International Communications in Heat and Mass Transfer 59 (2014) 75–87

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Multi-scale thermal analysis approach for the typical heat exchanger in automotive cooling systems☆

Yuqi Huang [a], Zhentao Liu [a], Guodong Lu [b], Xiaoli Yu [a,]*

[a] Power Machinery and Vehicular Engineering Institute, Zhejiang University, Hangzhou, PR China
[b] Zhejiang Yinlun Machinery Co., Ltd., Tiantai 317200, PR China

ARTICLE INFO

Available online 25 October 2014

Keywords:
Multi-scale coupled
Computational Fluid Dynamics (CFD)
Intercooler
Wind tunnel

ABSTRACT

CFD/CHT simulations have been widely used in simulating cross flow compact heat exchangers, but the calculating accuracy is commonly limited. The multi-scale thermal analysis approach was adopted to study an intercooler, one of the representative examples of cross flow heat exchangers in automotive cooling systems. With the application of mesh refinement and datum interpolation technique, the inside flow and heat transfer mechanism were analyzed, the pressure and temperature data were also calculated. The model was validated with experimental data based on wind tunnel tests, and the results show that the multi-scale coupled calculation is in good agreement with experimental values, especially in the heat transfer simulations. The research of the inside flow and heat transfer mechanism is capable of providing a basis for optimization of compact heat exchangers.

1. Introduction

The automotive cooling system consists of many compact heat exchangers such as radiator, oil cooler, intercooler, condenser and pump, and fan. It is an important subsystem to ensure the vehicular stability, and it has a direct effect on energy-saving and emission reduction. Since the heat exchangers directly affect the heat rejection capability of the system, there is a need to develop analysis procedures that can easily and accurately estimate the heat exchanger working performance.

Among all the heat exchangers for engine cooling applications, cross flow compact heat exchangers with different fins are most often used because of their higher efficiency. CFD/CHT simulations have been widely used for analyzing the flow and thermal characteristic of compact heat exchangers, but the calculating capacity is commonly limited by the computational conditions. The multiplicity of length scales, ranging from the fin thickness with dimensions in the 10^{-4} m scale, to the cooler size, almost in the 1 m scale range, makes the simultaneous resolution of all geometrical details expensive and often unrealistic. Under the current conditions, it is unlikely to cover the full heat exchanger scale by a detailed model. Then many researchers limit their analysis to a representative portion of the exchangers, obtain its own numerical heat transfer and friction correlations [1–4]. CFD tools are nowadays predicting with a high level of accuracy, yet the performance of a whole heat exchanger is hard to be accurate predicted only by the studying of one portion. The flow characteristic in a full exchanger is quite different from a portion and also worth studying.

Several strategies have been introduced to handle these issues. Some kind of distributed models have been implemented in the compact heat exchanger. Oliet et al. [5–8] did a series of researching and developing works about the vehicular heat exchangers. They resolved the smallest scales by using empirical correlations and validated their simulation by experiments [9]. Jung and Assanis [10] built a numerical model based on the thermal resistance concept. This model could effectively predict the heat exchange performance, and the calculation results were in good agreement with the experimental data. Compared to the CFD tools, the distributed models could be more efficient, and easy to be constantly revised and approved, but a little bit narrow in scope of applications. They are appropriate for a long-time, systematic research which is focused on a certain heat exchanger type.

Early in 70's last century, Patankar and Spalding [11] introduced the distributed-resistance concept to solve the flow field and enthalpy distribution of a steam generator. The flows in the generator are treated somewhat like those in porous media, and the solid objects are taken as distributed resistances to flow and heat transfer in the paper. This methodology could be applied in many kinds of engineering equipments that contain solid objects, such as heat exchanger [12,13]. From then on, the porous simplification turns to one of the most popular methods in simulating the whole cooling module [14–16]. Prithiviraj and Andrews [17–19] simulated a shell-and-tube heat exchanger. In their model, the tubes were treated by using distributed-resistance approach. Tang et al. [20] calculated an electronics cooling module by representing the internal offset fin structure with a Brinkman–Forchheimer-extended Darcy porous media model, and the simulated

☆ Communicated by W.J. Minkowycz

* Corresponding author.
E-mail addresses: huangyuqi@zju.edu.cn (Y. Huang), yuxl@zju.edu.cn (X. Yu).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2014.10.022
0735-1933/

