DESIGN OF AN INLET TRACK OF A SMALL I. C. ENGINE FOR SWIRL ENHANCEMENT

Jorge MARTINS, jmartins@dem.uminho.pt
Senhorinha TEIXEIRA, st@dps.uminho.pt
Stijn COENE, stijnxcoene@hotmail.com
Universidade do Minho, Escola de Engenharia, Guimarães, Portugal

Abstract. The purpose of this project was to re-design the inlet port of a small Internal Combustion Engine in order to enhance the production of turbulence by swirl. A good swirl promotes fast combustion and improves the efficiency. A small internal combustion engine was designed to be part of a very efficient vehicle to enter a consumption marathon. The engine should run at low speeds, in order to have low mechanical losses but the combustion should be fast, enabling good combustion efficiency. Therefore high turbulence should be produced prior to combustion within the cylinder, so swirl was induced by the inlet channel within the cylinder head. To perform this task the help of three software programs is required, Solid Works, Gambit and Fluent. The first was required to produce the CAD-geometry. To build the geometry it is important to bear in mind that there are some restrictions for casting and fuel injection. In the next step, Gambit meshes the geometry in the design by using a volume mesh. At last; Fluent calculates the flow within the engine by using a suitable turbulence model. Initially the current geometry was tested and proved to create low swirl, so the geometry was changed several times until reaching a good result in terms of generated swirl. The tests include just steady flow, where the air enters the inlet port and leaves the bottom of the cylinder continuously. The flow within the cylinder is examined at specific sections, namely at various heights of the cylinder (horizontal sections) and pathlines are also evaluated. The flow is calculated for various valve lifts at a specific engine condition.

Keywords: internal combustion engine; engine modelling; fluid dynamics

1. INTRODUCTION

Shell organises, on a yearly basis, an event called Shell-eco-marathon The purpose of this marathon is to race a vehicle on a particular track, at an average speed above 30 km/h with minimum fuel consumption. These vehicles are build by teams of students from different countries and universities. The University of Minho participates in this contest with its car, EconomicUM (Fig. 1). A new engine is being developed for this car, called the UMotor (Fig. 2). Part of the development of the head for this engine is herein revealed.
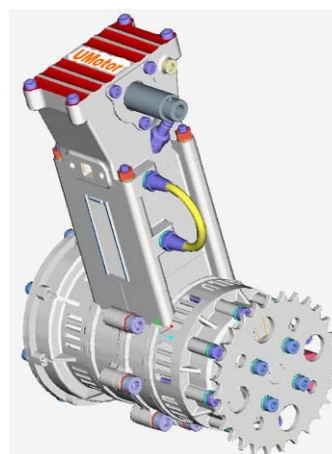

Figure 1- EconomicUM

Figure 2 - UMotor

To enhance the efficiency of an engine it is important to optimize thermal efficiency, which is obtained at the highest possible compression ratio. But if the compression ratio is too high, there is a chance to have knock, which should be avoided at all cost. A solution for this problem is to promote rapid combustion, to reduce the time available for the self-ignition to occur.

To promote rapid combustion, sufficient large-scale turbulence (kinetic energy) is needed at the end of the compression stroke because it will result in a better mixing process of air and fuel and it will also enhance flame development. However, too much turbulence leads to excessive heat transfer from the gases to the cylinder walls, and may create problems of flame propagation (Blair, 1999, Stone, 1989, Lumlay, 2001). The key to efficient combustion is

to have enough turbulence in the combustion chamber prior to ignition. This turbulence can be created by the design of the intake port (Blair, 1999).

There are two types of structural turbulence that are recognisable in an engine; tumbling and swirl (Fig.3). Both are created during the intake stroke. Tumble is defined as the in-cylinder flow that is rotating around an axis perpendicular with the cylinder axis.
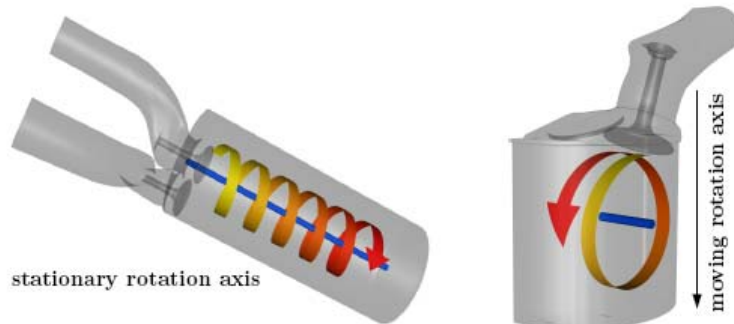

