DOI : https://doi.org/10.56452/208

ISSN: 0974-5823 Vol. 6 (Special Issue, Nov.-Dec. 2021)

International Journal of Mechanical Engineering

Investigation of Different Combustion chamber and Nozzle angle of GF 50I MP Diesel engine for performance Improvement using Computational Fluid Dynamics modeling

Apurva Indrodia[1] Dr. Mohit Diwan[2] Dr. Utsav Gadhia[3] Navneet Yadav[4] Bhavin Bhimani[5]

[1]Assistant Professor, Department of Mechanical Engineering, Ahmedabad Institute of Technology, Ahmedabad, Gujarat, India. apurvcader@gmail.com

[2]Associate Professor and HOD, Department of Mechanical Engineering, Ahmedabad Institute of Technology, Ahmedabad, Gujarat. diwanmohit@gmail.com

[3] Assistant Professor, Department of Mechanical Engineering, Gujarat Power Engineering and Research Institute, Mehsana, ,Gujarat Technological University, Gujarat. gadhiautsav@gmail.com

[4]Assistant Professor, Department of Mechanical Engineering, Ahmedabad Institute of Technology, Ahmedabad, Gujarat, India. Ns221990@gmail.com

[5]Assistant Professor, Department of Mechanical Engineering, Marwadi University, Rajkot, Gujarat. bhavinbhimani123@gmail.com

Abstract : Efficient energy utilization becomes the concentrative point among research group. Among this agriculture field is most popular. Direct injection diesel engine GF 50I MP is considered as reference model for analysis. Here work is concentrated on homogeneous combustion considering controlling parameters as nozzle angle and piston bowl geometry. Further reduction in emission parameters like NO_x, CO_2, HC is also a prime aim. Key parameters velocity and turbulence kinetic energy (TKE) is analyzed for same using CFD tool ANSYS. Different geometry and nozzle angle analysis leads to improved TKE and velocity at an angle of 60^0 and geometry with shallow depth of 19 mm at throat section and edge radius of 41 mm. Result shows that on the aspect of velocity and NO_x emission Modified geometry-1 and improved nozzle angle-2 (60°) is better among all the cases. Leads to form optimum geometry of piston and nozzle angle and reduction in NO_x and increase in HC.

Key Words—Diesel engine, Emission, Turbulence, Swirling, Combustion chamber geometry, Nozzle angle, CFD.

I. INTRODUCTION

An i.c. engine is one of the best available sources of energy in the field of automobile and agriculture. Major issue in the diesel is environmental problem and performance of diesel engine is enhancing by proper design of combustion chamber with proper nozzle spray angle. Flow and combustion chemistry of fuel in combustion chamber will effect on pollution emission from a single cylinder four stroke diesel engine. Apurva M indrodia et. al.[1] for more efficient combustion, less emission and soot mixture should homogeneous and burn completely. Combustion chambers with certain geometry spray angle of fuel and mixing process having great impact on combustion and emission. Jaychandars.[2,3] made study on combine effect of injection pressure and combustion chamber geometry on performance of biodiesel fueled diesel engine. Result shows that improvement in brake thermal efficiency and reduction in brake specific fuel consumption. in recent year, much progress has been made on study of fuel atomization and combustion simulation which provide guidance for prediction of data and better

Understanding of mixture formation and combustion process, Computational fluid dynamics (CFD) simulation predicts the data related to spray characteristics, mixture formation, emission data and combustion processes. It will reduce expenses. For conventional direct injection (DI) diesel engine distribution of mixture

