

Analysis of the fluid dynamic behaviour through the air collector following the installation of pressure outlets

R Bucevschi, V Socalici, A Budiul Berghian and C Birtok Băneasă

Politehnica University of Timisoara, Department of Engineering and Management, 5 Revolution Street, Hunedoara, 331128, Romania

E-mail: robert.bucevschi@yahoo.ro

Abstract. This study is a preliminary investigation to determine the feasibility of creating an experimental model of pressure analysis at different points in the air intake system of an internal combustion engine. The purpose of the CFD analysis is to determine whether or not the value of the pressure obtained from the air intakes is influenced by their geometry or position. The study presents both the results of the analysis and the steps and parameters used in the numerical modeling process. The software used to create the numerical model is Fluent from the suite of software provided by Ansys under the academic license.

1. Introduction

In the design process of the internal combustion engines, an important topic is the gas supply system to the combustion chamber. Engine performance is influenced in a considerable way by the efficiency of the intake circuit. Route design as well as interior surface roughness are two of the factors of influence in the system's effectiveness [1].

A poor efficiency of the intake system leads to a smaller amount of fresh air into the combustion chamber, in order to reduce the gas-dynamic resistances most of the performant intake systems have slight outlines and transitions in as small steps as possible between the adjacent segments [2]. The intake process starts from the combustion chamber, where due to the downward movement of the piston a partial vacuum is created with a lower value than the atmospheric pressure, the vacuum value being mainly influenced by the restrictions caused by the intake valve geometry [3]. The gradual decrease of the intake cross-sectional areas results in an increased resonance pressure for a given airflow [4]. Knowing that the appearance of Helmholtz resonance is a result of Venturi effect. Due to the Helmholtz resonance, by narrowing the inlet manifold cross section in a controlled way, we can achieve a pressure increase effect similar to the efficiency of a low pressure compressor, but this process can be obtained on a narrow range of engine functional regimes, the phenomenon being also influenced directly by the length of the intake circuit [5].

A positive contribution can be created by the existence of two or more "hot spots" under pressure on the air route. When the intake air speed is higher, the dynamic pressure that pushes the air inside the engine is increased. Dynamic pressure is proportional to the square of the intake air speed, so as making a narrower or a longer passage, the dynamic speed or pressure can be increased [6], [7].

The main factors that precludes the increase of the air volume required for an optimal combustion are the restrictions that can inhibit the flow of gases as well as the increase of the air intake



temperature on the circuit, a higher temperature leads to an poor oxygen concentration. In order to increase the fresh air flow into the cylinder, each element of the air path must be analyzed from the perspective of this perturbing factors.

2. Case of study

As an study subject was chosen the intake circuit of an automobile Ford Puma (Figure 1) with the following characteristics [8]:

- Produced between 1997-2002
- Types of available engines: 1.4 ccm, 1.6 ccm 16V, 1.7 ccm Zetec VCT;
- Powertrain disposal: transverse, front-wheel drive;
- Circulation of air through the air filter is crossflow type;
- The filter casing is made of prismatic plastic;
- The filter element is a panel type;
- The filter position is on the left-hand side (behind the headlamp);
- The air path from the entrance to the collector of the throttle valve is about 1500 mm.



Figure 1. Ford Puma

This study aims to analyze the influence and efficiency of reading data after the installation of two air outlets on the collector inner surface (Figure 2.a) located at the inlet of the intake circuit. The analysis is based on the results of the Computational Fluid Dynamics (CFD) simulations on the assembled geometry. The geometric model used was achieved with Inventor Mechanical Design assisted design software from the Autodesk software suite, and the numeric model was built using the Fluent software from the suite of softwares provided by Ansys [9], [10]. The studied assembly is composed by a venturi air collector, a air intake with road and a simple air intake (Figure 2.b).