Figure 3 - Difference between swirl and tumble [Garth, 2007]

Swirl is defined as the charge that rotates concentrically about the axis of the cylinder. While tumble is generally existent in 4-valve pent roof combustion chambers, swirl is more common on 2-valve heads, such as the one on this engine. If the inlet flow is brought into the cylinder with an initial angular momentum, it will create swirl (Blair, 1999, Stone, 1989).

As explained above, turbulence should not be too high, because otherwise it could lead to excessive heat transfer from the cylinder contents. Furthermore, a high swirl is also not desired, as the kinetic energy for the flow is obtained at expense of a reduced volumetric efficiency. An optimal swirl ratio is not only good for optimum combustion, but also for an optimal emission reduction (Stone, 1989).

There are two ways to create swirl, by valve design and by intake design (Heywood, 1988). The design needs a shape that creates turbulence during the intake stroke, and still lasts during the compression stroke.

There are two kinds of designs commonly used; these are the direct port and the helical port. We are aiming at producing swirl by developing a helical intake port. The idea of a helical port is that the air is brought in rotation prior to entering the cylinder, namely in the inlet port. The rotation is achieved by forcing the air flow around the valve stem, so that there is angular momentum created about the cylinder axis when the charge enters the cylinder.

Helical ports normally have a higher discharge coefficient than direct ports if the swirl levels are equivalent. Another advantage of helical ports is that they are not that sensitive to misplacements that can occur in casting. Also, a helical port produces better swirl than a direct port.

The purpose of this work was to create a suitable intake design; which would improve the engine efficiency by means of improving the generation of swirl. There is a further constrain for the intake port design which is the manufacturing (casting) requirements.

The intake design was developed using CFD techniques, which involve the use of three main software packages; Solid Works, Gambit and Fluent.

2. OPTIMAL GEOMETRY DESIGN

To make the head, sand casting is used to produce the parts. The material used for the cores and the moulds is produced by rapid prototyping or by using a core box, which is made of aluminium or wood and created on a 4-axis CNC machine. But there are some restrictions to build this:
- To produce the design it is not possible to produce very sharp angles because for milling the core box, the tool can't reach sharp angles.
- For the design, it is not possible to have undercuts, otherwise you can't open the moulds.
- Walls that are too thin to be casting should be avoided.

There are also some restrictions on the design of the inlet track:
- The casting restrictions apply.
- The fuel should be injected towards the inlet valve, impacting over its back. As the inlet is curved to create swirl, care must be taken to allow a straight path from the injector to the valve.
- Care must be taken to reduce pressure losses within the helical duct.

The optimization of the design was continuously improved using CFD studies in a sequence of several enhanced geometries, leading to an optimal final design (see Coene, 2008, to assess the full development). On the final (sixth) geometry (the only presented in this paper), the optimization of the design was applied in terms of design for manufacturing and swirl enhancement.

The cylinder capacity has $25cm^3$ and the flow was modelled for an engine speed of 4000 rpm, with a valve lifts of 1.5mm, 2.0mm and 2.5mm.

The dimensions of the engine are:
- Bore: 30,98mm
- Stroke: 33mm
- Cylinder capacity: 24.9cc
- Inlet area: 140.57mm²

After that the design was completed, it was possible to make the final assembly, which consists of (Fig. 4): the intake design with cylinder (grey); valve seat (yellow); valve (red) and valve guide (green).

