Thermo-hydraulicperformance of anartificially roughened SAH.

Anshuman Silori
Department of Mechanical Engineering, DIT University, Dehradun (UK)
Mob no.-9690123679
anshuman.silori@gmail.com

Jitesh Rana
Department of Mechanical Engineering, DIT University, Dehradun (UK)
Mob no.-9808410744
Jiteshrana95@gmail.com

Rajesh Maithani
Department of Mechanical Engineering, DIT University, Dehradun (UK)
Mob no-9760890622
Rajeshmaithani1@gmail.com

Sunil Chamoli
Department of Mechanical Engineering, DIT University, Dehradun (UK)

Abstract

In the article, CFD investigation has been carried out to maximize the heat transfer using roughness geometry in the shape of wavy delta winglets. The parameters varied in the current investigation is number of waves (ϕ) from 3–7.It has been found that Maximum enhancement in Nusselt number is found to be 131 for 5 waves winglets. For maximum value of friction factor at Reynolds number 0.11 for 7 waves.The data collected by the CFD analysis for a smooth duct was compared with that of experimental results and was found to be in good agreement. Hence the paper also verify the good scope of CFD in the future. It has also been recommended to use Winglets for enhancing the heat transfer coefficient.

IJSER

Introduction:

Solar Air heaters is one of the system which uses solar energy for heating, drying of concretes and seasoning of timbers etc. It has a very simple design which consists of an absorber plate to capture the photons and with the help of convection transfer this energy to air. But since there is formation of laminar sub-layer in the vicinity of the duct therefore a simple solar air heater has low heat transfer coefficient[1]. Various researches has been carried out and investigated that there islarge increase in heat transfer by roughness element,as it creates turbulence in the laminar sub-layer and there is greater mixing. On the other hand using roughness also increases friction thus increase of pumping power. Various

Investigation has been carried out using experimental as well as numerical methods. Arulanandam[2] investigated using CFD that if absorber plate is used of the low conductivity material such as polythene ,plastics etc. is used in solar air heater appreciable efficiencies can still be achieved.Chaube et al.[3] added roughness in the form of ribs and analyzed in Ansys 6., from his conclusion it seems that k-w turbulent model seems to be in close agreement with the result obtained from experiment.Varol and Oztop [4] using CFD carried out a numerical investigation for natural convection heat transfer and fluid flow inside the duct. It was concluded that heat transfer rate is maximum for wavy winglets than flat winglets.Kumar and Saini [5]in his CFD investigation found out that, Nusselt number is increasing with increase in Reynolds number and friction factor is decreasing with increase in Nusselt Number for all set of relative roughness heights.Soi et al. [6] in his investigation using CFD by roughness element of k-shaped concluded that deviation between CFD values and experimental was found to be 15% for Nusselt number and 20% for friction factor.Karmare and Tikekar [7] using CFD investigated that roughness element below the absorber plate increases the heat transfer appreciably. Rajput [8] on his CFD simulation using U-type turbulators investigated that ribs completely obstructed the laminar sub layer formed.Giri[9] investigated that

ISSN 2229-5518

Renormalization group,k-ξ model seems to be in good agreement with experimental one with high aspect ratio. Sharma and Thakur in his CFD investigation concluded that combination of swirling motion and detachment and reattachment of the fluid was responsible for increase in heat transfer.Gandhi and Singh [10] using CFD analysis investigated that with roughness element in the form of rib gives much flatter velocity profile while accompanying with high turbulence.

Nomenclature:

P Pitch of the rib (m)

e/D Relative roughness height

P/e Relative roughness pitch

Re Reynolds number

e Rib height (m)

η Thermo-hydraulic performance parameter

h Convective heat-transfer coefficient (W/m^2K)

f Friction factor of roughened duct

D Hydraulic diameter of duct (m)

N_g *Number of gaps*

Nu Nusselt number of roughened duct

Nu_s *Nusselt number of smooth duct*

2. CFD Investigation

In this article, CFD is used to investigate to analyses three-dimensional flow using numerical techniques. Since there is formation of secondary flow, 3-d investigation has been carried out to get the accurate results. In the current investigation, roughness geometry in the form of wavy winglets has been optimized.

2.1. Grid Generation:

In the present investigation, a 3-d fluid flow using wavy delta winglet as roughness element is used in the underside of the absorber plate. A 3-d numerical CFD investigation on Nusselt number and friction factor of solar air heater duct roughened with non-uniform cross-section transverse rib has been previously successfully investigatedby Sukhmeet Singh [11]. Hence, in the present investigation roughness element in the form of wavy delta winglet has been used. A typical diagram of the fluid domain is shown in Fig.1. In the present investigation, waves are varied in order of 3-7. Each of the geometry is simulated for five different Reynolds number in the order of 3000-18000 as reported by Gupta et al[12] keeping relative roughness height(e/D), relative roughness pitch(P/e) constant. The geometrical parameter for artificially roughened solar air heater is shown in Table 1. The range of operating parameters has been used in Table 2. Ansys 15.0 is used to generate fine mesh at vicinity of roughness and coarse mesh at the rest of the area.The fluid domain has been messed with uniform and non-uniform triangular mesh.

