Article citation info:
BIAŁY, M., PIETRYKOWSKI, K., TULWIN, T., MAGRYTA, P. CFD numerical simulation of the indirect cooling system of an internal combustion engine. *Combustion Engines*. 2017, **170**(3), 8-18. DOI: 10.19206/CE-2017-302

Michał BIAŁY
Konrad PIETRYKOWSKI
Tytus TULWIN
Paweł MAGRYTA

CE-2017-302

CFD numerical simulation of the indirect cooling system of an internal combustion engine

The paper presents an analysis of the fluid flow in the cooling system of an internal combustion engine with oposite pistons. The purpose of the work was to optimize the flow of fluid through the channels located in the engine block. Simulation studies and subsequent iterations were performed using Ansys Fluent software. Two-equation k-epsilon turbulence model was used in the simulation model. Boundary and initial conditions were taken from previously made simulations conducted in AVL Boost software. The average wall temperature of the cylinder and the temperature of the outer walls of the cylinder were assumed for simulations. The results of the analyzes were graphically illustrated by the speed streamline distribution of velocity fields and temperature.

Key words: *Ansys Fluent, combustion engine, computational fluid dynamics CFD, cooling system*

1. Introduction

CFD method is one of the most rapidly growing nowadays. It is used in all areas of life. Beginning with medicine, by simulating blood flow in major arteries and finishing with the engineering activities, i.e. simulations of the flow around the obstacle. CFD allows to reduce the cost of producing an item. Rather than producing and testing an object by experimentation, it can be tested using the simulation tools. CFDs provide the opportunity for modification of geometry, changing boundary conditions and observing their impact on the key parameters. Both fluid and energy flows can be analyzed [1, 3].

The internal combustion engine generates the energy contained in the gas pressure and the heat from the combustion of the fuel in each cycle. On one hand the energy of the gas pressure is converted to mechanical energy. On the other hand, heat energy must be drained out of the system.

Cumulating of heat energy can lead to an increase in the thermal loads of individual engine components, thus accelerating their wear and tear. Too low or too high heat energy directly affects the combustion process, significantly worsening it. Therefore, it was decided to carry out a numerical studies using the CFD method to calculate the amount of heat flowing from the combustion chamber to the cooling system, through the fluid jacket placed in the engine block. As a research object, a research three cylinder engine with a opposite pistons was used [5].

2. CFD simulations

The element that receives the heat from the system in the internal combustion engine is the fluid (working medium). There are two different methods of receiving the energy: forced or gravitational, air or coolant. In the presented engine with opposing arrangement of the pistons (Fig. 1) the cylinders are placed in the block (wet bushings). In each of the cylinders between intake and outlet windows, the cylinder wall is in contact with the cooling fluid. The heat generated during the combustion of the fuel is transported to the outlet manifold (through the outlet windows) along with the hot exhaust gases and to the cylinder walls. Then the energy is taken over by the cooling fluid and then transported to a radiator in which the energy is dispersed into the environment [6, 8].

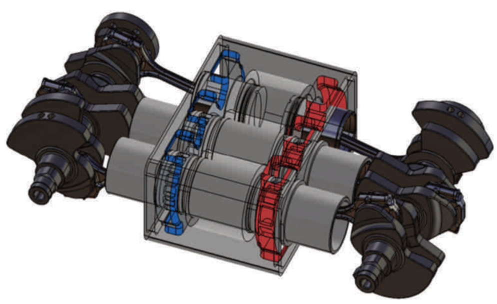


Fig. 1. CAD model of the engine

The engine with opposing pistons is a three-cylinder unit that will be used to drive lightweight gyrocopters. The engine will be characterized by a power of 100 kW, with a capacity of about 1600 cm^3, with a diesel cycle. The unit will have three cylinders with opposing pistons positions and two crankshafts facing each other. The engine will be equipped with a piston timing system and a direct diesel injection system to the inside of the cylinder.

CFD computer method was used to calculate the efficiency of a fluid jacket (Fig. 2) of the designed engine. The jacket geometrical model was created by "subtract" operation from the previously designed engine block, the "empty" part of the cooling channel was removed.

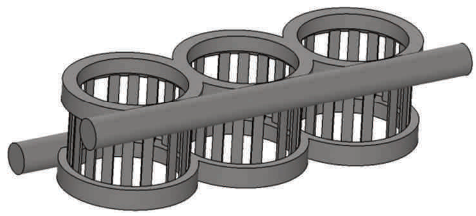


Fig. 2. CAD Model of fluid jacket

Computer fluid mechanics simulations were performed using ANSYS software in ver. 13.0.

The preparation of a simulation model using ANSYS software consisted of the following steps [7]:
- geometry preparation in CAD environment,
- geometry importing into the Geometry module,
- computational grid generation in Mesh module,
- assumption of the initial and boundary conditions,
- carrying out simulations of the heat flow,
- analysis of results.

2.1. Grid generation

The CAD model of the fluid jacket (Fig. 2) was transferred from CATIA V5 software in STP format. This is a standard format for exchanging product model data. STP files are used to store 3D image data in ASCII format, in accordance with ISO 10303-21 standards. Thus obtained geometry was imported to the Geometry module. This module allows for preparation of a model to generate a calculation grid and to modify the geometry, or perform boolean operations. In addition, the module can be coupled with CAD software, enabling live geometry updating in ANSYS after changes made in CAD model. Figure 5 shows the geometry model of the jacket in Geometry module [2].

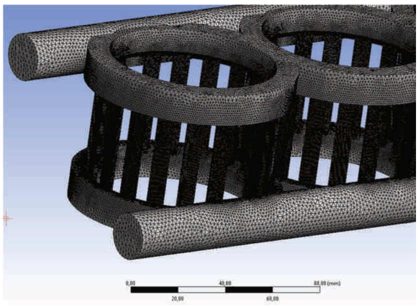


Fig. 3. Computational grid of fluid jacket model, with a minimum element size of 0.4 mm

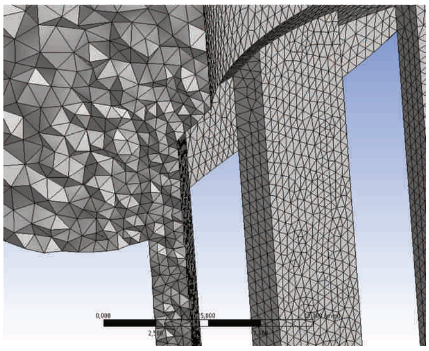


Fig. 4. Cross-section of computational grid of fluid jacket model, with a minimum element size of 0.4 mm

Such prepared geometry was imported into the Mesh module. This module allows to generate a computational grid based on the finite element method FEM. It is one of the most important steps in creating a CFD simulation. It depends on the correct definition of resolution, dimensions and number of calculation elements. This stage should be carefully analyzed as it is a parameter that has a measurable impact on the accuracy of the obtained results and the time of calculations. Optimal selection of grid resolution depends largely on the complexity of the analyzed object, its size and the thickness of the walls. Figure 3 shows the target grid of the model consisting of over 6,000,000 elements, and Fig. 4 shows the cross-section of the grid through the rib (channel along the cylinder axis).

