IOSR Journal of Engineering (IOSRJEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
PP 15-20

Optimization of Fuel Nozzles for Maximum Thrust using Multi-Disciplinary Optimization

Gowtham Dronamraju[1], Prof. Daksheeswara Reddy Chamala[2]
[1]*PG Student, Mechanical Engineering, Mallareddy College of Engineering & Technology, India*
[2]*Asst. Professor, Mechanical Engineering, Mallareddy College of Engineering & Technology, India*

Abstract: *Nozzles of various sizes and geometries are being used in various fields of engineering. A nozzle is a mechanical device used to convert chemical, thermal energy into kinetic energy. It converts low velocity, high pressure & high temperature gas into high velocity at the exit. It is basically a device used to control rate of flow, speed, direction, mass, shape and/or pressure of the fluid across a system. In an automobile, a nozzle is used to spray fuel air mixture to the piston cylinder through carburetor. A convergent- divergent nozzle is a most commonly used nozzle in aerospace and automobile industries. In this project, diesel and gasoline engine nozzlesare modeled to optimize for maximum performance using multidisciplinary optimization technique. Static structural, thermal and internal flow analyses have been done on the nozzle by varying the fuel inlet diameters, materials and velocities to study the effect of mass flow and thermal distributions of the discharge for diesel and gasoline engine nozzles. The optimization also considered two different materials Brass and Aluminium alloy.*
Keywords –*Convergent-Divergent Nozzle, Multi-Disciplinary optimization, Structural analysis, Mass flow rate*

I. INTRODUCTION

A nozzle is a mechanical device of varying cross section which controls the direction and characteristics of the fluid (Air or Water) flowing through it. Nozzles are used in rocket engines to expand and accelerate the combustion gases, burning propellants, so that the exhaust gases exit the nozzle at supersonic or hypersonic velocities. Nozzles come in a variety of shapes and sizes depending on the flow requirements and characteristics. It is very important to understand the performance characteristics of nozzles based on the particular application. Convergent divergent nozzles are the most commonly used nozzles in the industry. These nozzles can get the output velocities of supersonic speeds. 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.A 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).

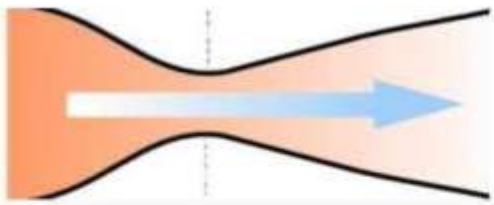

Fig.1 Convergent-Divergent nozzle

When the fluid flows through the nozzle, it exits at a higher velocity than its inlet velocity. This phenomenon occurs due to conservation of mass which states that the rate of change of mass equals to the product of density, area and velocity. m =P *A*V m = mass flow rate A = area of flow V = velocity of flow solving this equation using differentiation, we get the equation,

$$\frac{dA}{A} = \frac{dv}{v}(1 - M^2)$$

For subsonic flow M<1, increase in area causes velocity to decrease.
For supersonic flow M>1, increase in area causes velocity to increase
The Mach number M allows us to define flight regimes in which compressibility effects vary.

$$Mach\ number\ (M) = \frac{Velocity\ \ of\ the\ object}{velocity\ \ of\ sound}$$

There are two main approaches exist for the simulation of fluid– structure interaction problems:

1) Monolithic approach: In monolithic approach, the problem consists of the equations governing the flow and the displacement of the structure are solved simultaneously with a single solver. But, this approach becomes complicated but requires less time.

2) Partitioned approach: the equations governing the flow and the displacement of the structure are solved separately, with two distinct solvers. We use partitioned approach where fluid dynamic equations are solved using separate solvers i.e. Thermal and structural equations are solved using Finite element method in ANSYS. For industrial designing purpose this methodology is more reliable than the monolithic approach as design criteria for flow design and structural design is satisfied individually but it takes more time.

In this project nozzle is been modeled in 3D modeling software CREO. Structural analysis is done on the nozzles by varying the fuel nozzle diameter and the velocities of the flow. The analysis was done for nozzle diameters 30mm, 40mm and 50 mm and the inlet velocities are varied from 200m/s to 400m/s and also analysis is done using different nozzle materials using Brass and Aluminium alloy.

