TEST Engineering & Management

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

Modeling the Turbulent Flow and Heat Transfer in a Vertical U-Tube

Dr. Khudheyer S. Mushatet
Mechanical Engineering Department, Engineering College Thi-Qar University, Al-Nasiriya, Iraq
khudheyersalim@gmail.com

Ayad Ali Mohammad
Technical College/Al-Mussaib, Al-Furat Al-Awsat Technical University, Babil, Iraq, ayadia1@yahoo.com

Article Info
Volume 81
Page Number: 4893 - 4902
Publication Issue:
November-December 2019

Abstract:
In this work, three-dimensional incompressible turbulent flow and heat transfer in a circular tube of U-configuration has been investigated numerically. The influences using the vertical U-tube in various curvature radius ratios on thermal and hydrodynamic fields are presented in details. The case for U-tube is tested by using the tube with different curvature radius ratios (1, 1.5 and 2). The tube surface is subjected to a constant heat flux and the air is chosen to be the working fluid with turbulent flow under a range of Re. Number (10000 to 25000). The turbulent flow and heat transfer is governed by continuity, momentum and energy equations. The effect of turbulence is treated by a k-ϵ turbulent model. ANSYS fluent code (15.0) based on finite volume method is used to get the numerical results.

The obtained results of reducing the curvature radius ratio show an increasing in both Nusselt-number and friction factor as compared with those of direct tube, for all the considered values of Re. Number. It is discovered that reducing the curvature radius ratio by 100% enhance the heat transfer by about 5%, due to the strong intensification of the secondary flows.The optimum enhancement efficiency is ranged between 1.07 to 1.099 for using the circular cross-section U-tube, with Rc=1. The present numerical results are compared with empirical correlations and verified a comparatively good agreement.

Article History
Article Received: *5 March 2019*
Revised: *18 May 2019*
Accepted: *24 September 2019*
Publication: *24 December 2019*

Keywords: *turbulent, heat transfer, U-tube, numerical investigation, curvature radius ratio*

I. INTRODUCTION

U-tubes (a 180°bend tube linked by two segments of direct tube) are usually used in diverse engineering applications, such as heat exchangers, gas turbine blades, waste heat reboilers, nuclear reactors, evaporators and steam generators.

When any fluid travelswithin a directpipe, the velocity of fluid close to the centerline is greater than that closer to the tube wall. This effect is chieflynoticeable in laminar-flow but is exact for turbulent-flow. When the tube is boundedby 180°, all fluid particlesexperiencesa centrifugal force which is radially outward, proportional to the "square of the particles velocity" and inversely-proportional to the "radius of the curvature". Therefore, the more rapidly moving particles near the tube-centerlineare wont to move towards the wall, displacing the slower fluid particles, which in turn move back toward the centerline of the axis. This effect is to overlie a Secondary-flow pattern upon the primary-flow.

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

Due to the existence of the secondary flow, the heat transfer is higher in a curved tube than in acorrespondingdirectpipe under the same fluid flow circumstances.The heat transfer mechanism is more complicated that's the reason of using numerical investigation. [1]

In the tube bend, due to local imbalance among pressure gradient and inertial forces, the flow is characterized as a secondary flow/circulation. It is mainly depend up on the curvature of the pipe which is being a characteristic length of tube cross-section, for instance the radius or hydraulic radius, and the radius of curvature.

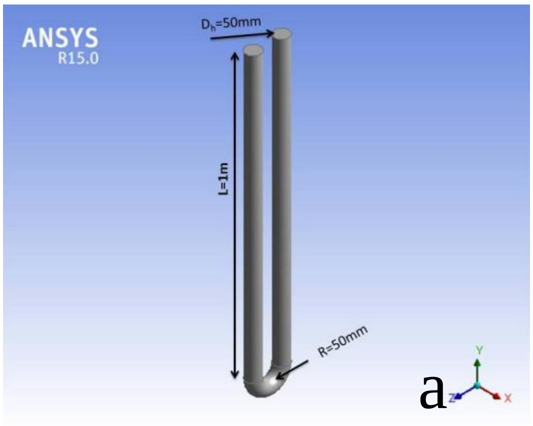
The characteristics of the secondary flow/circulation don'tonly depend on Re. Number and diameter, but also on the size of the tube and the cross-sectionalshape. In developing flows, e.g. in "the entrance region between a direct tube and a bend", they also depend on the distance from the bendentrance

The study of turbulent flow and fluid behavior in Vertical U-tube configurations has been a subject of interest among many researches. Several researchers have examined the fluid flow and heat transfer characteristics experimentally as well as computationally.

