13 The Simulations of Diesel Engine Spray Combustion and Two Phase Flow Field

Leonard Kuo-Liang Shih and Chung-Yi Lee
Department of Mechanical Engineering
National Yunlin University of Science and Technology
Taiwan, R.O.C.

Abstract

This research applied the KIVA-3V (Release 2) 3-Dimentional Reactive Flow Code developed by USA Los Alamos National Laboratory on the diesel engine combustion phenomena. The preprocessor (K3prep) constructs the grids on the intake manifold, the cylinder chamber, the bowl on piston surface, and the exhaust manifold. The experimental measured engine geometry data and engine operating conditions are then converted into the input file (itape 5).

The baseline case is set on 1800 rpm (engine speed) and equivalence ratio Ø=0.25 (engine load). The engine operating conditions are then changed to higher engine speeds and engine loads. The simulated cylinder pressures and heat release rates are compared to the experimental results. The dynamic output (0~720 CA) is post processed by K3POST Code and the Tecplot software. The flow streak line, the pressure filed, and the temperature distributions inside the cylinder chamber are first studied. Then the various gas concentration (NOx, HC, CO) and the turbulent kinetics distribution are then displayed in sequential crank angles.

The KIVA-3V results show the acceptable similarity of the cylinder pressures and heat release rates between simulation and experimental works. The chemical reactions and flow field inside the cylinder are demonstrated in 3-D continuous graphics. This study gives us powerful insight into the non-measurable phenomena inside the diesel engine.

Introduction

The fast development on the computer calculation capability and accuracy has made the numerical simulation technology a powerful tool on the modern engine design. In stead of using real engine test on engine design process, the simulation tool can shorten the design period.

There are basically three types of computer simulation program existing. The first type is the Thermodynamics model that analyzes the gaseous components and dynamics inside the combustion chamber. The second type is the Phenomenological model that deals with specific phenomena inside the combustion chamber. The third type is the multi-dimensional model that calculates the point-by-point detail data within the combustion chamber. The KIVA-3V is the third type code. The following sections will describe this model.

On 1995 Han and Reitz [1] applied the RNG k-ε turbulence model instead of the standard k-ε model on the diesel engine simulation. They found the RNG k-ε model has larger turbulence strength predicted which is relatively similar to the experimental results.

On the year 1998, the KIVA-3 program was released. Kong [2], et. al., modified the valve lift movement boundary conditions to simulate the intake-exhaust gasoline engine flow field inside cylinder and manifolds. Their results show appreciable cylinder pressure, average velocity and turbulence compared with the experiments.

On the year 2003, Wang, et. al. [3], applied the KIVA-3V code for the direct injection gasoline engine. They studied the stratification effect in the cylinder under various injected fuels. They found that if the tumbling ratio and the diffusion speed are increased the air-fuel mixing and combustion efficiency can be promoted. In the same time, Zhang, et. al [4], simulated the duel fuel (gas and diesel) combustion characteristics inside the diesel engine. The results show that the amount of diesel fuel injected, the injection timing, and the nozzle spray angle can all affect the engine performance and exhaust emissions.

Thus, this research will apply the newly released KIVA-3V (version 2) code on the simulation of engine inside flow field and combustion phenomena. The engine operating parameter will be modified and compares the simulation results to the experimental measurement to verify the accuracy of this code.

MODEL DESCRIPTION

Since 1970s the multi-dimensional concepts have been applied in the engine simulation field. The models include the mass- momentum, and energy conservation and the other transient gaseous combustion, cemical reaction, heat transfer and spray dynamics, etc. Both Eulerian and Lagrangian concepts are included in the program philosophy. The gaseous governing equations deal with the continuity, the momentum, the energy, the composition, the turbulence, and the chemical reaction phenomena. The liquid governing equations deal with the droplet movement, the spray dynamics, and the two phases flow phenomena. The other governing equations also deal with the droplet breakup–evaporation -coalescence, the shell ignition, the emissions formation, and the droplets –cylinder charge interactions.

System Hypothesis

The fundamental hypotheses in KIVA-3V include,

(1) The liquid spray is considered as Newtonian fluid.

