International Communications in Heat and Mass Transfer 73 (2016) 25–32

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Computational fluid dynamics simulation of heat enhancement in internally helical grooved tubes☆

Sogol Pirbastami, Samir F. Moujaes *, Samad Gharehdaghi Mol

Mechanical Engineering Department, University of Nevada Las Vegas, Las Vegas, NV 89154-4027, United States

ARTICLE INFO

Available online 2 March 2016

Keywords:
Heat transfer coefficient
Nusselt number
Friction factor
Grooved tube
Computational fluid dynamics

ABSTRACT

A computational fluid dynamics (CFD) investigation was carried out to study heat-transfer enhancement in four helically grooved tubes with different pitches, and compared with a smooth tube. The simulations were performed in the Reynolds number range of 4000–20,000 in helical rectangular groove tubes of 2-m length and 7.1-mm diameter under a constant heat flux of 3150 W/m^2. The predicted values for CFD in this current study were compared with previously published experimental data. The primary focus of this study involved evaluating the effect of groove pitch on heat transfer and friction factors. A thermal enhancement factor was defined to evaluate the performance of the internally grooved-tube models. The results revealed that by decreasing the pitch size from 130 mm to 7.1 mm at the same Reynolds number, both the Nusselt number and the friction factor increase. In addition, by increasing the Reynolds number for the grooved pipe, the Nusselt number increased as in the case of a smooth pipe. The highest Nusselt number was obtained for a smaller pitch size of 7.1 mm, but at the expense of a greater pressure drop compared to smooth tubes. An optimum value of the enhancement factor (η) was observed at about Re = 15,000 for all investigated grooves, and enhancement up to 20% was obtained for grooved tubes having a 7.1-mm pitch size.

1. Introduction

Heat-exchanger devices are widely used in industry such as air-conditioning systems, refrigeration, etc. These devices play a major role in energy consumption; therefore, increasing the performance of heat exchange in their components helps to reduce energy consumption, costs, and materials. Since the energy resources on Earth are decreasing and their costs are increasing, design of energy-efficient heat exchangers has a significant effect on energy conservation [1,2]. The technique to increase heat-transfer performance is called heat augmentation, which can be applied to pipe surfaces and other surfaces used in heat exchangers to improve their thermal performance. Enhancement techniques help increase heat convection in heat-exchanger devices by decreasing their overall thermal resistance [1]. However, increasing the rate of enhancement happens usually at the expense of an increase in pressure drop. Therefore, in any heat-augmentation design, the heat-transfer coefficient and the pressure drop need to be analyzed.

Heat-enhancement techniques are categorized into three groups: passive, active, and compound [3]. Active methods require external power and are expensive. In more passive techniques, there is no need for external power; rather, the geometry or surface of the flow channel is modified to promote turbulent flow and increase the heat-transfer coefficient. The compound method is a combination of more than one technique for increasing heat augmentation [4].

In recent years, the study of different techniques for heat-transfer enhancement in heat-exchanger devices has gained increased attention, and several research papers on heat-transfer enhancement have been published [1,5]. The use of grooved tubes is among the many kinds of enhanced surfaces that can provide heat-transfer enhancement [5–11]. In 2013, Aroonrat et al. [6] experimentally investigated the effect of pitch size in internally helical grooved tubes on heat transfer and flow characteristics in stainless steel tubes. The effect of the grooves showed that the Nusselt number and friction factor obtained from the helical grooved tubes were higher than those for smooth tubes [6]. Bilen et al. [12] experimentally investigated the effect of groove geometry on heat transfer and friction characteristics of fully developed turbulent flow in transversely placed grooved tubes with different shapes (circular, trapezoidal, and rectangular). A maximum heat-transfer enhancement was reported up to 63% for tubes having circular grooves, compared to smooth tubes. A correlation equation was then developed experimentally for the Nusselt number and friction factor for each tube. Thermal performance (η) for all grooved pipes in that study was in the range of 1.13–1.28. In 2013, Rahman, Zhen, and Kadir [13] carried out a computational fluid dynamics (CFD) analysis on the flow of a refrigerant (R22) through an enhanced copper tube with multi-start inner grooves. The simulation results were compared with published experimental results, and the overall heat transfer coefficient of enhanced tube was

☆ Communicated by W.J. Minkowycz.
* Corresponding author.
E-mail addresses: pirbasta@unlv.nevada.edu (S. Pirbastami), samir.moujaes@unlv.edu (S.F. Moujaes), gharehda@unlv.nevada.edu (S.G. Mol).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2016.02.007
0735-1933/

