ANNALS OF FACULTY ENGINEERING HUNEDOARA
– International Journal of Engineering
Tome XI (Year 2013) – FASCICULE 2 (ISSN 1584 – 2665)

[1.] Dan MOLDOVANU, [2.] Florin MARIASIU, [3.] Bogdan VARGA

THE INFLUENCES OF AIR TEMPERATURE ON THE INTAKE PROCESS BY NUMERICAL SIMULATION

[1-3.] TECHNICAL UNIVERSITY OF CLUJ-NAPOCA, ROMANIA

ABSTRACT: The intake process of an internal combustion engines is one of the most important. The importance of the intake process study is underscored by the need to generate superior engine performance, but also directly related to reducing emissions. There are a multitude of parameters that influence the intake process (functional, constructive, and environmental) that involving a series of complex studies and experiments. One of the easiest ways for researchers is to use computer modeling and simulation, which can provide early indications and new directions in the optimization of the study. This paper shows the influence of ambient temperature on the intake process of an internal combustion engine. The turbulence kinetic energy of the intake air was analyzed.

KEYWORDS: air intake process, air temperature, turbulence kinetic energy, computer simulation

INTRODUCTION

Functioning of the internal combustion engine is well established by the four major functional processes well known as the “motor cycle”: intake, compression, combustion and exhaust. Each of these processes has their importance, in order to increase overall energy efficiency of the internal combustion engine. And further, each process is influenced by several internal and external factors: physically, functional, constructive and nature of fuel [1,3,9].

The importance of cylinder air-charging process is given the necessity to fill the engine cylinder as large is possible quantity of fresh air, to achieve a greater amount of air-fuel mixture. As amount of air-fuel mixture will be higher, both functional motor performance (power) and dynamic characteristics will be higher [3]. Unfortunately, the structural characteristics and design of modern internal combustion engines have a number of features that have immediate effects on the quality of the filling process.

An ideal process of filling the engine cylinder with air should provide a laminar flow air intake for the entire length of the route. In practice, this is not possible, and evens more; the pattern of airflow through intake system (laminar or turbulent) is influence by the air inlet temperature. Air intake temperature influences (by specific physical processes) the parameters characterizing air (density), which further influences the degree of charging of the engine cylinder.

With few exceptions (variable tracks of the air inlet pipes or design characteristics of air filter – [7]), the design of air intake system is well determined, specifically designed to each specific engine that equips it. Specific restrictions related to the design of the air intake system occurred lately due to application of modern concepts of "downsizing", concepts that make intake systems to have a design process to high achievements and engine specification implementation.

Using computer simulation to design engine’s components, systems and study of functional processes is a tool widely used by researchers worldwide [2,3-8,10]. The main advantages offers using computer simulation consist in lower construction costs prototypes; develop an idea into a much shorter time than traditional experimental procedures; propose, conduct and analyze a variety of scenarios related to the characteristics of studied phenomena, etc.

For the simulation of fluid dynamic phenomena simulation software packages known collectively as CFD (Computer Fluid Dynamics) are used. Basic mathematical apparatus used in the study of fluid dynamic equations are based on Navier-Stokes equations.

Using Einstein notation the equations are written as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \ \text{ on } \Omega \qquad (1)$$

$$\frac{\partial u_j}{\partial t} + \frac{\partial u_i u_j}{\partial x_i} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\nu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \ \text{ on } \Omega \qquad (2)$$

Equation 1 represent the incompressibility and equation 2 the momentum, were S_c is the car surface, G is the ground surface, Ω a large volume around S_c and above G, $u_j(t,x)$, $p(t,x)$ and ρ are respectively the flow velocity, pressure and density. Because in real configurations the Reynolds number are very high is necessary a turbulence model to be added. The usually model embedded in CFD simulation software is k-ε turbulence model and with those considerations the equation 2 can be rewritten as:

$$\frac{\partial \bar{u}_j}{\partial t}+\frac{\partial \bar{u}_i \bar{u}_j}{\partial x_i}=-\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i}+\frac{\partial}{\partial x_i}\left[\left(\nu+\nu_t\right)\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right)-\frac{2}{3}\delta_{i,j}k\right] \tag{3}$$

where ν_t is the eddy viscosity and is related to the turbulent kinetic energy k and rate of dissipation ε given by:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

These equations take into account the various physical parameters to characterize the fluid flow; parameters that can predict future space-time behavior. In terms of the theme presented in this article, it is observed that air turbulence (and related turbulent kinetic energy) is an important factor in defining the flow regime, so the use of computer simulation is an easy alternative to analyze the influence of ambient temperature on the air charging process for a modern intake system of an internal combustion engine with four-cylinder.

