DIESEL ENGINE IN-CYLINDER CALCULATIONS WITH OPENFOAM

[1]Ervin Adorean*, [1]Gheorghe-Alexandru Radu

[1]Transilvania University of Brasov, Romania

KEYWORDS - diesel, engine, CFD, simulation, OpenFOAM

ABSTRACT - The reduction of diesel engines emissions is a challenging task. Reducing both soot and NOx at the same time is quite difficult, especially with no fuel consumption penalty. Reducing them both is possible by enhancing the air-fuel mixing process in order to improve combustion. Experimental research still does not give us enough information about the very complex thermo-and fluid dynamic processes that are occurring in a diesel engine. In addition, experimental research is still quite expensive and time consuming.

The objective of this work is to give support to experimental research by providing some insight into the complex phenomena that occur in the cylinder and the combustion chamber of diesel engines, using Computational Fluid Dynamics (CFD) modeling. The first task was to calculate the air flow through the inlet ports and cylinder of a diesel engine and the second one was to calculate combustion in a diesel engine.

A steady flow test rig setup for diesel engine ports development has been successfully simulated with OpenFOAM (a free CFD software). The flow through the inlet ports was simulated at different valve lifts. The CFD calculated values of flow coefficients and swirl ratios agree fairly well with the experimental results. By sectioning the domain, a detailed description of the flow field can be obtained. A step forward was the simulation of combustion in a direct injection diesel engine. This is a much more complex task, but the calculated in-cylinder pressure corresponds to the measured pressure.

INTRODUCTION

Combustion in diesel engines is influenced by the injection process and fuel jet interaction with the air inside the cylinder. Knowledge of the air movement inside the cylinder is of great importance for the improvement of the combustion process and it helps reduce the pollutants emissions of diesel engines.

There are many methods of studying the in-cylinder air movement, but they can be classified in two groups: experimental research and mathematical calculus models. Experimental research reproduces more or less the real operating conditions of the engine and usually it has high costs. Nevertheless, experimental validation is always necessary in the development of an engine. Amongst the mostly used experimental methods for in-cylinder flow study are the AVL – Thien or Tippelmann method of steady flow test rig, which outputs two coefficients which characterize the in-cylinder flow, and LDA (Laser-Doppler Anemometry), which can measure the velocity in some limited measuring points in the volume.

In the last few years the computing power of workstations and personal computers grew exponentially, and it continues to grow. Computational Fluid Dynamics (CFD) programs have also evolved. They became more robust, precise, and they can even calculate the combustion process in engines. The results obtained by CFD calculations agree well with experimental research. The costs of studying in-cylinder flow and combustion with CFD is only at a fraction of that of experimental research. After calculation with a CFD program, there are much more information available about the flow field then with experimental research such as: velocity fields, pressure, turbulent kinetic energy, etc. It is very important though that CFD calculations are always validated by experimental results.

RESULTS AND DISCUSSION

STEADY-STATE IN-CYLINDER CALCULATIONS

In this section the results of stationary air flow through the inlet pipes and cylinder of a diesel engine will be presented (bore : 130 mm, stroke : 150 mm), for four valve lifts : 4, 6, 8 and 10.2 mm. Among the studied parameters are velocity, pressure, turbulent kinetic energy, dissipation of turbulent kinetic energy, flow coefficient and swirl ratio. The last two parameters will be compared with experimental measurements on a Tippelmann steady flow test rig.

The boundary conditions used in the calculations are the same as in experimenrtal measurements. Amongst the used mathematical models are : SIMPLE velocity and pressure fields coupling, standard k-epsilon turbulence model, standard pressure, First Order Upwind momentum, energy and turbulent kinetic energy discretization. The pressure boundary conditions imposed at the entrance of the inlet pipes are the ones corresponding to atmospheric pressure during experimental measurements on the stationary flow bench. It is necessary to estimate the turbulence intensity and the turbulent length scale. The turbulence intensity in a duct is the ratio between the mean velocity deviation, u' and the flow mean velocity u_{mean}.

For the domain inlet, turbulence intensity is given by :

$$I = \frac{u'}{u_{mean}} \cong 0{,}16 \cdot (\mathrm{Re})^{-1/8}$$

$$v_m = \frac{m}{A_{in} \cdot \rho} \cong 47 \left[\frac{m}{s} \right]$$

The Reynolds number is :

$$\mathrm{Re} = \frac{\rho \cdot v_m \cdot D}{\mu} \cong 156000 \Rightarrow I \approx 4\%$$

where:
m – mass flow
v_m – mean velocity
D – equivalent diameter
ρ - density
μ - dynamic viscosity

The turbulent length scale may be considered equal to l=0.1D, where D is a characteristic dimension, in this case beeing the equivalent diameter of the inlet section of the pipes (≈ 50 mm).
Concluding, the inlet boundary conditions are:

- total pressure: 102125 Pa
- temperature: 296 K
- turbulence intensity: 4%
- turbulent length scale: 0.005 m.

For the domain outlet turbulence intensity is:

$$v_m = \frac{m}{A_{out} \cdot \rho} \cong 15 \left[\frac{m}{s} \right]$$

and the Reynolds number is:

$$\mathrm{Re} = \frac{\rho \cdot v_m \cdot D}{\mu} \cong 125000 \Rightarrow I \approx 4\%$$

The outlet boundary conditions are:

- static pressure: 98205 Pa
- temperature: 296 K
- turbulence intensity: 4%
- turbulent length scale: 0.013 m

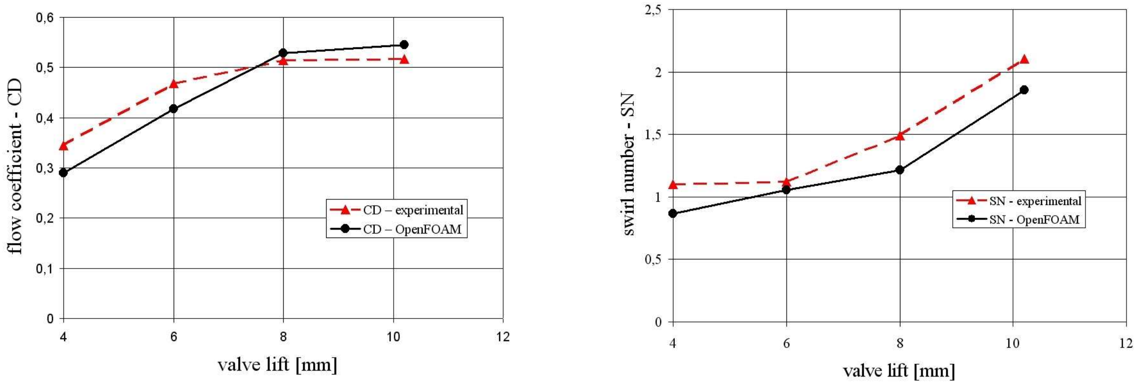
Boundary condition	Inlet	Outlet
Pressure [Pa]	102125	98205
Temperature [K]	296	296
Turbulence intensity [%]	4	4
Turbulent length scale [m]	0.005	0.013

