24th COBEM - 2017

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

COBEM-2017-1132
NUMERICAL ANALYSIS OF THE AIR FLOW IN INTAKE MANIFOLDS OF INTERNAL COMBUSTION ENGINES

Ana Marta de Souza
Hugo de Sanches Souza
Taynara Brito da Silva
Kaissar de Oliveira Nabbout
UFU – Universidade Federal de Uberlândia
Av. João Naves de Ávila, 2121, 38408-100 – Uberlândia, MG - Brazil
amsouza@ufu.br; hugosouza.meta@gmail.com, taynara.brito@yahoo.com.br, kaissar_on@hotmail.com

Abstract. *Optimizations on the performance of an internal combustion engine are important, because they are responsible for reducing fuel consumption and pollutant emissions. The intake manifold has a fundamental relevance on the engine performance, since it has to lead the air flow into the cylinders properly. In this way, the geometry of the ducts and plenum has a fundamental role in the engine performance. A good project of the intake manifold could help increase the load inside the cylinder, ensuring a higher volumetric efficiency. The engine torque and, consequently, power, are directly related to the volumetric efficiency. Furthermore, such project could minimize the head loss and avoid the cylinder to cylinder variability. The present work presents an analysis of the airflow in intake manifold through the Computational Fluid Dynamics (CFD) simulations using AVL/FIRE software. The airflow was investigated in three different intake manifolds, in which the principal difference is the cross section of the ducts. The results made it possible to verify the behavior of the air inside the plenum and the ducts, the recirculations formed, the head losses and the possible impacts in the engine operation.*

Keywords: *internal combustion engine, intake manifold, Computational Fluid Dynamic (CFD), AVL/FIRE.*

1. INTRODUCTION

The engine performance is measured by means of torque and power, specific fuel consumption and pollutant emissions. There are many ways to increase the torque and power, such as increasing the volumetric efficiency, the thermal efficiency and the air specific mass, changing the fuel or minimizing the mechanical losses. Among these, the volumetric efficiency has been much explored in the last times. The volumetric efficiency measures how efficiently the engine cylinders are filled of air according to its capacity. There are specific devices whose goal is to increase the cylinders filling, among which the most famous is the turbocharger. Turbocharged engines can display volumetric efficiency above 100%. Naturally aspirated engines rarely reach 100 % of volumetric efficiency, but it is possible to increase the usual values through the use of resonators, overlap valves, head project and intake and outtake manifolds project, etc.

The project of heads, intake and outtake manifolds is a common practice in engine tuning. It is possible to improve the air flow in the head and manifolds, taking advantage of the determined operational conditions and getting a better volumetric efficiency and, consequently, a better engine performance.

The intake manifold has as principal function to lead the air into the cylinders, ensuring adequate volume and velocity to the engine operation. There are many project parameters of an intake manifold that can influence the engine performance. However, the d port geometry has a fundamental role in the project of air admission systems. Length and diameter of the ducts can affect directly the torque and power, as it has been studied for many authors.

The present work has as main goal to analyse the air flow in an intake manifold, identifying the local recirculations, the head losses and the possible impacts on the engine performance. It was evaluated three manifolds with three different geometries of the ducts (circular, square and segmented).

2. INTAKE MANIFOLDS

The intake manifolds are responsible to lead the air flow into the cylinders properly. A good project of this important part can result in an increasing of the volumetric efficiency, which means gains of torque and power. Many parameters have to be considerate in the conception of the intake manifold, such as the length, the geometry, the way of distribution to the cylinders, etc.

In the case of the geometry, the usual is to use ducts with circular sections, although this work will compare three types of sections: circular, squared and segmented. Each section has different influence on the flow and its representation can be seen in the Fig. 1.

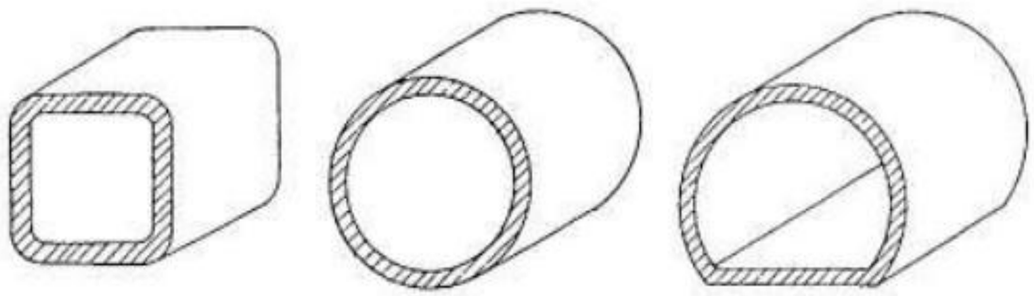

Figure 1. Duct sections: square/rectangular, circular or segmented, respectively (HEISLER, 1995).

