HUNGARIAN JOURNAL
OF INDUSTRIAL CHEMISTRY
VESZPRÉM
Vol. 39(1) pp. 27-30 (2011)

SIMULATION OF FLUID FLOW, COMBUSTION AND HEAT TRANSFER PROCESSES IN INTERNAL COMBUSTION ENGINES

T. MORAUSZKI[1✉], P. MÁNDLI[1], Z. HORVÁTH[1], M. R. DREYER[2]

[1]Széchenyi István University, Department of Mathematics and Computer Sciences
[2]Széchenyi István University, Audi Hungaria Department of Internal Combustion Engines
9026, Győr, Egyetem tér 1, HUNGARY
✉E-mail: mtamas@sze.hu

The better understanding and optimization of fluid flow, injection, mixing and combustion processes in internal combustion engines are of critical importance in regard to compliance of the emission standards. Presented here is a complex simulation method, which provides transient fluid flow results in a cylinder of a gasoline engine. The model includes moving boundaries. Through the intake port and open intake valve, air is flown into the combustion chamber. Fuel is injected. In the compression stroke, after closing the intake valve, the mixing process is shown. At the top dead center (TDC) ignition is set, which results in combustion of the fuel-air mixture in the power stroke. In the exhaust stroke the burnt gases leave the combustion chamber through the open exhaust valve and the exhaust port. With the results of this simulation, it is possible to determine the temperature and heat transfer coefficients on the walls of the combustion chamber, and to define the thermal load. This load can later be used in a structural analysis of the parts of the combustion chamber, where heat transfer and expansion is calculated.

Keywords: Internal combustion engines, Air flow simulation, Direct-injection, Spark ignition, Combustion

Introduction

Although internal combustion engine development has in certain aspects already approached its limits, just by looking ahead to the upcoming tightening of the emission norms, it is clear that further optimization is inevitable.

By now numerical simulations such as structural, thermal or fluid flow calculations have become fast, reliable and inexpensive tools in all fields of the vehicle development process.

To further broaden the frontiers, recently complex simulation methods have been developed, which simultaneously use more software and provide results of more interacting physical phenomena. These are sometimes called multiphysics applications, fluid-structure interaction (FSI) being a good example.

The main goal of this work is to develop and present a complex simulation method for evaluating important physical properties in the combustion chamber of a gasoline direct injection engine, such as air flow in the combustion chamber, direct injection of gasoline, mixture formation, spark ignition and combustion processes, heat generation and heat transfer from hot air to the surrounding solid walls. These tasks were solved by one of the computational fluid dynamics (CFD) software, Ansys Fluent. It should be noted that the presented methodology is capable to include combustion of different fuel materials under realistic operating conditions.

In-cylinder air flow with moving boundaries

As a slight simplification, a one-cylinder part is taken from a 2-liter inline four-cylinder naturally-aspirated gasoline direct injection spark ignition engine. The extent of the model is therefore limited to an intake and an exhaust port – with two intake and two exhaust valves – and a combustion chamber. The latter is bounded by the piston top surface and walls of the cylinder and cylinder head. *Fig. 1* shows the computational domain at a mid-stroke stage. The intake port is on the left side, while the exhaust port is to the right of each picture.

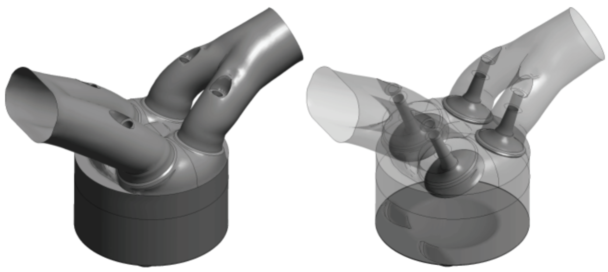

Figure 1: Shaded and partly transparent view of the computational domain

Construction of the fluid flow model

Transient, highly compressible and turbulent air flow has to be considered inside the cylinder. This makes the numerical simulation rather demanding in regards to computing power and time. Further complications arise because of the moving boundaries: some parts of the discretised model have to be remeshed in each and every time step.

