

A CFD Based Analysis of Solar Air Heater Having V-Shaped Perforated Blocks on Absorber Plate

Ashok Singh Yadav¹, Tarun Singh Samant², Lokesh Varshney³

¹ P.G. Scholar, Department of Mechanical Engineering, College of Technology, Pantnagar, Uttarakhand, India

² P.G. Scholar, Department of Mechanical Engineering, College of Technology, Pantnagar, Uttarakhand, India

³ Professor, Department of Mechanical Engineering, College of Technology, Pantnagar, Uttarakhand, India

Abstract - In this article a numerical investigation is conducted to analyze the 3-dimensional incompressible Navier–Stokes flows through the artificially roughened solar air heater for the Reynolds number ranges from 2000 to 20,000, the effect of geometrical parameters of the V-shaped perforated blocks on heat transfer and flow characteristics of rectangular duct, has been investigated. The governing equations are solved with a finite-volume-based numerical method. The commercial finite-volume based CFD code ANSYS FLUENT 14 is used to simulate turbulent airflow through artificially roughened solar air heater. The RNG $k-\epsilon$ turbulence model is used to solve the transport equations for turbulent flow energy and dissipation rate. The investigation encompassed the geometrical parameter namely, relative blockage height (e/H) of 0.4–1.0, relative pitch ratio (P/e) of 4–12 and open area ratio (β) of 5–25% at a fixed angle of attack (α) of 60° . The maximum enhancement in Nusselt and friction factor has been found to be 6.38 and 13.96 times to that of smooth duct, respectively.

Key Words: Solar air heater, Nusselt number, Friction factor, V-shaped blocks, Perforation, Rectangular duct

1. INTRODUCTION

A solar air heater is a type of energy collector in which energy from the sun is captured by an absorbing medium and used to heat air. Solar air heater has myriad of uses and applications. The main applications of solar air heater are drying of crops, seasoning of timber, space heating, fresh air ventilation etc. A solar air heater is very simple in design and requires little maintenance. It is observed that the heat transfer coefficient between the absorber plate and air of solar air heater is generally poor and this result in lower efficiency [1]. These air heaters have low thermal efficiency due to development of laminar sub-layer in

turbulent boundary layer on airflow side of the absorber plate [2]. The enhancement in thermal performance by breaking the laminar sub-layer by employing artificial rib roughness on the airflow side of absorber plate has widely been accepted as convenient and efficient method [3-5].

Earlier studies carried out on turbulators in the form of small wires used in solar air heater utilized different geometries like transverse ribs [6,7], W-shaped ribs [8], V shaped ribs [9], multi V-shaped ribs [10] and multi V-shaped ribs with gap [11]. Alam et al. [12,13] reported that larger turbulence promoters such as ribs, block, baffles, vortex generators and obstacles of larger height lead to relatively higher heat transfer with still have higher pressure drop penalty, which was the main drawback of such devices. Hot spots have also been found to develop in the wake of solid elements due to flow re-circulating [14,15] resulting in lowered heat transfer from these zones. This problem is overcome by using modified ribs/blocks/baffles in the form of perforated ribs/blocks/baffles by way of perforation where part of flow passes through these perforations. Hawang and Liou [15] investigated heat transfer in a channel with perforated fences and compared the thermal performance of three types of turbulence promoters namely, solid, perforated and slit types. It was reported that perforations in turbulence promoters substantially enhanced the heat transfer. Buchlin [16] studied the heat transfer characteristics in a channel with various perforated rib turbulators by means of an infrared scanning radiometer. Shaeri and Yaghoubi [17] numerically investigated the effect of perforated fin on heat transfer and pressure drop. Sara et al. [18,19] investigated the solid and perforated blocks attached on the heated surface in the duct. Karwa et al. [20,21] investigated the half and fully perforated baffles attached to the heated surface. Enhancement of 79–169% in Nusselt for fully perforated and 133–274% for half perforated baffles was found. Shin and Kawak [22] reported that wider hole gave higher thermal performance. Nuntadusit et al. [23] carried out detailed studies of six different perforated blocks (made by acrylic). Circular tube equipped with perforated twisted tape, having four different porosities ($R_p = 1.6\%, 4.5\%$,

8.9% and 14.7%) investigated on Nusselt number, friction factor and thermal performance. An enhancement in Nusselt number, friction factor and thermal performance factor was found to be 110–340%, 110–360% and 28–59% higher in comparison to plane tube, respectively [24].

