ATLANTIS PRESS
Advances in Engineering Research, volume 104
7th International Conference on Mechatronics, Control and Materials (ICMCM 2016)

The Similarity Theory of the Whole Flow Field Inside the Cylinder Piston Engine Simulation Method a Preliminary Study

Guangxu Ren[1,a], Xiujuan Liu[2,a], Chunguang Wang[3,a]
Aviation University of Air Force, Changchun, China
723704537@qq.com

Keywords: Piston engine; the whole flow field; simulation; model

Abstract. This paper illustrates the basic characteristics of the fluid flow inside the piston engine, the condition of clear piston engine full flow field research premise, puts forward a available for study of the turbulent flow field model, and based on this model, this paper expounds the basic methods of the whole flow field simulation.

Introduction

Piston engine cylinder of fluid movement, has a great influence on the performance of the engine. Overall movement within the cylinder and the turbulent flow, influence the cylinder filling amount of layered, combustion, heat transfer and clearing process. In order to predict filling quantity stratification, must understand and be able to predict the turbulent mixing process. Wall boundary layer heat transfer and flow control of quenching, turbulent structure and turbulence dimension of knowledge is the precondition of combustion analysis foundation.

Pollutants in different regions produce different, the analysis of the exhaust emissions need to be able to predict what fluid when the exhaust unit will be cleared out, what units will remain within the cylinder. Therefore, the comprehensive prediction need a great deal about the overall flow and turbulent flow in the cylinder and the knowledge of the boundary layer.

The Basic Characteristics Of Multiphase Flow In Piston Engine. Important in typical piston engine cycle flow characteristic is based on the measurement of starting working state, to work in the engine cylinder of photography, and for other flow with similar characteristics on the basis of the character of analogy.

In the admission process, through the inlet valve will flow from the valve seat and valve top flange apparent shear layer. The shear layer on the dynamics is unstable, for a lot of circular vortex broken at first, and then merging for the large scale vortex. The large scale vortex itself is broken into three dimensional turbulent flow movement. The result was a cone of turbulent jet containing scope of great turbulence scale, from the order of magnitude with jet thickness large eddy simulation to the small scale eddy dissipation of turbulent movement. The flow of inlet flow will cause a general circulation in the cylinder, it could also superposition produced by admission line into the cyclone flow. Shear layer from the top of valve and valve seat surface separation lead to produce recirculation region, and the downward motion of the piston in the cylinder wall to produce a boundary layer. The large scale circulation may cause smaller reflux in each corner. This type of back flow generally is sensitive to tiny changes in flow. So return the position and size may have a lot of cyclic variations.

If the engine has a separate inlet precombustion chamber, it would be more complicated. In a precombustion chamber flow will have a lot of the above characteristics, there will be also from the precombustion chamber, through the connection channel to the main flow of the combustion chamber. Of one or a few jet - step will be to complicate the movement within the cylinder.

When the inlet valve closing, the apparent shear layer disappears, but there was a turbulent flow produced by them. Large scale circulation driving small scale turbulent flow, which spread in the large-scale movement through the cylinder before exists in local area. The structure of the turbulent flow will change in the compression process. If there is no viscous effect, fluid particle will maintain its angular momentum is constant, so the compression stroke tends to increase the angular velocity of the gas. Small scale turbulent flow of these changes is caused by the piston compression and general circulation in the cylinder deformation produced by both. Near the top dead center, flow may be complicated by fluid jet. In a direct injection engine, the overall circulation may be increased or reduced because of fuel injection, specific see the position and direction of the nozzle, and small scale turbulent flow formed by injection process.

The ignition and combustion greatly depends on the flow structure, especially in layered filling volume of the piston engine. In this phase - particularly important is the small scale structure of turbulence, it influence the spread of flames. A dramatic increase in the temperature of combustion caused a sudden increase of fluid viscosity, therefore, in order to make the gas FenZi mixing, shall be appropriately increased turbulence decay rate, increasing the scale of the movement. In precombustion chamber type diesel engine, the reaction gases from precombustion chamber flow will change of turbulence structure in primary combustion chamber, and then in the main chamber combustion could lead to a reversal to the precombustion chamber.

During the period of expansion stroke, flow and subject to change due to deformation and continue to burn again, in the cylinder wall to form a boundary layer. Before the remnants of the process of the large scale circulation, affirmation will exist during this period. The overall properties may vary greatly depending on engine design, even in a given engine cycle also produces the process of change.

Fluid in the process of exhaust must be from the cylinder wall, it will form a large at the center of the vortex. During the period of combustion from quenching layer is formed on the surface, some areas in the exhaust process is expelled. These factors determine the fluid distribution, both inside and outside the cylinder at the end of the exhaust process.

In a word, the flow piston engine is composed of turbulent shear layer, boundary layer and recirculation region of complex system, this system is extremely unstable, there may be a lot of circulating, raise the temperature of burning make the fluid transport properties has great changes. Whether it is a large scale or small scale turbulent motion, is an important factor in the whole process adjustment. Accurate and detailed simulation of the flow of it is very important to study the combustion process.