The principle of this geometry is that the air flow first will be deflected to the right when it passes the bend (Fig. 5). Then, the air flows around the valve, creating an angular momentum for the intake flow.
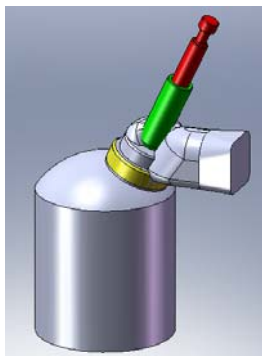

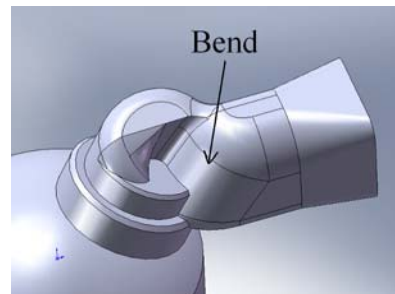Figure 4 - Assembly with the different parts

Figure 5 - Working of the helical port

3. MESH GENERATION

There is a major difference between the design for the production of moulds for casting and for the flow study. On the latter, the volumes such as valve guide and valve seat should be removed from the design (Fig. 6-a).

The meshes were created in Gambit, using the volumetric tetrahedral meshes, because the intake part is rather complex. The geometry is meshed for three different valve lifts: 1.5mm, 2.0mm and 2.5mm. The result of the mesh is given in Fig. 6, where it is also included a longitudinal section of the mesh (b). The results for a valve lift of 1.5mm, for 2.0 mm and 2.5mm are similar, only the position of the valve changes.

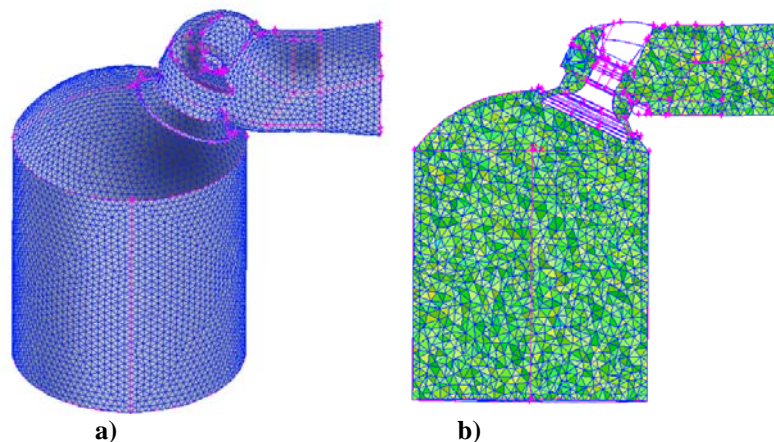**a)**   **b)**

Figure 6 - The meshed volume

The mesh has 146 270 elements. The quality of the mesh has been tested in terms of EquiAngle skew, EquiVolume skew and aspect ratio. The worst element in the mesh has an EquiAngle skew of 0.97 and is lower than the limit of 0.98. The worst element for EquiVolume skew is 0.99. The worst element has an aspect ratio of 109. These results show that the quality of the mesh is very good.

To examine the created design on swirl, a steady flow case is considered. In this simulation, there is no piston in the cylinder so that the bottom of the cylinder is open. At the intake, there is blow air blown through the cylinder and the air leaves the cylinder through its bottom. For that reason the bottom of the cylinder is defined as outflow in the boundary conditions panel. For the inlet of the air the velocity inlet is taken, because there you can define the flow rate by velocity and direction of the air at the inlet (in Fluent). The velocity inlet is only valid for incompressible flows, and the flow is supposed to be incompressible (Mach number lower than 0.3). It is shown that the results for this strategy are valid in comparison to experimental tests [Mahmood, 1996].

4. MODELLING:FLUENT

The domain discretization into control volumes is necessary for the finite volume method used in the FLUENT. This CFD software solves the equations of mass, momentum and the turbulence in each control volume, guarantying conservation for each variable.

The three-dimensional conservation equations for mass (equation 1) and momentum (equation 2) are:

$$\nabla \vec{v} = 0 \tag{1}$$

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

where $\vec{v}$ is the fluid velocity vector, ρ is the density, p is the static pressure and $\overline{\overline{\tau}}$ is the stress tensor.

The model assumes the fluid to be incompressible, steady and turbulent. For the turbulence model, the RNG k-ε model was selected, because it is proved that the predictions by this model display reasonable agreement with the experimental data [Xu, 2001]. The swirl dominated option is taken in this model. The boundary conditions were already defined in Gambit, but in Fluent it is needed to prompt the velocity and direction at the inlet. This velocity at the inlet was taken as 23m/s.

It was used the standard discretization method for the pressure and the second order upwind method to the momentum, turbulent kinetic energy and turbulent dissipation rate. The partial differential equations are approximated by a set of algebraic equations over the computational domain and the algebraic equations are then solved. Solutions are obtained iteratively using the segregated solver with the SIMPLE algorithm, and the convergence is accepted when the residuals are of 1E-05.

5. RESULTS

Results were calculated for valve lifts of 1.5mm, 2mm and 2.5mm, as these would be the values that may be used as maximum lift. Five cross sections were evaluated on different places in the cylinder.

In each cross section values for swirl number and swirl component were calculated. Two different values are calculated to assess the swirl intensity. The first, the swirl number, is the ratio of angular momentum to the axial momentum:

$$S = \frac{\text{angular momentum}}{\text{axial momentum}} = \frac{\int r \omega_t \vec{v}.d\vec{A}}{\overline{R} \int u \vec{v}.d\vec{A}} \tag{3}$$

This angular momentum is calculated in the centre of the swirl. (not on the cylinder axis).

The other is herein called the "swirl component" and is the swirl parameter relevant for experimental tests with a paddle wheel placed in the axis of the cylinder:

$$C_{sp5} = \frac{B^2 S_t \omega_s}{4Q} \tag{4}$$

The angular velocity is calculated in the centre of the cylinder (on the cylinder axis). The calculations are made in cross sections of the cylinder (surfaces perpendicular to the cylinder axis).

Pathlines were also evaluated, in order to better understand the flow.

5.1. 1.5mm valve lift

Fig. 7 shows five cross sections (from the top of the cylinder to its bottom). This gives a good view on the evolution of the swirl within the cylinder. For the case where the inlet valve is 1.5mm open (from its seat), it is clear that there is sufficient swirl. The vectors represent the velocity magnitudes (m/s) in the mesh points.
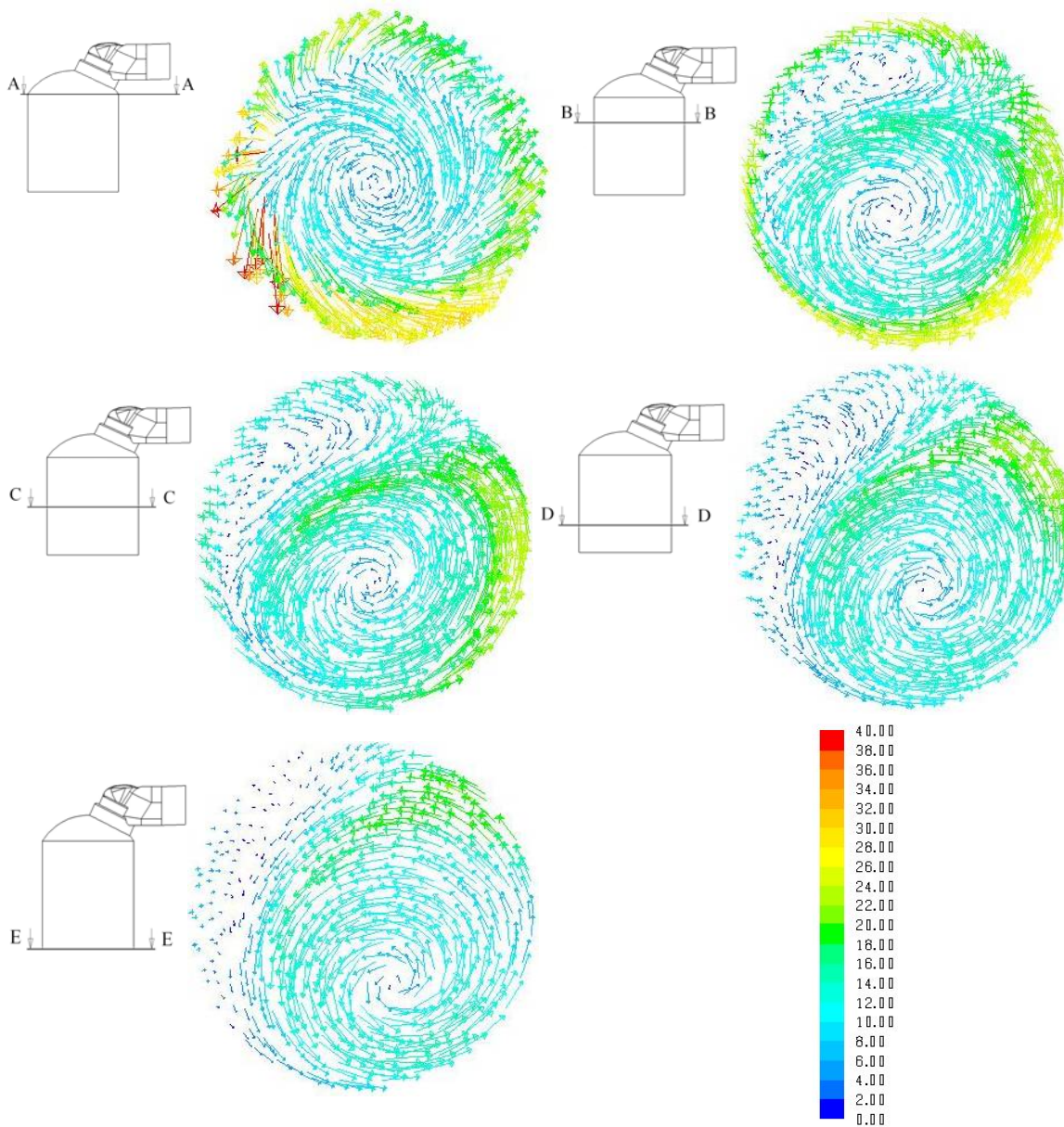
Figure 7 - Velocity field along the cylinder (1.5mm valve lift)