To keep the calculation speed within reasonable limits, the varying number of elements is kept in the 200–400.000 range during the whole calculation. With these restrictions it is possible to run a complex simulation (including combustion) in one day time, using 16 processor cores. Further refinement of the mesh might help to achieve better results, which in turn might slow the computation speed to unacceptable levels.

The chosen refinement level of the mesh does not allow boundary layers to be meshed properly therefore losing information on turbulence is inevitable. This effect was reduced by using the two-equation realizable k-ε turbulence model with enhanced wall functions [1].

During the calculation one four-stroke cycle is computed, this corresponds to a rotation of 720° crankshaft angle. On *Fig. 2* the events of the cycle are shown.

The starting angle is 360°: piston is at the TDC, intake stroke starts and the intake valve opens, following the valve lift curve (as seen on Fig. 2, part A). Air is passing through the inlet of the intake port, where free-flow pressure inlet boundary condition is defined, which corresponds to the intake process of a naturally-aspirated engine.

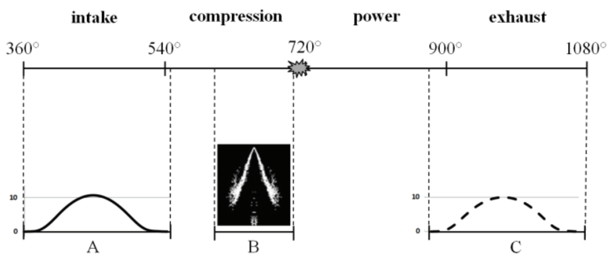

Figure 2: Timeline of events during the simulation; A: intake valve is open, B: gasoline injection, C: exhaust valve is open

At 550° the intake valve closes and during the compression and power stroke no air enters or leaves the combustion chamber, the air flow is guided by the piston motion and the effects of injection and combustion process.

In the exhaust stroke, starting at 900°, the exhaust gas leaves through the open exhaust valves. In the computation free-flow pressure outlet was applied at the end of the exhaust port. The lift curve of the exhaust valve can be seen on Fig. 2, part C. The events end at 1080°.

Dynamic mesh

The aforementioned events predispose the structure of the model: a dynamic mesh has to be generated, which consists of tetrahedral and hexahedral elements and continuously changes during the steps of the computation.

The dynamic mesh system has three types of meshes. Stationary parts are not involved in any movement (e.g. most of the intake and exhaust ports meshed with tetra elements). Rigid parts are moving, but not deforming (e.g. a layer of elements above the piston top surface). Deforming parts can change shape (e.g. regions around the valves). Here either tetra elements are created by remeshing or hexa elements are created by layering. On *Fig. 3* the process of mesh deformation can be seen. On the left picture the exhaust valve is closed, while on the right it is open: its walls are moved, the tetra mesh under it is remeshed and the hexa layers above it are multiplied as needed.

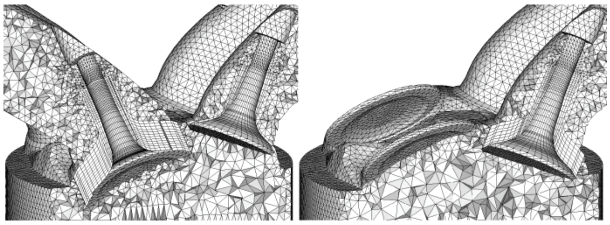

Figure 3: Two states of the dynamic mesh at the region of the valves

Piston movement is governed by basic engine data: crankshaft speed is 2000 rpm, piston stroke is 90 mm and connecting rod length is 150 mm. Intake and exhaust valve timing and lift is subject to change depending engine settings (e.g. speed, load) and has to be defined for each case with movement profiles.

Results of the fluid flow simulation

Examination of the fluid flow results at several stages of the cycle shows that the events happened as planned and are in balance with the normal behavior of a four-stroke engine.

Fig. 4 shows path lines of the airflow from all four strokes. On Fig. 4/a the air intake process is visible: the vacuum-effect of the downward moving piston sucks air through the intake valve. The shape of the piston top surface aids the development of a swirling flow.

In the compression stroke all valves are already closed and the piston moves upward. As the volume of the airspace becomes smaller, the results show the inevitable rise in the density of the material. At this stage work is done on the system through the crankshaft. As a consequence the internal energy, i.e. the mean temperature of the air goes up to 400 °C by the end of compression.

As Fig. 4/b and 4/c show, the air flow pattern continues to evolve, maintaining a high-swirl profile both in the compression and power stroke. Fig. 4/d is a good example of the exhaust process: gas leaves the combustion chamber through the open valves.

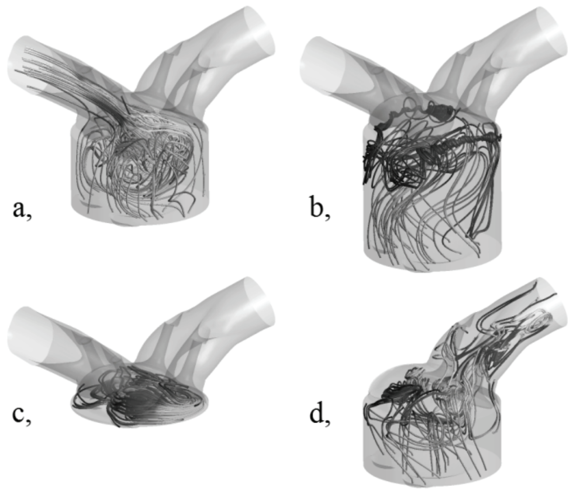

Figure 4: Path lines of fluid flow in a, intake; b, compression; c, power and d, exhaust stroke

As a short summary of the fluid flow process it can be said, that the results are as expected. Further evaluation and model possible refinement is necessary. Other engine setups are also to be computed and compared.

Combustion process

Construction of the combustion simulation

As the combustion model is introduced to the air flow simulation, the timeline of events can be further divided. On Fig. 2, part C is bounded by the start and end time of gasoline injection (in this engine setup it is between 600° and 700° in the compression stroke).

Injection is provided through the discrete phase model, where in the compression stroke 40 mg of gasoline droplets, discrete particles are injected with a relatively high velocity of 120 m/s [2]. Although the actual injector geometry is not modeled, the injection is parameterized as a solid cone with four particle streams and 25° cone angle. The injected material is octane, which is a major part of gasoline and its combustion properties are closely linked to gasoline itself.

The mixture formation in direct-injection engines happens in a relatively short duration of the compression stroke. Shortly after injection – at 708°, i.e. 12° before TDC – the spark ignition is modeled.

Spark is defined as a sphere at the position of spark plug (which itself is not included in the model). It has sufficient energy (0.1 J) to initiate combustion 12° before TDC.

Combustion is modeled with the partially premixed combustion model, which is a simple combination of the non-premixed [3] and the premixed [4] combustion models. As the software provides good initial settings to these combustion models, it is possible to calculate the flame front position, heat generation, unburnt and burnt mixture fraction properties without detailed knowledge of the combustion model. In order to further optimize the method, a deeper understanding of these models is desired. It should also be noted, that with user defined functions it is possible to include custom written extensions in the combustion calculations.

Results of the combustion simulation

The high-speed discrete octane-particles entering the combustion chamber have to be mixed with the high-swirl air flow. Despite the short duration (only 108° crankshaft angle) until the spark at 708°, the result shows a relatively even distribution of the injected particles. This makes it possible, that after the spark initiation a smooth combustion process can happen.

As the results of *Fig. 5* show, the flame front is constantly evolving after the spark ignition.