*A**b***Figure 2.** Intake collector*a-* intake collector position; *b-* CAD assembly model used

As input data, we used the air flow velocity on a single operating mode of the engine, namely at a rotation speed of 900 rpm. The air velocity is measured at the collector entrance using a Testo 410-2 model anemometer (Figure 3). As a result of the measurements it was established that the flow rate for the studied operating regime is 0.6 m / s. The simulation solution was generated on the basis of the Green-Gauss method, the elements used in the meshed geometric model are tetra-type solid. The size of the disposed elements is variable so that in the areas of interest they have a minimum size of 0.3 mm. Regarding the boundary conditions a contact with low friction coefficient was introduced on the separation surface between the fluid and the defined solid (Figure 4). The properties of the input data were assigned to the special surface created at the entrance of the collector, air density being assumed as a constant of 1,225 kg / m³. And the viscosity of 1.7894e-05 kg / m * s was also defined in the same way.

**Figure 3.** Anemometer Testo 410-2

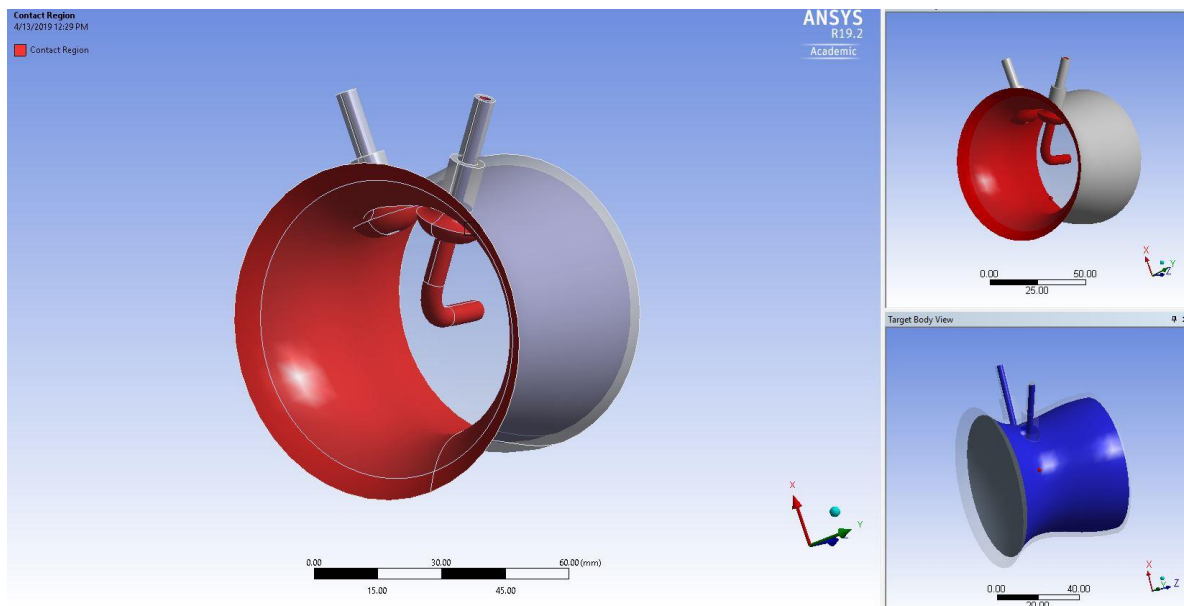


Figure 4. Defined contact surfaces between solid and fluid

3. Results and discussions

After solving 1000 iterations of the numerical model, the following results were obtained.

