Optimization of the Corrugated Pipe for the Enhancement of Heat Transfer

Vishal N. Gunake[1] Sanjay V. Kadam[2]
[1,2]Rajarambapu Institute of Technology Rajaramnager, Islampur

Abstract— By using the Passive method for enhancing the heat transfer with minimum pressure drop change in the geometrical modification of internally pasted rib's with length 15 mm and diameter of that fins are 1 mm that will help to disturb the laminar sub layer which is created in turbulent region in the boundary layer. By using modelling software CATIA V5, creating the smooth pipe model and models with different orientations of angle 00, 300, 450, 600, 900 in the direction of the fluid flow. These models imported in the software ANSYS version 15.0 with the extension. stp and created the fluid domains of different orientation of the pipes for fluid flow analysis. The Meshing of these fluid domains is required for fluid flow analysis for the CFD software. The accuracy of the meshing which is acceptable for the analysis is refers to Skewnees which is in the range of 0 to 1. The Computational Fluid Dynamics (CFD) with FLUENT as a solver used, the simulation of heat transfer and fluid flow analysis in the Laminar and turbulent region in tubes with internally rib's are done. From analysis it found that 30 deg orientation pipe gives more heat transfer with minimum pressure drop as compared with other orientations of pipe. Over all 33.5 % increment is achieved with using internal ribs in pipe as compared to smooth pipe.

Key words: CATIA V5, ANSYS 15-Computaional Fluid Dynamics (CFD)-FLUENT, boundary layer theory, Newton's low of cooling

I. INTRODUCTION

Heat exchangers have several industrial and engineering applications. The design procedure of heat exchangers is quite complicated, as it needs exact analysis of heat transfer rate and pressure drop estimations apart from issues such as long-term performance and the economic aspect of the equipment. The major challenge in designing a heat exchanger is to make the equipment compact and achieve a high heat transfer rate using minimum pumping power. They are broadly classified into three different categories:

1) Passive Techniques
2) Active Techniques
3) Compound Techniques.

A. Passive Techniques

These techniques generally use surface or geometrical modifications to the flow channel by incorporating inserts or additional devices. They promote higher heat transfer coefficients by disturbing or altering the existing flow behaviour (except for extended surfaces) which also leads to increase in the pressure drop. In case of extended surfaces, effective heat transfer area on the side of the extended surface is increased. Passive techniques hold the advantage over the active techniques as they do not require any direct input of external power. These techniques do not require any direct input of external power; rather they use it from the system itself which ultimately leads to an increase in fluid pressure drop. They generally use surface or geometrical modifications to the flow channel by incorporating inserts or additional devices. They promote higher heat transfer coefficients by disturbing or altering the existing flow behaviour except for extended surfaces.

B. Active Techniques

These techniques are more complex from the use and design point of view as the method requires some external power input to cause the desired flow modification and improvement in the rate of heat transfer. It finds limited application because of the need of external power in many practical applications. In comparison to the passive techniques, these techniques have not shown much potential as it is difficult to provide external power input in many cases. In these cases, external power is used to facilitate the desired flow modification and the concomitant improvement in the rate of heat transfer.

C. Compound Techniques

A compound augmentation technique is the one where more than one of the above mentioned techniques is used in combination with the purpose of further improving the thermo-hydraulic performance of a heat exchanger. When any two or more of these techniques are employed simultaneously to obtain enhancement in heat transfer that is greater than that produced by either of them when used individually, is termed as compound enhancement. This technique involves complex design and hence has limited applications.

II. METHODOLOGY

A. Boundary Layer Theory-Laminar boundary layer

Consider the flow of fluid having free-stream velocity (U), over a smooth thin plate which is flat and placed parallel to the direction for free stream of fluid as shown in Fig. Let us consider the flow with zero pressure gradients on one side of the plate, which is stationary.

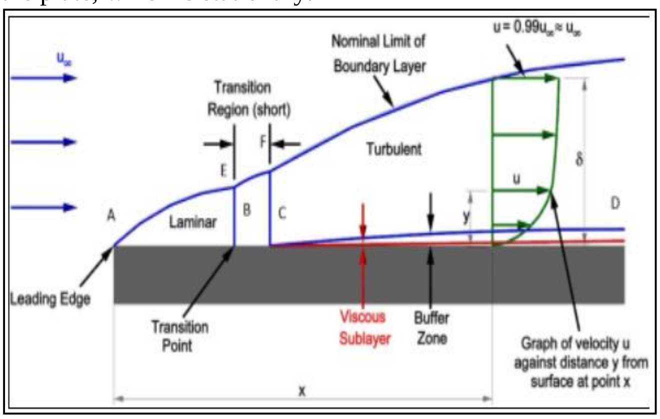

Fig. 1: Boundary layer

The velocity of fluid on the surface of the plate should be equal to the velocity of the plate. But plate is stationary and hence velocity of fluid on the surface of pate is zero. But at a distance away from the plate, the fluid is having certain velocity. Thus a velocity gradient is set up in the fluid near the surface of plate. This velocity gradient develops shear resistance, which retards the fluid. Thus the fluid with a uniform free stream velocity (U) is retarded in the vicinity of the solid surface of the plate and the boundary layer region begins at the sharp leading edge. At subsequent points downstream the leading edge, the boundary layer region increases because the retarded fluid is further retarded. This is also referred as the growth of boundary layer. Near the leading edge of the surface of the plate, where the thickness is small, the flow in the boundary layer is laminar though the main flow is turbulent this is shown by distance AB. The distance of B from leading edge is obtained from Reynolds number equal to for a plate. Because up to this Reynolds number the boundary layer is laminar. The Reynolds no is given by,

(Re) x = U*x/v

Where, x= distance from leading edge

U= free-stream velocity of fluid

ν = kinematic velocity of fluid

Hence, if the values of U and v are known, x can be calculated.

