Gas Flow Development Through Tandem Heat Exchangers Inside Exhaust Nozzle by Using Porous Medium Model

Liu Xiyue[1], *Zhang Jingzhou*[1,2*], *Lou Decang*[3], *Kang Yong*[3]

1. College of Energy and Power Engineering, Jiangsu Province Key Laboratory of Aerospace Power System, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, P. R. China;
2. Collaborative Innovation Center of Advanced Aero-Engine, Beijing 100191, P. R. China;
3. China Gas Turbine Establishment, Chengdu 610500, P. R. China

(Received 10 March 2015; revised 7 August 2015; accepted 24 August 2015)

Abstract: A computational study on the flow development through tandem double-U-shaped-tubes compact heat exchangers inside exhaust nozzle is presented. In order to simplify the computational process on modeling the flow field, the compact heat exchanger is modeled as a porous matrix by using an isotropic porous medium assumption, which makes two-dimensional numerical simulation realistic. With the use of an existed quadratic relation which connects the pressure drop with the inlet velocity in the external part of the heat exchanger, the permeability and drag coefficient in the porous medium model are determined and a corresponding computational method validation is also made. Two schemes of tandem double-U-shaped-tubes compact heat exchangers are numerically analyzed. In relative to the baseline scheme, the modified scheme is improved by smoothing the nozzle expansion, varying heat exchanger mounting angle and installing boat-tail ramp at the trailing edge of the last heat exchanger. The results show that the pressure losses due to the existence of local recirculation zones and inappropriate distribution of the flow field are reduced in the modified scheme. The pressure loss coefficient is decreased from 1.7% under the baseline scheme to 1.2% under the modified scheme.

Key words: compact heat exchanger; exhaust nozzle; porous medium; pressure loss; numerical simulation

CLC number: V231.1 **Document code**: A **Article ID**: 1005-1120(2016)02-0147-08

0 Introduction

The concept of an inter-cooled recuperative engine is schematically shown in Fig. 1. Between the low pressure compressor (LPC) and the high pressure compressor (HPC), the air is forced to flow through the intercoolers installed in the bypass duct. After HPC, the air is fed into a series of recuperators which are installed inside the exhaust nozzle of an aero-engine. The main target of heat recuperator is the exploitation of thermal energy from turbine exhaust gas for pre-heating the air before combustion and thus to decrease fuel consumption and pollutant emissions[1,2].

The heat exchangers installed in the exhaust nozzle were previously developed by many researchers[3-6]. A typical kind of the heat exchangers is also schematically represented in Fig. 1, that is, the double-U-shaped bundle heat exchanger. This kind of heat exchanger is regarded

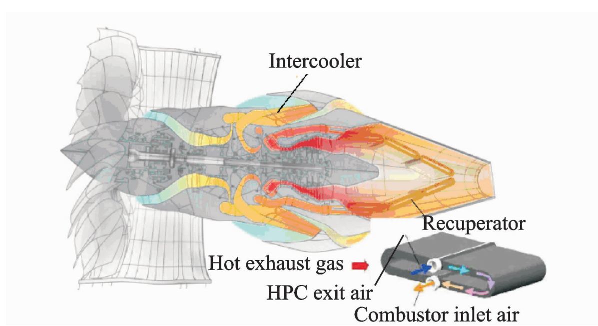

Fig. 1 Schematic of inter-cooled recuperative engine

* Corresponding author, E-mail address: zhangjz@nuaa.edu.cn.

How to cite this article: Liu Xiyue, Zhang Jingzhou, Lou Decang, et al. Gas flow development through tandem heat exchangers inside exhaust nozzle by using porous medium model[J]. Trans. Nanjing Univ. Aero. Astro., 2016,33(2):147-154.

http://dx.doi.org/10.16356/j.1005-1120.2016.02.147

as durable and low cost, which is constructed by special profiled tubes in order to achieve a minimum possible pressure drop in the exhaust nozzle system. The heat exchanger consists of two manifold tubes. The cooler air from HPC enters the upper manifold (distributor tube) from both sides and is distributed into the U-shaped profile tubes, which are brazed into the manifold tubes. The lower manifold (collector tube) collects the preheated air and then leads the preheated air back to the combustion chamber.

