# Effect of Transverse Flow in Porous Medium on Heat Exchanger Simulation

<table>
<thead>
<tr>
<th>Journal:</th>
<th>Transactions of the Canadian Society for Mechanical Engineering</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manuscript ID</td>
<td>TCSME-2019-0158.R1</td>
</tr>
<tr>
<td>Manuscript Type:</td>
<td>Article</td>
</tr>
<tr>
<td>Date Submitted by the Author:</td>
<td>31-Jul-2019</td>
</tr>
</tbody>
</table>
| Complete List of Authors: | Yu, Chao; Jilin University School of Mechanical and Aerospace Engineering  
Qin, Sicheng; Jilin University School of Mechanical and Aerospace Engineering  
Liu, Yang; Changchun Institute of Optics Fine Mechanics and Physics Chinese Academy of Sciences |
| Keywords: | heat exchanger, water cooled charge air cooler(WCAC), porous medium, fin element, computational fluid dynamics(CFD) |
| Is the invited manuscript for consideration in a Special Issue?: | Not applicable (regular submission) |
Effect of Transverse Flow in Porous Medium on Heat Exchanger

Simulation

Liu Yang¹, Yu Chao¹*, Qin Sicheng²


2. School of Mechanical and Aerospace Engineering, Jilin University, Changchun 130022, China.

Corresponding author: Yu Chao, 8613578712359, Changchun Institute of Optics, Fine Mechanics and Physics, Chinese Academy of Sciences, Changchun 130033, China.

Email: yuchao17@mails.jlu.edu.cn
Abstract: In order to improve the accuracy of heat exchanger CFD simulation, the transverse resistance coefficient of the cold side and the hot side of the water cooled charge air cooler is calculated by using the fin element method, and the influence of several common factors on the resistance coefficient of the fin element is analyzed. The transverse resistance coefficient obtained from the simulation of the fin element are substituted into the WCAC model and compared with the experimental data. It is found that the fin element method can simulate the flow field of the WCAC accurately, and the simulation results are closer to the experimental curve compared with the empirical method. It provides guidance for optimal design of heat exchanger and is helpful to shorten development time and save cost.

Keywords: heat exchanger; water cooled charge air cooler(WCAC); porous medium; fin element; computational fluid dynamics(CFD)
Introduction

As a typical heat exchanger, the intercooler is used to cool the high temperature and high pressure air from the supercharger, which can greatly reduce pollutant emission and improve the dynamic performance of the diesel engine [1]. At present, most researchers use fluent and other multidimensional simulation software to design and calculate heat transfer and pressure loss of heat exchangers [2-3]. Kumar NS [4] used ANSYS software to analyze the heat dissipation performance of fins of a fin tube intercooler. When aluminum was used, the surface heat transfer coefficient was 19.73% higher than that of copper. When changing from copper to bronze, the surface heat transfer coefficient decreases by 0.53%. Dong et al. [5] analyzed the flow field and pressure drop of gas turbine intercooler with the help of CFD technology. Uysal et al. [6] conducted a numerical study on the momentum and thermal characteristics of the connecting hose of a fiat engine intercooler. Cuevas et al. [7] carried out relevant experimental tests on cars equipped with low-pressure exhaust recirculation to determine the impact of triangular straight fins and shutter fins on the thermal and hydraulic performance of intercoolers. Hribernik Ale [8] established the NTU efficacy method and the 2D model method to calculate and analyze the properties of air cold-plate-fin cross-flow heat exchangers, and compared the advantages and disadvantages of the two methods.

However, it is difficult to simulate the whole structure of heat exchanger because of the complex structure and large number of fins. The huge data file makes the ordinary microcomputer computation efficiency low or cannot run the simulation at all. In order to reduce the simulation requirements and realize the rapid and effective flow field analysis of heat exchangers, most of the current simulation calculations need the help of porous media model. The simulation method adopted by most researchers is to downplay the influence of flows in the second and third directions.
and focus on the main flow direction of incoming flows in porous media. Guo L, et al. directly set the viscosity and inertia resistance coefficients in the second and third directions at two or three orders of magnitude in the main flow direction to simulate the resistance of fluid media in the two directions, roughly estimate the resistance coefficients in the second and third directions. And mainly considering the influencing factors of the main flow direction. However, according to this semi-empirical method, the magnitude of the resistance coefficient has a large span and does not indicate which simplified magnitude of fluid medium should be adopted. Therefore, many uncertain effects will be produced in the actual heat exchanger simulation calculation. Usually the fin height of heat exchanger is less than 1% of the flow direction length, and ignore the fluid medium in the direction of the fin height direction of the third tap will not cause great influence to the whole simulation. But compared with the third direction heat exchanger, there is usually a larger transverse size, and the accuracy of the simulation of the transverse shunt in the flow passage will directly affect the simulation accuracy of the heat exchanger.