Rowe (1970) [2] the heat transfer effect in a curvy pipe was experimentally inspected, due to secondary flow there is a complete exchange of fluid in the wall and the central axis line. There work primarilydedicated on a single phase flow in various pipe turns like (180^{o}U-turn and 45^{o}S-turn) attached to the end of a long directpipe. In a180^{o}turn it is found that, from the start of the turn, the secondary-flows growth to a maximum and then reduce to a steady value. This effect is described by relating the local total pressure gradient to the production of stream-wise vortices. **Moshfeghian (1975) [3]** studied the heat transfer through an 180^{o}bendpipe. A Re. Number range from (7300 to 27000) was examined for a U-tube with a radius of the bend(9.875inch),diameter (0.875inch) and(0.052inch) wall thickness. The test segmentwas heated by a direct current through the pipe wall. Investigation of the experimental data of the 180^{o}bendshowed that the local heattransfer coefficient (h) in the upstream segment of the turn is independent of exterior position. The exterior heat transfer was far from uniform in the turn, lesser at the inside of the turn and significantlygreater at the outside, resulting in a upper mean (h) as compared to a direct tube under comparablecircumstances. The distribution of external(h)(at any cross-section in the turn) is almost symmetrical about a plane enclosing the longitudinal axis of the pipe and the radius of the turn. **Azzola et al. (1986) [4]** used laser-dopplervelocimetry measurement to examine the turbulent flow in a 180^o pipe bend. Numerical simulations have reproduced with a gratifying degree of fidelity the measured evolution of the flow. Basing on a "semielliptic" truncation of the Reynolds equations the predictions of the flow development were presented, in the main part of the flow with the "standard (k-Ɛ) effective viscosity model" used to estimate the turbulent stress field. They found that the angle between 90^o of the bend and X/D = 5, the circumferential velocity profiles showeda reversalsecondary flow which are independent of the Re. Number. **Janyanti et al. (1993) [5]** performed "CFD"investigation to study the gas element motion in 90^o and 180^o circular cross-sectional shape oftube bends. They found that the motion of the elements was affected by the secondary-flow induced in the gas-phase due to bending, which causes the smaller elements to come out of the curvature without deposition. **Sudo et al. (2000) [6]** experimentally studied developing turbulent flow in a circular cross-sectional 180°curvature. The radius of the curvature was (104mm)and aRc=4, with an upstream and downstream pipes. Measurements of the radial, longitudinal and circumferential components of mean velocity, and corresponding components of the Re. Stress were achieved with a "hot wire anemometer" at a Re. Number of (6×10^4) and at different longitudinal stations. The velocity fields of the primary-flow and secondary-flow and the "Reynolds stresses" were established in the form of contour-map. It was displayed that the flows through the 180°and the 90°turned pipes are closely equivalent in their activities to the segment upstream from an angle of 60°turn. In the segment from the angle of 90°turn, the high-velocity regions occurredcloser to the upper and lower pipe walls. **Münch and Métais (2007) [7]**examined numerically the effect of the curvature radiusratio on the flow carrying out three Large Eddy Simulations (LES) for U-bends with (Rc/D_h=3.5, 6.5 and 10.5). It was observed that the reduction of the curvature radius ratio was attended by a strong amplification of the secondarytransverse of the fluid flows. They also presented that the secondary fluid flows strength was proportional to the radial pressure gradient strength. **Clarke and Finn (2008) [8]**deliberate the improvement of interior convection heat transfer following in a heat U-bend exchanger under laminar flow conditions. Numerical investigations were conducted using (FLUENT)computational fluid dynamics, to study the improvement of temperature profiles upstream, within and downstream of a U-curvature pipe. The model geometry used in this study consisted of a U-curvaturefollowed by a direct circular inlet pipe and leadby amatchingdirect circular outlet pipe. It was displayed that Dean-Vortices which is centrifugally induced secondary-flows, partlyreversed the temperature profile. It was found that "Nusselt-Number" values downstream of a U-bend pipe exceed "Nusselt-Number"values for a combined entrancecondition by more than (20%) for up to (20)pipe diameters down-stream. **Patel and Parekh (2014) [9]** carried out the design of optimum size and the temperature difference of U-type (shell and tube heat exchanger), also illustrated the general design consideration and design procedure. Different types of procedures were carried out for best design, by "LMTD" method and "NTU" methods. The results showed that keeping the hot water mass flow-rate constant at (0.401kg/sec) and varying cold water mass flow rate at (0.4785, 0.5173 and 0.5595 kg/sec), created an growth in cold water temperature of (6.8°C, 5.7°C and 4.9°C) respectively. **Nayak et al. (2017) [10]** numerically deliberate the flow and heat transfer characteristics in 180°U-turnedtube. In their work they deliberated(RNG k-ε) turbulence model. The pressure-drop and heat transfer examined for multiphase flow using "finite volume method". The results showed that for tube bend the (h) for smaller radius ratio is (53.28%) more than the larger radius ratio for the solid concentration of (10%) and velocity of (1 m/s), also the value of the (h)enhanced with rise in the component concentration and velocity due to the attendance of a secondary -flow in the bends.