In general, detailed analysis of the flow through a compact heat exchanger can be carried out by using computational fluid dynamics (CFD) directly[7]. As the heat exchanger consists of a large number of pipes, huge grid number is needed for the detailed grid meshes outside and inside the numerous characteristic flow passages. The need for computer CPU and memory requirements would be enormous, making the computational task nearly impossible. In order to overcome this problem, the compact heat exchanger can be modeled as a porous medium matrix having the same thermal and flow behavior as the original device[8-10]. This treatment eliminates the need to model all the characteristic passages, but the restriction is located to the need of having a realistic and representative pressure drop law through the heat exchanger, providing an un-expensive computational solution of the flow field especially when the interest is not so much focused on the flow in the direct vicinity of the heat exchanger. In this manner, the correct pressure-velocity relationship is the key and it should be accurately built up for the specific heat exchanger.

Missirlis et al. [11-16] made a series of experimental and computational study for the flow development through a heat exchanger for aero-engine applications. Detailed flow measurements using a 3-hole pitot-static probe were carried out on a 1 ∶ 1 scale model of the heat exchanger in order to measure the pressure drop through the heat exchanger and the velocity distribution behind it. Based on the experimental data, a quadratic relationship of pressure drop versus an effective local velocity was derived for the specific heat exchanger. During the numerical modeling, the heat exchanger matrix was modeled using an isotropic porous medium assumption. They also examined the applicability of the quadratic law in a variety of velocity inlet conditions configured by different angles of attack. For all the examined cases, the computational results agreed well with the experimental data.

A numerical study on the flow development through tandem double-U-shaped-tubes compact heat exchangers inside exhaust nozzle is conducted by using porous medium model in this paper. Firstly, the computational method is verified from a simple flow inside rectangular duct equipped with U-shaped tubes. Secondly, the flow fields inside two-dimensional nozzle equipped with tandem double-U-shaped-tubes heat exchangers are numerically analyzed. Based on the flow field analysis of the baseline scheme, a modified scheme is presented for reducing the pressure losses inside nozzle.

1 Mathematic Model for Porous Medium

A porous medium assumption is widely adopted for modeling the flow through a complex device. A more advanced version of the Darcy-Forchheimer pressure drop law is given by[16]

$$\boldsymbol{S} = \mu \boldsymbol{D} \boldsymbol{V} + \frac{1}{2} \rho \boldsymbol{F} \left| \boldsymbol{V} \right| \boldsymbol{V} \tag{1}$$

where $\boldsymbol{S}$ is the pressure gradient vector and $\boldsymbol{V}$ the velocity vector. $\boldsymbol{D}$ and $\boldsymbol{F}$ are the first- and second-order pressure loss coefficient matrix, respectively.

$$\boldsymbol{S} = \begin{bmatrix} \partial p/\partial x \\ \partial p/\partial y \\ \partial p/\partial z \end{bmatrix}, \quad \boldsymbol{D} = \begin{bmatrix} D_{xx} & D_{xy} & D_{xz} \\ D_{yx} & D_{yy} & D_{yz} \\ D_{zx} & D_{zy} & D_{zz} \end{bmatrix}$$

$$\boldsymbol{F} = \begin{bmatrix} F_{xx} & F_{xy} & F_{xz} \\ F_{yx} & F_{yy} & F_{yz} \\ F_{zx} & F_{zy} & F_{zz} \end{bmatrix}, \quad \boldsymbol{V} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}$$

This more advanced version of the Darcy-Forchheimer pressure drop law aims at accurately predicting the pressure drop behavior of an anisotropic porous medium towards the various flow directions. For the isotropic porous medium assumption, Eq. (1) can be simplified as

$$S_i = \frac{\mu}{K} V_i + \frac{1}{2} C\rho \left| V_i \right| V_i \tag{2}$$

where K is the specific permeability of the porous medium, μ the fluid dynamic viscosity, and C the drag coefficient.

In the flow filed computation by using a porous medium model, it is essential to define the parameters of K and C in Eq. (2). These parameters can be derived the following relationship of pressure drop versus an effective local velocity

$$\frac{\Delta p}{l} = \frac{\mu}{K} V + C\rho V^2 \tag{3}$$

or

$$\Delta p = a_1 V + a_2 V^2 \tag{4}$$

where a_1 and a_2 are the coefficients.

