Nusselt number and friction factor behaviour of circular wavy transverse ribs artificially roughened solar air heater

Sheshang S Chandel¹, Ajay Verma², Maheandera Prabu Paulraj³,
Santosh Kumar Sahu⁴

¹²Dept of Mechanical Engineering, Shri Shankaracharya Group of Institutions, Bhilai- 490020, C.G, India.
³⁴Fluid Mechanics and Machinery Laboratory, Discipline of Mechanical Engineering, Indian Institute of Technology Indore, Simrol- 453552, Indore, Madhya Pradesh, India.
E-mail: csheshang1611@gmail.com

Abstract. The present study deals about the fluid flow and heat transfer behaviours of circular wavy wall transverse rib artificially roughened SAH. The circular wavy ribs are used in the solar air (SAH) heater with different undulation (N) and constant amplitude (A) below absorbing plate at testing zone for the present numerical investigation. Heat transfer behaviour varies about number of undulation with the intention that the optimum undulation of circular wavy wall transverse rib is necessary for better performance of SAH. The FLUENT v 14.5 is used to solve the governing equations (mass, momentum and energy). The RNG (k-ε) model is utilized to capture the turbulent effects and also the SIMPLE algorithm is used to couple the pressure and velocity. The geometrical parameter, pitch (p/e) ranges from 7.14 to 14.28 and relative roughened height (e/D) ranges from 0.021 to 0.042 are taken. The Reynolds numbers (Re) is varied between 3800 and 18000 for the constant heat (q) flux of plate 1000 W/m². The results, Nusselt number (Nu/Nus) enhancement, and friction (f/fr) factor performance are presented.

Key words: solar air heater, circular wavy wall, CFD, friction factor, Nusselt number

1. Introduction
Solar air (SAH) heater is the low cost heat transfer mechanism which absorbs heat from sun rays and transfers as thermal energy. Solar air (SAH) heaters have a plate placed at upper surfaces of duct which takes temperature from aluminium plate and bottom surfaces are taken insulated to avoid the heat losses. The duct is enclosed with inlet and outlet. The air flows inside the duct takes the heat energy from the aluminium absorbing plate and transfer to a required place by using electric blower or fan. Now days, solar air (SAH) heater are mainly using for space heating application, process heating application, ventilation applications and many more. Various turbulence creators in the terms of different roughened surfaces, rib, fins and vortex rings are used for enhancing overall heat transfer reported by Yadav et al. [1]. Prasad and Saini [2] conducted the experiment by using circular wire which are placed close to the hot test shield and captured in to flow pattern near to the wall. And they found positive results in thermal performance with penalty in pressure drop. Aharval et al. [3] introduce inclined ribs geometries and reports enhancement of Nusselt (Nu) number for angle of (α) attack effects. The Fluent software help to predict the effect inside solar air heater (SAH) channel and
a range of two-dimensional artificially roughened geometries by the Chaube et al. [4] and A S Yadav [5]. The results shown on their studies are related to thermal performance in different shapes and parameter of roughness by using commercial computational (CFD) software FLUENT. S Kumar and R P Saini [6] used three-dimensional computational domain chosen for the study of fluid and heat characteristics of arc shape ribs. Hatami et al. [7] uses the cavity of circular wavy shape for natural convection and their results was positive for local Nusselts (Nu) number. In this present investigation, the wavy ribs roughened surface is taken for solar air drying applications. The wavy roughened surface is placed near the absorbing (aluminium) plate (upper wall of testing section). The forced turbulent effects are captured by CFD code (fluent version14.5).

