Emil TOPORCER

PTNSS–2011–SC–054

Flow modeling in an intercooler

Abstract: The paper deals with the numerical modeling of charging air flow and cooling air flow in a charge air intercooler. The intercooler is air/air type and is used for the four stroke compression ignition engine with the total displacement of 7,6 l. The CATIA - CAD (Computer Aided Design) software is used for modeling of the 3D space geometrical model of the intercooler. The preprocessor Gambit is used for the creating of the needed computational mesh and the Fluent - CFD (Computational Fluid Dynamics) software is then used for the simulation itself and the postprocessing. The charge air flow in the intercooler is evaluated through the simulation results. Consequently the places with the maximum values of the velocity magnitudes for the charging air flow and the cooling air flow can be found. The simulation shows that the maximum value of velocity magnitude for the charging air flow is in the area of the intercooler inlet. The maximum value of velocity between the cooling fins is found through the simulation. Through this the information for the optimization of the intercooler can be gained. The purpose of the simulation is to use the low temperatures of charging to obtain more effective energy utilization, the decrease of the environment ecological load compared with the standard solution.

Key words: *intercooler, flow modeling, charge air, coolant, velocity magnitude*

1. Introduction

The intercooler is the part of the system for research of the charging air temperature influence on the production of exhaust emissions. The virtual model of the mentioned intercooler can be seen in Fig. 1. It is designed for the four stroke CI engine with the engine displacement of 7,6 l and rated power of 132 kW at 2200 rpm.

The design of the intercooler is examined through the simulation of the charging air flow and also cooling air flow.

The 3D parametric modeler CATIA V5 is used for the model creation. Then the preprocessor Gambit is used for mesh preparation. Finely the simulation and visualization of results are performed within the professional CFD (Computational Fluid Dynamics) software Fluent.

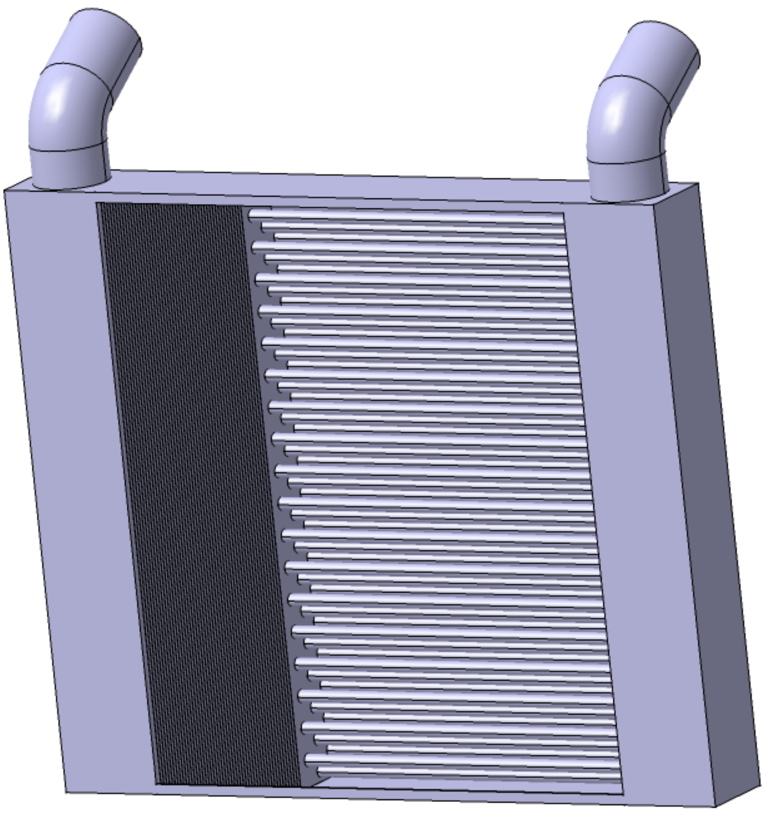

Fig. 1 Space 3D CAD model of the intercooler

2. Intercooler design

As mentioned above the intercooler is air/air type. The direction of charging air flow is perpendicular to the direction of the cooling air flow. The width of the intercooler is 570 mm, the height has the value of 470 mm. The distance between the axis of the pipes is 25 mm. The cooling fins have this parameters: height 462 mm, width 64 mm, thickness 0,2 mm. The distance between the cooling fins is 2,3 mm and there are total 165 cooling fins. The design consists of 54 pipes. The outer diameter of pipes is 10 mm and the thick of their wall is 1 mm. The active length of the pipes is 410 mm. The simplified model of the intercooler (see Fig. 1) was created because of the following use of computational software Fluent. The design of the intercooler shows the arrangement of the pipes and cooling fins – the pipes are horizontal and the fins are vertical.