2 Verification of Computational Method

2.1 Physical model

The structure of U-shaped tubes heat exchanger inside rectangular channel is taken from Ref. [11], which is shown in Fig. 2. The two-dimensional channel is 346 mm in length and 364 mm in height. This heat exchanger is formed by U-shaped tubes of elliptic cross section arranged in a 4/3/4 staggered configuration. The side view of the heat exchanger and the geometric details of the staggered arrangement of the characteristic flow passage are shown in Fig. 2. The manifold diameter is 56 mm and the supporting plate thickness is 2 mm.

2.2 Calculation method

The computational domain used for the heat exchanger matrix is subdivided into a number of porous blocks, such as four rectangular blocks and one bend side blocks, as shown in Fig. 3. Two manifolds and supporting plates are also modeled. Structured grids are constructed in the blocks of the heat exchanger matrix, also in the upstream and the downstream region of the heat

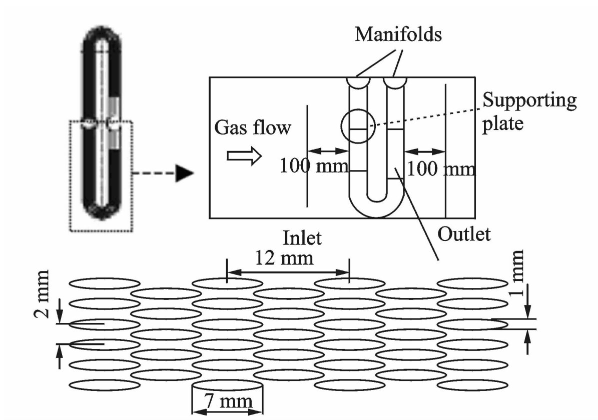

Fig. 2 Heat exchanger matrices with elliptic tubes

exchanger. While in the computational zone surrounding the bend portion of heat exchanger, unstructured grids are adopted. The grid generation is realized by using Gambit software. In the present computation, a grid system of approximate 210 000 points is used.

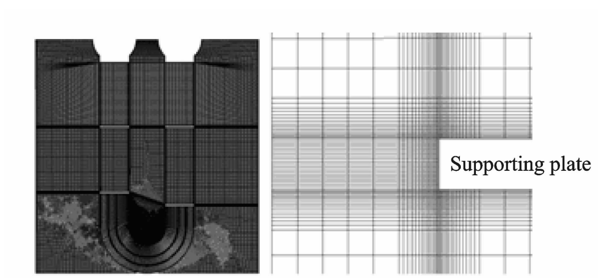

Fig. 3 Grid dividing (Local region in the right)

According to the experimental date[11], the relationship of static pressure drop and velocity is presented as

$$\Delta p = 7.171 V^2 + 10.088 V \tag{5}$$

Thus the specific permeability and the drag coefficient for this specific heat exchanger are determined from Eq. (5).

The boundary conditions of computational domain are specified as follows. The channel flow inlet is set as velocity inlet by giving the flow velocity ranged from 3 m/s to 11 m/s. A turbulence intensity of 5% and a turbulence length scale of 3% of the inlet hydraulic diameter are used. The flow outlet condition is set as pressure-outlet with the reference static pressure of 101 325 Pa. As the heat transfer is not taken into consideration and the flow is incompressible, the gas properties are set as constant with the density of 1.196 8 kg/m^3 and dynamic viscosity of 18.22×10^{-6} Pa • s.

The computation is accomplished based on Fluent software. The form of the pressure drop given by Eq. (5) is introduced into the flow solver in order to estimate an additional source term in the momentum equations. SST k-ω turbulence model is used to model turbulent flow. The standard of result convergence is that every residual accuracy is less than 10^{-5}.

2.3 Calculation results

Fig. 4 shows the reference static pressure (Relative to the ambient pressure) and flow vector plots through the heat exchanger. These plots refer to the maximum inlet velocity of 12 m/s. It is seen that a great pressure drop occurs across the heat exchanger. Additionally, the development of the wake behind the manifold is also observed.

The computed relationship of pressure drop versus the inlet velocity is presented in Fig. 5, where $\Delta p' = \frac{1}{2}\Delta p$. It is seen that the computed pressure drops inside the channel agree well with the experimental results presented in Ref. [11], with the maximum relative error of approximately 2.64%. It is confirmed that the present computational method is capable of predicting the pressure loss inside the channel equipped with the compact heat exchanger.

