

IMECE2010-40881

USE OF FLUENT FOR THE DEVELOPMENT OF A DI-SI ENGINE

Wim Vandenberghe
School of Engineering
University of Minho
4500-058 Guimaraes Portugal

Senhorinha F. C. F. Teixeira
School of Engineering
University of Minho
4800-058 Guimaraes Portugal

Jorge Martins
School of Engineering
University of Minho
4800-058 Guimaraes Portugal

Ricardo Falcao
School of Engineering
University of Minho
4800-058 Guimaraes Portugal

ABSTRACT

The recent surge of electric vehicles has put pressure on the development and manufacture of batteries. However, batteries are still expensive, bulky and heavy, creating the need for in-board electricity generation using an internal combustion engine, usually referred as “range extender”. This paper presents the initial development of a DI-SI engine to work as range extender, focusing on the interaction between fuel spray and airflow inside the combustion chamber.

To enable efficient combustion of lean and extra lean mixtures, a technique called stratified charge, is used. With direct injection spark ignition (DI-SI) engines it is important, under part load, to direct the fuel spray to the vicinities of the spark plug, enabling a fast and stable combustion of a lean mixture. A rich mixture region is created near the spark plug allowing an easy kernel formation and development.

There are three types of systems for “directing” the fuel spray towards the spark plug: wall guided, air guided and spray guided. The developed design is a mixture of wall and air guided systems and the idea is to inject the spray towards the piston crown and to divert it to the spark plug location by the barrel swirl existent within the combustion chamber at this time. The system development was carried out using CFD FLUENT code.

The study comprises three parts, the design of the components and its location (combustion chamber, piston crown, intake passage and injector location and aim), the air flow modelling and finally, the two phase modelling. A simple engine geometry and mesh were created in the Ansys CFD software. The air flow was considered to be transient, incompressible, Newtonian and

viscous turbulent. The turbulence model used was the standard $k-\epsilon$ model, since it is the most common, simple and well-known model of turbulence. The spray has been simulated using the Discrete Phase Model. The Lagrangian discrete phase model in Fluent™ follows the Euler-Lagrange approach, where the fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles through the calculated flow field. Preliminary results are now being obtained.

INTRODUCTION

To make the design of an internal combustion engine as efficient as possible, the goal is to obtain an optimization of the air-fuel mixture in the combustion chamber. Therefore the creation of turbulence is relevant in this process. The turbulence can be in form of swirl or tumble (Figure 1). Swirl rotates around the axis of the cylinder, while tumble does this around an axis perpendicular to the axis of the cylinder. [1] [2].

The engine used in this project will create tumble flow. This is possible by carefully designing the inlet to the engine.

Solid Works is used for the design of the engine. For the meshing of the model, Gambit is used and Fluent is used to do the simulation of the flow.

The purpose of the engine is to work as a “range extender” on electric vehicles. It works under the Miller-cycle [3], using direct injection, creating a stratified charge of lean mixtures during part load operation.

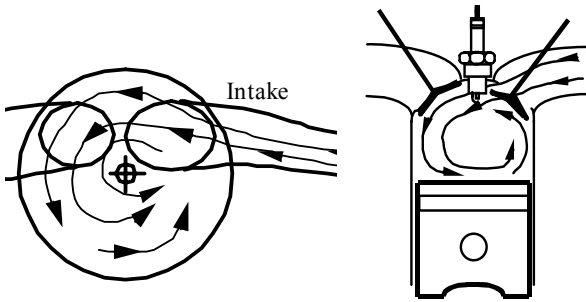


FIGURE 1. SWIRL AND TUMBLE

The injector is placed inside the cylinder head and injects the fuel, under high pressure, through a small nozzle. This happens when the piston is moving upwards, so when the air already inside the cylinder is under high pressure.

GEOMETRY AND MESH

GEOMETRY

Figure 2 shows the original design of one of the cylinders of the twin cylinder engine (Table 1).