According to Heisler (1995), square or rectangular sections provide greater opportunity to vaporize the precipitated fuel, spreading over the roof, walls and floor of the duct. Another benefit of a rectangular section is to prevent the load column from rotating as it moves through the duct, minimizing any centrifugal effect that would force the heavier liquid particles to be thrown against the collector walls. However, because of the exposed surface area is relatively larger for a given cross-sectional area, the friction drag is also larger, resulting in marginally lower volumetric efficiencies.

Circular sections provide the smallest surface area for any cross-sectional shape, and hence, less resistance to load flow. This provides the highest volumetric efficiency. However, the load column moving through a circular section tends to generate a longitudinal twisting or swirling which causes the fuel globules in the air stream to be thrown over the walls of the tube. As a result, the distribution of the blend through the section can be very uneven. Another disadvantage of the circular section is that the semicircular floor of the tube provides only a relative surface area for evaporation of the liquid fuel, thus achieving lower evaporation and mixing results of the fuel. (HESLER 1995).

Segmented sections are a solution that combine the high volumetric efficiency of circular ducts and the large rectangular duct area. Thus, the vaporization capacity is maximized while the turbulence of the charge column is minimized in these geometries. The semicircular section provides a large flat flat area for rapid evaporation of the fuel fluid content and secondly, the flat floor minimizes turbulence column loading, so the load retains its initial input density when flowing through of the duct. (HESLER 1995).

3. COMPUTATIONAL PROCEDURE

The simulations were performed with the AVL/FIRE software, which uses the finite volume method to solve the mass, movement and energy conservation equations. The problem simulated was the intake manifold with three different configurations for the cross-section that can be seen in the Fig. 2. The hydraulic diameter used in the configurations 1 and 2 was 32 mm, while it was 29 mm in configuration 3. Once the geometry has been created, the next step is import it into the software to create the surface. This step consists in define the edges and the regions of inlet/outlet presents in the surface. Afterwards, the mesh must be created. The program provides a versatile automatic tool capable to generate numerical grids in the most complex geometries assembled out of as many hexahedral elements as possible.

The meshes were constructed using the Fame Hexa, a tool of AVL/FIRE. The simulations were performed over the same conditions for the three configurations. Figure 3 shows the mesh used for the configuration 1 (circular cross-section).

a) Configuration 1 b) Configuration 2 c) Configuration 3

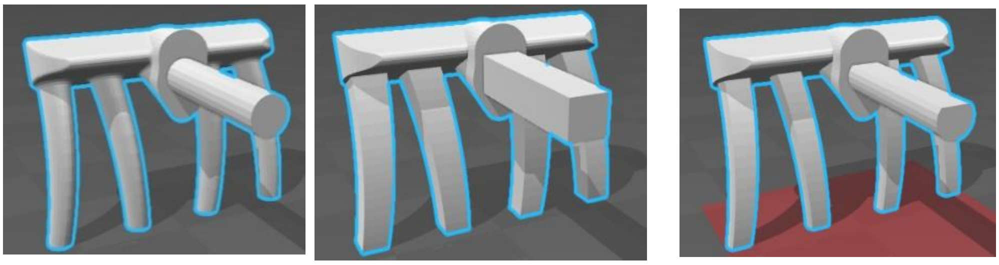

Figure 2. Intake manifold with ducts geometry adaptated.
a) Circular; b) square; c) segmented.

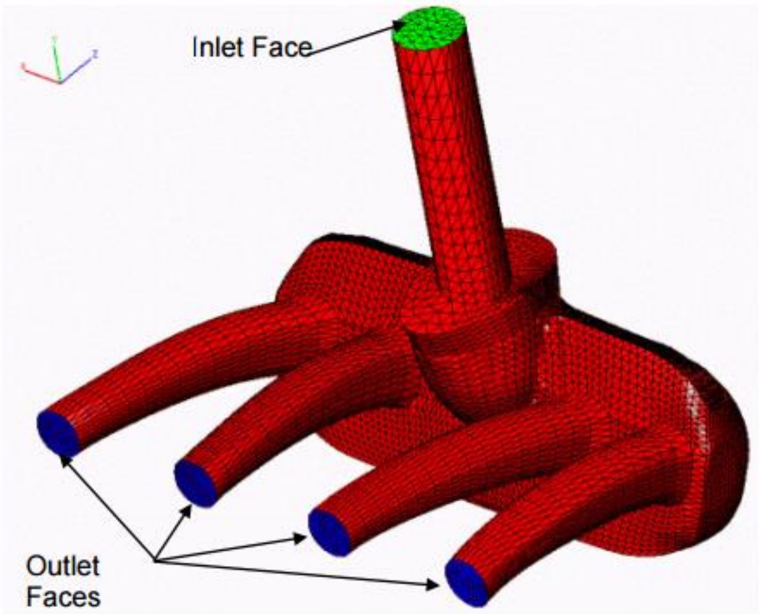

Figure 3. Mesh of the Intake Manifold. (AVL FIRE®)

The next step is define the boundaries conditions. In this problem only two boundaries conditions were necessary: mass flow for the inlet and static pressure for the outlet. They are shownd in Fig. 4 and Fig. 5 respectively.

