Investigation of the pressure drop inside a rectangular channel with a built-in U-shaped tube bundle heat exchanger

Xi-yue Liua, Jing-zhou Zhanga,b and Yong Shanα

α College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing, China; b Jiangsu Province Key Laboratory of Aerospace Power System, Nanjing University of Aeronautics and Astronautics, Nanjing, China

ABSTRACT

A simplified approach which utilizes an isotropic porous medium model has been widely adopted for modeling the flow through a compact heat exchanger. With respect to situations where the compact heat exchangers are partially installed inside a channel, such as the application of recuperators in an intercooled recuperative engine, the use of an isotropic porous medium model needs to be carefully assessed because the flow passing through the heat exchanger is very complicated. For this purpose, in this study the isotropic porous medium model is assessed together with specific pressure–velocity relationships for flow field modeling inside a rectangular channel with a built-in double-U-shaped tube bundle heat exchanger. Firstly, experiments were conducted using models to investigate the relationship between the pressure drop and the inlet velocity for a specific heat exchanger with different installation angles inside a rectangular channel. Secondly, a series of numerical computations were carried out using the isotropic porous medium model and the pressure–velocity relationship was then modified by introducing correction coefficients empirically. Finally, a three-dimensional (3-D) direct computation was made using a computational fluid dynamics (CFD) method for the comparison of detailed flow fields. The results suggest that the isotropic porous medium model is capable of making precise pressure drop predictions given the reasonable pressure–velocity relationship but is unable to precisely simulate the detailed flow features.

1. Introduction

The intercooled recuperative engine is regarded as an innovative concept for improving the fuel consumption...
and reducing the pollutants of high-bypass turbofan engines (Kyriyanidis, Grönsstedt, Ogaji, Pilidis, & Singh, 2011; Noppel, Lucisano, & Singh, 2009). Recuperative heat exchangers are installed inside the exhaust nozzle to exploit the thermal energy of external exhaust gas (relative to the heat exchangers) for pre-heating the internal air (relative to the heat exchangers) which is drawn from the compressor and then forced into the combuster. As recuperative heat exchangers operate under conditions involving high temperatures and pressure, their thermal-mechanical performance is very important (Schonenborn, Ebert, Simon, & Storm, 2006). Also, the additional pressure drops and weights due to the presence of recuperators affects the potential benefit of an innovative thermodynamics cycle; therefore, a lightweight and low-loss compact heat exchanger is essential in the practical application of intercooled recuperative aero-engines (Goulas et al., 2015; Xu, Kyriyanidis, & Grönsstedt, 2013).

One of the most promising heat recuperators used in aero-engines is the double-U-shaped tube bundle heat exchanger (Min, Jeong, Ma, & Kim, 2009). Although extensive investigations have been carried out concerning the flow development and overall heat transfer through the tube bundle geometry (Horvat, Leskovar, & Mavko, 2006; Runat, Janiga, Nobile, & Dominique, 2014; F. M. Wang, Zhang, & Wang, 2007), limited work has been performed on the flow and overall heat transfer through heat exchangers embedded inside an aero-engine exhaust nozzle, where the external flow passing through the tube bundle is more complicated due to various angles of attack. In recent years, a group at the Aristotle University of Thessaloniki and MTU Aero Engines conducted a series of experimental and numerical investigations on this topic. Missirlis, Yakinthos, Palikaras, Katheder, and Goulas (2005, 2007) investigated the pressure drop through the heat exchanger. Based on the experimental data, they derived some quadratic relationships of pressure drop versus local velocity for the specific heat exchangers. Yakinthos et al. (2007) optimized the design of multiple heat exchangers in an aero-engine exhaust nozzle, finding that the arrangement of the heat exchangers is very important to the mass-flow distribution for each heat exchanger, as well as to the pressure losses inside the exhaust nozzle. Albanakis et al. (2009) experimentally studied the effects of the inflow attack angle and the heat exchanger inclination angle on the flow performance in a wind tunnel, showing that the effect of the inflow attack angle on the pressure drop is significantly greater than that of the heat exchanger inclination angle. Kritikos et al. (2010) performed a numerical investigation on the thermal performance of a heat exchanger designed for aero-engine applications. In their numerical investigation, the heat exchangers were modeled as some isotropic porous medium matrices. More recently, Vlahostergios, Missirlis, Flouros, Albanakis, and Yakinthos (2015) experimentally studied the effect of turbulence intensity on the pressure drop and heat transfer in a staggered tube bundle heat exchanger, finding that an increase in turbulent intensity is helpful for reducing the total pressure drop of the external flow and enhancing the heat transfer for the heat exchanger. Liu, Zhang, Li, and Kang (2015) carried out an experimental investigation on the flow and heat transfer performance of a double-U-shaped tube bundle heat exchanger, and the effects of the tube profile and inclination angle on the pressure drop and heat transfer effectiveness were obtained.

As the pressure drop across the recuperators inside an exhaust nozzle is tightly associated with the potential benefits of using an innovative intercooled recuperative engine, the evaluation of the pressure drop is an important aspect in the design of recuperative heat exchangers. Generally, modeling the flow inside an exhaust nozzle with multiple built-in heat exchangers using a detailed computational fluid dynamics (CFD) simulation is hampered by the huge problem that vast computational grids are needed. Therefore, a simplified approach is more realistic, such as utilizing porous medium models (Benhamadouche & Laurence, 2003). By utilizing porous medium models, the flow modeling of heat exchangers specifically used in aero-engine nozzles was investigated by Missirlis et al. (2005, 2007, 2010) and Yakinthos et al. (2007, 2012).