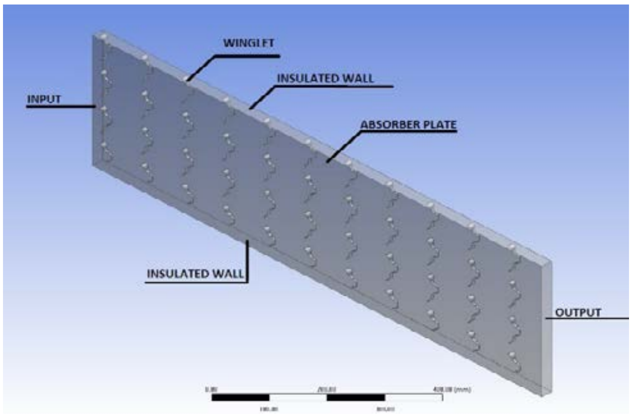

Fig.1. A typical diagram of the fluid domain used for CFD Analysis

Table 1. Geometrical Parameters of roughness

Geometrical parameter of artificial roughness for CFD analysis	
Test section length	1100mm
Winglet height	13mm
No. of waves	3,5,7,9
Angle of attack	60
Thickness of winglet	0.2mm
Hydraulic Diameter	46.1538
Area of cross section	300mm×25mm
Winglet length	35 mm
Number of gap	3
Transversal pitch	25 mm

Table 2.Operating Parameters

No.	Flow/winglet parameters	Range
1	Reynolds number (Re)	4000 -18000 (Six values)
2	Relative longitudinal pitch (P/H)	3(Fixed value)
3	Relative winglet cord length (c/H)	1.4 (Fixed value)
4	Relative blockage height (b/H)	0.5 (Fixed value)
5	Angle of attack (α)	60° (Fixed value)
6	Number of waves	3,5,7

2.2 Governing equations

IJSER

The flow pattern and heat transfer in any fluid domain can be describe by three basic equations namely Energy equation, Momentum equation and Continuity equation.It has been found in the investigation that roughness element in the form of winglet increases the heat transfer due to the turbulence at the expense of pressure drop. Therefore, thermal hydraulic performance has been calculated. Thermal performance of solar air heat concerns with the heat transfer while the hydraulic performance gives the pressure drop in the duct. Webb and Eckert [13] suggested a method for calculating thermo-hydraulic performance which evaluates the enhancement in the heat transfer compared with the smooth duct.

$$\eta = \frac{Nu/Nu_s}{\left(f/f_s\right)^{1/3}} \quad (1)$$

Nusselt number for smooth duct of a solar air heater and can be obtainedbyDittus–Boelter equation (Eq. 2) and friction factor is obtained by Blasius equation (Eq. 3).

$$Nu_s=0.023*Re^{0.}8*Pr0^{.4} \quad (2)$$

$$f_s=0.085*Re^{-0.25} \quad (3)$$

2.3 Boundary conditions

No-Slip Boundary conditions are assumed at the wall .A constant heat flux of 1000 W/m^2 is assumed at the absorber plate. The working fluid (air) is assumed to be at constant mean temperature of 300K .Velocity inlet condition is specified in the inlet of the duct and given velocity according to the selected Reynolds number. The range of the air velocity lies between 1-9.The pressure outlet boundary condition is specified at the outlet boundary conditions and rest all the wall are considered to be insulated.

2.4 Turbulence modelling

A detail investigation between choice of turbulence model with five different turbulence models, namely Standard k–ξ model, Renormalization-group k–ξ model, Realizable k–ξ model, the predictions were compared withavailable experimental data and the Renormalization-group k–ε model was found to be the best one. Therefore, Renormalizationgroup(RNG) k–ε turbulence model with ‘enhanced wall treatment has been selected for present study to predict the flow and forcedconvectioncharacteristics of a fully developed turbulent flowthrough artificially roughened solar air heater. Transport equationsand other details of each turbulence model can be found in [14].

2.5. Solution method

In the present numericalstudy RNG k–ξ turbulence model with ‘enhanced wall treatment’ isused. In the discretization of governing equations, SIMPLE (semiimplicitmethod for pressure linked equations) algorithm is usedin pressure–velocity coupling as suggested by Kumar and Saini[15]. The governing equationsfor mass and momentum conservation are solved with a segregated

3.4. Validation of CFD model

Tests for the confirmation of grid independence of the proposed model is first carried out by increasing the grid density until further enhancement shows a difference of less than 1% in two consecutive sets of results and then numerical results are validated by comparing with available analytical solutions or widely accepted numerical results.