Most of the studies on rib roughness of solar air heater duct have been carried out experimentally and only a few studies using computational fluid dynamics have been reported [25-30]. Two- dimensional CFD (computational fluid dynamics) analysis on rib roughened solar air heater duct using commercial software FLUENT 6.1 has been reported by Chaube et al. [31]. Heat transfer and fluid flow analysis have been done for five rib shapes viz. rectangular, square, chamfered, semicircular and circular rib. Shear stress transport k-u turbulence model was selected for analysis based on the comparison with experimental results available in literature. The inter-rib Nusselt number, streamlines and velocity vectors have been plotted. Results have been obtained in-terms of Nusselt number, friction factor and performance index. Two-dimensional CFD analysis for wedge shaped transverse rib roughened solar air heater duct done using CFD software FLUENT was found to have good agreement with experimental results for Nusselt number [32]. Yadav and Bhagoria [28] conducted 2-D CFD study using ANSYS FLUENT 12.1. From amongst the different turbulence models, the results obtained using RNG (Renormalization group) k-ε turbulence model were in good agreement with the Dittus-Boelter and Blasius empirical correlation. The effect of relative roughness height, relative roughness pitch and Reynolds number for transverse rib was investigated. The Nusselt number, friction factor and thermal enhancement factor were determined. Three-dimensional CFD investigation on rib roughened solar air heater duct using FLUENT 6.3.26 has been reported by Kumar and Saini [25] for arc-shaped rib roughness. RNG (Renormalization-group) k-ε turbulence model has been selected for analysis. The effect of geometrical rib parameters and flow Reynolds number on overall enhancement ratio has been determined. CFD simulation in 3D was performed for metal grit rib roughened solar air heater duct using CFD software FLUENT [26]. The investigation was done for different rib geometry parameters. The simulation was validated experimentally.

Conventional techniques used for the design and development of an artificially roughened solar air heater are expensive and time consuming. CFD approach has emerged as a cost effective alternative and it provides speedy solution to design and optimization of an artificially roughened solar air heater. CFD is a design tool that has been developed over the past few decades and will be continually developed as the understanding of the physical and chemical phenomena underlying CFD theory improves. The goals of CFD are to be able to accurately predict fluid flow and heat transfer in complex systems.

CFD uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. High speed computers are used to perform the calculations required to simulate the interaction of gases and liquids with surfaces defined by boundary conditions.

Literature reveals that creating the perforations in solid blocks enhanced the heat transfer rate due to acceleration of flow in the form of jets through these perforations which subsequently strikes the heated surface and mix with main flow. However in most of the cases blocks were oriented in the direction transverse to the flow. Earlier studies in the case of ribs showed that the V-shaping of the transverse ribs results in considerably higher heat transfer due to the generation of secondary flows. Consequently, it was proposed to investigate the effect of perforation in solid V-shaped blockages on heat transfer and fluid flow characteristics of rectangular duct. In this work, an attempt to bridge this gap by presenting a detailed CFD investigation of artificially roughened solar air heater having V-shaped perforated blocks on absorber plate in a rectangular duct.

2. DESCRIPTION OF COMPUTATIONAL MODEL

Solar air heater duct having V-shaped perforated blocks has been considered for CFD investigation (Fig. 1). The flow cross-section of the duct is 100mm × 20mm and duct length is 640mm, which is divided into three section , an entrance section, a test section and an exit section of lengths 245mm(>5√WH), 280mm and 115mm(>2.5√WH) respectively . V-shaped rectangular aluminium blocks having perforation were attached on the underside of the plate. Thickness of blocks was taken 3 mm because small thickness block was more advantageous [33].

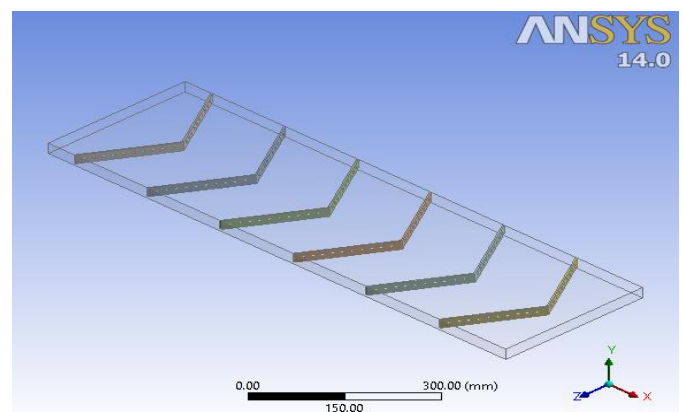


Fig -1: Geometry of the Model

The open area ratio is defined as ratio of the area of the perforation to the block frontal area, given by

$$\beta = \frac{n(\pi \times D^2 / 4)}{b \times e}$$

where n is the number of hole in one limb. This open area ratio could be changed either by changing the size of perforation for same number of perforations or by changing the number of perforations for same size of perforation. Sara et al. [18] suggested that changing the size of perforation was advantageous. In the current study, open area ratios were changed by changing the perforation holes size only.

