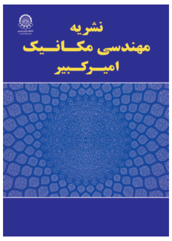

Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 50(4) (2018) 263-266
DOI: 10.22060/mej.2017.12159.5278

Numerical Study of Turbulence in Diesel Spray Combustion Using Large Eddy Simulation

M. R. Keshtkar, H. Khaleghi*, S. Yazdanparast, M. Khoramdel

Department of Mechanical Engineering, Tarbiat Modares University, Tehran, Iran

ABSTRACT: Under modern direct injection diesel engine conditions, the spray and combustion processes are known to be controlled by mixing. In large eddy simulation method, large eddies are solved directly and small scales are modeled, so it can potentially improve the predictive capability by better capturing the large-scale mixing of ambient air with the fuel vapor. In this paper, turbulent spray combustion is studied using the large eddy simulation method together with partially stirred reactor model using EPISO- SPRAY code. To develop this code, large eddy simulation method is applied in an Eulerian – Lagranian approach in which conservation equations of both phases are solved and then results of large eddy simulation are compared with those of Reynolds averaged of Navier-Stocks. Simulation of spray combustion using different sub-grid scale models of large eddy simulation method (Simple Smagorinesky and dynamic Smagorinesky) with partially stirred reactor combustion model are presented here. It is shown that with a fine mesh, results are in good agreement with experimental data. Results of non-reacting spray penetration length in gas environment and velocity profile during the intake stage are compared with related experimental data in order to validate the EPISO-SPRAY code performance. It is proved in this study that large eddy simulation results with relative fine mesh are much better than the Reynolds-averaged Navier–Stokes equations results. Results of reacting liquid spray using simple step kinetic and fuel vapor penetration length are compared with experimental data. It is shown that overall characteristics of diesel spray combustion such as liquid spray penetration and fuel vapor penetration are both in good agreement with experimental data using Reynolds-averaged Navier–Stokes equations or large eddy simulation models. Although small differences in the flame shape are seen with the two methods, maximum and minimum temperatures are predicted to be the same in both models.

Review History:

Received: 2 December 2016
Revised: 15 April 2017
Accepted: 21 May 2017
Available Online: 7 June 2017

Keywords:

Turbulence
Diesel spray combustion
Large eddy simulation
Reynolds averaged of Navier-Stocks equations
Diesel engine

1- Introduction

In recent decades, special attention has been paid to the topic of engine flow as one of the major areas of research. This is because of its highly complex features which is unsteady, compressible and turbulent as well as two phase and combustion. Engine in-Cylinder flow specifications can influence many mechanisms of the flow such as fuel spray penetration, evaporation and distribution in the combustion chamber, flame ignition and propagation, heat transfer and turbulent eddies. So choosing a proper method to model engine inflow, turbulence, chemical reactions and turbulence-chemical interaction is so important. In 2007 spray combustion using large eddy simulation and eddy dissipation model was carried out with KIVA-LES code by Hori et al. [1]. Significant differences were observed in the numerically computed heat release rates in comparison with the experimental data due to relatively large grid size which is not able to capture all energy spectrum containing scales. In this study the averaged jet and flame structure were captured well. In 2013, simulation of spray combustion using Large Eddy Simulation (LES) method and flamelet combustion model was performed by Hu et al. [2]. Bekdemir et al. [3] performed LES of diesel jet with tabulated chemical kinetics. Also, an unstructured non-uniform grid was used to obtain fine grid in the region of nozzle domain. Ignition delay time and flame lift-off height were in a good agreement with experimental data, but the unsteady development of the flame could not be captured well in this study. In 2013 and 2014, the LES method was applied to simulate diesel jet combustion using unsteady flamelet by Ameen et al. [4] and Ameen and Abraham [5], respectively. They studied unsteady flame evolution from unstable flame ignition until flame stabilization at the lift off height and concluded that ignition occurred in multiple spots around fuel jet and stabilization occurred in a specific flame lift-off height. In our study unlike the previous work, LES method (Dynamic smagorinesky based model) is applied to simulate turbulence in diesel spray combustion using Partially Stirred Reactor (PaSR) combustion model and single step chemical kinetics. Results of general flame characteristics are in good agreement with experimental data. It is proved in this study that the LES results with relatively fine grid, provides much better prediction than the Reynolds-averaged Navier–Stokes (RANS) results.

