CFD modelling wall heat transfer inside a combustion chamber using ANSYS forte

C. Plengsa-ard* and M. Kaewbumrung
Mechanical Engineering Department Kasetsart University, Chatuchak, Bangkok 10900, Thailand

* E-mail: fengcpp@ku.ac.th, Tel: 02-797-0999 Ext 1833

Abstract. A computational model has been performed to analyze a wall heat transfer in a single cylinder, direct injection and four-stroke diesel engine. A direct integration using detailed chemistry CHEMKIN is employed in a combustion model and the Reynolds Averaged Navier Stokes (RANS) turbulence model is used to simulate the flow in the cylinder. To obtain heat flux results, a modified classical variable-density wall heat transfer model is also performed. The model is validated using experimental data from a CUMMINS engine operated with a conventional diesel combustion. One operating engine condition is simulated. Comparisons of simulated in-cylinder pressure and heat release rates with experimental data shows that the model predicts the cylinder pressure and heat release rates reasonably well. The contour plot of instantaneous temperature are presented. Also, the contours of predicted heat flux results are shown. The magnitude of peak heat fluxes as predicted by the wall heat transfer model is in the range of the typical measure values in diesel combustion.

1. Introduction
It is known that heat transfer in diesel engines affects engine efficiency and emission. An increase of heat transfer to the combustion chamber walls is lower the in-cylinder pressure and the average gas temperature and this reduces the work per cycle transferred to the piston [1]. Thus, the magnitude of engine heat transfer is strongly dependent on engine efficiency. The changes in the gas temperature due to heat loss through the combustion chambers also have an impact on the pollutant emission formation. A higher in-cylinder temperature promotes the NOx emissions and the lower in-cylinder temperature causes the particulates during the exhaust. Since the engine heat transfer is important, the accurate predicted engine heat transfer results are needed.

There are many previous experimental and numerical studies on engine heat transfer reported in the literature. With experimental study, Hendricks [2] and Gingrich [3] showed experimental data of piston heat transfer under conventional diesel engine. Factors affecting heat transfer in a diesel engine also were investigated by Das [4]. The numerical modelling of engine wall heat transfer was researched and improved over the last few decades. Woshini [5] and Annand [6] proposed the empirical correlation to calculate the overall heat transfer coefficient. One-dimensional wall heat transfer model was introduced by Isshiki et al. [7] and Yang et al. [8]. The core regions and near wall layers were separated within these one-dimensional models. Heat flux results was predicted by solving an energy equation without using a heat transfer coefficient, while the core region was considered a global region with uniform properties without spatial distinction. However, in-cylinder core regions for internal combustion engine cannot be considered globally since there are local changes in the core regions due to turbulence,
swirling and tumbling. Also, in order to obtain an accurate values of heat flux, multi-dimensional model for core regions and one-dimensional heat transfer near walls are necessary. In these multi-dimensional models, conservation of mass, momentum and energy were directly solved in the core region, whereas the wall models were generally used within the wall layers. Since the mechanism of wall layers are similar to the physics in the boundary layers, the predicted heat fluxes were solved by using thermal boundary layer assumptions. Han et al. [9] derived the wall heat transfer model from the one-dimensional energy equation which included the variation of gas density and turbulent Prandtl number across the boundary layer. These models provided a better predicted heat flux results when compared with those of previous works.

Therefore, the purpose of this present work is to demonstrate how engine wall heat transfer model in the commercial code i.e., ANSYS forte predicts the heat flux results. The test case operates in diesel engine condition. The data from experiment is compared with those of heat flux prediction. The heat transfer in this study is based on the model work from Han et al. [9]. Also, the heat flux results through the combustion chamber walls due to gas phase heat convection is the major concern.

2. Wall heat transfer formulation

For ANSYS- Forte, the wall heat transfer model from Han et al. [9] is used to calculate the gas-phase heat transfer near wall regions. The analysis of the formulations of this model is initiated from the one-dimensional energy equation across the boundary layer which is described below in equation (1).

\[ c_p\rho \frac{\partial T}{\partial t} + c_p\rho \nu \frac{\partial T}{\partial y} = \frac{\partial}{\partial y} \left[ (k + k_t) \frac{\partial T}{\partial y} \right] + \frac{dP}{dt} + \rho Q \]  

\[ \text{(1)} \]

where \( \rho \) and \( T \) are the fluid density and temperature near walls respectively, \( c_p \) is the specific heat at constant pressure, \( p \) is pressure, \( \nu \) is fluid velocity, \( k \) is laminar thermal conductivity, \( k_t \) is turbulent thermal conductivity and \( Q \) is volumetric heat release rate. Several assumptions are introduced in the derivation of the model, including the following: (1) fluid velocity near walls is parallel to the wall, and only the temperature and density gradients normal to the wall are considered, (2) the gas is assumed to be ideal, (3) the pressure is uniform in space, (4) viscous dissipation is neglected, and (5) radiation heat transfer is not included. Based on these previous assumptions, the wall heat flux in the near wall regions is computed by equation (2).

