Monika Andrych-Zalewska, Wojciech Ambroszko, Łukasz Wielki, Krzysztof Ziora

Influence of air velocity at inlet to the intake system on flow parameters

JEL: R41 DOI: 10.24136/atest.2019.230
Data zgłoszenia: 28.01.2020 Data akceptacji: 10.02.2020

The article presents CFD (Computational Fluid Dynamics) analysis of the Ferrari 348 sports car intake system for three variants of air intake speed to the system. The article contains an analysis of the distribution of velocities and static pressures. In addition, local velocity and flow in the filtration chamber were investigated. The influence of air velocity at inlet to the intake system on the above-mentioned parameters was determined. The analysis is a prelude to the study of wave phenomena occurring in the intake system, affecting the fill factor of the cylinders. Ansys Fluent software was used for analysis.

Słowa kluczowe: intake system, CFD analysis, internal combustion engine, modeling, flow, wave phenomena, Ansys.

Wstęp

Modern spark-ignition engines must be characterized by large volumetric power indicators, which makes it possible to obtain a small mass of engine. This effect is possible by increasing the rotational speed and the cylinder filling ratio [5].

In terms of improvement of the fill factor, the following development tendencies are observed: optimization of intake systems (along with increasing the number of valves) and the use of controlled dynamic and compressor recharging (mainly turbo-recharging). The increase of the fill factor of cylinders with a fresh load requires a detailed knowledge and mapping of phenomena occurring in its intake system, including the process of filling the cylinder with the charge taking into account the fluid behavior during its movement in the above system and laws governing this process [8].

During engine operation, in the intake system there are also wave phenomena, which can be analyzed under certain assumptions and based on the theory of acoustic waves in the scope of their use for improving the cylinder filling ratio [18].

The fill factor is more efficiently influenced by the parameters of the intake system than by the exhaust system, which results in the characteristics of the torque and the useful power of the engine. The pressure at the end of the filling process is decisive. Using the dependence of the pressure of the end of the filling process on the charge flow rate in the downstream system, the system parameters can be selected so that the wave phenomena and inertia of the air flowing through the pipes and intake ducts lead to an increase in the pressure of the filling end [18].

The wave effect is associated with constantly occurring pressure disturbances forming a wave standing in the intake duct. These disturbances result from cyclic engine operation due to filling. When the amplitude of the initial pulse is high and the attenuation of the wave is small, then the standing wave influences the pressure during the flow through the open valve, and thus the filling of the cylinder, i.e. traveling the first pulse reflected from the open end of the conduit to the outlet, where the pulse changes pressure and thus affects the degree of filling of the cylinder [9].

The article presents CFD (Computational Fluid Dynamics) analysis of the Ferrari 348 GTC car's bottom system. Thanks to the discretization and numerical solution of partial differential equations describing the flow, it is possible to determine the distribution of velocity, pressure, temperature and other parameters in the fluid flow. Modern CFD pro-grams enable solving flows including viscosity and compressibility issues, multiphase flows, flows in which chemical reactions or combustion processes take place, flow through porous structures and flows in which the fluid is Newtonian or non-Newtonian liquid. It is possible to simulate the interaction between liquid and solid [13, 14].

The analysis of the influence of flow parameters such as velocity and pressure distribution is aimed at introducing further analysis of the intake system in terms of wave phenomena.

1.Model of the intake system

In order to simulate the flow of fluid in the intake manifold, the necessary parameters were calculated, assuming that the intake system was running for the travel of the vehicle at the maximum speed.

First, the speed of the fluid flow in the intake system was determined. The following parameters and operating conditions of the engine were adopted [1, 10–11]:

- engine speed n = 8000 rpm,
- engine displacement V_{ss} = 3.4 dm^3,
- power P = 281 kW (382 KM),
- volumetric efficiency η_v = 0.9,
- diameter of the flow cross-section d = 0.06 m,

and the function of the intensity of the air flowing through the intake system is expressed as:

$$Q = \frac{n}{2}\,\eta_v\,V_{ss} = 0{,}2\,\frac{m^3}{s} \qquad (1)$$

$$Q = 2\,\frac{\pi d^2}{4}\,v \qquad (2)$$

$$v = \frac{2Q}{\pi d^2} = 36{,}09\,\frac{m}{s} \qquad (3)$$

where: Q [m^3/s] – engine air demand,
v [m/s] – air flow speed.

Model representation of the intake system of the Ferrari 348 GTC engine

Result (Fig. 1) and its analysis was carried out in the Ansys Fluent program, for which the process of model discretization was done (Fig. 2) and a wall effect was defined using the program functions of the above simulation tool.

The following global grid settings were used in the analyzes: adaptive with a growth rate of 1.2, inflation options, smooth transition, type of elements (Tetra).

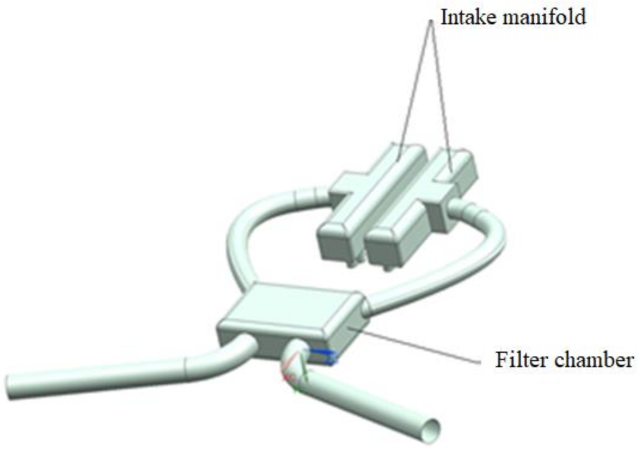

Fig. 1. View of the model of the intake system of the Ferrari 348 GTC sports car

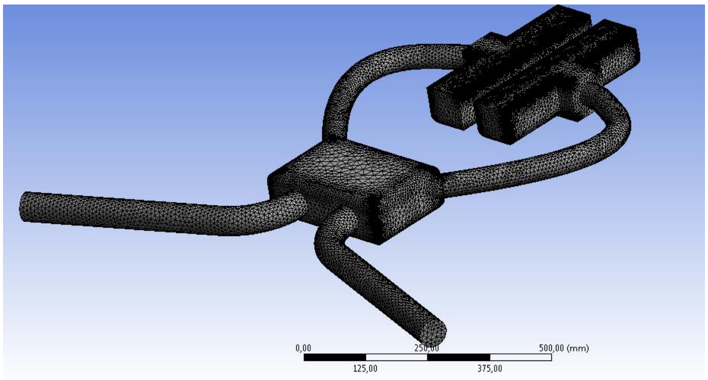

Fig. 2. Discrete model of the intake system of a Ferrari 348 GTC

Temperature and viscosity of air flowing in the tested intake manifold for atmospheric conditions were determined:

- air density ρ =1.2 kg/m^3,
- kinematic viscosity ν = 35•10^{-6} m^2/s,
- dynamic viscosity μ = 42•10^{-6} kg/(m•s),

2.Simulation of air flow in the engine's intake system

2.1.Determining the boundary conditions

Three cases of air velocity at the beginning of the Vp system were considered:

1. v_p = 0 m/s,
2. v_p = 36 m/s,
3. v_p = 94 m/s.

Considering the calculated demand of the air motor, the final air velocity in the intake system was defined as: v = 36.09 m/s. The intensity of turbulence was based on [2] as equal: 10%. The model was selected to simulate the flow k-omega SST, because the k-epsilon model is not able to capture appropriate behaviors turbulent in the boundary layer until fluid detachment [16].

The air temperature at the inlet to the intake system was assumed to be equal to T1 = 293 K, whereas the pressure in the intake system as (0.85-1.05 kPa).

The velocity distribution in the filtering chamber without filter was determined due to the lack of data on the filtering material (inertial resistance [m^{-1}] and viscous resistance [m^{-2}]). These are values determined experimentally.