II. Model Description

The physical domain consist of Vertical U-tube configuration, this configuration consists of three parts, i.e. the upstream tube, the U-bend and the downstream tube. The configuration consists of circular Vertical U-tube with different values of dimensionless parameter called curvature radius ratio "Rc". Three different bends have been studied depending on "Rc" (that is equal to the radius of the U-bend curvature to the hydraulic diameter of the tube, $Rc=R/D_h$) which is (1, 1.5, 2), as shown in Figure 1. The currentstudy is undertaken to study the effect of a 180°bend in a vertical tube on heat transfer enhancement and friction factor for different values of curvature ratio.



November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

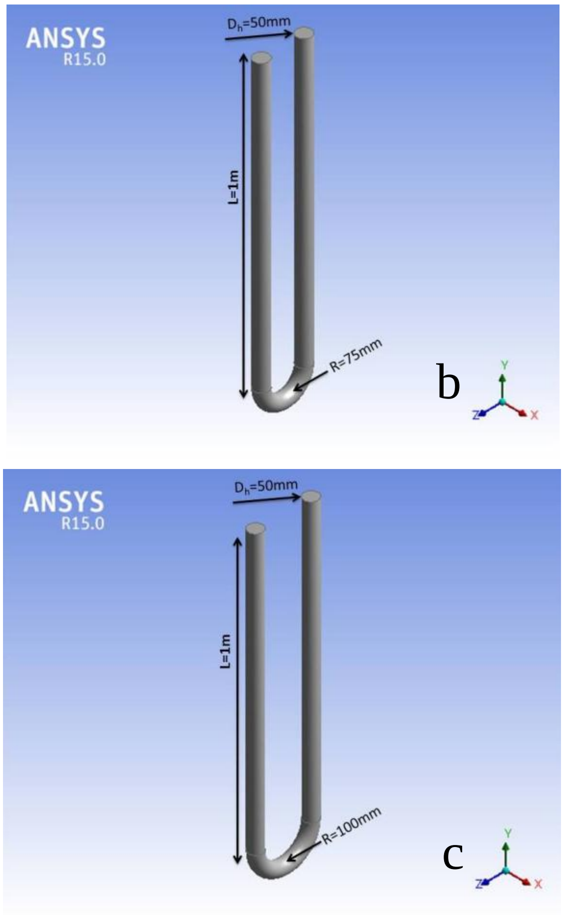


Figure 1. The geometry of the U-bend tube with different curvature radius ratio (a, b and c is Rc= 1, 1.5, 2)

Air flows in the tube, enters with a different velocities (3.136, 4.704, 6.272, 7.84 m/s) and the outlet is assumed to be an outflow boundary. Turbulent, isothermal, and steady state conditions will be considered to solve the flow field. The study has covered four Reynolds Numbers (1000, 1500, 2000 and 2500), and a constant PrandtlNumber equal to (0.707). Local heat transfer coefficients (h) were calculated from knowledge of local "heat flux" values along the test section which is ($1000 w/m^2$)

III. Mathematical Model and Numerical Analys

A. Assumptions

The following assumptions are used to simplify the suggested model solution:

- The fluid is "air" and has constant properties.
- The flow is assumed to be steady state.
- The flow isincompressible.
- The effect of gravity is neglected.
- Non slip flow is assumed.
- The dissipation of heat is assumed to be neglected.
- Negligible of pipe wall thickness.

B. Governing equations for the turbulent flow

The description of the turbulent fluid flow motion and heat transfer a U- tube bent is made byaverage differential equation of mass (continuity) momentum and energy equations.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (1)$$

Momentum equation in x-direction:

$$\rho\left(\frac{\partial u^2}{\partial x}+\frac{\partial uv}{\partial y}+\frac{\partial uw}{\partial z}\right) = -\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(2\mu_{eff}\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial u}{\partial z}\right) + \frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial w}{\partial x}\right) \quad (2)$$

Momentum equation in y-direction:

$$\rho\left(\frac{\partial vu}{\partial x}+\frac{\partial v^2}{\partial y}+\frac{\partial vw}{\partial z}\right) = -\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(2\mu_{eff}\frac{\partial v}{\partial y}\right) + \frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial y}\right) + \frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial w}{\partial y}\right) \quad (3)$$

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

Momentum equation in z-direction:

$$\rho\left(\frac{\partial wu}{\partial x}+\frac{\partial wv}{\partial y}+\frac{\partial w^2}{\partial z}\right) = -\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial w}{\partial x}\right) +\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(2\mu_{eff}\frac{\partial w}{\partial z}\right) +\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial z}\right) \quad (4)$$

