzle is used to inject the liquid propellant (liquid-diesel) into the ignition chamber, though it acts as a main part to send the required air-fuel ratio inside the chamber.

The main objective is to examine the results for different nozzle diameters and for different velocities to know the results like velocity outlet, pressure outlet and mass flow rate.

- Initially the nozzle is designed in a solid body. By using Nx shape studio, the inside diameter of nozzle is designed in a surface face.

- Now the surface body created is converted into Ansys file format
- Using workbench we conduct CFD analysis on the designed geometry.
- Working on different diameters and for different velocities will take place in this step
- Examining all the results and tabulated into a table and make a proposal of which one is better for manufacturing.

III. 3D MODELING OF ROTOR SHAFT

As it was focused on the nozzle parameters, we will design the whole assembly of fuel injector. We will use top down assembly approach for assembling whole geometry. Figure.1 Shows total body of fuel injector assembly.

VARIABLE	DIEMSION(MM)
Convergent diameter	6
length	27
Divergent diameter	0.4
Outlet hole	2.4

1. GEOMETRY OF BODY:

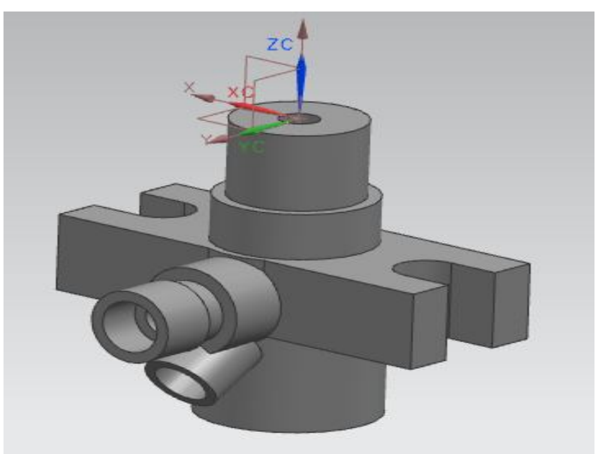

Figure.1.Total body used in fuel injector assembly

2. GEOMETRY OF DISTANCE PIECE:

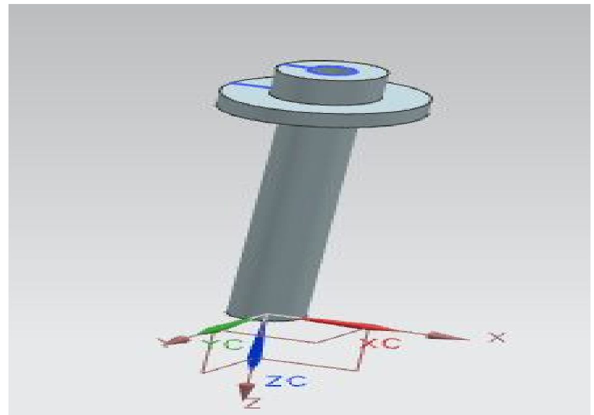

Figure.2.The piece used in fuel injector assembly

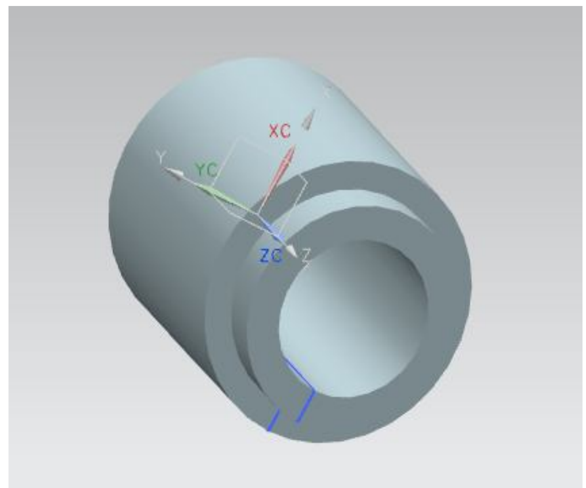

Figure.3.The piece used in fuel injector assembly

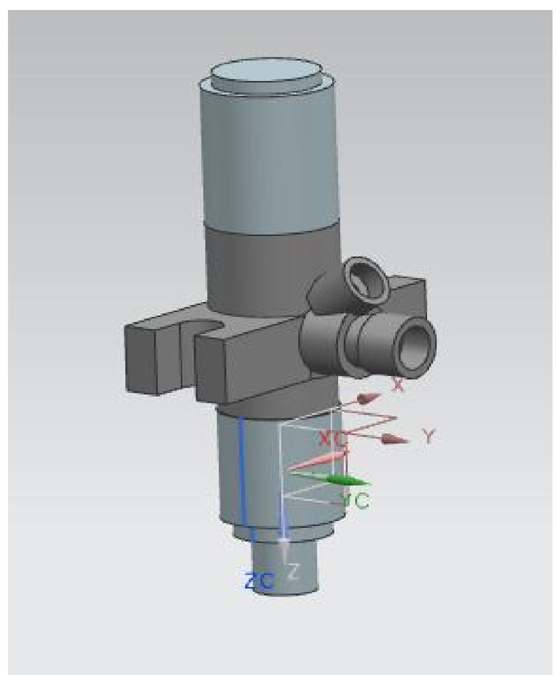

Figure.4 Shows the nozzle holder used in fuel injector assembly

IV. CFD ANALYSIS OF NOZZLE

Nowadays, Computational fluid dynamics (CFD) has become a progressively handy and robust tool for the numerical analysis involving transport processes.