II. LITERATURE SURVEY

Lei Liu et al [4] performed analysis and design of system for detecting fuel economy of a diesel engine nozzle. A comprehensive comparison of several traditional fuel economy detection systems has been made using ultrasonic flow detection which can work real time and detect automatically. Rajendran et al [5] had investigated flow structure of inside the cylinder with three dimensional Modeling of in cylinder flow. The results of the CFD simulation has been used to understand the effect of the different obstacles in the flow on the mass flow rate and the static pressure at the tip of the fuel tube. This would be helpful to analyze the pressure loss in the inlet, throat and outlet area. The results of this study is used for modification of throttle valve design. The carburetor body is remodeled with two throttle bodies replacing conventional throttle. Analysis has been performed to study flow field with modified design and results have been discussed. Zing Zhang et al [6] discussed on the spray characteristics as an important factor which affects the mixture formation and combustion process of the engine. Further, the team analyzed that the effect of nozzle-hole cone angle of the fuel injector directly influence the spray characteristics. In this paper, flow evolution process, spray distribution, fuel-air equivalence ratio distribution, temperature distribution and emissions formation have been discussed. The results show that the mixture formation quality can be improved by selecting the nozzle hole cone angle appropriately, and therefore combustion process be improved and emissions be reduced. Chirag et al [7] analyzed using the computational fluid dynamics programs CFX and obtained the pressure and velocity profiles for pre-selected locations in the fluid path of the injector. Locations of high and low velocity areas were determined to be performance inhibitors and possible failure points of a fuel injector geometry.

The above research papers did not analyze considering the effect of angle of fuel injectors vs the flow characteristics and also the change in material of the nozzle on the overall thermal distribution of the nozzle. Hence, this paper considered the opportunity to explore and suggest the optimal positions for fuel injectors in the gasoline engine nozzles and also the effect of materials (Brass (ASTM B21) and Aluminum alloy (A6061)

III. PROCEDURE

Tools Used

PTC CREOis a 3D modeling software used in various fields of engineering helpful in conceptual design and development of prototypes. It uses parametric modeling technique and is very effective in frequent design iterations and is compatible with all analysis soft wares.

ANSYS is a finite element simulation software to simulate engineering problems of various fields and provides solution using it in-built solvers. In this project, Static and CFD analysis have been done using Ansys simulation tool.FEA uses discrete elements to process the mechanical engineering problem. It can be applied to 1D, 2D and 3D problems. The model is split up into a number of smaller areas or volumes, which are called finite elements. Some of the modules are ANSYS Fluent, CFD, CFX, FENSAP-ICE and related software are Computational Fluid Dynamics software tools used by engineers for design and analysis.

Modelling Of Nozzle With Varying Diameter

Step: 1

DESIGN

The engine fuel nozzle has been designed in PTC Creo with varying nozzle diameters 50 mm, 40 mm and 30mm.

Fig.2 Nozzle with 50MM, 40 MM & 30 MM DIA

Static Analysis Of Diesel Engine Nozzle
Step: 2 **Meshing And Boundary Conditions**
Each nozzle has been meshed into a fine mesh as mentioned in the below figures and the boundary conditions of flow have been defined as A as inlet and B as outlet or exit

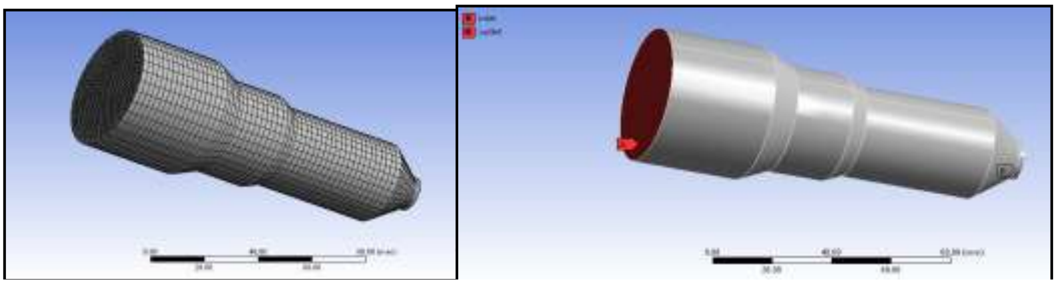

Fig.3 Meshing and defining boundaries

Step: 3 **SOLUTION:**
Now, considering each nozzle size, the inlet velocity has been varied from 200 m/s, 300 m/s and 400 m/s to plot static pressure, velocity and heat transfer coefficient