3. Results and Discussions:

The numerical analysis has been performed for artificially roughened solar air heater with rib winglets on the absorber plate and the results are presented in this section. The results have been compared with those obtained in case ofsmooth ducts operating under similar operating conditions to discuss the enhancement in heat transfer and friction factor on account of artificial roughness

3.1 Heat Transfer

It is observed from Fig. 2thatthe average Nusselt number of the roughened duct with respect to the smooth duct, increase with increasing values of Reynoldsnumber in all cases as expected. The velocity increases with

IJSER

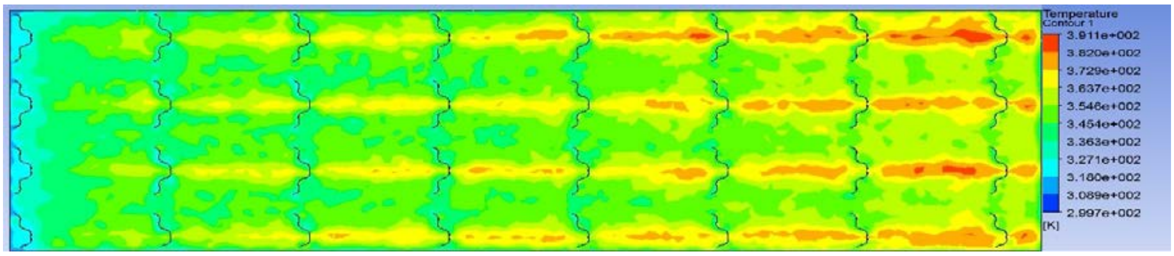

Fig.2 A typical figure of temperature contour for a 5 wave winglets

Increasing value of Reynolds number, which results in enhancedheat transfer rate. As the Reynolds number increases the roughness elements begin to project beyond the laminar sub-layer. Laminar Sub-layer thickness decreases with an increase in theReynolds number. In addition to this there is local contributionto the heat removal by the vortices originating from the roughnesselements. Therefore Nusselt number increases with Reynolds number as shown in Fig.3. The relative roughness pitch of 3 and Number of waves 5 provide maximum value of average Nusselt number. The temperature contour of the roughness is shown in Fig.2.

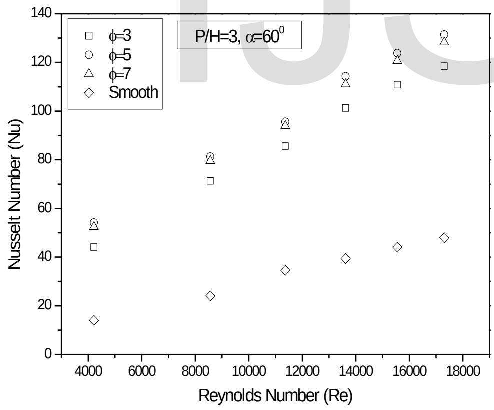

Fig.3. Variation of Nusselt Number for different Number of Waves

3.2. Flow friction

The flow blockage due to the presence of the equilateral triangular sectioned rib is a vital factor to cause a high pressure drop. In this case all configurations of investigated ribs cause a regular boundary layer separation and re-attachment. This is reflected in an increased pressure drop and therefore an increased friction factor in the roughened duct compared to a smooth duct. With higher blockage of the duct of a solar air heater, pressure drop increases with no significant increase in heat transfer. From this fig.4, it is seen that the average friction factor of the roughened duct with respect to the smooth duct, tends to decrease as the Reynolds number increases in all cases as expected because of the suppression of laminar sub-layer. The highest friction factor is obtained at relative roughness pitch of 3 and Number of waves 7.A typical velocity contour is shown in Fig. 5

IJSER

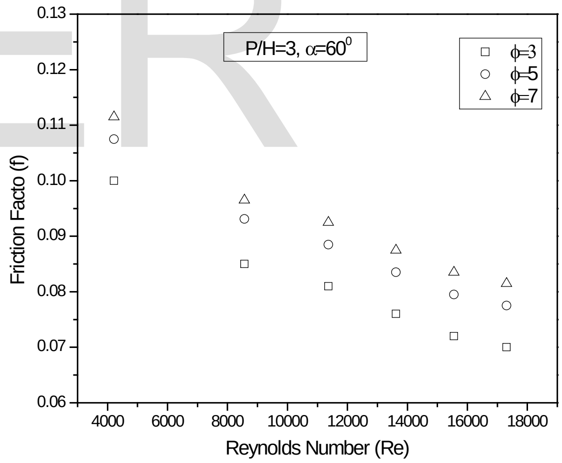

Fig.4 Variation of Friction Factor for different Number of Waves.