CFD provides understanding into flow design, which is laborious, costly or impossible to investigate using conventional techniques. It consists of simulation of multiphase flow, heat transfer, chemical reactions and particulate processes.

In CFD simulations, accuracy and reliability are the main factors upon which emphasize is given. It is broadly accepted that simulations performed by CFD are very susceptible to the various computational parameters that have to be set by the user.

Consequently, these studies of CFD verification and validation are very crucial, also as well as higher comprehensive studies can deliver effective guidance in the selection of computational variables for future CFD studies. In this paper, CFD analysis has been carried

V. GOVERNING EQUATIONS

The governing equations used by the CFD software package for this study are as follows:

Conservation of Mass (Continuity Equation):

$$\frac{D\rho}{Dt} + \rho\nabla.V_x = 0 \qquad (1)$$

Where,

$\frac{D\rho}{Dt} = \frac{\partial\rho}{\partial t} + V_x\nabla\rho$ is the material derivative.

Conservation of Momentum (equation of motion):

$$\rho\frac{DV_x}{Dt} = -\nabla p - \nabla.T + p.f \qquad (2)$$

Conservation of Energy (Temperature equation):

$$\rho C_p\frac{DT}{Dt} = \frac{Dp}{dt} - \nabla.q - \varphi \qquad (3)$$

Initially the geometry created in Nx cad is imported, can be seen in Figure.5

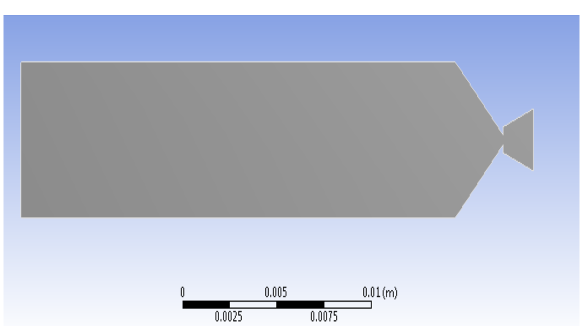

Figure.5. Geometry created in Nx cad

After importing the model must convert into finite elemental model for doing this process first we should create some named selections like fluid inlet, walls and fluid outlet. Next the mesh generation takes place.

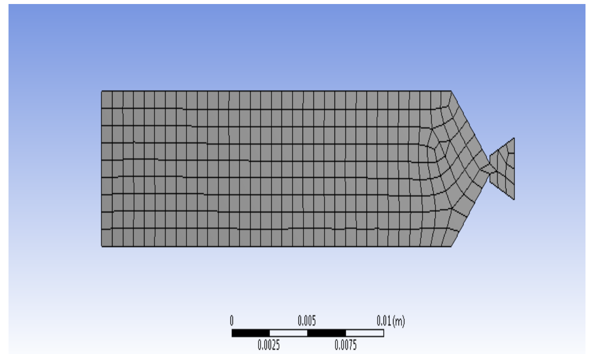

Figure.6 shows the finite elemental model of nozzle.

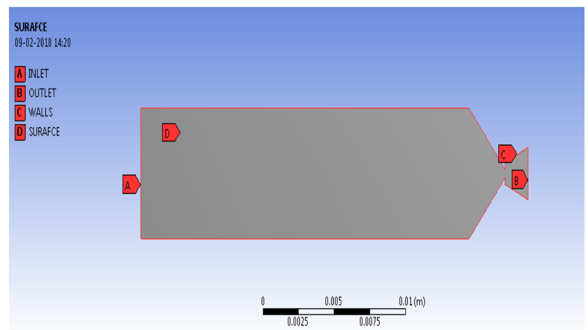

Figure.7 shows the surface model

The graph shows the convergence history for the nozzle with 6mm diameter with 200 m/s. The solution is converged at 68.

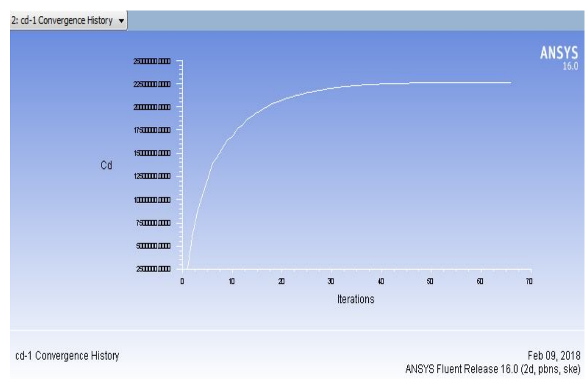

Figure.8 Graph showing the Convergence history of the Nozzle

Velocity outlet for the above convergence is 312.62 m/s.

The pressure outlet for the above convergence is 1.6950185e09 pa.

The mass flow rate for the above convergence is 0.071266174 Kg/s.

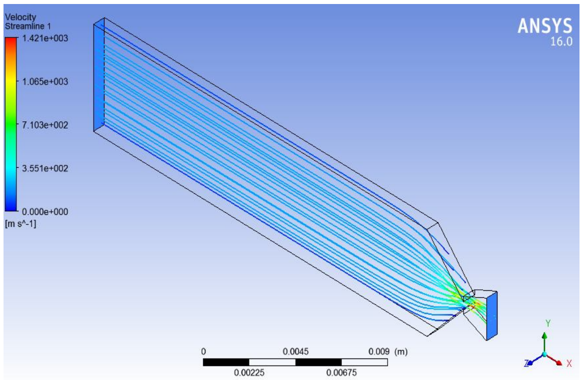

Figure.9 shows the velocity stream line for 200 m/s

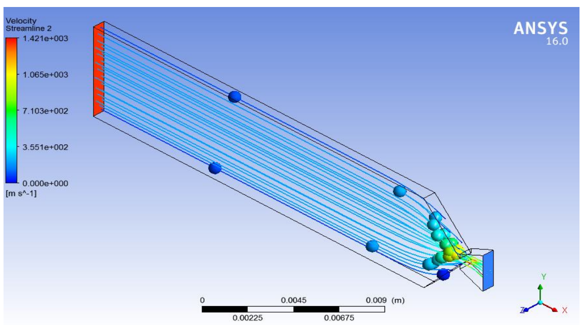

Figure.10 shows the pressure at 200 m/s

CFD Analysis of inside diameter with 6mm and velocity 300 m/s Inlet.

The graph shows the convergence history for the nozzle with 6mm diameter with 300 m/s velocity inlet. The solution is converged at 62.