The simulation is assuming the charging air inlet temperature of 155°C. The one cooling fin surface for heat transfer has the value of 0,050654 m^2. The charging air pressure before the intercooler has the value of 230000 Pa. Its mass flow rate has the value of 897 $kg.h^{-1}$. The total surface for the heat transfer from the side of cooling air is 8,9819 m^2.

The minimum flow cross-section from the side of the cooling air has the value of 0,017814 m^2.

3. Boosted air flow simulation

The model used for simulation preparation (Fig. 2) consists from the part of the intercooler because of the decrease in the demand on the used hardware. In the simulation itself only the part of the model shown in the Fig. 2 is used because of the same reason. The mentioned model is then used for the mesh creation in the preprocessor Gambit. The resultant mesh can be seen in the Fig. 3. The types of boundary conditions used are symmetry, velocity inlet, pressure outlet and wall. The final mesh consists of 1 323 792 tetrahedral cells. Then is the mesh imported into Fluent and the specific values for the single boundary conditions are specified. The value of velocity at the boosted air inlet has the value of 6,7 $m.s^{-1}$. The value of time step is 0,001 s.

The type of mathematical model used for turbulent flow simulation is renormalized groups - RNG $k-\varepsilon$ - turbulent model. This model is enough robust and suitable for the solution of turbulent flows.