CFD simulation software package used in this numerical study is FIRE, software developed by AVL List GmbH [11].

FIRE simulation package offers: a CFD environment that is specialized on automotive research and development, and an adjustable modeling depth, that also accommodates the users own equations and calculus and can be easily integrated in any computer system. Also the software can solve the most demanding flow problems, always taking into account the complex geometric shapes, physical and chemical modeling.

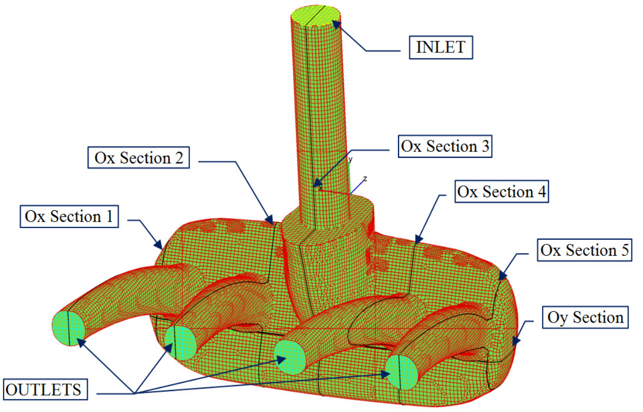


Figure 1. Built mesh, the selections for the inlet and outlet, and the sections perpendicular to the X axis (Ox) and the sections perpendicular to the Y axis (Oy)

EXPERIMENTAL RESEARCH

The modeling starts by exporting the intake pipe from any graphic model (3D Solid Works model, 3D Catia model), to an *.stl file, a format that can be imported in AVL FIRE. The next step is to make the selections and generate the mesh of model and define the boundaries (Figure 1).

Table 1. Boundary conditions for the inlet

Sel. for BC	BC_inlet	
Name of BC	BC_inlet	
Type of BC	Inlet/Outlet	
Inlet/Outlet	Mass Flow	
Activate Flow Direction	Deactivate	
Mass flow	0.0038 kg/s	
Fixed temperature	Yes	293.15 K
Fixed scalar	Yes	1

Table 2. Boundary conditions for the 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

The mesh had to be created so that the maximum number of nodes does not exceed 1 million, so that the simulation may run faster. Also, after generating the mesh it has to be checked so that there are no negative surfaces or negative volumes that can diverge the simulation.

After creating the mesh, a case is created on that mesh and the *.ssf (solver steering file) has to be edited, where all the input data is provided, the most important are: the governing equations and the 3D output file generation.

The only variables for this simulation were the mass flow for the inlet and the temperature, which were modified in accordance with the density of the intake air. The other inputs of the boundary conditions were kept the same for all simulations (Table 1 and Table 2).

RESULTS

After having all the simulation cases solved, the results can be extracted. In order to have a better view of the results, some sections were made and the Turbulence Kinetic Energy (TKE) was analyzed. There are five sections in planes that are perpendicular on the X axis (Ox) and one section perpendicular on the Y axis (Oy), placed as presented in Table 3. The Ox 4 section plane was automatically selected, and it follows the maximum value of the Turbulence Kinetic Energy.

Table 3. Position of the Cut sections

Sections	Ox 1	Ox 2	Ox 3	Ox 4	Ox 5	Oy
X position	0.114	0.041	0	-0.549 (Max value for TKE)	-0.113	0
Y position	0	0	0	0	0	0

Table 3. Centralized results of the Turbulence Kinetic Energy and Flow Velocity

Tempe-rature	Air Density	Mass flow	Turbulence Kinetic Energy	Flow Velocity
°C	kg/m^3	kg/s	m^2/s^2	m/s
35	1.1455	0.003710	0.5718	4.507
20	1.2041	0.003900	0.6024	4.611
10	1.2466	0.004038	0.7579	4.884
0	1.2922	0.004185	0.815	5.056
-10	1.3413	0.004344	0.8433	5.237
-20	1.3943	0.004516	0.9575	5.437

The results of the Turbulence Kinetic Energy and Flow Velocity were centralized in Table 4. An example of a 3D result of the Turbulence Kinetic Energy for a simulated temperature of 30°C, top view, is presented in Figure 2 and for -20°C is presented in Figure 3.