2.2. Initial and boundary conditions

Simulation studies of the amount of heat received from the combustion chamber to the cooling system were made using ANSYS software in FLUENT module. Calculations were based on pressure-based solver. It is used for calculating the streams of incompressible and slightly compressible flows, with low flow velocity. In this approach, the obtained solution of the equation of pressure or equation of pressure correction is obtained from the equations of continuity and momentum. The model also includes the energy equations that allows the flow of heat energy and temperature recording at selected points. A k-omega (2 equ), SST viscosity model was also selected. This model (k-omega), well reflects the turbulent flow in the boundary layer, but is very sensitive to turbulent magnitudes in free flow. As a coolant fluid, water was assumed, but as the material of engine block the aluminum was selected and the as the material of cylinder a steel was chosen.

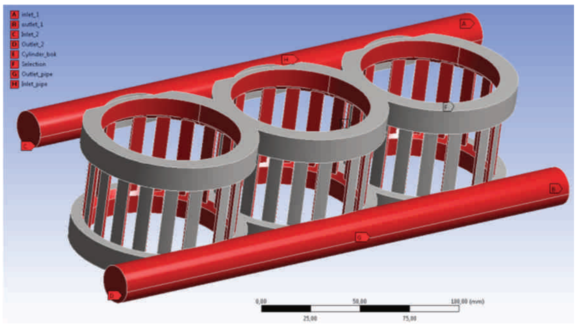


Fig. 5. The geometry model of the jacket in geometry module in Ansys Fluent

Boundary and initial conditions were taken from previously conducted simulations using AVL Boost software. The engine work was simulated for the state of operation of full load for a 4,000 rpm crankshaft rotational speed. The folowing assumptions were made:
- the cylinder wall surface:
  - wall temperature: 480 K (average cycle temperature),
  - wall thickness: 5 mm,
- engine block surface:
  - wall temperature: 323 K,
  - wall thickness: 5 mm,
- injectors placement surface („injectors” selection):
  - wall temperature: 480 K,
  - wall thickness: 5 mm,
- inlet surface („inlet” selection):
  - temperature: 363 K,
  - mass flow rate of the coolant: 0.3 kg/s,
  - turbulence intensity: 5%,
  - turbulence viscosity coefficient: 5%,
- outlet surface („outlet” selection):
  - temperature: 368 K (value assumed as the starting parameter),
  - turbulence intensity: 5%,

– turbulence viscosity coefficient: 5%,
– jackets surface outer/inner of cooling channels in block („wall" section):
– walls temperature: 323 K,
– wall thickness: 5 mm.

2.3. Calculations models

2.3.1. Influence of computational grid

In order to verify the influence of the size of the computational grid elements on the results of the simulations, a number of elementary mesh grid were developed for the geometry of the basic model. In all cases, the number of the elements was changed. Because of the flow phenomena, the most important place, with the smallest cross-sectional area, has been found, to be ribs running along the smooth surface of the cylinder.

Simulations were started with a mesh size of approximately 2 mm in the rib area, resulting in a total number of elements less than 600,000, ending in a 0.4 mm element size, resulting in a total number of approximately 6 million elements. Figure 4 shows the calculation grids with a minimum element size of 0.4 mm and figure 6 shows an intermediate mesh of 0.8 mm.

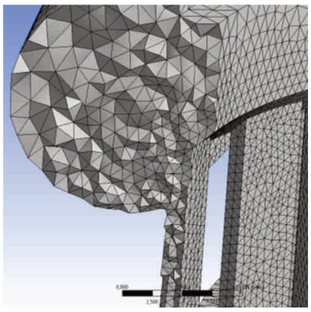


Fig. 6. Computational grid of fluid jacket model, with a minimum element size of 0.8 mm

2.3.2. Influence of turbulence

During the simulation a momentum equations and two-equation k-epsilon turbulence model were used. The k-epsilon model was adopted, with standard wall functions. Implicit coefficients such as were assumed [4]:
– C2-Epsilon = 1.9,
– TKE Prandtl Number = 1,
– TDR Prandtl Number = 1.2,
– Energy Prandtl Nubmer = 0.85.

In order to verify the effect of turbulence on the quality of simulation results, the turbulence intensity factor was changed. This ratio was in the range of 5 to 10%.

2.3.3. Geometrical calculations models

In order to optimize the flow of thermal energy during combustion of the fuel dose and transfer it to the cooling fluid, more than 20 geometric models were used for simulation tests. The different models differed in the intake and outlet diameters, the inlet and outlet manifold inclination angle, the water jacket splits individually for each cylinder, the enlargement and reduction of the fluid capacity above and below the ribs, the gradual gradation of the intake outlet channels, as to achieve a uniform flow in each of the cylinders or the assembly of the two inlet and outlet nozzles.

Among all analyzed models, the following versions can be distinguished:
– basic model used to determine the effect of calculating grid size and turbulence (Fig. 5),
– individual cylinder flow model (Fig. 7).

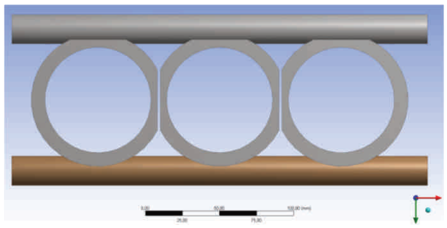


Fig. 7. Model with individual cylinder flow

– model with variable geometry of the intake and exhaust channels (Fig. 8),

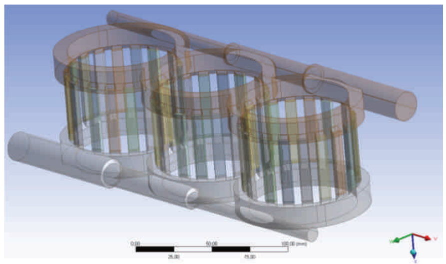


Fig. 8. Model with variable geometry of the intake and exhaust channels

– model for reducing flow resistance (Fig. 9),

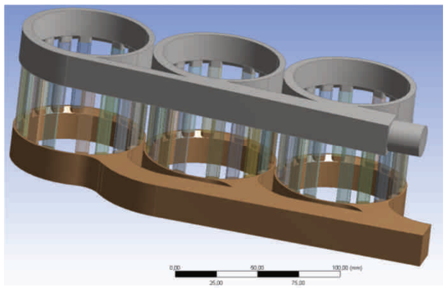


Fig. 9. Model for reducing flow resistance

– final model with the variable cross-section of the intake manifold at the cylinder inlet, double outlet channels with integrated joint outlet and inclination inlets of all ribs (Fig. 10).