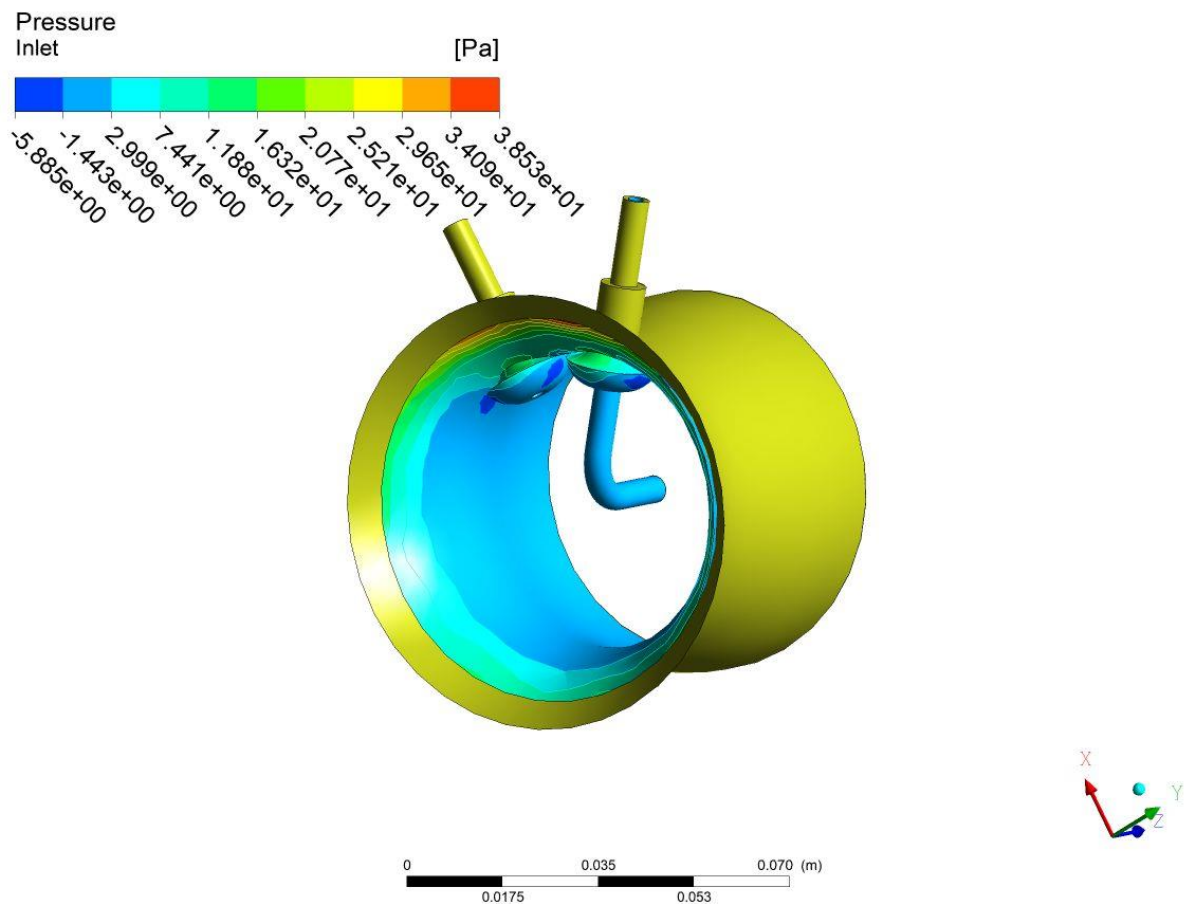


Figure 5. Pressure distribution on the collector inner surface

The pressure distribution chart on the inner surface shows an increase of the fluid pressure in the collector due to the gradual decrease of the cross sectional area, this being a result of Venturi effect occurrence.

