24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

# COBEM-2017-1131
# NUMERICAL STUDY OF FLUID DYNAMIC FLOW WITHIN AN INTERNAL COMBUSTION ENGINE CYLINDER

**Ana Marta de Souza**
**Antônio César Valadares de Oliveira**
**Enrico Temporim Ribeiro**
**Francisco José de Souza**
**Marcelo Colombo Chiari**
Universidade Federal de Uberlândia -UFU
Av. João Naves de Ávila, 2121, 38408-100 – Uberlândia, MG - Brazil
amsouza@ufu.br, antoniovaladaresmec@gmail.com, enrico.tribeiro@gmail.com, francisco.souza@ufu.br, m_chiari@hotmail.com

***Abstract.*** *Despite the significant advances in the automotive industry in recent year, there are still improvements to be achieved, especially concerning the conversion of chemical energy into mechanical energy, which is a maximum of nearly 40% currently. A larger engine efficiency would also result in lower fuel consumption and lower emissions. The development of a rapid and complete combustion is one of the factors that could contribute to improve the engine efficiency. It is known the flow within the cylinder is an important parameter and the turbulence flow in the combustion chamber may increase the rate of flame spread significantly, contributing to efficient combustion. According to Heywood (1988), the burning of fuel in engines of Cycle Otto is more influenced by the movement of charge in the cylinder than the fuel properties (which does not mean that does not require both). Turbulence may be created or enhanced by one or more patterns inside the cylinder, such as swirl and tumble. In this context, the main goal of this work is to investigate the influence of different geometries of air intake port on the cylinder flow. Techniques of Computational Fluid Dynamics (CFD) were applied through the AVL/FIRE software. The results allowed observe that small changes in the air intake port are able to promote different flow patterns, favoring the formation of swirl or tumble.*

***Keywords:*** *Internal combustion engine, Computational Fluid Dynamic (CFD), swirl, tumble*

## 1. INTRODUCTION

Internal combustion engines are thermal machines whose main function is to transform the chemical energy of the fuel into mechanical energy, and this process occurs through thermodynamic expansion and compression cycles. The proper functioning of the engine relies on several variables, among them the quality of the burning of the fuel. One of the factors influencing the combustion process in electric spark ignition engines is the movement of the fluid inside the combustion chamber. The generation of turbulence in suitable regions favor the mixing of the air with the fuel, providing rapid and quality combustion and, consequently, an improvement in engine performance.

In this context there is the proposal of research and search of different geometries of admission channels that give a greater efficiency to the system, and consequently reduce the consumption of fuel and the emission of pollutants in the atmosphere from the inadequate combustion. For an initial analysis, the use of computational methods of simulation is of interest, since they allow to analyze the dynamics of fluids in a cheap and fast way in relation to the experimental tests. Therefore, it is proposed to use the AVL FIRE software, which is specific to the area of CFD analysis in motors and whose use for carrying out studies such as the one proposed here is still recent.

## 2. BIBLIOGRAPHIC REVIEW

The turbulence inside the engines is directly related to the dimensions and geometry of the cylinder, the valve and the intake duct. The design of the engine head is of vital importance in the mixing process where the geometry and dimensions of its components are responsible for ensuring that the mixture is in the ideal firing conditions. Various formats of intake portholes can be developed to condition flows inside the cylinder, generating or potentiating the turbulence at the appropriate time. Figure 1 shows the main flow patterns, called swirl and tumble.

The swirl is the horizontal flow of charge inside the cylinder around its axis. This flow can be produced either by positioning the inlet duct to one side of the cylinder axis or by adding flow directing elements capable of generating the same effect. Spiral or propeller inlet conduits are used to promote swirl enhancement, which generally remains during the admission, compression and expansion processes. The tumble is the rotational flow of the charge within the cylinder about a horizontal axis, which normally extends to the final compression stage. This flow is produced by the positioning of the inlet duct and the inclination of the inlet valve(s).