The water-cooled intercooler contains both gas and liquid fluid media, and its structure and principle are relatively complex, so this paper chooses it as the carrier for simulation research. The viscous and inertial drag coefficients of rectangular cross-toothed fins were analyzed by building the main flow direction and transverse flow direction unit models of the hot and cold side of a water-cooled cooler. The resistance coefficients obtained by the solution method and the two groups of empirical resistance coefficients were respectively put into the overall model of water-cooled intercooler for Fluent simulation to calculate its pressure parameters. Finally, the calculation results are compared with the test data of intercooler, and the influence of transverse flow on heat exchanger design is studied.
1 Establishment of simulation model

1.1 Porous media model

A momentum source term is added to the momentum equation to simulate the action of porous media. Where $S_j$ is the source term of the momentum equation; $\mathbf{v}_j$ is the velocity vector; $D_{ij}$ and $C_{ij}$ are the matrix elements of viscous resistance coefficient and inertial resistance coefficient respectively. For simple homogeneous porous media, the momentum source term can be simplified into equation (2)\(^{[13]}\), Where $\frac{1}{\alpha}$ is the coefficient of viscous resistance and $C_2$ is the coefficient of inertia resistance.

\[
S_j = -\left(\sum_{j=1}^{3} D_{ij} \mu \mathbf{v}_j + \sum_{j=1}^{3} C_{ij} \rho_\text{mag} \mathbf{v}_j \right) \quad (1)
\]

\[
S_i = -\left(\frac{\mu}{\alpha} \mathbf{v}_i + C_2\frac{1}{2} \rho_\text{mag} \mathbf{v}_i \right) \quad (2)
\]

According to the semi-empirical simplified solution method provided by some researchers and the software Fluent, the coefficient of viscous resistance and inertial resistance in the second direction (Direction-2 in Fig.1, hereinafter referred to as D2) and the third direction (Direction-3 in Fig.3, hereinafter referred to as D3) can be amplified to 100 or 1000 times of that in the first direction for calculation.

1.2 Element model and boundary conditions

Water-cooled intercooler, a typical plate-fin heat exchanger, is composed of air chamber, water chamber, hot side cooling zone, cold side cooling plate and seal. The hot side has 10 layers of scattered tropics, while the cold side has 9 layers of hot plates. The air in hot side flows horizontally, while the coolant flows in a secondary U-shape in cold side. Theoretically, when solving the parameters of the porous medium of the heat exchanger, it is necessary to simulate all the flow paths on the hot and cold side of the heat exchanger, which requires an extremely large amount of
calculation, but the current development level of computer hardware is difficult to accomplish such a task. In this paper, the fin element method used to solve the drag coefficient simplifies the solution of a heat sink layer to a fin unit by setting the period and symmetrical boundary.

Fig. 1  Structure of the Water-cooled charge air cooler

Fig. 2  Transverse flow fin element of hot side

Taking the hot side of the intercooler as an example, the length of the main flow direction element is equal to 300mm of fin length. In order to study the influence of transverse shunt on the calculation, it is necessary to establish the transverse flow element of the hot side (Fig.2), whose passage length is 94mm of the fin width. The bulkhead is 0.6mm thick, the fin thickness is 0.2mm, the flow direction pitch on the hot side is 2mm, and the cold side is 3mm. The length of the inlet and outlet extension zone is 5 and 7 times of the hydraulic diameter of the fin, respectively. The geometric parameters of fins on the hot side and cold side are shown in Fig.3.

