

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

## CFD Simulation of Injection Mixer for CNG Engines

GINSE JOY<sup>1</sup>, DR. S.SANKAR<sup>2</sup>, RIXON K.L.<sup>3</sup>, SREEJITH K.V.<sup>4</sup>

M.Tech Scholar, Dept. of ME, NCERC, Pampady, Kerala, India<sup>1</sup>

H.O.D, Dept. of ME, NCERC, Pampady, Kerala, India<sup>2</sup>

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India<sup>3</sup>

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India<sup>4</sup>

**ABSTRACT:** The inadequate mixing of air fuel mixture in engines is one of the main reasons for insufficient engine performance and emission of harmful gases. In this project CFD analysis of injection manifold of the CNG engine is carried out using the RNG K-ε turbulent model. CFD results obtained from the existing model are compared with the modified model. The engine specifications are 125cc CNG engine with single nozzle fuel atomizer. In the modified model we are using cd atomizer as well as 3 nozzle atomizer for analysis. In this analysis the pressure, temperature and velocity contours inside the cylinder are compared for existing and modified model. We are carrying out 2d analysis.

**KEYWORDS:** ANSYS CFD, Fuel Injector.

### I. INTRODUCTION

With increase in awareness among the researches about the harmful effects of use of fossil fuels and their depletion at a higher rate due to excessive usage, the need to find alternative fuels has become the need of the day. There are large numbers of alternatives available but the selection of best out of the lot is also necessary. The use of compressed natural gas is continuously increasing due to some of its advantageous properties. As per research about 1.1 million CNG vehicles have been reported in India. CNG is actually methane gas stored at high pressure. The emissions from CNG engines or the compression products are carbon dioxide and water. Methane is the cleanest burning hydrocarbon as it is completely freed from contaminants during its processing. Due to its effective mixing it is highly recommended fuel. It can be used in place of petrol, diesel and LPG. As it is lighter than air it is safer than other fuels in the event of spill. CNG may be found above oil deposits, or may be collected from landfills or wastewater treatment plants where it is known as biogas.

CNG is used in traditional gasoline/internal combustion engine automobiles that have been modified or in vehicles which were manufactured for CNG use, either alone ('dedicated'), with a segregated gasoline system to extend range or in conjunction with another fuel such as diesel. In response to high fuel prices and environmental concerns, CNG is starting to be used also in tu-tuts and pickup trucks, transit and school buses, and trains.

### II. RELATED WORK

Nimit M. Patel [1] he has studied and replicated injection nozzle at different locations using Computational Fluid Dynamics software to get the best mixing in the combustion chamber of the engine. The other modification is done in the injection nozzle is the velocities of the fuel in combustion chamber. That also determined from the software. The length of bore and stroke of the engine is selected from the actual diesel engine and it has been modelled using Gambit software. From the various results the air and fuel mixing is very good and better when compared with the original injection nozzle. The velocity of fuel in combustion chamber gets from the simulation is highest for the new design. During the study for optimization of gas injector orientation there is relation is developed for better thermal efficiency.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

From the observation of his studies the gas injector at 225° position with reference to axial axis of intake manifold gives the most favourable angle based on better mixture. The study is focused at a constant speed (1800rpm) diesel engine under CNG-Diesel mode.

Hussein A. Mahmood[2] he has designed, manufactured and tested a new design for CNG mixture. And he has simulated the new design in the CFD simulation software. And for getting the optimization for the homogeneity in the new design he modelled 14 different models to investigate the different parameters like diameter, location and number of hole inside the mixer. From the simulation results model 7 gives the uniformity index (UI) values, where the UI values are 0.939 and 0.937. The efficiency for an air fuel ratio for a blend of hydrogen and compressed natural gas (AFRHCNG) = 51.31 and the air fuel ratio for compressed natural gas (AFRCNG) = 34.15 is get from the simulation results. And he has concluded in the paper as the efficient mixing of air and gaseous fuel get by prolonging the mixing time. However, the prolongation of mixing time is related directly to the nature of the flow. For non-turbulent or no vortex prolonging the mixing period does not affect the nature of the mixing. The gaseous fuel diffusion inside the mixer does not affect. Increasing the number of holes inside the mixer when the direction of holes is opposite that of the airflow.