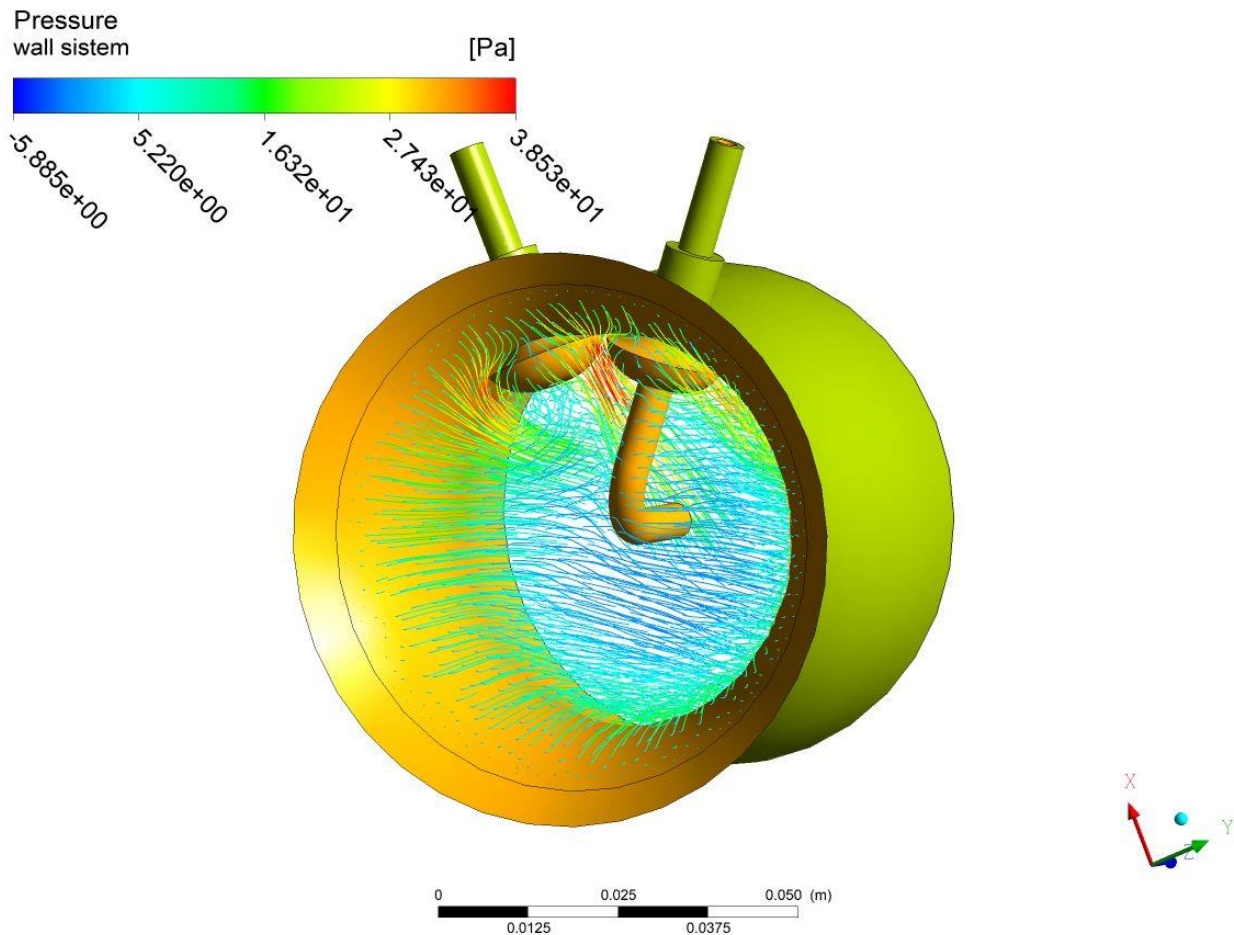


Figure 6. Fluid flow direction pressure diagram (inlet direction view)

In the simpler air intake case we can observe the appearance of small air vortex on the periphery side of the geometry, resulting a slight increase of the pressure currents in the external contact areas of the intake. But we can see that on the interest area, namely the communication port of the intake port, it can be noticed that the currents keep their flowing direction and their pressure is not influenced. As for the median intake port with rod, swirling currents formed in the collector mounting area do not influence the flow at the reading level in any way. A better perspective of the geometry influence on the flow can be obtained by analyzing the lateral view (Figure.7), it can be observed that in the pressure acquisition zone the value is not influenced in any way.

The results of the simulation obtained in the area in which the pressures will be read from the experimental model show close values of the pressures, more precisely in both cases, the pressure value was -1.44 Pa (Figure 9, Figure 10).

