International Communications in Heat and Mass Transfer 57 (2014) 48–52

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Simulation of heat transfer to separation Air flow in a concentric pipe☆

C.S. Oon [a,b,*], A. Al-Shamma'a [b], S.N. Kazi [a], B.T. Chew [a], A. Badarudin [a], E. Sadeghinezhad [a]

[a] Department of Mechanical Engineering, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia
[b] School of Built Environment, Liverpool John Moores University, Byrom Street, Liverpool L3 3AF, United Kingdom

ARTICLE INFO

Available online 16 July 2014

Keywords:
Numerical simulation
Heat transfer
Turbulent flow
Computational fluid dynamics
Backward facing step

ABSTRACT

Flow separations occur in various engineering applications. Computational simulation by using standard k-ε turbulence model was performed to investigate numerically the characteristic of backward-facing step flow in a concentric configuration. This research is focused on the variation of Reynolds number, heat flux and step height in a fully developed turbulent air flow. The design consists of entrance tube, and inner and outer tubes at the test section. The inner tube is placed along the entrance tube at the test section with an outer tube to form annular conduit. The entrance tube diameter was varied to create step height, s of 18.5 mm. The Reynolds number was set between 17,050 and 44,545 and heat flux was set between 719 W/m^2 and 2098 W/m^2 respectively. It is observed that the higher Reynolds number with step flow contributes to the enhancement of heat transfer. The reattachment point for q = 719 W/m^2 is observed at 0.542 m, which is the minimum surface temperature. The experimental data shows slightly lower distribution of surface temperature compared to simulation data. As for the same case in experimental result, the minimum surface temperature is obtained at 0.55 m. The difference between numerical and experimental result is 0.008 m. Finally, it can be inferred that utilizing the computational fluid dynamic package software, agreeable results could be obtained for the present research.

1. Introduction

One of the challenging types of fluid flow problem, from the perspective of computational fluid dynamics (CFD), is fluid flow involving flow separation and reattachment. Flow separation and reattachment is initiated by a sudden expansion in flow passage, it is also known as backward facing step, which is widely found in many engineering applications. Flow separation on a boundary surface occurs when the flow stream lines (the closest stream line to the boundary surface) breaks or separates away from the boundary surface and then reattaches at a different point. If the boundary surface has a finite dimension, then flow separation is expected due to the flow divergence over the downstream edge where the fluid flows away from the surface such as air flow across an airfoil. From the classical concept, viscosity induces flow separation, which is recognized as boundary layer separation [1].

Mixing of low and high thermal fluids happens in the reattachment flow region affects the heat transfer characteristic. Due to these phenomena, convection over forward and backward step geometries has been investigated by researchers [2]. Fig. 1 illustrates the backward facing step in a sudden expanded pipe.

In industries, rotating cylindrical surface in annular passage is commonly used. Thus, the knowledge of this type of flow passage has got special attention. The simplest representation of this geometry is an annulus space between two concentric-shaped surfaces [4,5]. Study of separation and reattachment flow was conducted first in late 1950s. Futher research was extended to various kinds of working fluids (i.e. nanofluids), boundary conditions and geometries [6–8]. In the advent of numerical codes and sophisticated instrumentations, the complex flow in and around the recirculation can be investigated. Besides the experimental and theoretical approaches, numerical simulation has established itself as the most practical and viable alternative to study and to understand different engineering problems.

2. Literature review

Numerical simulation is commonly used to investigate the heat transfer effect and characteristics of flow separation in a backward facing step. Many researchers have been working on complex flow separation encountered in engineering application. A lot of flow separation applications were extensively utilized in industry even though there are still lacks of knowledge on the information of the flow around the recirculation zone. Some of the earlier studies have been focused on understanding the parameters which affect the reattachment process in these flow phenomena considering the point of suppression and control of separation process. Other studies have added a major emphasis on observation and analysis of such a flow field [9,10].