Patel Nimit M[3] his investigation focused on engine performance based on variations in location of injector, number of holes in injectors and pressure. The results showed that the conversion of the diesel engine to a CNG engine reduced engine performance. For increase the power and decrease the exhaust gas emissions, the CNG engine is needs to improve. By converting of diesel engine to port injection CNG engine the spark ignition and improve of the gaseous fuel injector nozzle will give better performance and exhaust gas by variation hole geometry and various injection pressure. And also by these variations there will be reduction of exhaust emissions.

M. A. Shaikh[4] in this paper he has studied the different parameters that affects the performance of CNG engines. From his studies major problem of using of CNG engines in the road is the reduced power and torque. And this deficiency can avoid by swirl motions and these parameters are more emphasized in this study to enhance the performance of engine and so to negotiate mentioned difficulty. For the study of swirl motion the component for mix the fuel and air must be considered with various factors. The studies were conducted in Computational Fluid Dynamics (CFD). The study mainly focused on improving the optimum design by changing the swirl motion. And in the results varying the swirl intensity will give the enhancement in engine performance. This study is thus helpful to design Mixer and shows path if possibility of further modification in design exists. Concluded for mixer are throat diameters, number of holes at throat section and their diameter, inlet-outlet angle etc.

James Sevik[5] A his studies are focused on the design and execution of CNG mixer and to investigate the injection at different part loads. A Steady-state test was performed in the CFD for a single-cylinder with current gasoline Direct Injection engines. The tests are conducted with different orientation of injector. And also exhaust gas is recirculate for improving the emission control and its dilution tolerance is investigated in this paper. From the experimental results the variation of the start of injection give strong impact on the mixture formation process. Delayed injection timing increases the turbulence level in the cylinder but reduces the available time for proper mixing. Side injection provides more effective induced flow field and this improves the performance of the engine. The results of the study have shown

NG DI to be a viable option for a purpose built NG engine, especially in a part-load condition.

### III. METHODOLOGY

#### PROBLEM STATEMENT

In this project CFD analysis of injection manifold of the CNG engine is carried out using the RNG K-ε turbulent model. CFD results obtained from the existing model are compared with the modified model. The engine specifications are 125cc CNG engine with single nozzle fuel atomizer. In the modified model we are using cd atomizer as well as 3 nozzle atomizer for analysis. In this analysis the pressure, temperature and velocity contours inside the cylinder are compared for existing and modified model. We are carrying out 2d analysis.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

## SOLID MODELING

As we discussed earlier for injection mixer analysis we require a scaled prototype of the cart to carry out simulation in ANSYS. For analysis in CFD we require a 2D model which can be modelled using 2D modelling software like Creo, CATIA V 5, Solid works etc. any of these software can be used depending on one's skill. For this project work ANSYS Design modeller has been used to develop a 2D-Model of the engine cylinder. Various modelling tools are Line, Fillet, Curve, Trim etc. A engine cylinder 2D model was created using ANSYS Design Modeller.

## MESHING

The process of discretization model into number of elements is known as meshing. It is the most important step in a FEA analysis. The quality of mesh defines the accuracy of the result. We must require refining the meshing to get crisp and clear elements with uniformity. The meshing is done in ANSYS Mesh Generator. Before creating the mesh a fluid enclosure is created using CREATE > Enclosure Tool. Once we get into Mesh Generator we define Relevance, Sizing, Inflation, Smoothing, Refinement.

