Energy 103 (2016) 75–85

Contents lists available at ScienceDirect

Energy

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

Heat transfer and fluid flow characteristics in air duct with various V-pattern rib roughness on the heated plate: A comparative study

Anil Kumar, Man-Hoe Kim*

School of Mechanical Engineering, Kyungpook National University, Daegu, South Korea

ARTICLE INFO

Article history:
Received 2 November 2015
Received in revised form
4 February 2016
Accepted 24 February 2016

Keywords:
Convective heat transfer
Energy
Overall thermal performance
Rib roughness
CFD

ABSTRACT

This article presents three dimensional CFD (computational fluid dynamics) investigation of local Nusselt number and local friction factor ratios in an air channel with various V-pattern ribs as roughness elements employed on one wall. The various ribs such as, V-pattern rib, protrusion rib in V-pattern, dimpled rib in V-pattern, and V-pattern rib combined with groove have been considered in the study. The CFD analysis on heat transfer and fluid flow characteristics is conducted to investigate the overall effects of the various V-pattern rib shapes on the thermal hydraulic performance. The results found that V-pattern rib with a combination of groove roughness shapes provides better thermal hydraulic performance than other comparable V-pattern rib roughness shapes.

1. Introduction

The efficiency of compact heat exchangers can be improved by modifying the boundary layer developed on the heated surface. One of the well-known methods of modifying the boundary layer is to break the laminar viscous sub-layer formed on the heat transfer surface by creating artificial roughness in the form of repeated ribs, grooves or combination of ribs and grooves. The application of ribs enhances the heat transfer at the cost of increased value of friction factor and power penalty. Hence, the efforts of the researchers are always directed towards the proper selection of the shape and arrangement of ribs, which modifies the boundary layer, enhances the heat transfer coefficient with minimum pressure drop i.e. power penalty [1] and [2].

For comprehensive descriptions of several experimental research on fine wire ribs with different geometries, dimensions, and orientations, readers may refer to Han and Park [3] obtained experimental results for the local heat transfer and turbulence distributions of the flow around transverse ribs for a channel flow. Prasad and Saini [4] through an experimental study for the *Nu* and *f* for a fully developed turbulent flow in an air channel with a heated plate roughened by using transverse fine wires. Kiml et al. [5] through an experimental study on the effect of a rib-roughened rectangular passage on the Stanton number and pressure drop for a high-aspect-ratio channel. Cho et al. [6] performed experimental tests to determine the average velocity and turbulence distributions in the flow around discrete inclined ribs. Bhushan and Singh [7] enhanced the heat transfer in a air vessel using fixed protrusion rib. Jaurker et al. [8] studied the fully developed local heat transfer and friction characteristics in a rectangular air duct roughened by rib-grooved artificial roughness on one broad wall. Saini and Verma [9] reported the experimental prediction of the turbulent fluid flow and heat transfer characteristics in a rectangular air channel with a dimpled rough shape attached to the underside of a heated plate. Momin et al. [10] varied the angle of attack of the V-rib roughness and examined its effect on the Stanton number and pressure drop of an air duct. Yadav et al. [11] enhanced the heat transfer in a rectangular air duct by inserting protrusions rib-type disturbance elements adjacent to the principal wall.

CFD (computational fluid dynamics) is an additional approach to give details the problem of fluid flow and heat transfer in a rib roughened air channel. The reward of CFD in relation to experiments is that it is much cheaper and very less time consuming. A significant review of text on baffles rectangular channel exposed that very few studies used CFD [12–18]. Chaube et al. [12] numerical investigation two-dimensional effect of transverse circular

* Corresponding author. Tel.: +82 53 950 5576; fax: +82 53 950 6550.
E-mail addresses: anilkumar@knu.ac.kr (A. Kumar), manhoe.kim@knu.ac.kr (M.-H. Kim).