However, the real flow inside an exhaust nozzle with multiple built-in heat exchangers is very complicated; therefore, the utilization of a porous medium approach may be limited by several issues. Firstly, it is commonly known that the correct flow law is key to ensuring that the modeled porous medium has the same flow and thermal behavior as the original compact heat exchanger (Hayes, Khan, Shaaban, & Spearing, 2008). Most of the existing flow laws for determining the pressure–velocity relationships, the permeability and the inertia coefficient of porous matrices for different practical applications were derived from relatively idealized experimental conditions where the flow was mainly one-dimensional and the porous medium was nearly homogeneous and isotropic (Breugem, Dijk, & Delfos, 2014; Lage & Antohe, 2000; Lage, Antohe, & Nield, 1997; Naaktgeboren, Krueger, & Lage, 2012; Wilson, Narasimhan, & Venkateshan, 2004). With regard to the investigations of Missirlis et al. (2005, 2007), the U-bend matrix heat exchanger was arranged to occupy the entire cross-section of the tested wind tunnel and the elliptical tubes were closely operated as ‘flow straightening devices’ to make the flow aligned with the axial flow direction. In the real operating conditions of an intercooled recuperative engine, it can be seen that
the heat exchangers do not generally fully obstruct the flow section inside the exhaust nozzle. In this situation, the hot gas partially passes across the heat exchanger matrix, which makes a very different flow feature from the above cases. As the primary flow approaching the heat exchanger matrix has different attack angles corresponding to different ports of the heat exchanger matrix, defining the local velocity entering the heat exchanger matrix in determining the pressure–velocity relationships is challenging.

Secondly, the configuration of a real heat exchanger is generally anisotropic in its geometric features. To take anisotropic effects into account, some more advanced versions of the flow law have been presented (X. Wang, Thauvin, & Mohanty, 1999). Recently, Missirlis et al. (2010) developed a generalized porous medium model on the modeling of the flow through a compact heat exchanger by introducing a modified anisotropic formulation of the Darcy–Forchheimer pressure drop law. However, in practice, it is not easy to precisely determine the pressure drop coefficient matrix. Therefore, it is crucial to assess the reasonability of applying isotropic porous medium assumptions to the flow field modeling of the exhaust nozzle with built-in heat exchangers, either in the macroscopic behavior of the pressure drop or in detailed flow fields.

With these issues in mind, a preliminary effort is made in this paper to establish a practical engineering-based flow law as well as accurately assessing the isotropic porous medium model for flow field modeling inside a rectangular channel with a built-in U-shaped tube bundle heat exchanger. Three steps are involved in the work of the present investigation. Firstly, experiments were conducted to form an understanding of the relationship of the pressure drop and inlet velocity for a specific heat exchanger with different installation angles inside a rectangular channel. Secondly, a series of numerical computations were carried out based on the isotropic porous medium assumption. Thirdly, a three-dimensional (3-D) CFD method was adopted for the direct simulation of the flow field. A comparative discussion on the macroscopic behavior of the pressure drop and the detailed flow fields inside the channel is then presented for the porous medium model and the detailed numerical computations.

2. Experimental method and results

2.1. Experimental setup

The experimental setup is sketched schematically in Figure 1(a). It consists of three main components: the primary flow passage, the secondary flow supply passage, and the test section. The primary flow and the secondary flow are supplied by two individual compressors. Both flows are drawn through their corresponding standard flow meters and then forced into the test section.

Figures 1(b) and 1(c) show the schematic test section as well as the arrangement of the heat exchanger. The rectangular channel is 2300 mm in length, having a cross-section of 150 mm in the y-direction and 400 mm in the z-direction. The heat exchanger is set inside the rectangular channel at a certain inclination angle ($\alpha$). The inclination angles are set as 10°, 20° and 30°. In the lateral direction, the width of the heat exchanger matrix is 124 mm in order to take the gap flow in practical application into account.

The double-U-shaped tube bundle heat exchanger is adopted in the present investigation, as shown in Figure 2(a). This type of heat exchanger has two manifold tubes (namely the distributor tube and the collector tube). Between the distributor tube and the collector tube, a number of U-shaped tubes are distributed to form the core of the heat exchanger matrix. U-shaped tubes are arranged in a 4/3/4 staggered configuration, similar to that adopted by Missirlis et al. (2005, 2007). The heat exchanger matrix is 580 mm in length, 124 mm in width, and 160 mm in height. The side view of the heat exchanger is shown in Figure 2(b). According to the previous investigations (Horvat et al., 2006; Ranut et al., 2014), the elliptical-profile tubes have better comprehensive performance than the circular-profile tubes on the basis of pressure losses as well as heat transfer characteristics. Therefore, the U-shaped tubes with an elliptical profile are used in the present study. There was a total of 288 tubes arranged in 41 rows along the lateral direction (y-direction), as shown in Figure 2(c). All the tubes had the same wall thickness of 0.5 mm.

Two sets of experimental tests were made for the present investigation. Tests for determining the pressure drops inside the channel were performed under the cold condition. In these tests, the secondary flow supply and the heater for heating the primary flow were not used. The channel inlet velocity was adjusted within the range of 10 m/s to 20 m/s.

Pressure drop measurements were carried out at the inlet and outlet stations of the channel. A total pressure distribution measurement was carried out at a station 435 mm downstream of the heat exchanger matrix. The experimental measurements in this study were performed with the use of the total pressure rake. Each pressure rake has 20 pressure probes distributed at even intervals. By moving this total pressure rake along the lateral direction, the sectional area-averaged total pressures can be determined. As the channel was straight with a constant sectional area, the average velocity entering
into the channel is the same as that leaving the channel. Therefore, the total pressure drop may be approximately regarded as the static pressure drop through the heat exchanger model.

