Journal of KONES Powertrain and Transport, Vol. 17, No. 4 2010

INTERCOOLER FOR EXTREMELY LOW TEMPERATURES OF CHARGING

Emil Toporcer, Peter Tučnik

University of Žilina, Faculty of Mechanical Engineering
Department of Automotive Technology
Univerzitná 8215/1, 010 26 Žilina, Slovakia
tel.: +421 41 513 2653, +421 41 513 2660, fax: +421 41 5253016
e-mail: emil.toporcer@fstroj.uniza.sk, andrej.kovalcik@fstroj.uniza.sk

Jerzy Cisek

Cracow University of Technology, Faculty of Mechanical Engineering
Institute of Automobiles and Internal Combustion Engines
Jana Pawła II Street 37, 31-864 Krakow, Poland
tel.: +48 012 628 35 30
e-mail: cisek@mech.pk.edu.pl

Abstract

The paper deals with the optimization of the charge air intercooler of the turbocharged internal combustion engine with the focus on the extremely low temperatures of charge air. The software Fluent is used for the numerical simulation (CFD - Computational Fluid Dynamics) of the charge air flow.

The 3D space geometrical model of the charge air intercooler is created in the CAD (Computer Aided Design) software CATIA. The model is simplified according to the requirements of the computational mesh creation and numerical simulation. The model is then exported as the iges file and this file is subsequently imported into the preprocessor Gambit. The imported model is repaired and a suitable mesh is then created according to the computational means properties. The last step is the definition of the types of boundary conditions. The needed types of boundary conditions are applied to the single surfaces and volumes. The completed computational mesh is then exported from Gambit and imported into the CFD software Fluent for numerical simulation. The needed values of parameters are used for the calculation definition and the simulation is started. The simulation results show the charge air flow in the charge air cooler and the places with the maximum values of turbulences can be found and this information can be the input for the optimization design of the mentioned cooler.

Keywords: *intercooler, simulation, charge air flow, coolant flow, cooling fins*

1. Introduction

The charge air intercooler is designed for the four stroke compression ignition engine with the engine displacement of 7.6 l. The intercooler is liquid/air type. The aim is to acquire information regarding the dimensions and parameters of the mentioned intercooler for extremely low temperatures of charging air. At first it is evaluated the design from the side of coolant and the places of maximum turbulences are found. Next step is the evaluation of design from the side of charge air and the places with maximal values of turbulences are found again.

2. Intercooler design and parameters

The intercooler is the liquid/air type. The direction of air flow is perpendicular to the direction of coolant flow. The number of liquid draft is 3. Each draft consists of 5 pipes. The outer diameter of pipes is 10 mm and the thick of the wall is 1 mm. The active length of the pipes is 760 mm.

The dimensions of the cooling fin are: height 150 mm, width 50 mm, thickness 0.2 mm. The distance between cooling fins is 1.4 mm. The CAD model of the intercooler can be seen in the Fig. 1. The CAD software used for the model creation is CATIA V5. The air inlet and outlet and also coolant inlet and outlet can be seen in the mentioned figure. The Fig. 2 shows the inner design of the intercooler, where the cooling fins and the pipes can be seen.

