CFD Analysis Diesel Spray Mixing Nozzle in Various Angle

N.Kalaimani[a], M.Sundararaj[b], S.Elangovan[c]

a – Assistant Professor, Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Chennai.
B, c –Professor, Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Chennai.

Article History: Received: 11 January 2021; Accepted: 27 February 2021; Published online: 5 April 2021

Abstract: The main objective of this project is finding the optimized mixed ratio of the diesel spray mixer in the IC Engines, CFD Methodology is used for this analysis the spray angle variations give the various mixing ratios of the sprayer for better combustion ratios the turbulence will decide the best mixing efficiency, the turbulence, pressure and velocity results inside the mixing chamber is analyze through CFD methodology

Keywords: CFD, Diesel Spray mixing, Nozzle angles, Turbulence

INTRODUCTION

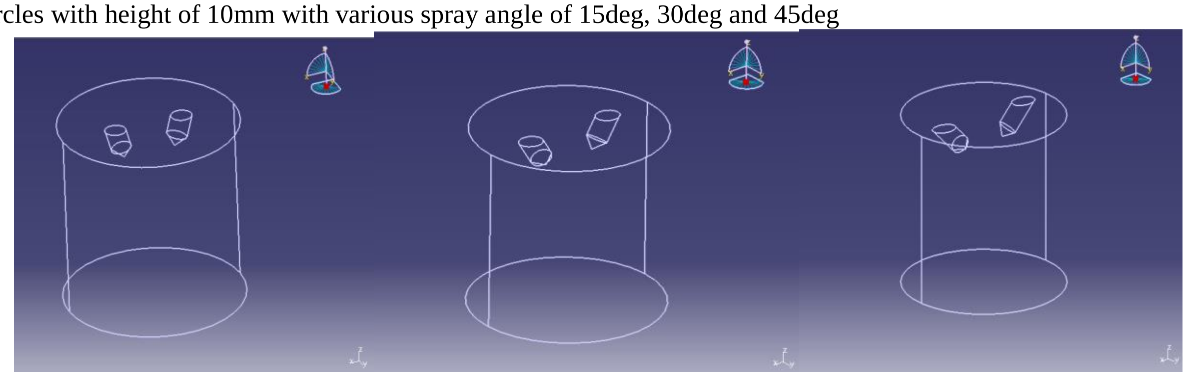
Fuel mixing ratio is very important in diesel engine combustion chamber to get proper combustion process, while creating the turbulence in flow, that turbulence mixing is mainly based on injector shape, angle and velocity of the flow from fuel tank. In improper mixing ratio inside combustion chamber, it creates combustion inefficiency, carbon deposit and unpleasant gases, this unpleasant gas creates air pollution.

In this research work we used circular nose cone for atomization process with various angle to get proper air fuel mixing ratio inside combustion chamber. Objective of this paper to simulate and analyze the flow characteristics in injector nozzle exit with help of CFD, the RANS k-Z SST model was used for turbulence, CATIA is used for model design based on given boundary condition.

The variation of mixing ratio is validated by CFD for different injection angle, and different pressure values. Finally, we compare the results for each angle and applied pressure value.

MODELING

The modelling diesel spray nozzle modelled in CATIA Software with the commands of shaft and groove with the mixing chamber dimensions of 20mm diameter 50mm height, the spray nozzle dimension is 3mm on both circles with height of 10mm with various spray angle of 15deg, 30deg and 45deg



15 DEGREE 30 DEGREE 45 DEGREE

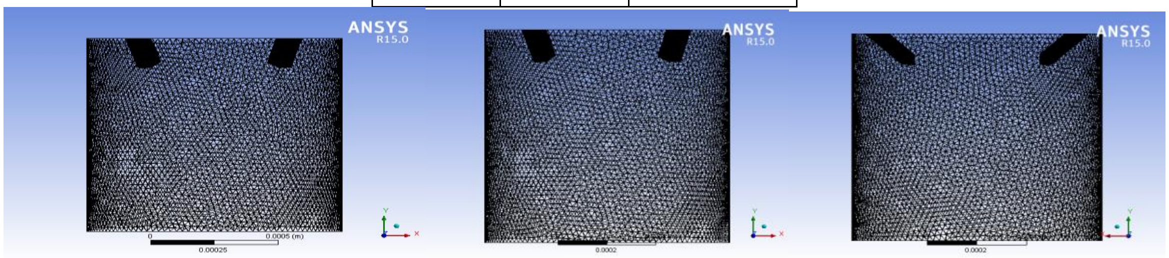
Fig 3D model Diesel Spray injectors in various angles

