Studies Regarding the Influence of the Squish In-Cylinder Movement of the Air in a Diesel Engine

Dan Moldovanu and Adela Ioana Borzan

Technical University of Cluj-Napoca, Cluj-Napoca, Romania
`{dan.moldovanu,adela.borzan}@auto.utcluj.ro`

Abstract. The objective of this paper is to establish the influence of the in-cylinder squish motion of the air on the turbulence inside the cylinder, which leads to a different output: effective power and torque.

The simulation was implemented in AVL FIRE ESE DIESEL in order to simplify the simulation and reduce the simulation time. The engine that was implemented is a Mercedes-Benz E220 engine type OM611 220 CDI, 92 kW, because the piston can be cut through to measure the precise shape so that the simulation is as close as possible to the real combustion chamber. In order to modify the squish, the distance from the piston to the cylinder head was modified (standard is 1 mm and it was modified to 0.5 mm, 1.5 mm and 2 mm), but the compression ratio was kept the same so as to not influence the performance indices.

Keywords: Squish motion, intake, CFD Simulation, AVL FIRE, AVL ESE DIESEL.

1 Introduction

Currently, the study of the in-cylinder movement of the air during the intake process for the Diesel engine has been done (a major concern) because the quality of the burning process and therefore the emission quantities can also be changed through a simple modifications of shapes for the piston and intake pipes (manifold), which is why many researchers have emerged in this direction [1, 2, 3]. The influence of the piston bowl was demonstrated by Jovanovic et al. [4, 5] and also many CFD studies were made in this direction [6].

From a simplified point-of-view, there are two types of ideal flow patterns in an engine cylinder: *swirl motion* (with the cylinder axis as the axis of rotation and the flow entering tangentially through the intake ports) and *tumble motion* (orthogonal to the cylinder axis, the axis of motion moves as the cylinder expands and stays halfway between the top cylinder wall and the cylinder head at the bottom). Both are rotational motions, however, the axis of rotation is different in each case. Depending on the type

International Congress of Automotive and Transport Engineering, CONAT 2016
Transilvania University of Brasov, 2016
ISSN 2069-0401

of engine, one of these patterns is considered optimal because it maximizes the mixture of injected fuel and air, resulting in a homogeneous combustion.

2 In-Cylinder Movements of the Air for the Diesel Engine

2.1 Swirl and Tumble Movement

The flow of air admitted into the engine cylinders is usually characterized by a swirl movement, represented in figure 1.a., the tumble movement, represented in figure 1.b., and turbulence intensity. Swirl and tumble are widely known controlled turbulences that may exist within the cylinder, either separately or combined. These movements are created during the intake stroke, the piston moves towards BDC and the fluid passes over the inlet valves. Depending on the design of the inlet movements, swirl and / or tumble are created and preserved (or slowly dissipated) into the cylinder when the intake valves are closed. Swirl motion lasts longer than tumble and can affect the post-oxidation process [7].

The swirl number is the angular velocity of swirl motion around the central axis of the cylinder, while the angular velocity of the tumble movement is perpendicular to the axis of the cylinder.

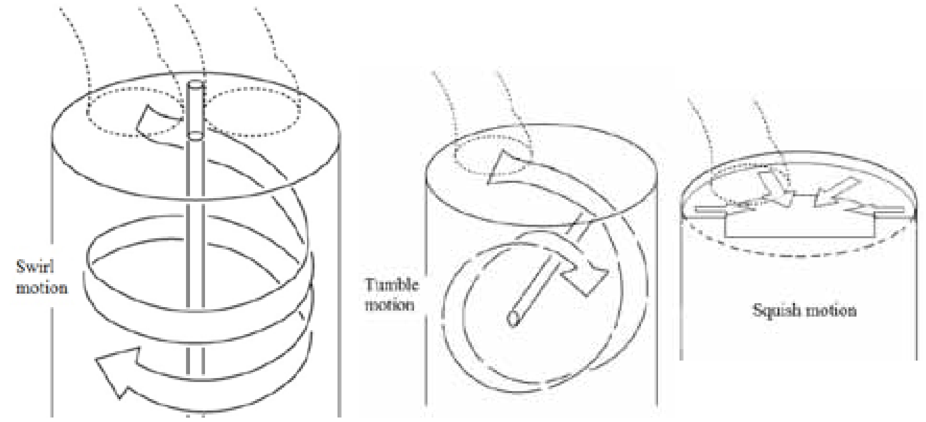
Where there are both these types of movements, they combine to create a single large vortex. Normally, the swirl motion is used in direct injection Diesel engines, while tumble is used in spark ignition engines.