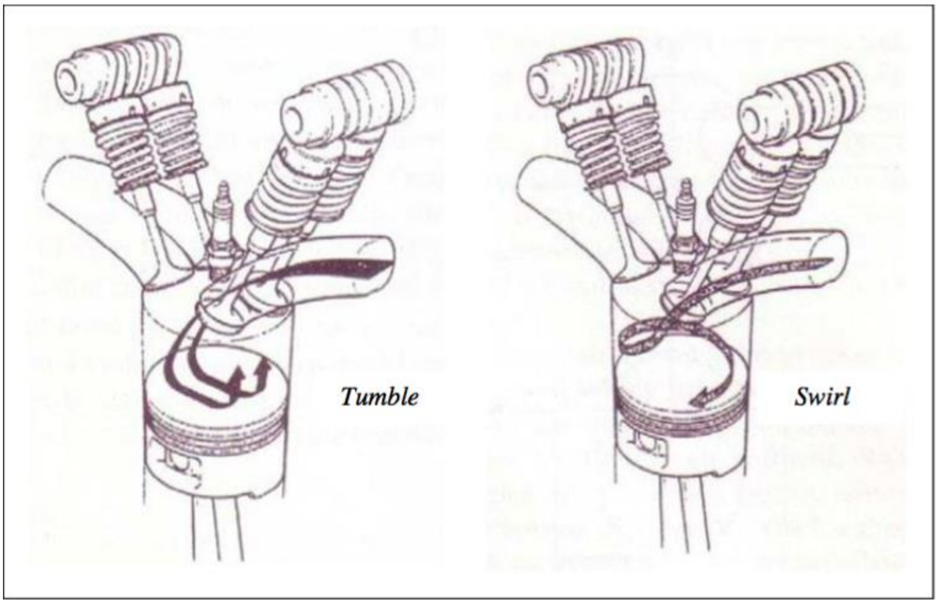


Figure 1. Fluid movements inside the cylinder: swirl and tumble (Lumley, 1999)

Normally, higher intensities of rotational flow imply improvements in power, torque and specific consumption. However, the main strategies to increase the intensity end up reducing the volumetric efficiency, so that not always increases in the vortices lead to improvements in the combustion efficiency. Thus, in an analysis for the optimization of an engine, all variables must be investigated carefully.

One way of quantifying the large swirl and tumble movements within the cylinder is by means of the swirl and tumble coefficients, which are each a comparison between the angular velocity of the rigid body rotation with the same amount of angular motion as the distribution of the speed presented by the flow, and the angular velocity of the rotation of the rigid body of the motor shaft.

The Equations (1) and (2) represent respectively SC (Swirl Coefficient) and TC (Tumble Coefficient).

$$SC = \frac{\overline{V_Z}}{2 \cdot \omega_E} \tag{1}$$

$$TC = \frac{\overline{V_Y}}{2 \cdot \omega_E} \tag{2}$$

Where $\overline{V_Z}$ and $\overline{V_Y}$ are the components $\vec{w}$ and $\vec{j}$ , respectively, of the average vorticities of the analysis planes and $\omega_E$ is the angular velocity of the crankshaft.

Equations for the calculation of the mass flow based on the imposed axis rotation:

$$\omega_E = \frac{C_{mp}}{2 \cdot s} \tag{3}$$

$$C_{mp} = \frac{\dot{m}}{\left[\frac{\pi \cdot D_c^2}{4} \cdot \rho_i\right]} \tag{4}$$

Where $\omega_E$ is the angular velocity of the crankshaft, $C_{mp}$ is the average piston speed, $s$ is the piston stroke, $\dot{m}$ is the mass flow, $D_c$ is the cylinder inside diameter and $\rho_i$ is the is the density of air inside the cylinder.

## 3. COMPUTATIONAL PROCEDURE

For the analysis previously mentioned three models were created in order to best identify the differences between each case. All models are based on the same cylinder and pipes volumes, however there is a difference in the angular positioning of the inlet and outlet ducts. All dimensions are described in table 1 and the inclination of the conduits can be seen in figure 2.