The results of the swirl and swirl parameter are in Table 1. For all the valve lifts, it was not possible to compare the numbers for the outflow with the other cross-sections for section E-E.

Table 1 - Swirl values (1.5mm valve lift)

Cross section	Swirl number	Swirl parameter C_{sp}
A-A	1,16	1,16
B-B	1,12	1,15
C-C	1,10	1,12
D-D	1,09	1,02

At the top of the cylinder, the centre of the swirl is aligned with the centre of the cylinder. In cross-section B-B a second swirl starts to develop near the side of the cylinder. The consequence is that the main swirl is not in the centre of

the cylinder anymore, producing a difference between the swirl number and the swirl parameter (calculated on the centre of the cylinder). The decrease in swirl is the highest between cross section A-A and B-B.

With pathlines (Fig. 8) one can follow the track of flow particles. The pathlines are coloured by velocity vectors (m/s). The pathlines are swirling through the geometry, showing the direction of the moving fluid. The existence of the swirl is evident from this figure.
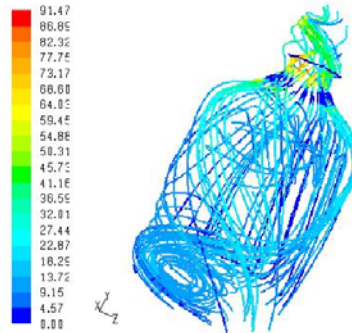

Figure 8 - Pathlines (1,5mm valve lift)