2. Problem definition

The two-dimensional rectangular SAH duct is considered for study the variations in flow pattern and heat transfer of circular wavy ribs artificially roughened solar air heaters. In present investigation the wavy circular ribs are introduced as a roughened surface and placed on a bottom of the absorbing plate. The flow parameters are taken similar as A S Yadav and J L Bhagoria [8] for predicting the behaviours of working fluid flow. Entrance section and exit section are taken 5√/WH and 2.5√/WH recommended by ASHRE Standard 93-2003[9]. The parameters of duct for entrance section length (L₁) 225mm, tested section length (L₂) 115mm and length (L₃) 115mm for exit section are taken with 100mm width and 20mm height. Computational domain with circular wavy walls is show in figure 1. Aluminium absorbing plate of 0.5 mm thick is placed at top wall of testing zone with GI wavy circle ribs. The pitch ranges between two ribs are taken as constant at 10mm and the three values are considered for rib height 0.7mm, 1.0mm and 1.4mm for analysis. The circular wavy rib function is taken from M Hatami, D Song and D Jing [7] and the function is

\[ r = r_{in} + A \cos(N(\hat{\xi})) \]

Here, \( r_{in} \) base radius of circle, \( A \) is amplitude taken as constant of 0.2mm and \( N \) is number of undulation which is in range from 4-12. The \( \hat{\xi} \) taken as rotational angle. The fluid and material properties are taken at room temperature. List of properties of aluminium and air are mentioned in Table 1 which are using in present study.

3. CFD Simulation

Computational Fluid (CFD) Dynamic code fluent (version 14.5) is used to capture the flow behaviours of fluid and characters of heat transfer for 2D artificially roughened solar air collecter; by using finite volume (FVM) scheme, general Navier-Stokes equations (mass and momentum) and governing equation for energy are solved. Steady condition is considered for present investigation, assuming fully developed and turbulence effect. The properties of present material and fluid are taken as constant throughout the investigation. No radiation effect and no slip at wall are assumed for analysis. The FLUENT v14.5 (CFD code) is used in present numerical investigation. The 2-D geometry is developed by using design modular of ANSYS; mesh generation is done by using ANSYS ICEM HEX tool which is present at ANSYS Workbench 14.5v and then this meshing model are used for further simulation process. The assemble desktop is used for simulating and the configuration, processor- Intel(R) Xeon(R) processor, Ram-32 GB, NVIDIA Quadro K620- with Professional 64-bit has been used for the present study.

3.1 Grid Generation

The different cell shapes with different element size are generated and tested to capture the turbulent effects on present study and are mentioned in figure 2. The triangle mesh, triangle and square combined mesh and square mesh are generated for proper capturing flow behaviour of fluid on computational domain.

| Properties                        | Working fluid (air) | Absorber plate (aluminum) |
|-----------------------------------|---------------------|---------------------------|
| Density, \( \rho \) (kg m\(^{-3}\)) | 1.225               | 2719                      |
| Specific heat, \( \text{Cp} \) (J kg\(^{-1}\) K\(^{-1}\)) | 1006.43             | 871                       |
| Viscosity, \( \mu \) (N m\(^{-2}\)) | 1.7894e-05          | -                         |
| Thermal conductivity, \( k \) (W m\(^{-1}\) K\(^{-1}\)) | 0.0242              | 202.4                     |
(a) Schematic diagram of computational domain, (b) Circular wavy rib with pitch 10, (c) Circular wavy rib with amplitude (A=0.2mm) and angle of rotation (ζ=45°).

**Figure 1.** Artificially roughened SAH (circular wavy ribs).

**Table 2.** Grid test for present analysis

| Mesh Size (mm) | No. of elements | Nu    | % error | f      | % error |
|---------------|-----------------|-------|---------|--------|---------|
| 0.26          | 137435          | 102.86| 0.012072| 104.64 | 0.012396| 4.13    |
| 0.25          | 145190          | 104.64| 0.012396| 106.37 | 0.012611| 2.61    |
| 0.24          | 160534          | 106.37| 0.012611| 108.07 | 0.012721| 1.70    |
| 0.23          | 175256          | 108.07| 0.012721| 108.85 | 0.012883| 0.86    |
| 0.22          | 191542          | 108.85| 0.012883| 110.88 | 0.012972| 1.25    |
| 0.21          | 208676          | 110.88| 0.012972|        |         |
Figure 2. Different cell shape with element size 2.3 mm, (a) 464594 cells, (b) 372958 cells, (c) 276010 cells, (d) 175256 cells.