Where μ_t is the eddy, or turbulent, viscosity.

$$\frac{\partial uT}{\partial x}+\frac{\partial vT}{\partial y}+\frac{\partial wT}{\partial z}=\frac{\partial}{\partial x}\left(\Gamma_{eff}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma_{eff}\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma_{eff}\frac{\partial T}{\partial z}\right) \quad (6)$$

To model the turbulence in the flow, The standard ***k*-*ε*** model which has two transport equations, one for ***k*** and one for ***ε*** is used.

$$\rho\left(\frac{\partial}{\partial x}(\boldsymbol{k}u)+\frac{\partial}{\partial y}(\boldsymbol{k}v)+\frac{\partial}{\partial z}(\boldsymbol{k}w)\right) = \frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_{\boldsymbol{k}}}\frac{\partial \boldsymbol{k}}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_{\boldsymbol{k}}}\frac{\partial \boldsymbol{k}}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_{\boldsymbol{k}}}\frac{\partial \boldsymbol{k}}{\partial z}\right)+G-\rho\varepsilon \quad (7)$$

For energy dissipation rate ($\boldsymbol{\varepsilon}$):

$$\rho\left(\frac{\partial}{\partial x}(\boldsymbol{\varepsilon}u)+\frac{\partial}{\partial y}(\boldsymbol{\varepsilon}v)+\frac{\partial}{\partial z}(\boldsymbol{\varepsilon}w)\right) = \frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_{\boldsymbol{\varepsilon}}}\frac{\partial \boldsymbol{\varepsilon}}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_{\boldsymbol{\varepsilon}}}\frac{\partial \boldsymbol{\varepsilon}}{\partial y}\right) +\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_{\boldsymbol{\varepsilon}}}\frac{\partial \boldsymbol{\varepsilon}}{\partial z}\right)+C_{1\boldsymbol{\varepsilon}}\rho\frac{\boldsymbol{\varepsilon}}{k}G - C_{2\boldsymbol{\varepsilon}}\rho\frac{\boldsymbol{\varepsilon}^2}{k} \quad (8)$$

Where G is referred to the generation term and is given [14]:

$$G = \mu_t\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+2\left(\frac{\partial w}{\partial z}\right)^2+\left(\frac{\partial v}{\partial y}\frac{\partial u}{\partial x}\right)^2 +\left(\frac{\partial v}{\partial z}\frac{\partial w}{\partial x}\right)^2 +\left(\frac{\partial v}{\partial z}\frac{\partial w}{\partial y}\right)^2\right] \quad (9)$$

Also,

$$\boldsymbol{k} = \frac{1}{2}\left(\overline{u'^2}+\overline{v'^2}+w'2 \quad (10)$$

$$\boldsymbol{\varepsilon} = \overline{e'_{ij}.e'_{ij}} \quad (11)$$

To determine the turbulent viscosity (μ_t), the turbulent kinetic energy($\boldsymbol{k}$) and the dissipation rate of the turbulent kinetic energy($\boldsymbol{\varepsilon}$) are chosen as the two properties. Where:

$$\mu_t = \rho c_\mu\frac{\boldsymbol{k}^2}{\boldsymbol{\varepsilon}} \quad (12)$$

Where: c_μ is a constant.

C. Boundary Conditions

The boundary conditions are specified for each zone of the computational domain as follow:

- Inlet boundary conditions
  - Uniform inlet velocity, U=Uin.
  - The flow is isothermal (T=Tin=300K).
- Wall Boundary Condition
  - The velocity at the walls is taken to be zero (no slip). (u=v=w=0 m/s) in the x, y and z direction.
  - A constant heat flux (q=1000W/m^2) concentrated on the whole surface area of the wall in the circular cross-section tube
- Outlet Boundary Condition: Zero gage pressure is specified at the outlet domain.

D. Hydrodynamic Parameters

- The Re. Number is defined by equation:

Re = inertial forces / viscous forces

$$\mathbf{Re} = \boldsymbol{\rho}\,\boldsymbol{U_{in}}\boldsymbol{D_h}/\boldsymbol{\mu} \quad (13)$$

Where D_h is the hydraulic diameter

$$\boldsymbol{D_h} = \frac{\boldsymbol{4A_{sec}}}{\boldsymbol{P_{er}}} \quad (14)$$

- Friction coefficient

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

The friction coefficient can be expressed in shear stress at the pipe surface is defined as:

$$C_f = \tau_w / \frac{1}{2}\rho U^2 \quad (15)$$

Where τ_w: is the wall shear stress and defined as:

$$\tau_w = \square\sqrt{(\frac{\partial u}{\partial y})^2 + (\frac{\partial w}{\partial y})^2} \quad (16)$$

- The bulk temperature.