2.2.Results simulation of processes occurring in the intake manifold

During the simulation, the total velocity distribution (Fig. 3) and the static pressure distribution were examined (Fig.4).

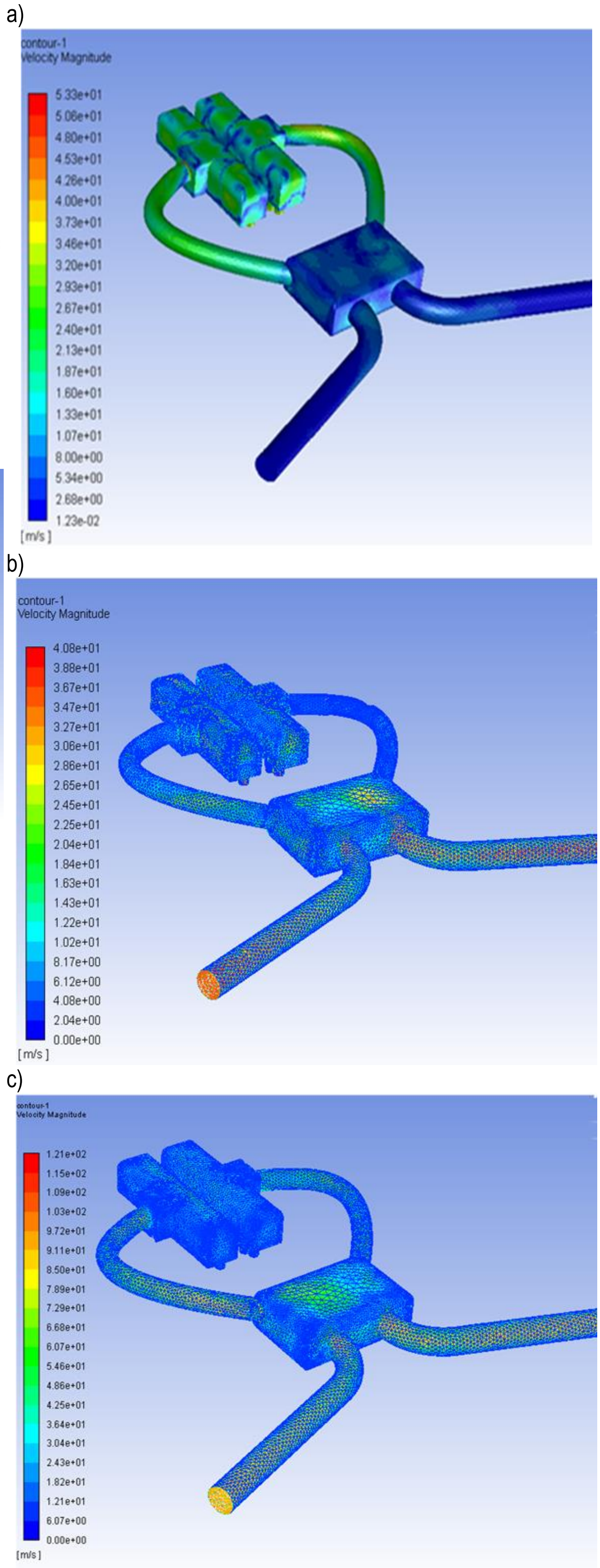

Fig. 3. Distribution of total velocity values a) v_p=0 m/s; b) v_p= 36 m/s; c) v_p= 94 m/s