Domain inlet and outlet boundary conditions

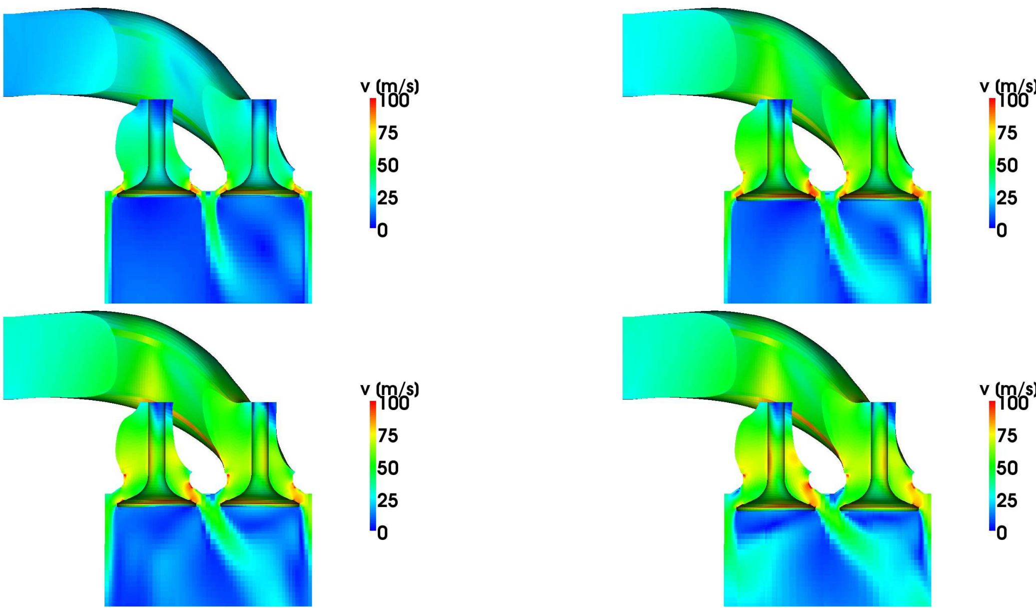
The velocities between the valve and valve stem grow as the valve lift increases. In the case of a 4 mm valve lift, the velocities are low, but at higher valve lifts, the air flow at the left valve follows the shape of the volume created between the valve and valve stem.
The flow velocities in a section found at 1.75D are also increasing for higher valve lifts and the flow patterns are more organized and the tangential pipe’s flow becomes the dominant one.

Hexahedral meshes for in-cylinder flow (left) and combustion (right) calculations

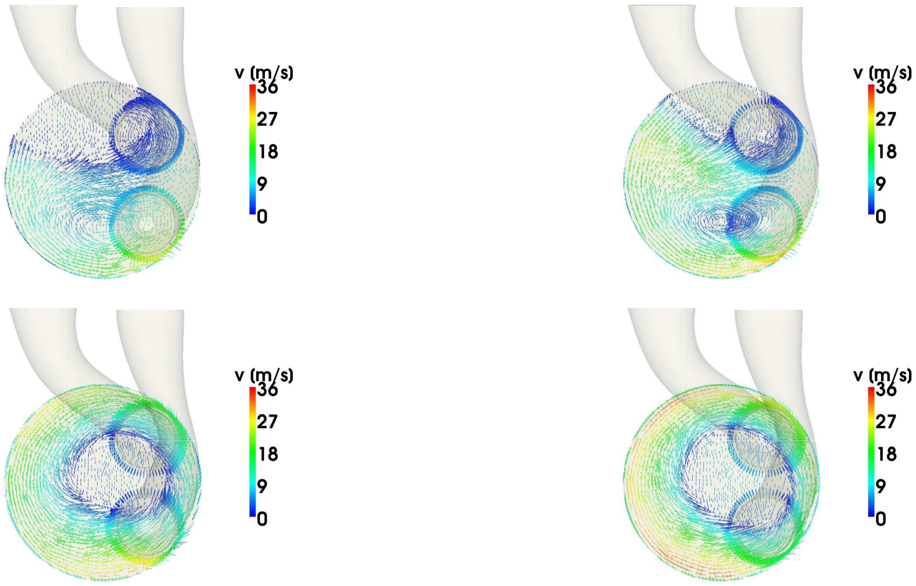
The two parameters that characterize the air flow through the inlet and outlet ports of an engine are the flow coefficient or coefficient of discharge – CD and the swirl number – SN. The CFD calculated values of these parameters agree fairly well with the experimental values, obtained in a Tippelmann steady flow test rig.