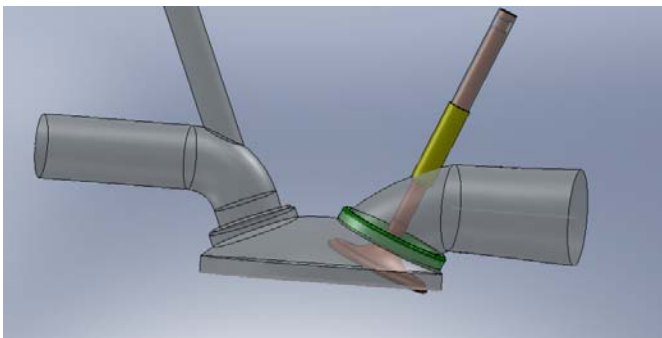


FIGURE 2. ORIGINAL ENGINE DESIGN

For the actual simulations, however, a simplified model is used, to make the meshing less complicated and the calculation faster. Since the focus only lies on the inlet and compression strokes, everything involving the exhaust stroke can be erased. Therefore the exhaust valve and ducts were eliminated.

TABLE 1 Engine physical parameters

Engine	
Bore	75mm
Stroke	66mm
Cylinder capacity	250cc
Connecting rod length	50mm
Piston stroke cutoff	30mm
Intake valve angle	67°

DYNAMIC MESH

To be able to run calculations on this model, it has to be meshed first. This was executed in GAMBIT, and since in this case there are moving parts involved (piston, valve) the dynamic mesh option was employed.

For the valve, combustion chamber and inlet channel, tetrahedral volumes have been used because the model geometry in that location is too complicated to use the simpler hexagonal volumes. Those volumes were used to mesh the piston. To simulate the piston moving down and upwards, its mesh creates new volumes, using the layering option in the dynamic mesh menu.

The boundary conditions have to be put in GAMBIT. This is needed to define where the fluid can pass and where it can't. Faces not defined will automatically be considered as walls. The inlet needs to be indicated as well and the piston has to be defined in order to be set to move.

To initiate the simulation in Fluent, the mesh has to be imported and the dynamic mesh has to be set. There are three methods: spring-based smoothing, remeshing and layering [4].

In the spring-based method, the entity of the mesh volumes could be compared to a sponge. The nodes of the mesh will move in case of a movement of a boundary. So the mesh volumes get bigger or smaller, according to the movement of the piston.

The remeshing method marks the faces that have to be remeshed according to the minimum and maximum length scales and the maximum cell skewness. Those faces are usually close to the moving boundaries.

The third method, layering, is used to add or remove layers of neighboring cells to a moving boundary. This is a motion suitable for linear movement [5] and so, this is used to simulate the movement of the piston. Layers are being added, when the piston goes down, subsequently they will be removed during the compression stroke.

Figure 3 shows the mesh of the model of the cylinder at crank angle 0°, 90° and 180°.

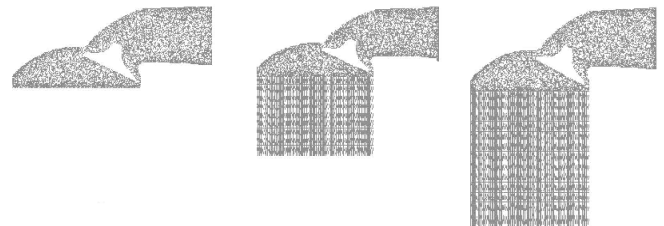


FIGURE 3. MESH MOTION AT 0°, 90° AND 180°

In the dynamic mesh menu, the in-cylinder model has to be activated to simulate the movement of the piston. There, the values that characterize the engine have to fill in.

MATHEMATICAL MODEL

FLUENT uses the finite volume method to solve the equations of the model. This is the most commonly used method in CFD. The software solves the equations of mass, momentum and turbulence in each control volume, guarantying conservation for each variable. The three-dimensional conservation equations for mass (1) and momentum (2) are:

$$\nabla \vec{v} = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) \quad (2)$$

\vec{v} : fluid velocity vector
 ρ : density
 P : static pressure
 $\vec{\tau}$: stress tensor

The method requires a discretization of the domain into control volumes. In the finite volume method, the partial equations are approximated by a set of algebraic equations that are written over the computational domain and then solved [D]. As solver, the SIMPLE algorithm was chosen. To have more precise results the equations were solved with the second order upwind. To simulate turbulence, the realizable k- ϵ model was used. This is a relatively new model, but it's the most common, simple and well known turbulence model. [6]