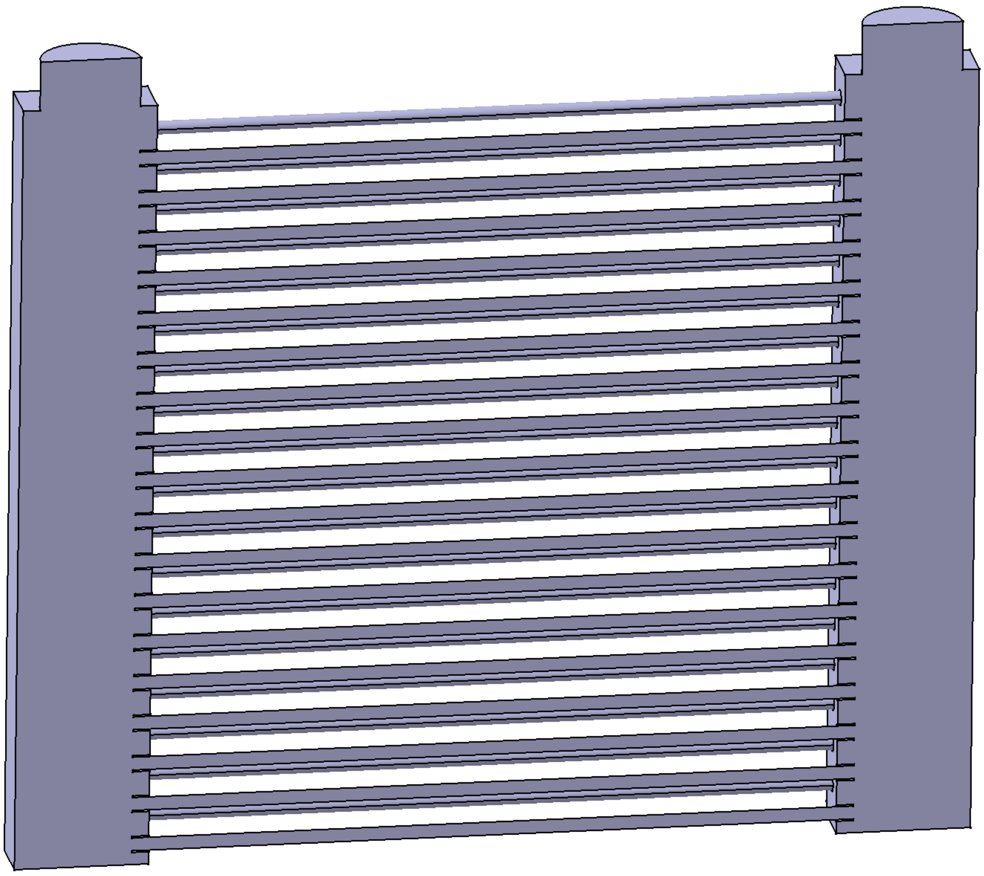

Fig. 2 The model for the simulation

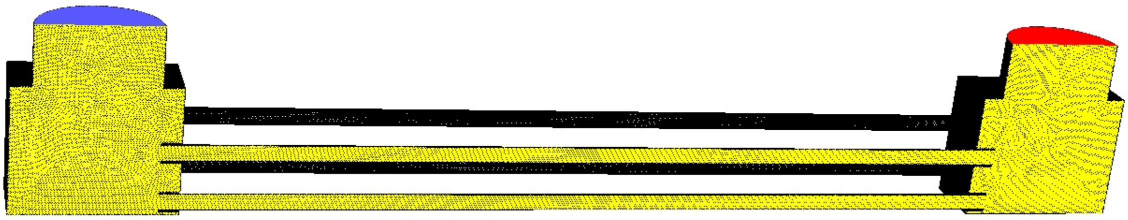

Fig. 3 The computational mesh

The equation for the motion transfer has this form:

$$\frac{\partial}{\partial t}(\rho \bar{u}_i)+\frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j)=$$

$$=\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right)-\left(\frac{2}{3}\mu_{eff}\frac{\partial \bar{u}_i}{\partial x_i}\right)\right]-\frac{\partial \bar{p}}{\partial x_i}+\rho g_i+F_i \qquad (1)$$

and transport equations have following forms:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho \pi_j k)=$$

$$=\frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)+\mu_t S^2-\rho\varepsilon \qquad (2)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}(\rho\mu_j\varepsilon)=$$

$$=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}\mu_t S^2-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R \qquad (3)$$

The simulation results can be seen in Fig. 4, 5 and 6.

Fig. 4 shows the velocity vectors distribution for the boosted air flow.

As we can see the maximum values of the velocity magnitude are in the area of the inlet into the pipes and reach the 91 m.s^{-1} (see Fig. 5).

The distribution of the velocity vectors in the area of outlet from the pipes into the common chamber can be seen in Fig. 6 and it can be seen that the maximum values of velocity magnitudes reach about 81 m.s^{-1}.

The simulation is performed on the PC with the Intel Core 2 CPU 6700 (2,66 GHz) and with the RAM of 3 GB. There was no problem with the numerical stability of the simulation.

Through the gained information the design of the intercooler can be verified and subsequently optimized.