In the present tests, the standard flow meter had a nominal accuracy of 99% with reference to the current mass flow rate measurement; thus, the measured error for the inlet velocity is estimated below ±1.5% with respect to the corresponding factors (such as the air density and the inlet area). In regard to the pressure measurement, the pressure probe was pre-calibrated to have an accuracy of approximately 98.5% for the reference pressure (relative to the ambient pressure). Thus, the pressure drop is estimated below ±3% for the current tested range.

In addition, heat transfer tests were also carried out to determine the heat transfer effectiveness of the heat exchangers. In these tests, the mass flow rate of the primary flow was fixed at 0.29 kg/s and the temperature was about 760 K. The mass flow rate of the secondary flow was adjusted within the range 0.02 kg/s to 0.06 kg/s. The temperature of the secondary flow is about 293 K.

The heat transfer effectiveness of a heat exchanger is defined as

$$\eta = \frac{T_{S,\text{outlet}} - T_{S,\text{inlet}}}{T_{P,\text{inlet}} - T_{S,\text{inlet}}}$$

where the subscripts P and S represent the primary flow (or external flow) and secondary flow (or internal flow), respectively.

The inflow temperature ($T_{S,\text{inlet}}$) and outflow temperature ($T_{S,\text{outlet}}$) of the secondary flow were measured.
using two temperature probes located at the distributor inlet and collector outlet, respectively. The inflow temperature ($T_{P, \text{inlet}}$) of the primary flow was measured with a temperature rake with 20 temperature probes distributed at even intervals. To estimate the thermal balance of a heat exchanger, the outflow temperature ($T_{P, \text{outlet}}$) of the primary flow is also measured by a temperature rake.

According to the heat transfer theory, the heat flux released by the primary flow and the heat flux accepted by the secondary flow are calculated by Equations (2) and (3), respectively:

$$Q_P = (mc_p)P(T_{P, \text{inlet}} - T_{P, \text{outlet}})$$  \hspace{1cm} (2)  

$$Q_S = (mc_p)S(T_{S, \text{outlet}} - T_{S, \text{inlet}})$$  \hspace{1cm} (3)  

where $m$ is the mass flow rate and $c_p$ is the specific heat.

In an ideal situation, the heat flux released by the primary flow would be balanced by the heat flux accepted by the secondary flow. However, in the real tests, the heat

**Figure 2.** Double-U-shaped tube bundle heat exchanger model: (a) the experimental model, (b) a side view schematic, and (c) the arrangement of the elliptical tubes.
losses from the primary flow channel are not eliminated completely. Here, the relative error is defined as

$$\sigma = \frac{Q_P - Q_S}{Q_P}$$  \hspace{1cm} (4)

From the tested data, it is estimated that the non-balance error is within 3.5% in the presented test range.

### 2.2. Test results

Figure 3(a) presents the varying tendencies of the pressure drop inside the rectangular channel versus the mean inlet velocity for different specific inclination angles. It can be seen that the relationship of the pressure drop versus the channel inlet flow velocity takes on a quadratic form. In general, the total frontal area of the heat exchanger is enlarged as the heat exchanger inclination angle increases. Therefore, the blockage effect of the heat exchanger matrix on the channel flow is more significant, resulting in a greater pressure drop or a larger pressure drop variation as the slope varies with the inlet velocity.

Some actual relationships reflecting the pressure drop versus the inlet velocity within a rectangular channel for elliptical-profile tubes in the heat exchanger are directly derived from the present experimental data, as illustrated by the following:

$$\Delta p = 0.9338 U_{P, \text{inlet}}^2 + 0.987 U_{P, \text{inlet}} (\alpha = 10^\circ)$$  \hspace{1cm} (5)

$$\Delta p = 1.388 U_{P, \text{inlet}}^2 + 1.64 U_{P, \text{inlet}} (\alpha = 20^\circ)$$  \hspace{1cm} (6)

$$\Delta p = 1.827 U_{P, \text{inlet}}^2 + 1.859 U_{P, \text{inlet}} (\alpha = 30^\circ)$$  \hspace{1cm} (7)

where $U_{P, \text{inlet}}$ is the inlet velocity of primary flow.

By comparison with the investigation on elliptical tubes conducted by Missirlis et al. (2005), it is found that the presented pressure drop or the varying gradient of the pressure drop versus the flow velocity is somewhat less than that presented in Missirlis et al, in which the U-bend matrix heat exchanger occupies the entire cross-section of the tested wind tunnel and the elliptical tubes are closely operated as ‘flow-straightening devices’. According to Missirlis et al., when the average streamwise airflow velocity is increased from 3.18 m/s to 10.85 m/s, the static pressure drop varies from 249.8 Pa to 2303.3 Pa. However, in the present study, when the inlet velocity is increased from 10 m/s to 18 m/s, the pressure drop varies from 110 Pa to 345 Pa for an inclination angle of $10^\circ$, 165 Pa to 500 Pa for an inclination angle of $20^\circ$, and 210 Pa to 650 Pa for an inclination angle of $30^\circ$. As the heat exchanger only partially occupied the channel cross-section, the blockage effect of the heat exchanger matrix was weaker relative to the situation where the channel flow passes entirely through the matrix heat exchanger.
also be seen that the heat transfer effectiveness significantly increases with an increase in the heat exchanger inclination angle for a specific $(\rho U)_{p, \text{inlet}}/((\rho U)_s)_{\text{tube}}$. For example, the heat transfer effectiveness of a heat exchanger with a 30° is improved by about 40 to 50% relative to a heat exchanger with a 10° inclination. Apparently, the improvement in heat transfer effectiveness due to the enlargement of the heat exchanger inclination contributes to an external convective heat transfer enhancement.

