Cold flow simulation of an internal combustion engine with vertical valves using layering approach

G Martinas\textsuperscript{1}, O S Cupsa\textsuperscript{1}, L C Stan\textsuperscript{2} and A Arsenie\textsuperscript{2}

\textsuperscript{1} CERONAV, 69 A Pescarilor Str. 900581, Constanta, Romania
\textsuperscript{2} Maritime University of Constanta, 104 Mircea cel Batran Str., Constanta, Romania

E-mail: georgemartinas@ceronav.ro

Abstract. Complying with emission requirements and fuel consumption efficiency are the points which drive any development of internal combustion engine. Refinement of the process of combustion and mixture formation, together with in-cylinder flow refinement, is a requirement, valves and piston bowl and intake exhaust port design optimization is essential. In order to reduce the time for design optimization cycle it is used Computational Fluid Dynamics (CFD). Being time consuming and highly costly caring out of experiment using flow bench testing this methods start to become less utilized. Air motion inside the intake manifold is one of the important factors, which govern the engine performance and emission of multi-cylinder diesel engines. Any cold flow study on IC is targeting the process of identifying and improving the fluid flow inside the ports and the combustion chamber. This is only the base for an optimization process targeting to increase the volume of air accessing the combustion space and to increase the turbulence of the air at the end of the compression stage. One of the first conclusions will be that the valve diameter is a fine tradeoff between the need for a bigger diameter involving a greater mass of air filling the cylinder, and the need of a smaller diameter in order to reduce the blind zone. Here there is room for optimization studies. The relative pressure indicates a suction effect coming from the moving piston. The more the shape of the inlet port is smoother and the diameter of the piston is bigger, the aerodynamic resistance of the geometry will be smaller so that the difference of inlet port pressure and the pressure near to piston face will be smaller. Here again there is enough room for more optimization studies.

1. Introduction
During their evolution pistons evolved along with engines. They became lighter and shorter. They use smaller skirts, the cylindrical body of the piston. Aluminum alloys contains more silicon than before pistons being made of that and using this material will reduce expansion by thermal and improves resistance to heat.

Use of different piston crowns or top, the part that is subject to combustion and the part that enters the combustion chamber, is the most advancement way to improve piston design. Most of the older pistons were flat, but many of the bowls on top with today features influence differently the combustion process. Diesel engines use mainly the piston bowl. There is not an ignition phase at diesel engines, so the combustion chamber may beformed by the piston crown itself. The pistons with differently shaped crowns function with direct injection are becoming very popular and are nowadays often used for diesel and gasoline engines, [1].
The objective of the present study is the simulation of the CFD functioning on the IC engine with inlet valve, on a manifold with an intake and a piston which uses dynamic mesh. This is the cold flow analysis.

Giving a shape to the airflow in the transient engine cycle with no reactions is a process which is involved by the cold flow analysis. The interaction of the fluid dynamics with the moving geometry of the induction process must be accurately accounted and thus the goal will be achieved: capturing the mixture formation process. This way it can be determined the changing features of the jet of the airflow tumbling into the cylinder with a swirly motion. The process happens when the air enters the intake valves and when it is passed out through the opening and closing exhaust valves. It can also be determined the turbulence production due to swirl and tumble which are generated by the compression and squish, [2].

The information mentioned in the above paragraph is very useful from the combustion and flame propagation point of view, which are influenced by the conditions in the cylinder which is situated at the end of the compression stroke. During the power stroke, the high turbulence levels are facilitated first, by the rapid propagation of the flame and second, by the complete combustion. The right air/fuel ratio of the combustion is ensured by the airflow which is highly turbulent and well mixed. The level of charge stratification can also be assessed by the CFD.

However, reality is not reflected by the flow characteristics which happen during the power and exhaust strokes, because the significant thermodynamic changes which accompany combustion are not included in the cold flow simulations. In terms of validation, even if transparent cylinders and pistons have been used to obtain accurate information about some engine configurations, it is not easy to observe data about PIV (Particle Image Velocimetry) or LDV for cycling engines, due to the analysis of the port flow.

For above mentioned approach, in ANSYS FLUENT 15, can be considered three steps [3]:
1. Mesh the zones properly, after decomposing into the different zones, the geometry. Wishing to apply various strategies regarding mesh motion, to various regions during one simulation, model was divided into various zones.
2. Setup journal is used in setting the engine case in ANSYS FLUENT
3. Perform a transient IC simulation

The cold flow analysis of the combustion chamber is the starting point for the optimization of the geometric configuration of the piston and combustion chamber geometry, [4].

2. CAD and Finite Volume Analysis (FVA) Model of the IC

The goal of this paper is to simulate the aerodynamics of the air induction and end compression stages for a vertical valves IC engine. Therefore the CFD will have as fluid domain the negative of the In-Cylinder (the combustion chamber along with its ports) as given below (CAD):

![Figure 1. In cylinder and inlet port CAD.](image)

Only the inlet port and valve were modeled for the air admission process.
Once the piston/valve is moving, the newly created volumes need to be re-meshed using the dynamic meshing function. Three groups of mesh motion methods can be used by ANSYS Fluent. In order to update in the deforming regions the volume mesh considering motion defined at the boundaries: Smoothing Methods; Dynamic Layering; Remeshing Methods. For vertical valves IC the Dynamic layering is involved. [5]

The meshing of the fluid domain has to have defined certain zones with specific mesh requirements, as given in figure2. The mesh requirements are:
1 fluid-piston- any mesh, 2 fluid-ch-lower- layered mesh, 3 fluid-ch-upper -any mesh, 4 fluid-ch-rootname -layered mesh, 5 fluid-rootname-ib -layered mesh, 6 fluid-rootname-vlayer - layer mesh

![Figure 2. Sketch of Decomposition and Zone Names.](image)

The fluid to be considered in simulation will be air with standard properties. The engine functioning/geometric parameters are: Crank shaft speed 1200 rpm; Crank period 720 deg; Crank radius 25 mm; Connecting rod radius 90 mm, Piston diameter 55 mm.[6]

3. CFA Simulation Results
The results analysis will cover the following crank angles (CA):
CA = 372 (deg) in which the inlet valve is opening;
CA = 445 (deg) in which the piston is moving downwards at halfway of its course;
CA = 535 (deg) where the piston is fully displaced for the air admission phase and inlet valve is closed.