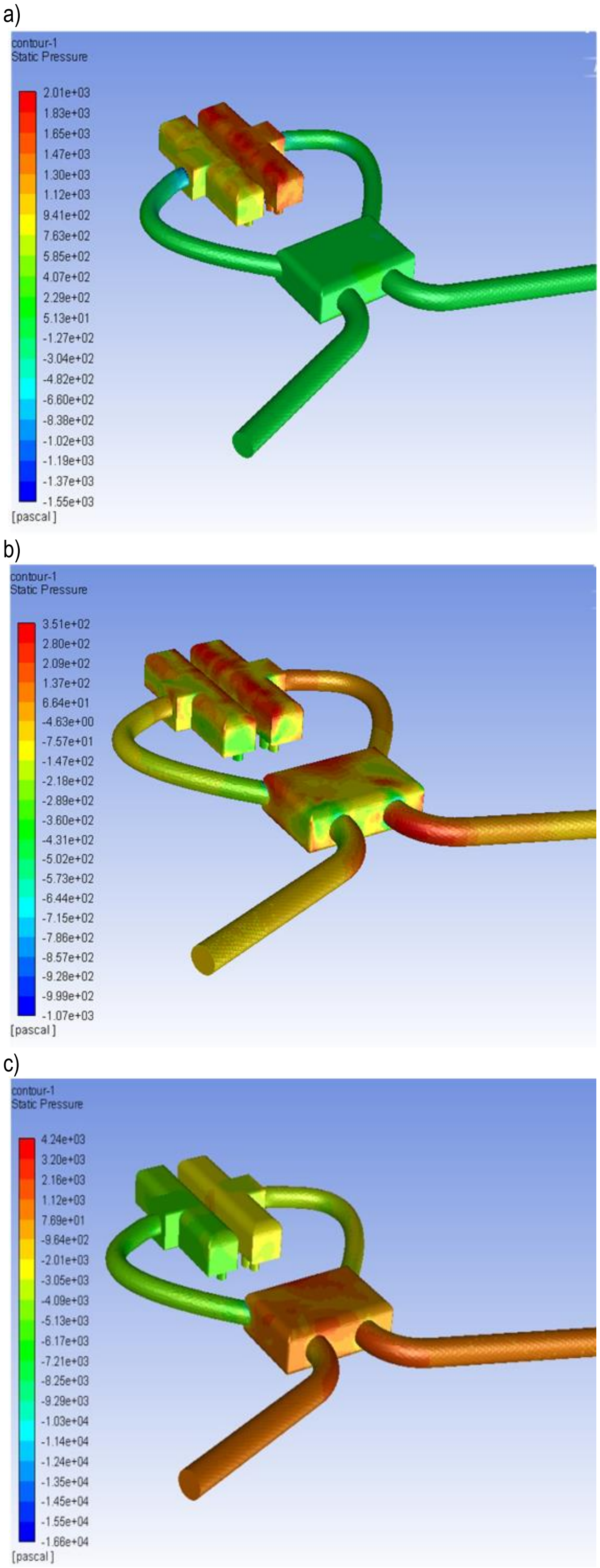

Fig. 4. Static pressure distribution a) v_p=0 m/s; b) v_p= 36 m/s; c) v_p= 94 m/s

Based on the analysis of Figs. 3 and 4, it can be stated that at 0 m/s air inflow, dynamic engine charge can be observed. In the case of speeds v_p = 36 and 94 m/s, the share of this phenomenon in the process of filling cylinders is smaller or harmful.

Speeds and turns as well as directions of velocity vectors over the entire chamber area and current lines between the influence and outflow of fluid in the filtration chamber were determined (Fig. 5, 6).