Nomenclature

A	Surface area of the test section (mm^2)
C_p	Specific heat at constant pressure (J/kg K)
di	Tube inner diameter (mm)
e	Grooved depth (mm)
f	Friction factor
h	Heat transfer coefficient (W/m^2 K)
k	Thermal conductivity (W/m K)
L	Length of the test tube (mm)
m	Mass flow rate (kg/s)
Nu	Nusselt number
η	Thermal performance factor
P	Pressure (Pa)
p	Grooved pitch (mm)
Q	Heat transfer rate (W)
q″	Heat flux (kW/m^2)
Re	Reynolds number
r	Radius (mm)
T	Temperature (°C)
u	Flow velocity at the inlet of test tube (m/s)
w	Grooved width (mm)
z	Axial location along tube (m)

Greek symbols

μ	Dynamic viscosity (kg/m s)
ρ	Density (kg/m^3)
ΔP	Pressure drop (Pa/m)
Φ	Dissipation function
ε	Dissipation rate

Subscripts

avg	Average
I	Inlet
O	Outlet

found to be higher compared to a smooth tube [13]. In 2015, Liu, Xie and Simon [14] studied the heat transfer performance of turbulent flow (Re: 10,000–25,000) in a square duct with cylindrical grooves. Their objective was to find a desirable design for a better enhancement rate and minimum pressure-drop penalties.

The purpose of the current research is to develop a predictive model using a CFD approach to describe heat enhancement of internal rectangular helically grooved tubes in an otherwise smooth tube, and validate the results with the work of Aroonrat et al. [6] and extrapolate on these results.

2. Numerical method and procedure

2.1. Geometry and grid generation

Several cylindrical tubes with 2-m lengths and internally rectangular grooves were simulated in this current study. Tubes with a pitch length of 203, 254, and 305 mm were used to validate the CFD model with the published experimental data, in the range of Re 4000–10,000. Tubes with pitch sizes of 7.1, 12.7, 50, and 130 mm were used to expand the CFD results in Re ranges 4000–20,000. Geometrical models of the CFD simulation were generated using SolidWorks and imported to the CFD software using STARCCM+ for analysis. The details of each test section and a schematic diagram of the grooved tube are provided in Table 1 and Fig. 1, respectively.

The dimensions of the constant heat flux test sections were the same as the grooved channels of the experimental work [6] which is used for validation of the current study. A polyhedral mesh was used as the core

Table 1
Details of the test section runs.

Tube	Groove width (w)	Groove depth (e)	d_i (mm)	Pitch length (mm)	Length (mm)
GT 7.1	0.2	0.2	7.1	7.1	2000
GT 12.7	0.2	0.2	7.1	12.7	2000
GT 50	0.2	0.2	7.1	50	2000
GT 130	0.2	0.2	7.1	130	2000
GT 203	0.2	0.2	7.1	203	2000
GT 254	0.2	0.2	7.1	254	2000
GT 305	0.2	0.2	7.1	305	2000

mesh, and prism layer mesh with grid adoption for $y+\approx 1$ at an adjacent wall region was used to resolve the laminar sub-layer, as shown in Fig. 2. Before, running the simulation, the best mesh was chosen by a grid independent study which is explained in Section 3.1.

2.2. Governing equations and turbulence model

The phenomenon under consideration was governed by the steady 3-D form of the continuity, the incompressible Navier–Stokes equations, and the energy equation, all of which are solved by STARCCM+ software. In this study, the conservation laws were expressed in Cartesian coordinates as follows [15]:

Conservation of mass:

$$\frac{D\rho}{Dt}+\rho\nabla.\overline{V}=0. \tag{1}$$

Conservation of momentum (Navier–Stokes equations):

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho g_x-\frac{\partial p}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right]\quad \text{(x direction)} \tag{2}$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]\quad \text{(y direction)} \tag{3}$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]\quad \text{(z direction).} \tag{4}$$

Conservation of energy:

$$\rho c_p\frac{DT}{Dt}=\nabla.k\nabla T+\mu\varphi \tag{5}$$

where the dissipation function in Cartesian Coordinates is [15]:

$$\varphi=2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2 +\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)^2. \tag{6}$$

In the present work, the *k*–*ε* model was chosen for the simulation [16,17]. In the *k*–*ε* two equation model, the velocity and length scale of turbulence are defined with two additional partial-differential equations, one for the turbulent kinetic energy *k* and the other for the dissipation rate ε and an algebraic demonstration for the eddy viscosity [16,17] is also given as in Eq. (9). The final form of *k* equation is:

$$\rho\frac{\partial k}{\partial t}+\rho\langle u_j\rangle\frac{\partial k}{\partial x_j}=2\mu_t\langle s_{ij}\rangle\frac{\partial\langle u_i\rangle}{\partial x_j}-\rho\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]. \tag{7}$$