\[ q_w = \frac{\rho u \cdot c_p T \ln \left( \frac{T}{T_w} \right)}{\left\{ 2.1 \cdot \ln \left( y^+ \right) + 2.5 \right\}} \]

\[ \text{(2)} \]

\[ y^+ = \frac{u^* y}{\nu} \]

\[ \text{(3)} \]

\[ u^* = \left( \frac{c_p}{\mu} \right)^{0.25} \cdot \kappa \cdot e^{0.5} \]

\[ \text{(4)} \]

where \( q_w \) is heat flux on the wall, \( u^* \) is friction velocity, \( y^+ \) is the dimensionless distance, \( T_w \) is the wall temperature, \( \nu \) is the kinematic viscosity, \( \kappa \) is turbulent kinetic energy and \( C_p \) is equal to 0.09.

3. Turbulence and Combustion models

In order to simulate in-cylinder flows, turbulence is modeled using the RNG k-ε turbulence model in ANSYS forte commercial code. These consider velocity dilation in the \( \varepsilon \)-equation and spray induced source terms for both k and \( \varepsilon \) equations. For the combustion model, ANSYS forte uses a detailed chemistry CHEMKIN PRO solver. A skeletal reaction mechanism for n-heptane fuel, which has similar ignition characteristics with diesel fuel, is used to simulate diesel fuel chemistry. The mechanism
includes 36 species and 76 reactions. However, the physical properties of the fuel in this work are represented with those of tetra-decane in the calculations.

4. Validation setup

4.1 Engine geometry and operating condition

The measurement data from a single cylinder direct injection 4 stroke diesel engine by Cummins N-series with 2.34L displacement is employed to validate computational results in this study.[10] A summary of the main geometrical data is listed in table 1, while a schematic diagram of the engine is presented in figure 1. Also, the engine condition for the test case is summarized in table 2.

| Table 1. Cummins N-series specification.[10] |
|---------------------------------------------|
| Geometry data                               |
| Engine type       | DI diesel         |
| Displacement (L)  | 2.34              |
| Bore (cm)         | 13.97             |
| Stroke (cm)       | 15.24             |
| Swirl ratio       | 0.5               |
| Connecting rod length (cm) | 30.48        |
| Number of cylinders | 1                  |
| Fuel injector type | Common rail      |
| Number of nozzles | 8, equally space |

![Figure 1. A schematic diagram of Cummins N-series engine [10]](image-url)
Table 2. The operating engine conditions [10]

| Engine speed (rpm) | 1200 |
|--------------------|------|
| IMEP (bar)         | 4.4  |
| Intake pressure (kPa) | 233 |
| Intake temperature (K) | 384 |
| SOI (°ATDC)        | -7   |
| DI duration (CA°)  | 10   |

4.2 Numerical approach

In this paper, sector symmetry is assumed and periodic boundary conditions were applied for each injector hole. A 45 degree sector mesh was used to represent one-eighth of the engine combustion chamber. There were 60 cells in the radial direction, 42 cells in the azimuthal direction and 62 cells in the axial direction, with 6 cells in the squish region at the top dead center. This mesh resolution, which can be seen in figure 2, has been found to give adequately grid-independent results. Figure 3 illustrates the variation of simulated in-cylinder pressure on the test engine condition for the three different mesh sizes. It can be seen that there is no significant difference between grid II and grid III. To save computational time, grid II is chosen for the simulation in this research. Moreover, the computational domain consisted of one section of the modeled engine cylinder which is used for simulations between inlet valve close (IVC) and exhaust valve open (EVO). Therefore, the model computes only the closed volume part of the engine cycle. The simulations are started at the intake valve closure (IVC) with uniform in-cylinder mixture distribution assumptions. The swirl is initialized based on the wheel flow velocity profile and the turbulent kinetic energy is initialized at IVC by scaling with the mean piston speed. The pressure and temperature at IVC are given based on the experimental data. The boundary conditions of turbulent kinetic energy are the Neumann boundary conditions. The averaged wall temperature was 460 K for the test case.