Flow coefficient (left) and swirl number (right)



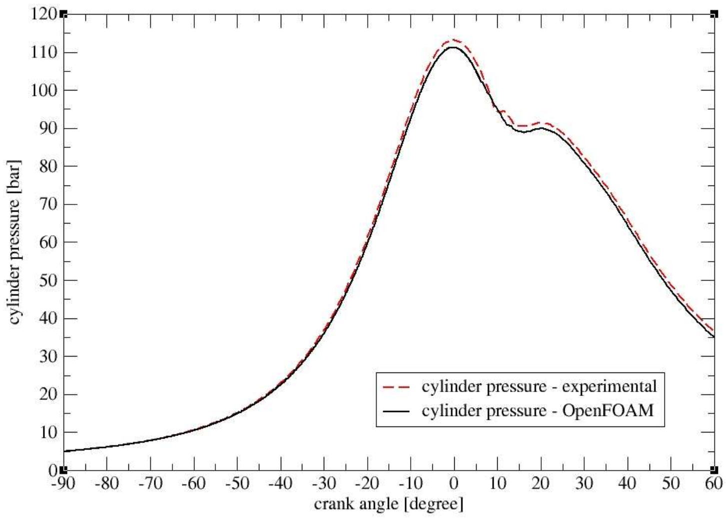
Contours of velocity magnitudes at 4, 6, 8 and 10.2 mm valve lift (left to right, top to bottom)



Vectors of velocity magnitudes at 4, 6, 8 and 10.2 mm valve lift (left to right, top to bottom)

COMBUSTION CALCULATIONS

Combustion calculations were done for a 6 cylinder direct injection heavy-duty diesel engine, with a bore of 123 mm and a strike of 156 mm.
Some of the mathematical models employed in OpenFOAM for the combustion simulations were: k-epsilon turbulence model, KH-RT (Kelvin Helmholtz – Rayleigh Taylor) wave breakup model, standard OpenFOAM coalescence and vaporization models, Chalmers PaSR (Partially Stirred Reactor) combustion model, n-heptane (C_7H_{16}) as Diesel fuel surrogate with a simple reaction mechanisms: 5 species and 1 reaction.
The calculated cylinder pressure agrees well with the experimental pressure. Once the validation is done, the influence of different parameters on combustion may be studied: injection pressure, injector nozzle inclination angle, swirl ratio and injection timing.



Experimental and OpenFOAM calculated cylinder pressure

Temperature (left) and temperature isosurface (right)

CONCLUSIONS

In this paper only a selection of the results are presented. It has been proved that although not widely used yet, OpenFOAM is a CFD code capable of modelling diesel engine in-cylinder phenomena, be it steady state flows or combustion.
CFD programs also offer more information about the flow field than experimental research. It is very important to define a reliable methodology of work for these kind of simulations, methodology which can be applied for any configuration of diesel engine simulations.
The larger differences obtained for the flow coefficient at high valve lifts might be due to the used turbulence models. This will have to be further investigated.

REFERENCES

(1) G. A. Megias, Caracterizacion del flujo de aire en el cilindro de motores diesel mediante calculo tridimensional – Tesis doctoral, CMT Motores Termicos, Universidad Politecnica de Valencia, 2003.
(2) E. Adorean, Modelarea traiectului canalizatiei de admisie si analiza cu elemente finite a unor fenomene din canalizatie, Referat nr. 2 din cadrul pregatirii Tezei de Doctorat, Facultatea de mecanica, Universitatea Transilvania din Brasov, Brasov, 2006.
(3) A. Horvath, Application of CFD numerical simulation for intake port shape design of a Diesel engine, Department of Physics, Szechenyi Istvan University, Gyor, 2002.
(4) P. Taskinen, Modeling of spray turbulence with the modified RNG k-epsilon model, Energy and Process Engineering, Tampere University of Tehnology.
(5) J. Benajes, X. Margot, J. V. Pastor, A. G. Megias, Threedimensional calculation of the flow in a DI diesel engine with variable swirl ports, Automotive Transportation Conference Exposition, Barcelona, SAE paper 2001-01-3230, 2001.
(6) Jasak H, Weller HG and Nordin PAN (2004) In-Cylinder CFD Simulation Using a C++ Object-Oriented Toolkit. SAE paper 2004-01-0110.
(7) Nordin PAN (2001) Complex Chemistry Modeling of Diesel Spray Combustion. PhD thesis, Chalmers University of Tehnology.
(8) Payri F, Benajes J, Margot X and Gil A (2004) CFD Modelling of the In-Cylinder Flow in Direct-Injection Diesel Engines. Computers & Fluids: vol 33: pp 995-1021.