The bulk temperature is described as:

$$T_b = \frac{\int Ti\,\rho\,dVi}{\int \rho\,dVi} = \frac{\sum_{i=1}^{n} Ti\,\rho\,|Vi|}{\sum_{i=1}^{n} \rho\,|Vi|} \quad (17)$$

- The wall temperature: The wall temperature is described as:

$$T_w = \frac{1}{A}\int Ti\,dAi = \frac{1}{A}\sum_{i=1}^{n} Ti\,|Ai| \qquad (18)$$

- The pressure drop, was calculated by surface integral by means of area weighted average:

$$P = \frac{1}{A}\int PdA \quad (19)$$

- The coefficient of heat transfer is defined as:

$$h = \frac{q''}{Tw - Tb} \quad (20)$$

Where q'' is the heat flux

- The Nusselt-number: The Nusselt-number is defined as:

$$Nu = \frac{h\,.\,D_h}{K} \quad (21)$$

- The radius curvature ratio (Rc) :

$$Rc = R/D_h \quad (22)$$

Where (R) is the tube curvature radius

E. Grid independency

A grid independence study is made to choose the optimum grid to get a better solution. The present results consider three different value for the grid independency investigate. The results are summarized in Table 1 show the specified grid for all shapes. The Nusselt-number is determined for all shapes at Re =10000.It is found that there is a small change in the average value of Nusselt-number. The grid resolution investigation reveal that ST, CUT/Rc=1, CUT/Rc=1.5 and CUT/Rc=2, become independent of the grid at cells 541,082 , 1,197,774 , 1,237,194 and 1,336,384 cells respectively.

TABLE I. DIFFERENT GRIDS AND THEIR NUSSELT-NUMBER AND FRICTION FACTOR RESULTS FOR DIFFERENT STUDIED CASES.

No.	Case	No. of grid elements	*Nu*	*Nu*$_{deviation}$	*f*	*f*$_{deviation}$
1	ST	465,094	37.1807	0.017957	0.03399	0.02335
		541,082	37.86056	0.044694	0.03480	0.04284
		683,636	39.63186		0.03636	
2	CUT	1,008,000	40.05469	0.011956	0.03738	0.02853
		1,197,774	40.53936	0.021926	0.03848	0.02572
		1,480,347	41.44816		0.039499	
5	CUT with Rc=1.5	959,889	39.2036	0.027685	0.03685	0.02435
		1,049,622	40.31984	0.002167	0.03777	0.01117
		1,237,194	40.4074		0.03820	
6	CUT with Rc=2	1,191,095	39.16034	0.035202	0.03681	0.03559
		1,273,857	40.58917	0.01863	0.03817	0.01659
		1,336,384	41.35972		0.03881	

F. Numerical solution

The numerical method of discredited governing equations can be solved by writing a code or the problem can be solved by using one of the commercial (CFD) code. ANSYS FLUENT 15.0

TEST Engineering & Management

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

software is used to calculate the current numerical investigation, Since Fluent is generally used in this field and it is considered one of the greater among other available codes. With a RNG (k-Ɛ) model, the governing equations where approximatedby usingfinite volume approach. A double-precision and pressure-based solver was used in the numerical computation. A non-slip boundary condition is adopted on pipe surface. The SIMPLE algorithm is used for pressure–velocity coupling. A second-order upwind scheme was adopted on the discretization of all terms.

IV. Results and Discussion

In this section, the numerical results have been addressed to verify the intensification of forced convection in a Vertical U-tube configuration with different shapes. The hydraulic diameter of the U-tubes is (0.05m). The case of the circular cross section U-tube with a different curvature radius ratio "Rc" (1, 1.5, 2) is discussed, and the results compared with the direct tube.

The effect of the curvature radius ratio on the average Nusselt-number variation for different values of Re. Number is described in Fig. 2.It is clearly observed that the Nusselt-number acquired from the three various curvature radius ratios are increase with increasing Re. Numberand decreasing the curvature radius ratio, which is agreed with [7]. The lowest curvature radius ratio (Rc=1) indicated the maximum increase in Nusselt-number as compared with other curvature radius ratio. The results verified that the enhancement in the mean Nusselt is found to be 7.9%, 7.4% and 7 % for Rc=1, 1.5 and 2 respectively as compared with the direct tube, the decrease of the curvature radius was accompanied by a strong intensification of the secondary transverse flows. It can be noted that using of the U-bend configuration leads to greater heat transfer for all the considered curvature radius ratio values. This is due to the development of secondary flows at the bend section. This flow mixing disturbs the boundary layer and as a result intensification the heat transfers.