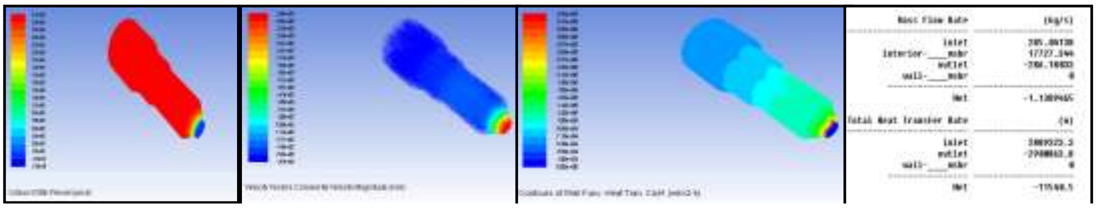

Fig.4 Static Pressure, Velocity, Mass flow rate and HT coefficient rate plots for Nozzle Dia. 50 mm& Velocity at Inlet 200 m/s

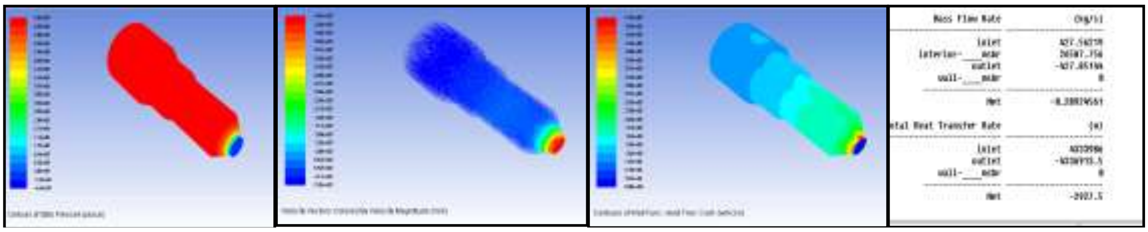

Fig.5 Static Pressure, Velocity, Mass flow rate and HT coefficient rate plots for Nozzle Dia. 50 mm & Velocity at Inlet 300 m/s

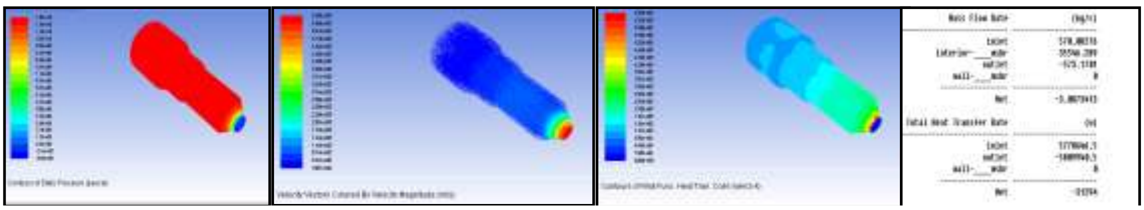

Fig.6 Static Pressure, Velocity, Mass flow rate and HT coefficient rate plots for Nozzle Dia. 50 mm & Velocity at Inlet 400 m/s

Similarly, the same plot have been plotted for nozzle diameters 40 mm, 30 mm and 20 mm to understand the static pressure, velocity, heat transfer coefficient and Mass flow rate and heat transfer rates.

Thermal Analysis Of Diesel Engine Nozzle
Step: 4
Now, the temperature and heat flux distribution has been plotted using thermal analysis for Cu alloy (Brass) and Al alloys

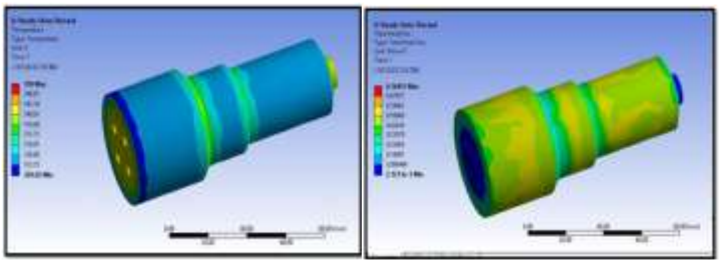

Fig.7 Temperature and heat flux plots for Brass metal

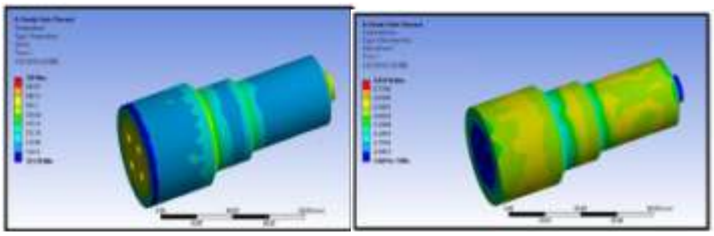

Fig.8 Temperature and heat flux plots for Al alloys

FATIGUE ANALYSIS OF DIESEL ENGINE NOZZLE

Step: 5The nozzle has been analyzed with lifing, Damage tolerance and understand safety factor