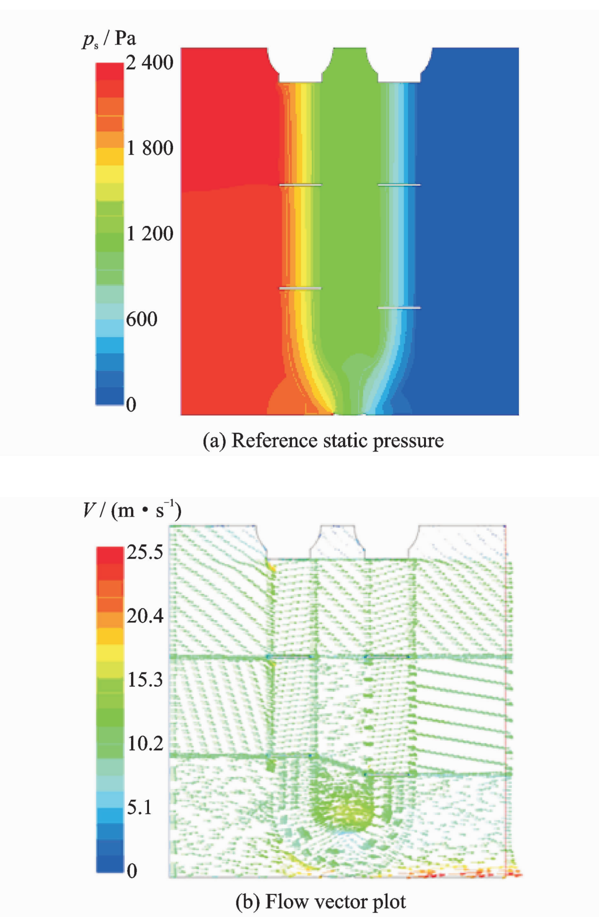

Fig. 4 Computed flow fields

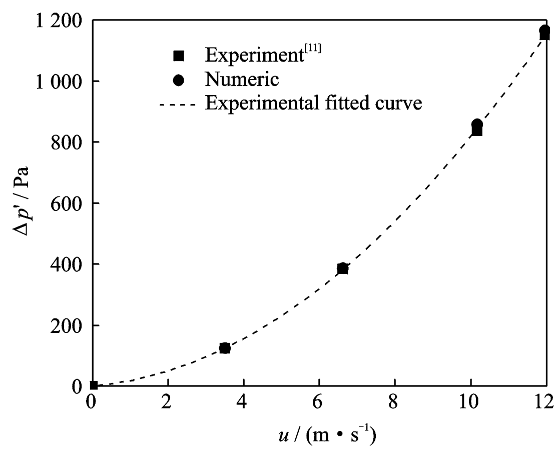

Fig. 5 Comparison between the presented computation and experimental data[11]

3 Internal Flow Inside Nozzle

3.1 Baseline scheme

The nozzle is simplified as a two-dimensional variable cross-section channel. The computational domain is shown in Fig. 6(a). The length of computational domain is 5 500 mm with inlet height of 500 mm and outlet height of 330 mm. The length and height of the central cone are 1 650 mm and 276 mm, respectively. Inside the nozzle three double-U-shaped-tubes heat exchangers are installed in tandem. These heat exchangers are as modeled as three porous matrices. Their original arrangement is also shown in Fig. 6 (a). The front heat exchanger is arranged horizontally with mounting angle of 0°. Both the middle and the rear heat exchangers are inclined with 20°. Each heat exchanger unit has the same configuration, as shown in Fig. 6(b). The U-shaped-tubes arrangement of each heat exchanger unit is the same as the structure in Section 2. The con-