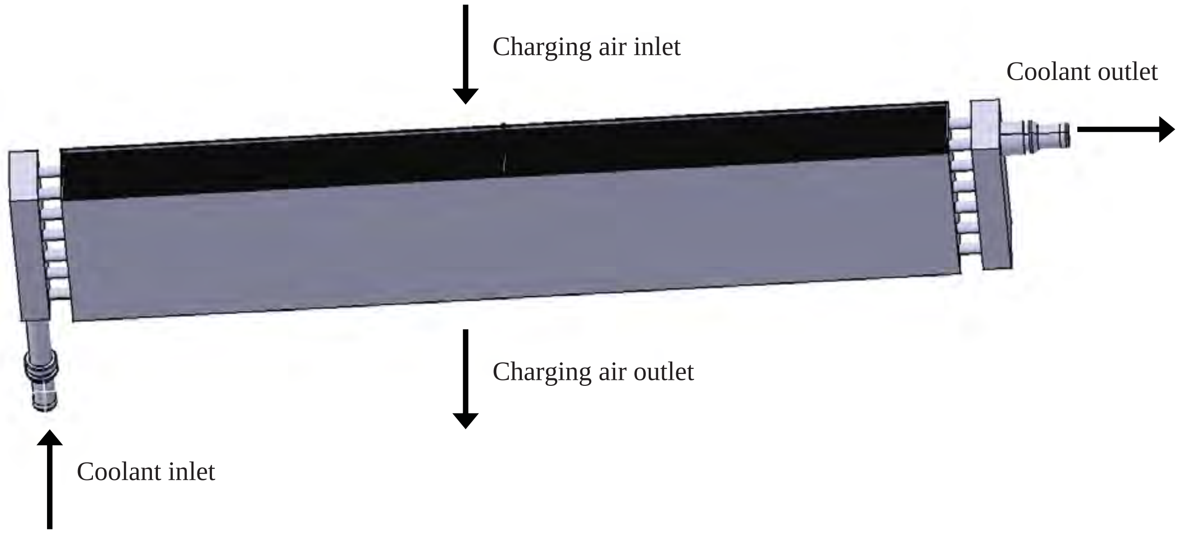

Fig. 1. CAD model of the intercooler

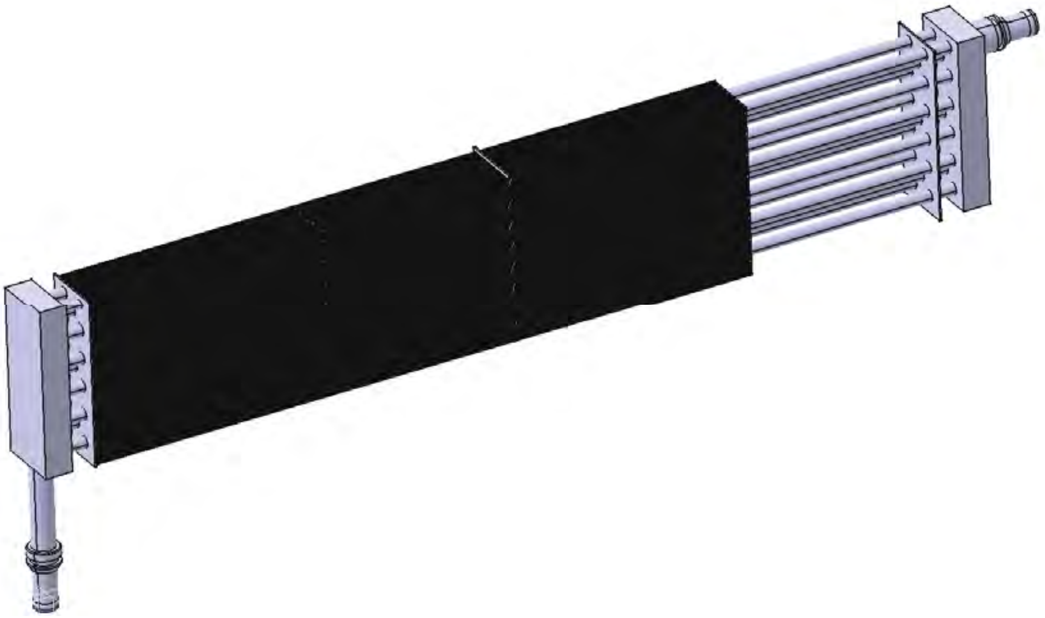

Fig. 2. Inner design of the intercooler

The simplified calculation of the intercooler is assuming the charging air inlet temperature of 157°C. The boosted air pressure before the intercooler has the value of 230 000 Pa. Air mass flow rate has the value of 897 $kg.h^{-1}$. Coolant mass flow rate is 1138 $kg.h^{-1}$. The value of one cooling fin surface for heat transfer is 0.012644 m^2. Total intercooler surface for heat transfer from the side of air has the value of 6.4 m^2. Used value of heat transfer coefficient on the side of air has the value of $\alpha_L = 78\ W.m^{-2}.K^{-1}$. For the coolant heat transfer coefficient was used value of $\alpha_W = 8158\ W.m^{-2}.K^{-1}$. Resultant heat transfer coefficient for the intercooler is calculated according the following formula:

$$k = \frac{1}{\frac{1}{\alpha_L} + \frac{S_L}{S_W} \cdot \frac{1}{\alpha_W}} = \frac{1}{\frac{1}{78} + \frac{6.393965}{0.286513} \cdot \frac{1}{8158}} . \tag{1}$$