3. Computational procedures

3.1. Computation based on the isotropic porous medium assumption

In the present study, the heat exchanger matrices are modeled based on the assumption that they are an isotropic porous medium. The flow in the computational domain is considered to be three-dimensional incompressible turbulence for the clear fluid and porous medium zones. For the clear fluid zone, the momentum transport is modeled with a Navier–Stokes equation, as shown in Equation (8). For the porous medium zone, the extended Darcy model is adopted (Yang, Zeng, Wang, & Nakayama, 2010), as shown in Equation (9):

$$ V \cdot \nabla V = -\frac{1}{\rho} \nabla p + (\nu + U_t) \nabla^2 V \tag{8} $$

$$ \frac{1}{\varphi^2} (V \cdot \nabla) V = -\frac{1}{\rho} \nabla p + \frac{\nu + U_t}{\varphi} \nabla^2 V - S \tag{9} $$

where $V$ is the velocity vector, $\rho$ and $\nu$ are the fluid density and kinematic viscosity, respectively, $\varphi$ is the porosity, $\nu_t$ is the turbulent kinematic viscosity, and $S$ is an additional source term in the momentum equation which is defined as:

$$ S = \frac{\nu}{K} V + C|V|V \tag{10} $$

where $K$ is the specific permeability and $C$ is the inertia coefficient of the porous medium.

The porosity of a $U$-shaped tube bundle heat exchanger is determined by $\varphi = 1 - \text{Vol}_{\text{tubes}}/\text{Vol}_{\text{matrix}}$, where $\text{Vol}_{\text{tubes}}$ represents the total volume of the tubes and $\text{Vol}_{\text{matrix}}$ represents the volume of the heat exchanger matrix. For the presented heat exchanger configuration, the porosity is about 0.53. The permeability ($K$) and the inertia coefficient ($C$) in the porous medium model are determined according to the specific relationship between the pressure drop and the effective velocity, which is discussed in the next section.

In the computation based on an isotropic porous medium assumption, the computational domain is subdivided into a number of zones, such as the manifold tube solid zones, the heat exchanger matrix porous zones, the upstream clear fluid zone, the downstream clear fluid zone, and the clear fluid zone around the exchanger matrix. Structured grids are constructed in the heat exchanger matrix and in the upstream and downstream zones of the heat exchanger, while unstructured grids are constructed in the computational zone around the heat exchanger. These two meshes are merged together to form a ‘hybrid’ mesh with a non-conformal interface boundary between them. Figure 4(a) shows local grids in the vicinity of the heat exchanger for the computation based on a porous medium assumption. In the current computation, grid sensitivity tests were pre-conducted by applying the grid convergence index (GCI) method (Celik et al., 2008). For this purpose, three numbers were adopted for coarse (600,000 points), fine (900,000 points) and highly fine (1,200,000 points) grids. The corresponding $y^+$ value is 2.5, 1.5 and 0.8, respectively, generally fulfilling the viscous clustering requirement. The discretization error is within 4% for the pressure drop, therefore a grid system with 900,000 points was chosen for the computation based on a porous medium assumption.

In the computation, the boundary conditions of the computational domain were specified as follows. The flow inlet of the wind tunnel was set as the velocity inlet by giving the flow velocity. A turbulence intensity of 5% was used, which is determined from an empirical formula of turbulence intensity $- I \equiv u'/u_{\text{avg}} \approx 0.16 \text{ Re}^{-1/8}$, where $u'$ is the fluctuating velocity, $u_{\text{avg}}$ is the time-averaged velocity, and $\text{Re}$ is the Reynolds number – as illustrated in the Fluent 6.3 User’s Guide (Fluent, 2006). The flow outlet condition is set as the pressure outlet with a static reference 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 constants with a density of 1.1968 kg/m$^3$ and a dynamic viscosity of $18.22 \times 10^{-6}$ Pa.s.

The flow field was computed using the commercial CFD solver Fluent v6.3. It is well known that accurately modeling turbulent flow is a very complex process. Although the two-equation turbulence models have some drawbacks relative to the more advanced turbulence models (such as the $v^2f$ normal velocity relaxation model and the Direct Numerical Simulation/Large Eddy Simulation time-variant models), they are still recommended as useful turbulence models due to their advantage of obtaining a sufficient degree of accuracy at a low cost (Galván, Reggio, & Guibault, 2011). In the present investigation, four two-equation turbulence models comprising standard $k-\varepsilon$, realizable $k-\varepsilon$, RNG (Renormalization Group) $k-\varepsilon$, and SST (Shear-Stress Transport) $k-\omega$ were selected for comparison. A detailed comparison for the computational results corresponding...
to different two-equation turbulence models is presented in the next section. As a result of the comparison, the SST $k-\omega$ turbulence model developed by Menter (1994) was chosen to model the turbulence for the computations in this study, and the flow near-wall region was modeled using enhanced wall functions.

In the current computations, a SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm was used to couple pressure and velocity. Convergence is achieved when the reduction in all residuals of five orders of magnitude has been met. Generally, the computation time required for each case is approximately 20 hours using a PC with a 3.40 GHz Intel core processor.

3.2. Computation based on a 3-D CFD approach

A 3-D detailed flow field simulation was also carried out using CFD directly. For this computation, very fine grids with unstructured meshes are needed. From mesh-independent tests, approximately 8,750,000 points with a maximum $y^+$ of 1.8 were adopted. Figure 4(b) shows the meshing situation on a typical plane crossing the tube bundles. Unstructured grids were constructed in the characteristic flow passages between elliptical tubes, and structured grids were constructed in the upstream and downstream regions of the heat exchanger. These two meshes were merged together to form a 'hybrid' mesh, with a non-conformal interface boundary. The grid generation was realized using Gambit.

The computations were also performed using Fluent v6.3. The boundary conditions of the computational domain were same as those mentioned above. The SST $k-\omega$ turbulence model was selected and convergence is achieved when the reduction in all residuals of five orders of magnitude is met. Generally, the computation time required for each case is
approximately 120 hours on the same PC as mentioned above.