(2) The droplets are spherically symmetric and suitable to the KH-RT droplet breakup model.

(3) The temperature inside the droplet is homogenous

(4) The gaseous fluid is ideal gas.

(5) Diffusion Fick's law is applicable in molecular level.

(6) The cylinder charge match the "1/3 rule".

COMPUTATIONAL METHODOLOGY

This research applied the KIVA-3V program to simulate the flow field and combustion phenomena inside the combustion chamber of the YAMAHA ME-200F diesel engine. The simulation methodology has been introduced earlier in this paper. The program structure and the grid generation is discussed here.

Program Structure

KIVA-3V is written by the USA Los Alamos National Lab. The program can be cut down into three major parts: the Pre-processor, the main code, and the post processor. The flow chart of this program is illustrated in Figure 1.

The pre-processor (K3PREP) includes the grid generation and the boundary conditions set up. The main program (KIVA-3V) completes all the models described earlier in this paper. The post-processor (K3POST) transfer the result into image output. However, this research will use the Tecplot software for the output organization.

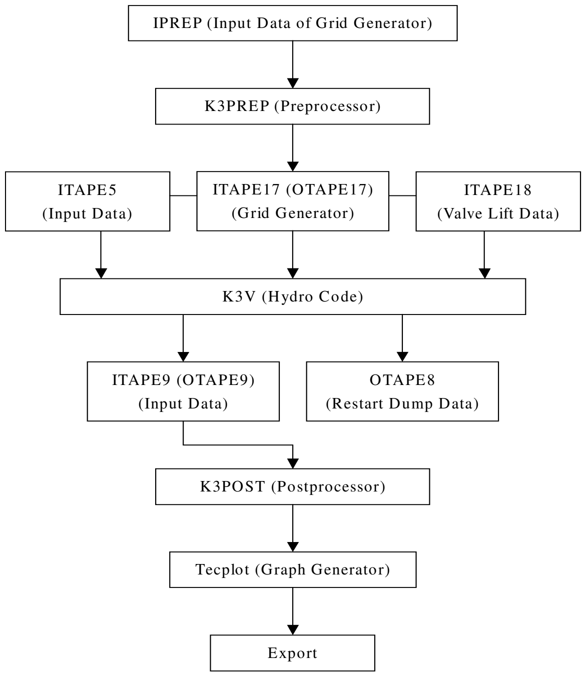


Figure 1 KIVA-3Vprogram flow chart.

Grid Generation

The intake manifold, the combustion chamber, and the exhaust manifold are divided into many small six-face cells. The "reshape" subroutine in the program line these small cells as a "zone" and the "patch" subroutine line all zones to become the complete shape of the simulation domain. The intake and exhaust valves geometry, the bowl shape of the piston surface, the cylinder head shape can all be accurately re-generated in this program. Figures 2 illustrate the 2-D and 3-D grid structure.

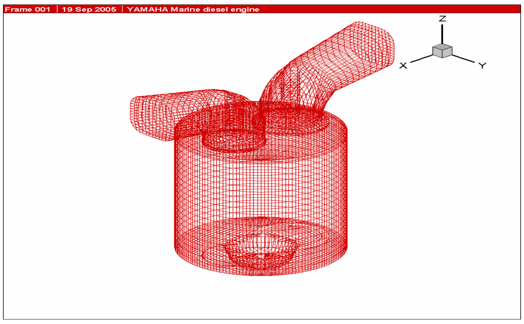


Figure 2 KIVA-3V 3-D grid.

RESULTS AND DISCUSSION

The marine inboard diesel engine used in this study is specified in Table 1. This is a three cylinder direct injection diesel engine without turbocharger.

Engine Type	3-cylinder, four strokes, DI
Combustion Chamber Shape	Round dish squish at TDC
Bore - Stroke	91.5×100 mm
Engine Displacement	1973 cc
Compression Ratio	18.2：1
Connecting Rod Length	168 mm
Start of Injection	24°BTDC
End of Injection	11°ATDC
Intake Valve Open/Close	16°BTDC, 52°ABDC
Exhaust Valve Open/Close	56°BBDC, 12°ATDC

Table 1 The engine specifications of YAMAHA ME200F diesel engine.