The calculated value is 64 W.m^{-2}.K^{-1}. The values for specific heat capacity are 1005 J.kg^{-1}.K^{-1} for air and 4186 J.kg^{-1}.K^{-1} for coolant. The air outlet temperature and coolant outlet temperature dependency on the coolant inlet temperature is demonstrated in the Tab. 1.

Tab. 1. Air outlet temperature and coolant outlet temperature dependency on the coolant inlet temperature

Coolant inlet temperature (°C)	Air outlet temperature (°C)	Coolant outlet temperature (°C)
0	43.2	21.5
-5	40.8	17
-10	38.4	12.5
-15	36	7.9
-20	34	3.3
-25	32.4	-1.4
-30	31.2	-6.2
-35	30.4	-11

3. Coolant flow simulation

The CAD model of the space where the coolant flows is shown in Fig. 3.

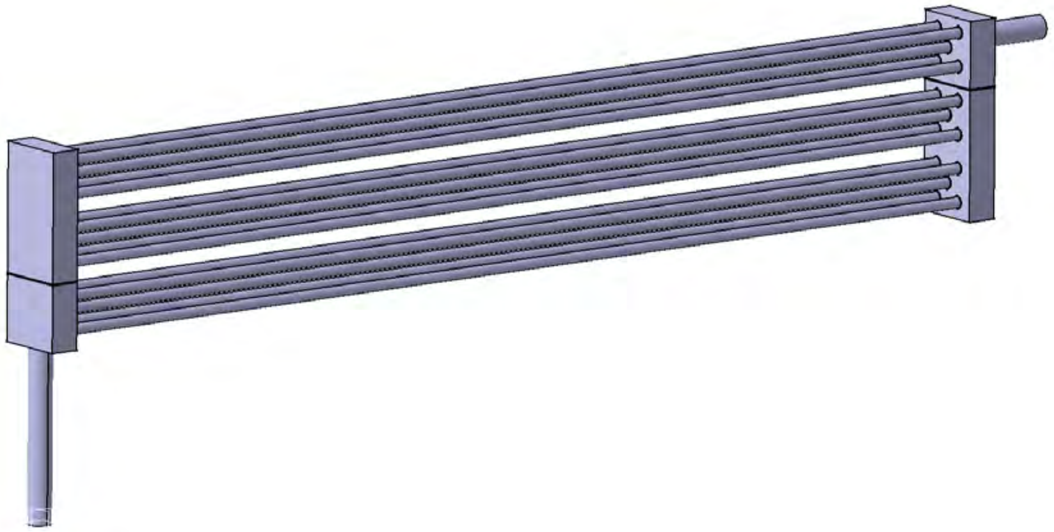

Fig. 3. Space for coolant flow

For the simulation is used one half of this space (Fig. 4) because of its symmetry and resultant possible decreasing in the requirements of the hardware parameters for simulation.

The simulation mesh is created in the preprocessor GAMBIT and then imported into the Fluent for simulation. Mesh can be seen in Fig. 5. The value for boundary conditions and other needed parameters are set. The mesh consists of 1399065 tetrahedral cells. For calculation value of velocity at coolant inlet is 1.26 m.s^{-1}.