The Reynolds number has been varied from 2000 to 20,000. The investigations have been carried at five levels of Reynolds number in this range. The investigation was done using software ANSYS CFD 14.0. ANSYS Design Modeler was used for preparing the 3-D fluid domain for analysis. Initially a coarse mesh was used to resolve the flow. The steady flow analysis was done using governing equations of energy, momentum and continuity [34]. The solver used was segregated with finite volume based algorithm and second order upwind scheme was chosen for momentum, continuity and energy equations. For discretization of governing equations, SIMPLE algorithm was chosen [35]. A constant heat flux condition of 1000 W/m² was applied on ribbed surface, while for other walls adiabatic boundary condition was applied. All the other walls are considered to be completely insulated with zero heat flux. No slip condition is applied to all the 'walls'. A uniform air velocity is introduced at the inlet while a pressure outlet condition is applied at the exit. At the exit, a pressure outlet boundary condition is specified with a fixed pressure of 1.013×10⁵ Pa. The convergence limit of 10⁻⁵ for residual of continuity equation, 10⁻⁵ for the residuals of velocity components and 10⁻⁸ for residuals of energy was taken. After obtaining initial results, the mesh was made finer at specific locations to obtain the results.

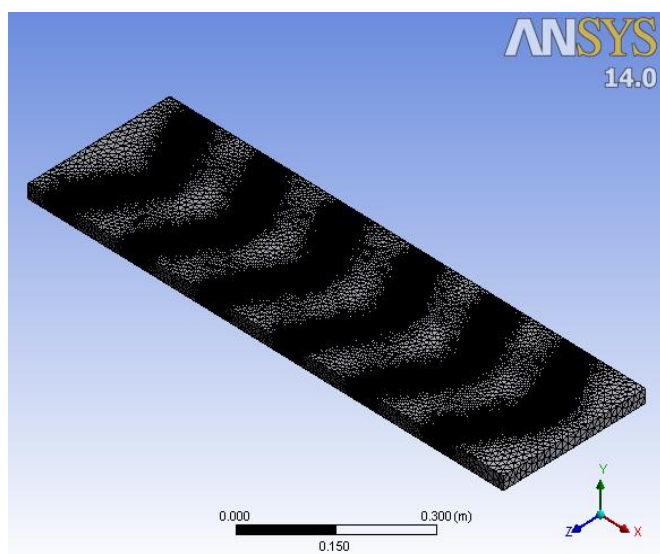


Fig -2: Meshing of the Geometry

Table -1: Range of parameters.

S.No.	Parameters	Range
1	Reynolds number (Re)	2000-20,000 (Ten values)
2	Relative blockage height (e/H)	0.4-1.0 (Four values)
3	Relative pitch (P/e)	4-12 (Five values)
4	Angle of attack (α)	60° (Fixed)
5	Open area ratio (β)	5-25% (Five values)

Table -2: Thermo-physical properties of air and absorber plate for CFD analysis.

Properties	Air	Absorber plate and V-Shaped ribs(aluminum)
Density, ' ρ ' (kgm ⁻³)	1.7705	2719
Specific heat, ' C_p ' (Jkg ⁻¹ K ⁻¹)	1006.35	871
Viscosity, ' μ ' (Nsm ⁻²)	1.8415e-05	-
Thermal conductivity, ' k ' (Wm ⁻¹ K ⁻¹)	0.0262	202.4

3. SELECTION OF FLOW FIELD, TURBULENCE MODEL AND MODEL VALIDATION

Eight turbulence models viz. Standard k- ϵ model with standard and enhanced wall treatment, Renormalization k- ϵ model with standard and enhanced wall treatment, Realizable k- ϵ model with standard and enhanced wall treatment, SST (Standard and Shear Stress Transport) k- ω were tested for smooth duct [37]. The Nusselt number obtained for smooth duct is compared with Nusselt number for smooth duct as given by Dittus-Boelter empirical correlation in eq. (1) below:

$$Nu = 0.023Re^{0.8}Pr^{0.4} \dots\dots\dots(1)$$

[37] shows the variation of Nusselt number with Reynolds number using different turbulence models. Also shown is the plot of calculated Nusselt number values determined from eq. (1) for comparison. The Nusselt number determined by using turbulence model RNG k- ϵ with enhanced wall treatment were found to result in least average absolute deviation from the values of Nusselt number obtained from empirical correlation given in equation (1). Keeping this in view, the turbulence model

RNG k-ε with enhanced wall treatment was selected for investigation on roughened duct.

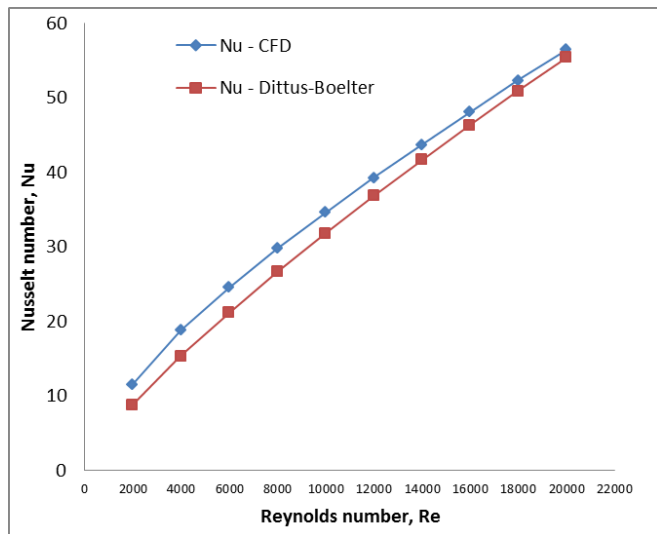


Chart -1: Comparison of experimental and predicted values of Nusselt number for smooth surface