DOI : https://doi.org/10.56452/208

of air and fuel is non-uniform. Charles Roberts et.al[4] investigated that since last 10-15 years diesel industries facing problem of smoke production through use of ever increasing fuel injection pressure. Therefore it is necessary to understand effect of number of nozzle holes and arrangement of nozzle is important to relate fuel flow characteristic. de risi[5] identified effect of combustion chamber geometry configuration and noticed trends of producing NO_x and soot, by different nozzle angle. The main problem of using CFD is it required very high computational time for engine cycle simulation and expertise by their user. Equally pressure, temperature and equivalence ratio affect the combustion and emission parameter like NO_x, HC etc. As fuel mass fraction increase temperature increases. So that burned gas equivalence ratio becomes leaner by mixing with excess air reduces the NO emission. [6]Combustion chamber having low squish area generate high turbulence which helps in generates better combustion. [7] Tumble motion generates good turbulence. Better turbulence can be generating by good intake port design. Re-entrant combustion chamber helps in improving the atomization of fuel reduces the emission. [8]

A. Nomenclature:

u, v, w - Velocity in x, y, z direction (m/s)
ρ – Density of fuel (kg/m^3)
k- Turbulence Kinetic energy
$\mathcal{E}$- Turbulence dissipation rate
υ – Kinematic viscosity
μ – Dynamic viscosity
NO_x- Nitrogen oxide
a_x– Acceleration in x- Direction
h – Enthalpy
e- Total energy
m_f – Mass of fuel
q^c and q^s are source terms related to chemical reaction
CFD- Computational Fluid Dynamic
ma – Mass of air
t- Time (sec)
p- Pressure (Pa)
T- Temperature (K)
τ–Stress (m/s^2)
HC- Hydrocarbon
f_x – External force

II. ROLE OF COMPUTATIONAL FLUID DYNAMIC (CFD):

CFD analysis or Computational Fluid Dynamics has emerged to be one of the most important fields of study that plays a crucial role in the modern engineering environment. The importance of CFD analysis in engineering and manufacturing industries includes effortlessly calculatingthe fluid forces and understanding the impact of gas or liquid on the performance of a product.Effective CFD analysis simplify efficient and quick simulation of heat transfer and fluidflow of a product, part or structure to determine its performance level across diverse fluid forces. In computational modelling of turbulent flows, one common objective is to obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modelled. For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modelling approaches excessively expensive. The primary approach in such

DOI : https://doi.org/10.56452/208

cases is to create numerical models to approximate unresolved phenomena. CFD analysisis proving to be a necessary design tool in the industry Expanding the details of flow physics to the eyes, cost control and change times. High reliability and precision CFD results offer considerable independence from expensive testing process.

III. MESHING AND BOUNDARY CONDITION OF BASELINE GEOMETRY:

The 2-D Model shown in fig.-1 is used for analysis. Simulated engine has direct injected Diesel Fuel with single hole injector and hemispherical bowl Combustion chamber. Meshing is performed in commercial software package GAMBIT for better meshing. For simulation quad (Map) mesh is used. As by analyzing the geometry is free from mesh types and number of elements. Fig.1 shows geometry with boundary condition and meshed geometry when piston at top dead center (TDC) and injection parameter are given in table-1 for GF 50 I DI diesel engine. Nozzle angle for fuel spray is 55°.

TABLE I DETAIL OF FUEL SUPPLY FOR BASELINE GEOMETRY

Parameter	Magnitude
Nozzle angle	55°
Mass flow rate (kg/s)	0.0015
No. of nozzles	1 with single hole