## CFD SOLVER (SET UP)

This is the main part of CFD where our model solved and analysed according to the given boundary. Special care should be taken while defining the steps in the solver. The set up of the solver varies with the type of analysis we are performing. It takes up the most part of time of CFD analysis. It is here where the size of generated mesh comes into play. Finer the mesh more the time will it consume to solve our problem because the equations are solved for each and every element, smaller the element size more will be the number of elements for a given model. The solver setup includes following steps :-

1. Type of analysis: For our injection mixer analysis we carrying out 2D model analysis which pressure based with absolute velocity formulation. We are doing steady state analysis
2. Models: here the type of numerical model used for the analysis is defined. For injection mixer analysis Viscous based numerical model. The type of viscous based model used is realizable k-epsilon and scalable wall function model
3. Materials: here the material for the model is defined. In this work the analysis is carried out on methyl - alcohol liquid which is the surrounding fluid. Here properties of the material can be retrieved from ANSYS Library.
4. Boundary conditions: this is the core part of ANSYS Solver here the initial boundary conditions are defined. Without defining these variables we cannot initialize the solution that is why they are called initial boundary values. The initial boundary values for our analysis are:-
  - a. Inlet fluid pressure: - 105 Pa
  - b. Wall: - Stationary
5. Formulation: - implicit
  - a. Flux type: Roe-FDS
  - b. Gradient: - Least Square Cell Based
  - c. Flow: - second order upwind
  - d. Turbulent kinetic energy :- first order upwind
  - e. Turbulent dissipation rate :- first order upwind.

After carrying out all these initial set-ups run the calculation with required number of iterations. More the number of iterations the obtained result will be more close to the actual values.

## IV. RESULTS AND DISCUSSIONS

The ANSYS CFD analysis of the existing CNG engine model with single C-D nozzle is carried and different contours are like pressure, velocity and temperature plotted to analyse the mixing of fuel as shown in figure below. From the velocity contour shown in the fig we can see that the velocity of the injected air-fuel mixture is very low and non-uniform which results in non-uniform temperature distribution inside the cylinder and also the pressure inside the cylinder is very low.

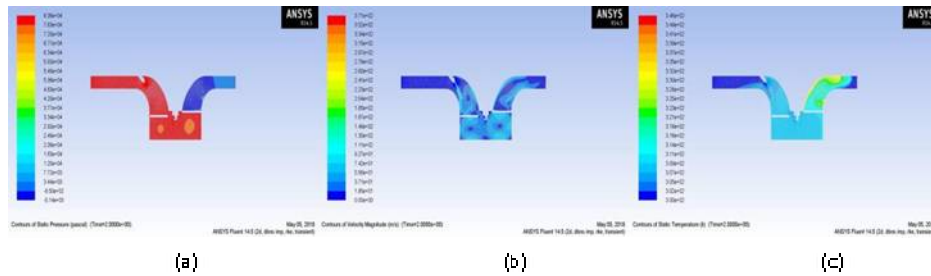


Fig. 2. (a) contours of static pressure in single nozzle (b) contours of velocity in single nozzle (c) contours of temperature in single nozzle.

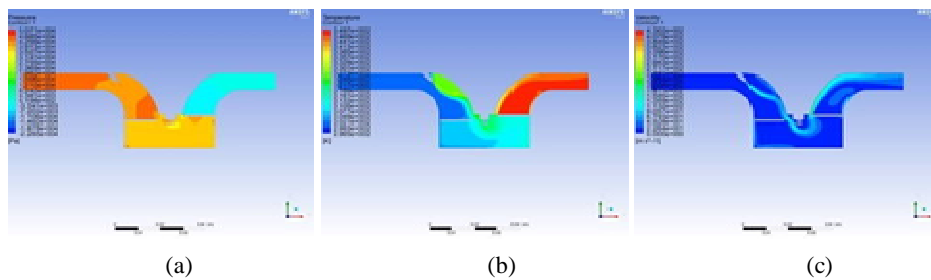


Fig. 3 (a) contours of pressure in cd nozzle nozzle (b) contours of temperature in cd nozzle nozzle (c) contours of velocity in cd nozzle