Generated meshes are tested with varying element size from 2.1 mm to 2.6 mm. The square mesh performance, mesh type (d) found better as compare to mesh types (a), (b) and (c). Uniform square mesh type (d) with 2.3 mm element size are found suitable in present study. The less than 1% error is considered for Nusselt (Nu) number and for (f) friction factor. All grid independence tests are done for rib height 1.4mm, pitch distance 10 mm with undulation (N) 8.

3.2 Governing Equation

Here the mass, momentum and the energy equations have been solved based on initial and boundary condition by using the commercial FVM-CFD coding. The governing equations are presented in the form of finite difference mode as below.
Mass
\[ \frac{dp}{dt} + \nabla \cdot (p \vec{v}) = 0 \]  \hspace{1cm} (1)

Momentum
\[ \frac{d(p\vec{v})}{dt} + \nabla \cdot (p\vec{v}\vec{v}) = \rho \vec{g} - \nabla p + \nabla(\vec{T}) + \vec{f} \]  \hspace{1cm} (2)

Energy
\[ \frac{d(pE)}{dt} + \nabla \cdot (\nu (pE + p)) = \nabla \cdot (k\nabla T + (\vec{t} \cdot \vec{v})) \]  \hspace{1cm} (3)

Some non-dimensional numbers are involved in our present study: Reynolds number, Nusselt number & Friction factor which can obtain by:

Reynolds Number: \[ \text{Re} = \frac{\rho v D_h}{\mu} \]  \hspace{1cm} (4)

Here, \( D_h \) is a hydraulic diameter.

Average Nusselt (Nur) number is calculated for roughened absorbing surface which can be obtained by:

\[ \text{Nu} = \frac{h D_h}{k} \]  \hspace{1cm} (5)

Here, \( h \) is convective heat transfer coefficient.

Friction (fr) factor are depended upon the pressure (\( \Delta P \)) drop due to given roughened absorbing surface across the testing section. It can be calculated by:

\[ f_r = \left( \frac{\Delta P}{2 \nu v^2} \right) \]  \hspace{1cm} (6)

Here, \( \Delta P \) is pressure drop at testing section.

The two dimensional rectangular geometry at the x-y plane having inlet and the outlet at either end sides, the duct is separated into different section; entrance, testing and exit section. Testing section upper surface is taken as absorber plate (aluminum material). The working temperature of fluid and material are taken at ambient temperature. So the property of fluid and the materials are also taken at ambient temperature. The inlet fluid temperature is taken 300 degree Kelvin.

### Table 3. Boundary conditions

| Boundary conditions                  |
|--------------------------------------|
| Duct inlet                          | velocity-inlet          |
| Duct outlet                          | Pressure-outlet at atmospheric pressure |
| Absorbing Plate                      | wall (constant heat flux) 1000W/m² |
| Bottom wall                          | Insulated (adiabatic)/ no slip |

The wide ranges of models are present in Ansys (FLUENT) for capturing the turbulent effect. The selection of model doesn’t depend upon only the fluid flow; the accuracy and the computational time satisfaction also needed. Figure 3 (a) describes the results of various turbulent models and compared their individual results with the predicted fluid flow for smooth (s) duct which was calculated from Dittus-Boelter correlation eq. (8). The renormalization (RNG) group k- \( \epsilon \) model found less error of 2.5% from predicted values as compare to the others turbulent models.

### 3.3 Model selection and Validation

Numerically approach to fluid dynamic and thermal analysis problem are solved with the help of CFD (FLUENT v14.5) code are common nowadays. Present, investigation involves wide range of turbulent models which are SST model, k- \( \omega \) model, standard k- \( \omega \), standard k- \( \epsilon \), k- \( \epsilon \) RNG model, and k- \( \epsilon \) realizable models for the present analysis and the best model is chosen for analysis. The predicted results of smooth duct are also validated. After comparing, the turbulent model renormalization (RNG) group k- \( \epsilon \) is found to be best for the investigation. And also the existing results of semi-
circular rib SAH [8] have been taken to compare with present roughened geometry for validation and the similarity solution obtained and it have been presented in figure 3. (b).