Inlet	Sel. for BC	BC_inlet	
	Name of BC	BC_inlet	
	Type of BC	Inlet/Outlet	
	Inlet/Outlet	Mass Flow	
	Activate Flow Direction	Deactivate	
	Massflow	0.0038	kg/s
	Fixed temperature	Yes 293.15	K
	Fixed scalar	Yes 1	
	Fixed turbulence	Yes	
		Turb. ref. velocity= 0	m/s
		% of mean velocity = 0	
		Turb. kin. energy = 0.02	m^2/s^2
		Turb. length scale = 0.001	m
		% of hydraulic diameter and **Turb. diss. rate** are calculated from **Turb. kin. energy** and **Turb. length scale**.	

Figure 4. Inlet conditions.

Outlet	Sel. for BC	BC_outlet	
	Name of BC	BC_outlet	
	Type of BC	Inlet/Outlet	
	Inlet/Outlet	Static Pressure	
	Pressure	100000	Pa
	Activate Flow Direction	Deactivate	
	Fixed temperature	No	
	Fixed scalar	No	
	Fixed turbulence	No	

Figure 5. Outlet conditions.

4. RESULTS

The three geometries adopted present different results for the velocity development in the inlet. Figure 6 shows the behavior of the velocity vector for ducts with the same hydraulic diameter.

Circular Section Square Section Segmented Section

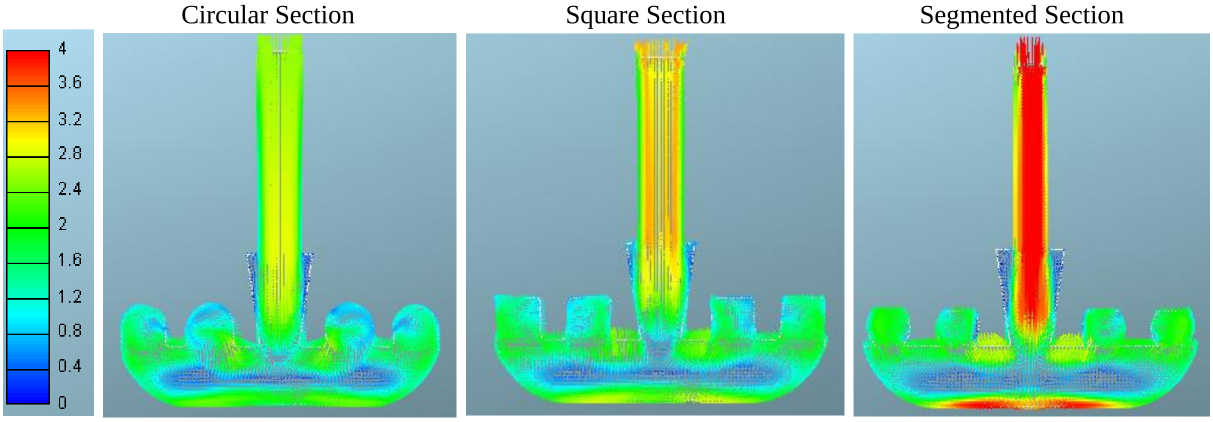

Figure 6. Cut in the Z-plan showing the velocity vectors.

It is noticeable that, in the case of the square section, the velocity variation between the center and the sides of the duct is greater than in the circular section. This is mainly because the square geometry presents a relatively larger exposed surface area for the same cross section, resulting in a greater frictional drag. In addition, the flow velocities in the square duct were larger than those presented in the circular section. The circular section, on the other hand, provides a smaller surface area and, consequently, less resistance to load flow.

A second analysis that can be performed through Fig. 6 is the comparison between the segmented section and the other sections, since this section presents higher velocities in the inlet due its smaller hydraulic diameter.

By analyzing the central duct of the intake manifold with circular geometry, it is possible to identify, as shown in Fig. 7, that the higher velocity occurs at the center of the duct, with lower velocities at the sides of the duct due to the viscous effect. The flow through the central duct favors the formation of two large vortices. This occurs due to the difficulty of the fluid to remain adhered to the surface of the plenum, since the air reaches the lower wall of the plenum with high velocity, being distributed to the sides and forming the vortices.