Corresponding author, E-mail: khaleghi@modares.ac.ir

2- Numerical Method

In our computer code which is named EPISO-SPRAY, Eulerian - Lagrangian method has been used to simulate two-phase diesel spray in engine. In order to solve the gas phase, conservation equations of mass, momentums, energy, as well as k and ε equations or LES method based on Smagorinesky or dynamic Smagorinesky equations, are used and Pressure Implicit with Splitting of Operator (PISO) algorithm is employed to resolve the coupling between the pressure and velocities. Heat and mass transfer model for liquid diesel spray evaporation in high-temperature gaseous environments is applied to the solver. Using finite volume method, diffusion terms are discretized by central difference while convection term is discretized using hybrid scheme. Spray is assumed to be fully atomized after injection. O'Rourke and Bracco model [6] is used for droplet-droplet collision and Reitz and Diwakar model [7] is applied for secondary droplet break-up. Bai and Gosman model [8] is used for spray wall impingement. Also, the combustion model is PaSR with single step chemical kinetics.

3- Results and Discussion

In order to show grid independency, numerical calculations for engine geometry as seen in Fig.1 and for the radial profiles of axial mean velocity in 36 degrees After Top Dead Center (ATDC) at the distance 10 mm from the cylinder head were made with four different mesh sizes of 30 × 80, 45 × 120, 90 × 240 and 135 × 360. It was observed that a 90× 240 grid is sufficient for a grid independent results.

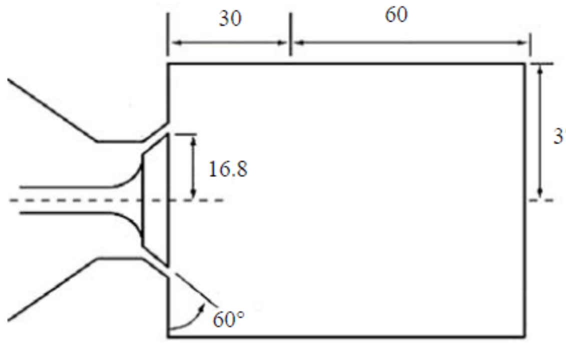

Figure 1. Schematic of Internal Combustion Engine (ICE) geometry

Fig. 2 compares the radial profiles of axial mean velocity in 36 degrees ATDC at the distance 10 mm from the cylinder head with the experimental results of Morse et al. [9] and the results obtained from the smagorinesky, dynamic smagorinesky and standard k-ε model.

As seen in Fig.2, LES method in the calculation of mean axial velocity predicts more accurate results than model particularly in the region near the wall in the intake stroke. Also as seen in Fig.2, The k-ε model is unable to capture the maximum velocity and LES method of dynamic smagorinesky based is more accurate than LES method of smagorinesky based.

To validate the results of spray performance in gaseous environment, comparisons of spray tip penetration length between the numerical and experimental observations of Ref. [10] for trapped pressure of 25 bar and trapped temperature