Table 1. Dimensions of each model.

| Case | Diameter of cylinder (mm) | Cylinder height (mm) | Cross-sectional area of conduits (mm$^2$) | Inclination of conduits |
|---|---|---|---|---|
| 1 | 95,5 | 86,7 | 907,92 | 0° |
| 2 | 95,5 | 86,7 | 852,71 | 20º |
| 3 | 95,5 | 86,7 | 695,64 | 40º |

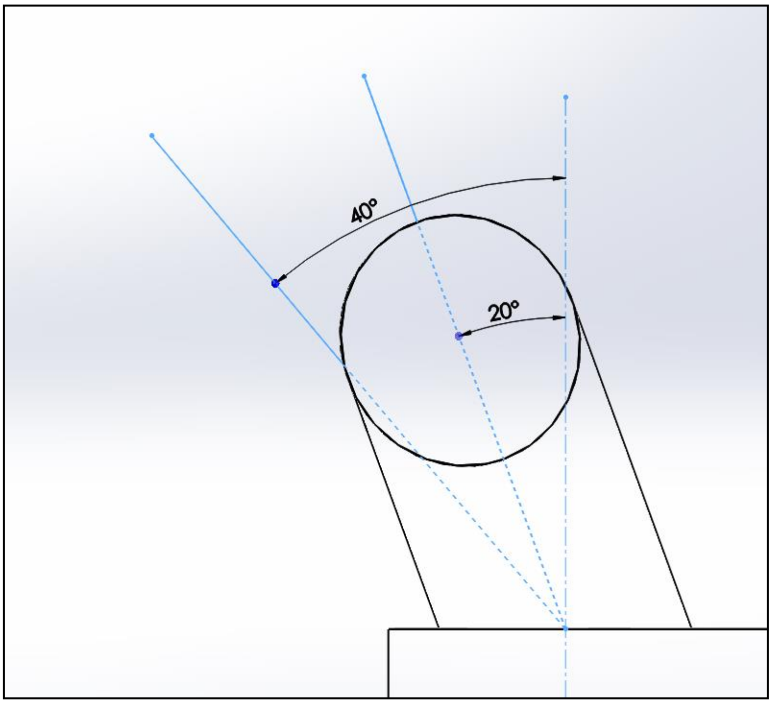


Figure 2. Schematic of the inclination of the conduits

To proceed with the study, it was necessary to create meshes from the designed models. The meshes are represented in figure 2 and were created using the Fame HEXA software provided by AVL AST with characteristics according to table 2.

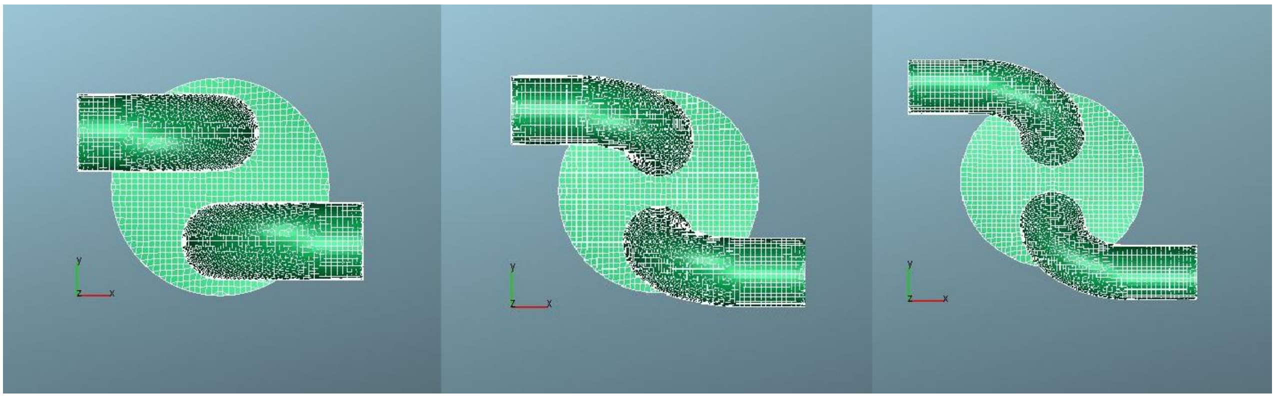

Figure 3. Generated meshes

Table 2. Mesh creation parameters.