The Whole Flow Field in the Piston Engine Simulation Method. The flow field in the piston engine simulation methods have partition simulation, the whole flow field simulation and large eddy simulation. In the whole flow field simulation, the description with partial differential equation has been average amount, and everywhere in the flow in the same equation. In the unsteady incompressible flow, using the time average. Variables including velocity field, at least average turbulence may also include a variety of parameters, such as turbulent kinetic energy and turbulent stress tensor. For compressible flow and other thermal state variables must also be calculated. For the periodic flow piston engine, average time must plug in, to ensure that the average.

In the whole flow field simulation of turbulent flow quantity of each average need each model. These models must reflect all characteristics of turbulent motion scale. Rho flow control equation of statistical nature from dominating density, velocity, the unit mass of internal energy e UI and the concentration of unit mass of ya basic equation is derived. These equations are written in the briefly way to:

$$\frac{D}{Dt}\begin{pmatrix}1\\ u_i\\ e\\ Ya\end{pmatrix}=\begin{pmatrix}0\\ 0\\ Q\\ Sa\end{pmatrix}-\frac{\partial}{\partial x_j}\begin{pmatrix}0\\ \tau_{ij}\\ q_j\\ Jaj\end{pmatrix} \tag{1}$$

$$\frac{Df}{Dt}\equiv\frac{\partial(\rho f)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j f) \tag{2}$$

Type D/Dt is quality convection operator.

Thermal energy Q comprises a viscous term and caused by chemical reaction source term. Regardless of Q or material source Sa will depend on the chemical reaction rate equation, the latter must be known to solve the problem. Thermodynamic state equation is needed. In addition, the structure of diffusion flux equation must be specified.

Studies have shown that in addition to the diffusion of heat conduction, the spread of various substances may also constitute a part of the spread of the internal energy flow qj.

In the whole flow field simulation, according to the equation (1) the average equation variable. The development of the most turbulent model is conducted using the time average.

The analysis of flow in engine cylinder, need to consider compressibility. One way to from beginning to end using simple average. Another is the quality of the weighted average, namely FuLei average, it makes the compressible flow equations of average looks completely same as the average equation of incompressible flow. Such, FuLei average equation without a new model must add additional items. So that it is successful for incompressible flow model can be extended to directly method FuLei compressible flow equations. Of course this promotion is pure mathematics; To confirm the promotion from the physical need through experimental verification.

Composite phase - FuLei average is the basis of analyzing the engine flow, so a little more detail here. We will use curly braces said that the average process:

$$\{\rho(x,t)\} = \lim_{N\to\infty} \frac{1}{N} \sum_{n=1}^{n} \rho(x, t+n\tau) \tag{3}$$

τ is the loop cycle. We also represented as { ρ }= $\tilde{\rho}$. Then the ρ is decomposed into $\rho = \tilde{\rho} + \rho'$. With pointed bracket said the quality of the weighted average process, with the top quality of the weighted average, short stroke said that

$$\tilde{\rho}(\underline{x},t) < f(\underline{x},t) >= \lim_{N\to\infty}[\frac{1}{N}\sum_{n=1}^{N}\rho(\underline{x},t+n\tau)\overline{f}(\underline{x},t+n\tau)] = \tilde{\rho}\overline{f} \tag{4}$$

We put all flow variables are decomposed into (besides density and pressure). Average of equation (1), available:

$$\frac{\bar{D}}{Dt}\begin{pmatrix}1\\ \bar{u}_i\\ \bar{e}\\ \bar{Y}a\end{pmatrix} = \begin{pmatrix}0\\ 0\\ \{\bar{Q}\}\\ \{Sa\}\end{pmatrix} - \frac{\partial}{\partial x_j}\begin{pmatrix}0\\ \{\tau i_j\}\\ \{q_j\}\\ \{J_{aj}\}\end{pmatrix} - \frac{\partial}{\partial xj}\begin{pmatrix}0\\ \tilde{\rho}u_i'u_j'\\ \tilde{\rho}e'u_j'\\ \tilde{\rho}Y_a u_j'\end{pmatrix} \tag{5}$$

Type in the following equation, we can obtain

$$\frac{\overline{Df}}{Dt} \equiv \frac{\partial}{\partial t}(\tilde{\rho}\overline{f}) + \frac{\partial}{\partial x_j}(\tilde{\rho}\overline{u}_j\overline{f}) \tag{6}$$

Equation (5) on the left side of the variables, $\tilde{\rho}$ 、 $\overline{u}_i$ 、 $\overline{e}$ and $\overline{Ya}$, and therefore do not need to model. But the right of all the various, particularly on behalf of the turbulent flow at the end of the various, include turbulent fluctuation, must establish a model with variable expression.

Momentum equation containing $-\tilde{\rho}\overline{u}_i'\overline{u}_j'$,which represent turbulent stress. In the numerical simulation of stress equation, using model lists these stress equation and can be solved. Described to equation (5) add two speed and length scale of turbulence of partial differential equations, and then express other amount of turbulent flow in these dimensions.