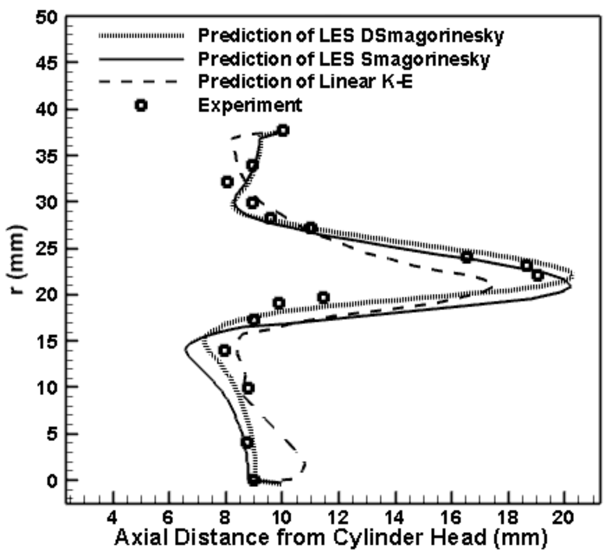

Figure 2. Comparison of axial mean velocity in 36 degrees ATDC at the distance 10 mm from the cylinder head with different turbulence models

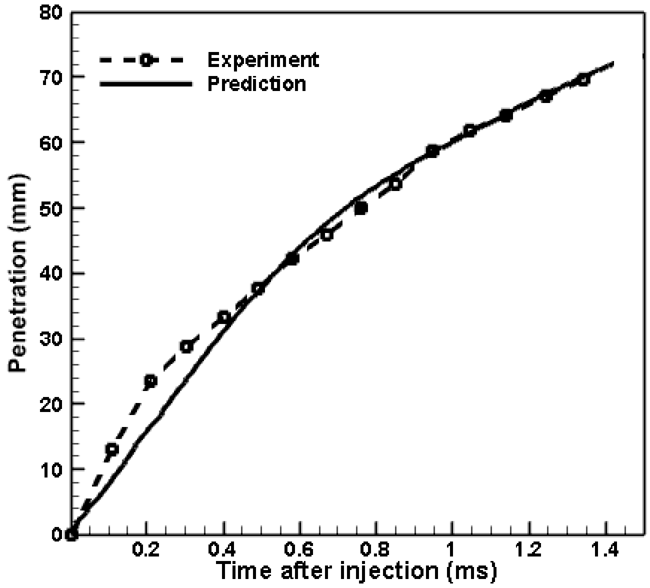

Figure 3. Comparison of experimental [10] and predicted spray tip penetration for *P*=25 bar and *T*=673 K

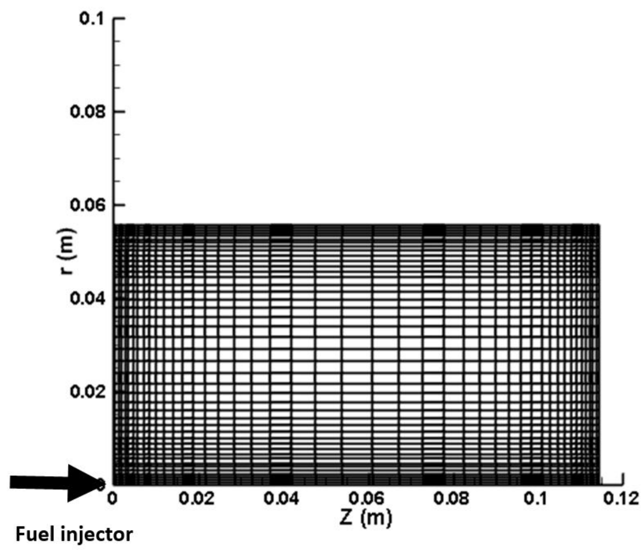

Figure 4. Computational grid

of 673 K are made as seen in Fig. 3. It is observed from Fig. 3 that during the initial stages of the injection period the penetration rates obtained from the present study are mainly under predicted. This is because of our fully atomized assumption of spray right after injection. After this stage, the spray tip penetration is slightly over predicted.

As seen in Fig.5, results of reacting liquid spray using simple step kinetic and fuel vapor penetration length are compared with experimental data. Geometry and grid configuration of this study is seen in the Fig.4. It is shown that overall characteristics of diesel spray combustion such as liquid spray penetration and fuel vapor penetration are both in good agreement with experimental data of Sandia national laboratory [11] using RANS or LES methods.