Meshing of the spray nozzle is used to find the best mixing point in every mm the unstructured mesh methodology is used the tetrahedral elements make more elements, in this nozzle spray analysis tetrahedral elements are used the nodes and elements details shown below in the table

Table 1 Mesh Details of Spray nozzles

MESH	NODES	ELEMENTS
15 DEG	78851	423536

30 DEG	78835	423498
45 DEG	117660	634890



15 DEGREE 30 DEGREE 45 DEGREE

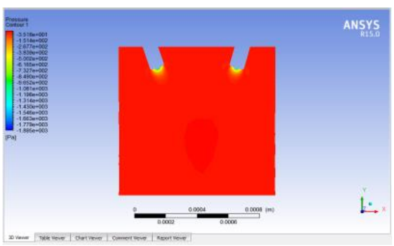
Mesh images of spray nozzles

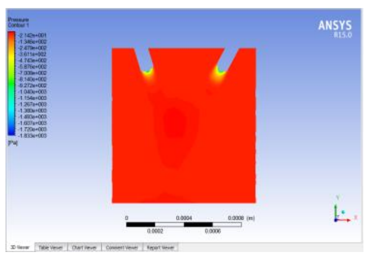
CFD Simulation

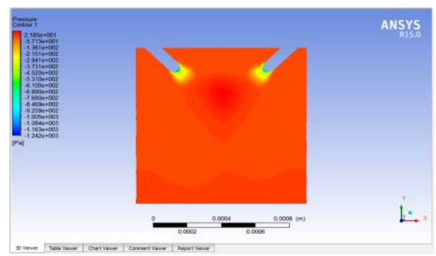
The CFD Simulation of the Diesel spray nozzle will analyze through Finite Volume Method of CFD Methodology, Navier Stokes equation is used to pressure and velocity contours K-epsilon turbulence model is used for find the turbulence energy semi implicit pressure linked equation is used for the analysis

Results and discussion:

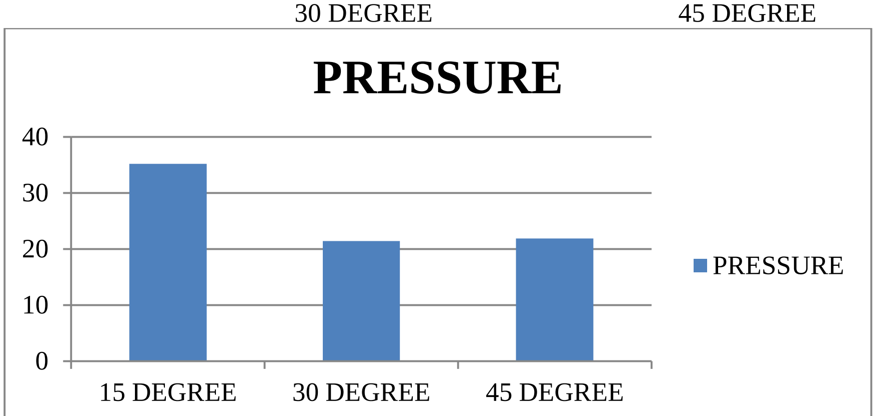
Pressure contours:





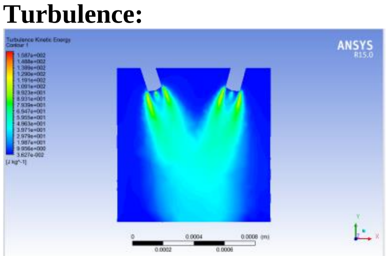


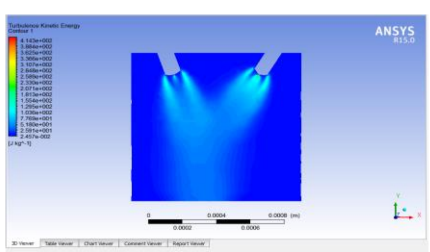
15 DEGREE 30 DEGREE 45 DEGREE

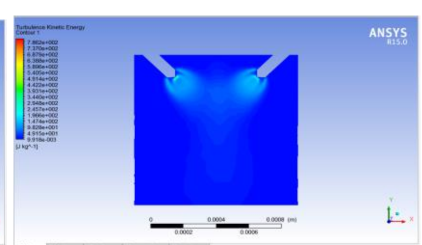


VARIOUS ANGLE	PRESSURE
15 DEGREE	35.18
30 DEGREE	21.42
45 DEGREE	21.85

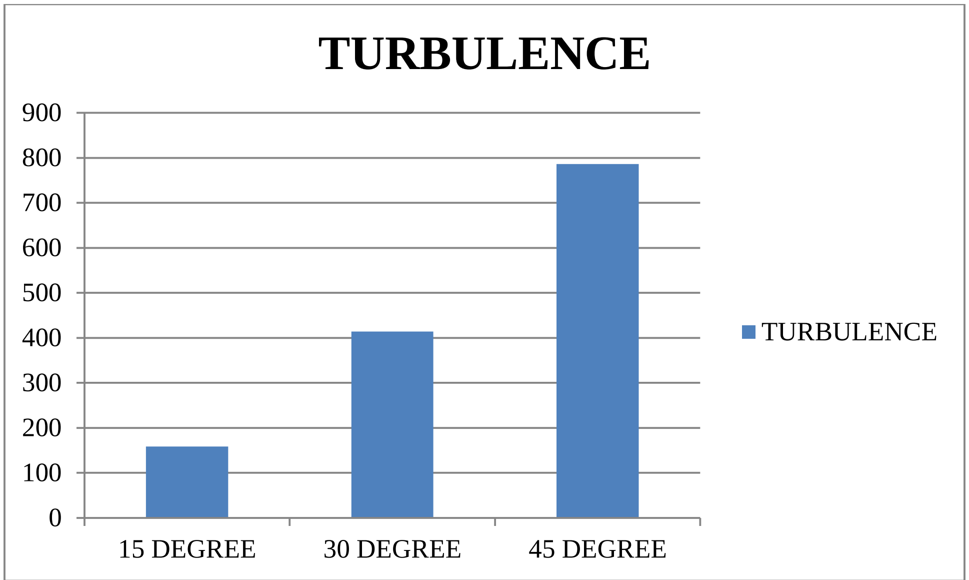
Turbulence:





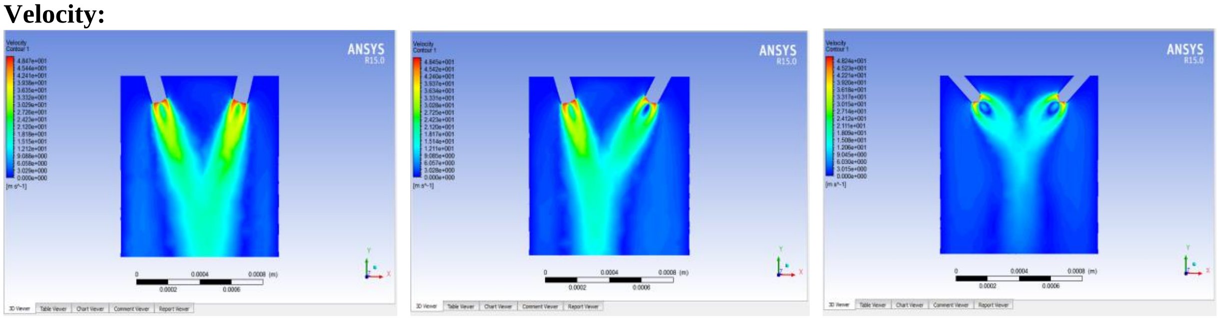


15 DEGREE 30 DEGREE 45 DEGREE

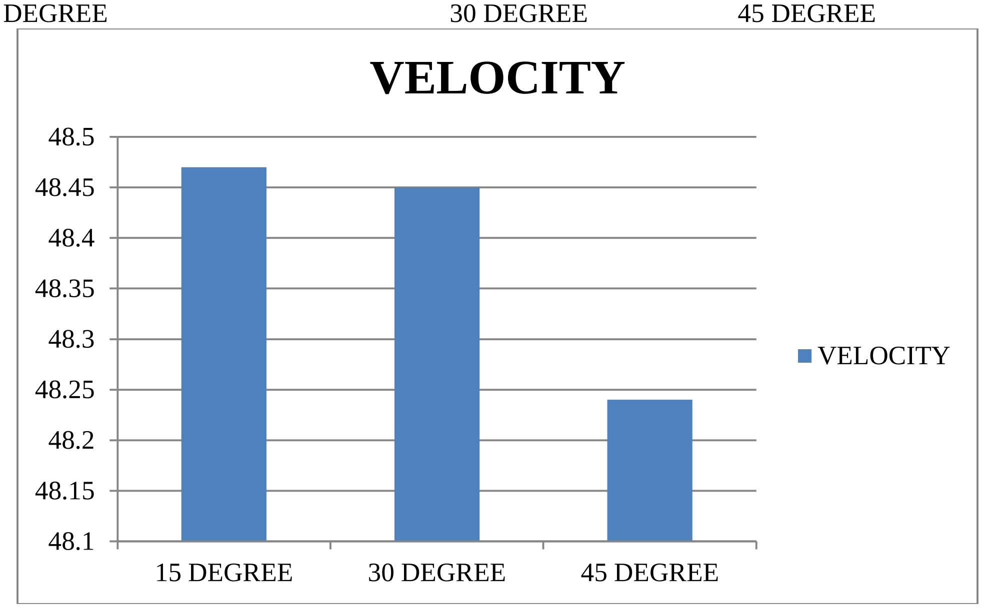


VARIOUS ANGLES	TURBULENCE
15 DEGREE	158.7
30 DEGREE	414.3
45 DEGREE	786.2

Velocity:



15 DEGREE 30 DEGREE 45 DEGREE



VARIOUS ANGLE	VELOCITY
15 DEGREE	48.47
30 DEGREE	48.45
45 DEGREE	48.24

Conclusion:

The Diesel Engine Spray nozzle analysis done in various nozzle angles of 15 deg, 30 deg and 45 deg of nozzle angles, the CFD analysis of spray mixing nozzle done in ANSYS Fluent software the results of turbulence kinetic energy will decide the mixing rate if the turbulence energy increase the mixing rate will increase and the velocity is less the mixing time will be increase these conditions will satisfied in the model of 45 deg spray nozzle models

REFERENCES

1. Salvador, F. J., J-V. Romero, M-D. Roselló, and J. Martínez-López. "Validation of a code for modeling cavitation phenomena in Diesel injector nozzles." Mathematical and Computer Modelling 52, no. 7 (2010): 1123-1132.
2. Winklhofer, E., E. Kull, E. Kelz, and A. Morozov. "Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions." In ILASS-Europe 2001, 17 International Conference on Liquid Atomization and Spray Systems. 2001.
3. Chaves, H., and F. Obermeier. "Correlation between light absorption signals of cavitating nozzle flow within and outside of the hole of a transparent diesel injection nozzle." Proc. ILASS-EUROPE. Manchester, UK (1998): 6-8.
4. Pratama R. H., Sou A., Wada Y., and Yohohata H., “Cavitation in mini-sac nozzle and injected liquid jet” In THIESEL 2014 Conference on thermo and fluid dynamic processes in direct injection engines. Valencia, Spain (2014).
5. Salvador, F. J., Jorge Martínez-López, M. Caballer, and C. De Alfonso. "Study of the influence of the needle lift on the internal flow and cavitation phenomenon in diesel injector nozzles by CFD using RANS methods." Energy conversion and management 66 (2013): 246-256.
6. Jasak, Hrvoje. "Error analysis and estimation for the finite volume method with applications to fluid flows." (1996).
7. Menter, F. R. "Influence of freestream values on k-omega turbulence model predictions." AIAA journal 30, no. 6 (1992):1657-1659.
8. Versteeg, Henk Kaarle, and Weeratunge Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson Education, 2007.
9. H. W. Wang, L.B. Zhou, D. M. Ziang, Z. H. Huang (2000), “Study on the performance and emission of a compression ignition engine fuelled with dimethyl ether,” Proceedings of the institution of mechanical engineers Part D Journal of Automobile Engineering, Vol. 214, pp 101-106.
10. A Kowalewicz, MWojtyniak (2004),”New alternative fuels for I.C. Engines- A review,” Journal of KONES International Combustion Engines, Vol. 45,pp 358-368.
11. MitevRanjan, Karan Kumar, Subrata Kr. Ghosh,” Mine ventilation in a board and pillar mines using CFD”. International Journal of Emerging Technology and Advanced Engineering, Vol. 3(3), ICERTSD 2013, pp 389-393.