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail addresses: oonsean2280@siswa.um.edu.my, oonsean2280@yahoo.com (C.S. Oon).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2014.07.008
0735-1933/

Nomenclature

Re_x	Reynolds number
U	velocity of the fluid
ρ_f	density at film temperature
μ_f	dynamic viscosity at film temperature
X	distance along horizontal axis
h_x	convection heat flux
T_{sx}	local surface temperature
T_{bx}	local bulk air temperature
q_c	convection heat flux
$T_{out,ave}$	average outlet temperature
$T_{in,ave}$	inlet temperature
Nu_x	local Nusselt number
K_f	thermal conductivity
D_h	turbulent length scale
L	length of the passage
w	width of the passage
k	turbulence kinetic energy
ω	specific dissipation rate
G_k	generation of turbulent kinetic energy
G_ω	generation of specific dissipation rate
T_k	effective diffusivity of k
T_ω	effective diffusivity of ω
Y_k	dissipation of k due to turbulence
Y_ω	dissipation of ω due to turbulence
S_k	user-defined source term
S_ω	user-defined source term

A recent study examined the turbulence forced convection heat transfer over double forward facing step in 2006 [11]. The Navier–Stokes and energy equations were solved numerically by Computational Fluid Dynamics technique. The solutions were obtained by using the commercial FLUENT code which utilizes the finite volume method [12]. Effects of step heights, step lengths and the Reynolds number on heat transfer and fluid flow were investigated. Results showed that the second step can be used as a control device for both heat transfer and fluid flow. Other researchers focus on experimental study on the effects of sudden contraction and expansion on characteristics of flow and heat transfers in turbulent condition [13,14].

3. Methodology

3.1. Design description

An experiment was conducted before the numerical simulations to verify the accuracy and reliability of numerical simulation results. Fig. 2 shows the experimental setup conducted by Togun et al. [15]. The experimental investigation was focused on the effect of separation flow on the local and average convection heat transfers. The experimental set-up consists of inner constant diameter center pipe, outer unheated concentric entrance pipe and constant diameter heated concentric pipe at test section. The outer tube of test section was made of aluminium having 83 mm inner diameter and 600 mm heated length, which was subjected to a constant wall heat flux boundary condition. The investigation was performed in a Re range of 17050–44545, heat flux varied from 719 W/m^2 to 2098 W/m^2 and the step height of 18.5 mm, which is $d/D = 1.80$.

3.2. Mesh independent study

Mesh independent study was considered to authenticate the results of numerical data obtained from software GAMBIT and FLUENT. The computational domain was being meshed using GAMBIT software. Three different types of mesh have been created for mesh independent study (interval size 4, 4.5 and 5). Constant parameter of step height, $s =$ 18.5 mm and heat flux, $q = 2098$ W/m^2 is used in this study for Reynolds number between 17,050 and 44,545.

3.3. Surface roughness study

Study of the effect of surface roughness on heat transfer was taken into consideration. The surface of the heated wall is set to the surface roughness data measured by Kazi et al. for different materials commercially available. Then, the models are simulated with same parameters and conditions to study the effect of surface roughness. Reynolds number of 17,050 and heat flux of 2098 W/m^2 were considered in this case. Table 1 shows roughness height for different materials.

3.4. Data analysis method

The following section will discuss about the equations employed in the numerical simulation. The equations enable the researcher to the evaluate heat transfer to the flowing air in an expanded annular passage while the test tube is subjected to uniform wall heat flux.

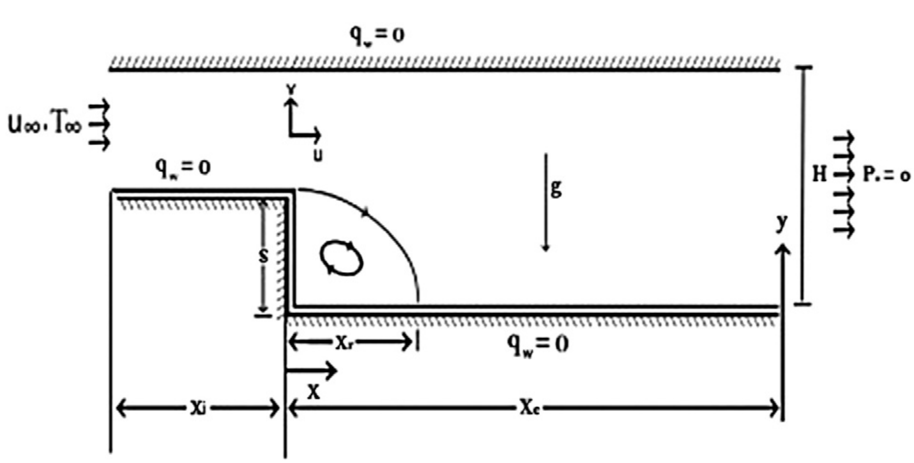

Fig. 1. Schematic diagram for backward facing step [3].