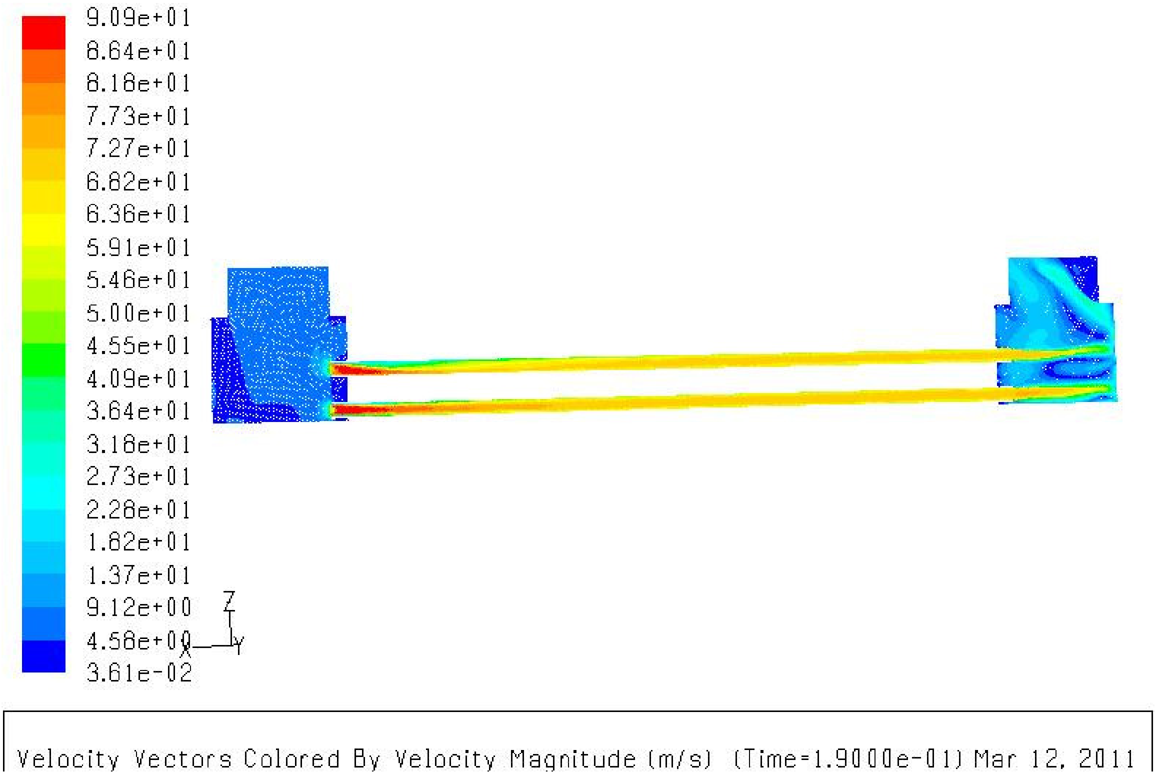

Fig. 4 Velocity vectors for the boosted air flow

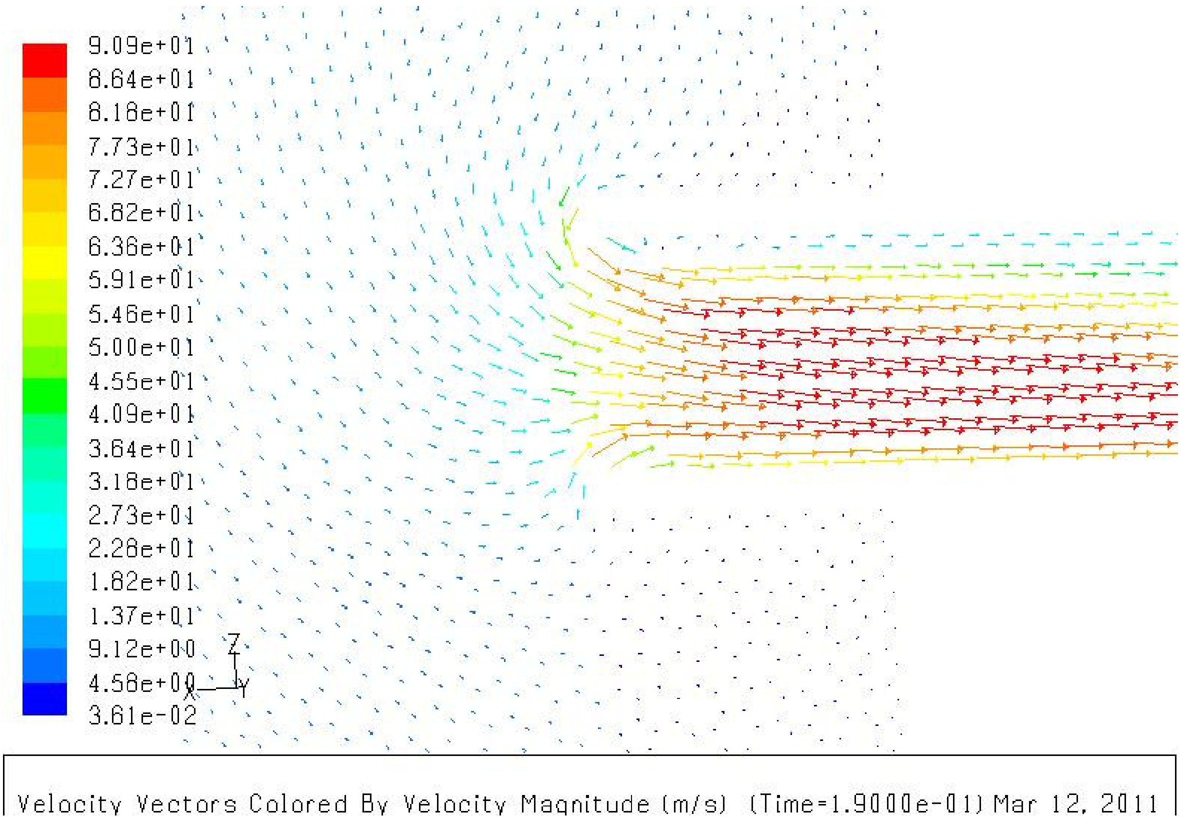

Fig. 5 Detail of the velocity vectors for the boosted air flow in the area of inlet into the pipe