Dittus–Boelter equation: $\text{Nu}_f = 0.023\text{Re}^{0.8}\text{Pr}^{0.4}$ (7)

![Figure 3. (a) Selection of turbulent model, (b) Validation.](image)

The validation result describes and also comparisons of the prediction fluid flow result of smooth duct. Friction ($f_c$) factor of smooth duct can be predicted by using Blasius equation (8).

Blasius Equation: $f_c = 0.0791 \text{Re}^{-0.25}$ (8)

The turbulent kinetic energy ($\text{K.E}$) with the RNG ($k − \varepsilon$) transport model [10] as defined as

$$\frac{\partial}{\partial \xi_1} (\rho k u_i) = \frac{\partial}{\partial \xi_1} \left( \alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial \xi_1} \right) + G_k - \rho \varepsilon$$ (9)

$$\frac{\partial}{\partial \xi_1} (\rho \varepsilon u_i) = \frac{\partial}{\partial \xi_1} \left( \alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial \xi_1} \right) + C_{1_\varepsilon} \frac{\varepsilon}{k} (G_k) - C_{2_\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon$$ (10)

Where, $G_k$ is the (K.E generation) energy due to mean velocity gradient, $G_k$ energy developed by average velocity gradient as defined as

$$G_k = -\rho u_i \frac{\partial u_i}{\partial \xi_1}$$ (11)

$\mu_{\text{eff}}$ is the effective turbulent ($\tau$) viscosity, $\mu_{\text{eff}}$ is define as

$$\mu_{\text{eff}} = \mu + \mu_t$$

Where $\mu_t$ is the turbulent ($\tau$) viscosity, $\mu_t$ is define as

$$\mu_t = \rho C_\mu \frac{\varepsilon^2}{k}.$$ (12)

Where $C_\mu$ is Constant, $\alpha_k$, $\alpha_\varepsilon$ represents Prandtl number for kinetic energy $k$ & dissipation, $\varepsilon$.

The present constant values are having default values form Launder and Spalding 1972 [11]: $C_{1_\varepsilon} = 1.42, C_{2_\varepsilon} = 1.68, \ C_\mu = 0.0845, \ \alpha_k = 1.39$ and $\alpha_\varepsilon = 1.39$.

The turbulent model RNG K-$\varepsilon$ find more appropriate to capture the turbulent effect for this investigation. The conversing residual for continuity equation are taken as 10-3 and 10-6 for momentum and energy. The SIMPLE scheme is selected for pressure- velocity coupling and second orders upwind are selected for velocity and energy [Paulraj et al. 12 and 13]. The constant velocity
inlet is taken at entrance zone. Atmospheric pressure is selected at exit of the duct and plate is maintained at constant heat (q) flux of 1000W/m².

![Graph](image1)

Figure 4. (a) Nu, (b) f, variations for different relative e/D and P= 10mm.

![Graph](image2)

Figure 5. Turbulent Kinetic Energy of N=8 at Re 18000

![Graph](image3)