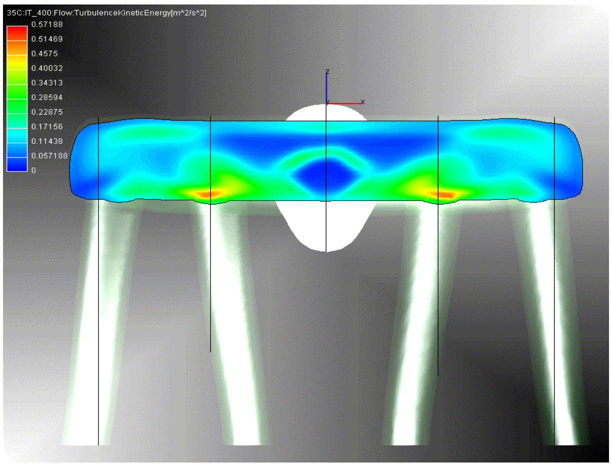


Figure 2. Top view of 3D result of the Turbulence Kinetic Energy for a simulated temperature of 30°C

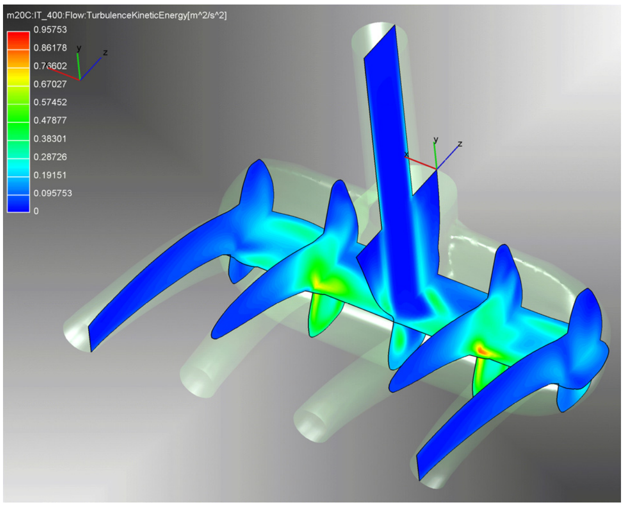


Figure 3. 3D view of the Turbulence Kinetic Energy in the selected sections, for the -20°C air temperature

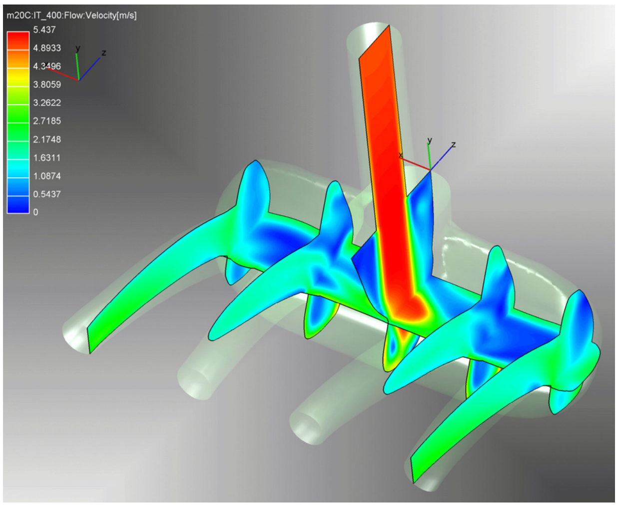


Figure 4. 3D view of the Flow Velocity in the selected plane cuts

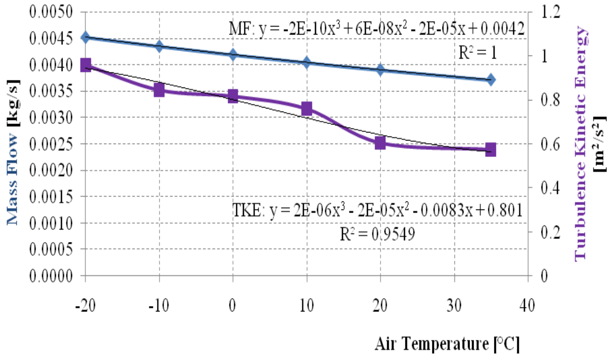


Figure 3. Turbulence Kinetic Energy and the Mass Flow with the air temperature rise

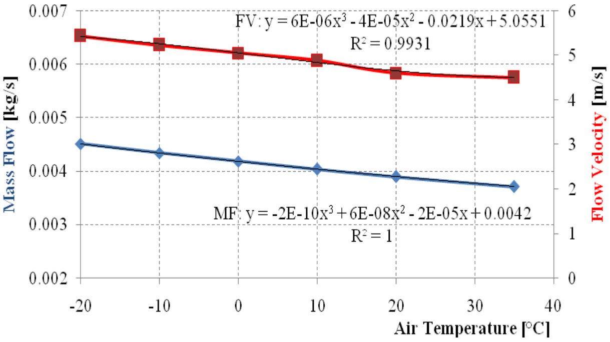


Figure 5. Flow Velocity and the Mass Flow with the air temperature rise

The Turbulence Kinetic Energy was represented with the Mass Flow, in Figure 4, depending on the rise of the intake air temperature.