Figure 2. CFD 45 degree sector mesh of a simulated engine.
5. Results and discussion

For model validation, comparison between computed and measured in-cylinder pressure and heat release rate as predicted by ANSYS forte are shown in figure 4. The start of injection is 7 degrees before the top dead center. Good agreement with the magnitude of peak in-cylinder pressure can be seen in the figure, while the over prediction of the cool flame location occurs. The trend of heat release curve matches with the conventional diesel combustion. The predicted peak heat release rate during the premixed burn occurs slightly before the top dead center and the overestimation of calculated heat release rate can be seen during the mixing control combustion regime.

![Figure 3. Variation of simulated in-cylinder pressure with crank angles for the test case with three different mesh sizes](image)

![Figure 4. Comparisons of computed and measured in-cylinder pressure and heat release rate with crank angles. The start of injection is 7 degrees before TDC](image)
Figure 5. Comparisons of predicted local heat flux with crank angles near a piston head

Figure 6. Comparisons of predicted local heat flux with crank angles near a piston bowl

Figures 5 and 6 illustrate the comparisons of local heat flux prediction near a piston head and a piston bowl respectively. Both pictures provide the spatial effects along the primary spray axis. The magnitude of predicted heat flux results is varied with crank angles and is small during the compression and strokes. In figure 6, the location of point 4 near a piston bowl shows the lowest values of predicted heat flux because the diesel spray actually targets beyond this point. At point 1, the highest values of predicted heat flux occurs near a piston head close to an injector due to the high temperature from the flame during the combustion. At the locations of point 5 and 6, the magnitude of predicted heat flux is higher when compared with other locations. The distributions of heat transfer rate for three area of the combustion chamber are presented in figure 7. As can be seen, most of heat transfer loss to combustion chamber walls are from the piston bowl. This is true because the flame fronts will reach the area of
piston during the combustion events. Figures 8 and 9 show the contour plots of calculated in-cylinder temperature and heat flux results at 5 degrees after the top dead center.

**Figure 7.** Distributions of heat transfer rate for the three area of the combustion chamber

**Figure 8.** Contours of predicted peak in-cylinder temperature at 5 degrees after TDC
The maximum temperature is about 2300 K which is in a range of typical conventional diesel engine. Moreover, the non-uniform of temperature and surface heat flux is presented. The location of locally maximum and minimum values of temperature and heat flux results on the piston bowl can be captured by the wall model. The contour plots of wall heat flux information are consistent with the local heat flux results in figure 6.

![Heat Flux Contours](image)

**Figure 9. Contours of calculated heat fluxes at 5 degrees after TDC (top view)**

6. Conclusions
The wall heat transfer models from Han et al.[10] is tested. The model is validated using experimental data from a CUMMINs engine operated with a conventional diesel combustion. The operating engine condition is tested at 5 degrees before TDC. Comparisons of simulated in-cylinder pressure and heat release rates with experimental data shows that the model predicts the cylinder pressure and heat release rates reasonably well. The model can capture the location of the cool flame and main heat release. The contour plot of predicted temperature and computed heat flux results are presented. Comparisons of predicted heat flux near a piston head and a piston bowl are shown. The magnitude of peak heat fluxes as predicted by the wall heat transfer model is in the range of the typical measure values in diesel combustion.

Acknowledgements
This material is based upon work supported by Faculty of Engineering Kasetsart University. In addition, the authors are grateful for the intellectual support from Mr. Mongkol Kaewbumrung, a PhD. candidate from Applied Combustion Lab, Mechanical Engineering Kasetsart University.

References
[1] Jia M, Gingrich E, Wang H, et al. 2016 Int J Engine Res 17(3) pp 331-346
[2] Hendricks T, Splitter D, et al. 2014 Int J Engine Res 15(6) pp 684-705
[3] Gingrich E, Ghandhi J B, Reitz R D 2014 SAE Technical Paper 2014-01-1182
[4] Das S and Roberts CE 2013 SAE Technical Paper 2013-01-0875
[5] Woschni G 1967 SAE Technical Paper 670931
[6] Annand W J D 1963 Proc. of the Institution of Mechanical Engineers vol. 177 pp 973-996
[7] Ishihiki, N and Nishiwaki N 1970 Proc. of the 4th Int. Heat Transfer Conference, FC3.5 pp 1-10
[8] Yang J, Pierce P, Martin, J, et al. 1988 SAE Technical Paper 881314
[9] Han Z and Reitz R D 1997 Int. J. Heat and Mass transfer 40(3) pp 613-625
[10] Singh S, Reitz R D and Musculus M P B 2006 SAE Technical Paper 2006-01-0055