| | |
|---|---|
| Maximum Cell Size (mm) | 0.0025 |
| Boundary Layers | 1 |
| Boundary Layer Optimization | Volume Optimizer |
| Smoothing | Volume Optimizer |

After the meshes were generated, the edges of the geometry were set according to the software pattern, which are defined by a sudden variation of the direction angle between consecutive cells. In this case they would be considered an edge if the angle was greater than 40 degrees. From these sections, it is possible to select the inlet and outlet face.

The next step is to set up the *Solver Steering File*, starting with the run mode where steady state simulation was chosen. Then the boundary conditions were set. For the inlet section, the mass flow condition was chosen and its value was calculated with Eqs. (3) and (4), which returned a flow of 0.3062 kg/s. Both turbulence and temperature were fixed scalars. The outlet edge condition was set as static pressure with value of 100000 Pa, representing the atmosphere. Air properties were used for this simulation. Table 3 shows the initial conditions for the simulation.

Table 3. Initial conditions

| | |
|---|---|
| Pressure (Pa) | 100000 |
| Temperature (K) | 293.15 |
| Turbulence Kinetic Energy ($m^2/s^2$) | 0.001 |
| Turbulence Length Scale (m) | 0.001 |
| Turbulence Dissemination Rate ($m^2/s^3$) | 0.00519615 |

Discretization settings were adjusted according to the software pattern by extrapolating boundary values and using the Least Square Fit method for calculation of derivatives. Simple mode was active. On Equation Control, k-zeta-f was the chosen turbulence method and fluid was considered incompressible. Adopted convergence criteria was 0.0001 units for both pressure and momentum.

After configuring these settings all three simulations were run and post processed.

## 4. RESULTS AND DISCUSSION

To analyze the data, for each case was created a cut in the XY plane, using the AVL IMPRESS tool, to best visualize the velocity vectors direction. Also, a perspective view of each model was created showing streamlines to represent predominant flow directions in the cylinder. Through these figures, it is possible to make a qualitative analysis of the in-cylinder fluid dynamic.

Figure 4 below shows the plane cut and the streamlines perspective for case 1, without conduit inclination. It is possible to observe that tumble is the most present flow structure through the streamlines, despite it is possible to see swirl movement in the vectors of the plane cut. Also, there are different structures scales inside the cylinder and the homogenous distribution of velocity suggests low swirl.

