Renewable and Sustainable Energy Reviews 92 (2018) 405–425

Contents lists available at ScienceDirect

Renewable and Sustainable Energy Reviews

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

A review of artificial roughness geometries employed in solar air heaters

Inderjeet Singh[a,*], Sukhmeet Singh[b]

[a] Department of Mechanical Engineering, Punjab Agricultural University, Ludhiana 141004, India
[b] School of Renewable Energy Engineering, Punjab Agricultural University, Ludhiana 141004, India

ARTICLE INFO

Keywords:
Solar air heater
Artificial roughness
Heat transfer
CFD
Reynolds number
Thermohydraulic

ABSTRACT

Enhancement of heat transfer in the solar air heater ducts can be achieved by breaking laminar sub-layer near the absorbing surface and can be efficiently done by employing ribs as roughness elements. However, this gain is accomplished at the expense of increase in pressure drop. This paper presents the detailed review of various investigations made on artificial roughness geometries summarizing their outcomes. Attempt have been made to study the effect of various influencing roughness and flow parameters on the thermal and hydraulic performance of solar air heater through the flow visualization.

1. Introduction

The role of energy becomes increasingly important to fulfill needs of modern societies and to sustain fast economic and industrial growth worldwide. The rapid depletion of fossil resources has necessitated an urgent search for alternative sources of energy. Solar energy is available freely and an indigenous source of energy provides a clean and pollution free atmosphere. Energy consumption has been multiplying at one of the fastest rates in the world due to population growth and economic development. In view of world's depleting fossil fuel stocks and environmental threats, development of renewable energy sources has received momentum. From many alternatives, solar energy stands out as the brightest long-term resource for meeting continuously increasing demand of energy.

Solar air heater (SAH) is a device in which energy from sun is captured by absorbing surface and the thermal energy is extracted by the air flowing over it [1]. SAH is the cheapest way of solar energy conversion and used for various applications as space heating, drying of crops, and other industrial applications. A typical SAH is simply designed and requires less maintenance. However, they have poor heat transport between absorber and fluid due to development of laminar sublayer which results in a lower efficiency [2,3]. The heat transfer coefficient can be significantly improved by disrupting the laminar sublayer and inducing turbulence adjacent to the absorber plate by providing artificial roughness [4]. However, it is done at the cost of extra pressure drop which upturns the pumping power requirements.

The use of repeated ribs as roughness elements underside the absorber is one of the convenient and most efficient method for heat transfer augmentation. A lot of experimental as well as few Computational Fluid Dynamics (CFD) explorations [4] are reported so far to evaluate the influence of roughness elements on the thermal and frictional performance of roughened SAH duct. Further, attempts had been made to optimize the rib roughness parameters.

This article presents a comprehensive review of various investigations carried out with the purpose of obtaining maximum heat transfer improvement and least pumping power penalty. The outcomes of these investigations are discussed along with the optimum parameters obtained and the reported correlations of Nusselt number and friction factor.

2. Performance of solar air heater duct

2.1. Thermal performance

Thermal performance of SAH duct is expressed as the convective heat transport between the absorber and the working medium i.e. air (Fig. 1). The thermal efficiency of a typical SAH duct is low due to low value of convective heat transfer coefficient (h) due to laminar sublayer formation close to the absorber plate.

The rate of useful energy gain by the air flowing through SAH duct may also be calculated by using the following equation:

$$Q_u = mC_p(T_o - T_i) = hA_c(T_{pm} - T_{am}) \tag{1}$$

Nusselt number for a smooth duct can be obtained by Dittus-Boelter Equation [5]:

$$Nu = 0.023 Re^{0.8} Pr^{0.4} \tag{2}$$

The heat transfer coefficient (h) can be increased by the application

* Corresponding author.
E-mail addresses: inderjeetsingh.pau@gmail.com, inderjeet-me@pau.edu (I. Singh).

https://doi.org/10.1016/j.rser.2018.04.108
Received 19 December 2017; Received in revised form 21 April 2018; Accepted 21 April 2018
1364-0321/

Nomenclature

D	hydraulic diameter of duct (mm)
A_p	effective collector area (m^2)
e	rib height (mm)
L	length of duct (mm)
P	pitch (mm)
W	width of duct (mm)
H	height of duct (mm)
p'	staggered pitch (mm)
T_a	ambient temperature, K
V	wind speed, m/s
n	number of glass covers
I	solar radiation intensity, W/m^2
m	mass flow rate (kg/s)
ΔP	pressure drop (Pa)
T_i	inlet temperature (K)
T_o	outlet temperature (K)
T_{pm}	mean plate temperature (K)
T_{am}	mean air temperature (K)
ρ	density of air (kg/m^3)
α	angle of attack (degree)