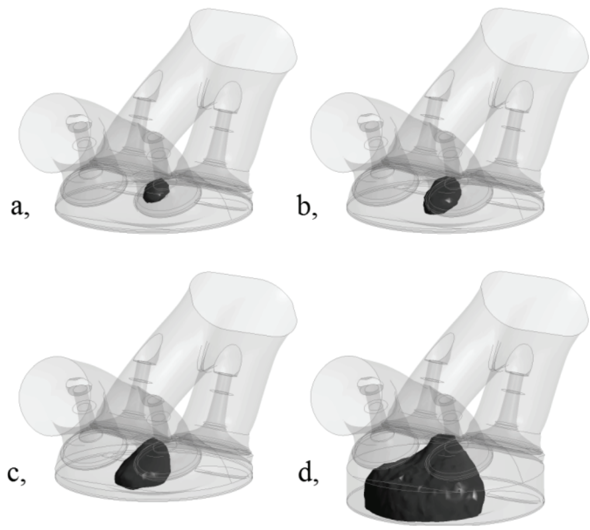

Figure 5: Evolution of the flame front: 8° after spark initiation (a,) and further 10-20-30° later (b, c, d,)

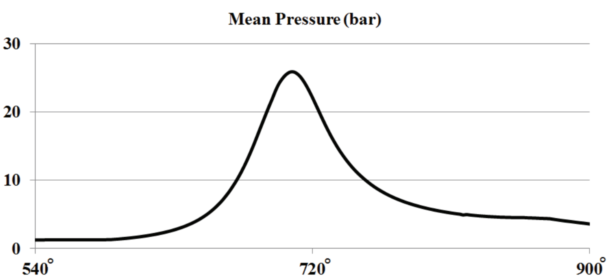

Figure 6: Mean pressure in the combustion chamber during the compression (540–720°) and power (720–900°) stroke

As seen in *Fig. 6*, the mean cylinder pressure has its peak at TDC, right after spark ignition at 25 bar. For a naturally-aspirated engine running at 2000 rpm, mid-load these results are in the expected range [5]. So is the cylinder mean temperature, which rapidly rises to 1100° C.

Conclusions and outlook

A simulation method was developed to model in-cylinder fluid flow processes bounded by moving walls. Injection, mixture formulation, spark ignition and combustion was also included. Delivered results show good correlation with literature values.

Further evaluation of the method is necessary: mesh refinement possibilities should be considered, important parameters need fine-tuning and the extent of human interventions during the calculation should be reduced. Comparison to measurements and parametric studies should be carried out.

Method extension possibilities include implementation of custom defined combustion methods. A model for turbocharged engines is of great interest, too. In this case, overpressure has to be defined at the intake inlet.

In every simulation case, the results can further be used: wall temperatures and heat transfer coefficients can be exported to a finite element (FE) solver. Mapping CFD-results to similar but not identical FE-meshes can also be done.

With FE-calculations heat transfer and thermal expansion of the surrounding parts (e.g. piston, engine block) can be calculated. Expansion has a significant role in piston and piston ring dynamics. It can and will influence the mechanical friction losses of the piston-rings-liner system, therefore taking expansion into consideration while doing friction simulations is of great importance.

ACKNOWLEDGEMENTS

TAMOP-4.2.1/B-09/1/KONV-2010-0003: Mobility and Environment: Research in the fields of motor vehicle industry, energetics and environment in the Central- and Western-Transdanubian Regions of Hungary. The Project is supported by the European Union and co-financed by the European Social Fund.

REFERENCES

1. H. C. CHEN, V. C. PATEL: Near-Wall Turbulence Models for Complex Flows Including Separation, AIAA Journal, 26(6), (1988), 641–648
2. F. ZHAOA, M.-C. LAIA, D. L. HARRINGTON: Automotive spark-ignited direct-injection gasoline engines, Progress in Enegry and Combustion Science, 25, (1999), 437–562
3. Y. R. SIVATHANU, G. M. FAETH: Generalized State Relationships for Scalar Properties in Non-Premixed Hydrocarbon/Air Flames, Combustion and Flame, 82, (1990), 211–230
4. V. ZIMONT, W. POLIFKE, M. BETTELINI, W. WEISENSTEIN: An Efficient Computational Model for Premixed Turbulent Combustion at High Reynolds Numbers Based on a Turbulent Flame Speed Closure, J. of Gas Turbines Power, 120, (1998), 526–532
5. J. B. HEYWOOD: Internal Combustion Engine Fundamentals, McGraw-Hill, 1988, 371–383