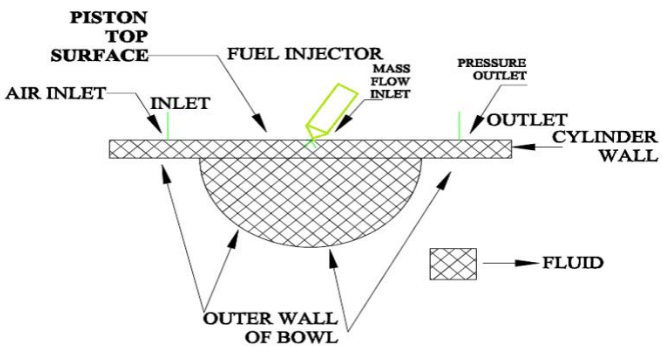

Fig. 1 a) Boundary condition of Baseline geometry

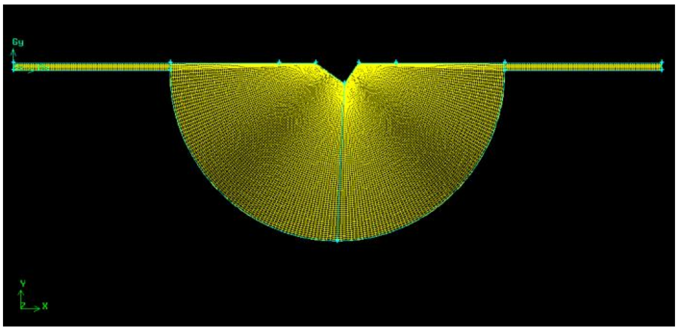

Fig. 1 b) Meshing of Baseline geometry

DOI : https://doi.org/10.56452/208

IV. Methodology

A. Experimental Setup at Industry:

Experimental work is carried out at field marshal group of companies. They are manufacturing many types of products like Diesel engine, Petrol engine, Kerosene engine, Diesel generating sets, various types of pumps and lubricating products oil and grease etc. This group is one of the largest manufacturers of Diesel engine used in agriculture.

Experimental facility available in industry is as shown in fig. 2 for testing purpose.

Fig.2Experimental Set up

TABLE II SPECIFICATION OF ENGINE

Parameter	Magnitude
Engine type	GF 50I MP Four stroke single cylinder Diesel engine
Crank shaft speed (RPM)	1500
Bore (mm)	85
Stroke (mm)	80
Cubic capacity (ltrs.)	0.455
Nozzle Diameter (mm)	9.7
Fuel	Diesel ($C_{10}H_{26}$)
Compression Ratio	18:1
Fuel consumption (kg/s)	0.0015

B. Experimental result obtained from Engine testing:

Results are obtained for emission parameter like NO_x, CO_2 and HC are given below in table 3.

DOI : https://doi.org/10.56452/208

TABLE III EMISSION TESTING DATA

Parameter	No_x (ppm)	CO_2 (g/kwh)	HC (g/kwh)
Magnitude	442.4	1.50	0.38

V. Analysis Using CFD Package Fluent:

TABLE IV Temperature Detail for Baseline geometry:

Parameter	Temperature Magnitude (K)
Air inlet	298
Fuel inlet	333
Piston bowl	420
Exhaust outlet	363
Cylinder Head	315

A. Governing equation used for combustion of fuel:

In Diesel engine, the fuel is sprayed directly into the cylinder through the fuel injection nozzle. The fuel is then broken into a number of droplets. These droplets undergo collision processes, exchanging momentum and energy with the high temperature and pressure surrounding gases inside the cylinder. Finally the droplets evaporate into vapor and mix with air.

- Governing equations required for combustion process in diesel engine are:

a) Continuity equation
b) Momentum equation
c) Energy equation
d) Species transport equation

- Continuity equation:Forincompressible fluid:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad (1)$$

- Momentum equation:

The momentum equations is derived based on Newton's second law of motion, states 'the rate of change of momentum of a fluid particle equals to the sum of forces on the particle.

X-component:

$$\rho \frac{Du}{Dt} = \frac{\partial (p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \qquad (2)$$

Where,

ρ = density (kg/m^3)

p = Pressure

τ = stress

x, y and z are the direction

- Energy equation:

The momentum equations is derived based on First law of thermodynamics, states 'the rate of change of an energy of a fluid particle is equal to sum of the rate of heat addition to the fluid particle and the rate of work done on the particle.'

$$\left[\frac{\partial(\rho e)}{\partial t} + \nabla\left(\partial \underset{u}{\rightarrow} e\right) = -p\nabla \underset{u}{\rightarrow} - \nabla\left[-k\nabla T - \rho D \sum_{m} h_m \nabla\left(\rho \frac{m}{\rho}\right)\right] + q^c + \rho\mathcal{E} + q^s\right]$$

(3)

Where, q^c *and* q^s are chemical heat release and fuel spray interaction.