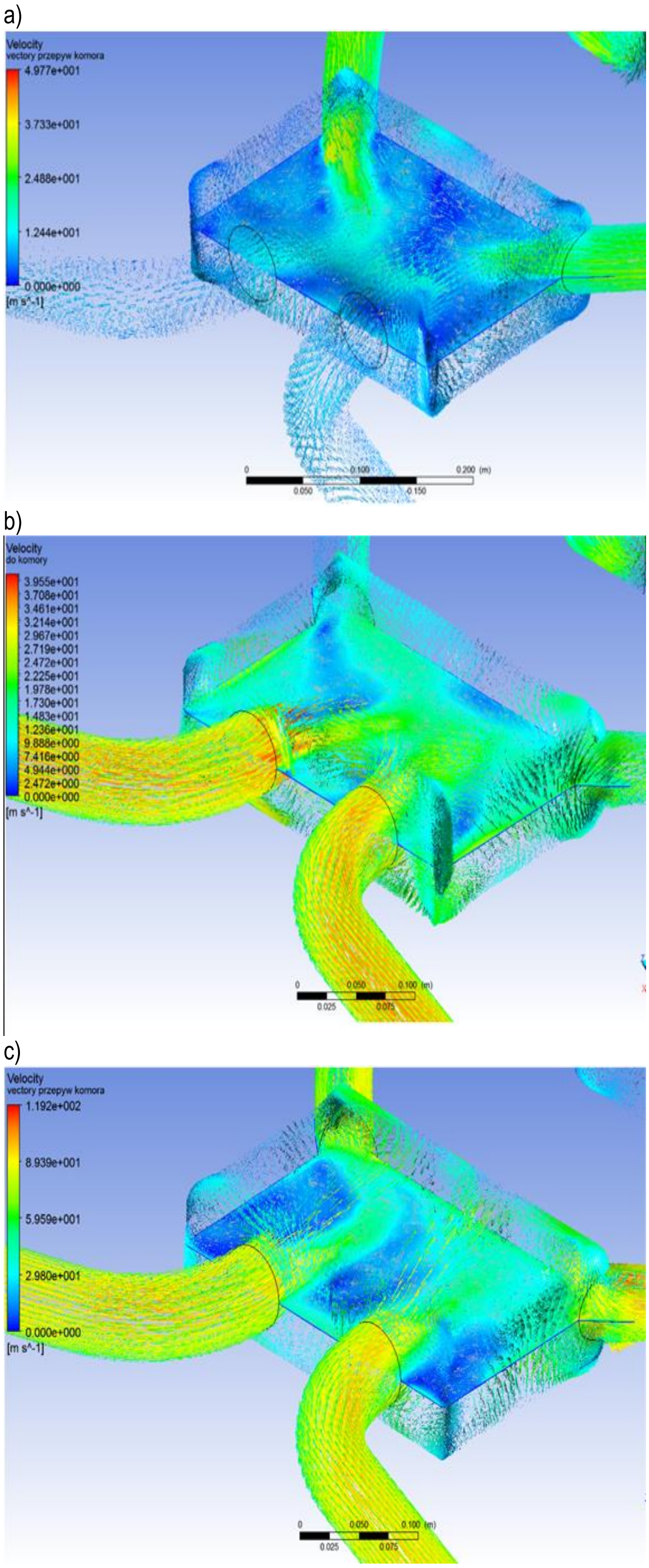

Fig. 5. Presentation of velocity and directions and returns of velocity medium in the filter chamber a) v_p=0 m/s; b) v_p= 36 m/s; c) v_p= 94 m/s

a)