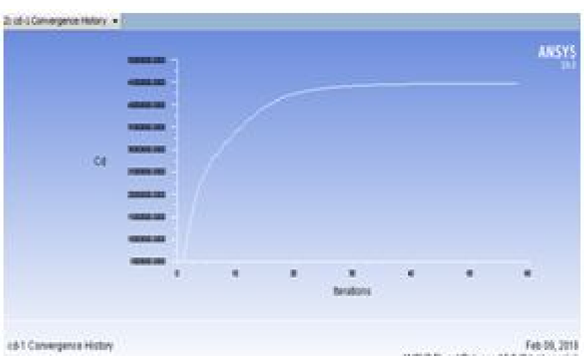

Figure.11.Graph showing the Convergence history of the Nozzle.

The velocity outlet for the above convergence is 435.92 m/s can see in figure 5.9.

The pressure outlet for the above convergence is 3.3234025e09 pa can see in fig 5.10.

The mass flow rate for the above convergence is 0.16108704 Kg/s.

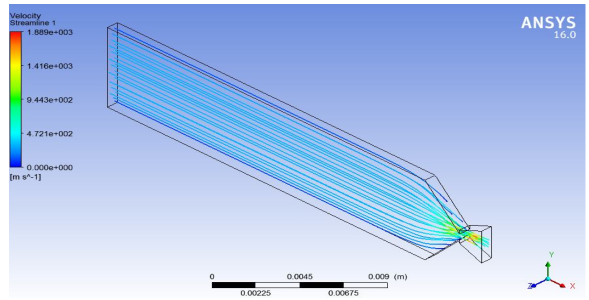

Figure.12 shows the velocity stream line for 300 m/s

The graph shows the convergence history for the nozzle with 4mm diameter with 400 m/s velocity inlet. The solution is converged at 101.

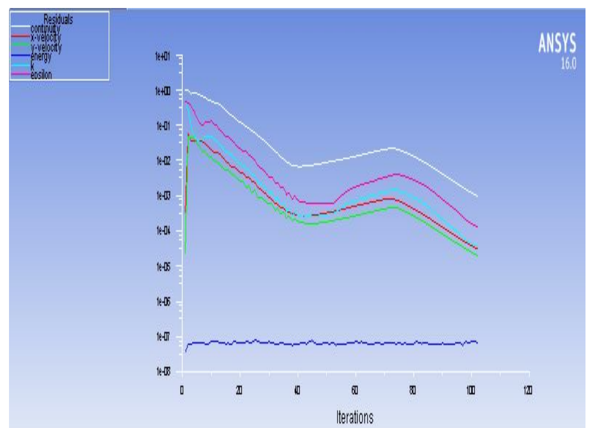

Figure.13 convergence history for 400m/s for 4mm inside diameter

V. RESULT

Nozzle Dia.	Inlet Velocity (m/s)	Pressure (pa)	Velocity (m/s)	Mass Flow Rate (kg/s)
6mm	200	1.70E+09	312.62	0.07126617
	300	3.32E+09	435.92	0.16108704
	400	5.91E+09	581.56	0.15797424
5mm	200	8.63E+09	267.37	0.07910919
	300	1.94E+09	400.32	0.11932373
	400	3.44E+09	534.22	0.15486145

VI. CONCLUSION

Nozzles used in diesel engines are manufactured in variety of shapes and sizes depending on the application where it is used. This is very important for understanding the performance characteristics of engine for nozzles. Convergent - divergent nozzle is the most commonly used nozzle in car engines since it used as a propellant which can be heated in combustion chamber. In this paper, the convergent divergent nozzle changing the different nozzle diameters and different fluids at different velocities is used for examining the fluid flow in nozzle. The convergent divergent nozzle having different nozzle diameters are designed by examining the results like pressure, velocity and mass flow rate values.

REFERENCES

[1] Vimal Kumar Pathak and Sumit Gupta, "Study of Nozzle Injector Performance Using Cfd", International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.4, No.3, August 2015.

[2] Amir Khalid, "Computational Fluid Dynamics Analysis of High Injection Pressure Blended Biodiesel", International Research and Innovation Summit (IRIS2017).

[3] Chirag J.Padhya, "CFD Analysis & optimization of fuel injector by changing its geometry", IJIRST –International Journal for Innovative Research in Science & Technology, Volume 1, Issue 7, December 2014.