During the engine cycle (the time in which the swirl motion is created and the valves are closed) the swirl movement changes between compression and combustion. At the end of the compression stroke, the rotation of the swirl motion is larger and the flow of air is forced into the combustion piston bowl. The radius of the swirl motion is reduced, increasing the angular velocity. When the piston moves toward BDC, the opposite is true. The flow also slows down due to friction with the combustion chamber walls [7].

2.2 Squish Movement

The region where the squish motion occurs, represented in figure 1.c., is located on the outer radius of the flat area of the piston. This area is between the piston and the cylinder head, designed to be of approximately 1 mm, which is important for a Diesel engine. When the piston reaches TDC (Top Dead Center) the squish movement also contributes to guide the fuel that is injected. For a given compression ratio (higher compression ratio offers higher efficiency at a certain level), the heat transfer at TDC is relatively high and by reducing the surface, the heat transfer is also diminished (the surface, the heat transfer, …).

Air trapped in the volume between the piston and the cylinder head is forced toward the center of the cylinder and creates a powerful flow. This flow (squish) can slightly influence combustion process timing after fuel injection.



a) b) c)

Fig. 1. In-cylinder motion of the air: a) Swirl, b) Tumble and c) Squish

3 Materials and Method

3.1 Simulation Software

The software that was used to simulate the combustion process is AVL FIRE ESE DIESEL.

AVL FIRE is a multifunctional software (the latest generation 3D CFD - Computational Fluid Dynamics). It is developed and continuously improved to solve the most demanding problems in terms of geometric complexity, physical and chemical. CFD is a branch of fluid mechanics which uses numerical methods and algorithms to solve and analyze problems that involve fluid flows.

In order to simulate the 3D flow, mixture formation, combustion and pollutant formation, the CFD modeling engine with direct injection is used. It also makes it possible to analyze the interaction between fuel and air movement admitted into the combustion chamber. The main advantage of modeling is that it minimizes the CFD simulation time, specifically the numerous studies executed in a relatively short time, based on computing power. The main interest in flow visualization is admitted cylinders extraction and analysis of virtual motion swirl and tumble motion [8].

Generally the level of information that can be provided by a visualization technique increases with the size of the input data.

3.2 Simulated Engine

The simulated engine is the OM611 220 CDI, 92 kW (Mercedes-Benz E220). This engine was chosen because the engine is available for further practical analysis. The simulations were made for different distances between the piston and the cylinder head at TDC, keeping the same compression ratio by slightly changing the profile of the piston bowl.

The steps that followed in AVL FIRE ESE DIESEL for a single simulation were:

- the General Data for the engine were introduced (engine layout, number of cylinders, compression ratio, crank radius, connecting rod length and piston offset);
- the piston parameters were introduced from an actual section of a piston;
- the injector properties were introduced;
- the mesh was generated (2D and 3D) and checked for irregular faces;
- the data for calculation were introduced (crank angle for the start and end of the simulation, engine speed, boundary conditions for the piston, liner and axis);
- fluid properties and initial conditions were introduced;
- the solver control parameter tree was completed (k-zeta-f turbulence model, energy output and no two-stage-pressure correction) and the maximum number of iterations was set to 100;
- output control was set to ensure proper 2D and 3D results;
- combustion model was set to Coherent Flame Model - ECFM-3Z;
- the spray was set, taking into account the fuel consumption at full throttle for the given engine speed;
- the simulation was started and the results were exported using Report Generator.

The first simulation was made for a 0.5 mm distance between the piston and the cylinder head, where a maximum velocity of the intake air was 86.778 m/s at 714 deg CA. The simulation can be seen in figure 2.