Fig. 3  Geometric parameters of the fin

As the overall model size of intercooler is large and its requirements for mesh quality are not high, the maximum mesh size in Gambit is generally less than 0.8, in order to meet the simulation requirements. The precision of viscous resistance coefficient and inertial resistance coefficient will determine the accuracy of the porous media model, which is greatly affected by the mesh quality of the finned element model, so the mesh quality of the element model should be improved as much as possible[14]. The hot side unit body model meshed by Gambit is established at 1/5 of the flow
path length of the hot side main flow direction (Direction-1 in Fig. 1 hereinafter referred to as D1).

Five boundary layers with a starting thickness of 0.01 mm and an increment of 1.01 are established on the surface of the fin. The unit body model is divided into grids with grid meshing size 0.1. Finally, the number of final grids in the hot side D1 direction unit is 3147600. The grid density is of 807 cells/mm³. Since the structure of each part of the three kinds of unit bodies on the hot side D2 and the cold side D1 and D2 is similar to that of the hot side main flow unit, only the number of grids is different, so it will not be described. the physical property parameter Settings are as follows:

The hot pressurized air side is 120 °C; the dynamic viscosity is $22.84 \times 10^{-6} \text{ kg/m} \cdot \text{s}$; the density is 2.656 kg/m³; the coefficient of thermal conductivity is 0.0325 w/m · k. The cold side is Glycol cooling fluid; the dynamic viscosity is $7.14 \times 10^{-4} \text{ kg/m} \cdot \text{s}$; the density is 1013 kg/m³; coefficient of thermal conductivity is 0.4043 w/m · k[15-16]. In this research, the $k$-$\varepsilon$ turbulence model is used for simulation by Fluent, the transport equation of the standard $k$-$\varepsilon$ model is $^{[16]}$:

$$
\left\{ \begin{array}{l}
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon + S_k \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{2\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon
\end{array} \right. \tag{3}
$$

The following values are used for standard $k$-$\varepsilon$ model:

$C_{1\varepsilon}=1.44$, $C_{2\varepsilon}=1.92$, $C_{3\varepsilon}=0.09$, $\sigma_k=1.0$, $\sigma_\varepsilon=1.3$.

Fig. 4  Mesh generation of hot side 1/5 fin element in main flow

2 Analysis of drag coefficient of fin element

2.1 Influence of flow length on drag coefficient

Taking the hot side D1 direction unit as an example, the influence of the period boundary and the symmetrical boundary on the calculation is studied. The actual flow length in the D1 direction
of the hot side dispersing zone of the intercooler was 300mm. In order to verify whether reducing the runner length will have an impact on the calculation, the 1/5 runner length cell model described in section 1.2 and the full-size cell model of 300mm are respectively simulated by Fluent. The standard for determining the convergence is set as that each residual is less than $1 \times 10^{-7}$ to ensure the calculation accuracy of the resistance coefficient, and there is no or minimal pressure fluctuation in the pressure inlet and outlet sections\cite{17}. According to equation (2), MATLAB is used to fit the calculated pressure drop and velocity data, as shown in Fig.5. The fitting results are shown in equations (4) and (5), and the $v$ is velocity in equation. The goodness of fit of the two curves is 1. The fitted velocity pressure drop curve must cross the origin, that is the closer the conic constant term is to the origin the higher, the simulation accuracy\cite{18}. According to equation (2), the viscous resistance coefficient and inertial resistance coefficient are calculated as shown in Tab.1. It can be seen that the resistance coefficient of 1/5 flow channel length element is basically the same as that of full-size element when the mesh mass reaches $1 \times 10^{-10}$. In this paper, through multiple simulation calculations, it is found that the difference between the resistance coefficient and the full-size model gradually increases when the flow passage length of the hot-side element decreases to 50mm. It is considered that the simulation difference between the full-size model A and the simulation difference $\Delta$ scaling model is affected by the scaling factor C and the grid quality B (as shown in equation 6). The simulation value of scaling model is the function of mesh quality B and scaling factor C, which can theoretically shorten the flow path length of the cell body indefinitely when the mesh quality is infinitely high. However, because the mesh quality is difficult to control due to the different complexity of the model, the size of the model needs to be adjusted according to the actual mesh quality to improve the solving efficiency.
Fig. 5  Fitting results of the fin element

The full-size model: \( \Delta P = 8.36v^2 + 155.9v + 6.73 \)  \hspace{1cm} (4)