Fig. 3. shows the influence of the curvature radius ratio on the average friction factor variation for different values of Re. Number. It can be noticed that the friction factor acquired from the three various curvature radius ratios are reduced with increasingRe. Number. It is viewed from the figure that there is decrease in the friction factor for utilizing Rc = 1.5, 2, and there is a rise in the friction factor for the lower curvature radius ratio, particularly at Rc=1. The gained results indicated that the increase in the friction factor is found to be 1.8% and 2.4 % for Rc=1, as compared with the curvature radius ratio Rc=1.5 and 2. Decreasing the (Rc) of the bend causes more improvement of the secondaryflow. The secondaryflow consequencesa very high frictional loss in the tube bend compared to direct tubes under similar circumstances. It can be observed that use of the U-bend configuration leads to more friction factor for all the considered (Rc) values.

The pressure losses occurred in a bend is caused by two factors, momentum exchanges and friction, resultant from a change in the direction of flow. These factors depend on the curvature ratio, bend angle and the Re. Number. The influence of the curvature radius ratio on the pressure drop variation for different values of Re. Number is shown in Fig. 4. It can be examined that the pressure losses obtained from using three different curvature radius ratios are increased with the increase of Re. Number. The lowest curvature radius ratio (Rc=1) denoted the highest growth in pressure drop as contrasted with other curvature radius ratios. The collected results illustrated that the rise in the pressure loss for Rc=1 as compared with the curvature radius ratios Rc=1.5 and 2 is 1.8% and 4.2%, respectively. The rise in pressure loss with U-bend configuration is greater than the direct tube, this is due to the centrifugal force that move the fluid from the bend outside around the tube wall towards the bend inside, in addition to the transverse velocity components of a secondary flow in the bends.

Fig. 5displays the effect of the curvature radius ratio on the wall temperature (for the whole surface area of the wall) in versus Re. Number. It is demonstrated that the mean wall temperature reduces with rise in Re. Number, since the Re. Number is proportional directly to the revolving speed or pressure, and the increase in the fluid speed lead to a declines of the wall temperature [6]. The minimum curvature radius ratio Rc=1 indicated the lowest average wall temperature as contrasted with other curvature radius ratios. The obtained results showed that the increase in the wall temperature is found to be 0.8% and 1.1% for the curvature radius ratios Rc=1.5 and 2, respectively as compared with Rc=1. This reduce of wall temperature is due to the secondary flows. The increase in wall temperature with U-bend configuration is greater than the directtube, this is due to the secondary flow that detected to drive cold fluid from the central of the tube towards the external tube walls. With extra circulation, the thermal boundary layer is reduced and aabrupt drop in fluid temperature at the surface of the tube happens within the curvature

Fig. 6established the effect of the curvature radius ratio on the enhancement efficiency variation for versus Re. Number. As can be shown in this figure, the variety of enhancement efficiency almost increases with rising of Re. Number. It is detected that the enhancement efficiency increases as curvature radius ratio decreased reaches its maximum value at Re. Number 20000, particularly at higher Re.Number 20000, The lowest curvature radius ratio (Rc=1) indicated the greatest increase in enhancement efficiency as contrasted with other curvature radius ratios.

The gained results demonstrated that the increase in the enhancement efficiency for Rc=1 as compared with Rc=1.5 and 2 is 0.7% and 1.5 %, respectively. It is demonstrated that the enhancement efficiency increased with a decreasing (Rc) and increasing Re. Number. When using U-bend configuration, the enhancement in heat transfer as Re. Number increase is higher than the direct tube where at high Re. Number, the flow has sufficient interchange the heat transfer with U-bend configuration and consequently the enhancement efficiency increase

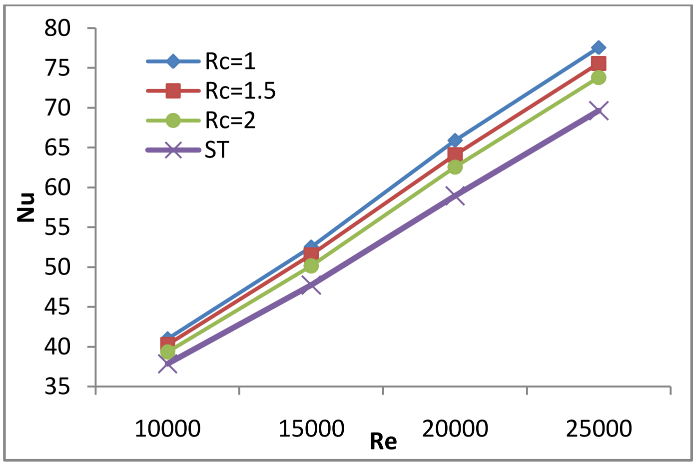


Figure. 2. Effect of curvature radius ratio on Nusselt-number for circular cross section U-tube