Dimensionless parameters

Re	Reynolds number
e/D	relative roughness height
P/e	relative roughness pitch
W/w	relative roughness width
W/H	duct aspect ratio
e^+	roughness Reynolds number
d/w	relative gap position
g/e	relative gap width
l/s	relative grit length
L/e	relative longway length of wire mesh
S/e	relative shortway length of wire mesh
p'/P	relative staggered pitch ratio
l/s	relative grit length
f	friction factor of roughened duct
f_s	friction factor of smooth duct
Nu	Nusselt number of roughened duct
Nu_s	Nusselt number of smooth duct
St	Stanton number
η_{th}	thermal efficiency
η_{eff}	effective efficiency

of artificial rib roughness on the air flow side of absorber plate and thereby cause increase in the thermal efficiency given by Eq. (3).

$$\eta_{th} = \frac{Q_u}{IA_p} \tag{3}$$

2.2. Hydraulic performance

The air flowing through the SAH duct undergoes frictional losses and hence accounts for the extra energy in form of mechanical power that has to be supplied to the blower to circulate air properly in the duct. The hydraulic performance for the fully developed turbulent flow can be represented by friction factor which is given by:

$$f = \frac{2(\Delta P)_d D_h}{4\rho L v^2} \tag{4}$$

Further using above equations mechanical power can be computed by Eq. (5) [6]:

$$P_m = \frac{m(\Delta P)_d}{\rho} \tag{5}$$

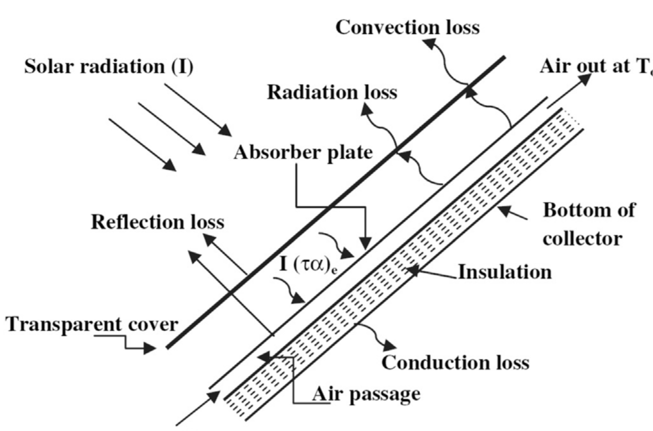

Fig. 1. A conventional flat plate SAH.

2.3. Thermo-hydraulic performance

The overall enhancement in the performance of a roughened SAH duct can be determined by considering thermal and hydraulic characteristics simultaneously in contrast to the SAH with smooth duct. A thermohydraulic performance parameter given by [7] is used to compare the roughened and smooth surfaces in terms of Nusselt number and friction factor ratios. Therefore, thermohydraulic performance of a SAH is determined by:

$$Thermohydraulic\ performance = \frac{(Nu/Nu_s)}{(f/f_s)^{\frac{1}{3}}}$$

3. Concept of artificial roughness

In conventional flat plate SAH's the laminar sub layer has to be disturbed for enhancing the heat transfer by inducing turbulence adjacent to the absorber plate surface. This can be effectively done by the employment of artificial ribs on the air flow side of the absorber. However, the use of artificial roughness may result in high pressure loss due to friction and hence more power requirements for pumping of fluid [8–10].

For the investigation of the effect of artificial roughness elements, SAH is usually modeled as rectangular channel with one wall comprising ribs on the air flow side while other three walls are kept smooth. In the present study, duct has a length, width and height of 1000 mm, 300 mm and 25 mm respectively and hydraulic diameter of 46.15 mm. The roughened side is subjected to uniform heat flux of 1000 W/m^2.

For the better understanding of the flow phenomena and effect of the geometrical and flow parameters, a CFD analysis has been carried out in ANSYS Fluent 15.0. 3-D Finite volume based numerical method using turbulence model RNG k-ε with enhanced wall treatment has been selected for present CFD analysis. The governing equations have been discretized with second order upwind scheme. The convergence criteria for momentum and turbulence equations is kept 10^{-5} and 10^{-10} for energy equation.

For the validation and verification of the CFD methodology, the results for circular transverse rib have been investigated using the proposed CFD method and compared with the experimental results reported by Gupta et al. [18]. Fig. 2 presents the comparison of Nusselt number computed by the proposed CFD method with the results