In Fig. 8, it is possible to see in more detail, through the velocity vectors, the large recirculation formed in the plenum. This recirculation tends to impair the intake of air into the central cylinders of the intake manifold. The disadvantage of this intake manifold configuration is that the central and lateral cylinders have different flows due to the formation of recirculations. As direct effects on the engine, there is cyclical variability, that is, at each cycle the engine tends to draw different amounts of air, and therefore works with different loads from cycle to cycle. Although this cycle-to-cycle variation is an undesired phenomenon, it becomes quite common, due precisely to the configurations of intake manifolds.

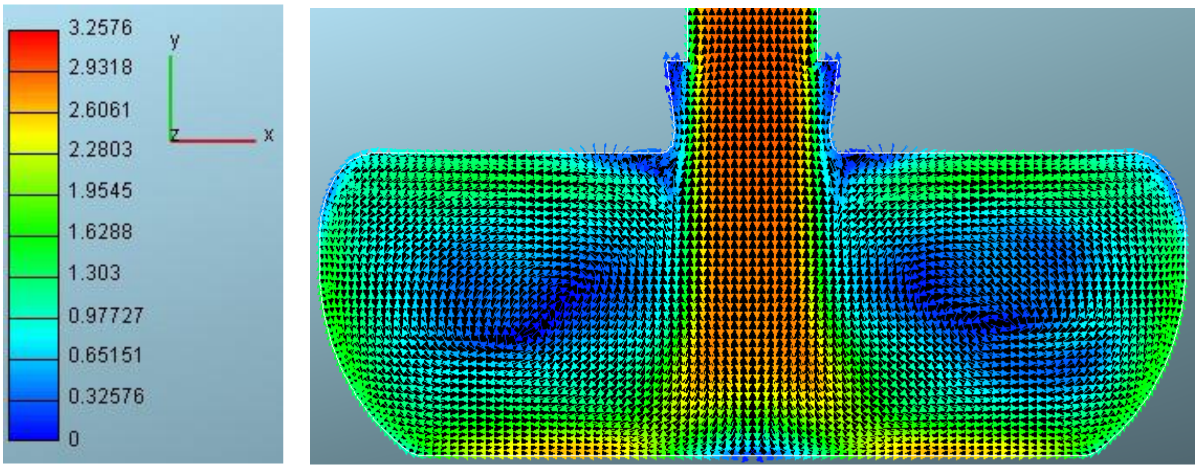

Figure 7. Gradient of velocity vectors in the duct with circular section.

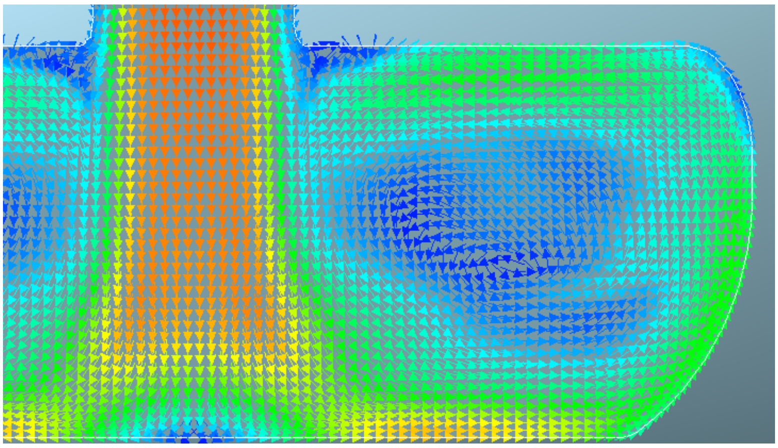

Figure 8. Velocity vectors showing the air recirculation in the circular pipe.

Analyzing this same cut for the segmented section, it is possible to notice that the recirculations in this scenario are smaller, compared to those presented for the circular geometry, as can be observed in Fig. 9.

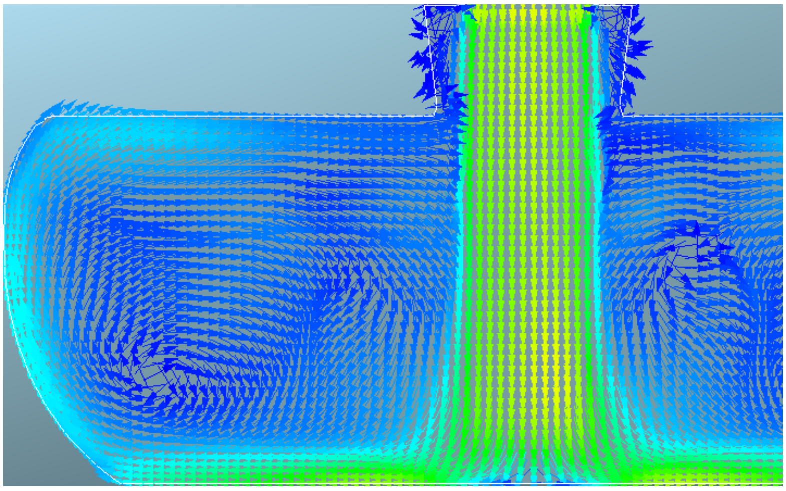

Figure 9. Velocity vectors showing the air recirculation in the segmented pipe.