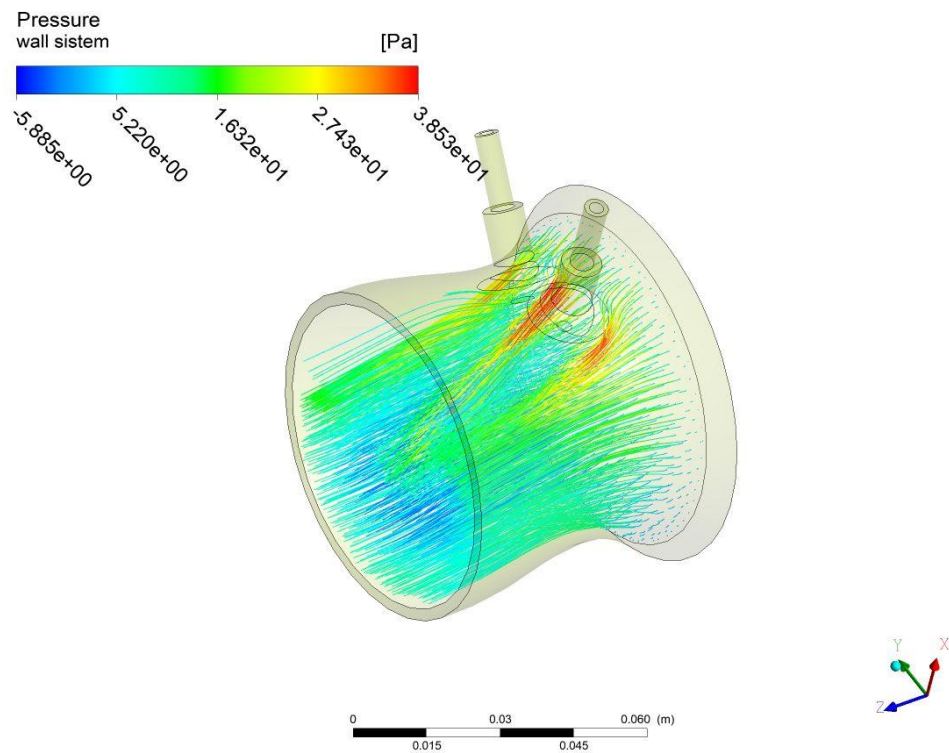


Figure 7. Fluid flow direction pressure diagram (lateral view)