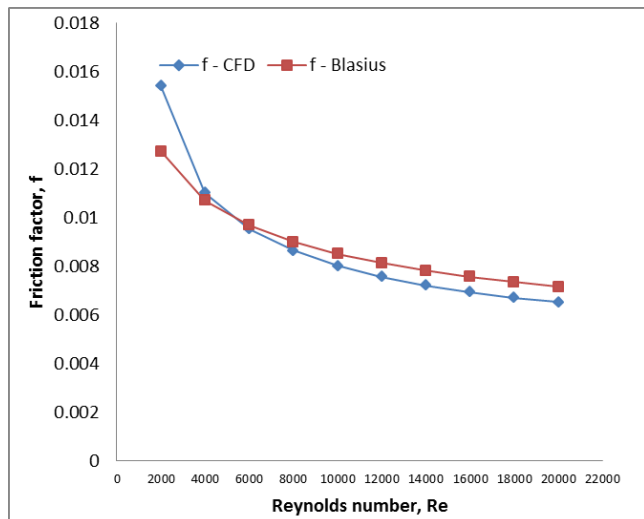


Chart -2: Comparison of experimental and predicted values of friction factor for smooth surface

4. RESULTS AND DISCUSSION

Nusselt number and friction factor values for rectangular duct with perforated V-blocks, computed by using the CFD analysis for various sets of values of geometrical parameters of blocks are discussed below.

4.1 Effect of Reynolds number (Re)

Chart -3 and Chart -4 shows the effect of Reynolds number (Re) on Nusselt number (Nu) and friction factor (f) for perforated V-shaped blockages with $e/H = 0.8$, $P/e = 8$, $\beta = 20\%$, solid V-shaped blockages with relative blockages height $e/H = 0.8$, $P/e = 8$ and smooth duct.

Nusselt number is observed to increase with increasing Reynolds number (Re) for all cases.

It can be seen that perforated blockages yield higher heat transfer to that of solid blockages. Perforated V-shaped blockage leads, to approximately, 35% improvement in Nusselt number in comparison to that of V-shaped solid blockages. Though, friction factor is decreased due perforation in blockages.

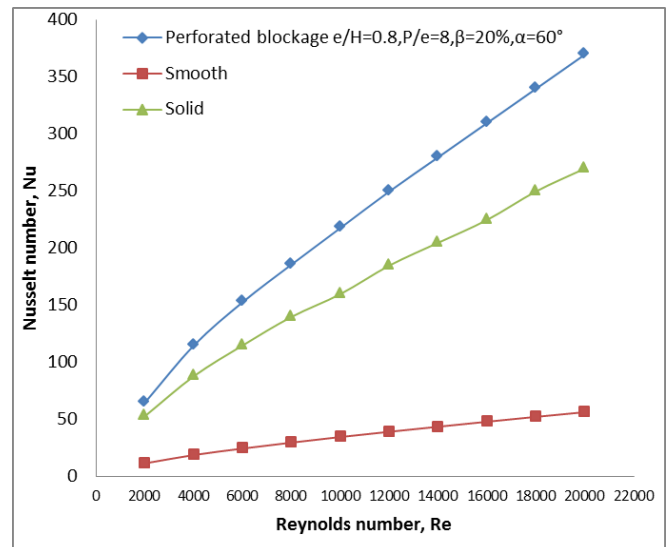


Chart -3: Effect of Reynolds number on Nusselt number

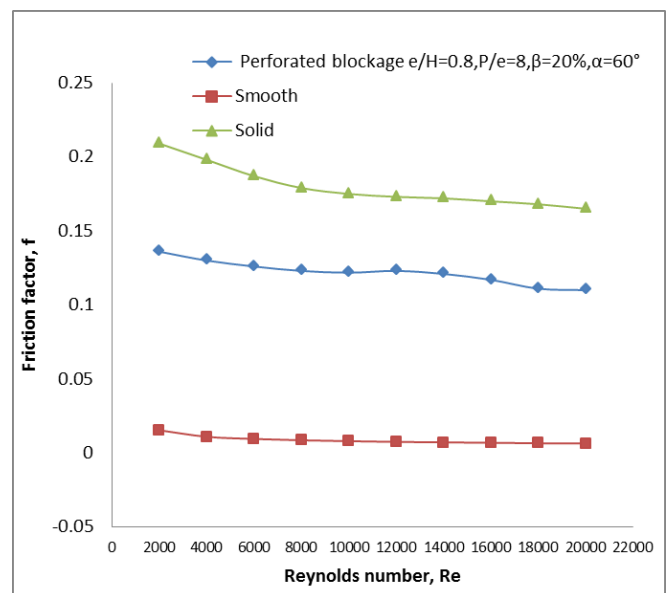


Chart -4: Effect of Reynolds number on friction factor

4.2 Effect of relative blockage height (e/H)

Chart -5 shows the effect of relative blockage height (e/H) on Nusselt number for given values of other fixed geometrical parameters. This figure shows that Nusselt number increases with relative blockage height (e/H),

attains maxima at a relative blockage height of 0.8 and further increase of blockage height, the value of Nusselt number slightly decreases. The value of Nusselt number is higher for relative blockage height ratio of 0.8 and lower for relative blockage height ratio of 0.4.