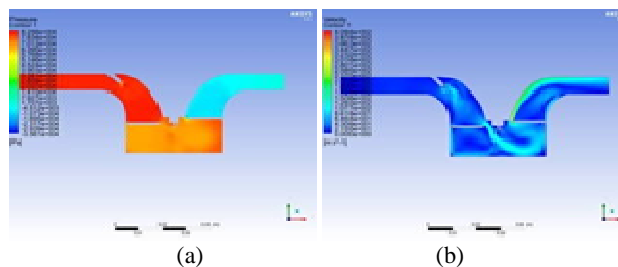


Fig. 4 (a) contours of pressure in 3 holes nozzle (b) contours of velocity in 3 hole nozzle

In the second model we use the single ventury nozzle to analyse the mixing of fuel and the contours are again plotted. In this model relatively high mixing velocity is achieved as compared to single C-D nozzle and also the temperature distribution and pressure is relatively high In the third model we use three nozzle injector, here themixing velocity is the highest of the previous two models and the temperature distribution is alsouniform.The pressure obtained inside the cylinder is high and uniform among the other two as depicted by the contours.

## V. CONCLUSION

In this project we have done ansys CFD analysis on CNG engine nozzles to know the varies character of the CNG gas depending up on the nozzle that is used to inject totally three types of nozzles are used in this study each and every process are taken carefully to understand the character of the fluid as per the study We can conclude both the cd nozzle and the three holes nozzles are a good choice because the nozzles are spraying the fluid in high velocity in all over the inlet so the fuel can mix with the air. So we here by conclude the cd nozzle or the three holes nozzles are a better choices for any fuels.

# International Journal of Innovative Research in Science, Engineering and Technology

*(A High Impact Factor, Monthly, Peer Reviewed Journal)*

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

## REFERENCES

- [1] Nimit M.Patel, Dr. Ashwin D.Patel, "A Parametric Study for Optimization of Gas Injector Orientation and Its Effect on Dual Fuel Engine Using CNG as Fuel", International Journal of Applied Engineering Research ISSN 0973-4562 Volume 12, Number 19 (2017) pp.8556-8559
- [2] Hussein A. Mahmood, "New Design of a CNG-H<sub>2</sub>-AIR Mixer for Internal Combustion Engines: An Experimental and Numerical Study", Energies 2017, 10, 1373
- [3] Patel Nimit M. A.D.Patel, "Conversion of Diesel Engine to Port Injection CNG Engine Using Gaseous Injector Nozzle Multi Holes Geometries Improvement: A Review, International Journal of Automotive Engineering", Vol. 6, Number 3, Sept 2016
- [4] M. A. Shaikh, Aditya H. Shukla, "Optimization of natural gas mixture design by computational method for improving swirl effect to obtain enhancement of SI engine performance: A Review", International Journal of Current Engineering and Technology, E-ISSN 2277-4106, P-ISSN 2347-5161, 2016
- [5] James Sevik, et al., "Influence of Injector Location on Part-Load Performance Characteristics of Natural Gas Direct-Injection in a Spark Ignition Engine", SAE International, 2016-01-2364, 2016
- [6] Supee, et al., "Effect of Compressed Natural Gas (CNG) Injector Position on Intake Manifold towards Diesel-CNG Dual Fuel (DDF) Engine Performance", Journal Teknologi (Sciences & Engineering) 70:1 (2014), 107-115, EISSN 2180-3722
- [7] T. Yusaf, et al., "Effect of Compressed Natural Gas Mixing on the Engine Performance and Emissions", International Journal of Automotive and Mechanical Engineering (IJAME), ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online); Volume 8, pp. 1416-1429, July-December 2013
- [8] Shahrir Abdullah, et al., "Numerical Analysis of the Combustion Process in a Compressed Natural Gas Direct Injection Engine", Journal of Applied Fluid Mechanics, Vol. 1, No. 2, pp. 65-86, 2008. filling and object removal by exemplar-based image inpainting.", IEEE Transactions on Image Processing, vol. 13, no.9, pp. 1200-1212, 2004.