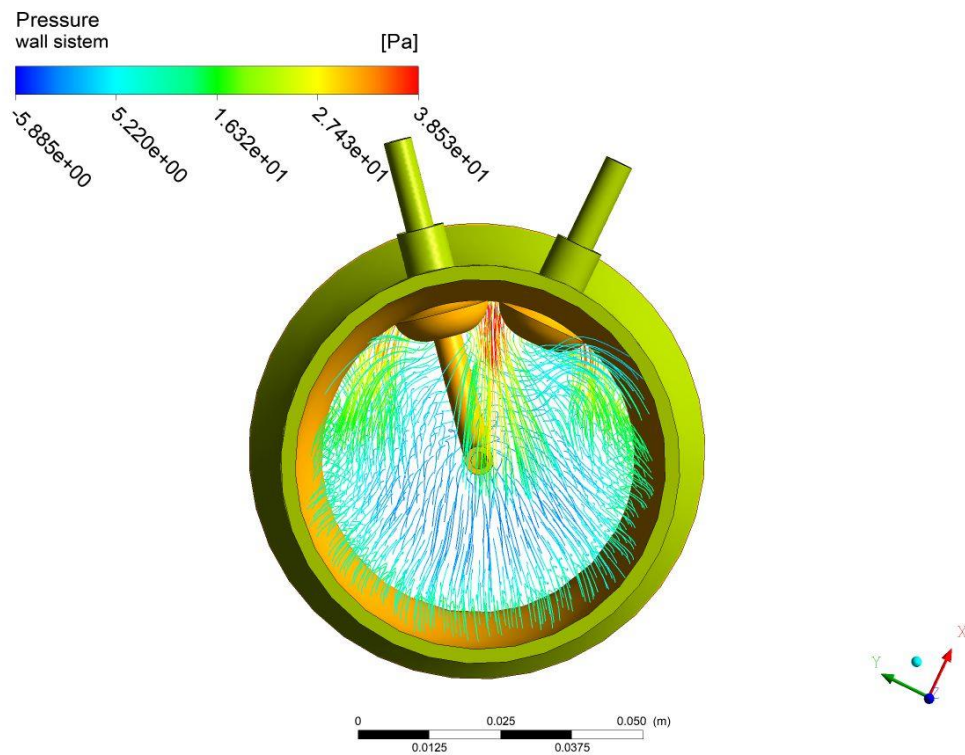


Figure 8. Fluid flow direction pressure diagram (outlet direction view)