The baseline case of this study is set at engine speed of 1800 rpm and engine load of φ=0.25. Instead of using engine output percentage as the engine load, the fuel air equivalence ratio is applied in this study. This is because the equivalence ratio can "accurately" reveal the total fuel energy injected into the cylinder. Thus, the simulated output can be precisely explained on the base of energy conservation.

In order to compare the simulation results to the experimental measurements, the engine speed and the engine load are selected as listed in Table 2. The cylinder charge pressure, the heat release profile, and the cylinder charge temperature distribution are first revealed in the following discussion. Secondly, the TKE, the streak line, and the droplets distribution in the combustion chamber are then shown in this paper. Finally the combustion emissions (HC, CO, NO_x) distributions in the combustion chamber are discussed.

	Low	Mid	High
Engine Speed, Load	(1800rpm ，φ=0.25)	(2300rpm ，φ=0.35)	(2800rpm ，φ=0.45)
Total Fuel Injected (g)	0.011504	0.015696	0.019829

Table 2 Engine operating conditions.

Comparisons Between Simulation and Experimental Cylinder Pressures & Heat Release Rates

Figure 3 shows that the simulation pressure is higher than the experimental data. Basically the KIVA-3V code combines many phenomenological models and can be affected by the initial model constants severely. The peak pressure occurs at 364 CA ATDC that is consistent with the experimental result. The heat release rate calculation is through a standard model [5].

Figure 4 illustrates the heat release pattern calculated from the experimental and simulation cylinder pressure. The simulated data shows lower peak and earlier heat release at 342 CA ATDC compares to 350 CA ATDC for the experimental case.

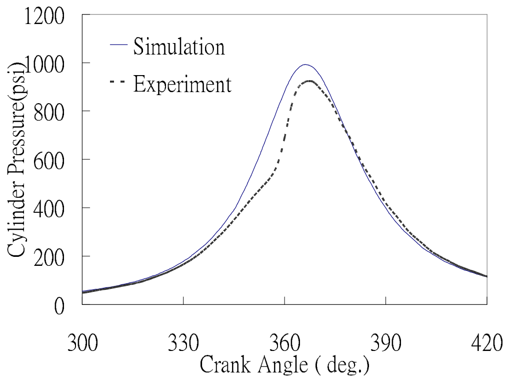


Figure 3 Comparison of cylinder pressures from simulation and experimental results.

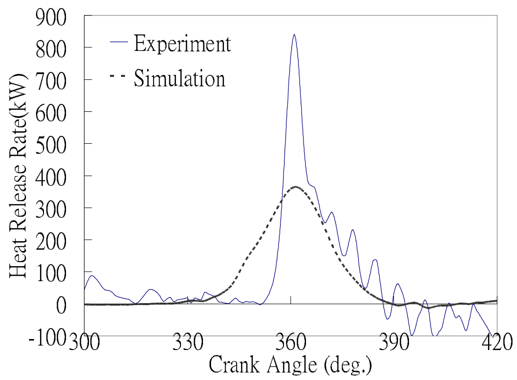
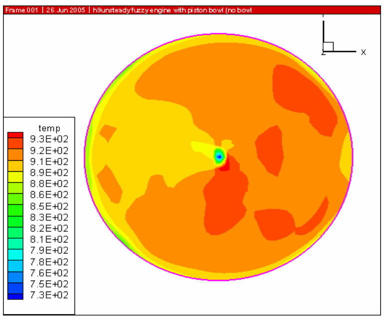


Figure 4 Comparisons Between Simulation and Experimental Heat Release Rates

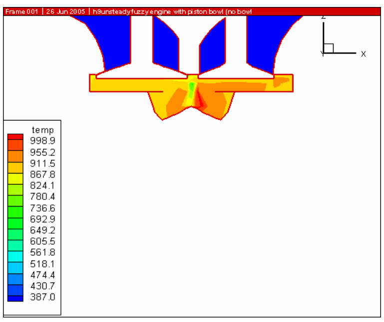
Cylinder Charge Temperature Distributions In Cylinder

In this section the cylinder charge temperature is illustrated at various crank angle (340 and 370). The X-Y plane is defined at the middle of the engine squish area.