- Species transport equation:

$$\frac{\partial\rho_m}{\partial t} + \nabla\left(\rho_m \underset{u}{\rightarrow}\right) = \nabla\left[\rho D\nabla\left(\frac{\rho_m}{\rho}\right)\right] + \rho^c{}_m + \rho^s \quad (4)$$

Where, $\rho^c{}_m$ *and* ρ^s are source term due to chemical reaction and fuel spray. [9]

B. Chemical process involved in combustion process:

Chemical reaction of combustion process produced by burnt gases is a function of mean local quantities computed by software for oxidation of methane with oxygen to form vapor and Carbon dioxide is considered. Fuel used for combustion is Diesel ($C_{10}H_{26}$). Following species is used for combustion of Diesel fuel.

$C_nH_p + O_2 \rightarrow CO_2 + H_2O$ (5)

$2C_{10}H_{26} + 37O_2 \rightarrow 20CO_2 + 34H_2$ (6)

Ansys Fluent predict local mass fraction of each species through solution of convection diffusion equation for the i^{th}species.

$$\partial/\partial t\ (\rho y_{i)} + \nabla(\rho \vec{v} y_i) = -\nabla \vec{Ji} + R_i + S_i \qquad (7)$$

Where, S_i = Net rate of production of species $i^{th}$by chemical reaction.

R_i = Rate of creation by addition from dispersed phase plus any user defined function.

C. CFD Results obtained for Baseline geometry:

Result is simulated in Ansys Fluent 16.0. There are Three Nozzle angle and Two Geometry is analyzed. Angle of fuel spray is taken as 55° baseline angle and two other angles as modified angle-1(50°) and modified angle-2 (60°).

- Results for 55° Nozzle angle:

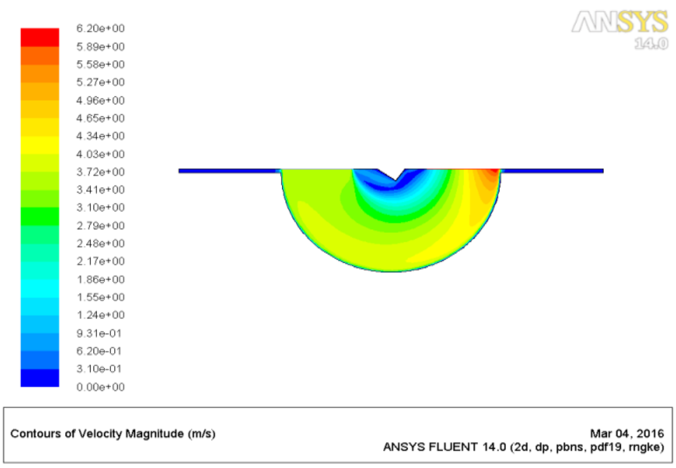

Fig.3 a) Velocity contour at 55° Nozzle angle

DOI : https://doi.org/10.56452/208

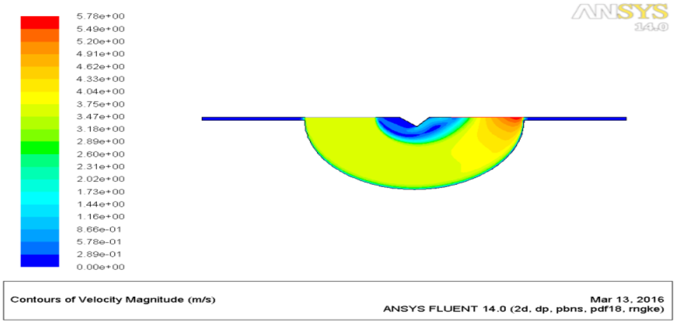

Fig.3 b) Velocity contour at 50° Nozzle angle

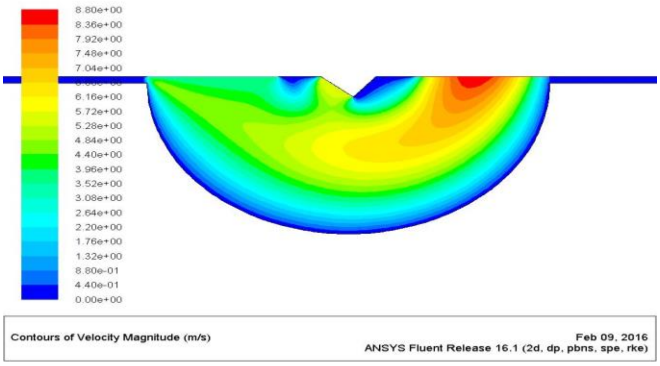

Fig.3 c) Velocity contour at 60° Nozzle angle

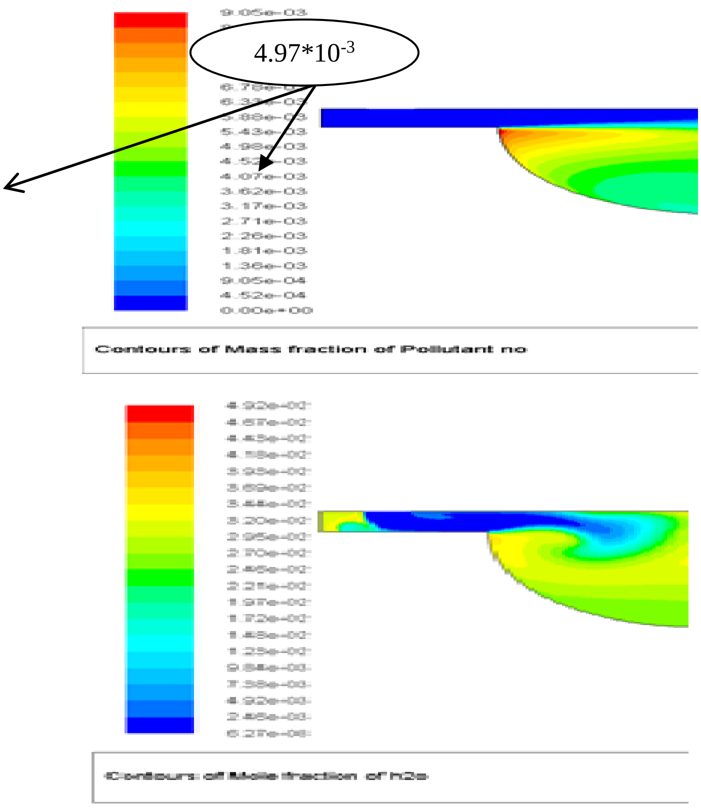

FIG. 4 a) NO mass fraction b) H_2O mole fraction

$$Noxppm = \frac{No\_Mole\_FractionX10^6}{1 - H_2O\_Mole\_Fraction}$$

DOI : https://doi.org/10.56452/208

D.CFD Results obtained for Modified Geometry-1:

- Modified Geometry is shown in fig. 5. In this geometry Shape of piston bowl is changed. Depth of bowl is made as shallow depth. This geometry gives better result than Baseline geometry. Velocity increase and NO_x emission decrease in this geometry.