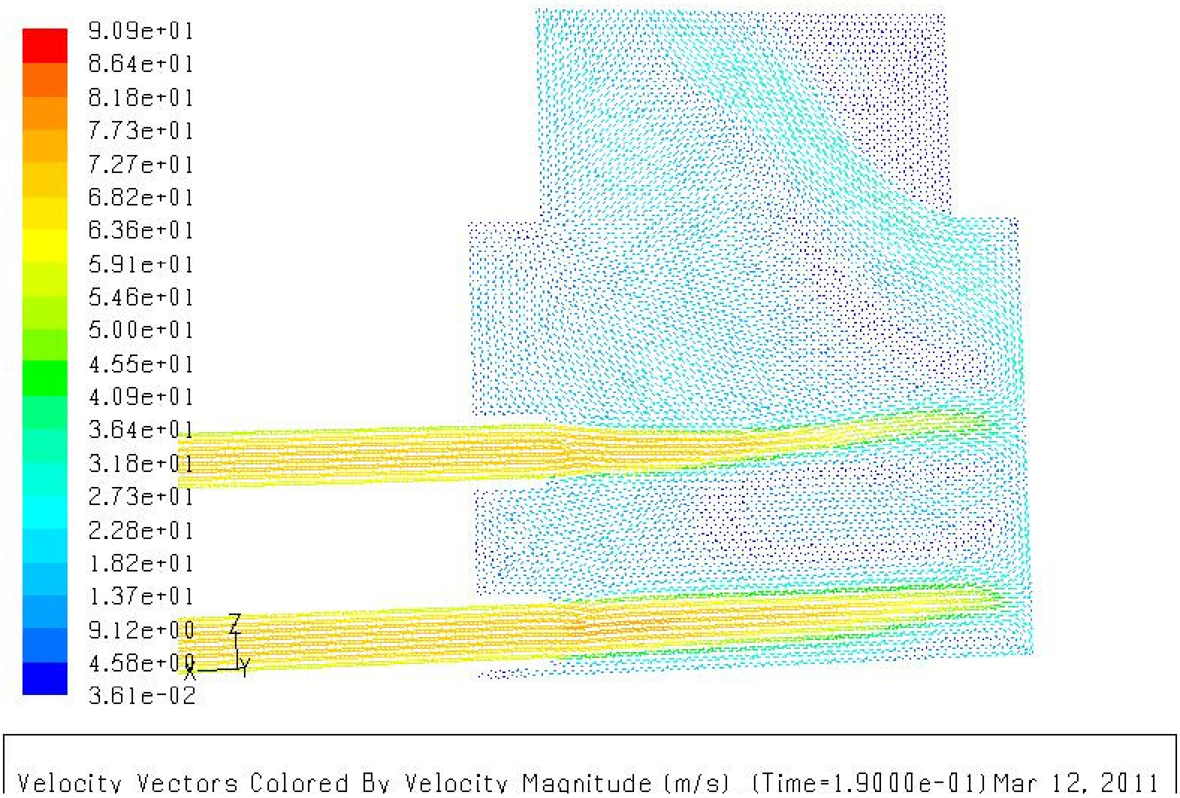

Fig. 6 Detail of the velocity vectors for the boosted air flow in the area of outlet from the pipes

4. Cooling air flow simulation

The CAD model used for the mesh generation within the preprocessor Gambit can be seen in Fig. 7.