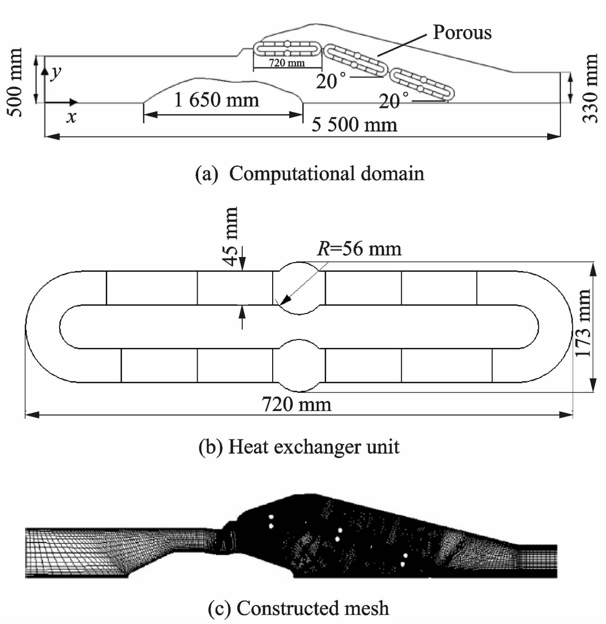

Fig. 6 Baseline scheme of tandem heat exchangers arrangement

structed mesh is shown in Fig. 6(c), composed of approximately 850 000 computational cells.

Velocity-inlet and pressure-outlet boundary conditions are used with the inlet velocity of 22 m/s and outlet pressure of 101 325 Pa.

Fig. 7 presents the velocity vector inside the nozzle for the baseline arrangement scheme of tandem heat exchangers. It is seen that a large recirculation zone appears in the vicinity of the front horizontal heat exchanger unit, downstream the nozzle expansion section. The reason is mainly due to the abrupt expansion of nozzle surface. Additionally, the presence of the horizontal heat exchanger unit aggravates the flow separation because it affects the flow expansion to the nozzle surface. Therefore, little primary flow is driven to pass through the front heat exchanger. A small recirculation zone is also observed downstream the rear heat exchanger. The primary flow is forced to pass through the inclined heat exchanger units, resulting in wake flow behind the rear heat exchanger.

Fig. 8 shows the contour plots of the velocity components together with the static and total pressures. It is clearly shown from Figs. 8(a), (b) that local streamwise velocity in the zone on top of the front horizontal heat exchanger unit appears negatively and the local vertical velocity passing through the front heat exchanger is very low simultaneously. Both the streamwise velocity and the vertical velocity passing across the middle and rear heat exchanger units are significantly higher than these for the front horizontal heat exchanger due to the non-zero angle of attack. After penetrating the heat exchangers, the primary flow is accelerated inside the convergent section of the nozzle. Owing to the obstruction of the heat exchanger units, the primary flow is expelled to

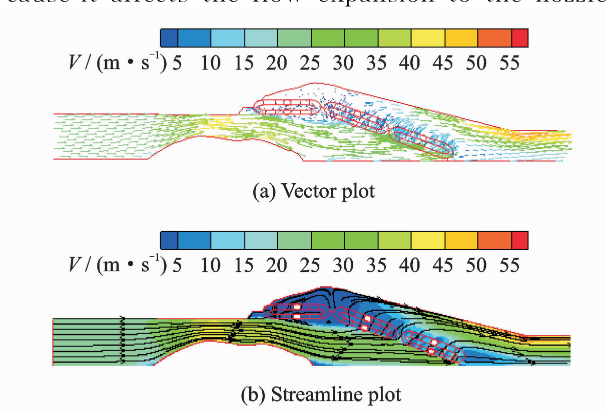

Fig. 7 Velocity vector inside exhaust nozzle for baseline scheme of tandem heat exchangers