http://dx.doi.org/10.1016/j.energy.2016.02.149
0360-5442/

Nomenclature

d_d	Print diameter of dimple rib, *m*
d_p	Print diameter of protrusion rib, *m*
D	Hydraulic diameter of channel, *m*
e	Rib height, *m*
e/D	Relative rib height
e/d_d	Ratio of dimple depth to print diameter
e/d_p	Ratio of protrusion depth to print diameter
E	Energy, *J*
f_{ave}	Average friction factor of rough wall
fs_{ave}	Average friction factor of smooth wall
f_L	Local friction factor of rough wall
fs_L	Local friction factor of smooth wall
H	Depth of channel, *m*
I	Solar intensity, W/m^2
k	Turbulent kinetic energy, m^2/s^2
M_t	Turbulent Mach number
Nu_{ave}	Average Nusselt number of rough wall
Nus_{ave}	Average Nusselt number of smooth wall
Nu_L	Local Nusselt number of rough wall
Nus_L	Local Nusselt number of without rib wall
P	Pitch of the rib, *m*
P/e	Relative rib pitch
p	Pressure, *Pa*
Pr	Prandtl number
Pr_t	Turbulent Prandtl number
Re	Reynolds number
S	Mean rate of strain tensor
u_i	Velocity in x_i-direction, m/s
$\vec{v}$	Overall velocity vector, m/s
W/H	Channel aspect ratio
y^+	Dimensionless distance from walls

Greek symbols

α	Angle of attack, degree
β	Thermal expansion coefficient, $1/K$
μ	Dynamic viscosity, Ns/m^2
μt	Turbulent viscosity, Ns/m^2
ρ	Density, kg/m^3
η	Thermo-hydraulic performance parameter
η_O, β_O	Model constant
ε	Turbulent kinetic energy dissipation rate, m^2/s^3
σ_k	Prandtl number for k
σ_ε	Prandtl number for ε
$C\mu$, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$	RNG $k-\varepsilon$ model constant

Subscript

CFD	Computational fluid dynamics

vortex type ribs an artificially roughened air channel. They observed that the performance of the transverse circular vortex type ribs wall channel was better than that of smooth wall channels. Kumar and Saini [13] carried out a two-dimensional CFD analysis of heat transfer and fluid flow characteristics through an arc shaped rib tabulators air channel. Karmare and Tikekar [14] numerical investigation three-dimensional effect of metal grit type rib an artificially roughened air channel. Yadav and Bhagoria [15] numerically investigated the overall thermal performance of an air channel with transverse rib fitted to the heated wall. Kumar and Kim [16] numerically investigated the overall thermal performance of a solar air channel with multi V-pattern rib to combine staggered rib fitted to the heated wall.

Singh et al. [17] reported the numerical prediction results for the turbulent fluid flow and heat transfer characteristics of solar air heater with transverse ribs. Dongxu et al. [18] reported numerical predictions for the turbulent fluid flow and heat transfer characteristics of a solar air duct with multi V-type rib roughness shape attached to the underside of the heater plate.

In this work, CFD analyses have been carried out on a air duct having various V-pattern rib roughness (simple V-pattern rib, dimpled rib in a V-pattern, protruding rib in a V-pattern and V-pattern rib with grooves) on the underside of the absorber plate, to evaluate the local Nusselt number ratios, local friction factor ratios and consequently the thermohydraulic performance of the roughened channels. The open literature does not contain a similar comprehensive CFD study on a rectangular channel with various V-ribs on the heated plate. Various rib roughness shape rectangular channels are shown in Table 1.

2. Numerical analysis

In the present CFD study, three-dimensional fluid flow through the artificial roughened air duct having different V-pattern roughness on the underside of the heated plate is simulated. The simple V-pattern rib, protrusion rib in V-pattern, dimpled rib in V-pattern, and V-pattern rib with grooves are considered on the underside of the heated plate while other sides are considered as smooth surfaces. The commercial finite-volume based CFD code ANSYS Fluent 6.3.26, has been used to simulate fluid flow and heat transfer. Computational domain, grid generation, boundary conditions, governing equations, and selection of appropriate model are presented in detail in the following sub-sections.

2.1. Numerical models

The three dimensional air channel, has a length, L, equal to 1000 mm while the height H is set equal to 25 mm and width of channel W is 300 mm, the hydraulic diameter, $D = 4A/P = 2H$, is equal to 46.15 mm. The heated plate is made up by aluminum and a steady heat flux to 1000 W/m^2 has been functional as per experimental observation [7–11]. The arrangement of rib roughness elements in the form of V-pattern rib, Dimpled rib in V-pattern, protrusion rib in V-pattern and V-pattern rib with groove roughness attached on the underside side of the heated plate has been careful. The explanation domain used for CFD examination has been generated as shown in Fig. 1. The range of flow and roughness parameters for this CFD based analysis is listed in Table 2.

2.2. Governing equations

A three-dimensional numerical simulation of the conjugate heat transfer and fluid flow was conducted using the CFD code ANSYS Fluent. The simulation modeling was carried out to predict and explain the experimental observations. CFD codes enabled to numerically solve the heat transfer rate, and flow, mass, and energy balances in complicated flow shapes such as a V-ribs air channel. The conservation equations for continuity, momentum, and energy can be written as follows [19–21]: