International Journal For Advanced Research In Science & Technology
A peer reviewed international journal www.ijarst.in
ISSN: 2457-0362
IJARST

CFD Analysis of Diesel Engine Nozzle at Different Velocities and Divergent Angles

MR.MD MAZHAR NEHAL[1], Mr. Y.SURESH KUMAR [2],

[1]M.TECH, STUDENT, DEPARTMENT OF MECHANICAL ENGINEERING, NOVA COLLEGE OF ENGINEERING AND TECHNOLOGY, JANGAREDDY GUDEM, PIN: 534 447, WEST GODAVARI DIST. ANDHRA PRADESH.

[2]HOD&ASSISTANT PROFESSOR, DEPARTMENT OF MECHANICAL ENGINEERING, NOVA COLLEGE OF ENGINEERING AND TECHNOLOGY, JANGAREDDY GUDEM, PIN: 534 447, WEST GODAVARI DIST. ANDHRA PRADESH.

Abstract

Nozzle is a mechanical device used to convert chemical thermal energy generated in the combustion chamber into kinetic energy by controlling the direction or characteristics of the fluid flow ie; the low velocity, high pressure, high temperature into high velocity gas of lower pressure and temperature. There are two main type of injector nozzles, Hole and Pintle. Hole type nozzles are commonly used in direct injection engines. They can be single-hole or multi-hole and they operate at very high pressures up to 200 atm. They give hard spray. Which is necessary to penetrate the highly compressed air.

In this project, we are modelling (BOSCH DLLA 146P2563) convergent divergent nozzle changing with different nozzle angles and analyzing the convergent divergent nozzle with different velocities to determine the pressure drop, heat transfer coefficient, velocity and mass flow rate for the fluid by using CFD technique. in this process to check object strength here safety factor value calculated with ansys workbench structural analysis process, from this analysis it is easy to understand how much pressure object can withstand. If object has minimum safety factor above 1.5 then object is safe at applied boundary conditions.

Based on all these results finally, thesis conclude with optimum results with required tables and graphs.

Keywords: ANSYS, CFD technique, CATIA software, deformation, stress, strain, safety factor, Convergent-Divergent Nozzle, Diesel engine, Fuel injector, Flowrates.

CHAPTER 1 INTRODUCTION

1.1 DIESEL ENGINE

The diesel engine, named after a person Rudolf Diesel, is an internal combustion engine in which ignition of the fuel is caused by the elevated temperature of the air in the cylinder due to mechanical compression.

Diesel engines work by compressing only air, or air plus residual combustion gases from the exhaust. Air is inducted into the chamber during the intake stroke, and compressed during the compression stroke. This increases the air temperature inside the cylinder to such a high degree that atomised diesel fuel injected into the combustion chamber ignites.

The diesel engine has the highest thermal efficiency of any practical internal or external combustion engine due to its very high expansion ratio. Diesel engines may be designed as either two-stroke or four-stroke cycles. [1]

They were originally used as a more efficient replacement for stationary steam engines. Since the 1910s, they have been used in submarines and ships. Use in locomotives, buses, trucks, heavy equipment, agricultural equipment and electricity generation plants followed later. In the 1930s, they slowly began to be used in a few automobiles.

1.1 PROBLEM STATEMENT

EXISTING

As per the base paper referred, the flowrates considered to perform CFD analysis are as follows:

At the inlet velocity of 15.5 m/sec, the maximum velocity and stagnation pressure developed at the outlet are 97.027 m/sec and 35155 N/m^2 respectively. At the inlet velocity of 50m/sec, the maximum velocity and stagnation pressure developed are 52.007 m/sec and $0.165e^{17}$.

Lastly, the nozzle with inlet velocity 15.5 m/sec is selected as optimal one.

PROPOSED

For the nozzle BOSCH DLLA 146P2563, we are going to take the inlet velocities are 15.5 m/sec, 25.5 m/sec and 35.5 m/sec with divergent angles 135^0, 1200, 105^0.

The optimal nozzle will be selected based upon the analysis in CFD.

1.2 OBJECTIVES:

Selecting a design parameter for convergent divergent nozzle to vary the geometry and boundary conditions of model. Thus, to increase the velocity by decreasing pressure.

To optimize the model by making use of CFD technique in ANSYS

LIETRATURE REVIEW

In 2016, Fuying Xue, Fuqiang Luo, Huifeng Cui, Adams Moro, Liying Zhou [6]

- In this paper, mathematical theory and model associated with the cavitation and turbulence is presented.
- The turbulence flow pattern and cavitation evolution for nozzle holes of an asymmetric multi-hole diesel injector were replicated using the multi-phase two fluid flow approach where the effect of injection conditions on bubble number density was considered.
- The fuel flow characteristics in each nozzle hole were simulated and the effect of cavitation and turbulence were analyzed.

- It was observed that the nozzle angles and needle lifting process influenced the cavitation, mass flow rate and flow velocity. In 2016, F.J. Salvador, J. De la Morena. Martínez-López. Jaramillo [7]

- In this paper, an investigation of the compressibility effects in nozzle flow simulations has been carried out for injection pressures up to 250 MPa.

- To do so, the fluid properties (including density, viscosity and speed of sound) have been measured in a wide range of boundary conditions.
- These measurements have allowed to obtain correlations for the fluid properties as a function of pressure and temperature.

- Then, these equations have been incorporated to a CFD solver to take into account the variation of the fluid properties with the pressure changes along the computational domain.

- The results from these simulations have been compared to experimental mass flow rate and momentum flux results, showing a significant increase in accuracy with respect to an incompressible flow solution.

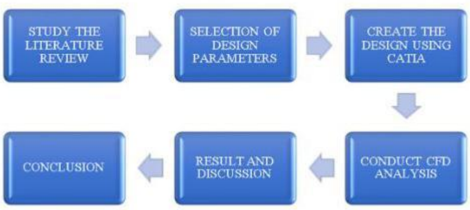
- In 2016, Zhixia He, Zhengyang Zhang, Genmiao Guo, Qian Wang, Xianying Leng, Shenxin Sun [8]

- This paper, particular attention was focused on the transient flow characteristics in the real-size diesel nozzle.

- An experimental study under different pressures was conducted to analyze the evolution of cavitation inside diesel nozzle, and it was found that higher injection pressure leads to earlier cavitation inception.

- The bubble "suction" from orifice exit at the end of injection and the bubble "discharge" at the initial stage of the next injection were observed as well.
- Moreover, two types of "string cavitation" were observed and the "string cavitation" as a special cavitations phenomenon which considerably boosts the spray angle was investigated in details.
- It was found that the occurrence of "string cavitation" has a strong relationship with the location of needle, the injection pressure, and the shape of sac. Furthermore, the effects of these three factors on the occurrence regularity of the "string cavitation" were also investigated.

3.1 METHODOLOGY



In all of these approaches, the same basic procedure is followed.
During pre-processing The geometry (physical bounds) of the problem is defined.

The volume occupied by the fluid is divided into discrete cells (the mesh). Themesh may be uniform or non-uniform.

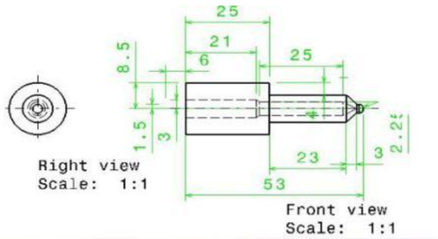
The physical modelling is defined – for example, the equations of motion + enthalpy + radiation + species conservation

Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

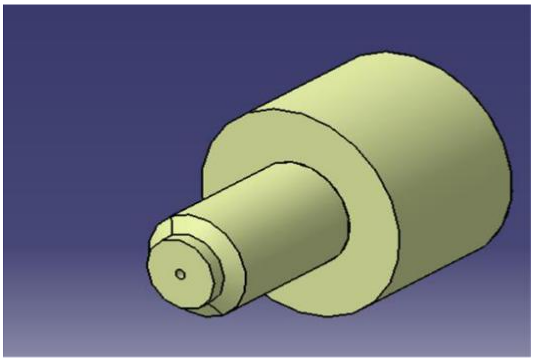
The simulation is started and the equations are solved iteratively as a steady-state or transient.

Finally, a postprocessor is used for the analysis and visualization of the resulting solution.

3.2.1 2D DRAWING OF NOZZLE



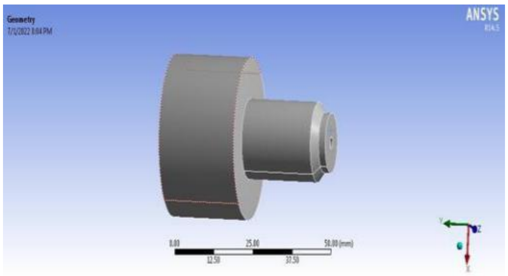
3D MODEL OF NOZZLE

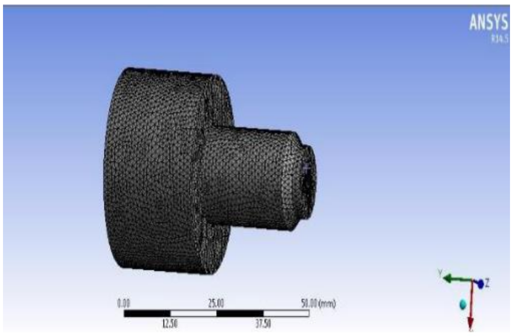


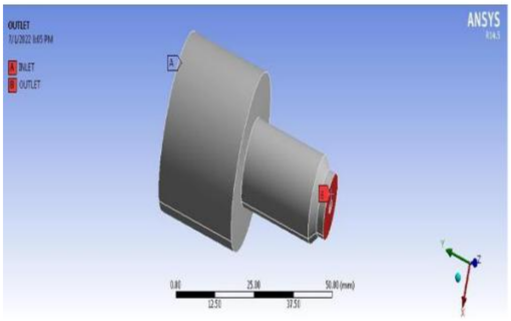
3.2.1 CFD ANALYSIS OF DIESEL ENGINE NOZZLE

FLUID - DIESEL

- **At angle-45^{0}**
- **Velocity inlet = 15.5m/s**







Update project>setup>edit>model>select>energy equation (on)>ok

Materials> Materials > new >create or edit >specify fluid material or specify properties > ok

Select fluid

Boundary conditions>inlet>enter required inlet values

Solution > Solution Initialization > Hybrid Initialization >done

Run calculations > no of iterations = 10> calculate > calculation complete>ok

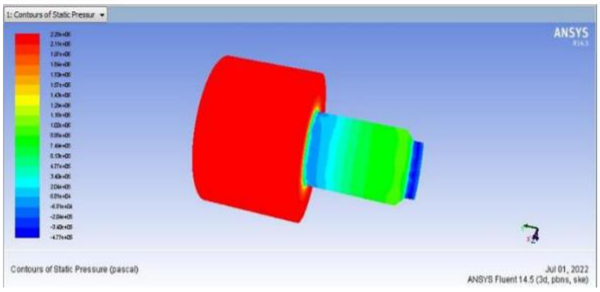


Fig 3.7 - Pressure Plot

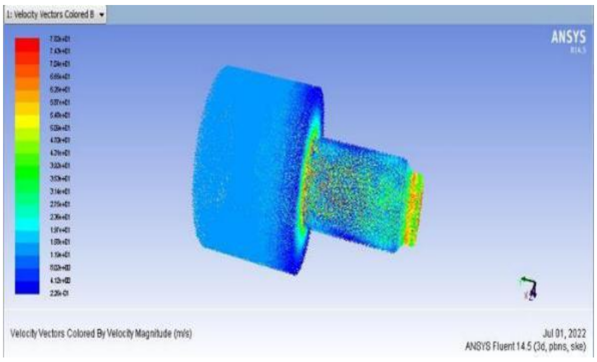


Fig 3.8 – Velocity

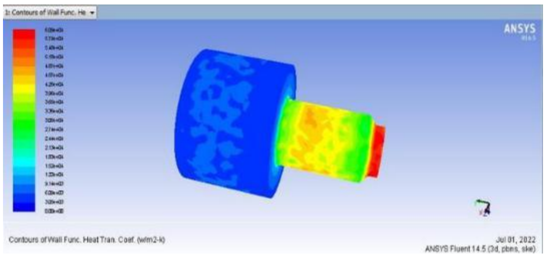


Fig 3.9 - Heat Transfer Coefficient

```
Mass Flow Rate                      (kg/s)
-------------------- --------------------
               inlet             5.1545849
 interior-partbody              -2.7459452
              outlet            -5.1530056
       wall-partbody                     0
 ------------------- --------------------
                 Net         0.0015792847
```

Fig 3.10 - Mass Flow Rate

CHAPTER 4

RESULTS

Angle	Inlet velocity(m/s)	Pressure (pa)	Velocity (m/s)	Heat transfer coefficient	Mass flow rate (kg/sec)
45	15.5	2.25e+06	7.82e+01	6.09e+04	0.00157
	25.5	5.37e+06	1.37e+02	8.32e+04	0.00115
	35.5	1.24e+07	1.92e+02	1.00e+04	0.0704
60	15.5	3.34+06	1.05e+02	4.9e+04	0.012572
	25.5	8.77+06	1.72e+02	7.30e+04	0.0281
	35.5	1.63e+07	2.42e+02	9.75e+04	0.040214
75	15.5	4.19e+06	1.24e+02	6.51e+04	0.01485
	25.5	1.02e+07	2.00e+02	9.37e+04	0.006624
	35.5	2.26e+07	2.77e+02	1.86e+05	0.04163

CONCLUSION

Nozzles come in a variety of shapes and sizes depending on the mission of engine, it is very important to understand the performance characteristics of diesel engine. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber. In this project the convergent divergent nozzle changing with different nozzle angles at different velocities.

We modelled convergent divergent nozzle changing with different nozzle diameters. By observing the CFD analysis of diesel engine nozzle the pressure, velocity, heat transfer rate and mass flow rate values are increases by increasing the inlet velocities and increasing the nozzle angle. To get clear idea about design, each design undergoes structural analysis process, from this process safety factor has been calculated, all the values are above 1.5, so design is safe in all applied boundary conditions, and this pressure don’t damage the object, and it is under control.

So, it can be concluded the diesel engine nozzle efficiency were more when the nozzle angle 75 deg.

REFERENCES

[1] Thermal engineering by RK Rajput

[2] https://en.wikipedia.org/wiki/Diesel_engine#:~:text=The%20diesel%20engine% 2C%20named%20after, ignition%20engi ne%20(CI%20engine).

[3] https://en.wikipedia.org/wiki/Boundary_ element_ method [4] https://in.search.yahoo.com/search;_ylt=AwrPrnYExr9iwEAE3wy9HAx.?p=C ATIA + software + wiki & type=E210IN826

[4]G0&fr=mcafee&fr2=p%3As%2Cv%3%2Cm%3Apivot&stype=web

[5] M. Volmajer, B. Kegl, "Experimental and numerical analysis of fuel flow in the diesel engine injection nozzle", Journal of Kones. Combustion Engines, Vol. 8, No. 1-2, 2001.

[6] Numerical analyses of transient flow characteristics within each nozzle hole of an asymmetric diesel injector. In 2016, Fuying Xue, Fuqiang Luo, Huifeng Cui, Adams Moro, Liying Zhou

[7] Comparative study of the internal flow in diesel injection nozzles at cavitating conditions at different needle lifts with steady and transient simulations approaches. In 2016, F.J. Salvador, J. De la Morena. Martínez-López. Jaramillo

[8] Visual experiment of transient cavitating flow characteristics in the real size diesel injector nozzle. In 2016, Zhixia He, Zhengyang Zhang, Genmiao Guo, Qian Wang, Xianying Leng, Shenxin Sun.

[9] Sanghoon Lee, Sungwook Park, "Spray atomization characteristics of a GDI injector equipped with a group-hole nozzle", Fuel 137 (2014) 50–59