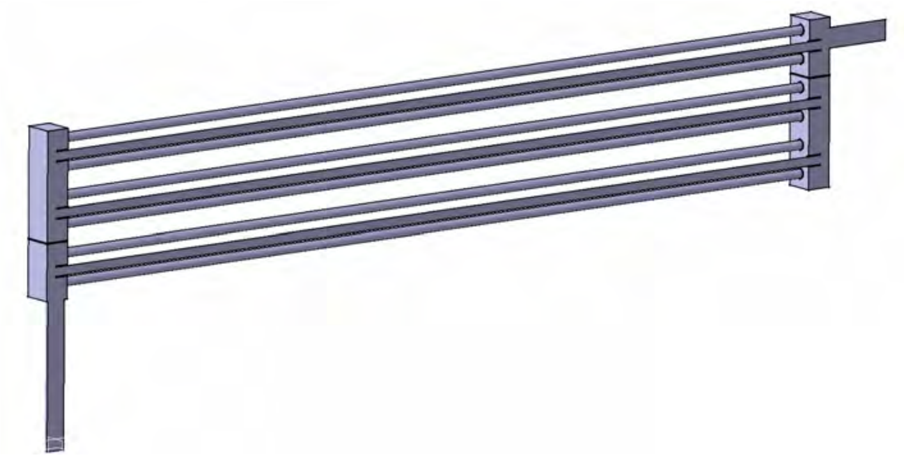

Fig. 4. 3D model used for mesh generation

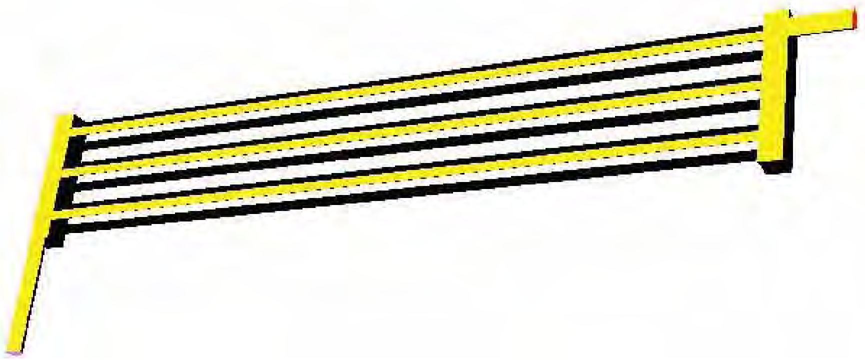

Fig. 5. Computational mesh for coolant flow

The results can be seen in Fig. 6 and 7. Fig. 6 shows the detail of the velocity vectors distribution in the area of first draft pipe in the proximity of coolant inlet. Fig. 7 shows the velocity vectors distribution in the area of coolant outlet from the second draft pipe. It can be seen that the maximum value of velocity is about 2.6 m.s^{-1}.

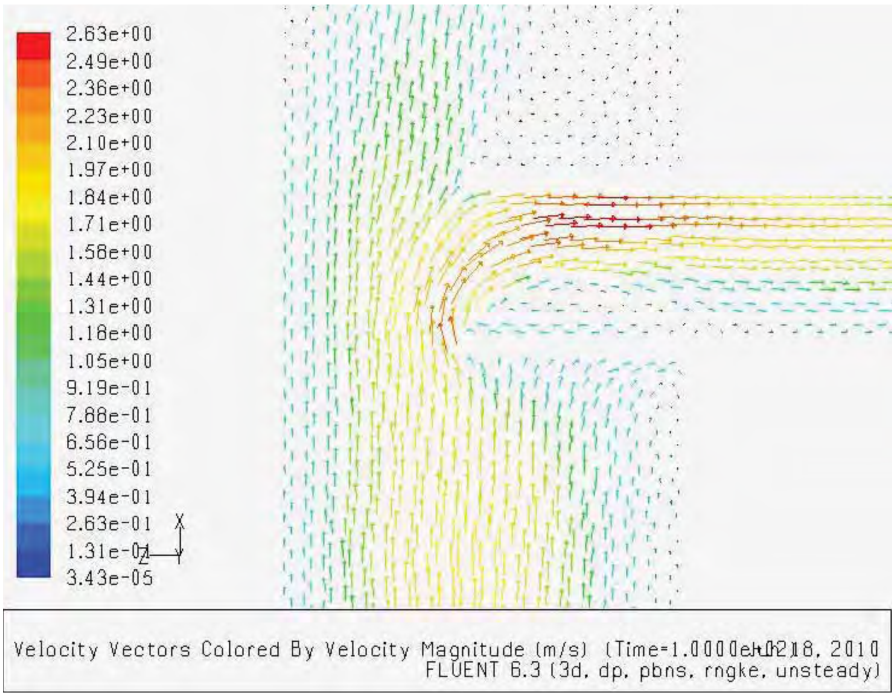

Fig. 6. Velocity vectors in the inlet of the first draft pipe

4. Charging air flow simulation between the cooling fins

The CAD model of the space between two cooling fins was created again in the CATIA. Gambit was used for mesh generation and Fluent for the calculation and postprocessing.