5.2. 2.0mm valve lift

A similar pattern can be found for a wider valve opening (Fig. 9), although of somehow lower intensity.
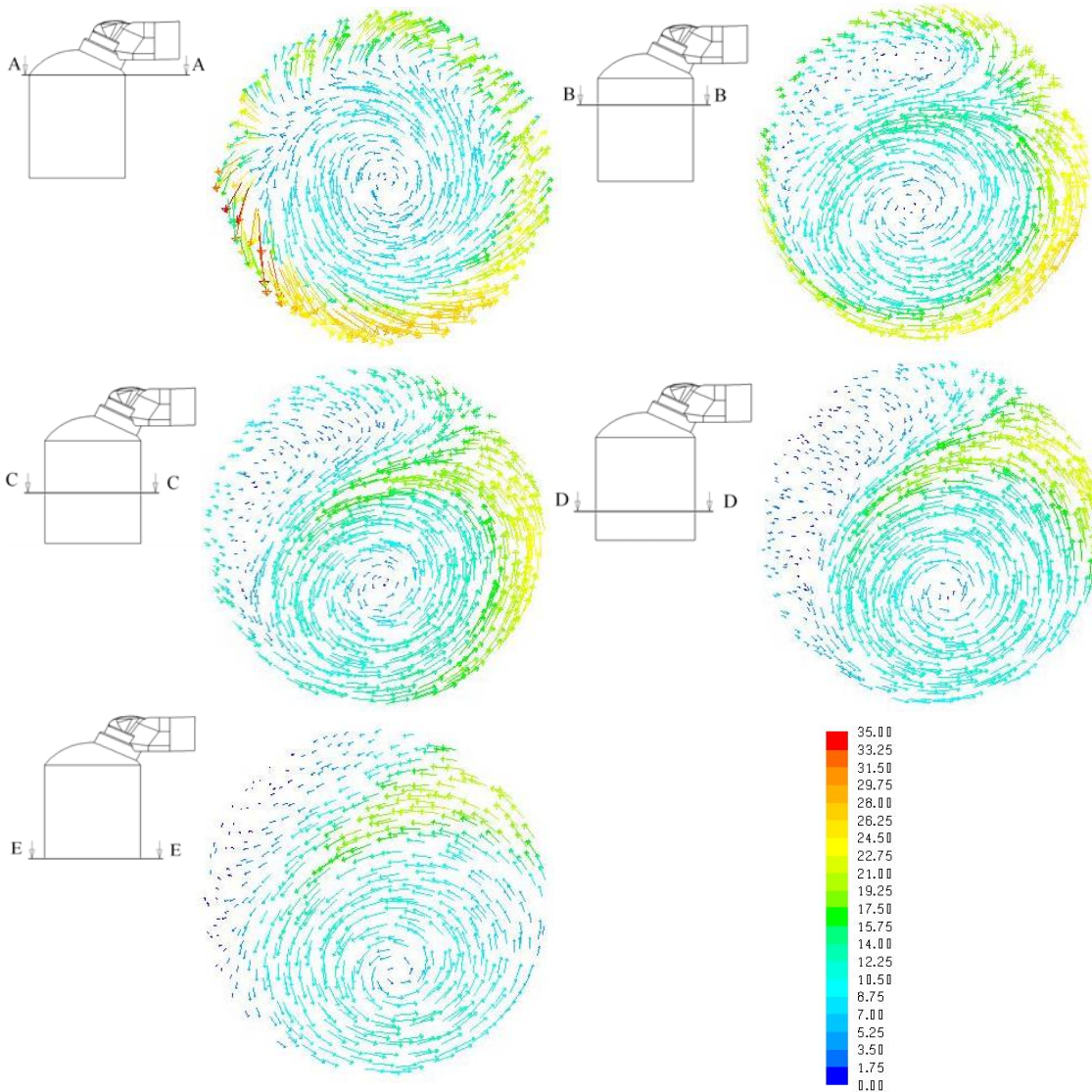

Figure 9 - Velocity field along the cylinder (2.0mm valve lift)

The results of the swirl and swirl parameter can be seen in Table 2.

Table 2 - Swirl values (2.0mm valve lift)

Cross section	Swirl number	Swirl parameter C_{sp}
A-A	1,07	1,06
B-B	0,99	1,02
C-C	0,98	0,99
D-D	1,00	0,93

The cross sections for a valve lift of 2mm have similar patterns as with a valve lift of 1.5 mm. There is also a second swirl developing at the section B-B, and the centre of the main swirl moves away from the centre of the cylinder. The only difference is that the velocities are a bit lower, which results in a lower swirl.

There is a large decrease in swirl from cross section A-A to B-B, higher than with a valve lift of 1.5mm.

When looking at the pathlines within the cylinder (Fig. 10), it is possible to realise that there is a lot of turbulence in the cylinder.
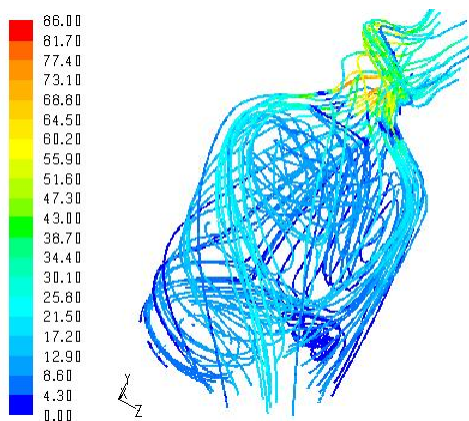

Figure 10 - Pathlines (2.0mm valve lift)

5.3. 2,5mm valve lift

Again, for the valve lift of 2.5 mm, a pattern occurs (Fig. 11) similar to the previous two cases, with a second swirl developing near the edge of the cylinder. However, this swirl starts to develop right at the top section (A-A) of the cylinder.