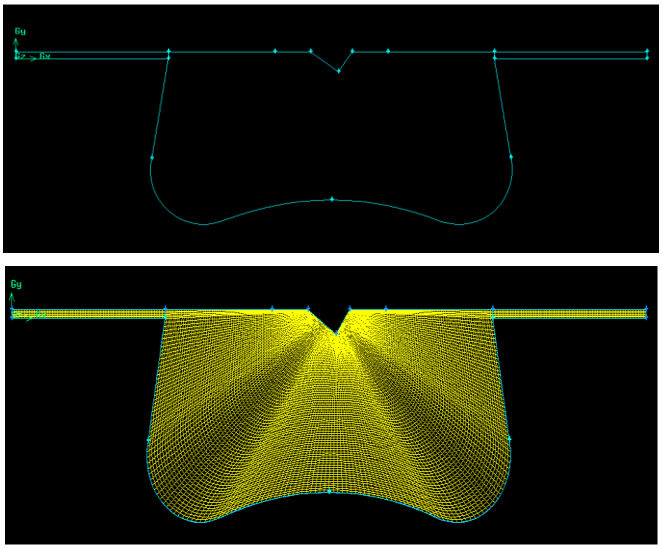

Fig. 5 Boundary and Meshing of modified geometry-1

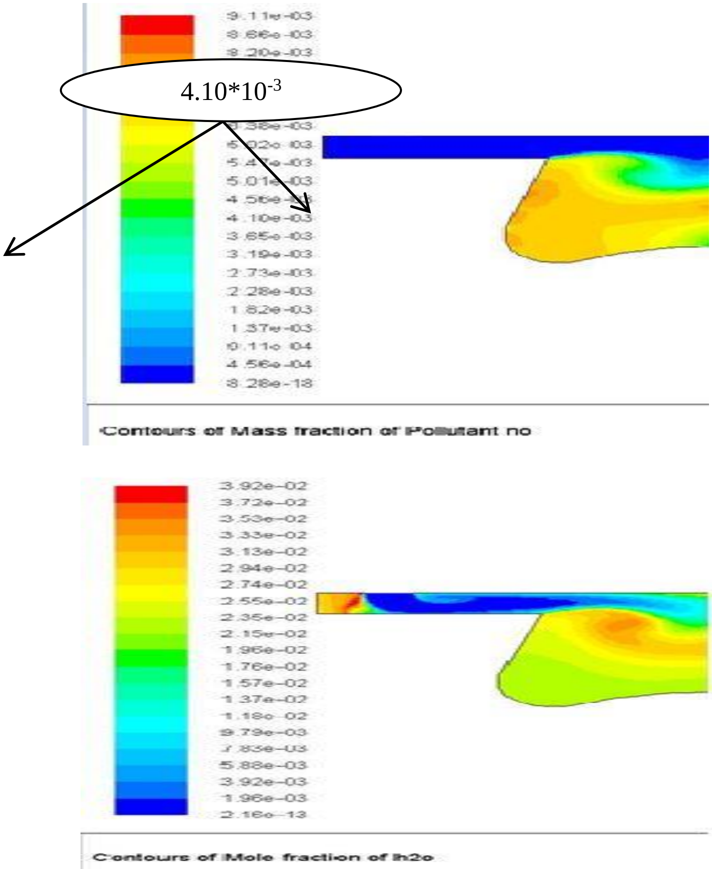

Fig. 6 a) NO mass fraction b) H_2O mole fraction

VI. Result and Discussion

A. Validation of Experimental Result with Simulation for Baseline Geometry (55°) Nozzle angle:

TABLE V Comparison of Experimental and Simulation data

Parameter	NO_x (ppm)	CO_2(g/kwh)	HC (g/kwh)
Experimental	442.4	1.50	0.38
Simulation	498	-	-

Table VI % Error in simulation of Baseline geometry

Parameter	NO_x (ppm)	% Error
Experimental	442.4	12.56 % ↑
Simulation	498	

TABLE VII Result comparison for Modified geometry-1 and Baseline geometry:

Parameter	NO_x (ppm)	% Error
Baseline Geo.	498	6.87 ↓
Modified geo-1	412	

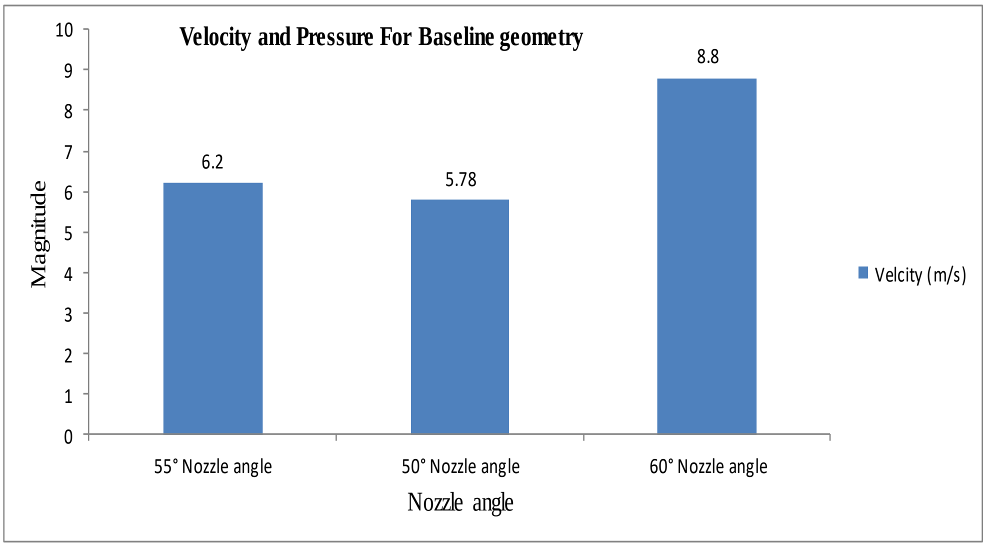

Fig. 7 Velocity for Baseline geo. with different nozzle angle

DOI : https://doi.org/10.56452/208

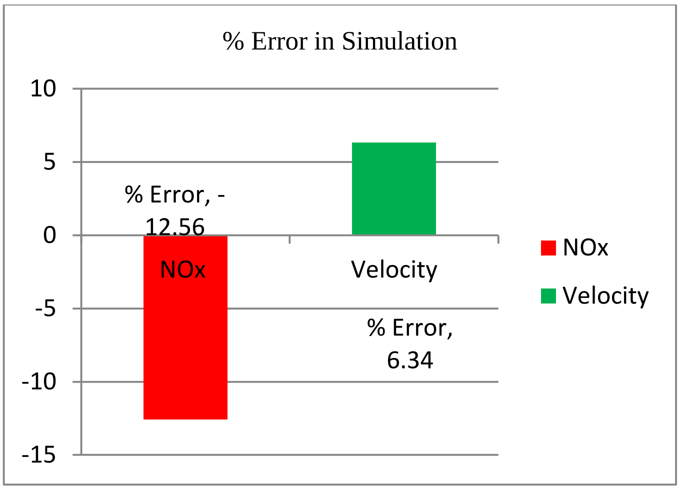

Fig. 8 % Error in simulation result for Baseline geometry

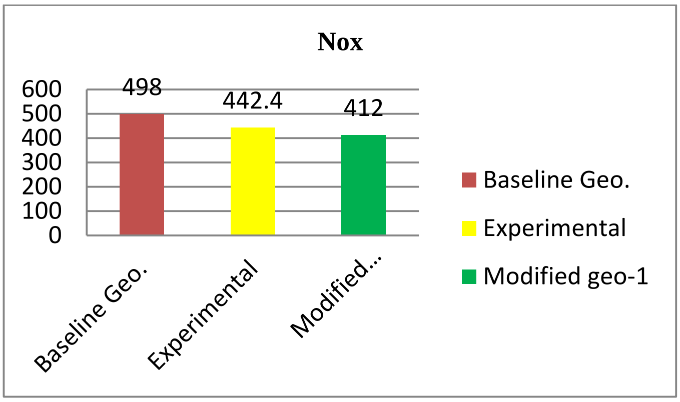

Fig. 9 NO_x emission Comparison

- CONCLUSION

Research in the field of automobile and agriculture mainly focus on increment in efficiency and Reduction of emission.

By using modified geometry-1 velocity should be increased 6% and NO_x emission reduced upto 7%.

From above Analysis and Experimental investigation it is concluded that Modified geometry-1 and Nozzle angle of 60° is better among all the results.