Equation turbulent model using control unit mass turbulence kinetic energy equation. $k = \overline{u}_i\overline{u}_j / 2$,Control equation of k can be derived: the u_i equation of equation (1) multiplied by the u_j, minus from the equation in equation (5) equation in The Times of equation, then the results are the average. Results to find k equation is:

$$\frac{\bar{D}\bar{k}}{Dt} = \tilde{\rho}(p-e) - \frac{\partial}{\partial x_j}J_k \tag{7a}$$

p is a quality of the turbulent flow rate:

$$p = -\overline{u}_i\overline{u}_j\frac{\partial\overline{u}_i}{\partial x_j} \tag{7b}$$

In the actual experiment, all equation model with a dissipative e model equation is used as the second turbulent flow equation. Due to the energy dissipation rate is not controlled by the viscosity, and the supply of large eddy simulation of the control of the rate of dissipation of energy for small scale, which itself can also can adjust the size to deal with this. So, if it is a large eddy simulation turbulence length scale, dissipation on k and should be fixed scale:

$$e \propto k^{2/3} / l \tag{8}$$

In equation turbulent model, all unknown amount of turbulent flow with turbulent velocity scale $k_1/2$ and the length of the feet of $k_2/3/e$ expression, or expressed in k and e. Therefore, this model is called k-e model. A general method includes defining a turbulent viscosity:

$$\mu_T \equiv Co\tilde{\rho}k^2 / e \tag{9}$$

Then the equation (5) and (7) in the turbulent stress a quasi Newton method was used to simulate:

$$\tilde{\rho}\overline{u_i' u_j'} = \tilde{\rho}\frac{2}{3}k\delta_{ij} + \frac{2}{3}\mu_T \nabla \overline{u}\delta_{ij} - 2\mu_T \overline{S}_{ij} \tag{10}$$

In the type $\overline{s}_{ij}$ is a $\overline{u}_i$ strain rate

$$\overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \tag{11}$$

Viscous stress term in the momentum equation with Newton structural equation to estimate. They include additional associated with the pulsation of molecular viscosity turbulence.

In all kinds of conveying turbulent diffusion term in the equation, turbulent diffusion coefficient is used to simulate the diffusion fluxes of phi, use the simulation:

$$J_{\Phi i} = -\frac{\mu_T}{\delta_\phi} \cdot \frac{\partial \phi}{\partial x_i} \tag{12}$$

Finally, the model with a e transport equation. E the accuracy of the equation can be used to properly handle navier Emmanuel - stokes equations are derived, which is to establish e equation is useful guide. All e equation model have this form:

$$\frac{\overline{D}e}{Dt} = W - \frac{\partial H_i}{\partial x_i} \tag{13}$$

With other type of diffusion flux Hi diffusion with the same method to simulate.

The above equation model has been successfully used in free and shear layer under the constraint of a wall. Application of the above changes, this kind of model even in the near wall region can also be a good simulation. However, if you want to put the calculation has been to the wall, then in the vicinity of a wall to use very thin computing grid, so the usual procedure is to send wall near the fitting a hypothetical "general" wall layers form. But in the engine, the wall boundary layer have no time to slack to local equilibrium, so the surface treatment is a potential source of error.

Summary

Through the above analysis, from the study of flow model can predict the flow model of the structure of the flow of the burning point so far, should be able to give the simple used in the turbulent combustion model parameter values. In a stratified charge quantity system, a good flow model should be able to predict filling quantity distribution of ignition time. If can construct a model to deal with the combustion process, then, should be at least can predict the pressure changes over time course and the estimated cycle flow pattern. If the model can properly deal with quenching layer and the circulation flow, is also likely to predict pollutant discharge time process.

In the type of experimental research model is the most useful, is only one factor change and all other experiments. Other basic experiment including the unsteady compressible boundary layer, compressible flow china-africa steady flow field of the experiment, and only from data is complete turbulent flow field including combustion and expansion of the experiment.

Is worth questioning, model validation should depend on the actual engine experiment, record, cylinder pressure and exhaust gas flow field changes over time. During the experiment should be based on the parameters of the system changes, forecast analysis model with the trends and laws of design parameters.

References

[1] R.J.T abaczynski, "Turbulence and Turbulent Combustion in Spark Ignition - Engincs." Prog. Energy Combust. Sci., Vol. 2, pp. 143-165197-6:

[2] W.C.R eynoynolds. Combustion simulation. Stanford university in the United States. Liu Yijun. 1987.8.

[3] Li-xing zhou. Fog liquid fuel combustion efficiency of modeling and analysis problem. Gas turbine combustor experimental technique project academic conference papers anthology. Shanghai mechanical engineering society. Beijing: China's industrial press, 1964.

[4] Chen Danzhi. Modeling law of low speed of steady state condition of combustion system. Journal of engineering thermal physics, 1980, 1 (4)

[5] Qing-guo liu. The application of pneumatic atomizer combustion performance test research. Combustion science and technology, the series of 2, 1984