The velocities are weaker that at the smaller valve openings, leading to swirl values almost 2/3 of the value for the 1.5mm valve opening (Table 3, Fig. 11).

Table 3 - Swirl values (2.5mm valve lift)

Cross section	Swirl number	Swirl parameter C_{sp}
A-A	0,73	0,74
B-B	0,65	0,68
C-C	0,70	0,66
D-D	0,69	0,63

The swirl in cross section A-A doesn't start exactly in the centre of the cylinder anymore, and consequently the swirl number and the swirl component have not the same value at the top section A-A.

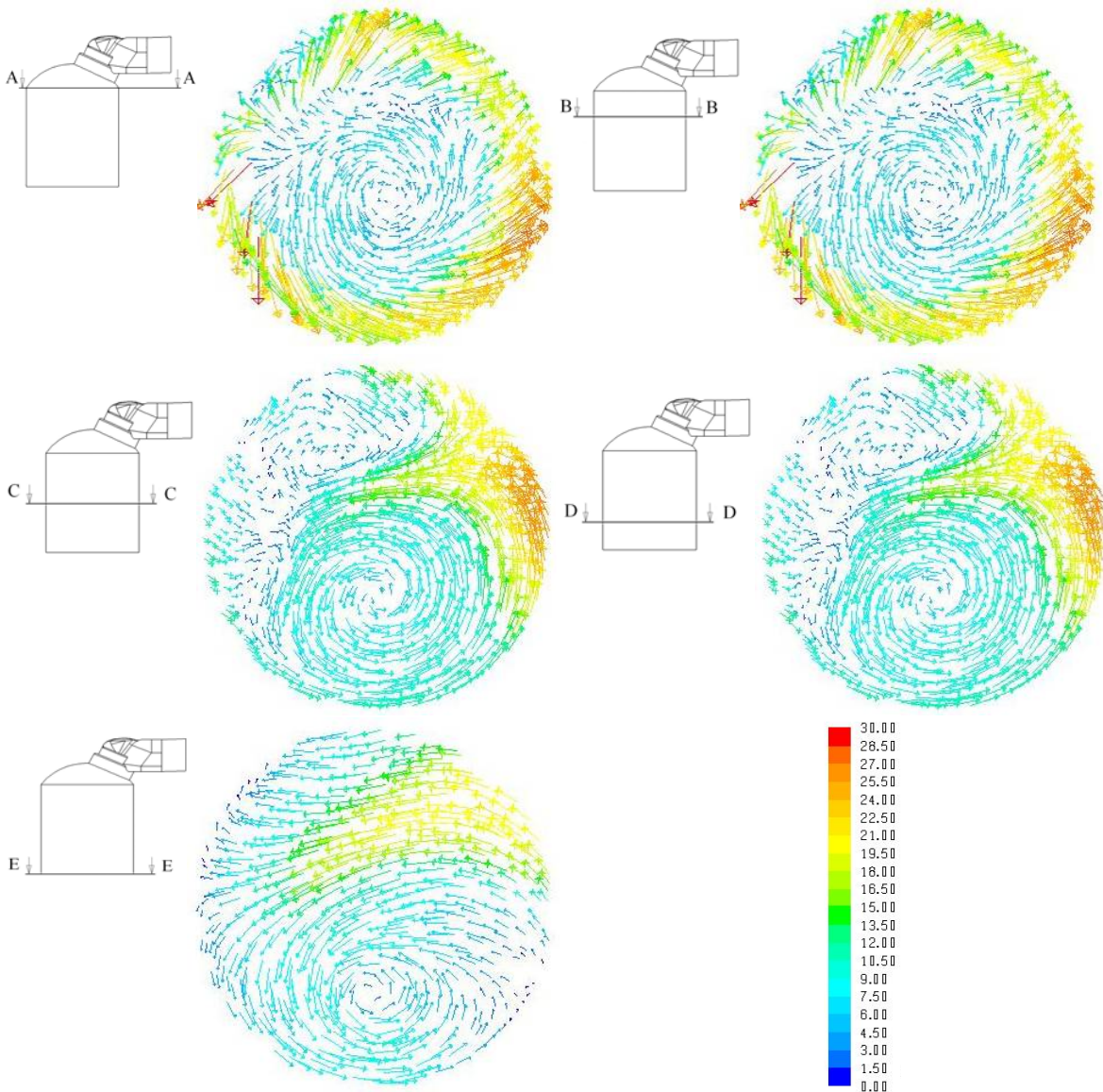The pathlines (Fig.12) show the usual pattern of a rotating flow inside the cylinder.