The results can be seen in Figs. 8 and 9. The maximal values of velocity are in the proximity of pipes and reach 13.3 ms^{-1}.

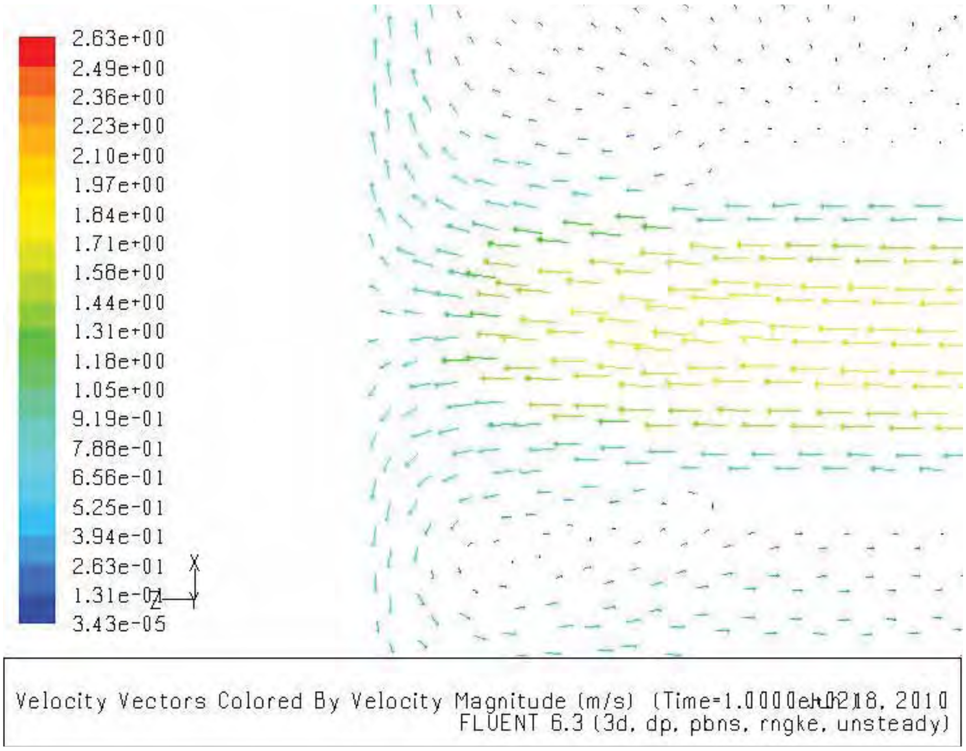

Fig. 7. Velocity vectors in the outlet of the second draft pipe

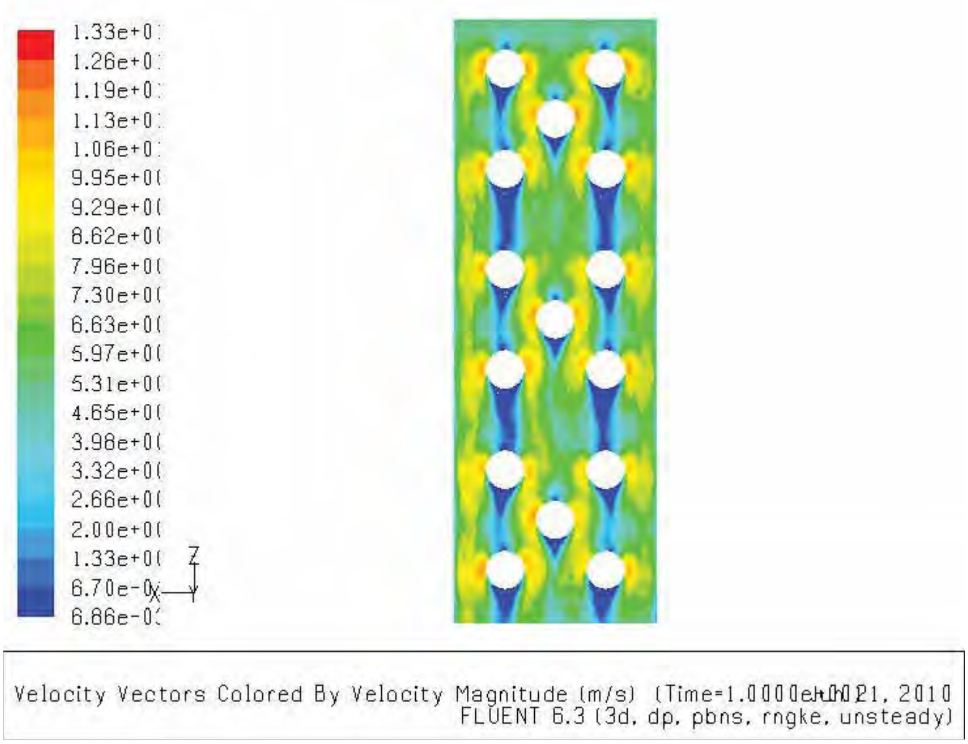

Fig. 8. Velocity vectors between the cooling fins

5. Charging air flow simulation in the intercooler assembly

The CAD model corresponding to the space under consideration can be seen in Fig. 10 together with the mesh. The symmetry of the mentioned space is used. The computational mesh consists of 1456233 tetrahedral cells. The results can be seen in Figs. 11 and 12. The maximal values of velocity are in the area of inlet and reach 120 m.s^{-1} (Fig. 12).