Effect of relative blockage height on friction factor is shown in Chart -6 by keeping the other geometrical parameters fixed. This figure shows that friction factor increases with relative blockage height.

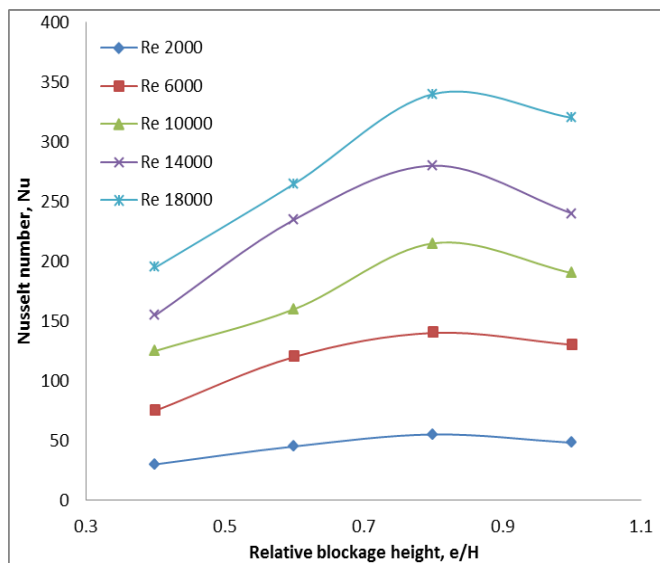


Chart -5: Effect of relative blockage height on Nusselt number

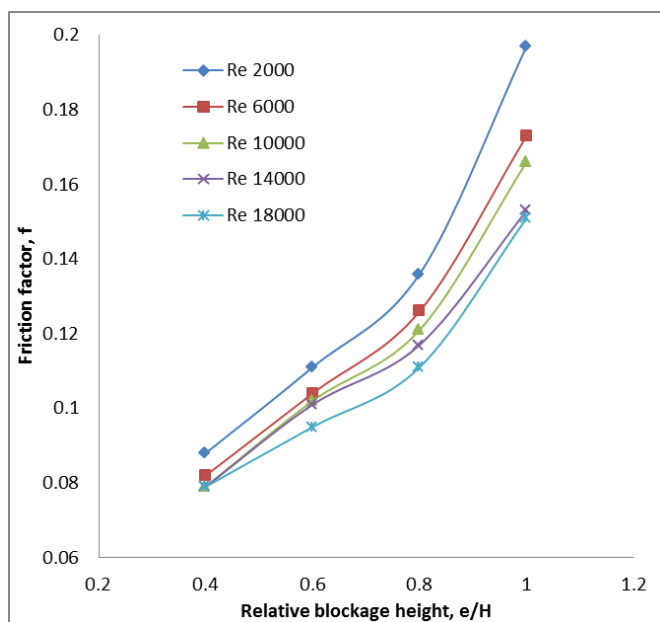


Chart -6: Effect of relative blockage height on friction factor

4.3 Effect of relative pitch (P/e)

Chart -7 shows the effect of relative pitch ratio (P/e) on Nusselt number for fixed values of other geometrical parameter. It can be observed that Nusselt number attained maximum values corresponding to relative pitch of 8. It appears that the flow separation may occur downstream of a rib and re-attachment of free shear layer may occur if relative roughness pitch ratio (P/e) lies in the range of 8–10 and maximum heat transfer occurs in the vicinity of re-attachment point.

Maximum heat transfer is obtained by keeping the relative pitch ratio of 8 in the current study. This may be the optimum relative pitch ratio for this block configuration and flow condition. Chart -8 shows the effects of relative pitch on friction factor at a fixed value of relative blockage height of 0.8 and open area ratio of 20%. It appears to be the friction factor decreases with increasing the relative pitch ratio for all Reynolds number.