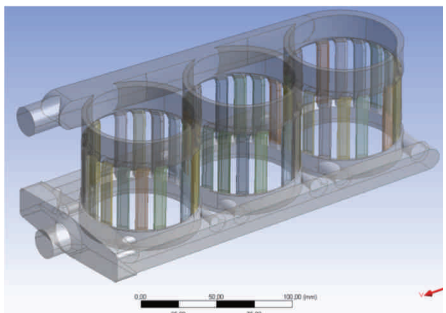


Fig. 10. Final model

2.4. Simulations results

Figures 11 to 17 show the results of the simulation studies in the form of the distribution of velocity fields of the working medium and the temperature fields on the model walls in the cylinder region.

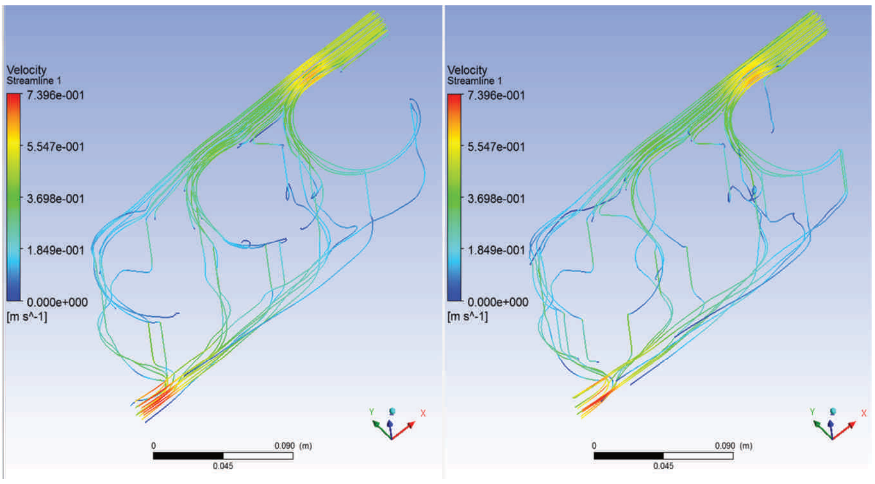
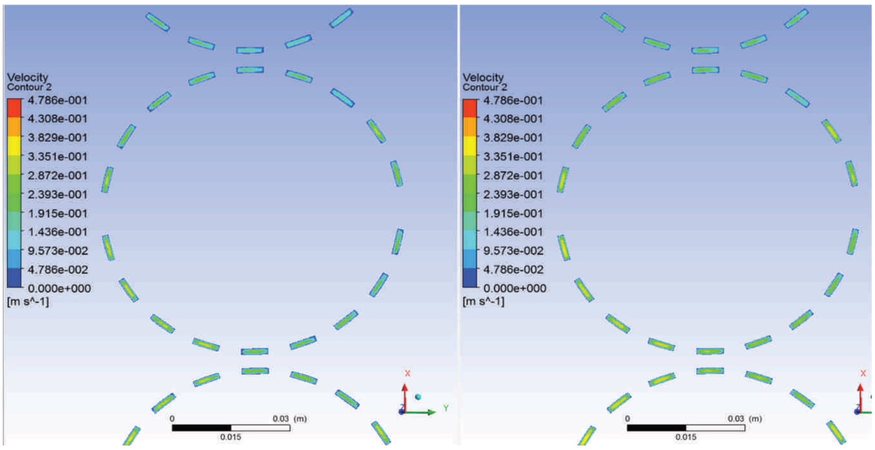
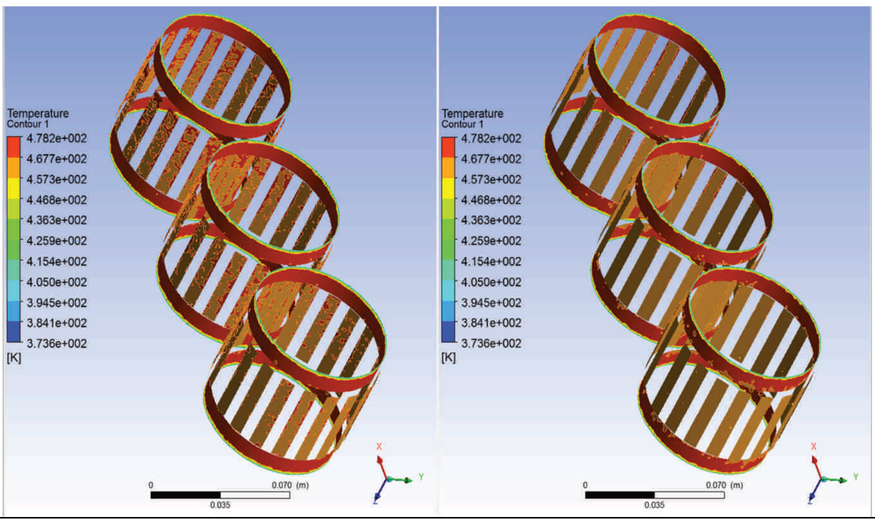
<table>
<tr><td colspan="2">Minimum element size:</td></tr>
<tr><td>0.6 mm</td><td>0.4 mm</td></tr>
<tr><td colspan="2">Distribution of the speed streamline

</td></tr>
<tr><td colspan="2">Distribution of velocity fields in cross-section through ribs (cross section in middle part)

</td></tr>
<tr><td colspan="2">Distribution of temperature fields on cylinder walls

</td></tr>
</table>

Fig. 11. Distribution of velocity and temperature fields for the computational grid with a minimum element size of 0.8 mm (left side) and 0.6 mm (right side)