4. Computational results and discussion

4.1. Validation of the isotropic porous medium model

As illustrated in previous work, an accepted practice for determining the permeability and the inertia coefficient of a homogeneous isotropic porous medium is to use the Hazen–Dupuit–Darcy model, as given expressed in Wilson et al. (2004) as:

\[ \frac{\Delta p}{l} = \frac{\mu}{K} U + C\rho U^2 \]  

(11)

or

\[ \Delta p = a_1 U + a_2 U^2 \]  

(12)

where \( \mu \) is the fluid dynamic viscosity, \( a_1 \) and \( a_2 \) are the first- and second-order coefficients in the pressure drop formula, respectively, and \( l \) is the characteristic length of the porous medium.

This treatment seems very simple. However, it is extremely realistic for the simple flow passing through a homogeneous isotropic porous medium with a primarily one-dimensional flow. Looking at Figure 1, it can be seen that the flow field inside a rectangular channel with a built-in U-shaped tube bundle heat exchanger is different from the simple flow passing through a homogeneous isotropic porous medium. As the heat exchanger matrix is partially installed inside a channel, the local airflow velocity and attack angle approaching the front of the heat exchanger matrix are different, corresponding to different ports of the heat exchangers. Furthermore, the flow inside the heat exchanger matrix obviously behaves in two-dimensional (2-D) or even 3-D ways. In this situation, the utilization of an isotropic porous medium model undoubtedly involves some simplifications.

With regard to the actual pressure drop, given the laws derived from the experimental data – as indicated in Equations (5) to (7) – it is the case that the overall pressure drops inside the channel come from the pressure loss passing through the porous matrix, the pressure loss passing through the gap between the heat transfer matrix and the primary flow channel, and the pressure loss due to the frictional stress on the channel walls. Unfortunately, it is tremendously difficult to determine these individual pressure losses. In the current investigation, a simplification was made by neglecting the other pressure losses or treating the pressure drop as being entirely attributed to the porous matrix. For this reason, the pressure–velocity relationships (Equations (5) to (7)) applied in the use of an isotropic porous medium model are referred to as the ‘original pressure–velocity relationships’.

On the basis of these original pressure–velocity relationships, the permeability and the inertia coefficient for an isotropic porous medium can be approximately derived according to Equations (11) and (12). In this process, two questions are encountered: the definition of the airflow velocity approaching the porous matrix and the definition of the characteristic length of the porous matrix. Due to the partial obstruction of the heat exchanger matrix inside the channel, the local airflow velocity will be different for different parts of the heat exchanger matrix. For convenience, the channel inlet velocity was selected as the mean airflow velocity entering the heat exchanger matrix and the heat exchanger thickness (about 140 mm) was chosen as the characteristic length.

The permeability and the inertia coefficient used in a porous medium model together with the original pressure–velocity relationships for different heat exchanger inclinations are summarized in Table 1. As these values are deduced from the original pressure–velocity relationships with some simplified treatments, they cannot represent the native properties of a heat exchanger matrix, but can be regarded as nominal properties of a heat exchanger matrix just for use in the computations. From Table 1, it can be seen that the permeability decreases and the inertia coefficient increases as the heat exchanger matrix inclination angle increases.

Based on the above simplifications, some computations were carried out for the comparison of the chosen four two-equation turbulence models: the standard \( k-\varepsilon \), realizable \( k-\varepsilon \), RNG \( k-\varepsilon \), and SST \( k-\omega \) models. Figure 5(a) presents the total pressure distribution on the symmetrical line of a sectional plane located at \( x = 725 \) mm or 435 mm downstream of the heat exchanger matrix. Here, the primary inlet velocity is 13.9 m/s and the inclination angle of the heat exchanger matrix is 30°, while the dashed lines mark the positional borders of the heat exchanger unit. It can be seen that the total pressure computed by the isotropic porous model together with the original pressure–velocity relationship is much lower than that of the experimental data among the range bounded by the dashed lines. It can also be seen that no major differences are found in the pressure drop simulations between these turbulence models. It is

| Inclination angle \( \alpha \) | Permeability \( K \) | Inertiacoefficient \( C \) |
|-----------------------------|---------------------|-------------------|
| 10°                         | \( 2.58 \times 10^{-6} \) | 11.1              |
| 20°                         | \( 1.79 \times 10^{-6} \) | 16.7              |
| 30°                         | \( 1.37 \times 10^{-6} \) | 21.8              |
Figure 5. Comparisons of the flow field simulations of the four turbulence models: (a) the total pressure distribution at \( x = 725 \text{ mm} \) and \( z = 0 \text{ mm} \) and (b) the streamwise velocity distribution at section of \( x = 725 \text{ mm} \).

Suggested that the reasons for such a significant deviation are primarily due to the assumption of an isotropic porous medium, as well as the simple treatments on the use of an isotropic porous medium model. Figure 5(b) shows the streamwise velocity distributions at section \( x = 725 \text{ mm} \). It can be seen that the streamwise velocity in the low-velocity zone computed with the SST \( k-\omega \) turbulence model is slightly higher than the corresponding values computed with the other turbulence models. Hence, the SST \( k-\omega \) turbulence model was selected for the final computations.

Figure 6 presents the comparison of the computed pressure drops and the measured pressure drops for different inclination angles of the elliptical-tube heat exchangers, with the computational results obtained by the isotropic porous medium model together with the pressure–velocity relationships (Equations (5) to (7)) marked as the original pressure–velocity relationships. Additionally, the computational results based on 3-D detailed computations are marked as ‘3-D CFD’.