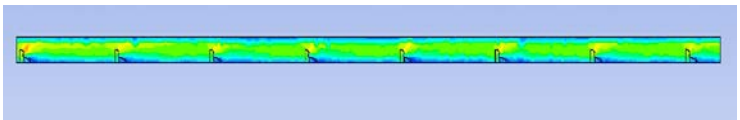

Fig.5 A typical figure of velocity contour for a 5 wave winglets

Thermo-hydraulic performance:

The Thermo-hydraulic performance of the selected artificially roughened duct is shown in Fig.6.It can be seen that the maximum Thermo-hydraulic performance is obtained for

Relative longitudinal pitch of 3 and number of waves 5.

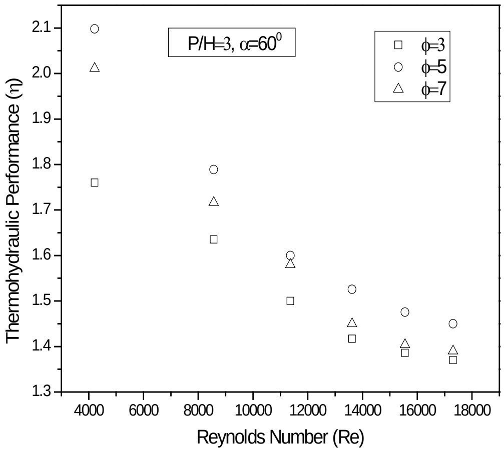

Fig.6. Variation of Thermo-hydraulic performance for different Number of Waves.

.3.4. Conclusions

In this article a three -dimensional CFD model of an artificially roughened solar air heater having delta winglets on the absorber plate has been proposed and used to predict the heat transfer and flow friction characteristics. Using this approach a detailed study is performed to analyze the impact of parameters on the thermal and hydraulic performance of an artificially roughened solar air heater by varying number of waves.

The major conclusions of this article are as follows:

1. The average Nusselt number tends to increase as the Reynolds number increases in all cases.
2. The maximum Nusselt number has been found to be 131 corresponds to relative roughness pitch (P/e) of 3 in the range of parameters investigated.
3. The maximum friction factor has been found to be 0.11 corresponds to relative roughness pitch (P/e) of 7.

The CFD velocity contour, temperature contours are analyzed to draw a conclusion of obtaining higher heat transfer enhancement.

References

[1] Twidell J, Weir T. Renewable energy: sources. 2nd ed.. New York: Taylor & Francis; 2006.

[2] Arulanandam SJ, Hollands KGT, Brundrett E. A CFD heat transfer analysis of the transpired solar collector under no-wind conditions. Solar Energy 1999;67(1–3):93–100.

[3]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. Renewable Energy 2006;31:317–31.

[4]Varol Y, Oztop HF. A comparative numerical study on natural convection in inclined wavy and flat-plate solar collectors. Building and Environment 2008;43:1535–44.

[5]Kumar S, Saini RP. CFD based performance analysis of a solar air heater duct provided with artificial roughness. Renewable Energy 2009;34:1285–91.

[6]Soi A, Singh R, Bhushan B. Effect of roughness element pitch on heat transfer and friction characteristics of artificially roughened solar air heater duct. International Journal of Advanced Engineering Technology 2010;1(3): 339–46.

[7]Karmare SV, Tikekar AN. Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD. Solar Energy 2010;84:409–17.

[8]Rajput RS. Heat transfer and fluid flow analysis of inverted U-type turbulator in a solar air heater duct by CFD. M.Tech. dissertation. MANIT Bhopal, India;

[9]Giri AK. CFD analysis of reattachment point, heat transfer and fluid flow of a solar air heater duct provided with artificial roughness. M.Tech. dissertation.

[10]Gandhi BK,Singh KM .Experimental and numerical investigations on flow through wedge shape rib roughened duct. The Institution of Engineers(India) Journal—MC 2010;90:13–8January.

[11]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 March 2015

[12]D. Gupta, S.C. Solanki, J.S. Saini, Thermohydraulic performance of solar air heaters with roughened absorber plates, Sol. Energy 61 (1) (1997) 33–42.

[13]R.L. Webb, E.R.G. Eckert, Application of rough surface to heat exchanger design, Int. J. Heat Mass Transfer 15 (9) (1972) 1647–1658.

[14]B.E. Launder, D.B. Spalding, Lectures in Mathematical Models of Turbulence, Academic Press, London, England, 1972.

[15] S. Kumar, R.P. Saini, CFD based performance analysis of a solar air heater duct provided with artificial roughness, Renewable Energy 34 (2009) 1285–1291]

IJSER

IJSER