3.1. Results for CA = 372 (deg)

![Figure 3. Velocity fields for CA=372 deg.](image)

![Figure 4. Pressure fields for CA=372 deg.](image)

When the inlet valve is opening the air is rushing inside the cylinder having high velocities up to 38.6 m/s near the valve seat. The air is starting to fill the cylinder with speeds varying from 2.5 to 5 m/s. Immediately under the valve face is detected a blind zone where the air velocity is null. One of the first conclusions will be that the valve diameter is a fine tradeoff between the need for a bigger diameter
involving a greater mass of air filling the cylinder, and the need of a smaller diameter in order to reduce the blind zone. Here there is room for optimization studies [7].

The relative pressure indicates a suction effect coming from the moving piston. The more the shape of the inlet port is smoother and the diameter of the piston is bigger, the aerodynamic resistance of the geometry will be smaller so that the difference of inlet port pressure and the pressure near to piston face will be smaller. Here again there is enough room for optimization studies, [8].

Figure 5. Turbulence kinetic energy for CA=372 deg.

The bigger the turbulence is, the better is for the functioning of the internal combustions engines. That is for the impact of the turbulence upon a better fuel-air mixing leading to finer droplets of fuel which finally is increasing the contact surface between fuel and air and the reaction rate. The turbulence energy (max. 28.8 J/kg.) is bigger in the valve seat zone and smaller in the blind zone under the valve seat.

3.2. Results for CA = 445 (deg)

As the air is continuing fill the IC space, its velocity continues to increase as well. The maximum velocity becomes 69.1 m/s in the same zone of valve seat and the blind area is decreasing in dimension under the push of the air inside.

The pressure distribution tends to be more uniform across the IC, with peaks on the valve boundary where the air is still pushing upon.

The maximum value of turbulence kinetic energy is relatively decreasing to 106.4 J/kg, but the volume of turbulent regions is increasing instead. Opposed to the inlet valve face the turbulence is minimum as an effect of the blind zone, [9].

Figure 6. Velocity fields for CA=445 deg.  
Figure 7. Pressure fields for CA=445 deg.
3.3. Results for CA = 535 (deg)

The inlet valve is closed and no more air is accessing the cylinder. Therefore the velocity field near the valve seat is smaller than before and the maximum velocities are registered near the piston face, with peak values of 11.9 m/s.

The pressure fields tend to redistribute with peaks near the opposed to valve zone of the cylinder.

The peak values of the turbulence kinetic energy are diminishing to values of 17.9 J/kg.
4. Conclusions

Air motion inside the intake manifold is one of the important factors, which govern the engine performance and emission of multi-cylinder diesel engines.

Any cold flow study on IC is targeting the process of identifying and improving the fluid flow inside the ports and the combustion chamber. This is only the base for an optimization process targeting to increase the volume of air accessing the combustion space and to increase the turbulence of the air at the end of the compression stage.

One of the first conclusions will be that the valve diameter is a fine tradeoff between the need for a bigger diameter involving a greater mass of air filling the cylinder, and the need of a smaller diameter in order to reduce the blind zone. Here there is room for optimization studies.

The relative pressure indicates a suction effect coming from the moving piston. The more the shape of the inlet port is smoother and the diameter of the piston is bigger, the aerodynamic resistance of the geometry will be smaller so that the difference of inlet port pressure and the pressure near to piston face will be smaller. Here again there is enough room for more optimization studies.

References

[1] Bakar R A and Ismail A R 2008 Computational Visualization and Simulation of Diesel Engines Valve Lift Performance Using CFD Semin, Automotive Focus Group Malaysia American Journal of Applied Sciences 5(5) pp 532-539
[2] Bhagat A R and Jibhakate Y M 2012 Thermal Analysis and Optimization of I.C. Engine Piston Using Finite Element Method International Journal of Modern Engineering Research 2(4) pp 2919-2921
[3] ANSYS 15.0 Help Library
[4] Alkidas A C 1989 Performance and emissions achievements with an uncooled heavy duty, single cylinder diesel engine SAE 890141
[5] Alkidas A C 1987 Experiments with an uncooled single cylinder open chamber diesel SAE Paper 870020
[6] Mattarelli E, Rinaldinia C A and Golovitchev V I 2014 2013 CFD-3D analysis of a light duty Dual Fuel (Diesel/Natural Gas) combustion engine Energy Procedia 45 pp 929 – 937
[7] Centeno González F O, Mahkamov K, Silva Lora E E, Andrade R V and Jaen R L 2013 Prediction by mathematical modeling of the behavior of an internal combustion engine to be fed with gas from biomass, in comparison to the same engine fueled with gasoline or methane Original Research Article, Renewable Energy 60 pp 427-432
[8] Myagkov L L, Mahkamov K, Chainov N D and Makhkamova I 2014 Alternative Fuels and Advanced Vehicle Technologies for Improved Environmental Performance Advanced and conventional internal combustion engine materials 11 pp 370-392
[9] Lu G and Li L 2011 Study on Combustion Parameters of Liquefied Petroleum Gas Engine Energy Procedia 12 pp 897 – 905