Figure 6. Velocity Magnitude of N=8 at Re 18000.
4. Result and Discussion

Average Nusselt (Nu) number and friction (f) factor are presented and comparing with various parameters for showing the optimum effects of thermal characteristics. The comparisons are done with existing results of the semi circular rib and the smooth (s) SAH duct. From the figure 4 (a), describe the deviations in Nu value and pressure drop (\(\nabla P\)) by various e/D in different Reynolds (Re) number. The Reynolds (Re) number 18000 gives the maximum Nu value with minimum friction in all the set of parameters taken. It is also noticed that the Nu value increases when e/D, roughness height increased. However, when we seen the higher Reynolds (Re) number, there is no huge variation in (Nu) Nusselt number noticed. From figure 4 (b) describes the influence of e/D on friction factor and the range from 0.042 to 0.021 relative roughened surface at different (Re) Reynolds number and compared the results with predicting smooth (s) duct results. The friction factor gives higher values when the relative roughened surface get increase and also lower Reynolds (Re) number values give increased pressure drop values because viscous layer get suppressed. Figure 5 & figure 6 describing the contour of (\(\tau\)) turbulent K.E and velocity magnitude. The maximum turbulent (K.E) kinetic energy noticed at wavy ribs leading edge (except first rib) and which indicates that the higher turbulence occurs at near wavy ribs tailing edge. Also the K.E decreases along the flow direction. From figure 6, the higher velocity (\(\upsilon\)) magnitude is noticed at wavy ribs leading side and lower velocity magnitude observed at tailing side. This velocity gets affected because of wavy surface and the low pressure regions take place at tailing edge. Turbulent kinetic energy (K.E) & dissipation increased due to increases of intensity of the turbulent. Figure 7 (a) describe the Nu and undulation (N) in different Reynolds (Re) number. The undulation (N) of value 4 gives lower Nusselt (Nu) number as compare to higher value for undulation (N) of 8 and 12 for Reynolds (Re) number range 3800-5000 because of undulation effect of wavy rib. It is also seen at figure 7 (b) the friction (f) factors for undulation (N) of value 4 are lower for Re 3800 and 5000 as compare undulation (N) value 8 and 12. The lower friction (f) factor gives the lower effect of heat transfer. The higher Reynolds (Re) number of range 8000- 18000 gives higher Nusselt (Nu) number and lower friction as compare to other values of N. Figure 8 shows the fluid behaviors in presence of circular wavy rib. The turbulence is taking place at the tailing side of the wavy rib.

![Figure 7](image-url)
Figure 8. Stream line for N=8 at Re-18000.

Figure 9. (a) Nusselt number (Nur/Nus) enhancement and (b) Friction factor (fr/fs) enhancement comparison between wavy rib and Semi-circular.

Figure 9(a) describes the Nusselt number (Nu_r/Nu_s) enhancement of wavy rib and semi-circular rib. The wavy ribs find fewer enhancements variation for lower Reynolds (Re) number of 3800-5000 because of effect of undulation (N). It is also noticed at Figure 9(b) the friction factor (f_r/f_s) enhancement of wavy ribs are having less enhancement values as compare to the semi-circular rib. The enhancement values of the wavy rib are higher for Reynolds (Re) number range 8000-18000 with the penalty of higher friction factor. The maximum Nusselt number (Nu_r/Nu_s) enhancement found for Reynolds (Re) number is 1.055 times of semi-circular ribs at Re 8000 with the friction factor (f_r/f_s) enhancement of 1.04 times of semi-circular ribs.

5. Conclusion
Circular wavy ribs are placed near to the absorbing surface for evaluating the friction factor and Nusselt number effect in two-dimensional rectangular channel flow. Reynolds (Re) number, relative roughened (e/D) height, relative roughened (P/e) pitch, and undulation (N) parameters are taken and compared. The height of ribs disturbed the fluid flow pattern, hence gives the higher heat transfer characters but with the penalty of pressure drop. The relative roughened height 0.042 at Reynolds (Re) number 18000 for N=4 gives maximum average Nusselt (Nu) number value 119.19 with the penalty of 0.0166 friction factor. It is found, the circular wavy roughened surface placed near to the
absorbing plate with N=4 gives higher heat transfer performance for higher Reynolds number as compare to N=8 and N=12. In present investigation the e/D=0.042, N=4 and the P/e=7.14 gives higher Nusselt number (Nu/Nu,) enhancement Reynolds (Re) number range 8000-18000 as compare to the semi-circular roughness. At Re-8000 the wavy surface gives 1.055 times enrichment in the Nu as compare to semi-circular roughened surface and 2.791 times of the smooth channel SAH.