The fuel is injected into the combustion chamber at 336 CA BTDC; thus, at 340 CA BTDC the center of Figure 5 shows a small area of low temperature. This is because the evaporation process of liquid fuel actually cool down the charge temperature. The left portion of Figure 5a has lower gas temperature.



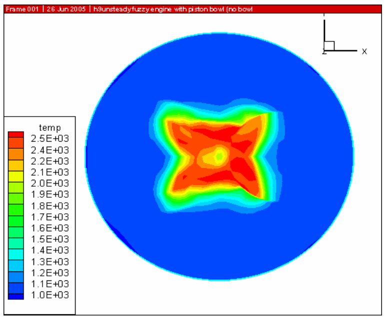
(a) X-Y Plane



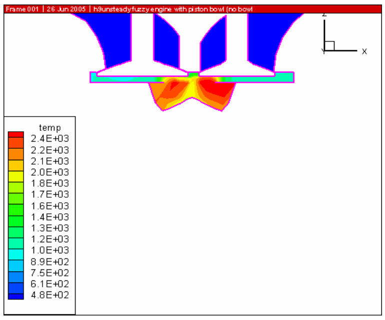
(b) X-Z Plane

Figure 5 Temperature distributions at 340 CA BTDC.

Figure 6 shows the continuous flame fulfill the whole piston bowl and part of the combustion chamber squish area. After the end of fuel injection (at 371 CA ATDC) the flame temperature has reaches 2050 ^{0}K at 390 CA ATDC.



(a) X-Y Plane



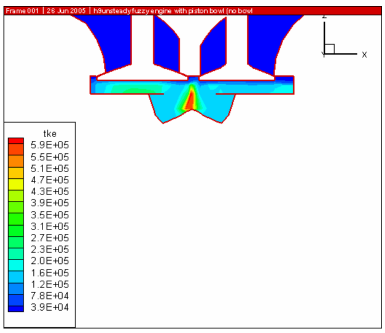
(b) X-Z Plane

Figure 6 Temperature distributions at 368 CA ATDC.

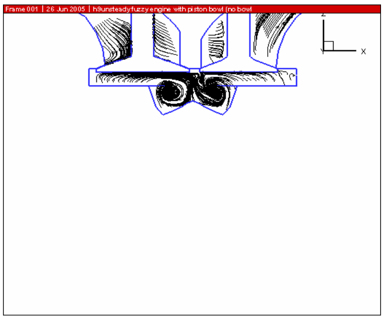
TKE Distribution and Streakline

The turbulent energy distribution (TKE) and streakline shown in this section provide the essential information of how the fuel injection interacts with the cylinder charge. The following figures show the TKE and streakline.

The fuel starts to inject at 336 CA BTDC so that the TKE shown in Figure 7a clearly indicates the strengthened jet flow turbulence near the piston bowl center. Figure 7b also demonstrates the two opposite tumbling gas motions inside the piston bowl due to fuel injection.



(a) TKE

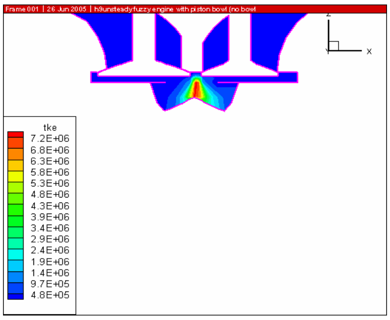


(b) Streakline

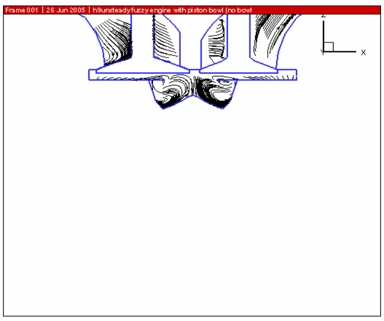
Figure 7 TKE and streakline at 340 CA BTDC.