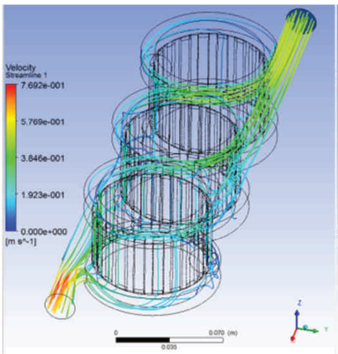
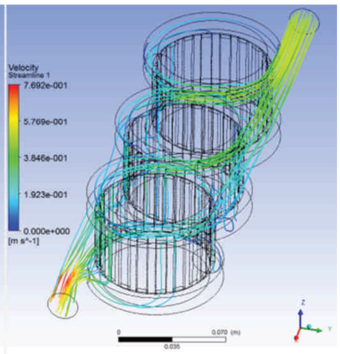
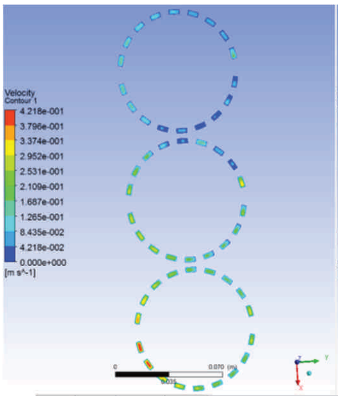
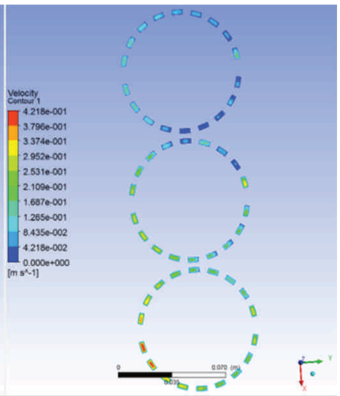
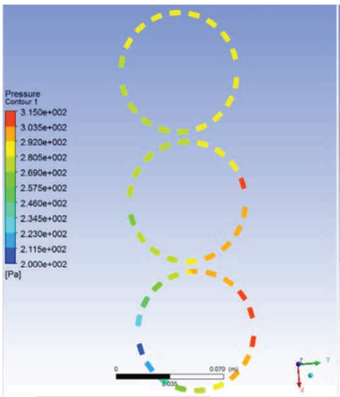
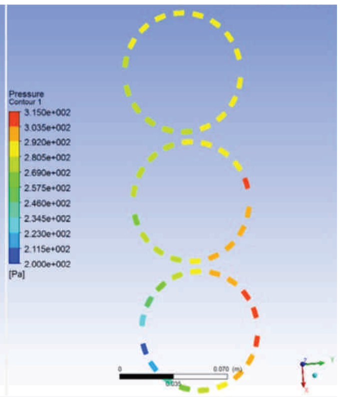
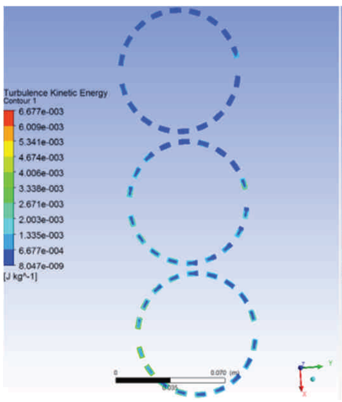
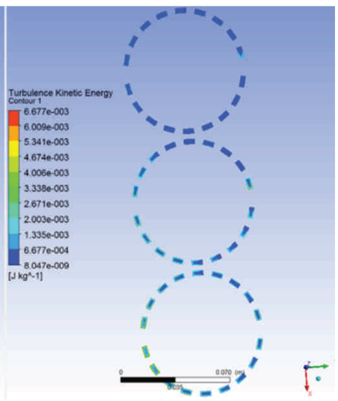
Turbulent Intensity **5%**	Turbulent Intensity **10%**
Distribution of the speed streamline	
	
Distribution of velocity fields in cross-section through ribs (cross section in middle part)	
	
Distribution of pressure fields in cross-section through ribs (cross section in middle part)	
	
Distribution of kinetic energy of turbulence	
	

Fig. 12. Visualization of the effect of turbulence intensity on the computational model

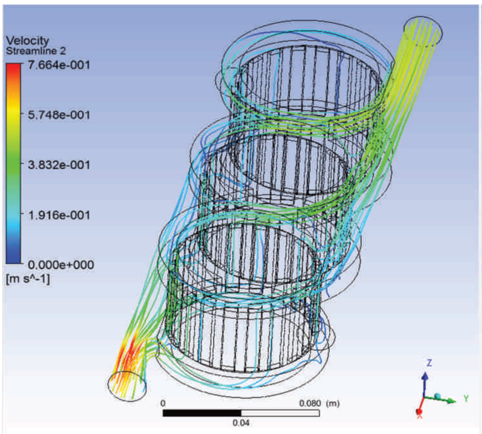
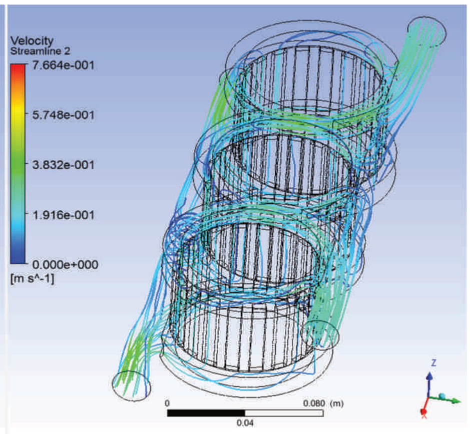
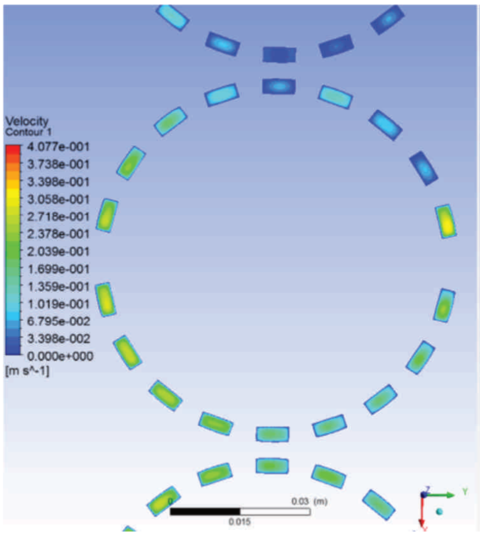
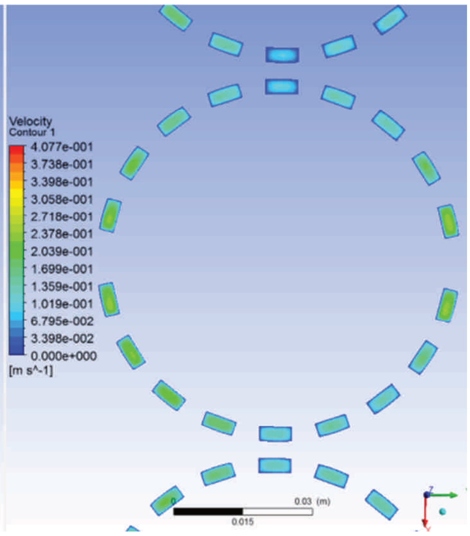
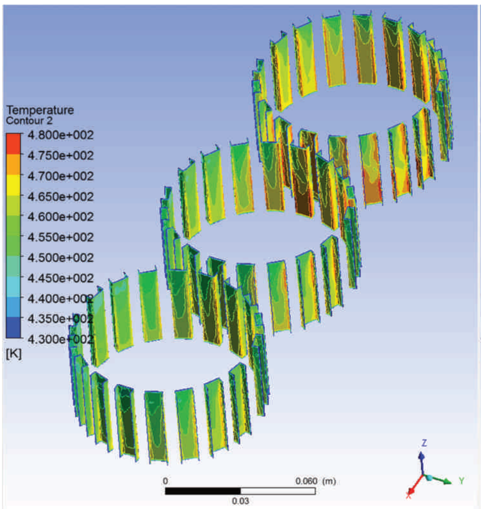
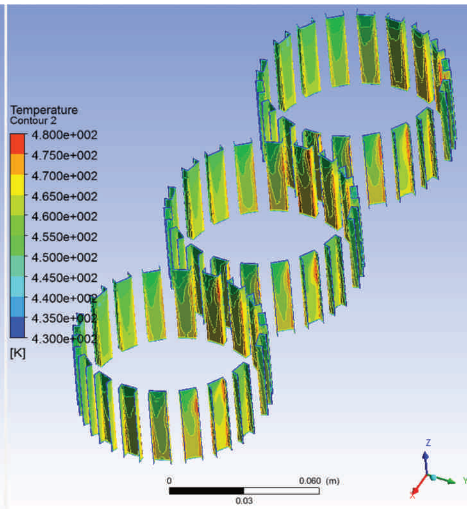
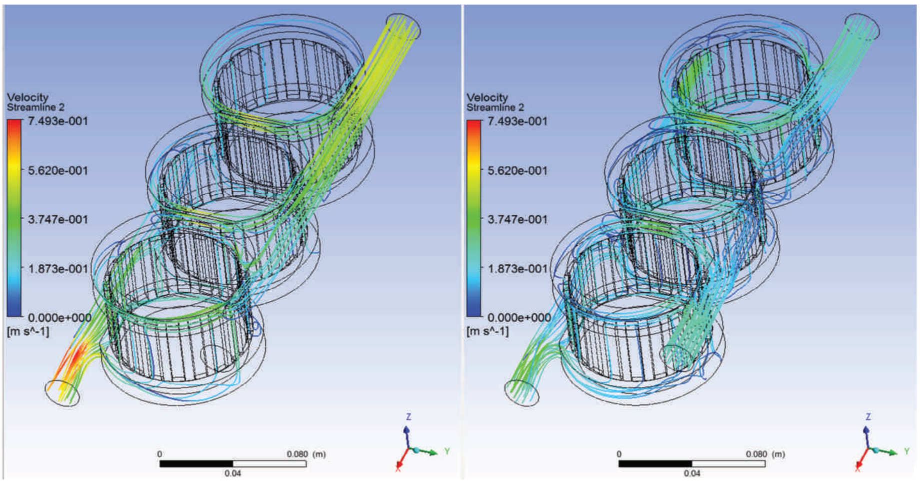
<table>
<tr><th colspan="2">Basic model</th></tr>
<tr><th>1 inlet / 1 outlet</th><th>2 inlets / 2 outlets</th></tr>
<tr><td colspan="2">Distribution of the speed streamline</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Distribution of velocity fields in cross-section through ribs (cross section in middle part)</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Distribution of temperature fields on cylinder walls</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