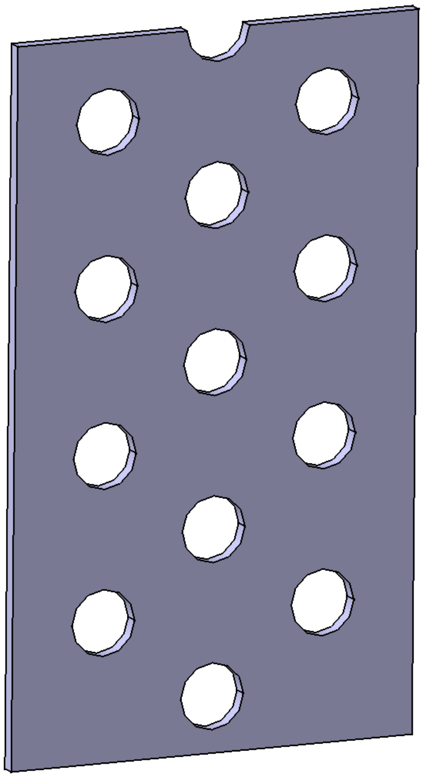

Fig. 7 CAD model of the space between the cooling fins

The model consists of the part of the space between the two cooling fins. The cooling air flows through this space. The mentioned model contains again only the part of the whole space because of the decrease in the demand on the used hardware. The detail of the resulting computational mesh can be seen in Fig. 8. The types of the boundary conditions used are symmetry, velocity inlet, pressure outlet and wall. The mesh consists of 848 529 wedge elements. Then is again the mesh imported into Fluent and the specific values for the single boundary conditions are specified. The value of velocity at the cooling air inlet is 5,5 m.s^{-1}. The value of time step is 0,001 s.

The simulation results can be seen in Fig. 9 and 10.

Fig. 9 shows the total results of the velocity vectors distribution between the cooling fins. As we can see the maximum value of the velocity magnitude is 15,2 m.s^{-1} and can be found in the proximity of pipes.

In Fig. 10 it can be seen the detail of the velocity vectors distribution within the mentioned space of the intercooler. We can see also the orientation of the velocity vectors.