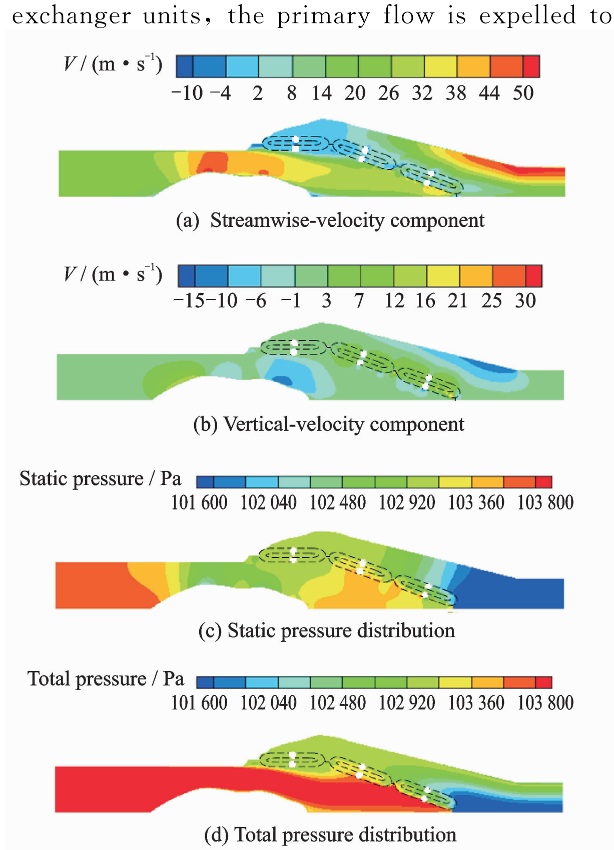

Fig. 8 Flow fields inside exhaust nozzle for baseline scheme

nozzle surface in the vicinity of the nozzle outlet, resulting in stronger streamwise velocity and inward velocity. As regards as the pressure distribution is concerned, the static pressure drop is well illustrated in Fig. 8(c). The total pressure distribution at the nozzle outlet is non-uniform. A strong total pressure gradient from the nozzle surface to the nozzle center is exhibited.

3.2 Modified scheme

Considering the flow field features of the baseline arrangement scheme, it is inferred that two main faults are involved in this preliminary scheme. Firstly, a large recirculation zone in the vicinity of the front horizontal heat exchanger contributes additional flow loss. Simultaneously, low penetration capacity of the primary flow into the front horizontal heat exchanger is unfavourable for the heat transfer. Secondly, the wake flow downstream the rear heat exchanger in the vicinity of the nozzle outlet is also unfavourable for the flow recovery.

In order to attenuate the impact of low-speed zone and recirculation zone on aerodynamic performance, a modified scheme is presented, as shown in Fig. 9. Three modifications are made in the modified scheme. Firstly, the nozzle surface expansion is smoothed for weakening the recirculation zone in the vicinity of the front heat exchanger. Secondly, the mounting angle of the front heat exchanger is changed from 0° to 17° for increasing the flow penetration through the front heat exchanger. As the inclination angle of the front heat exchanger increases, the trailing position of the front heat exchanger declines in relative to the nozzle central line. Therefore, the positions and mounting angles of the middle and rear heat exchangers should be adapted accordingly. The mounting angle of the middle heat exchanger remains as the original angle and the

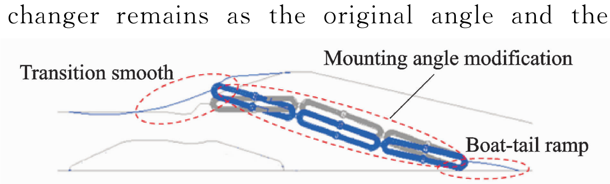

Fig. 9 Modified scheme of heat exchangers arrangement

mounting angle of the middle heat exchanger is adjusted from 20° to 13°. Thirdly, a boat-tail ramp is added near the trailing edge of the rear heat exchanger for eliminating the wake flow downstream the rear heat exchanger.

Fig. 10 presents the velocity vector inside the nozzle for the modified arrangement scheme of tandem heat exchangers. By comparison with the baseline case (See Fig. 7), it is seen that the primary flow penetration through the front heat exchanger is enhanced obviously and the wake flow downstream the rear heat exchanger is nearly eliminated under the modified scheme. Thus it is concluded that the modified scheme is more reasonable for improving the flow performance inside the nozzle.

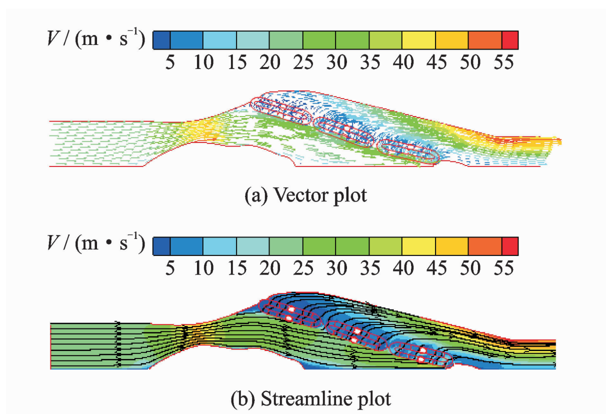

Fig. 10 Velocity vector inside exhaust nozzle for modified scheme of tandem heat exchangers