In the computations based on the isotropic porous medium model together with the original pressure–velocity relationships, the permeability and the inertia coefficient are derived from Equations (5) to (7). As mentioned above, the pressure drops considered in the original pressure–velocity relationships are the overall pressure drops inside the channel. Consequently, the isotropic porous medium model together with the original pressure–velocity relationships will overestimate the pressure drop inside the channel because it is responsible for enlarging the flow drags passing through the heat exchanger matrix in the computation.
It can be seen from Figure 6(a) that when the heat exchanger matrix inclination angle is 10° and the isotropic porous medium model is used in combination with the original pressure–velocity relationship, the computed pressure drops are higher than the corresponding experimental values. For example, under an inlet flow velocity of 18.4 m/s, the difference between the computed pressure drop and the experimental value is about 109 Pa, giving a relative error of around 32%. As the inclination angle of the heat exchanger increases, the computed pressure drops seems fit well with the experimental values, as can be seen in Figures 6(b) and 6(c). When the heat exchanger matrix inclination angle is 20°, the computed pressure drop corresponding to an inlet flow velocity of 18.4 m/s is approximately 13% higher than the experimental value, and when the heat exchanger matrix inclination angle is 30°, the computed pressure drop corresponding to an inlet flow velocity of 18.4 m/s is approximately 10% higher than the experimental value.

Considering that the derivation of the permeability and internal resistance coefficient from the original pressure–velocity relationship is based on the overall pressure drop but not the ‘real’ drop due to the flow passing through the heat exchanger matrix, a modification relative to the original pressure–velocity relationship could be made by introducing a correction coefficient. According to the fact that the relative difference between the computed pressure drop using the isotropic porous medium model in combination with the original pressure–velocity relationship and the experimental pressure drop decreases as the heat exchanger matrix inclination angle increases, some correction coefficients were introduced to the original pressure–velocity relationships as follows:

\[
\Delta p = 0.6 \times (0.9338 U_{\text{inlet}}^2 + 0.987 U_{\text{inlet}}) \quad (\alpha = 10^\circ)
\]

\[
\Delta p = 0.85 \times (1.388 U_{\text{inlet}}^2 + 1.64 U_{\text{inlet}}) \quad (\alpha = 20^\circ)
\]

\[
\Delta p = 0.95 \times (1.827 U_{\text{inlet}}^2 + 1.859 U_{\text{inlet}}) \quad (\alpha = 30^\circ)
\]

Equations (13) to (15) are thus referred to as the ‘modified pressure–velocity relationships’. It can be seen that the correction coefficient for the heat exchanger with an inclination angle of 10° is much less than that for an inclination angle of 30°. This treatment accords with the physical features in a sense; when the heat exchanger matrix inclination angle is 10°, the additional flow losses except for the flow loss passing through the porous matrix are mostly contributed by the pressure drop inside the channel, because less channel flow is forced to pass through
the heat exchanger matrix relative to the case where the inclination angle is greater. This could explain the tendency for the relative difference between the computed and experimental pressure drops to decrease as the heat exchanger matrix inclination angle increases. By adopting the modified pressure–velocity relationships, it can be seen in Figure 6 that the numerical computation based on the isotropic porous medium assumption is capable of making precise pressure drop predictions.

It can also be seen in Figure 6 that the detailed numerical computation gives better flow modeling. In most of the computational cases, the relative difference between the detailed numerical computation and the experimental data is within 10%.

Figure 7 presents the total pressure distribution at \( x = 725 \text{ mm} \) or at 435 mm downstream of the elliptical-tube heat exchanger matrix and \( z = 0 \text{ mm} \), where the dashed lines mark the positional borders of the heat exchanger unit. It can be seen that the total pressure distribution computed by the isotropic porous model (either together with the original pressure–velocity relationship or the modified pressure–velocity relationship) is relatively uniform in correspondence to the heat exchanger matrix zone. In regard to the experimental data, the total pressure increases rapidly from the location of \( y = -50 \text{ mm} \) to \( y = 50 \text{ mm} \) for an inclination angle of \( 10^\circ \), \( y = -100 \text{ mm} \) to \( y = 100 \text{ mm} \) for an inclination angle of \( 20^\circ \), and \( y = -150 \text{ mm} \) to \( y = 0 \text{ mm} \) for an inclination angle of \( 30^\circ \). This total pressure distribution feature is well captured by the 3-D CFD computation. The reason for the isotropic porous model causing such deviation from the experimental results is hard to explain, but is perhaps due to the strong anisotropic effect of the heat exchanger matrix in such applications.

### 4.2. Comprehensive discussion on the detailed flow field

In this section, a comprehensive discussion on the detailed flow field is presented. Here the computations based on the isotropic porous medium model are performed by using the modified pressure–velocity relationships, and the flow inlet velocity is fixed at 14 m/s.

Figure 8 shows the local flow trace passing through the heat exchanger matrix at the central \( x-y \) plane. It can clearly be seen that the flow trace is very complicated where the heat exchanger matrix is partially installed inside a channel. Also, it is notable that the detailed flow trace passing through the heat exchanger matrix simulated by the ‘porous medium’ assumption is somewhat different from that generated by the 3-D CFD approach. With the isotropic porous medium assumption, the wake...
modified pressure-velocity relationship

(a) $\alpha = 10^\circ$

modified pressure-velocity relationship

3-D CFD

(b) $\alpha = 20^\circ$

modified pressure-velocity relationship

3-D CFD

(c) $\alpha = 30^\circ$

3-D CFD

Figure 8. Flow trace passing through the heat exchanger matrix at the central $x-y$ plane for (a) $\alpha = 10^\circ$, (b) $\alpha = 20^\circ$, and (c) $\alpha = 30^\circ$.