The 1/5 model: \( \Delta P = 1.713v^2 + 3.12v + 1.566 \) \hspace{1cm} (5)

\[ |A - \delta(B, C)| = \Delta \] \hspace{1cm} (6)

Tab. 1  Resistance coefficient of different flow passage length

2.2 Influence of boundary conditions on drag coefficient

The influence of periodic boundary and symmetric boundary on the calculation was studied by taking the hot side D1 element as an example. When boundary conditions are set in fluent, periodic boundary and symmetric boundary are set respectively on the side of the element body, as shown in Fig.6. However, the premise of using periodic boundary is that the shape, number of nodes and mesh generation of the two surfaces must be completely consistent, which is greatly restricted by the model structure, while the symmetric boundary does not need to be considered. The resistance coefficient and inertial resistance coefficient of symmetric boundary model as shown in Tab.2.

Tab. 2  Resistance coefficient of symmetric boundary

The periodic boundary of the finned part in the 1/5 element body model in Fig.3 was changed to a symmetric boundary for simulation and the calculation results were fitted as shown in equation
(7). There is no significant difference between the viscosity resistance coefficient and inertia resistance coefficient of Tab.1 and Tab.2. This is because the rectangular staggered fin is a regular structure, and its porosity will not change regardless of the use of periodic or symmetric boundary. Although the model cross section with symmetric boundary is consistent with the model with periodic boundary, the distribution of pressure field between them is almost approximate.

2.3 Boundary layer analysis of element model

The turbulence model of high Reynolds number is a standard model for fully developed turbulence [19]. It is suitable for solving flows in turbulent core region, while the solution of the near wall part needs the help of wall function. The boundary layer density required by different wall functions is also different. In theory, the boundary layer can be replaced by continuous densification of the grid near the wall surface, but this will increase the number of grids in the whole model and reduce the solution economy, so the boundary layer thickness needs to be calculated [20]. In this paper, the maximum Reynolds number on the cold side and the hot side is 2200 and 5000 respectively, which means the boundary layer effect on the hot side needs to be considered. In this paper, the cell model used the enhanced wall function to calculate the boundary layer thickness, when the $y^+$ is 1. The thickness of the starting layer is 0.01mm, and there are 5 boundary layers in total.

$$y^+ = \frac{\rho H y_p}{\mu}$$  \hspace{1cm} (8)

$$\mu_e = \sqrt{\frac{\tau_w}{\rho_w}}$$  \hspace{1cm} (9)

The comparison of static pressure cloud diagram at the center section of the unit body is shown in Fig.7. It can be seen that the static pressure cloud diagram of the unit model with boundary layer is clearly arranged on the side of the near wall. However, the static pressure cloud diagram without
boundary layer is quite different from the real flow field inside the intercooler. That is because the near wall part and the main flow field are solved according to the non-viscous flow. After calculation, the model resistance coefficient of the element without boundary layer is shown in Tab.3, which is significantly different from the data in Tab.1.

Fig. 7  Influence of boundary layer on the pressure field

Since the total thickness of the five-layer boundary layer which is half of the mesh size of the main channel area is about 0.05 mm, omitting the boundary layer is equivalent to completely ignoring the wall friction. However, the fin in the flow field is a typical longitudinal flow around the plate, and the incoming flow on the surface of the fin will be subject to non-negligible friction resistance. If the influence is ignored, the calculation result of the resistance coefficient will be greatly deviated.

Tab. 3  Resistance coefficient without boundary layer

2.4 Comparison of drag coefficients in different directions

The heat side and cold side D2 direction element model are respectively established and their respective viscous resistance coefficients and inertial resistance coefficients are simulated according to the above methods, as shown in Tab.4. The fitting results are shown in equations (10) ~ (12). According to the results obtained by the element solution method, the inertial resistance coefficient in the D2 direction of the hot side is 7.3 times of that in the D1 direction, and the viscous resistance coefficient is 1.3 times of that in the main flow direction. The inertial resistance in D2 direction on the cold side is 18.7 times of that in D1 direction, and the viscous resistance is 3.1 times of that in
the main flow direction.