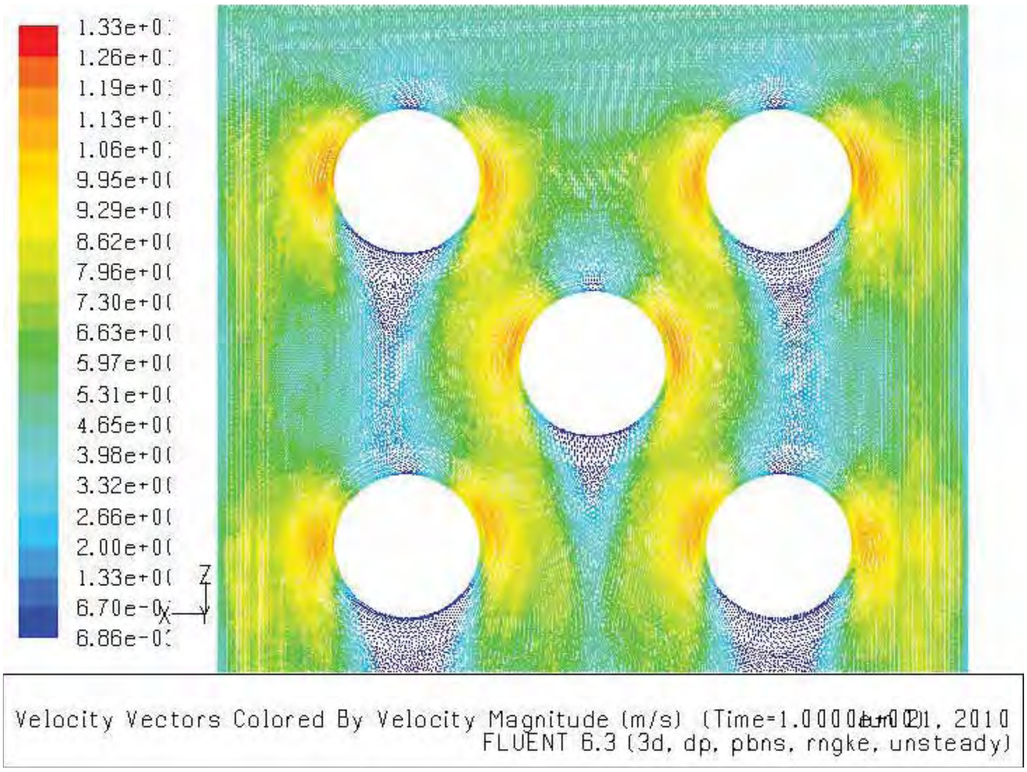

Fig. 9. Detail of velocity vectors between the cooling fins

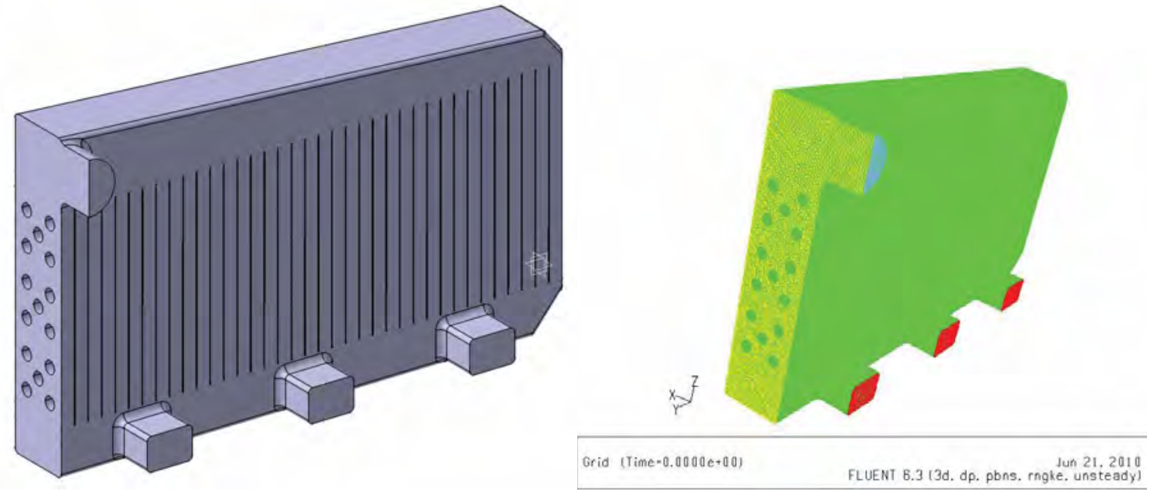

Fig. 10. 3D model used for mesh generation (on the left) and corresponding mesh (on the right)

6. Conclusion

The paper deals with the numerical simulation of air and coolant flow in the intercooler of the turbocharged internal combustion engine. CATIA was used for corresponding models creation and Fluent for simulation.