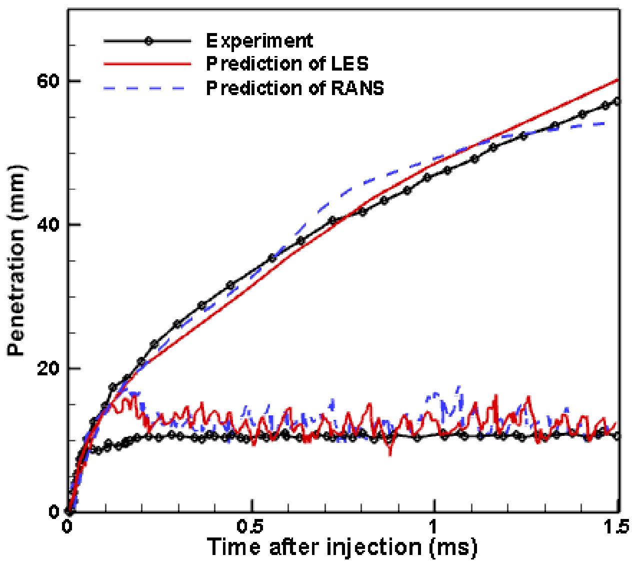

Figure 5. Comparison of reacting liquid spray and fuel vapor penetration length using different turbulence method with experimental data [11]

4- Conclusions

According to the results, LES methods of Smagorinesky and dynamic Smagorinesky based is more accurate than standard in predicting radial profile of mean axial velocity in the intake stroke. Maximum and near wall velocity is not captured by standard as well as LES method. Reacting fuel and vapor tip penetration length results of two methods are in good agreement with experimental data.

References

[1] T. Hori, T. Kuge, J. Senda, and H. Fujimoto, "Large eddy simulation of diesel spray combustion with eddy-dissipation model and CIP method by use of KIVALES," SAE Technical Paper 0148-7191, 2007.

[2] B. Hu, C. J. Rutland, and T. A. Shethaji, "A mixed-mode combustion model for large-eddy simulation of diesel engines," Combustion Science and Technology, vol. 182, pp. 1279-1320, 2010.

[3] C. Bekdemir, L. Somers, L. de Goey, J. Tillou, and C. Angelberger, "Predicting diesel combustion characteristics with large-eddy simulations including tabulated chemical kinetics," Proceedings of the Combustion Institute, vol. 34, pp. 3067-3074, 2013.

[4] M. Ameen, V. Magi, and J. Abraham, "Modeling the transient structure of reacting diesel jets using large eddy simulation," in 8th US National Combustion Meeting, University of Utah, 2013.

[5] M. Ameen and J. Abraham, "RANS and LES study of lift-off physics in reacting diesel jets," SAE Technical Paper 0148-7191, 2014.

[6] P. O'Rourke and F. Bracco, "Modeling of drop interactions in thick sprays and a comparison with experiments," Proceedings of the Institution of Mechanical Engineers, vol. 9, pp. 101-106, 1980.

[7] R. D. Reitz and R. Diwakar, "Effect of drop breakup on fuel sprays," SAE Technical Paper 0148-7191, 1986.

[8] C. Bai and A. Gosman, "Development of methodology for spray impingement simulation," SAE Technical Paper 0148-7191, 1995.

[9] A. Morse, J. Whitelaw, and M. Yianneskis, "Turbulent flow measurements by laser doppler anemometry in a motored reciprocating engine," Imperial College Mechanical Engineering Department Report FS/78/24, 1978.

[10] M. S. Yule AJ, Tham SY, Aval SM, "Diesel spray structure," ICLASS-85, London, UK, 1985 Jul

[11] Available: http://www.sandia.gov/ecn/dieselspraycombustion.php.

Please cite this article using:

M. R. Keshtkar, H. Khaleghi, S. Yazdanparast, M. Khoramdel, Numerical Study of Turbulence in Diesel Spray Combustion Using Large Eddy Simulation, *Amirkabir J. Mech. Eng.*, 50(4) (2018) 823-836.

DOI: 10.22060/mej.2017.12159.5278