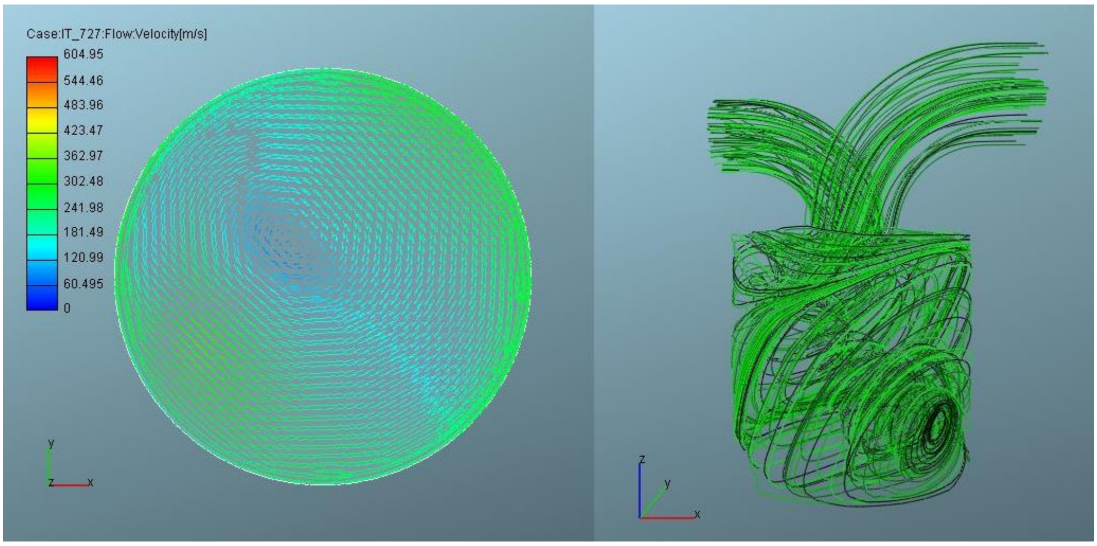


Figure 4. XY plane cut and streamline representation of case 1 results

In figure 5 the representation of the XY plane cut and the streamline for case 2 are shown. Similar to the previous case result tumble is still the most predominant structure of the in-cylinder flow, nevertheless swirl has increased and it is possible to see more lines projections in the XY plane. Velocity of the vectors has slightly increased which shows more swirl movement in this section of the cylinder, which is even better for dispersing fuel mixture and flame wall in combustion. Another way to observe swirl increase is the enhancement of the low velocity area in the middle of the cut plane.

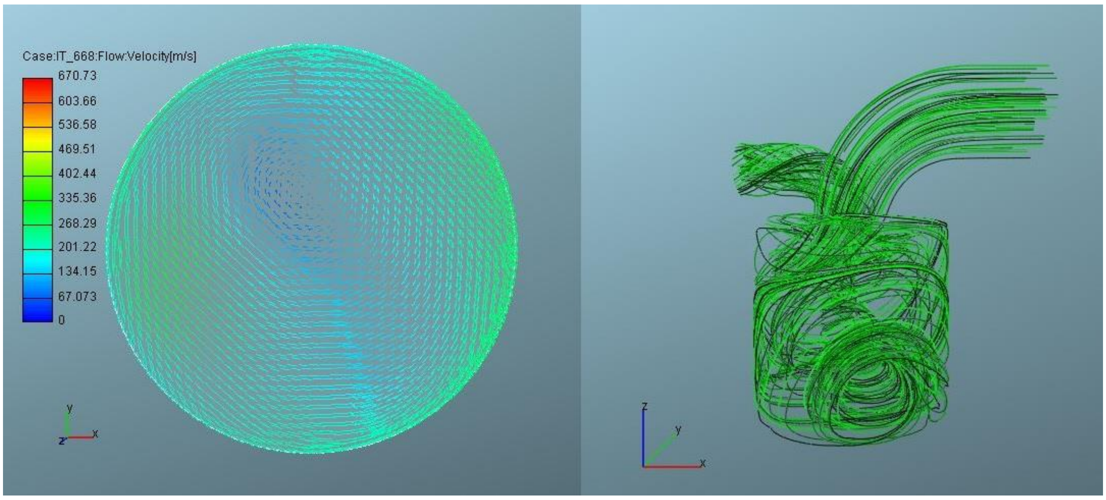


Figure 5. XY plane cut and streamline representation of case 2 results

The last case simulated has its results displayed in figure 6. Different from the other cases, swirl is more clearly identified. Both structures are present, however this type of turbulence is even more significant than tumble, showing that inclining the inlet duct provides a better swirl formation without reducing the other. There was an increase of swirl velocity as seen in the plane cut. Also, there is a greater area of lower flow velocity in the center, indicating that the gases inertia makes the fluid to follow the tendency of staying in the peripherical area of the cylinder, which suggests more swirl movement.