SPRAY SIMULATION

To simulate the injection, the *discrete phase model* (DPM) was used. This model is capable of simulating a discrete second phase, in addition to the solving of transport equations for the continuous phase. This second phase consists of spherical particles dispersed in the continuous phase. The trajectories of the droplets were computed in the Lagrangian phase. So were the exchange of heat, mass and momentum. The DPM has a lot of possibilities which allows fluent to calculate events such as particle separation and classification, spray drying, aerosol dispersion, bubble stirring of liquids, coal combustion and liquid fuel combustion. When simulation of turbulence for the particles is wanted, either the *stochastic tracking model*, or the *particle cloud model* needs to be activated. The first model includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods. The particle cloud model tracks the statistical evolution of a cloud of particles about a mean trajectory. The Gaussian probability density function (PDF), about a mean trajectory, represents the concentration of particles within the cloud. *Coupling between the discrete phase and continuous phase* is a model available, in stochastic tracking, for the generation of dissipation of turbulence in the continuous phase. The continuous phase always effects the

discrete phase, but with this model the effect of the discrete phase also has an impact on the continuum. This was realized by alternately solving the continuous and discrete phase equations until the solutions in both phases have stopped changing.

Another possible option to apply is the *droplet collision model*. This will give a more realistic result of the movement of the particles. This option estimates the number of droplet collisions and their outcome in a computationally efficient manner. Next, the injection had to be defined. The initial conditions that need to be put in were: starting position, velocities, number of particle streams, injection timing, flow rate, injection type.

SOLUTION PROCEDURE

The first step was to read the mesh file into FLUENT. After that it's advised to *check* the mesh. This grid check lists the minimum and maximum x and y values of the mesh and reports on a number of other grid features that are checked. If there would be reported any negative volumes, FLUENT would not be able to calculate the simulation. Time needs to be *transient* because in this case there are moving objects so the results are time dependent. For the model, in order to have turbulence, the *realizable k- ϵ model* was chosen. The next step was to choose the *materials*: air and fuel for the injection. Boundary conditions have been put in GAMBIT. In the *dynamic mesh* menu, dynamic mesh needs to be activated, *smoothing* and *layering* enabled and the *in-cylinder* model needs to be activated as well. There the correct data for the engine need to be filled in. In the dynamic mesh zones menu, the piston needs to be indicated as a rigid body, the "piston-full" profile needs to be chosen, in order to make the piston move the right way. The direction of movement needs to be indicated as well (Y-axis). To simulate the injection, the discrete phase model (DPM) needs to be activated, as mentioned before.

RESULTS AND DISCUSSION

GEOMETRY

Due to some problems with simulating the movement of the valve, the first simulation was made starting from a simplified geometry. In this new geometry, the focus laid on the moving of the piston and the injection of fuel during the compression stroke. For this new geometry the valve was removed and a ring was designed to replace it. This ring was considered as a pressure inlet, during the inlet stroke.

At crank angle 180°, this ring changes from a pressure inlet to a wall, so that during the compression stroke no fluid can escape through it.

This change of boundary condition was accomplished by creating an event at 180°, in the dynamic mesh menu.

The parameters for the injection (Table 2) were all provisional because in this case the emphasis was laid on the testing of the different options from the injection. [7] [8].

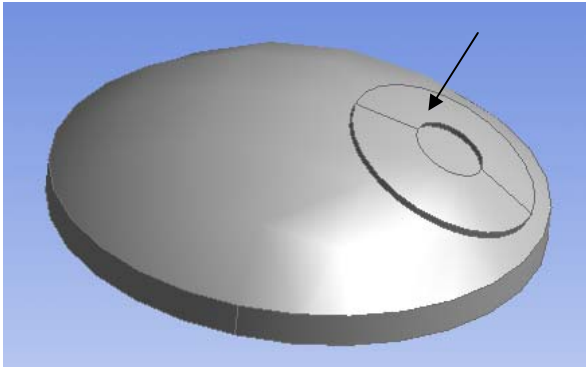


FIGURE 4. SIMPLIFIED GEOMETRY WITH INLET-RING

In different calculations, more and more options were added, so that in the end, turbulence of the injection, *two way turbulence coupling*, *droplet collision* and *droplet breakup* were all enabled at the same time.