Figure 9. Detailed flow fields in the $x-z$ plane for $y = 0$ and $\alpha = 10^\circ$ simulated using 3-D CFD.

recirculation induced by the flow passing through the distributor tube is obviously stronger than that simulated by the detailed numerical computation in the case of a heat exchanger matrix inclination angle of 10° or 20°. The local flow trace is altered by using the isotropic porous medium assumption in comparison with the detailed numerical computation. It seems that the flow inside the inner region of the heat exchanger matrix does not flow out from the upper-rear U-shaped tube bundle easily, resulting in a larger wake recirculation and a low-velocity region. This alteration of the flow trace is perhaps associated with the presented quadratic relationships, which do not consider the anisotropic effect inside the porous matrix.
Figure 9 shows the detailed flow fields in the $x$–$z$ plane where $y = 0$ mm with an elliptical-tube heat exchanger matrix inclination angle of $10^\circ$. For the front U-bend tubes, the flow entering into the staggered tubes seems like a parallel flow in the middle part. The flow velocity inside the gap between the heat exchanger matrix and the side wall is obviously higher than that passing through the heat exchanger due to a larger flow drag in the heat exchanger matrix. The flow attack angle is deflected outside the side wall for the outer lines of the tubes by comparison with the middle part. After passing through the front U-bend tubes, the flow is concentrated in the inner region. As the collecting tubes of the heat exchanger cause the flow blockage, a local region with a higher flow velocity occurs, while for the rear U-bend tubes, the flow enters into the staggered tubes with different flow attack angles corresponding to the different parts. At the rear edge of the heat exchanger, there is a recirculation zone which is similar to the backward-facing step flow. Figure 9 illustrates that the flow passing through the heat exchanger matrix is complicated when the heat exchanger does not obstruct fully the flow section inside the channel.

Based on this preliminary research, it is concluded that the isotropic porous medium assumption is capable of making precise pressure drop predictions given the reasonable pressure–velocity relationship. However, it is also suggested that the isotropic porous medium model is unable to precisely simulate the detailed flow features in a rectangular channel with a built-in double-U-shaped bundle heat exchanger.

5. Conclusions

Experimental tests were conducted to investigate the pressure drop inside a rectangular channel with a built-in double-U-shaped elliptical-tube bundle heat exchanger. The influence of the heat exchanger inclination angle on the pressure drop and heat transfer effectiveness has been illustrated. As the inclination angle of the heat exchanger increases, the blockage effect of the heat exchanger matrix on the channel flow becomes more significant, resulting in a greater pressure drop and a larger pressure drop variation of slope depending on the inlet velocity. The heat transfer effectiveness of a heat exchanger with an inclination angle of $30^\circ$ is improved about 40 to 50% relative to the heat exchanger with an inclination angle of $10^\circ$.

In the flow simulation inside a rectangular channel with a built-in double-U-shaped tube bundle heat exchanger which uses the isotropic porous medium assumption, a preliminary effort was made by introducing some simple treatments to the flow law for modeling the pressure drop inside the heat exchanger matrix, such as selecting the channel inlet velocity as the mean airflow velocity entering the heat exchanger matrix, choosing the heat exchanger thickness as the characteristic length for derivation of the permeability and the inertia coefficient, and modifying the pressure–velocity relationship by introducing an empirical correction coefficient. Although these simplifications are not theoretically strict, the effort is somewhat acceptable in the sense of providing a computationally affordable solution in terms of engineering applications. Indeed, it has been proved that numerical computation using the isotropic porous medium assumption is capable of making precise pressure drop predictions given the reasonable pressure–velocity relationship. However, it is also suggested that the isotropic porous medium model is unable to precisely simulate the detailed flow features in a rectangular channel with a built-in double-U-shaped bundle heat exchanger. For all the examined cases, the computational results based on the 3-D CFD approach are in close agreement with the experimental data, both for the pressure drop and the total pressure distribution downstream of the heat exchanger matrix.

In the current investigation, the specific permeability and the inertia coefficient in the porous medium are assumed to be isotropic. These assumptions are not true for ensuring that the modeled porous medium has the same flow behavior as the original compact heat exchanger due to the complicated 3-D flow feature. Thus, a more advanced anisotropic porous medium model needs to be developed for more accurately modeling this specific heat exchanger application.

Disclosure statement

No potential conflict of interest was reported by the author.

Funding

This work was supported by the Fundamental Research Funds for the Central Universities [grant no. 3082015NP2015204].

References

Albanakis, C., Yakinthos, K., Kritikos, K., Missirlis, D., Goulas, A., & Storm, P. (2009). The effect of heat transfer on the pressure drop through a heat exchanger for aero engine applications. Applied Thermal Engineering, 29(4), 634–644.

Benhamadouche, S., & Laurence, D. (2003). Les, coarse LES, and transient RANS comparisons on the flow across a tube bundle. International Journal of Heat and Fluid Flow, 24(4), 470–479.

Breugem, W. P., Dijk, V., & Delfos, R. (2014). Flows through real porous media: X-Ray computed tomography. Experiments,
and Numerical Simulations. *Journal of Fluids Engineering*, 136(4), 040902.

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H., & Raad P. E. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering*, 130, 078001.

Fluent Inc. (2006). User’s Guide 6.3, Fluent Inc.

Galván, S., Reggio, M., & Guibault, F. (2011). Assessment study of $k$-$\varepsilon$ turbulence models and near-wall modeling for steady state swirling flow analysis in draft tube using Fluent. *Engineering Applications of Computational Fluid Mechanics*, 5(4), 459–478.

Goulas, A., Donnerhack, S., Florous, M., Missirlis, D., Vlahostergios, Z., & Yakinthos, K. (2015). Thermodynamics cycle analysis, pressure loss, and heat transfer assessment of a recuperative system for aero-engines. *Journal of Engineering for Gas Turbines and Power*, 137(4), 041205.