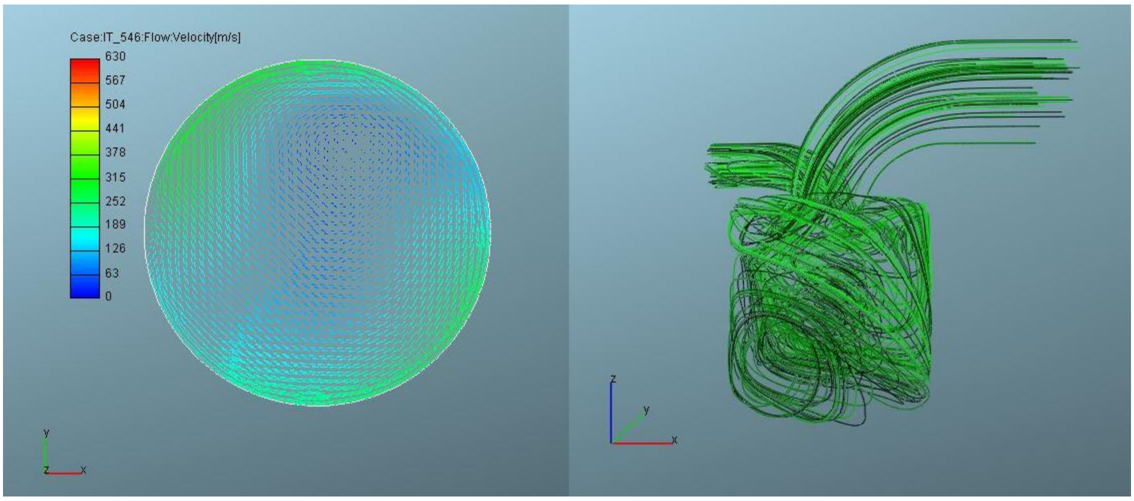


Figure 6. XY plane cut and streamline representation of case 3 results

Table 4 quantifies the effects observed in figures 4, 5 and 6. Through Eqs. (1) and (2) both swirl and tumble coefficients were calculated by collecting vorticity values in points of the cylinder plane. It assures that swirl is constantly increasing in the cases while tumble stays approximately constant. Therefore, increasing the conduit inclination causes a growth in the swirl coefficient without producing significant effect on tumble formation. Nonetheless, the greater area of low velocity in the center may harm the capacity of mixture diffusion inside the cylinder and also the propagation of flame wall.

Table 4. Swirl and tumble coefficients for each case

| Case | Swirl Coefficient (SC) | Tumble Coefficient (TC) |
|---|---|---|
| 1 | 7.14e-04 | -6.10e-02 |
| 2 | 2.93e-03 | -7.08e-02 |
| 3 | 1.40e-01 | -2.51e-02 |

## 5. CONCLUSIONS

The studied cases presented above demonstrates that increases in the inclination angle of inlet ducts produce a higher level of swirl, which was shown both qualitatively and quantitively. Greater swirl formation can be very beneficial for internal combustion engines, since it can raise the rate of diffusion which enhances efficiency, especially when operating with lean mixtures. However, a very elevated swirl effect can be harmful, considering it can diminish the amount of species transportation in the central area where there is a low velocity field.

This research was the first step studying the geometric influence of the intake ports on in-cylinder flow. Due to the complexity of simulating transient species, this work focuses on a steady state model and will be improved in the future. With the addition of moving meshes, valves and air fuel mixture a more approximate model can be calculated, including a transient one.

## 6. ACKNOWLEDGEMENTS

The authors thank FAPEMIG for the support to this work.

## 7. REFERENCES

AVL FIRE®, “Primer Getting Started: Intake Manifold (900)”, AVL FIRE® VERSION 2014. <http://www.avl.com>.

Camata, M. B., 2017. “Análise de diferentes geometrias de retificador de fluxo no desempenho de um sensor tipo "impulsive swirl meter” utilizado para medição de cabeçotes.”Master thesis, Universidade de São Paulo, São Paulo.

Fonseca, L. G., 2014. “Caracterização do escoamento de ar em um motor de combustão interna utilizando mecânica dos fluidos computacional.” Master thesis, Universidade Federal de Minas Gerais, Belo Horizonte.

Heywood, J. B. “ Internal Combustion Engine Fundamentals”. McGraw-Hill International Editions, Sigapura. (1988).

Lumley, J. L., “Engines: an Introduction”, Cambridge University Press, Cambridge, 1999.

## 8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.