- ACKNOWLEDGMENT

First author is grateful to Mr.RameshkumarBhoraniya HOD of Mechanical Department.I am also thankful to my guides for their valuable support. We are also thankful to Marwadi education foundation, Rajkot for providing best facility and kind support for work and encouragement, support and co-operation during the entire work. We are also thankful to Gujarat forging pvt. ltd. for providing experimental facility and sharing of their experience of work.

DOI : https://doi.org/10.56452/208

REFERENCES

[1] A.M. Indrodia, N.J. Chotai, B.M. ramani. Investigation of different combustion chamber geometry of diesel engine using CFD modeling of in-cylinder flow for improving the performance of engine. 5th International & 26th All India Manufacturing Technology, Design and Research Conference (AIMTDR 2014) December 12th–14th, 2014, IITGuwahati, Assam, India.

[2] S. Jaichandar, P. Senthil Kumar, K. Annamalai. Combined effect of injection timing and combustion chamber geometry on the performance of a biodiesel fueled diesel engine. Energy 47 (2012) 388e394.

[3] S. Jaichandar, K. Annamalai. Combined impact of injection pressure and combustion chambergeometry on the performance of a biodiesel fueled diesel engine.Energy (2013) 1-10.

[4]] Charles E. Roberts, Chad Stovell, Rainer Rothbauer, Darius Mehta. Advancement in Diesel combustion system design to improve the smoke BSFC-tradeoff.C.E.Roberts et al./International Journal of Automotive Engineering 2 (2011) 55-60

[5] Arturo de Risi, Teresa Donateo, Domenico Laforgia, Optimization of the combustion chamber of direct injection diesel engine, SAE 2003-01-1064.

[6] HelgiFridriksson, Bengt Sunden, ShahrokhHajireza, Martin Tuner. "CFD Investigation of Heat Transfer in a Diesel Engine with Diesel and PPC Combustion Modes". JSAE 20119177 SAE 2011-01-1838.

[7] Stefania Falfaria, Federico Brusiania, Piero Pellonia. "3D CFD analysis of the influence of some geometrical engine parameters on small PFI engine performances – the effects on the tumble motion and the mean turbulent intensity distribution". Elsevier Energy Procedia 4 (2014) 701 – 710.

[8] R. ThundilKaruppa Raj, R. Manimaran. "Effect of Swirl in a Constant Speed DI Diesel Engine using Computational Fluid Dynamics". ISSR journal, Vol. 4(4) 2012, PII: S2180 -1363(12)4214.

[9] R. Bhoobathi, ShaikAmjad, and Dr. R. Rudramoorthy, Diesel Engine Combustion Simulation using Computational Fluid Dynamics, Proc. of. Int. Conf. on Advances in Mechanical Engineering 2010.

[10] Haditaghavifar, Mohammad taghishervani-tabar, Majid abbasalizadeh, "Numerical study of the effects of injector needle movement and nozzle inclination angle on the internal fluid flow and spray structure of a group-hole nozzle layout", 17 april,2015 journal of applied mathematical modeling.

[11] Zhenyuzhang, Chngluzhao, Zhaoyixie, Fujunzheng, Zhenfengzhao, "Study on the effect of the nozzle diameter and swirl ratio on the combustion process for an opposed-piston two stroke diesel engine", the 6th international conference on applied energy, ICAE 2014. 61(2014) 542-546. Journal of fuel.

[12] C.D. Rakopoulos, G.M. Kosmadakis, E.G. Patriots. "Investigation of piston bowl geometry and speed effects in a motored HSDI diesel engine using a CFD against a quasi-dimensional model". 15780 Athens, Greece, Energy Conversion and Management 51 (2010) 470–484.

[13] Tarek M. Belal, El Sayed M. Marzouk, Mohsen M. Osman. "Investigating Diesel Engine Performance and Emissions Using CFD". Energy and Power Engineering, 2013, 5, 171-180.Journal of scientific research.

[14] P. U. Bhuva, U. C. Arvadia, A. A. Katariya, "Experimental investigation work of diesel engine by developing combustion chamber geometry, International journal of science technology and engineering", Vol.1, issue 12, june 2015. ISSN 2349-784.