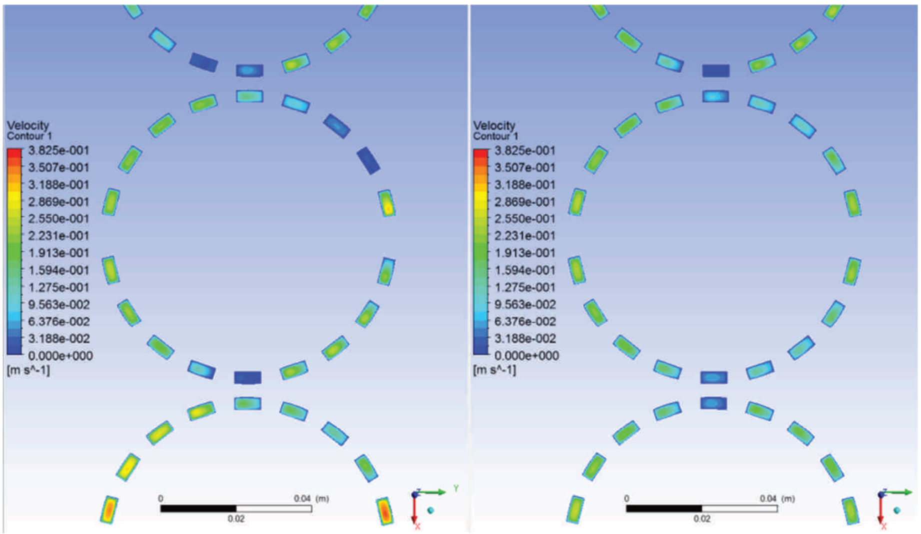
Fig. 13. Comparison of distribution velocity fields across the ribs and temperature fields for the basic model, one inlet and outlet (left side) and two inlets and outlets (right side)

Model with individualized flow in cylinders	
1 inlet / 1 outlet	**2 inlets / 2 outlets**

Distribution of the speed streamline



Distribution of velocity fields in cross-section through ribs (cross section in middle part)



Distribution of temperature fields on cylinder walls

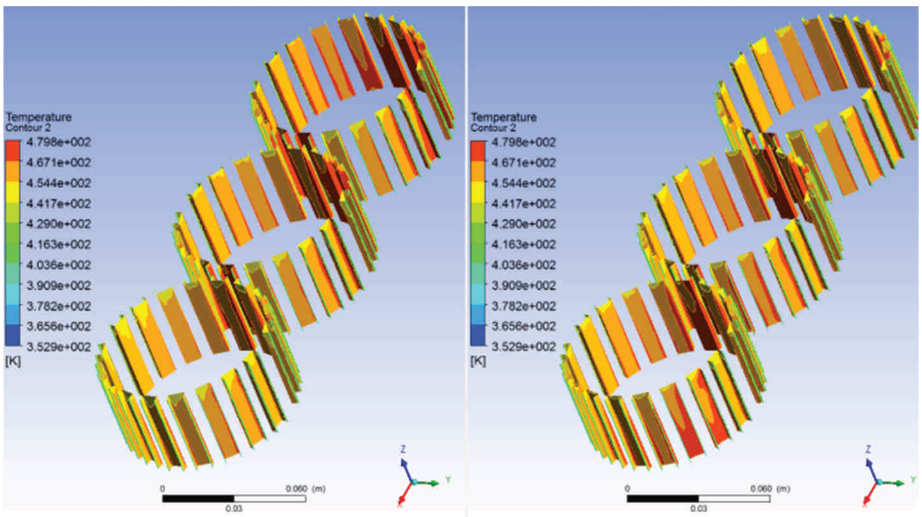


Fig. 14. Comparison of distribution of velocity fields across ribs and temperature fields for model with individualized flow in cylinders, one inlet and outlet (left side) and two inlets and outlets (right side)