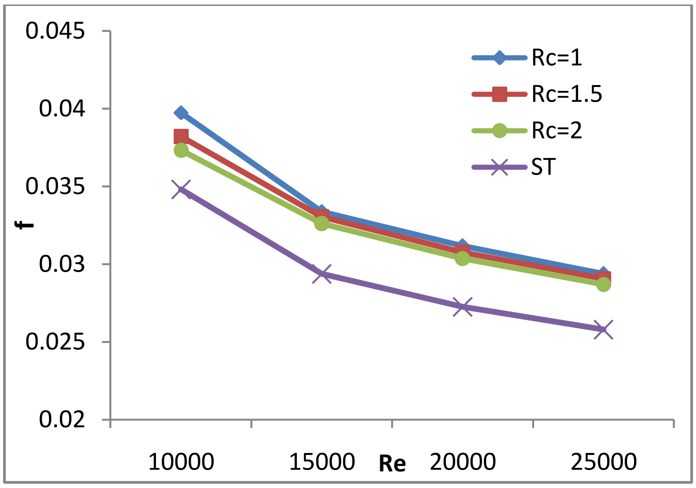


Figure.3. Effect of curvature radius ratio on friction factor variation for circular cross section U-tube

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

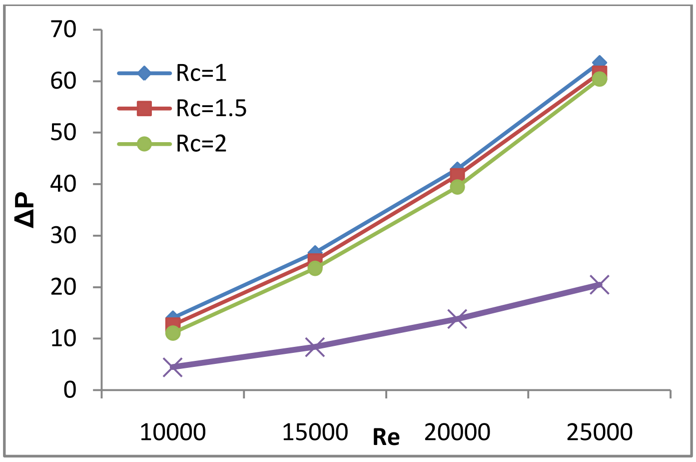


Figure.4. Effect of curvature radius ratio on pressure drop variation for circular cross section U-tube

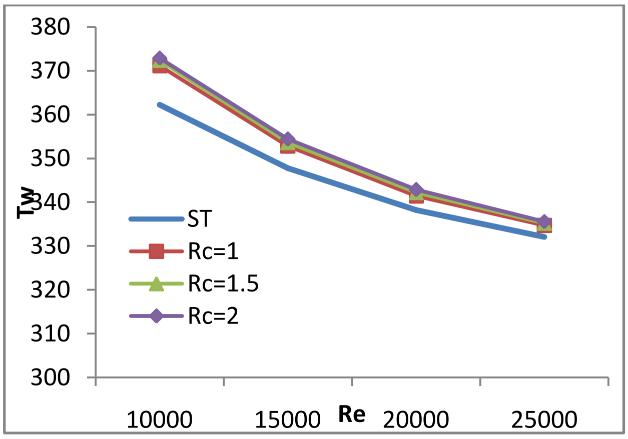


Figure.5. Effect of curvature radius ratio on wall temperature variation for circular cross section U-tube

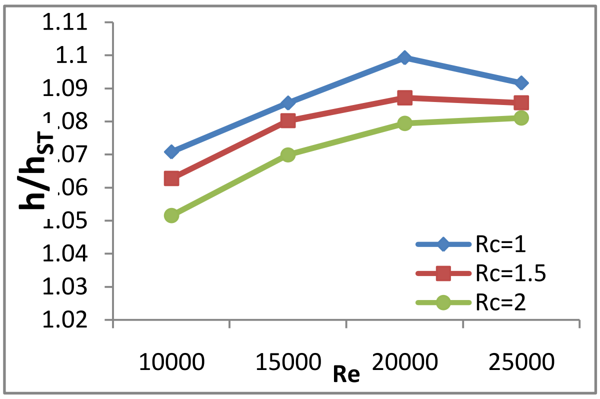


Figure.6. Effect of curvature radius ratio on enhancement efficiency for circular cross section U-tube