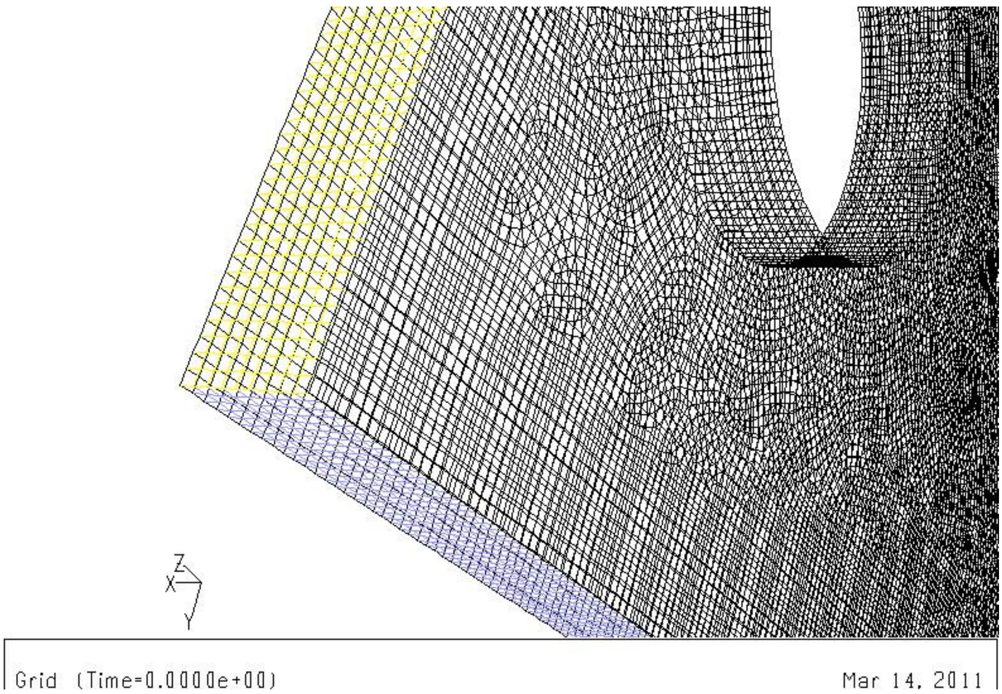

Fig. 8 The detail of the computational mesh

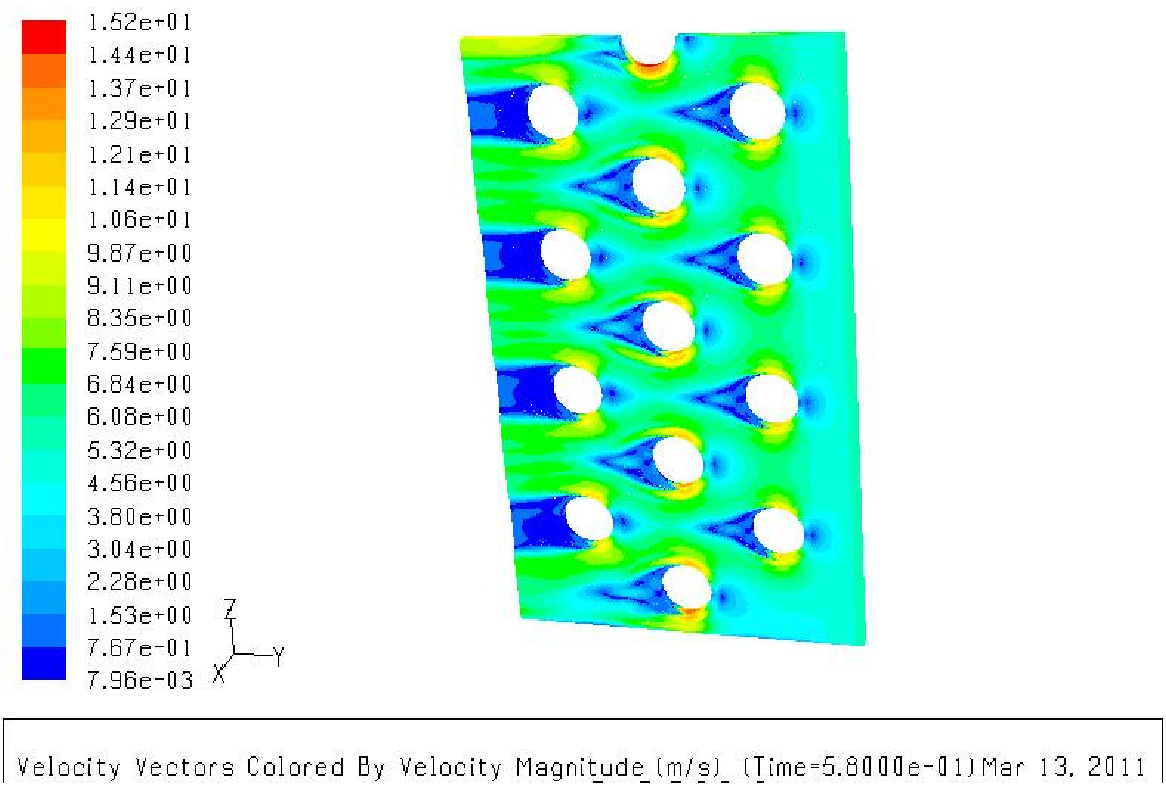

Fig. 9 Velocity vectors distribution between the cooling fins (the velocity inlet is from the right side)