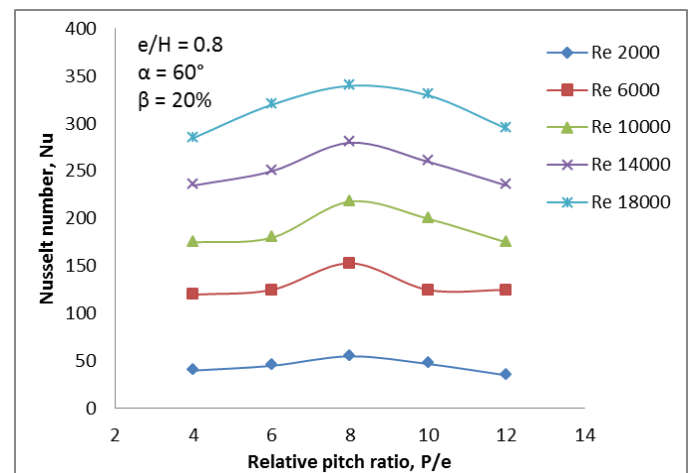


Chart -7: Effect of relative pitch ratio on Nusselt number

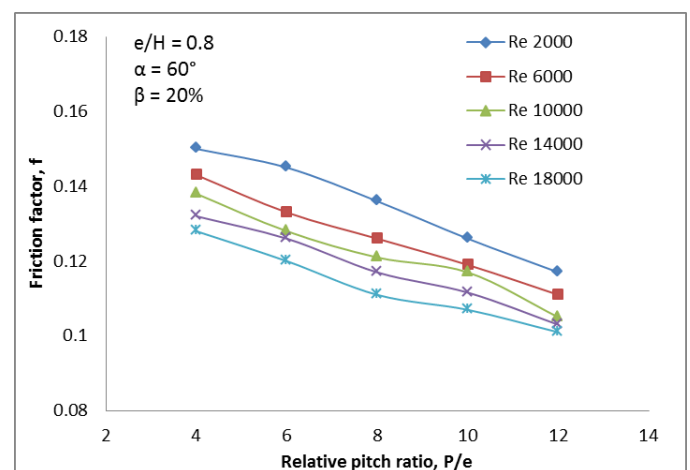


Chart -8: Effect of relative pitch ratio on friction factor

4.3 Effect of open area ratio (β)

Chart -9 shows the effect of open area ratio on Nusselt number at relative pitch ratio (P/e) of 8, relative blockage height (e/H) of 0.8 and angle of attack (α) of 60°. This

figure shows that Nusselt number increases with increase in the open area ratio up to 20%, outside which it decrease with increase in the open area ratio. The value of Nusselt number is higher for open area ratio of 20% and lower for open area ratio of 5%. This reason may be explained by the effect of increasing the jet diameter. For the given axial distance, radial expansion of the jet will be more for higher jet diameter. Thus, in case of blocks of multiple jet perforations, large enough radial expansion of jet interact with neighbouring jet with each other, this would lead to the higher degree of flow mixing and help the reduce the separated flow zone downstream of each block, thus yielding the higher heat transfer when open area ratio increase up to 20%. When open area ratio increase beyond 20%, the flow velocity through the hole will reduce, which may not be strong enough to accelerate the flow through the hole, this would weaken the flow turbulence and flow mixing, hence the heat transfer reduce significantly.

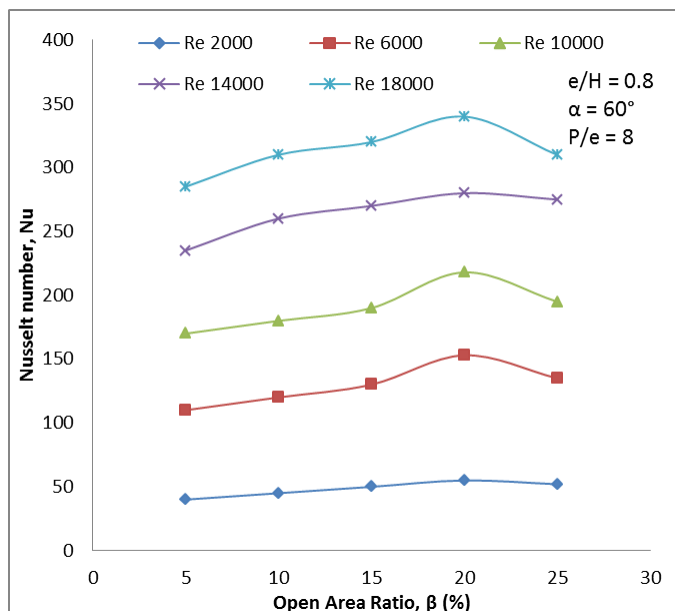


Chart -9: Effect of open area ratio on Nusselt number

Chart -10 shows the effect of open area ratio on friction factor at same other parameters. This figure displays that friction factor decreases with increase in open area ratio because higher open area ratio offers low resistance to air, thus resulting in lower friction factor.