TABLE 2 Simulation parameters

Injection	
x-position	-36mm
y-position	0mm
z-position	0mm
Start crank angle	250°
Stop crank angle	355°
x-axis	1
y-axis	-2
Velocity magnitude	80m/s
Cone angle	28°
Radius	0.00045mm
Total flow rate	0.00385 kg/s

The creation of the tumble flow is seen in figure 5 to 8. At crank angle 45° it is still in the initial phase, to keep on growing until 180° (Figure 8).

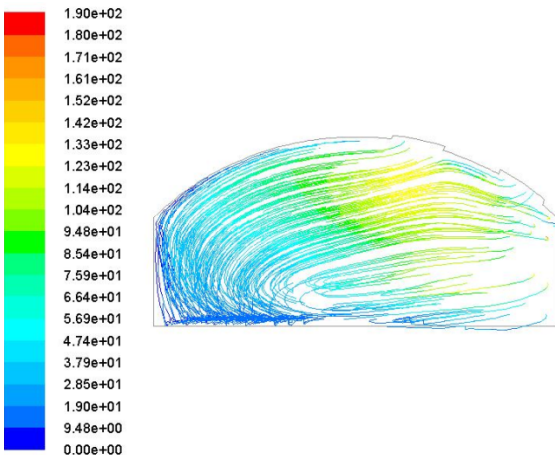


FIGURE 5. VELOCITY PATHLINES AT 45°

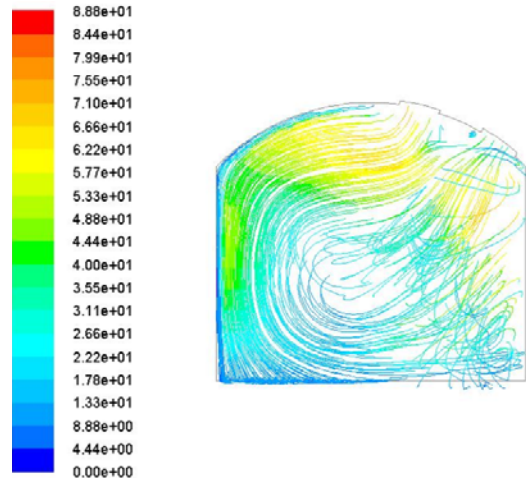


FIGURE 6. VELOCITY PATHLINES AT 90°

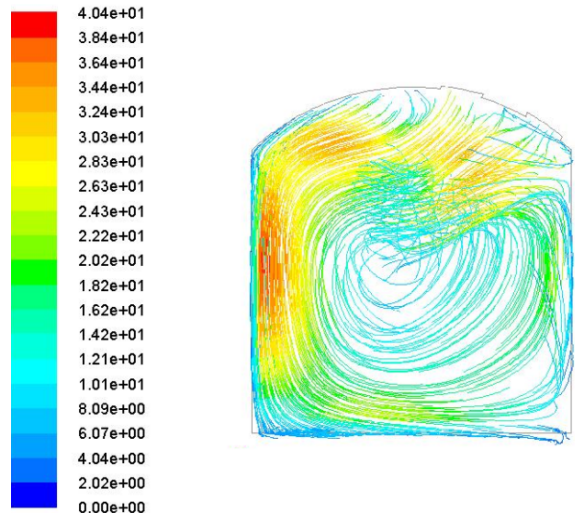


FIGURE 7. VELOCITY PATHLINES AT 135°

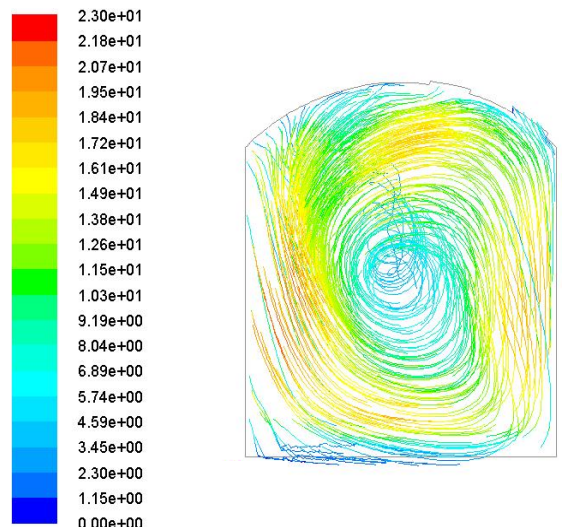


FIGURE 8. VELOCITY PATHLINES AT 180°

After the bottom dead center, the piston will move upwards, forcing the tumble to decrease in size (figure 9 and 10).