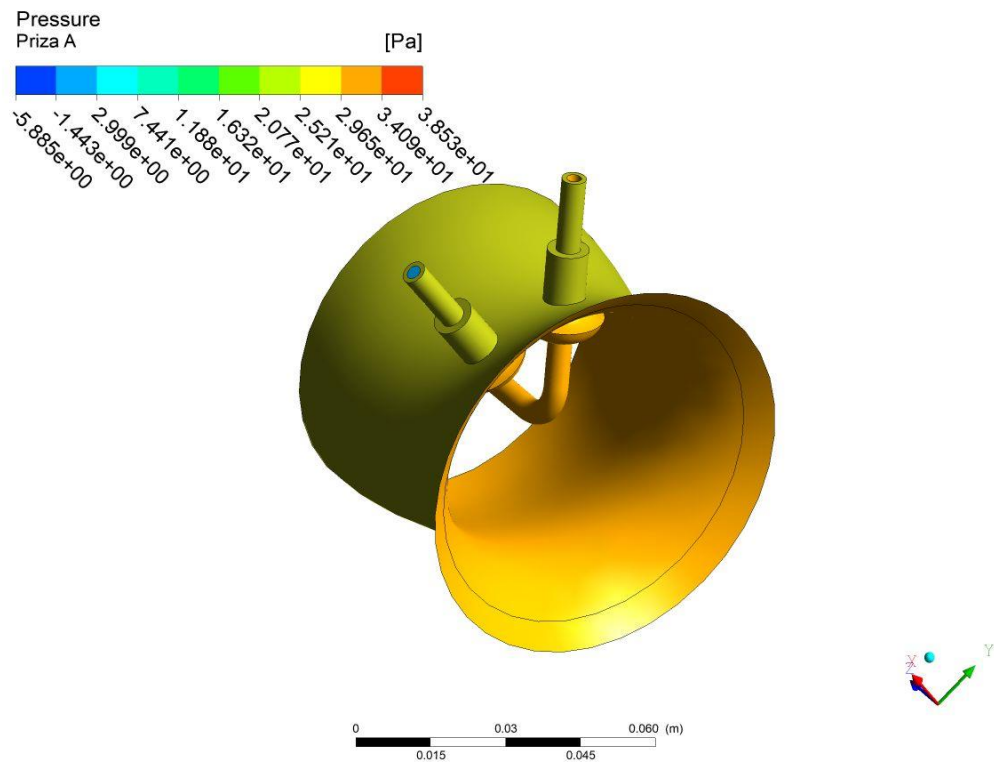


Figure 9. Result of the pressure dropping out of the short outlet

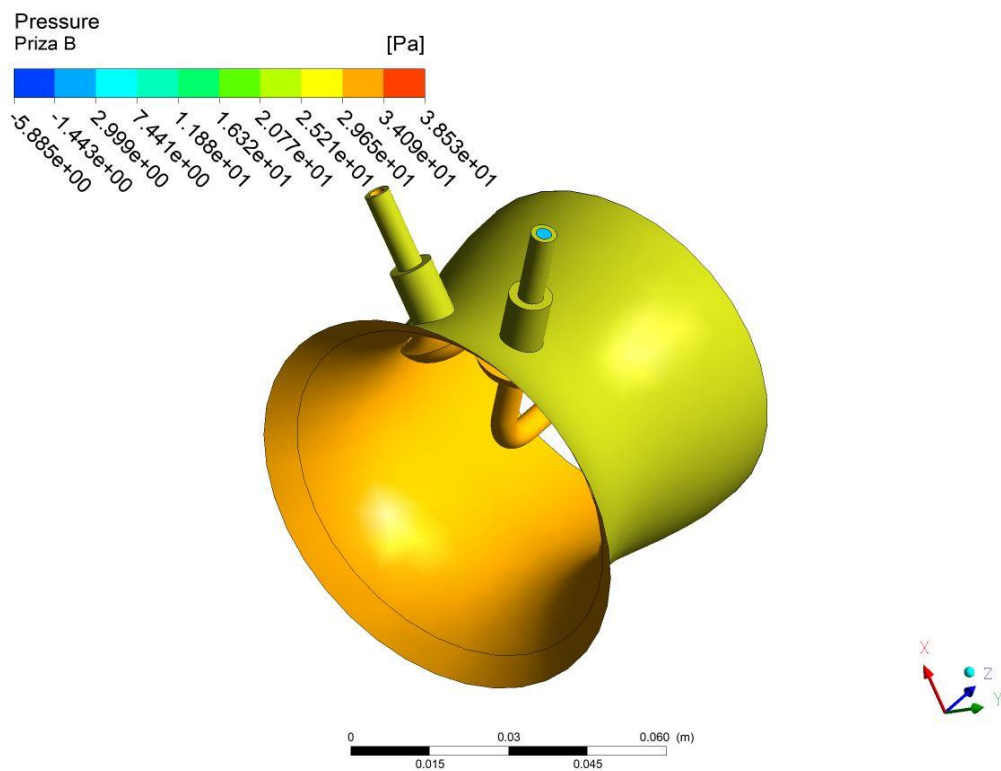


Figure 10. Result of the pressure dropping out of the median outlet

4. Conclusions

In conclusion we can say that the results of the simulation demonstrate that the pressure values obtained by mounting the two pressure intakes are not influenced in a negative way by their geometry. The influences on fluid flow direction or pressure are not likely to influence the final results. Under these circumstances it can be considered feasible to manufacture the experimental stand.

References

- [1] Ceviz M A and Akın M 2010 Design of a new SI engine intake manifold with variable length plenum, *Energy Conversion and Management* **51**(11) 2239-2244
- [2] Birtok Băneasă C, Budiul Berghian A, Socalici A and Bucevschi R 2019 Simulation of thermal transfer through the polyamide intake manifold, *Materiale plastice* **56**(1) 191-194
- [3] Wimmer A, Pivec R and Sams T 2000 *Heat Transfer to the Combustion Chamber and Port Walls of IC Engines-Measurement and Prediction*, SAE Tech. Paper No.2000-01-0568
- [4] ***Wikipedia Intake manifold, https://ro.wikipedia.org/wiki/Intake_manifold
- [5] ***Wikipedia Helmholtz resonance, https://en.wikipedia.org/wiki/Helmholtz_resonance
- [6] Schurov S M and Collings N 1994 *A Numerical Simulation of Intake Port Phenomena in a Spark Ignition Engine Under Cold Starting Conditions*, SAE Tech. Paper No. 941874
- [7] Birtok Băneasă C 2019 *Research regarding the use of advanced materials for the optimization in the intake process of the internal combustion engines*, Politehnica University of Timisoara, Romania, Doctoral Thesis
- [8] ***<https://www.evo.co.uk/ford/puma>
- [9] ***<https://www.autodesk.eu/collections/product-design-manufacturing/overview>
- [10] ***ANSYS Homepage, <https://www.ansys.com/products>