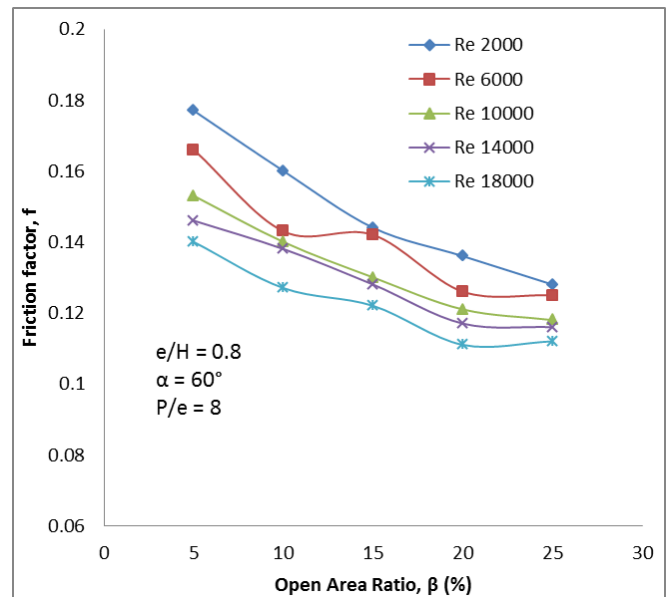


Chart -10: Effect of open area ratio on friction factor

5. CONCLUSIONS

Based on the investigation of heat transfer and friction characteristics of a rectangular duct with V-shaped perforated blocks on the heated surface subjected to uniform heat flux, it can be stated that Nusselt number and friction factor are strongly depended on the open area ratio, relative blockage height and relative pitch ratio. Following conclusions are drawn:

1. Providing the perforation in V-shaped blockages average enhancement in Nusselt number for perforated V-shaped blockages is found to be 33% higher than solid blockages while friction factor of perforated blockages is decreased by 32% of the value as found in solid blockages.
2. When compared with the smooth duct, the presence of V-shaped blockages with perforation holes yields Nusselt number up to 6.38 times while friction factor rises up to 13.96 times.
3. Maximum enhancement of Nusselt number occurs at open area ratio (β) of 20%, relative blockage height (e/H) of 0.8, and relative pitch (P/e) of 8, while maximum friction factor is found corresponding to open area ratio (β) of 5%, relative blockage height (e/H) of 1.0, and relative pitch (P/e) of 4.

REFERENCES

- [1] A.S. Yadav, J.L. Bhagoria, Renewable energy sources - an application guide, Int. J. Energy Sci. 3 (2) (2013) 70-90.
- [2] Bhatti MS, Shah RK. Chapter 4. In: Kakac S, Shah RK, Aung W, editors. Handbook of single-phase convective heat transfer. New York: Wiley; 1987.

- [3] Kumar A, Saini RP, Saini JS. A review of thermohydraulic performance of artificially roughened solar air heaters. *Renew Sustain Energy Rev* 2014; 37: 100-22.
- [4] Sharma SK, Kalamkar VR. Thermo-hydraulic performance analysis of solar air heaters having artificial roughness - a review. *Renew Sustain Energy Rev* 2015; 41:413-35.
- [5] Yadav AS, Bhagoria JL. Heat transfer and fluid flow analysis of solar air heater: a review of CFD approach. *Renew Sustain Energy Rev* 2013;23:60-79.
- [6] Prasad BN, Saini JS. Effect of artificial roughness on heat transfer and friction factor in a solar air heater. *Solar Energy* 1988;41:555-60.
- [7] Gupta D, Solanki SC, Saini JS. Heat and fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates. *Solar Energy* 1993;51:31-7.
- [8] Lanjewar A, Bhagoria JL, Sarviya RM. Heat transfer and friction in solar air heater duct with W-shaped rib roughness on absorber plate. *Energy* 2011;36:4531-41.
- [9] Ebrahim Momin A-M, Saini J, Solanki S. Heat transfer and friction in solar air heater duct with V-shaped rib roughness on absorber plate. *Int J Heat Mass Transfer* 2002;45:3383-96.
- [10] Hans VS, Saini RP, Saini JS. Heat transfer and friction factor correlations for a solar air heater duct roughened artificially with multiple v-ribs. *Solar Energy* 2010;84:898-911.
- [11] Kumar A, Saini RP, Saini JS. Experimental investigation on heat transfer and fluid flow characteristics of air flow in a rectangular duct with Multi V-shaped rib with gap roughness on the heated plate. *Solar Energy* 2012;86:1733-49.
- [12] Alam T, Saini RP, Saini JS. Use of turbulators for heat transfer augmentation in an air duct - a review. *Renewable Energy* 2014;62:689-715.
- [13] Alam T, Saini RP, Saini JS. Heat and flow characteristics of air heater ducts provided with turbulators - a review. *Renewable Sustainable Energy Rev* 2014;31:289-304.
- [14] Kim KY, Kim SS. Shape optimization of rib-roughened surface to enhance turbulent heat transfer. *Int J Heat Mass Transfer* 2002;45:2719-27.
- [15] Hwang JJ, Liou TM. Heat transfer in a rectangular channel with perforated turbulence promoters using holographic interferometry measurement. *Int J Heat Mass Transfer* 1995;38:3197-207.
- [16] Buchlin J. Convective heat transfer in a channel with perforated ribs *Transfert de chaleur par convection dans un canal muni de pontets perforés*. *Thermal Sci* 2002;41:332-40.
- [17] Shaeri MR, Yaghoubi M. International journal of heat and fluid flow numerical analysis of turbulent convection heat transfer from an array of perforated fins. *Int J Heat Fluid Flow* 2009;30:218-28.
- [18] Sara ON, Pekdemir T, Yapici S, Yilmaz M. Heat-transfer enhancement in a channel flow with perforated rectangular blocks. *Heat Fluid Flow* 2001;22: 509-18.
- [19] Sara ON, Pekdemir T, Ersahan H. Thermal performance analysis for solid and perforated blocks on a flat surface in a duct flow. *Energy Conversion Manage* 2000;41:1019-28.
- [20] Karwa R, Maheshwari BK, Karwa N. Experimental study of heat transfer enhancement in an asymmetrically heated rectangular duct with perforated baffles. *Int Commun Heat Mass Transfer* 2005;32:275-84.
- [21] Karwa R, Maheshwari BK. Heat transfer and friction in an asymmetrically heated rectangular duct with half and fully perforated baffles at different pitches. *Int Commun Heat Mass Transfer* 2009;36:264-8.
- [22] Shin S, Kwak JS. Effect of hole shape on the heat transfer in a rectangular duct with perforated blockage walls. *J Mech Sci Technol* 2008;22:1945-51.
- [23] Nuntadusit C, Wae-hayee M, Bunyajitradulya a, Eiamsa-ard S. Thermal visualization on surface with transverse perforated ribs. *Int Commun Heat Mass Transfer* 2012;39:634-9.
- [24] Bhuiya MMK, Chowdhury MSU, Saha M, Islam MT. Heat transfer and friction factor characteristics in turbulent flow through a tube fitted with perforated twisted tape inserts. *Int Commun Heat Mass Transfer* 2013;46:49-57.
- [25] Kumar S, Saini RP. CFD based performance analysis of a solar air heater duct provided with artificial roughness. *Renew Energy* 2009;34:1285-91.
- [26] Karmare SV, Tikekar AN. Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD. *Sol Energy* 2010;84:409-17.
- [27] Boukadoum AB, Benzaoui A. CFD based analysis of heat transfer enhancement in solar air heater provided with transverse rectangular ribs. *Energy Proc* 2014;50:761-72.
- [28] Yadav AS, Bhagoria JL. A CFD (computational fluid dynamics) based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate. *Energy* 2013;55:1127-42.
- [29] Yadav AS, Bhagoria JL. A CFD based thermo-hydraulic performance analysis of an artificially roughened solar air heater having equilateral triangular sectioned rib roughness on the absorber plate. *Int J Heat Mass Transf* 2014;70: 1016-39.
- [30] Yadav AS, Bhagoria JL. A numerical investigation of square sectioned transverse rib roughened solar air heater. *Int J Therm Sci* 2014;79:111-31.
- [31] Chaube A, Sahoo PK, Solanki SC. Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. *Renew Energy* 2006;31:317-31.