The hot side D2: \[ \Delta P = 19.86v^2 + 6.6v + 1.5716 \] (10)

The cold side D1: \[ \Delta P = 2868.8v^2 + 824.13v + 1.9082 \] (11)

The cold side D2: \[ \Delta P = 30086v^2 + 1429.6v + 2.1855 \] (12)

Tab. 4 Resistance coefficient of hot side and cold side

3 Experiment contrast analysis

In order to verify the influence of transverse flow on the simulation calculation of intercooler, the three resistance coefficient combinations in Tab.5 are used for the overall simulation analysis of intercooler respectively. In the direction of D2 of the first set of data, the viscous resistance coefficient and inertial resistance coefficient obtained by the unit body fitting in chapter 2 are taken. The second group is 100 times of direct amplification in D1 direction. The third group is 1000 times larger in the D1 direction. The data in Tab.4 is used for the resistance coefficient of the hot side D1 direction and the cold side D1 direction, without considering the influence of fluid flow in the D3 direction. The experiment is constructed as shown in Fig.8. The experimental results of water-cooled intercooler are compared with the overall simulation results of intercooler with several combinations of different drag coefficients as shown in Fig.11 and Fig.12.

Fig. 8 Experimental acquisition process

Tab. 5 Lateral resistance coefficient combination in D2

The viscous resistance coefficient and inertial resistance coefficient of the above Tab.5 were
input into the porous zone model to simulate the whole model of the intercooler at different inlet speeds., and the porosity of the hot side and the porosity of the cold side of the water-cooled intercooler were 0.889 and 0.859, respectively. The static pressure cloud diagram of the intercooler when the coolant inlet velocity is 1.09 m/s and the pressurized air velocity is 4.5 m/s is shown in Fig.9.

Fig. 9  The contours of static pressure

The results of the bench test of water-cooled intercooler were compared with the overall simulation results of several intercooler combinations with different drag coefficients in Tab.5. The comparison of cooling fluid lateral pressure is shown in Fig.10. The D2 lateral resistance coefficient curve obtained by using transverse element body is the closest to the test curve. The second and third groups of data fitting curves using the empirical method have a large deviation from the test value in the velocity interval. The critical value of the resistance coefficient is within 100 times of the main flow direction. After reaching the critical value, the lateral flow is basically in a suppressed state, so the curves of the second and third groups in Fig. 10 are substantially coincident. The average deviation between simulation value and test value of the three combinations is 7.88%, 13.15% and 13.95% respectively.

Fig. 10  The pressure loss comparison in cold side

Fig. 11  The pressure loss comparison in hot side

Compared with the cold side, the hot side is greatly affected by transverse flow. The average
deviation of simulation value and test value of the three groups on the hot side is 8.17%, 12.81% and 14.27%, respectively. Combined with Fig. 10 and 11, the lateral flow on the cold side is not obvious, while a considerable part of the flow on the hot side turns into lateral flow, which has a great influence on the calculation. As the pressure drop of heat exchanger is mainly the pressure loss along the way, it is considered that it is mainly affected by the following three points: The first is the inlet velocity of the fluid medium. The maximum velocity of hot side pressurized air is 10.8 times that of coolant. Second, ethylene glycol dynamic viscosity is far higher than that of pressurized air. Third, the characteristic length of the fin itself is different. This leads to the fact that when the fluid medium is a gas, it is easy to flow to the lateral shunting, while the liquid medium is not easy to shunting. That is, only a small lateral resistance coefficient is required for simulation of liquid medium, while a large lateral resistance coefficient is required for simulation of gas medium. It can be seen that the empirical method directly expands by two or three orders of magnitude according to the first direction to estimate the transverse resistance coefficient, which is more suitable for simulating the liquid-liquid heat exchanger. Although the method of solving the element body to obtain the lateral resistance coefficient improves the accuracy of simulation, it should not be ignored that it requires a lot of extra work to solve the lateral resistance coefficient alone. However, by defining the lateral resistance coefficient in an empirical way, the work efficiency can be improved. According to specific engineering problems, it is necessary to decide whether to pursue high-precision quantitative analysis or simple qualitative analysis.

4 Conclusion

In this research, the viscous and inertial resistance coefficients of heat exchangers are solved by finned element method. The results show that the transverse flow has a certain influence on the
simulation calculation of heat exchanger, which is closely related to the physical parameters of fluid medium and the geometric ruler of fin. Both the empirical method and the element method can accurately describe the flow field inside the heat exchanger, but the latter has higher accuracy and the simulation results are closer to the experimental values. The empirical method is more suitable for simple analysis with low accuracy requirement, while the element solution method is suitable for quantitative analysis with certain accuracy requirement.