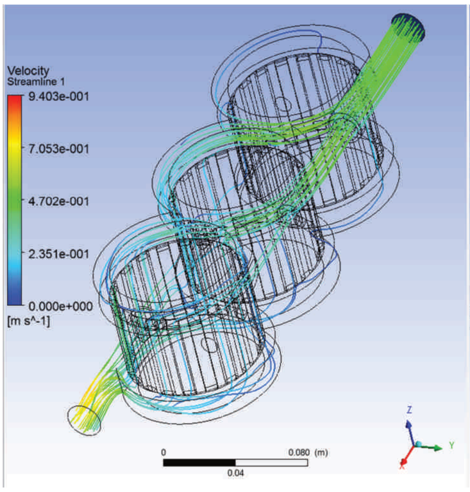
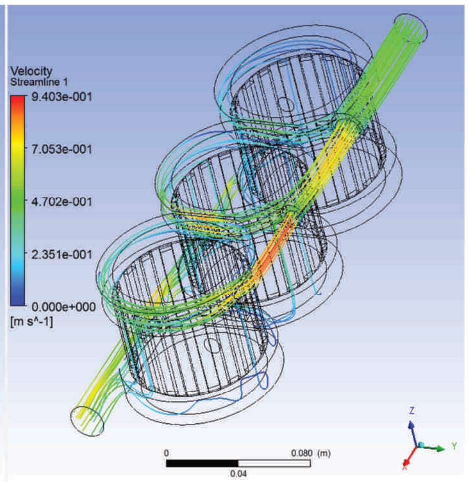
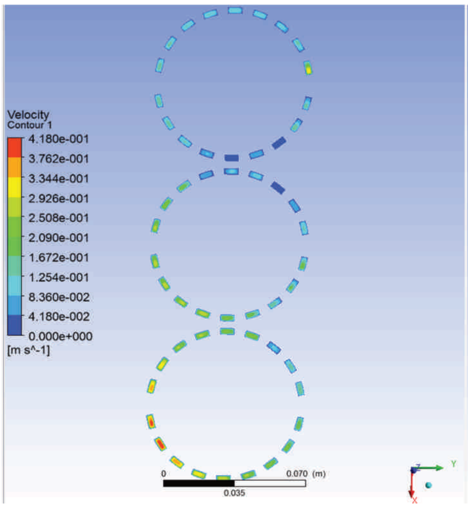
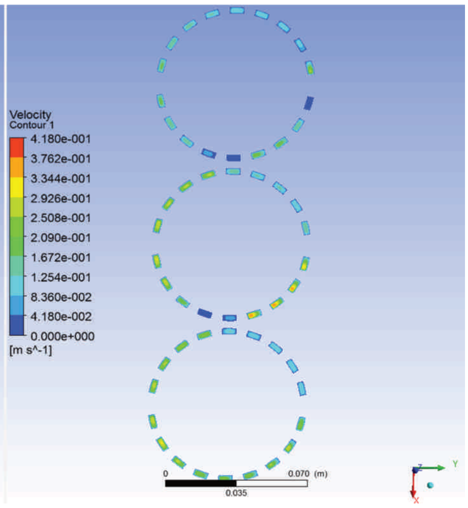
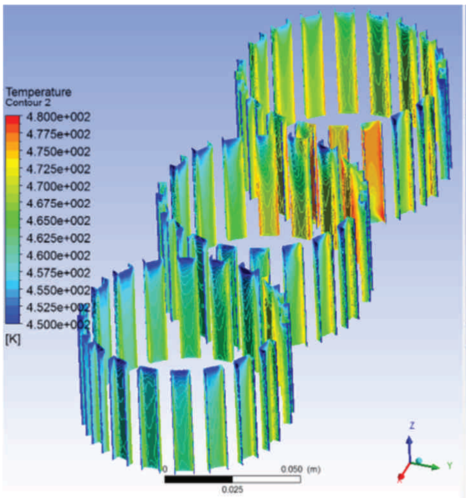
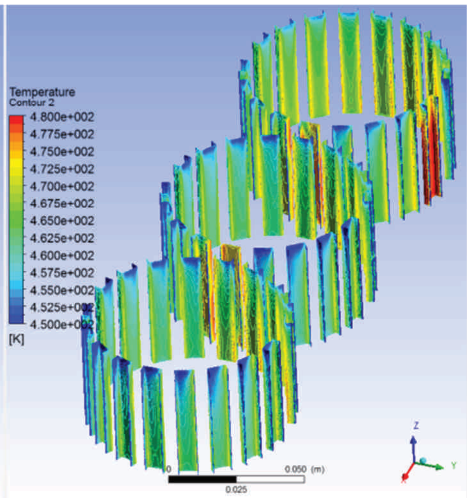
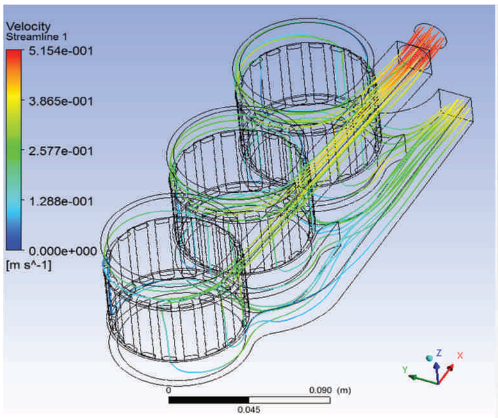
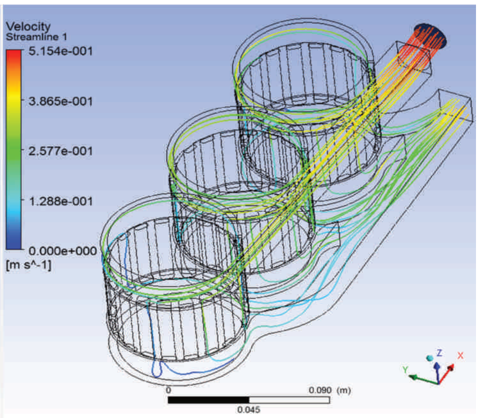
<table>
<tr><th colspan="2">Model with the variable geometry of the intake and outlet channels</th></tr>
<tr><th>Graduated collectors</th><th>Graduated collectors, split cylinders</th></tr>
<tr><td colspan="2">Distribution of the speed streamline</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Distribution of velocity fields in cross-section through ribs (cross section in middle part)</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Distribution of temperature fields on cylinder walls</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Fig. 15. Comparison of distribution of velocity field across ribs and temperature fields for model with variable geometry of the intake and outlet channels