- [32] Gandhi BK, Singh KM. Experimental and numerical investigations on flow through wedge shape rib roughened duct. J Inst Eng India J 2010;90:13-8
- [33] Hwang JJ, Lia TY, Liou T-M. Effect of fence thickness on pressure drop and heat transfer in a perforated-fenced channel. Int J Heat Mass Transfer 1998;41:811-6.
- [34] Yadav AS, Bhagoria JL. A CFD (computational fluid dynamics) based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate. Energy 2013;55:1127-42.
- [35] Patankar SV. Numerical heat transfer and fluid flow. Washington DC: Hemisphere; 1980.
- [36] Lewis MJ. Optimizing the thermohydraulic performance of rough surfaces. Int J Heat Mass Transfer 1975;18:1943-8.
- [37] Sukhmeet Singh. Bikramjit Singh, V.S. Hans, R.S. Gill. CFD (computational fluid dynamics) investigation on Nusselt number and friction factor of solar air heater duct roughened with non-uniform cross-section transverse rib. Energy xxx (2015) 1-9.



Lokesh Varshney obtained his doctoral degrees in Mechanical Engineering from Indian Institute of Technology, Roorkee, Uttarakhand.

He was the former chairman of The Institute of Engineers (India) Pantnagar centre (Uttarakhand). He has a total research and teaching experience of more than 30 years. He is currently working as professor in the Mechanical Engineering department of College of Technology, G. B. Pant University of Agriculture and Technology, Pantnagar and has published a number of research papers. He has a vast experience of guiding M. Tech. and Ph. D. students.

BIOGRAPHIES



Ashok Singh Yadav obtained his **bachelor's degree (B.Tech.)** in Mechanical Engineering from Moradabad Institute of Technology, Moradabad (U.P.), in the year 2011 and M. Tech. in Thermal Engineering from G. B. Pant University of Agriculture and Technology, Pantnagar, Uttarakhand in the year 2015. He was former lecturer in the Mechanical Engineering department of Apex Institute of Technology, Rampur, (U.P.) His areas of interest are finite element analysis and renewable energy.



Tarun Singh Samant obtained his **bachelor's degree (B.Tech.)** in Mechanical Engineering from Kumaon Engineering College Dwarahat (Almora), Uttarakhand, in the year 2012 and M. Tech. in Thermal Engineering from G. B. Pant University of Agriculture and Technology, Pantnagar, Uttarakhand in the year 2015. His areas of interest is renewable energy and heat transfer.