NUMERICAL INVESTIGATION OF TURBULENT HEAT TRANSFER IN A SUDDEN-EXPANSION FLUID FLOW

MOHSEN MOTAHARINEZHAD¹a, SAEID SAHRAKI¹b, REZA TIRGARc, HOSEIN BISADId

¹a¹b¹School of Automotive Engineering, Iran University of Science and Technology (IUST), Tehran, Iran
¹c¹National Iranian Gas Company, Tehran, Iran

ABSTRACT
In this study the heat transfer to the fluid flow with sudden expansion in an annular passage with numerical simulation is investigated and validated. Separation, backward facing step flow, and fully developed flow take place at constant heat flux. We have considered the separation and reattachment of air flow through a sudden expansion in an annular passage. Wall of test section considered to be uniformly heated from the beginning of the expansion. The effects of variation of Reynolds number of inlet air flow from 17050 until 44545 and heat flux from 719 w/m² until 2098 w/m² on heat transfer rate are investigated. We find with numerical simulation that the reattachment point extends further with increase in velocity and decrease in heat flux. The reattachment point occurs at the maximum of local heat transfer. The results have been validated by experimental data.

KEYWORDS: Heat Transfer, Numerical Simulation, CFD, Turbulence

The study of flow separation from the surface of a solid body, and the determination of global changes in the flow field that develop as a result of the separation, are among the most fundamental and difficult problems of fluid dynamics. Flow separation occurs when the boundary layer travels fast enough against an adverse pressure gradient that the speed of the boundary layer relative to the object falls almost to zero (Anderson, 2004; Tocher, 2013). A necessary condition for flow separation is the adverse pressure gradient. Generally, there are two types of flow separation, separation at external and that at internal flow (Chang, 1970). Separation fluid flows are extensively used in industrial applications even though there is still a lack of knowledge about information of the flow around the recirculation zone (Ashurst et al., 1980). Boundary layer separation occurs when the portion of the boundary layer closest to the wall or leading edge reverses in flow direction. The separation point is defined as the point between the forward and backward flow, where the shear stress is zero. The overall boundary layer initially thickens suddenly at the separation point and is then forced off the surface by the reversed flow at its bottom (Wilcox, 2007). The separation reattached flows are characterized by the interaction between vortices and the solid surface. Meanwhile, the separated flow without reattachment is characterized by the interaction between vortices shed from the separation points (Kiya et al., 1975). This phenomenon is relevant to some engineering applications, such as flow over airfoils at large angles of attack, in channels where area suddenly increases, in wide angle diffusers (Goldstein et al., 1970; Paulo J, 2003). Study of separation and reattachment flow was conducted first in the late 1950s. With the development of advanced instrumentations and numerical codes, the investigations are more facilitated to study complex three dimensional flows in the recirculation area. Guo et al (2001) numerically solved the unsteady turbulent flow behind the axi-symmetric sudden expansion.

M.P. Escudier et al (2002) have studied experimentally turbulent flow through a plane sudden expansion. In agreement with previous studies, the flow pattern downstream of the expansion was found to be asymmetric about the XZ center plane. They showed that the maximum axial turbulence intensity occurs in the upper recirculation region with values as high as 26% at the bulk velocity at inlet while the maximum transverse intensity at this location was only 14%.

Paulo and Oliveira (2003) has studied the asymmetric flows of viscoelastic fluids in symmetric planar expansion geometries. He has investigated experimentally the local and average heat transfer to a simultaneously developing air flow in a horizontal, inclined, and vertical concentric cylindrical annulus. Numerical study of self-induced transonic flow oscillations behind a sudden duct enlargement have been done by Thomas Emmert et al., (2009). They studied the transonic flow passing a sudden enlargement of cross section by using large-eddy simulations based on an explicit relaxation filtering. A direct numerical simulation has been performed to study wall-driven flow over a backward-facing step at a relatively low Reynolds number based on the step height and the upper-wall velocity by George et al., (2010). In the recirculation zone a large negative skin friction coefficient was observed beneath the core of primary separation bubble by them. The effect of step height on heat transfer to a radially outward expanded air flow stream in a concentric annular passage was studied experimentally by S.N. Kazi et al., (2011). They have found that an increase in the local heat transfer coefficient was obtained against heat flux. Also the effect of step variation is prominent on heat transfer at the separation region which increases with the rise of step. Hussein Togun et