Fig. 11 shows the contour plots of the velocity components together with the static and total pressures for the modified arrangement scheme. By comparison with the baseline case (See Fig. 8), it is well demonstrated that the flow field is improved. Besides the advantages of enhancing primary flow penetration through the front heat exchanger, and eliminating wake flow downstream the rear heat exchanger, the velocity and pressure distributions at streamwise section seems more uniform than the baseline case.

In order to evaluate the flow performance improvement quantitatively, a pressure loss coefficient is presented as follows

$$\xi=\frac{\Delta p}{p_{\text{inlet}}} \tag{6}$$

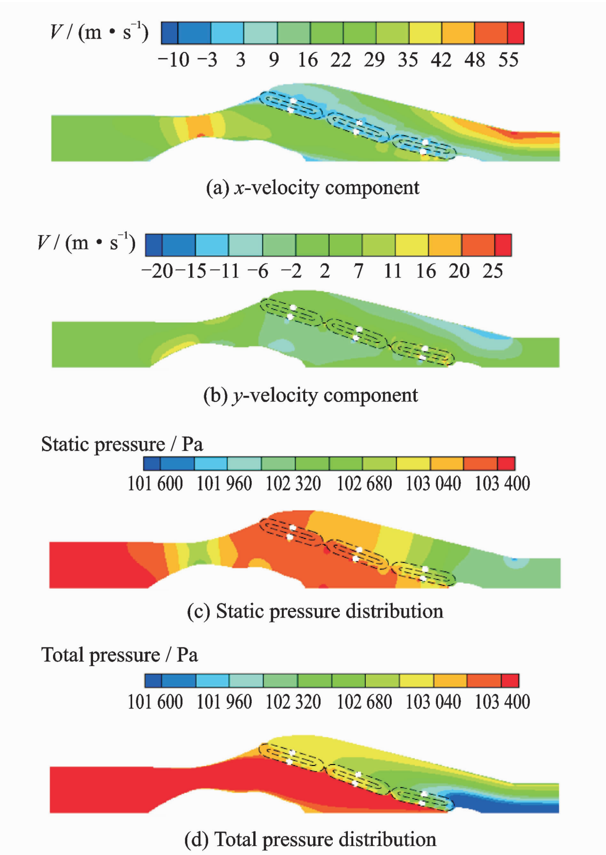

Fig. 11 Flow fields inside exhaust nozzle for modified scheme

where Δp is the total pressure drop between the inlet and outlet of the nozzle and p_{inlet} the averaged total pressure at the nozzle inlet.

Table 1 presents the comparison of pressure losses. From the computational results, the pressure drop under the modified arrangement scheme is reduced approximately 500 Pa in relative to the baseline scheme. The pressure loss coefficients for the baseline scheme and the modified scheme are 1.7% and 1.2%, respectively.

Table 1 Comparison of pressure losses

Scheme	p_s/Pa	p_T/Pa	ξ/%
Baseline	2 479	1 743	1.7
Improved	1 911	1 210	1.2

4 Conclusions

The flow development through the compact heat exchanger inside a two-dimensional channel is investigated with the use of computational fluid dynamics on the base of porous medium approach. Numerical method verification shows that porous medium model is capable of modeling the flow field through the compact heat exchanger.

Two schemes of tandem double-U-shaped-tubes compact heat exchangers are numerically analyzed. The flow features inside the nozzle are well illustrated by using the porous medium model. Based on the computational results for the baseline scheme, a modified scheme is presented by smoothing the nozzle expansion, modifying heat exchanger mounting angle and installing boat-tail ramp at the trailing edge of the rear heat exchanger. This modified scheme is capable of reducing the pressure drop by improving the flow fields inside the nozzle.

References:

[1] KYPRIANIDIS K, GRONSTEDT T, OGAJI S, et al. Assessment of future aero engine designs with intercooled and intercooled recuperated cores [J]. ASME Journal of Engineering for Gas Turbines and Power, 2011, 133(1): 011701-1-10.