Nomenclature

F_h [m] Fin height
F_p [m] Fin pitch
δ [m] Fin thickness
L_d [m] Fin length along the flow direction
b [m] Amplitude of the wavy-fin
L [m] Wavelength of the wavy-fin
f [–] Friction factor
j [–] Heat transfer factor
Re [–] Reynold's number
D_h [m] Hydraulic diameter of the fin
V_c [m/s] Average velocity of the flow field
ΔP_i [Pa] Pressure differential produced by the circulation area abruptly narrow
ΔP_e [Pa] Pressure differential produced by the circulation area abruptly widening
T [K] Temperature
b_c [m] The feature size of the fins
m [–] The dimensionless coefficient of fin height
h [–] The heat transfer coefficient
Q [J] Total heat amount
F_1 [m^2] The direct-heat-transfer surfaces area
F_2 [m^2] The secondary heat-transfer surfaces area
α^{-1} [m^{-2}] Viscous resistance
C_2 [m^{-1}] Inertial resistance
γ [–] Porosity
k [–] Turbulence kinetic energy
ε [–] Turbulence dissipation rate
G_k [–] Generation of turbulence kinetic energy
v_{mag} [m/s] Velocity magnitude
S_i [–] The source term for the ith (x, y, or z) momentum equation
ρ [kg/m^3] Density
μ [Pa·s] Dynamical viscosity
E_f, E_s [J] Total fluid energy and total solid energy
K_{eff} [–] Effective thermal conductivity of the medium
$\bar{\bar{\tau}}$ [Pa] Stress tensor
$\vec{v}$ [m/s] Overall velocity vector
Δn [m] The thickness of porous medium

results agreed well with the measured temperature profile. This method could also be used in simulating an automotive heat exchanger and the key was estimating the porosity, and the viscous and inertial resistance values, although the simulated heat dissipation sometimes was far from the real values because all porous parameters were deduced from pressure characteristics.

In order to balance the calculating accuracy and computational efficiency, the multi-scale coupling method has been put forward in several cases. Wang and Li [21] adopted the multi-scale CFD approach in gas–solid two-phase flow simulation. Cheng et al. [22] elaborated on the multi-scale method and application in wing-store simulation. Due to its outstanding effectiveness, the multi-scale coupled approach has been introduced into the simulation of heat exchangers. Carluccio et al. [23] employed the coarse mesh and porous model to analyze a compact cross flow heat exchanger as a first step, then set up submodels according to the fin structures, creating very fine mesh, and studying the characters of internal turbulent flow and local heat transfer.

On the basis of the existing research, this paper deals with an intercooler, one of the representative examples of cross flow heat exchangers in automotive cooling systems. With the application of mesh refinement and datum interpolation technique, the pressure and temperature data are calculated. Commercial computational fluid dynamic (CFD) codes ANSYS FLUENT 13 based upon the finite volume method are used to make the simulation, and the results were validated with experimental data based on wind tunnel tests.

2. Numerical simulations

2.1. The intercooler

A plate-fin intercooler is taken as the study object. Hot-pass and cold-pass are arranged alternately (photo and size are shown in Fig. 1).

As shown in Fig. 1, the size of the core is 300 mm by 300 mm by 80 mm. The thickness of the partitions between hot-pass and cold-pass is 0.6 mm and the fin thickness is 0.2 mm.

Because considerable dirt and dust are contained in low-altitude air, the serrated, louvered, and punctured fin structures are not suited to the cold side of the intercooler. The present intercooler consists of plane-fins in hot-pass and wavy-fins in cold-pass, and fin types are shown in Fig. 2.

Plane-fin $F_h = 6.5$ mm; $F_p = 2.5$ mm
Wavy-fin $L_d = 80$ mm; $L = 12$ mm; $b = 1.8$ mm; $F_h = 7.5$ mm; $F_p = 4.25$ mm

2.2. Multi-scale methodology

To meet the goals of high efficiency and accurate estimation, a multi-scale-coupled approach was utilized to study the internal flow and heat transfer progress. The method comprised the following steps:

Step 1. Simulate the entire intercooler. The porous media model is used to decrease the model complexity and computational effort.
Step 2. Analyze the flow and heat transfer characters, fix on key locations, and extract the regional distributed data.
Step 3. Model the key regions with all detailed fin structures. Calculate the detailed local model, interpolating the regional data extracted by step 2, studying the micro-flow and heat mechanism.
Step 4. Synthesize the simulation results, interpolate the data computed by detailed local models back to the entire model, and then obtain the performance predictions.

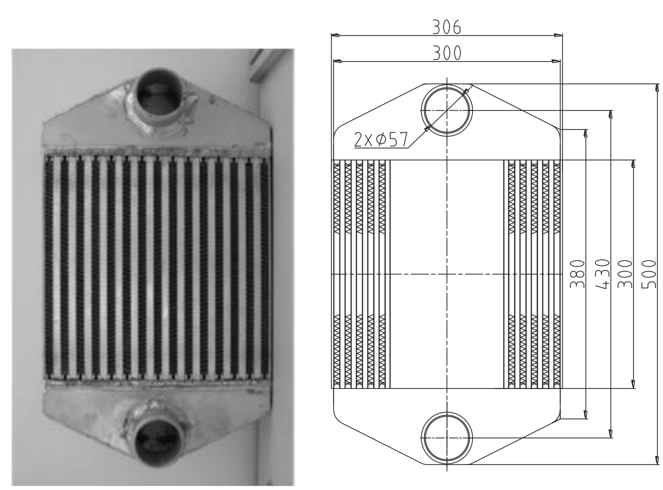

Fig. 1. Photo and the size of intercooler.