Hayes, A. M., Khan, J. A., Shaaban, A. H., & Spearing, I. G. (2008). The thermal modeling of a matrix heat exchanger using a porous medium and the thermal Non-equilibrium model. *Experimental Thermal and Fluid Science*, 47(10), 1306–1315.

Horvat, A., Leskova, M., & Mavko, B. (2006). Comparison of heat transfer conditions in tube bundle cross-flow for different tube shapes. *International Journal of Heat and Mass Transfer*, 49(5–6), 1027–1038.

Kritikos, K., Albanakis, C., Missirlis, D., Vlahostergios, Z., Goulas, A., & Storm, P. (2010). Investigation of the thermal efficiency of a staggered elliptic-tube heat exchanger for Aeroengine applications. *Applied Thermal Engineering*, 30(2–3), 134–142.

Kyprianidis, K. G., Grönstedt, T., Ogaji, S. O. T., Pilidis, P., & Singh, R. (2011). Assessment of future aero-engine designs with intercooled and intercooled recuperated cores. *Journal of Engineering for Gas Turbines and Power*, 133(1), 011701.

Lage, J. L., & Antohe, B. V. (2000). Darcy’s experiments and the deviation to nonlinear flow regime. *Journal of Fluids Engineering*, 122(3), 619–625.

Lage, J. L., Antohe, B. V., & Nield, D. A. (1997). Two types of nonlinear pressure-drop versus flow-rate relation observed for saturated porous media. *Journal of Fluids Engineering*, 119(3), 700–706.

Liu, X. Y., Zhang, J. Z., Li, G. T., & Kang, Y. (2015). Investigation on flow and heat transfer performance of double U-shaped-tubes modeled heat exchanger. *Acta Aeronautica et Astronautica Sinica*, 36(12), 3832–3842.

Menter, F. R. (1994). Two-Equation Eddy viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598–1605.

Min, J. K., Jeong, J. H., Ma, M. Y., & Kim, K. S. (2009). High temperature heat exchanger studies for applications to gas turbines. *Heat and Mass Transfer*, 46(2), 175–186.

Missirlis, D., Donnerhack, S., Seite, O., Albanakis, C., Sideridis, A., Yakinthos, K., & Goulas, A. (2010). Numerical development of a heat transfer and pressure drop porosity model for a heat exchanger for aero engine applications. *Applied Thermal Engineering*, 30(11-12), 1341–1350.

Missirlis, D., Yakinthos, K., Palikaras, A., Katheder, K., & Goulas, A. (2005). Experimental and numerical investigation of the flow field through a heat exchanger for aero engine applications. *International Journal of Heat and Fluid Flow*, 26(3), 440–458.

Missirlis, D., Yakinthos, K., Storm, P., & Goulas, A. (2007). Modeling pressure drop of inclined flow through a heat exchanger for aero-engine applications. *International Journal of Heat and Fluid Flow*, 28(3), 512–515.

Naaktgeburnen, C., Krueger, P. S., & Lage, J. L. (2012). Inlet and outlet pressure-drop effects on the determination of permeability and form coefficient of a porous medium. *Journal of Fluids Engineering*, 134(5), 051209.

Noppel, F., Lucisano, D., & Singh, R. (2009). Performance of the clean exhaust engine concept. *Journal of Engineering for Gas Turbines and Power*, 131(3), 031201.

Ranut, P., Janiga, G., Nobile, E., & Dominique, T. (2014). Multi-Objective shape optimization of a tube bundle in cross-flow. *International Journal of Heat and Mass Transfer*, 68, 585–598.

Schonenborn, H., Ebert, E., Simon, B., & Storm, P. (2006). Thermomechanical design of a heat exchanger for a recuperative aero engine. *Journal of Engineering for Gas Turbines and Power*, 128(4), 736–744.

Vlahostergios, Z., Missirlis, D., Florous, M., Albanakis, C., & Yakinthos, K. (2015). Effect of turbulence intensity on the pressure drop and heat transfer in a staggered tube bundle heat exchanger. *Experimental Thermal and Fluid Science*, 60, 75–82.

Wang, F. M., Zhang, J. Z., & Wang, S. F. (2007). Study of flow characteristics inside rectangular channel with different Fin fins. *Acta Aeronautica et Astronautica Sinica*, 28(1), 37–41.

Wang, X., Thauvin, F., & Mohanty, K. K. (1999). Non-Darcy flow through anisotropic porous media. *Chemical Engineering Science*, 54(12), 1859–1869.

Wilson, L., Narasimhan, A., & Venkateshan, S. P. (2004). Turbulent flow hydrodynamic experiments in near-compact heat exchanger models with aligned tubes. *Journal of Fluids Engineering*, 126(6), 990–996.

Xu, L., Kyprianidis, K., & Grönstedt, T. (2013). Optimization study of an intercooled recuperated aero-engine. *Journal of Propulsion and Power*, 29(2), 424–432.

Yakinthos, K., Missirlis, D., Palikaras, A., Storm, P., Simon, B., & Goulas, A. (2007). Optimization of the design of recuperative heat exchangers in the exhaust nozzle of an aero engine. *Applied Mathematical Modelling*, 31(11), 2524–2541.

Yakinthos, K., Missirlis, D., Sideridis, A., Vlahostergios, Z., Seite, O., & Goulas, A. (2012). Modelling operation of system of recuperative heat exchangers for aero engine with combined use of porosity model and thermo-mechanical model. *Engineering Applications of Computational Fluid Mechanics*, 6(4), 608–621.

Yang, J., Zeng, M., Wang, Q. W., & Nakayama, A. (2010). Forced convection heat transfer enhancement by porous pin fins in rectangular channels. *Journal of Heat Transfer*, 132(5), 1–8.