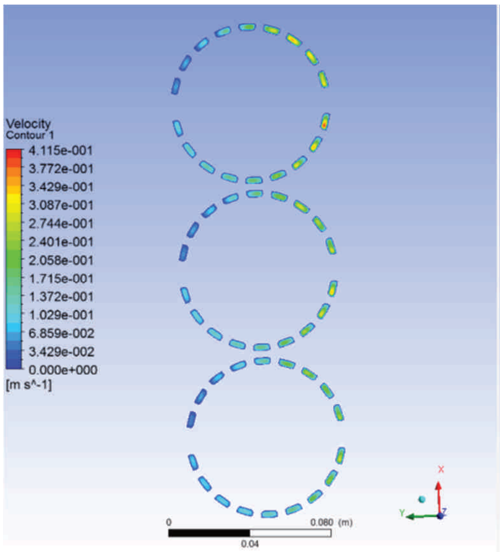
Model with reduction of flow resistance	
Without splitting the cylinders	**With cylinder splitting**

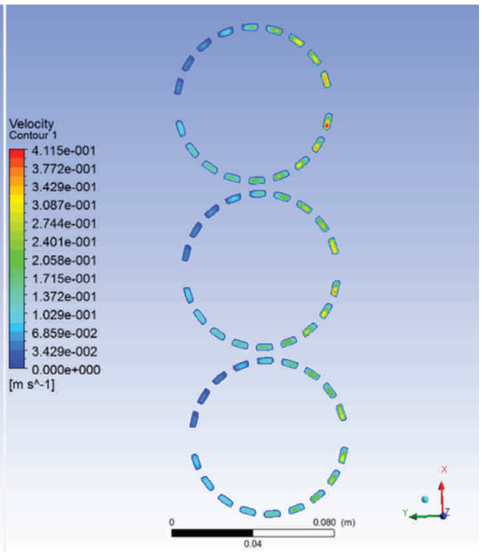
Distribution of the speed streamline



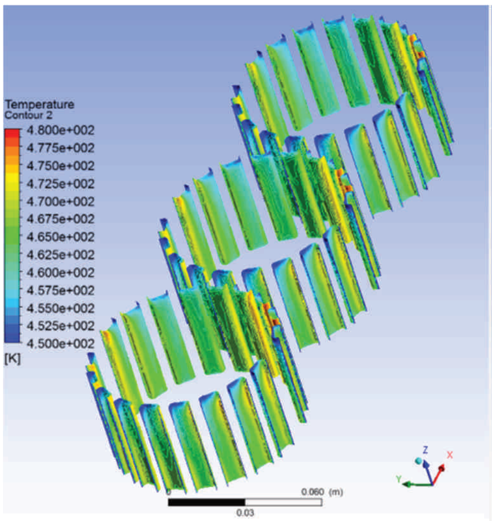


Distribution of velocity fields in cross-section through ribs (cross section in middle part)





Distribution of temperature fields on cylinder walls



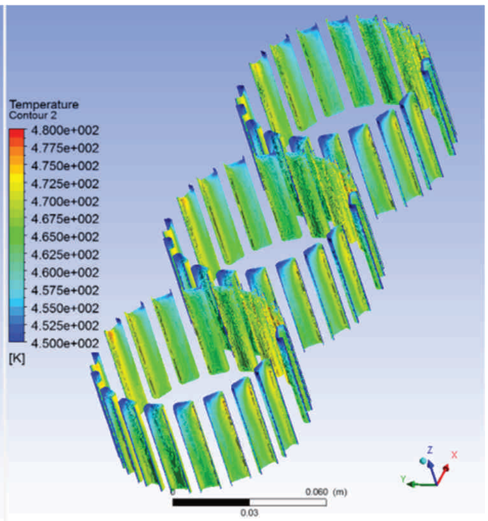


Fig. 16. Comparison of the distribution of velocity fields across the ribs and the temperature fields for the model with reduction of flow resistance

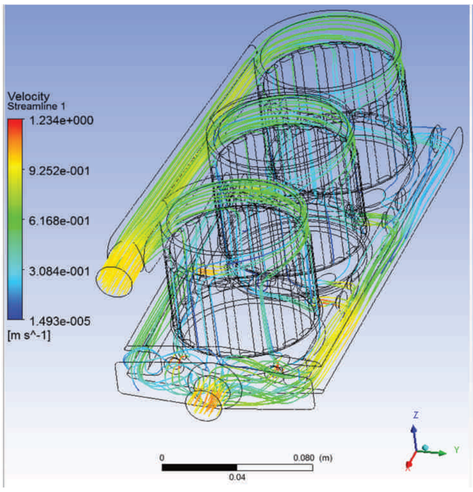
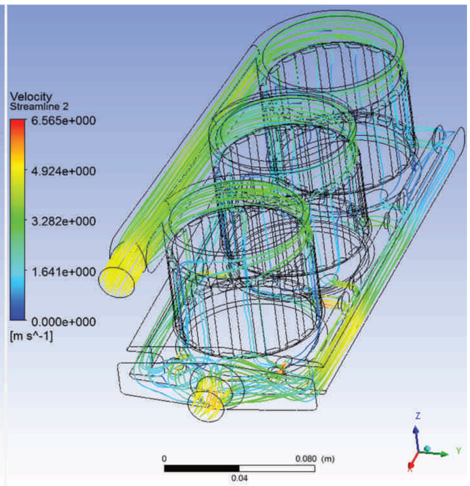
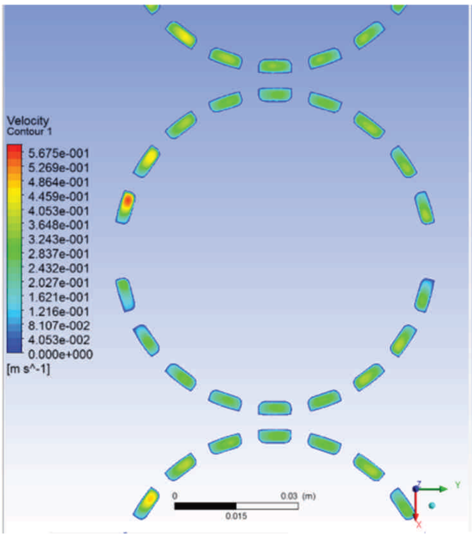
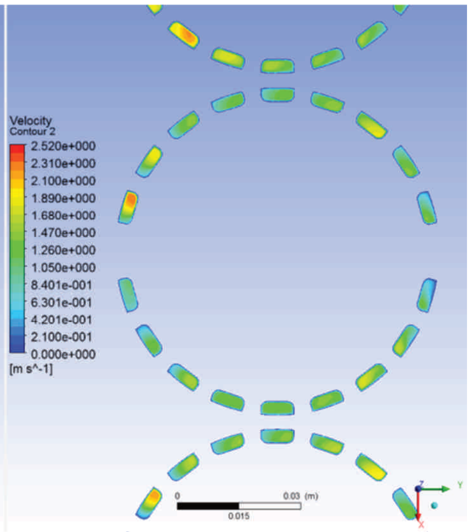
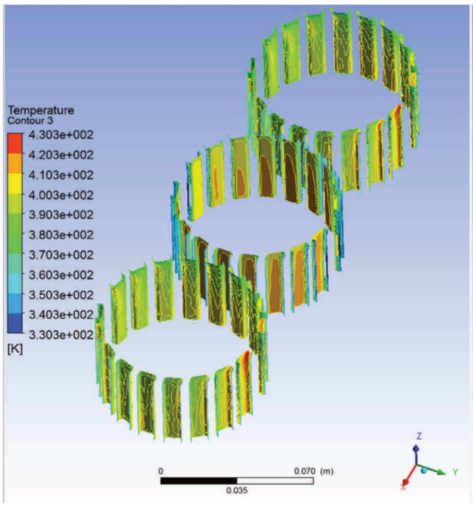
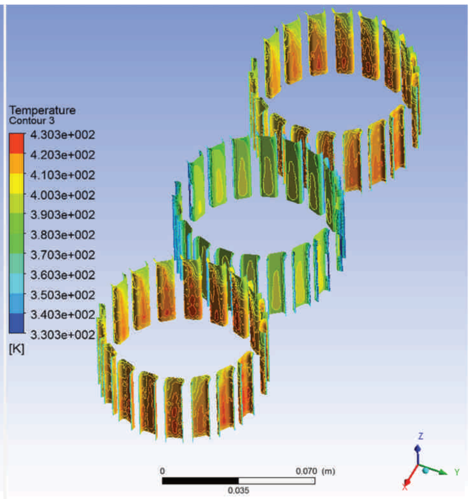
<table>
<tr><th colspan="2">Final model</th></tr>
<tr><th>m = 0.15 kg/s</th><th>m = 1.5 kg/s</th></tr>
<tr><td colspan="2">Distribution of the speed streamline</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Distribution of velocity fields in cross-section through ribs (cross section in middle part)</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Distribution of temperature fields on cylinder walls</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Fig. 17. Comparison of distribution of velocity fields across the ribs and temperature fields for the final model, for two mass flow rates