Since the piston at the end of compression stroke has very small squish area between piston surface and cylinder head; thus, the TKE distribution show in the following Figure 8a has very small turbulent strength in these figures.

The streaklines; however, are still shown in the squish area since the flow is not “fixed” inside the squish region. The same situation occurs in the intake and exhaust manifolds.



(a) TKE



(b) Streakline

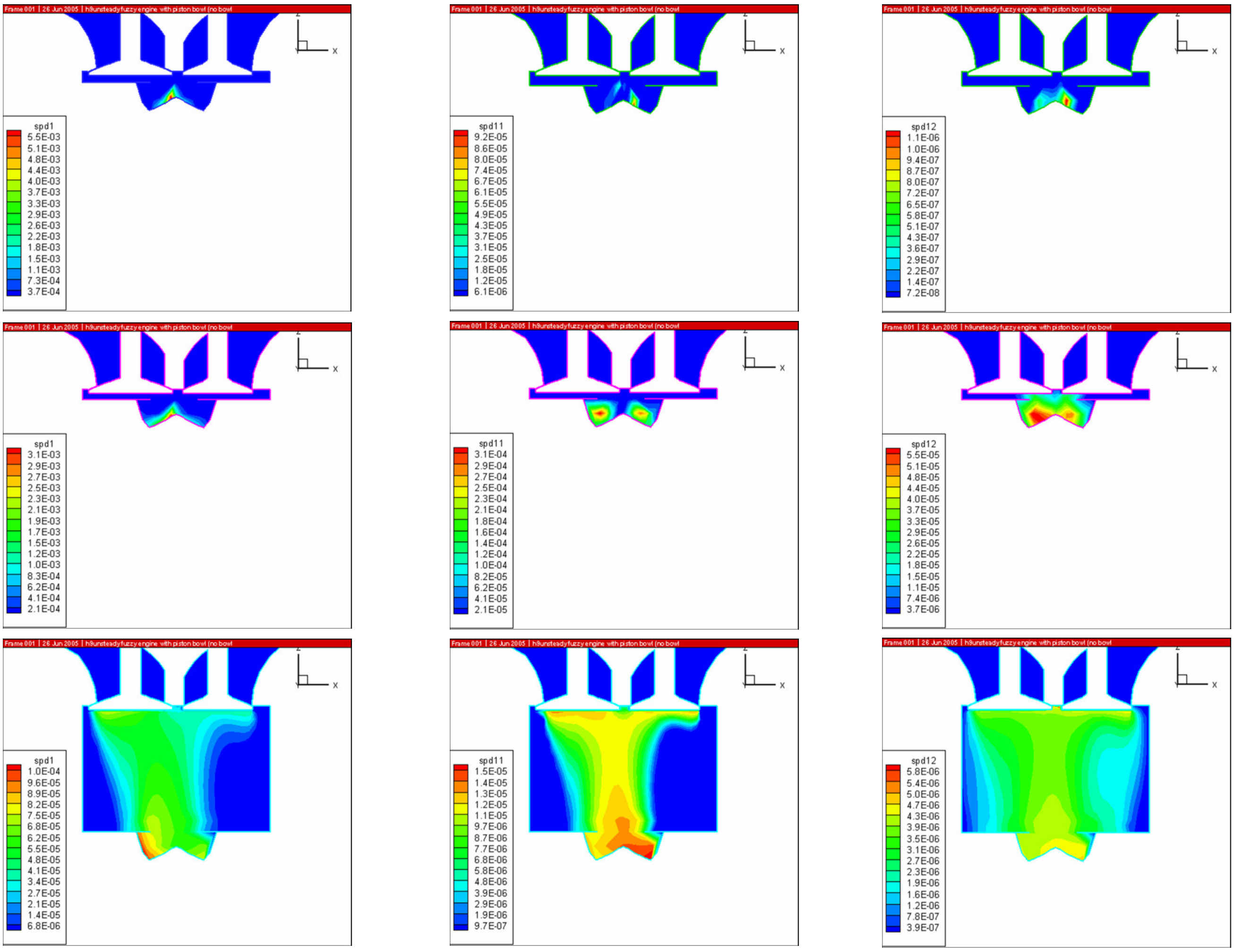
Figure 8 TKE and streakline at 368 CA ATDC.