Also, a 3D view of the Flow Velocity is presented in Figure 5. Figure 6 shows the Flow Velocity and the Mass Flow dependence with the intake air temperature rise.

CONCLUSIONS

For all the graphs that were made, the third degree equation of the trend-line that follows the lines was made, and also the R^2 value was displayed.

The coefficient of determination R^2 is used in the context of statistical models whose main purpose is the prediction of future outcomes on the basis of other related information. It is the *proportion of variability in a data set that is accounted for by the statistical model. It provides a measure of how well future outcomes are likely to be predicted by the model.*

From graphs in figures 3 and 5, we can observe that the R^2 value from the Mass Flow is 1, for the Turbulence Kinetic Energy, the value is 0.9549 and for the Flow Velocity, the value is 0.9931. We can understand that the Mass Flow is linear; it varies proportionally with the intake air temperature. But the Turbulence Kinetic Energy and the Flow Velocity are not linear; the simulation results show that while the Mass Flow rises linearly with the decrease of the intake air temperature, the Turbulence Kinetic Energy has a bigger deviation than the Flow Velocity.

REFERENCES

[1.] Băţaga, N., Burnete, N., Motoare cu ardere internă, Vol. I şi II, Litografia UTC-N, Cluj-Napoca, 1995.

[2.] Brady, J.M (2003), A Simple high efficiency S.I engine design, SAE Paper No: 2003-01- 0923.

[3.] Burnete, N., Naghiu A., Rus I., ş.a., Motoare Diesel şi Biocombustibili pentru transportul urban, Editura Mediamira, Cluj-Napoca, 2008, ISBN 978-973-713-217-8.

[4.] Ceviz, M.A., (2007), Intake Plenum Volume and its influence on the engine performance, cyclic variability and emissions, Energy Conversion and Management, 48, pp. 961-966.

[5.] Khan I.T., Jahanzaib M., (2011), Modelling and Flow Behavior Analysis of Inner Plenum using Computational Fluid Dynamics (CFD), European Journal of Scientific Research, 56(3), pp.346-353.

[6.] Mamat, R., Abdullah, N.R., Xu, H., Wyszynski, M.L., Tsolakis, A., (2009), Effect of Air Intake Pressure Drop on Performance and Emissions of a Diesel Engine Operating with Biodiesel and Ultra Low Sulphur Diesel (ULSD) International Conference on Renewable Energies and Power Quality (ICREPQ'09) Valencia (Spain), 15th to 17th April, 2009.

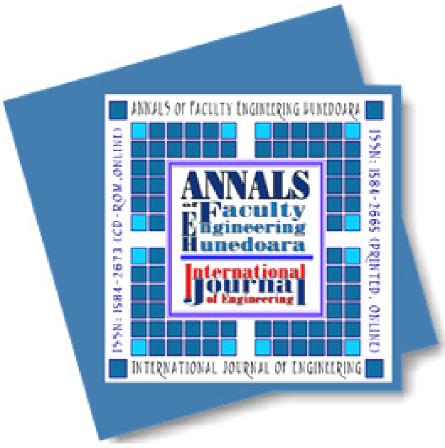
[7.] Mariasiu, F., MOLDOVANU, D., KOCSIS, L., CFD simulation of ultrasonic conditioning influences on biodiesel injection process, Research Journal of Agricultural Science, 43 (1), 327-333, 2011.

[8.] Moldovanu, D., Burnete, N., CFD SIMULATION OF A SINGLE CYLINDER RESEARCH ENGINE WORKING WITH BIODIESEL, Thermal Science, International Scientific Journal, citation impact factor 1,09. Acceptance date: 2012-01-07, Reference DOI: 10.2298/TSCI111004021M.

[9.] Negrea V.D. (1997), Motoare cu ardere interna. Procese. Economicitate. Poluare, vol.1, ed.Sidona, Timisoara.

[10.] Patil, A. S., Halbe, V. G., Vora, K.C., (2005), A System Approach to Automotive Air Intake System Development, SAE Technical Series 2005-26-011.

[11.] *** AVL List GmbH (2009) Fire User Manual, AVL List GmbH, Graz, Austria, www.avl.com.



ANNALS of Faculty Engineering Hunedoara

- International Journal of Engineering

FACULTY OF ENGINEERING HUNEDOARA,
5, REVOLUTIEI, 331128, HUNEDOARA, ROMANIA
http://annals.fih.upt.ro