The next analysis refers to the outlet ducts, lateral and central, which receive air from the plenum and take it to the combustion chamber. Figure 10 shows how the flow flows from the plenum to the four cylinders of the intake manifold with circular geometry ducts. It is possible to identify the difference between the flow of the lateral cylinders and the central cylinders, due to the cyclical variability caused by the recirculations that occur in circular geometries.

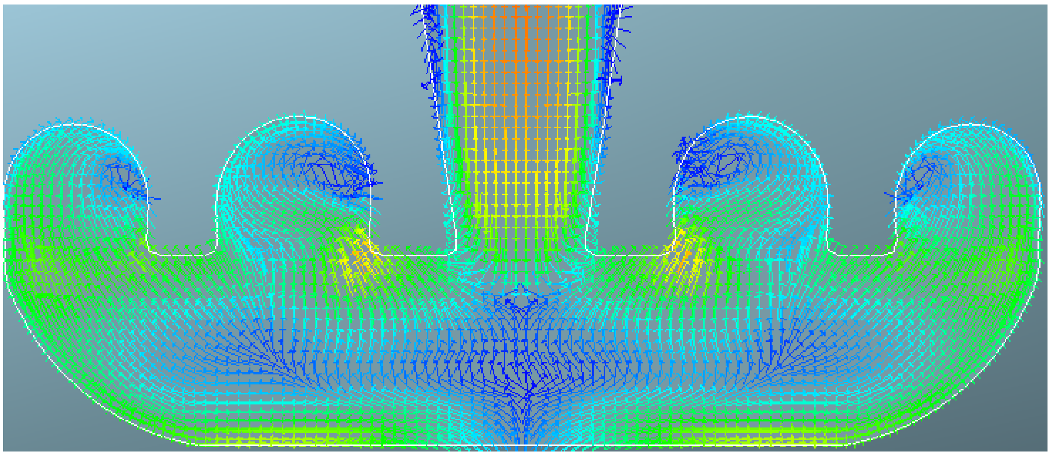

Figure 10. Velocity vectors in the central and lateral outlet ducts with circular cross-section.

Figure 11 illustrates the inlet configuration in one of the central outlet ducts with circular geometry showing a slight swirling in the plenum, small detachment of the boundary layer upon entering higher velocities in the inlet duct of the combustion chamber.

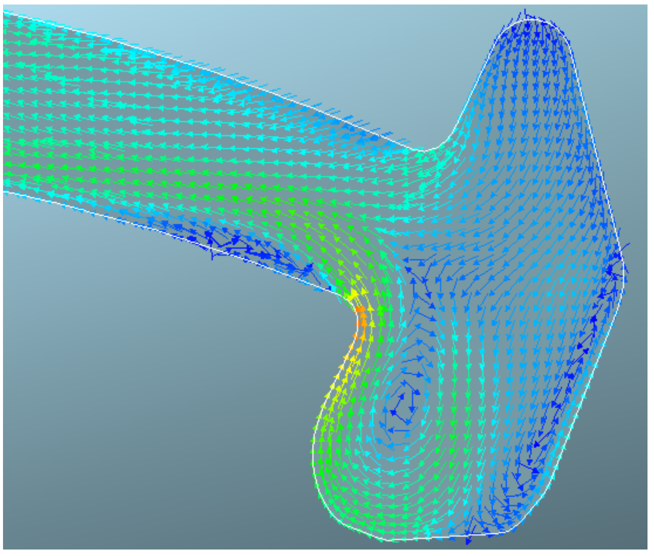

Figure 11. Velocity vector in the inlet of the circular central cylinder.

The circular geometric configuration contributes to the formation of small vortices in the inlet duct, there being small detachment of the boundary layer with rapid recomposition at the lower edge, better geometric configuration for flow stability and greater mass flow per unit of time compared to the other configurations analyzed.

Figure 12 presents the velocity configuration in the inflow of one of the lateral ducts of the outlet from the collector to the intake chamber, evidencing a lower recirculation condition with greater ease of flow than observed in the previous central configuration, contributing to the disadvantage of cyclic variability with cycle-to-cycle load difference.

All the symbols and notation must be defined in the text. Physical quantities must be expressed in the SI (metric) units. In order to avoid units braking in the next line use *Ctrl+Shift+space* between the numerical value and the unit. Mathematical symbols appearing in the text must be typed in *italic* style.