Fig. 7. presents the velocity vector for the case of circular Vertical U-tube and the effect of the curvature radius ratio variation, for air flowing with Re = 15000. Due to viscous effects, the velocity of flow in upstream tube domain, as presented in the Figure, velocity will increase by going away from the tube wall toward its center reached its highest value at the center, the frame a, b and c illustrates the velocity Vector for the circular Vertical U-tube with curvature radius ratio Rc=1. When the fluid inters the U-bend area the tangential velocity rises and swirl flow is produced. As shown from the figure, when the curvature radius ratio decreases the transverse velocity increase in the U-bend area. For the present model, at 90° bend, the maximum transverse velocity was approximately 6.04m/s, 5.97m/s and 5.87m/s for the 1, 1.5 and 2 curvature radius ratios, respectively

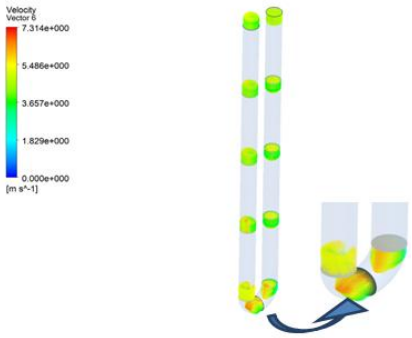


Figure 7.velocity vector for circular U- tube for Re=15000 of curvature radius ratio Rc=1

November-December 2019
ISSN: 0193-4120 Page No. 4893 - 4902

V. Conclusions

The heat transfer intensification in a heated vertical U-tube with different configurations has been numerically investigated .In this section, the following conclusions can be collected.

A. The heat transfer in the tube could be augmented highly by using U-bend configuration.

B. On comparing the performance in terms of heat transfer between direct tube and U-bend tube, U-bend tube yields better performance and heat transfer.

C. The lower curvature radius ratios of the vertical U-tube accomplish better intensification of heat transfer rate and enhancement efficiency .

D. The enhancement efficiency for a circular cross-section U-tube, with lower curvature radius ratio is found to be higher as compared with all other curvature radius ratio.

E. The optimum enhancement efficiency is ranged between 1.07 to 1.099 for using the circular cross-section U-tube, with Rc=1.

F. The enhancement in the mean Nusselt for utilizing the different curvature radius ratios is found to be 8.6%, 8.08% and 7.59 % for Rc=1, 1.5 and 2, respectively as compared with the direct tube.

References

[1] M. Moshfeghian, 'Fluid Flow and Heat Transfer in U-Bends', PhD thesis, Oklahoma State University, USA(1978)

[2] Rowe M., 'Measurements and computations of flow in pipe bends', J Fluid Mech;43:771-783(1970).

[3] M. Moshfeghian, The Effect of A 180° Bend on Turbulent Heat Transfer Coefficient in a Pipe', M.Sc thesis, Oklahoma State University, USA(1975)

[4] J. Azzola, J. Humphrey, H. Lacovides, B. Launder, Developing turbulent flow in a U-bend of circular cross-section: measurement and computation. J Fluids Eng;108:214-221(1986).

[5] S. Jayanti, M. Wang, F. Mayinger, Gas-particle flow through bends. IMechE C461; 24:161-166(1993).

[6] K. Sudo, M. Sumida, H. Hibara ,'Experimental investigation on turbulent flow through a circular-sectioned 180° bend', Experiments in Fluids, 28(1), pp. 51–57(2000).

[7] C. Münch, and O. Métais, "Large Eddy Simulations in Curved Square Ducts: Variation of the Curvature Radius," Journal of Turbulence, Vol. 8, No. 28, pp. 1-18(2007).

[8] R. Clarke and D.P. Finn, Numerical Investigation of the Influence of Heat Exchanger U-Bends on Temperature profile and Heat Transfer of Secondary Working Fluids, Proceedings of the 5th European Thermal-Sciences Conference, The Netherlands(2008).

[9] D. B. Patel, J. R. Parekh, 'Design and Experimental Analysis of Shell and Tube Heat Exchanger (U-Tube)', International journal of advanced research in engineering, science & management, 1 (5), pp.1-8(2015)

[10] B. B. Nayak, D. Chatterjee, A. N. Mullick, Numerical prediction of flow and heat transfer characteristics of water-fly ash slurry in a 180° return pipe bend, International Journal of Thermal Sciences, Volume 113, March, Pages 100-115(2017).

[11] V.L. Streeter and E.B. Wylie, Fluid mechanics,First SI Metric Edition ,5th printing ,ISBN:0-07-548015-8.

[12] K.L. Kumar, Engineering fluid mechanics, Seventh Edition, ISBN:81-219-0100-6(2001).

[13] H. M. A. AL-Aboodi, Three –Dimensional numerical study of laminar and turbulent flow in sudden expansion channel, M.Sc. Thesis, university of basrah,(2010).

[14] H. Mahdi, Numerical and Experimental study of Enhancement of heat transfer in Roughened Ribbed Duct, PHD thesis, Department of technical Eduction, University of technology, Iraq(2004)