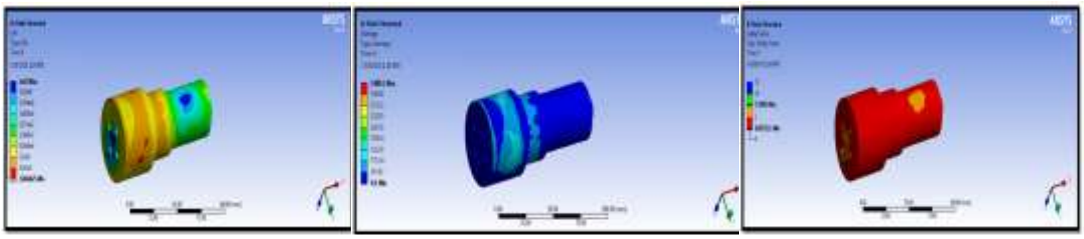

Fig.9 Lifing, damage and safety factor for the nozzle

Cfdanalysis Of Engine Nozzle
Step: 6 Boundary conditions for CFD analysis of Convergent- Divergent Nozzle for gasoline engine

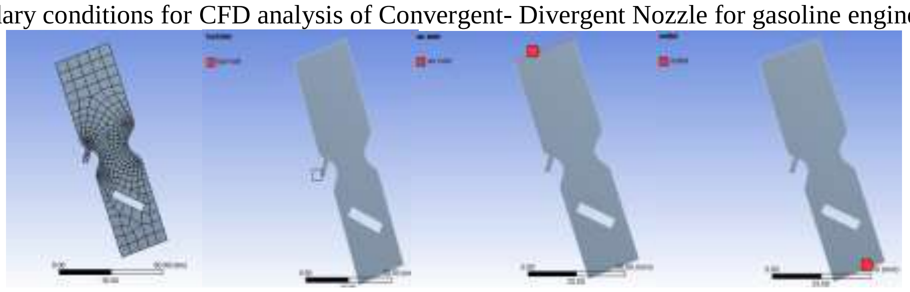

Fig.10 CFD modeling, Defining Boundary Conditions such as air & fuel inlet & exit

Step: 7 CFD analysis of Convergent- Divergent Nozzle –results
K-Epsilon standard model is considered with standard wall functions and Prandtl number is 1.

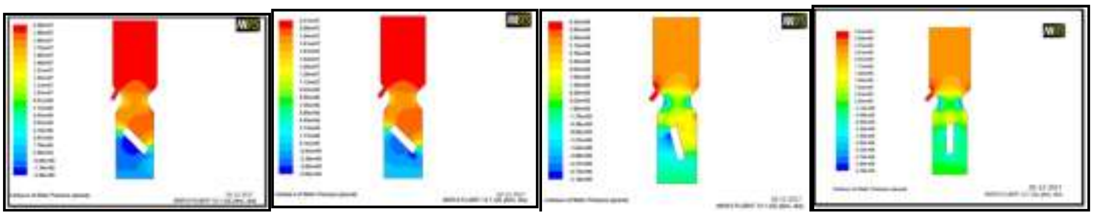

Fig.11 Static pressure at throttle angle 45, 60, 75 and 90 degrees

IV. TABLES AND RESULTS

All the results have been plotted for Static, Thermal and Lifing analysis have been recorded in the table 1 & 2 respectively for nozzle diameters 50 mm, 40 mm and 30 mm and materials for figures 3 to 9

Nozzledia. (mm)	Inlet velocity (m/s)	Pressure (Pa)	Velocity (m/s)	Heat transfer coefficient (w/m2-k)	Mass flow rate (kg/s)	Heat transfer (W)
50	200	3.12e+09	2.98e+03	3.59e+05	1.138945	11540.5
	300	6.96e+09	4.46e+03	5.10e+05	0.289245	2927.5
	400	1.25e+10	5.99e+03	6.56e+06	3.087343	31294
40	200	4.53e+09	3.58e+03	3.76e+05	1.0457764	10600
	300	1.03e+10	5.38e+03	5.30e+05	2.192199	22219
	400	1.83e+10	7.17e+03	6.80e+05	2.9847107	30249
30	200	1.04e+10	5.36e+03	6.90e+05	0.16120148	1634.3125
	300	2.34e+10	8.05e+03	8.05e+03	0.44642	4520.625
	400	4.18e+10	1.07e+04	1.25e+06	0.8333587	8450.75