[2] GONG Hao, WANG Zhanxue, KANG Yong, et al. Performance calculation and analysis of intercooled recuperated aero-engine [J]. Journal of Aerospace Power, 2014, 29(6): 1453-1461. (in Chinese)

[3] SCHOENENBORN H, ELBERT E, SIMON B, et al. Thermo-mechanical design of a heat exchanger for a recuperative aeroengine [J]. ASME Journal of Engineering for Gas Turbine and Power, 2006, 128(4): 736-744.

[4] YAKINTHOS K J, MISSIRLIES D K, PALIKARAS A C, et al. Heat exchangers for aero engine applications: IMECE2006-13667 [R]. Chicago IL, United States: ASME, 2006.

[5] MIN J K, JEONG J H, MA M Y, et al. High temperature heat exchanger studies for applications to gas turbines[J]. Heat Mass Transfer, 2009, 46(2): 175-186.

[6] KIM Y G, CHOI B I, KIM K, et al. Performance analysis and optimal design of heat exchangers used in high temperature and high pressure system[J]. International Journal of Aeronautical and Space Sciences, 2010, 11: 19-25.

[7] BENHAMADOUCHE S, LAURENCE D. LES, coarse LES, and transient RANS comparisons on the

flow across a tube bundle[J]. International Journal of Heat and Mass Transfer, 2003, 24(4): 470-479.
[8] DENG Bin, LI Xin, TAO Wenquan. Application of porous media and distributed resistance modeling in numerical simulation of shell-and-tube heat exchanges[J]. Journal of Engineering Thermophsics, 2004, 25(S): 167-169. (in Chinese)
[9] TOMIMURA T, HAMANO K, HONDA Y, et al. Experimental study on multi-layered type of gas-to-gas heat exchanger using porous media[J]. International Journal of Heat and Mass Transfer, 2004, 47 (21): 4615-4623.
[10] HAYES A M, KHAN J A, SHAABAN A H, et al. The thermal modeling of a matrix heat transfer exchanger using a porous medium and the thermal non-equilibrium model[J]. International Journal of Thermal Science, 2008, 47(10): 1306-1315.
[11] MISSIRLIS D, YAKINTHOS K, PALIKARAS A, et al. Experimental and numerical investigation of the flow field through a heat exchanger for aero-engine applications[J]. International Journal of Heat and Fluid Flow, 2005, 26(3): 440-458.
[12] MISSIRLIS D, YAKINTHOS K, STORM P, et al. Modeling pressure drop of inclined flow through a heat exchanger for aero-engine applications[J]. International Journal of Heat hand Fluid Flow, 2007, 28(3): 512-515.
[13] YAKINTHOS K, MISSIRLIS D, PALIKARAS A, et al. Optimization of the design of recuperative heat exchangers in the exhaust nozzle of an aero engine [J]. Applied Mathematical Modeling, 2007, 31 (11): 2524-2541.
[14] ALBANAKIS C, YAKINTHOS K, KRITIKOS K, et al. The effect of heat transfer on the pressure drop through a heat exchanger for aero engine applications [J]. Applied Thermal Engineering, 2009, 29(4): 634-644.
[15] KRITIKOS K, ALBANAKIS C, MISSIRLIS D, et al. Investigation of the thermal efficiency of a staggered elliptic-tube heat exchanger for aeroengine applications[J]. Applied Thermal Engineering, 2010, 30(2/3): 134-142.
[16] MISSIRLIS D, DONNERHACK S, SEITE O, et al. Numerical development of a heat transfer and pressure drop porosity model for a heat exchanger for aero engine applications[J]. Applied Thermal Engineering, 2010, 30(11/12): 1314-1350.

Mr. **Liu Xiyue** is a Ph. D. candidate of Nanjing University of Aeronautics. His research interests are combustion and heat transfer.

Dr. **Zhang Jingzhou** is a professor and doctoral supervisor in Nanjing University of Aeronautics and Astronautics. His research interests focus on combustion and heat transfer.

Dr. **Lou Decang** is a senior engineer in China Gas Turbine Establishment. His research interests focus on combustion and heat transfer.

Mr. **Kang Yong** is a researcher in China Gas Turbine Establishment. His research interests focus on combustion and heat transfer.

(Executive Editor: Xu Chengting)