Distributions of HC, CO, and NO_X Inside the Combustion Chamber

The hydrocarbon distribution in the combustion chamber can reveal the characteristics of ignition and combustion behavior after the onset of chemical reaction. The first column figures shown in Figures 9a to 9c demonstrate the history of HC concentration distribution. At the start of injection the HC vapor is concentrated near the injector nozzle (Figure 9a). There is some residual fuel vapor near the fuel injection impingement site inside the piston bowl. Since the impinged fuel jet makes the piston surface to form a liquid film so that the HC concentration is relatively condensed near the bowl region (Figure 9b). After the piston moves downward (Figure 9c) the unburned HC/cylinder charge mixture remains in the core region in the combustion chamber. Also there is very few HC vapor around the cylinder liner.

Another gas concentration that is used to explain the combustion in the engine is the CO emission. Since the chemical reaction takes place after 344 CA BTDC (Figure 9d), the CO emission occurs most probably at the region outside the injection path where the condensed HC vapor causes insufficient oxygen to complete the oxidation reaction (Figure 9e). Besides, inside the injector path the extremely condensed HC vapor basically allow very minor chemical reaction to take place (Figure 9f). This is why when the first and second columns in Figure 9 are compared, the position of CO emission region seems to be just outside the HC emission region. After the injection cause “round up” HC vapor cloud inside the piston bowl, Figure 9e indicates the insufficient of oxygen occurs near the bowl’s two symmetric “centers” where the HC vapor can not be sufficiently mixed with fresh air.

The third column pictures in Figure 9 show the NOx concentration in the cylinder. NOx is formed under high temperature environment and takes time to form. The high NOx region implies there is sufficient fuel and air in this region and the turbulence strength is strong enough to mix these two elements within combustible range. Figure 9g shows there is still no chemical reaction takes place. After the spray impingement inside the piston bowl causes mixing to take place and vigorous combustion to take place; thus, Figure 9h shows the high chemical reaction rate (NOx formation) near the bowl region. After the piston start the expansion motion (Figure 9i), the fast declined charge temperature stops the NOx formation.



HC Concentration (a)~(c) CO Concentration (d)~(f) NOx Concentration (g)~(i)

Figure 9 Y-Z Plane HC, CO, and NOx distributions inside the combustion chamber.

CONCLUSIONS

This research applies the KIVA-3V code to simulate the spray dynamics and combustion phenomena. The following conclusions are summarized.

1. The chemical reaction model is not capable to reveal the heat release profile of a diesel engine.

2. The charge temperature, the TKE distributions and the streak line in the cylinder shows all minor flow characteristics in the engine which is very useful for engine geometry design.

3. The history of HC, CO, and NOx concentrations distribution inside the combustion chamber makes the possibility of knowing the detailed combustion processes.

REFERENCE

[1] Z. Han and R. D. Reitz, "Turbulence Modeling of Internal Combustion Engines Using RNG k-ε Models," Combustion. Science and Technology, 1995, Vol. 106, pp.267-295.

[2] Kong, Song-Charn., et. al., "Three - Dimensional Computations of the Overall Flow Process In a Four -Stroke Engine," The 8th Combustion Technology and Application Symposium, Taiwan, pp. 333~338, 1998。

[3] Dongyao Wang, Chia-fon Lee, and Yangbing Zeng, "Modeling of Air Fuel Mixing in a stratified Gasoline Direct injection Engine using Multicomponent Fuel Representation" SAE Paper,NO.2003-01-0067,2003

[4] Youtong Zhang, Song-Charng Kong, Rolf D.Reitz, "Modeling and Simulation of a Dual Fuel (Diesel/Natural Gas) Engine with Multidimensional CFD" SAE Paper, NO.2003-01-0755, 2003

[5] Leonard Kuo-Liang Shih and Tien-Chiu Hsu., 2004, " Predictions of Diesel Engine's Performance by the DESP Program with the Combustion Chamber Coated with Metals or Ceramics" SAE Paper No. 2004-01-2697.