¹Corresponding author
al., have reviewed the experimental studies of turbulent heat transfer in separated flows (2011). A Lagrangian-Eulerian approach for the dispersion of solid particles in a suddenly expanded fluid flow have been reported by Mergheni et al., (2012). At their study the fluid was investigated based on the Eulerian method. Also, a Lagrangian approach was also applied to calculate the particle trajectories. A numerical simulation of heat transfer to separation fluid flow has been done by C.S. Oon et al. (2012). They used the standard k-□ turbulent model for simulation. In this study, the effect of flow separation on heat transfer process in a flow with turbulent separation in two circular pipe has been investigated numerically by SST k-omega model. This model uses the k-w model near the wall and the k-□ model for the remaining sections. The effects of changing in flow speed at the inlet section and changing in the heat flux on heat transfer at separation region have been investigated. The results show that the SST k-omega model is accurate enough to model the heat transfer to the turbulent separation fluid flow.

TEST SECTION

The test section was designed to have an axisymmetrical separated and reattached air flow in an annular passage. The schematic offest section is shown in Figure 1. Test section includes of two horizontal circular pipes. Horizontal inner pipe has been made of aluminum with constant outside diameter. The outer horizontal pipe has been made of aluminum and has a step at flow direction. The surface of outer pipe from step location until exit has been subjected to a constant heat flux. Air flow when passes from step, will experience separation, reattachment and redevelopment while is subjected to a constant heat flux. The effects of changing the inlet heat flux and flow velocity in the inlet section of the pipes on heat transfer rate at the separation zone have been investigated. Dimensions of the test section are shown in the Table 1.

![schematic of test section](image)

**Table 1. Dimensions of the test section**

<table>
<thead>
<tr>
<th>Inner pipe</th>
<th>Outer pipe of entrance section</th>
<th>Outer pipe at test section</th>
</tr>
</thead>
<tbody>
<tr>
<td>D=22 mm</td>
<td>D=46 mm</td>
<td>D=83 mm</td>
</tr>
<tr>
<td>L1=1500 mm</td>
<td>L1 =500 mm</td>
<td>L2 =600 mm</td>
</tr>
</tbody>
</table>

**NUMERICAL PROCEDURE**

The following equations are used to evaluate the heat transfer to the air flow passing the annular pipe with expanded cross section. The outer pipe surface has been exposed to a fix heat flux. Incompressible fluid equations are continuity and momentum equations, in general.

**GOVERNING EQUATIONS OF FLUID FLOW**

Contd

Momentum conservation equations are

\[
\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = -\frac{\partial P}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) + \frac{\partial}{\partial y}(\rho vu) + \frac{\partial}{\partial y}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)
\]

Where u and v are velocity components in the directions x and y, P is pressure, \(\rho\) is the fluid density and \(\nu = \mu / \rho\) is the kinematic viscosity. These equations are solved by finite volume method.

**TURBULENCE MODEL**

Due to the turbulent nature of the flow and for modelling this turbulent flow, a suitable turbulence model is needed.
According to the number of differential equations for associating the turbulent stresses with the averaged velocities or gradient, turbulence models can be divided into zero-equation, one-equation, and two-equation models. Two-dimensional models for rotational and return flows provide acceptable results. In this study, SST k-omega model which uses the k-omega model near the wall and the k-□ model for the remaining sections has been used for modeling.

**SST K-Omega Model**

\[ q = F\delta(T_{sx}^4 - T_w^4) \]  \hspace{1cm} (19)

Where \( \delta = 5.699 \times 10^{-8} \text{ watt/m}^2 \text{K}^4 \) is the Stefan-Boltzmann constant, \( F \) is the shape factor, \( T_{sx} \) is the surface local temperature and \( T_w \) is the average temperature of the inner pipewall. Heat transfer coefficient is calculated by

\[ h = \frac{q_d}{(T_{sx} - T_{bx})} \]  \hspace{1cm} (20)

where \( T_{bx} \) is the temperature of the air mass. Local Nusselt number, \( Nu_d \) can be calculated by

\[ Nu_d = \frac{h_x d}{K_f} \]  \hspace{1cm} (21)

Local Nusselt number, \( Nu_x \), can be calculated obtained by the following equation

\[ Nu_x = \frac{h_x X}{K_f} \]  \hspace{1cm} (22)

The average temperature of the air mass is calculated by

\[ T_s = \frac{1}{L} \int_{x=0}^{x=L} T_{sx} \, dx \]  \hspace{1cm} (23)