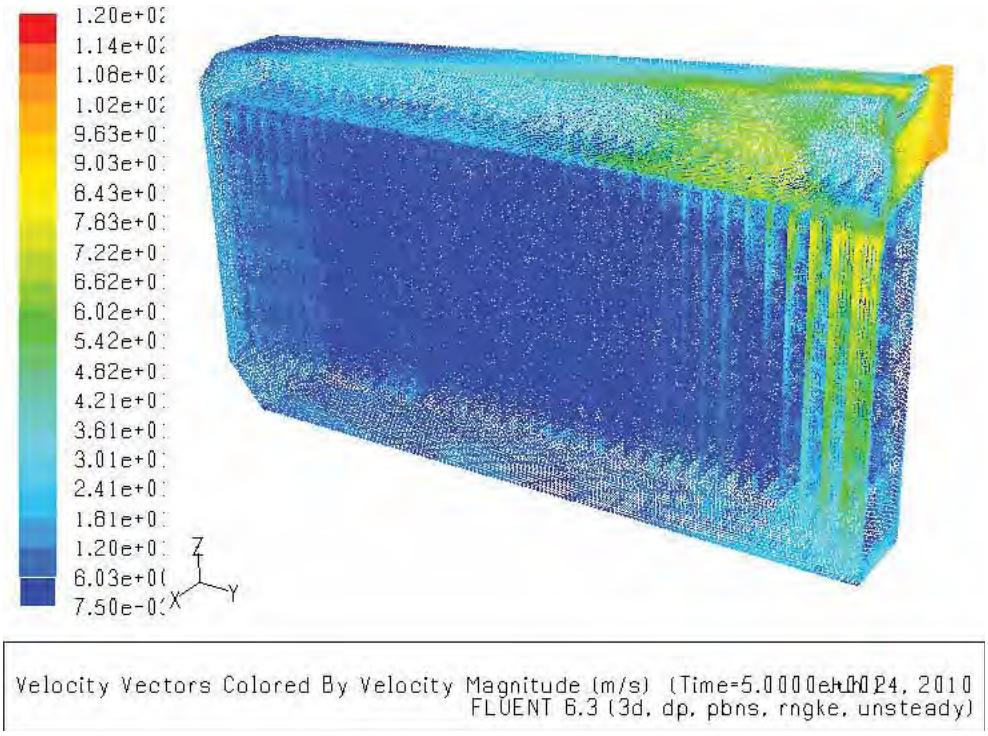

Fig. 11. Velocity vectors in the intercooler assembly

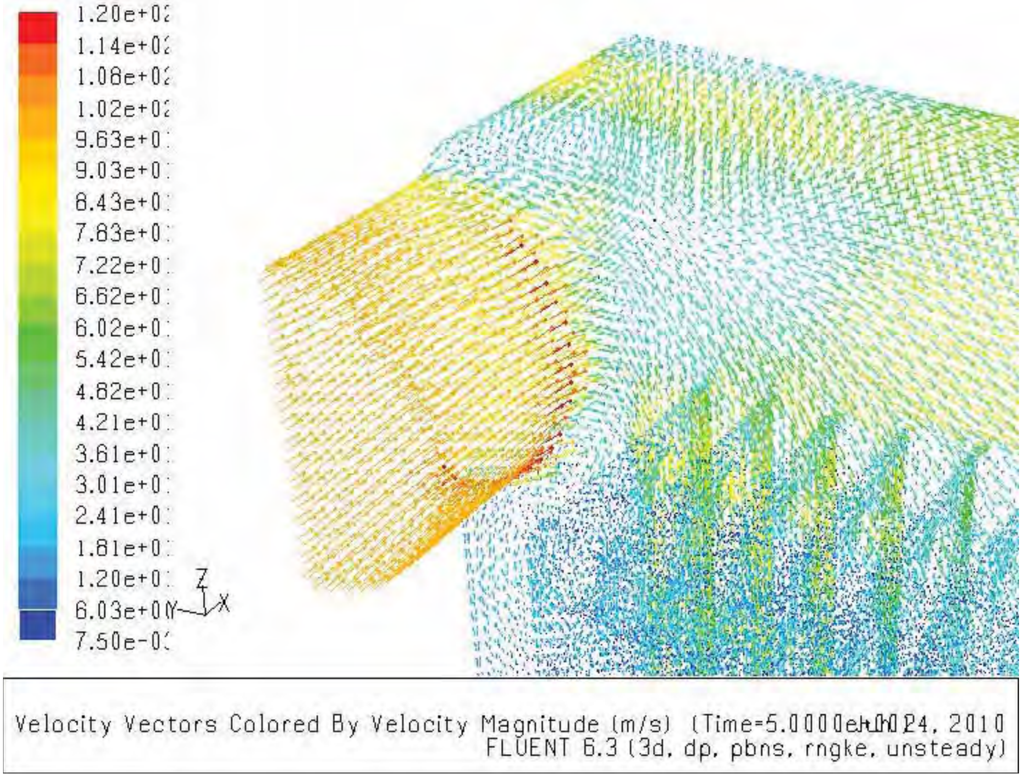

Fig. 12. Detail of velocity vectors in the area of inlet

From the coolant flow simulation results it can be seen that the maximum values of velocity magnitude is about 2.6 m.s^{-1} in the area of pipe inlet. Other result from the simulation is that the maximum values of velocity magnitude for charging air flow is about 120 m.s^{-1} in the area of intercooler inlet. Simulation shows that the maximum values of velocity between cooling fins are about 13.3 m.s^{-1}.

Acknowledgements

This contribution was created within the framework of the project SK-PL-0035-09 (SK-PL-0035-09/8027/2010), 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.

References

[1] Isteník, R., Barta, D., Mucha, W., *Influence of the wheels on the automobile dynamics*, Communications – scientific letters of the University of Žilina, EDIS ŽU, ISSN 1335-4205, 1/2004, pp.26-28, Žilina 2004.

[2] 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, SPU Nitra, ISBN 978-80-8069-942-0, Nitra 2007.

[3] SK-PL-0035-09, (SK-PL-0035-09/8027/2010).

[4] VEGA 1/0554/10.