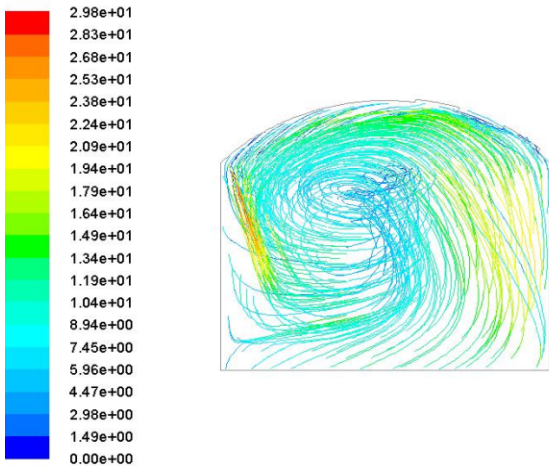


FIGURE 9. VELOCITY PATHLINES AT 270°

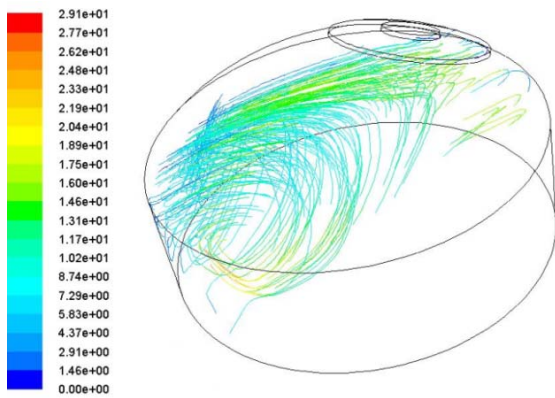


FIGURE 10. VELOCITY PATHLINES AT 310°

During the compression stroke, the tumble guides itself towards one part of the combustion chamber, as seen in Figure 10.

Figure 11 and 12 show the results of the injected particles. One can see how they move towards one half of the combustion chamber, due to the direction of the tumble.