Local Reynolds number based on the hydraulic diameter, \( Re_d \), is calculated by

\[ Re_d = \frac{\rho_f U D_h}{\mu_f} = \frac{UD_h}{v} \]  \hspace{1cm} (24)

Where \( v \) is the kinematic viscosity and \( \rho_f \) is the fluid density and \( \mu_f \) is the dynamic viscosity. The hydraulic diameter is calculated by

\[ D_h = \frac{4A}{p} \]  \hspace{1cm} (25)

The circular hydraulic diameter is calculated by
In this simulation, inlet air temperature and pressure are 25 °C and 101325 Pa, respectively. Also, we have \( \rho_{\text{air}} = 1.225 \text{ kg/m}^3 \) and \( v = 17.894 \times 10^{-6} \text{ m}^2/\text{sec} \).

### NUMERICAL MODELING

For turbulence modeling, SST k-omega turbulence model has been used. The mesh of the simulation domain is a non-uniform rectangular one and is thicken near the wall. The mesh of modelling domain consisted of 194,518 nodes. To study the effect of changing the heat flux and flow velocity rate on heat transfer rate, the heat flux on the tube outer test section varies from 719 watt/m\(^2\) to 2098 watt/m\(^2\) and Reynolds number varies from 17050 to 44545. The computational conditions are given in Table 2. Air at the inlet section will be assumed at the ambient temperature (25°C) and pressure (101.325 KPa). Velocity of air flow at the inlet section will be controlled by using of Reynolds Number. The properties of inlet air is \( \rho_{\text{air}} = 1.169 \text{ Kg/m}^3 \), kinematic viscosity = \( \nu = 15.854 \times 10^{-6} \text{ (m}^2/\text{s)} \).

### Table 2. Computational conditions

<table>
<thead>
<tr>
<th>Value</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>101325</td>
<td>Pressure (Pa)</td>
</tr>
<tr>
<td>1.7894 \times 10^{-5}</td>
<td>Viscosity (Pa.s)</td>
</tr>
<tr>
<td>1.23</td>
<td>Density (Kg/m(^3))</td>
</tr>
<tr>
<td>10^{-6}</td>
<td>Residual Error</td>
</tr>
<tr>
<td>SST k-omega</td>
<td>Turbulence model</td>
</tr>
</tbody>
</table>

### RESULTS

In this study, the heat transfer to the air flow with a sudden expansion in an annular pipe has been studied numerically. Figures 2 and 3, show the surface temperature changes and heat transfer coefficient changes with axial distance for different Reynolds numbers for \( q = 2098 \text{ W/m}^2 \), respectively. As shown, the heat transfer coefficient reaches its maximum amount at sudden expansion position, then decreases to reach the exit section. Also, due to the turbulent heat transfer at expansion section, the surface temperature reduces to a minimum temperature, then increases to a maximum temperature and at the test exit section it declines due to the exit heat loss. The minimum temperature is at the stagnation point of the flow. The location of minimum temperature is dependent on the downstream velocity. The stagnation point will move away by increasing the Reynolds number. Figures 4 and 5 show the temperature distribution and velocity magnitude contours, where colors ranging from blue to red indicate low amount to high amount. Figure 6 shows the temperature changes with heat flux along the axial distance. It can be seen that the surface temperature increases by increasing the rate of heat flux. Numerical results have been verified by experimental data of reference (Hussein Togun et al., 2011).

![Figure 3. Changes in surface temperature versus axial distance from the edge of the step for various Reynolds numbers and \( q = 2098 \text{ W/m}^2 \) (compared with the experimental data of Ref. (Hussein Togun et al., 2011))]
Figure 4. Heat transfer coefficient changes with the change in the Reynolds number along the axial distance from the edge of the step ($q = 2098 \text{ W/m}^2$)

Figure 5.

Figure 6. Contour of the temperature distribution along the pipe

Figure 7.

Figure 8. Velocity magnitude distribution along the pipe

Figure 9.
CONCLUSION

It can be seen that increasing the flow rate reduces the surface temperature along the pipe. The heat transfer coefficient reaches its maximum amount at sudden expansion position, then decreases to reach the exit section. The surface temperature increases by increasing the rate of heat flux. Also, the results show that the SST k-omega model is accurate enough to model the heat transfer to the turbulent separation fluid flow.

REFERENCES


TocherB.; 2013. Bifurcation contrasts between plane poiseuille flow and plane magnetohydrodynamic flow, Doctor of Philosophy, Aston University


laden using the Eulerian-Lagrangian approach, Thermal Science, 16(4):1005-1012.