3. Summary and conclusions

During the simulations tests of the heat flow from chemical reactions that's occurs during the fuel combustion, more than twenty different channel designs were analyzed in the engine cooling system. Each model was analyzed in the range of two to four design variants, i.e. for different flow rates and different combinations of inlet and outlet of the working medium. This article presents five representative models. These models were constructed using CAD software, using CATIA V5. Subsequent versions differed in shape and size of the inlet, outlet channels, number of inlets and outlets channels, arrangement of inlet and outlet channels (on one and other side of the model), shape of channels along the cylinder axis etc.

Using ANSYS software, a computational grid was developed, numerical research was performed and results were transformed into color graphs. During the numerical tests for all models, identical initial and boundaries conditions and the size of the computational grid in the critical elements, as in the basic model, were assumed. Boundary and initial conditions were established on the basis of previously conducted research.

Based on the analysis of the results of the simulation tests, it can be stated that the distribution of the flow of the working medium is not evenly distributed in the heat transfer space from the cylinder (vertical ribs). This condition can be observed in different velocities distributions for all investigated cases. Depending on the calculation version, the most intensive flow occurs in the area of the intake channel and in the ribs between the intake and outlet channels (between the cylinders) the velocity of the medium drops almost to zero. This directly translates into uneven temperature distribution on the cylinder walls, where for the extreme case, the difference for a single cylinder reaches almost 50°C.

The unevenness of the heat transfer from the cylinder wall will result in an increase in mechanical stress between the same wall in different thermal conditions.

The smallest spread of velocity values (and thus the increase in thermal stability - the reception of heat energy from the walls) is observed in the channels located along the cylinder axis for the last, final case. Therefore, this version seems to be the most optimum version. The smallest spread of the velocity fields between cylinders reflects in the smallest temperature difference on the cylinder walls. This state will directly result in an even mechanical stress distribution in the cylinder block in the area of the individual cylinders.

Acknowledgement

This work has been realized in the cooperation with The Construction Office of WSK "PZL-KALISZ" S.A." and is part of Grant Agreement No. POIR.01.02.00-00-0002/15 financed by the Polish National Centre for Research and Development.

Nomenclature

CAD computer aided design
CFD computational fluid dynamics
FEM Finite Elements Method

Bibliography

[1] CHOUGULE, A.B., SURESH, R. Pusher configured turboprop engine oil cooler ejector performance: CFD analysis and validation. *Proceedings of the 6th International and 43rd National Conference on Fluid Mechanics and Fluid Power*. December 15-17, 2016, MNNITA, Allahabad, U.P., FMFP2016–PAPER NO. 35.

[2] CZYŻ, Z., PIETRYKOWSKI, K. CFD model of the CNG direct injection engine. *Advances In Science And Technology Research Journal*. 2014, **23**(8), 45-52.

[4] CZYŻ, Z., ŁUSIAK, T., MAGRYTA, P. Badania numeryczne CFD wpływu usterzenia na charakterystyki aerodynamiczne. *Transactions of The Institute of Aviation – Prace Instytutu Lotnictwa*. 2013, **232**, 3-14.

[4] FOGLA, N., BYBEE, M., MIRZAEIAN, M. et al. Development of a K-k-E phenomenological model to predict in-cylinder turbulence. *SAE International Journal of Engines*. 2017, 3.

[5] GRABOWSKI, Ł., CZYŻ, Z., KRUSZCZYŃSKI, K. Model numeryczny układu chłodzenia zespołu napędowego wiatrakowca. *Logistyka*. 2014, **6**, 4169-4178.

[6] GRABOWSKI, Ł., CZYŻ, Z., KRUSZCZYŃSKI, K. Numerical analysis of cooling effects of a cylinders in aircraft SI engine. *SAE 2014 International Powertrain, Fuels & Lubricants Meeting*. 20-23, October 2014, Birmingham.

[7] NURSAL, R.S., HASHIM, A.H., NORDIN, N.I. et al. CFD analysis on the effects of exhaust backpressure generated by four-stroke marine diesel generator after modification of silencer and exhaust flow design. *ARPN Journal of Engineering and Applied Sciences*. 2017, **12**(4).

[8] ROMANOV, V. A., KHOZENIUK, N.A. Experience of the diesel engine cooling system simulation. International Conference on Industrial Engineering, ICIE 2016, *Procedia Engineering*. 2016, **150**, 490-496.

Michał Biały, MEng. – Faculty of Mechanical Engineering at the Lublin University of Technology.

e-mail: *M.Bialy@pollub.pl*

Tutus Tulwin, MEng. – Faculty of Mechanical Engineering at the Lublin University of Technology.

e-mail: *T.Tulwin@pollub.pl*

Konrad Pietrykowski, DEng. – Faculty of Mechanical Engineering at the Lublin University of Technology.

e-mail: *K.Pietrykowski@pollub.pl*

Paweł Magryta, MEng. – Faculty of Mechanical Engineering at the Lublin University of Technology.

e-mail: *P.Magryta@pollub.pl*