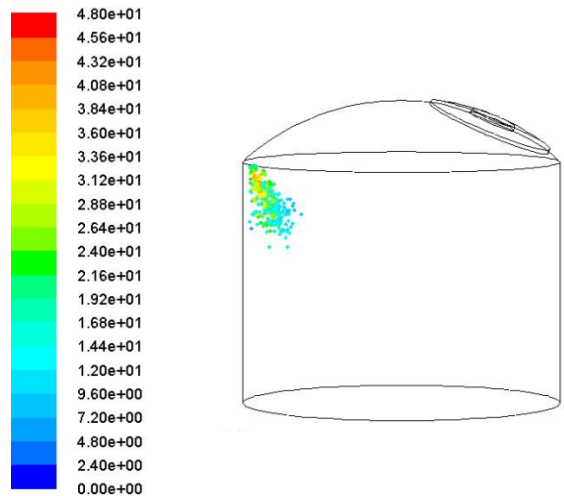


FIGURE 11. VELOCITY PARTICLE TRACKS AT 255°

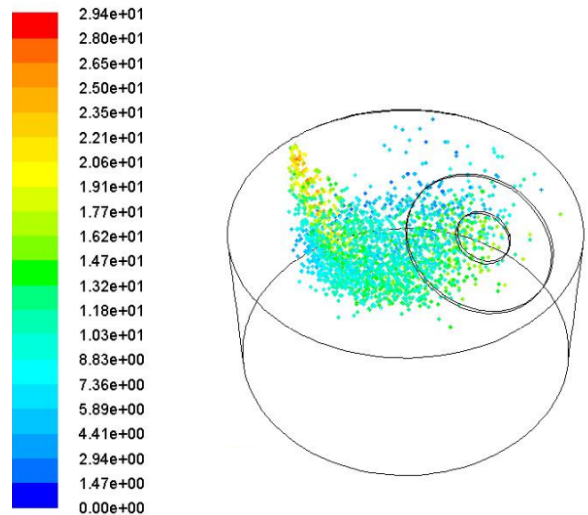


FIGURE 12. VELOCITY PARTICLE TRACKS AT 290°

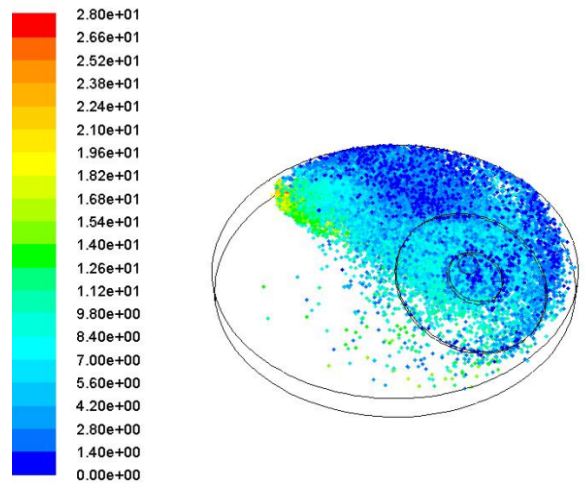


FIGURE 13. VELOCITY PARTICLE TRACKS AT 355°

As said before, this paper reflects the first phase of the design and development of a twin-cylinder DI-SI engine. The results obtained and presented are just the first attempt at the running of the model, including the use of moving parts and liquid spray injection. Therefore, they do not tend to express the design as an optimized solution. This will be a topic for a subsequent paper.

CONCLUSIONS

This paper shows the work leading to the design and development of a SI-DI engine, with the intention of optimizing its mixture preparation and stable combustion. With this purpose, the creation of tumble turbulence is enhanced within the intake flow.

Solid Works was used to draw the design, Gambit to create the mesh and Fluent calculates the flow of the fluids.

In the preliminary results, it is already clear how much the tumble affects the movement of the injected fuel particles.

The main purpose of this the work so far was to obtain knowledge on how to simulate piston movement and how to simulate the injection. The next goal is to make the valve moving and to create a new case with moving piston, injection and the moving valve.

ACKNOWLEDGMENTS

This research project was supported by POCI/EME/59186/2004 and MIT-Pt/EDAM-SMS/0030/2008.

REFERENCES

- [1] Martins, J., Teixeira, S.F.C.F. and Coene, S. Design of an inlet track of a small IC engine for swirl enhancement. Proceedings of COBEM 2009, Brasil (2009).
- [2] Jorge MARTINS, "Motores de Combustão Interna – 2nd edition", (ISBN: 972-8953-82-X) Publindústria, Porto, 2006
- [3] MARTINS, J.G., UZUNNEANU, K., RIBEIRO, B. and JASANSKY, O. Thermodynamic Analysis of an Over-Expanded Engine, SAE Technical Paper Series, nº 2004-01-0617, in 'Modeling of Spark Ignition Engines', SAE (ISBN Number: 0-7680-1366-6) (2004).
- [4] Ferreira, A.C.M., Teixeira, J.C.F., Teixeira, S.F.C.F. Contributions to the study of blood flow in the abdominal aorta and its Branches, ASME 2009.
- [5] Fluent 6.3 User's Guide, Fluent Inc, September (2006)
- [6] Teixeira, S.F.C.F., Martins, J. and Vanhaelst, J. Dynamic modeling of the inlet track of a small IC engine using Fluent. (in submission)
- [7] Wigley G., Goodwin M., Pitcher G., Blonde D.. Imaging and PDA analysis of a GDI spray in the near-nozzle region, Experiments in Fluids, Vol. 36, pp. 565-574 (2004).
- [8] Wigley G., Pitcher G., Law D., Schneider B. and Rogers S. Effect of Compression Pressure on the Spray Morphology of GDI Pressure-Swirl Injectors he Fifth International Symposium

on Diagnostics and Modeling of Combustion in Internal Combustion Engines, July 1-4, 2001, Nagoya (2001).