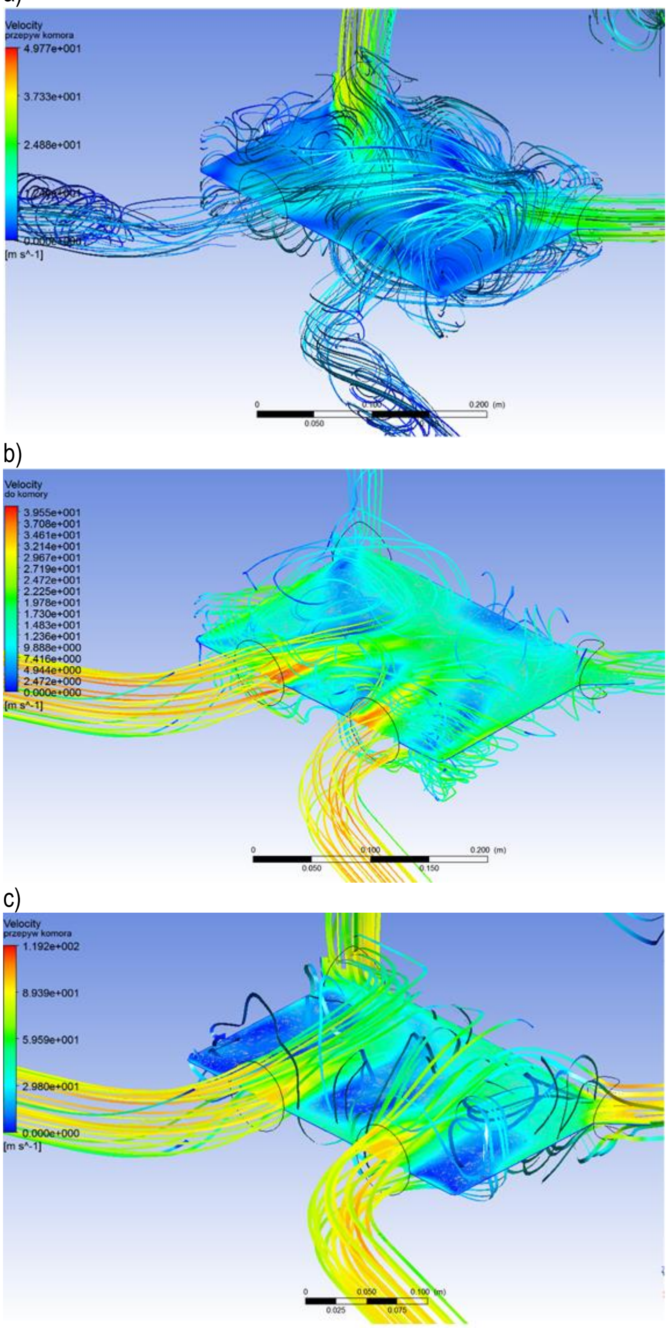

Fig. 6. Current lines between the inflow and outflow of medium in the filter chamber a) v_p=0 m/s; b) v_p= 36 m/s; c) v_p= 94 m/s

Analyzing Figs. 5 and 6, it can be concluded that the worst case is the inflow velocity v_p = 36 m/s, because the fluid turbulence that occurs at that time reduces the volumetric efficiency.

Summary

The analysis of simulation data presented in the article showed that the air velocity at the inlet to the engine's inlet system has a significant impact on the total distribution of air velocity in the system, static pressure and turbulence. The velocity vectors and flow directions graphically represent the places where the air has the highest flow velocity values. The obtained test results also indicate unevenness of the air flow. In the case when the air velocity at the inlet is 0 m/s, the distribution of the total velocities and static pressures is without significant turbulence or turbulence. As in the case of study no. 2, for v_p = 36 m/s. This is caused by the demand of the engine in the air and by the capacity of the inlet system. Consider the increasing pressure at the inlet and the filter chamber when considering the third case.

In the CFD calculation methods, the interpretation of results is an important aspect. These methods give results that in many cases are demonstrative and not always consistent with the physical characteristics of the phenomenon. They can be used as preliminary tests for real-world tests, initial selection of structural parameters and designing changes. By extending knowledge in the field of frequency analysis of pulsation of pressure with the influence of work, design and thermodynamic parameters on the spectrum of the signal, a tool can be created for effective and quick assessment of changes in the system.

List of markings

A_{min} [mm^2] – minimum surface
d [m] – diameter of the flow cross-section
n [rpm] – engine speed
P [kW] – engine power
Q [m^3/s] – engine air demand
T1 [K] – air temperature at the inlet to the intake system
v [m/s] – air flow speed
v_p – air speed at the beginning of the intake system
V_{ss} [dm^3] – engine displacement
δ [mm] – the length of the element parameter
η_v – volumetric efficiency
μ [kg/(m•s)] – dynamic viscosity
ν [m^2/s] – kinematic viscosity
ρ [kg/m^3] – air density