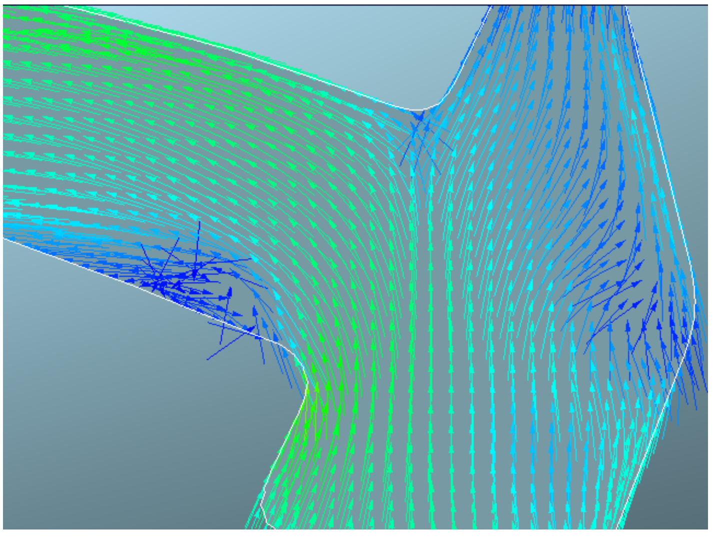

Figure 12. Velocity vector in the inlet of the circular lateral cylinder.

The velocity configuration shown in the square section duct shown in Fig. 13 shows a comparatively larger region of whirling in the plenum compared to the circular section. It is also possible to observe large detachment of the flow layer relative to the lower wall of the chamber intake duct with large vortex region, illustrating upstream vectors, possibly disrupting the duct entrance.

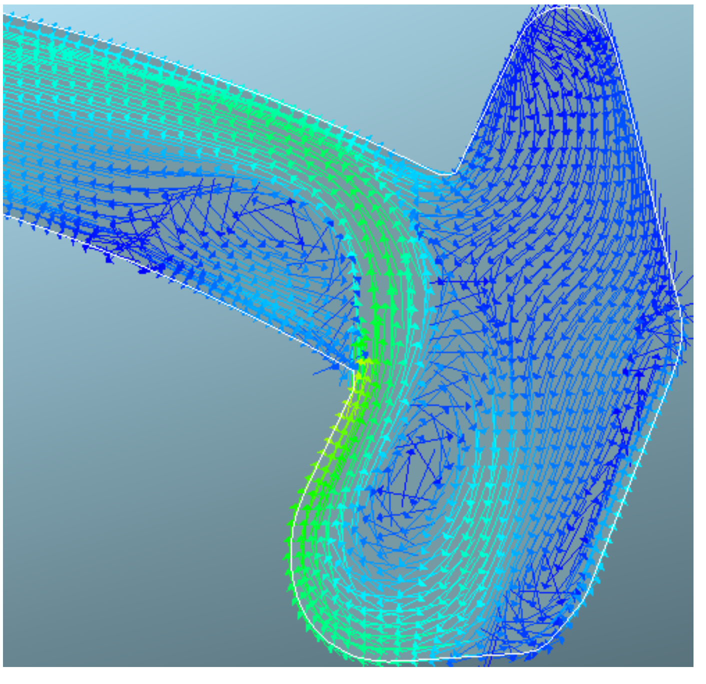

Figure 13. Velocity vector in the inlet of the square central cylinder.

The segmented section duct has the highest velocity values and the smallest recirculations are formed in the passage from the plenum to the outlet cylinder. In the square section duct, it is possible to identify the largest recirculation in the passage from the plenum to the outlet duct, which impairs the flow of air into the duct.

In order to compare the flows in the outlet ducts of the collector, the Fig. 14 shows a comparison of the speeds of the three geometries on a same scale. It is possible to visualize more clearly that the segmented section duct is the one that presents greater facility in conducting the air into the outlet ducts, generating less recirculations.

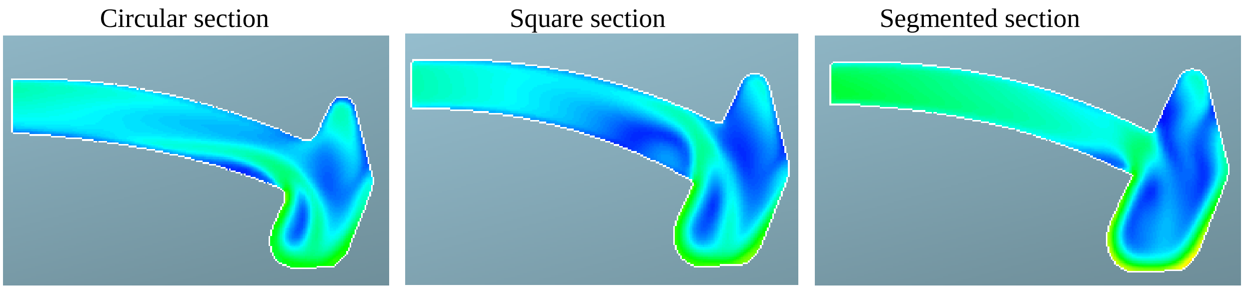

Figure 14. Comparison of the scalar velocities in the central cylinders.