Reference


Tab. 1  Resistance coefficient of different flow passage length

Tab. 2  Resistance coefficient of symmetric boundary

Tab. 3  Resistance coefficient without boundary layer

Tab. 4  Resistance coefficient of hot side and cold side

Tab. 5  Lateral resistance coefficient combination in D2

Fig. 1  Structure of the Water-cooled charge air cooler

Fig. 2  Transverse flow fin element of hot side

Fig. 3  Geometric parameters of the fin

Fig. 4  Mesh generation of hot side 1/5 fin element in main flow

Fig. 5  Fitting results of the fin element

Fig. 6  Section of the fin element

Fig. 7  Influence of boundary layer on the pressure field

Fig. 8  Experimental acquisition process

Fig. 9  The contours of static pressure

Fig. 10  The pressure loss comparison in cold side

Fig. 11  The pressure loss comparison in hot side

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Greek symbols</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v$</td>
<td>flow velocity vector</td>
<td>$\nu$</td>
</tr>
<tr>
<td>$s_i$</td>
<td>the source term</td>
<td>$\mu$</td>
</tr>
<tr>
<td>$D_{ij}$</td>
<td>viscous resistance coefficient</td>
<td>$\rho$</td>
</tr>
</tbody>
</table>
\[ C_{ij} \quad \text{inertial resistance coefficient} \]
\[ G_k \quad \text{kinetic energy caused by the average velocity gradient} \]
\[ G_h \quad \text{turbulent energy generated by the buoyancy effect} \]

<table>
<thead>
<tr>
<th>Subscripts</th>
<th>hot</th>
<th>hot side</th>
</tr>
</thead>
</table>

Tab. 1  Resistance coefficient of different flow passage length

<table>
<thead>
<tr>
<th>Inertial resistance</th>
<th>Viscosity resistance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full size model</td>
<td>21.5</td>
</tr>
<tr>
<td>1/5 model</td>
<td>21</td>
</tr>
</tbody>
</table>

Tab. 2  Resistance coefficient of symmetric boundary

<table>
<thead>
<tr>
<th>Symmetrical boundary inertial resistance</th>
<th>Symmetrical boundary viscous resistance</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>21.27</td>
<td>23423815</td>
</tr>
</tbody>
</table>

Tab. 3  Resistance coefficient without boundary layer

<table>
<thead>
<tr>
<th>Borderless inertial resistance</th>
<th>Borderless viscous resistance</th>
</tr>
</thead>
<tbody>
<tr>
<td>71</td>
<td>22750657</td>
</tr>
</tbody>
</table>

Tab. 4  Resistance coefficient of hot side and cold side

<table>
<thead>
<tr>
<th>D1</th>
<th>D2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inertial resistance</td>
<td>Viscous resistance</td>
</tr>
<tr>
<td>hot side</td>
<td>21</td>
</tr>
<tr>
<td>cold side</td>
<td>47</td>
</tr>
</tbody>
</table>
Tab. 5  Lateral resistance coefficient combination in D2

<table>
<thead>
<tr>
<th></th>
<th>Hot side</th>
<th>Cold side</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inertial resistance</td>
<td>Viscous resistance</td>
</tr>
<tr>
<td>1</td>
<td>154.2</td>
<td>29790384</td>
</tr>
<tr>
<td>2</td>
<td>2100</td>
<td>2275065700</td>
</tr>
<tr>
<td>3</td>
<td>21000</td>
<td>22750657000</td>
</tr>
</tbody>
</table>
Fig. 1  Structure of the Water-cooled charge air cooler

Fig. 2  Transverse flow fin element of hot side

Fig. 3  Geometric parameters of the fin

Fig. 4  Mesh generation of hot side 1/5 fin element in main flow
Fig. 5  Fitting results of the fin element

Fig. 6  Section of the fin element

Fig. 7  Influence of boundary layer on the pressure field
Fig. 8  Experimental acquisition process

Fig. 9  The contours of static pressure

Fig. 10  The pressure loss comparison in cold side
Fig. 11  The pressure loss comparison in hot side