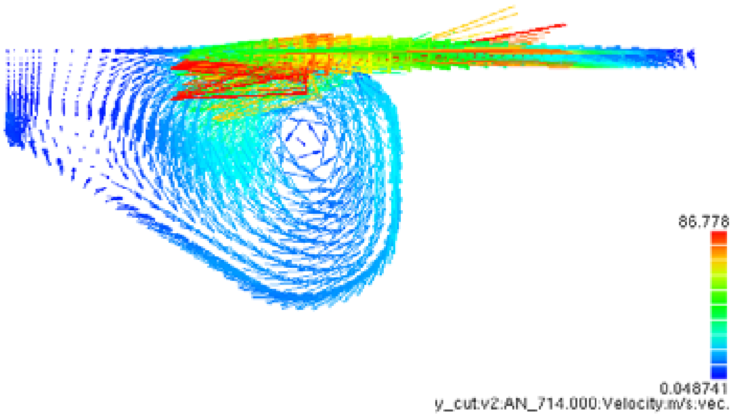


Fig. 2. Maximum velocity of the air for 0.5 mm gap at 714 deg CA

The second simulation was made for a 1 mm distance between the piston and the cylinder head, where the maximum velocity of the air was obtained at 710 deg CA (56.949 [m/s]), as shown in figure 3.

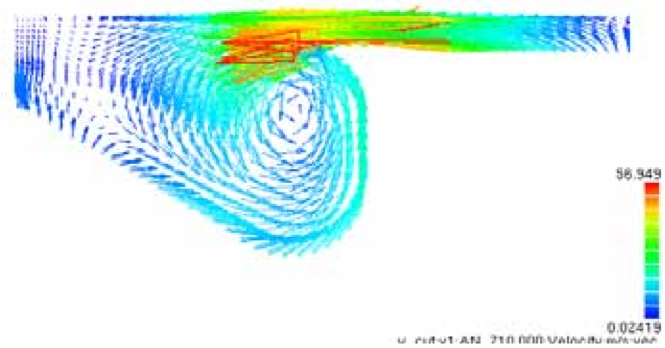


Fig. 3. Maximum velocity of the air for 1 mm gap at 710 deg CA

The third simulation was made for a 1.5 mm distance between the piston and the cylinder head, and the forth simulation for a 2 mm distance. The simulations show a velocity of 40.76 [m/s] at 710 deg CA, and 29.734 [m/s] at 708deg CA respectively (figures 4 and 5).

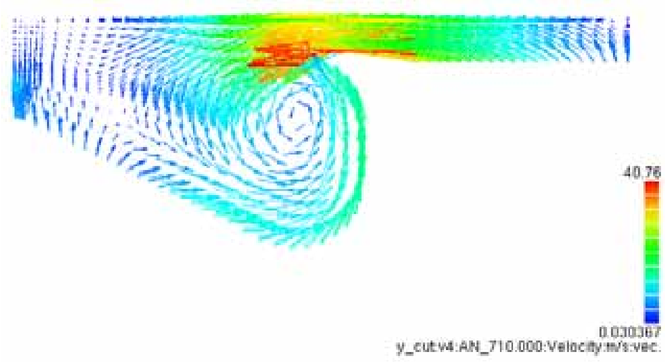


Fig. 4. Maximum velocity of the air for 1.5 mm gap at 710 deg CA

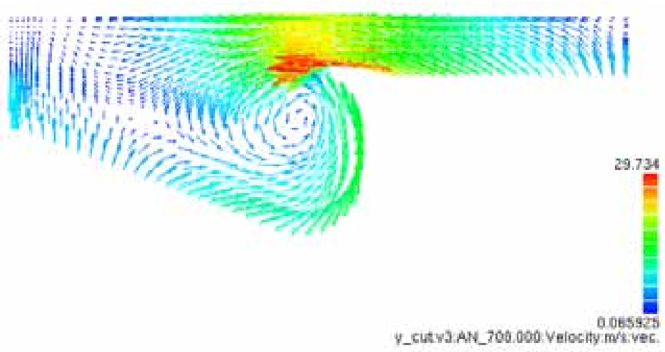


Fig. 5. Maximum velocity of the air for 2 mm gap at 708 deg CA

4 Results

The results of the simulations are presented in table 1 and figure 6. In Table 1, the maximum values for each simulation were outlined with a green background.