Table 1. Velocity, Static Pressure, Heat transfer Coefficient, Massflow rate and HT rate for various diameters of nozzles

Material	Temperature (K)		Heat flux(W/mm2)
	Min	Max	
Brass(ASTM B21) Cu 60.0, Zn 39.25, Sn 0.75	320.02	350	0.76451
Aluminum Alloy (A6061)	323.59	350	0.87036

Table 2. Temperature and Heat flux values for Brass and Aluminium materials

For CFD analysis of C-D nozzleof a gasoline engine, the throttle plate and fuel inlet angles have been modified to understand the flow characteristic and got the below results for the plots shown in figures 10 and 11.

Fuel discharge angle (Degrees)	30	35	40	45
Static Pressure (Pa)	$1e^6$	$6.75\ e^5$	$7.4\ e^5$	$8.79\ e^5$
Velocity (m/s)	$6.41\ e^1$	$6.21\ e^1$	$6.33\ e^1$	$6.37\ e^1$

Table 3. Static Pressure and Exit velocity for C-D nozzle with varying fuel nozzle discharge angle

Throttle Plate Angle (deg.)	45	60	75	90
Static Pressure (Pa)	2.087	2.21 e7	4.22 e6	3.43 e6

Table 4. Static Pressure for C-D nozzle with varying throttle plate angle

V. CONCLUSION

From the above analysis, the following conclusions can be derived.

Varying the nozzles to a variety of shapes and sizes depending on the type of engine, the performance characteristics of nozzle have been analyzed for optimal performance. In this paper, the characteristics of the nozzle are analyzed by changing different nozzle diameters and fluids at different velocities. Nozzle were modeled changing nozzle diameters.

By observing the static, thermal and fatigue analysis of diesel engine nozzle the pressure, velocity, heat transfer rate and mass flow rate values are increases by increasing the inlet velocities and decreasing the nozzle dia. So it can be concluded the diesel engine nozzle efficiency were more when the nozzle dia. decreases. The highest mass flow rate (2.98 kg/s) was fount when the nozzle diameter is 40 mm and inlet velocity is 400 m/s. By observing the thermal analysis, heat flux is more for aluminum alloy compared with brass material.

For gasoline engine carburetor nozzle, the max static pressure occurred at 45 degree angle of fuel nozzle angle at the throat of C-D nozzle, whereas for maximum velocity of the fuel occurred at 30 degree angle.

ACKNOWLEDGEMENTS

I sincerely thank Prof. Ch. Daksheeswara Reddy &Dr. PHV Sesha Talpa Sai, Head of Department, Mechanical Engineering Department, Mallareddy College of Engineering & Technology, Hyderabad for their valuable guidance in compiling this paper.

REFERENCES

[1]. OB Zimmerman, “IC Engines and their tractors”, Forgotten Books Publishing, pp 493-494, 1988.
[2]. Robert Bosch, “Distributor Type Diesel Fuel Injection Pumps: Bosch Technical Instruction”, Bosch GmbH, p23-71, 2003.
[3]. Challen, “Diesel Engine Reference Book” Reed Educational and Professional Publishing Ltd., Second Edition, pp.260-301, 1999.
[4]. Lei Liu et al, “Analysis and Design of the System for Detecting Fuel Economy"IEEE. 2010 International Symposium on Intelligence Information Processing and Trusted Computing, 28-29 Oct. 2010.
[5]. Rajendran, S & Purushothaman, K. (2014). Analysis of fuel flow in internal combustion engine using -CFD. International Review of Mechanical Engineering. 8. p480-487.
[6]. Jing, Zhang et al, Numerical simulation on effects of nozzle hole cone angle on combustion and emissions in a diesel engine, 2011 Conference on Electric Information and Control Engineering, 15-17 April 2011.
[7]. Chirag J. P1adhya, Ashish R. Patel, “CFD Analysis & Optimization of Fuel Injector by changing it’s Geometry -A Review” – International Journal for Innovative Research in Science & Technology| Volume 1 | Issue 7 | December 2014 ISSN (online): 2349-6010