Figure 15 represents the previously discussed effect of the cyclic variability in the central output ducts. It is possible to observe the greater ease of flow with higher flow velocities in the end ducts in all the studied geometries, but with a lower intensity effect in the segmented section, where the turbulence of the load columns is minimized.

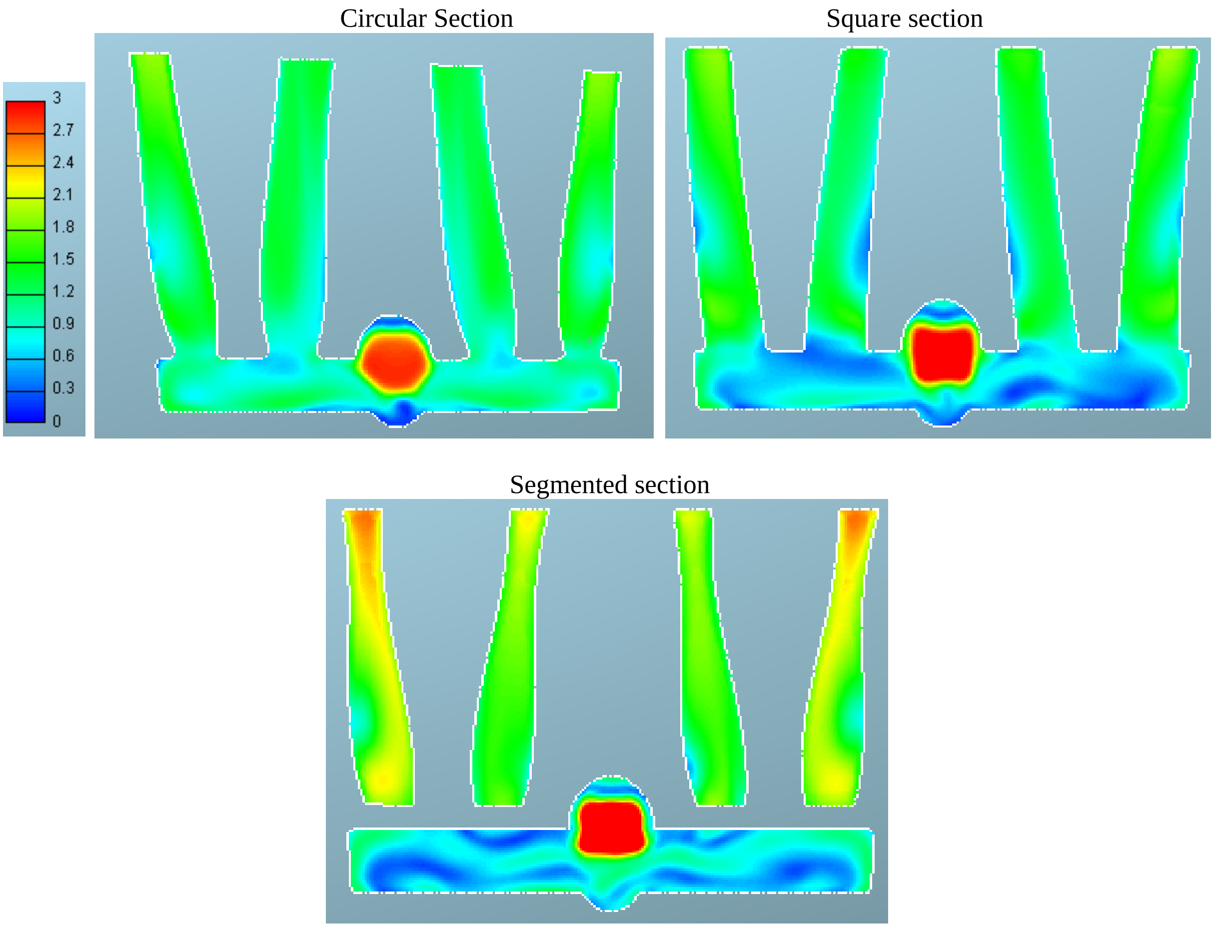

Figure 15. Comparison of scalar velocities in a plan cross-sectional to the y-axis.

In this work, there are no significant variations of the pressure in the ducts as can be seen in Fig. 16. It is important to remember that only the flow inside the intake manifold, not connected to a running engine, is being observed. It is known when the engine draws air from the collector pressure waves are formed, having a pulsed flow, which is not the objective of the present study.