Table 1. Values for maximum velocity of the air inside the combustion chamber, in all four simulated cases

	Piston position [deg CA]	700	702	704	706	708	710	712	714	716	718
Max. velocity of the air in the combustion chamber [m/s]	TDC gap 0.5 mm	43,83	49,89	56,46	64,05	72,07	79,93	85,59	86,77	77,33	49,21
	TDC gap 1 mm	37,05	41,41	45,68	50,25	54,26	56,94	56,57	51,89	40,61	26,48
	TDC gap 1.5 mm	29,83	32,72	35,5	38,17	40,16	40,76	38,86	33,88	24,86	19,52
	TDC gap 2 mm	23,58	25,38	27,11	28,71	29,73	29,54	27,4	23,15	16,33	16,19

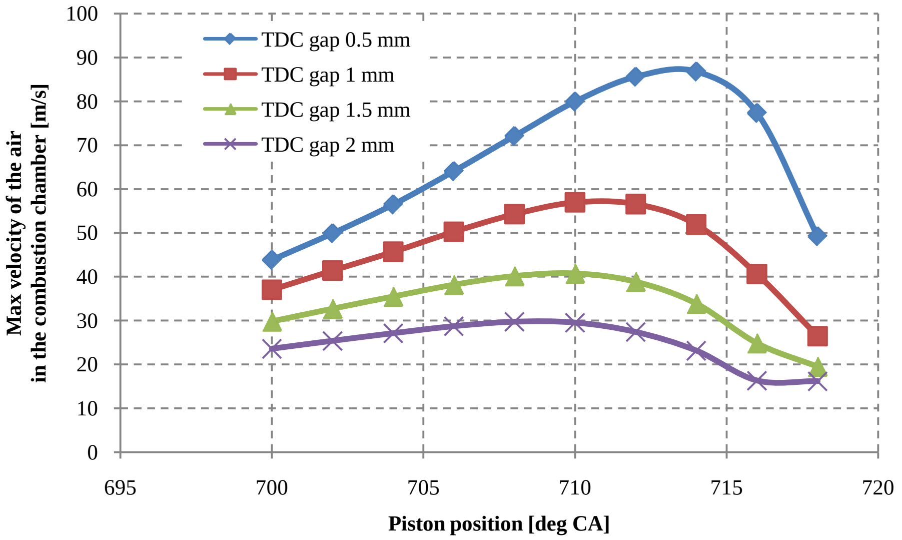


Fig. 6. Maximum velocity of the air in the combustion chamber for all simulation cases

The effective power was also monitored and the values are presented in Table 2.

Table 2. Values for effective power after the simulations with different gaps

Piston position [deg CA]	0.5 mm	1 mm	1.5 mm	2 mm
Effective power [kW]	87.975	91.899	88.058	76.122

5 Conclusion

When trying to modify the squish motion of the air, all velocities and turbulences change inside the combustion chamber, therefore the performance of the engine differs.

Knowing the maximum velocity of the air inside the combustion chamber, the effective power of the engine was also monitored to see consequent modifications. The maximum power of the engine was obtained at 1 mm gap. For a 1.5 mm gap the power was lower with 4.17% and for a 2 mm gap the power was lower with 17.16%. For a gap of 0.5 mm, the expected tendency of the power would be a small rise, but the simulation has shown a lower power with 4.26%, therefore the recommended value for the TDC gap is 1 mm.

Future studies will be made on the same engine to see the influence of the TDC gap and implicitly the influence of the squish on pollutant emissions.

Acknowledgements. The paper has been developed in the period of internal research project at Technical University of Cluj-Napoca, with the reference code number UTCN 11/1.2/2015, in which some specific activities were financed.

References

1. Bencherif, M., Mohand T., Abdelkrim L. Turbulence-combustion interaction in direct injection diesel engine, Thermal Science 18.1, 17-27, (2014)
2. Falfari, Stefania, Federico B., Gian M. B. Numerical Analysis of In-Cylinder Tumble Flow Structures–Parametric 0D Model Development. Energy Procedia 45, 987-996, (2014)
3. Ghodke, P. R., Jiwak G. S. Investigation of diesel engine for low exhaust emissions with different combustion chambers, Thermal Science 00 (2015)
4. Jovanovic, Z. S., Živanovic, Z. M., Sakota Z. B., Tomic, M. V., Petrovic, V. S., The effect of bowl-in-piston geometry layout on fluid flow pattern. Thermal Science 15.3 (2011)
5. Jovanović, Z. S., Petrovic, V. S., Tomic, M. V., The effect of combustion chamber geometry layout on combustion and emission. Thermal Science 12.1 (2008)
6. Payri, F., Benajes, J., Margot, X., Gil. A., CFD modeling of the in-cylinder flow in direct-injection Diesel engines. Computers & fluids 33.8 (2004)
7. Lumley, J., Engines – an introduction, Cambridge University Press, (1999)
8. *** AVL FIRE ESE DIESEL Tutorial, v. (2009)