### Nomenclature

| Symbol | Description |
|--------|-------------|
| A_s   | Surface area of absorbing plate, m² |
| D     | Hydraulic diameter, mm |
| C_p   | Specific heat of operational fluid, J/kg K |
| e     | Height of the rib, mm |
| h     | Heat transfer coefficient, W/m²K |
| H     | Duct height, mm |
| I     | Intensity, W |
| k     | Thermal conductivity, W/mK |
| L     | Channel length, mm |
| L_1   | Entry zone length, mm |
| L_2   | Length of test zone, mm |
| L_3   | Length of exit zone, mm |
| m     | Mass, Kg/s |
| P     | Pitch in mm |
| P/e   | The relative roughness pitch |
| Pr    | The Prandtl number |
| Re    | The Reynolds number |
| S     | Smooth |
| T_a   | Ambient temperature, K |
| T_i   | Temperature at inlet, K |
| T_o   | Outlet Temperature, K |
| T_w   | Wall temperature, K |
| ν     | Velocity of fluid, m/s² |
| W     | Channel width, m |
| W/H   | Channel aspect ratio |

| Greek Symbol | Description |
|--------------|-------------|
| µ             | Dynamic viscosity, Ns/m² |
| ζ             | Rotational angle, degree |
| ε             | Dissipation rate |
| ω             | Specific dissipation rate |
| δ             | Transition sub-layer thickness, m |
| k             | Turbulent kinetic energy, m²/s² |
| τ             | Turbulent |
| A             | Amplitude |
| e/D           | Relative roughness height |
| f             | Friction factor |
| f_r           | Friction of roughened surface |
| f_s           | Friction of smooth surface |
| N             | Number of undulation |
| Nu            | Nusselt number |
| Nu_r          | Nusselt number for roughened surface |
| Nu_s          | Nusselt number of smooth surface |
| i             | Inlet |
| o             | Outlet |
| r             | Roughness |
| s             | Smooth |

### Dimensionless Parameter

- e/D: Relative roughness height
- f: Friction factor
- f_r: Friction of roughened surface
- f_s: Friction of smooth surface
- N: Number of undulation
- Nu: Nusselt number
- Nu_r: Nusselt number for roughened surface
- Nu_s: Nusselt number of smooth surface

### Subscripts

- i: Inlet
- o: Outlet
- r: Roughness
- s: Smooth
Reference

[1] Yadav S, Paulraj MP, Sahu SK. Proceedings of the ASME 2018 Power Conference (POWER2018); June 24-28, Florida, USA, pp. V002T12A006; 12 pages, DOI: 10.1115/POWER2018-7231.

[2] Prasad BN, Saini JS. Solar Energy 1988; 41(6):555-60.

[3] Aharwal KR, Bhupendra K. Gandhi, Saini JS. Int. J. Heat and Mass Trans 52 (2009) 5970–5977.

[4] Chaube A, Sahoo PK, Solanki SC. Renew. Energy 2006; 31:317-31.

[5] Yadav AS, Bhagoria JL. Energy 55 (2013) 1127-1142.

[6] Kumar S, Saini RP. Rene. Energy 2009; 34: 1285-91.

[7] Hatami M, Song D, Jing D. Int. J. Heat and Mass Trans 98 (2016) 758–767.

[8] Yadav AS and Bhagoria JL. Numer. Heat Trans, Part A, 65: 679–698, 2014.

[9] ASHRAE Standard 93. Method of test to determine the thermal performance of solar collector. 30329. Atlanta, GA: American Society of Heating, Refrigeration and Air Conditioning Engineers; 2003.

[10] ANSYS FLUENT 14.5. Documentation. Ansys, Inc; 2003-04.

[11] Launder BE and Spalding DB, Lectures in mathematical models of turbulence, London, New York, Academic Press, 1972.

[12] Paulraj MP, Parthasarathy RK, Taler J, Taler D, Oclon P, Vallati A, Heat Trans. Research 2018; 49(12): 1151–1170.

[13] Paulraj MP, Byon C, Vallati A, Parthasarathy RK, Heat Trans. Eng 2018; DOI: 10.1080/01457632.2018.1513629.