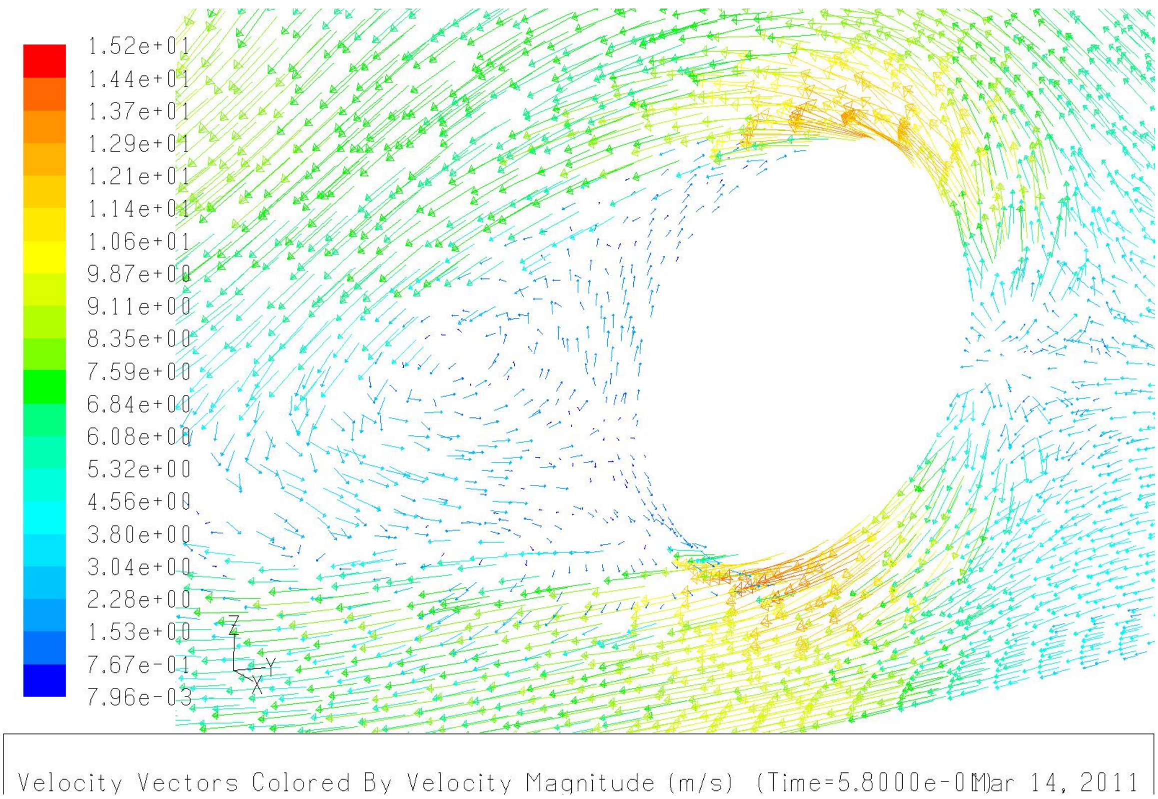

Fig. 10 The detail of the velocity vectors distribution between the cooling fins

5. Conclusion

The boosted air flow simulation shows that the maximum of velocity magnitude for the charging air flow is in the area of the inlet into the pipes and has the value of 91 $m.s^{-1}$ (see Fig. 5). The maximum value of the velocity magnitude in the area of the outlet of the pipes is 81 $m.s^{-1}$ (see Fig. 6).

The cooling air flow simulation shows that the maximum of the velocity magnitude between the cooling fins reaches the value of 15,2 $m.s^{-1}$ and can be found in the proximity of pipes.

The mentioned information give us the basis for the correct design verification and also optimization.

Both simulations are the part of the simulation performed within the project which is aimed to use the low temperatures of charging to obtain more effective energy utilization and to decrease of the environment ecological load.

The contribution was created within the framework of the project SK-PL-0035-09, which is supported by the Agency for Support of Science and Technology of the Slovak republic and the project VEGA 1/0554/10, which is supported by the Ministry of Education of the Slovak republic.

Nomenclature/*Skróty i oznaczenia*

CAD Computer Aided Design

Bibliography/*Literatura*

[1] Hudák, A., Barta, D.: Analýza prúdenia chladiaceho média v okolí valca motora Z8004C, In: Hydraulika a pneumatika: časopis pre hydrauliku, pneumatiku a automatizačnú techniku, ISSN 1335-5171, vol. 7, No. 1-2, 2005, p.52-54

[2] Chalamonski, M.: Analysis of thermal centre failuring, Eksploatacja i niezawodnosc, No. 4(36)/2007, ISSN 1507-2711.

[3] Kondepud, D.: Modern thermodynamics, John Wiley and Sons, 1998

[4] Kovalčík, A.: Pressure and flow measurement in a nonconventional energetic system, In proceedings of international conference TRANSCOM 2009, ISBN 978-80-554-0031-0, section 6, EDIS - University of Zilina, Zilina, 2009

[5] Kovalčík, A., Sojčák, D., Zvarková, D.: Combined cycle plants for cogeneration in industrial power station, In proceedings of international conference TRANSCOM 2009, ISBN 978-80-554-0031-0, section 6, EDIS - University of Zilina, Zilina, 2009

[6] Labuda, R., Isteník, R.: Vplyv dynamiky zmeny režimu spaľovacieho motora na jeho prevádzkové veličiny (Influence of combustion engine regime change dynamic on its operational quantities), In proceedings of international conference KOKA 2005, ISBN 80-01-03293-0, CTU Prague, Prague, 2005, p.173-178

[7] Labuda, R., Isteník, R., Hlavňa, V.: How to influence CO_2 production by style of vehicle control, New trends in construction and exploitation automobiles, Vehicles 2007, ISBN 978-80-8069-942-0, SPU Nitra, Nitra, 2007

[8] Sojčák, D., Hlavňa, V., Barta, D.: A structure of the cooling system, In: Communications, ISSN 1335-4205, Vol. 7, No. 4, 2005, p.23-26

Mr Toporcer Emil, Ing., PhD. – fellow in the Faculty of Mechanical Engineering at University of Žilina.