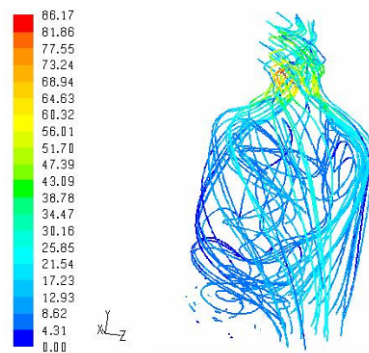Figure 11 - Velocity field along the cylinder (2.5mm valve lift)

Figure 12 - Pathlines (2,5mm valve lift)

6. CONCLUSIONS

A computer model created in GAMBIT was used for FLUENT simulations. Simulations were made to measure the swirl on a spark ignition internal combustion engine.

Looking at the cross sections of the three tests (inlet valve opened at 1.5, 2.0 and 2.5 mm), it can be seen that the progress of the swirl with the different valve lifts is the same. The swirl starts at cross section A-A in the centre of the

cylinder. In cross section B-B a second swirling starts to settle pushing the main swirl away from the centre of the cylinder. This second swirling flow stays on approximately the same location until the flow reaches the bottom of the cylinder.

When looking at the values of the swirl, an increase of valve lift results in a decrease of both the swirl number and the swirl component. The best results for swirl are reached for a valve lift of 1.5mm. This is as expected, because with a valve lift of 1.5mm there is less area between valve and valve seat where the air can pass resulting in higher speed and angular momentum.

Comparing the swirl number with the swirl component, the values of both numbers at the top are almost the same. This is because both numbers are calculated in the same way, but the swirl number is calculated in the centre of the swrl and the swirl component is calculated in the centre of the cylinder (axis). At the top of the cylinder the centre of the swirl is in the centre of the cylinder and thus the values are approximately the same.

Swirl numbers around the unit (1) produce the required amount of swirl, so this engine will be designed to have a valve opening between 1.5 and 2.0 mm, depending on the pressure drop it creates.

7. ACKNOWLEDGEMENTS

8. REFERENCES

Blair, G. P. , 1999, *Design and Simulation of Four-Stroke Engines*, Warrendale (U.S.A), SAE

Coene, 2008, *Inlet Design for Optimal Swirl in an IC Engine*, MSc thesis, University of Minho, Portugal

Fluent 6.3 User's Guide, Fluent Inc, September 2006

Heywood, J.B., 1988, *Internal Combustion Engine Fundamentals*, McGraw-Hill

Lumley, J. L., 2001, *Early work on fluid mechanics in the IC engine*, Annual Reviews, Annual Review of Fluid Mechanics Volume 33, Issue 1

Mahmood, Z., Chen, A., Yianneskis, M. and Ganti, G., 1996, *On the structure of steady flow through dual-intake engine ports*, International journal for numerical methods in fluid, vol. 23, 1085-1 109

Smits, J. J. M. , 2006, *Modeling of a fluid flow in an internal combustion engine*, Eindhoven

Stone, R. , 1989, *Motor Vehicle Fuel Economy*, Middlesex (England), Macmillan education LTD

Thien, G., 1965, '*Entwicklungsarbeiten an Ventilkanalen von Viertakt – Dieschmotoren*', Osterreichische Ingenieur Zeitschrift. Jahrgang 8, Heft. 9. (in German)

Xu, J, Yao, Q., Cao, X., Cen, K. , 2001, *Algebraic stress model with RNG ε- equation for simulating confined strongly swirling turbulent flows*, Journal of Thermal Science, Science Press, Volume 10, Number 1