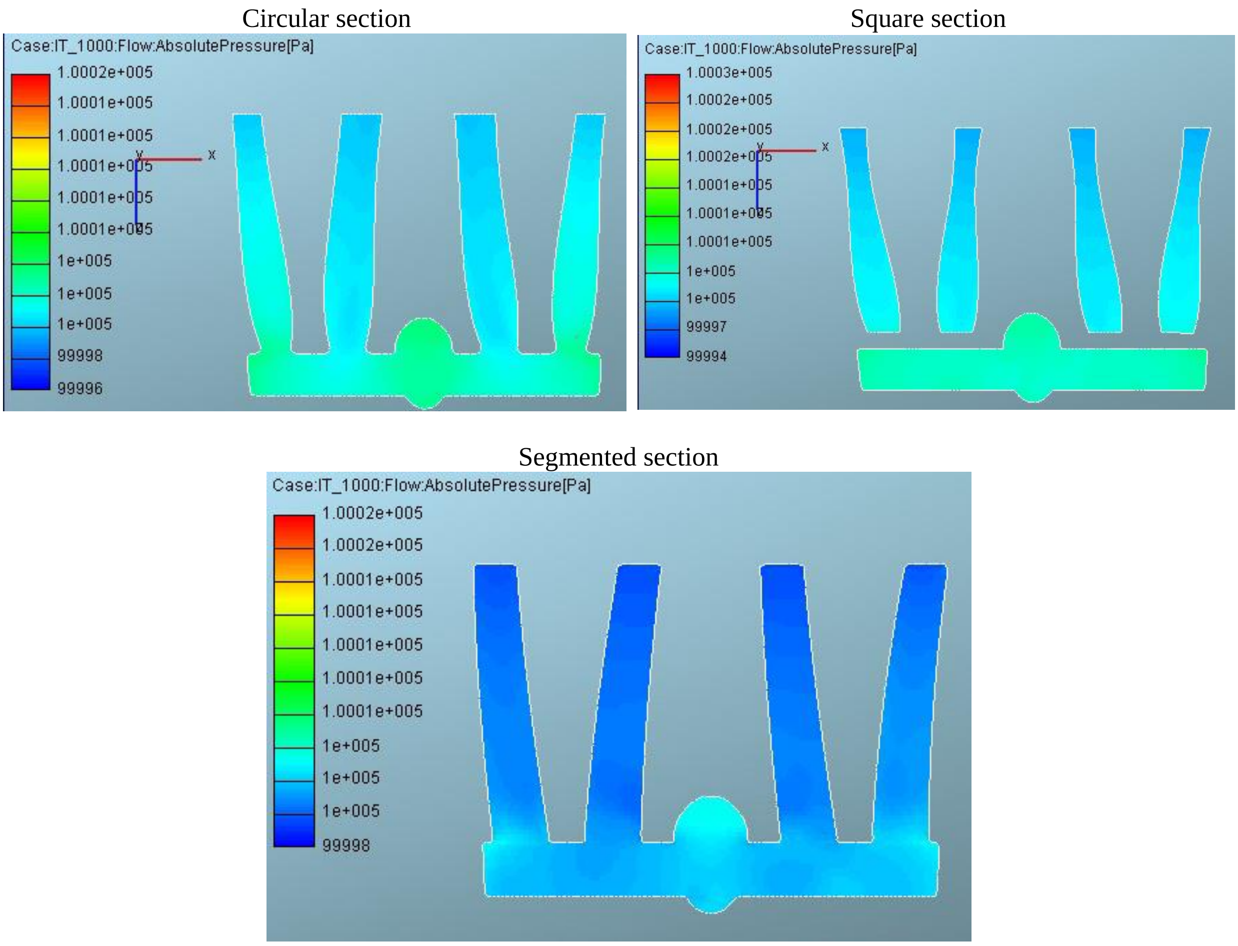

Figure 16. Pressure in the plenum and the outlet pipes.

5. CONCLUSION

In the intake manifolds analyzed it was observed the formation of two largest vortices in the plenum, which harm the air flow through the central ducts. The configuration 1 and 3 reprensent lower resistance to the air flow when compared to the configuration 2, wich means a better filling of the cylinders for circular and segmented profiles.

6. ACKNOWLEDGEMENTS

The authors thank FAPEMIG and AVL/FIRE® for the support to this work.

7. REFERENCES

AVL FIRE®, “Primer Getting Started: Intake Manifold (900)”, AVL FIRE® VERSION 2014. <http://www.avl.com>.
J. B. Heywood, Internal Combustion Engine Fundamentals, McGraw-Hill International Editions, Sigapura. 1988.
G. R. Souza, Estudo experimental e numérico do sistema de admissão de um motor de combustão interna. Doctorate Degree thesis. Universidade de São Paulo. 2010.
Y. A. Çengel and J.M. Cimbala. Mecânica dos fluidos - 3.ed. 2007.

8. RESPONSIBILITY NOTICE