Bibliography

1. Teodorczyk A., Rychter T.: Teoria Silników Tłokowych. Wydawnictwa Komunikacji i Łączności, Warszawa 2018.
2. ANSYS-Fluent-Tutorial-Guide_r170.pdf.
3. Stryczek S., Napęd hydrostatyczny T. 1 Elementy, Wydawnictwa Naukowo-Techniczne, Warszawa 2005.
4. Mączyński J., Mechanika płynów, Państwowe Wydawnictwo Naukowe, Warszawa, 1966.
5. Luft S., Podstawy budowy silników, Wydawnictwa Komunikacji i Łączności, Warszawa 2003.
6. https://auto.ferrari.com/en_EN/sports-cars-models/past-models/348-gt-competizione/ (data aktualizacji: 05.04.2019).
7. Święcicki K., Konstrukcja układu dolotowego silnika spalinowego, Modelowanie Inżynierskie, tom 26, nr 57, Polskie Towarzystwo Mechaniki Teoretycznej i Stosowanej, ISSN 1896-771X.
8. Merkisz J., Pielecha J., Radzimirski S., Emisja zanieczyszczeń motoryzacyjnych, Wydawnictwa Komunikacji i Łączności, Warszawa 2012.
9. King F.R.B.: The inertia theory of engine breathing. Automotive Engineering No. 3–5/1967.
10. https://autokult.pl/t/50447,ferrari-348 (data aktualizacji: 12.01.2019).
11. http://zss.lublin.eu/wp-content/uploads/2016/09/2.5-Proces-spalania.pdf (12.01.2019).
12. Wajand J. A., Wajand J. T., Tłokowe silniki spalinowe średnio- i szybkoobrotowe, Wydawnictwa Naukowo-Techniczne, Warszawa, 2003.
13. ANSYS Meshing User's Guide_r130.pdf. ANSYS 2013.
14. https://docs.plm.automation.siemens.com/tdoc/nx/10/nx_help#ui

d:id627236 (data aktualizacji: 13.01.2019).
15. https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_ug/flu_ug_sec_bc_porous_ media.html (data aktualizacji: 13.01.2019).
16. FLUENT User's Guide. ANSYS 2013.
17. Bakuniak W., Symulacje przepływowe kolektora dolotowego dla bolidu Formuła Student, Praca dyplomowa inżynierska, Poznań 2015.
18. Dykier M., Flakiewicz M., Analiza częstotliwości drgań czynnika roboczego w układzie dolotowych silnika ZI, Czasopismo Techniczne, Wydawnictwo Politechniki Krakowskiej, z. 8-M/2008.

Wpływ prędkości powietrza na wlocie do układu dolotowego na parametry przepływu

W artykule przedstawiono analizę CFD (obliczeniowej dynamiki płynów) układu dolotowego samochodów sportowych Ferrari 348 dla trzech wariantów prędkości wlotu powietrza do układu. Artykuł zawiera analizę rozkładu prędkości i nacisków statycznych. Ponadto zbadano lokalną prędkość i przepływ w komorze filtracyjnej. Określono wpływ prędkości powietrza na wlocie do układu dolotowego na wyżej wymienione parametry. Analiza jest wstępem do badania zjawisk falowych zachodzących w układzie dolotowym, wpływających na współczynnik wypełnienia cylindrów. Do analizy użyto oprogramowania Ansys Fluent.

Słowa kluczowe: układ dolotowy, analiza CFD, silnik spalinowy, modelowanie, przepływ, zjawiska falowe, Ansys

Authors:
dr inż. **Monika Andrych-Zalewska-** Wrocław University of Science and Technology, Faculty of Mechanical Engineering, Division of Automotive Engineering,
Braci Gierymskich 164 street, 51-505 Wroclaw
e-mail: monika.andrych@pwr.edu.pl
phone: +48 713477918
dr inż. **Wojciech Ambroszko-** Wrocław University of Science and Technology, Faculty of Mechanical Engineering, Division of Automotive Engineering, Braci Gierymskich 164 street, 51-505 Wroclaw
e-mail: wojciech.ambroszko@pwr.edu.pl
inż. **Łukasz Wielki-** Wrocław University of Science and Technology, Faculty of Mechanical Engineering,
e-mail: lukaszwielki@gmail.com
inż. **Krzysztof Ziora-** Wrocław University of Science and Technology, Faculty of Mechanical Engineering,
e-mail: k01zorbo@gmail.com