B. *Turbulent Boundary Layer*

If the length of the plate is more than the distance x, the thickness of boundary layer will go on increasing in the downstream direction. Then laminar boundary layer becomes unstable and motion of fluid within it, is disturbed an irregular which leads to a transition from laminar to turbulent is called transition zone. This shown by distance BC in fig. further downstream the transition zone, the boundary layer is turbulent and continues to grow in thickness. This layer of boundary is called turbulent boundary layer, which is shown by the portion FG in fig.

C. *Laminar Sub-layer*

This is the region in turbulent boundary layer zone, adjacent to the solid surface of the plate as shown in fig. In this zone, the velocity variation is influenced only by viscous effects. Though the velocity distribution would be a parabolic curve in the laminar sub-layer zone, but in view of the very small thickness we can reasonably assume that velocity variation is linear and so the velocity gradient can be considered constant. Therefore, the shear stress in the laminar sub-layer would be constant and equal to the boundary shear stress (Ʈ). Thus the shear stress in the sub- layer is,

Ʈ = μ*(du/dy)

= (μ * u/y)

III. SMOOTH PIPE ANALYSIS

A. *Observation of Smooth Pipe by CFD Analysis*

Observations:

1) Length of pipe =1.68m
2) Diameter of pipe =0.028m
3) Water flow rate =1 to 7 LPM

B. *Calculations and validating by using Dittus bolter co-relation*

Average surface temperature = 52

Bulk mean temperature = $(T_i + T_o)/2$ = 42.37°C

$(Q_{th}) = m\, C_p\,(T_o - T_i)$ …… (1)

Where,

m = mass flow rate of water in (kg/sec).

C_p=specific heat of water (J/kg k).

T_i = Initial temperature of water in°C

T_o = Final temperature of water in °C

Q_{th}=0.119 x 4182 x (43.75-41) watts =1368.55 watts

Therefore, Q_{th}= hA (T_{avg}-T_{bm})….. (2)

Hence, heat transfer coefficient (h) from equation (1) & (2)

$$h = \frac{Qth}{A\,(Tavg - Tbm)}$$

$$h = \frac{1368.55}{0.1478*(52-42.37)}$$

h = 961.52 w/m^{2}k (By experiment)

Validating using dittus bolter co-relation

Nu = 0.023 x (Re)$^{0.8\,x}$ (Pr)$^{0.4}$

Where,

Nu=Nusselt number

Re=Reynolds number

$P_{r=}$Prandtl number

Reynolds number (Re) is given by,

$$R_e = \frac{\rho v d}{\mu} = \frac{998.2*0.1936*0.028}{0.001} = 5411.26$$

$$P_r = (\mu * Cp)/k = \frac{0.001*4182}{0.6} = 6.97$$

Nu = Nu = 0.023 x (Re)$^{0.8}$ x (Pr)$^{0.4}$ .. (3)

Putting values of Re, P_r in equation (3)

Nu =0.023 x (5411.26)$^{0.8}$x (6.97)$^{0.4}$

Nu =48.49

Nusselt number can also be given by equation

$$Nu = \frac{hxD}{k} \quad h = \frac{Nuxk}{D} = \frac{48.49\,x\,0.6}{0.028}$$

=1039.08 w/m^2°C ….. (By co-relation)

C. *Comment on Result*

From above calculation conclusion is that there is not much difference in both CFD analysis and co-relation value obtained, we get the 7-8 % of difference in CFD analysis and the co-relation value. Hence dittus bolter co-relation is satisfied. Hence for CFD analysis can be done on other flow domains.

IV. DESIGN AND ANALYSIS OF DIFFERENT FLOW DOMAINS

A. *Creation of the Model in CATIA V5*

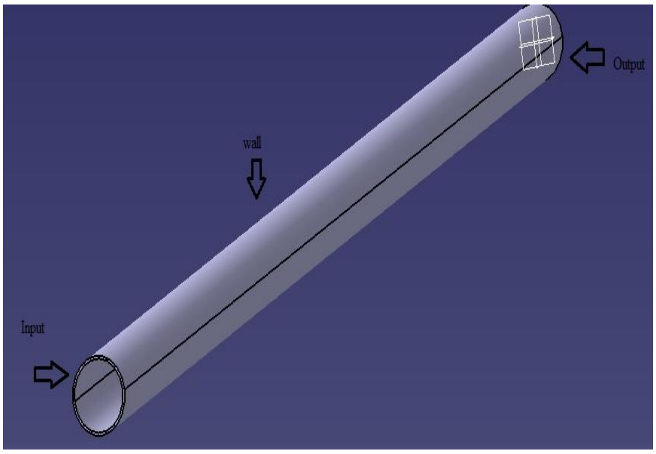

Fig. 2: smooth pipe

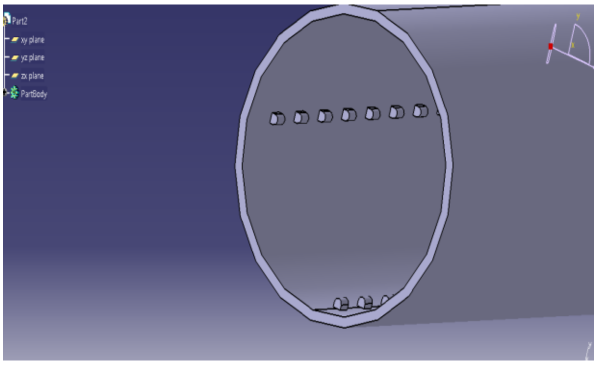

Fig. 3: Deg Orientation Pipe

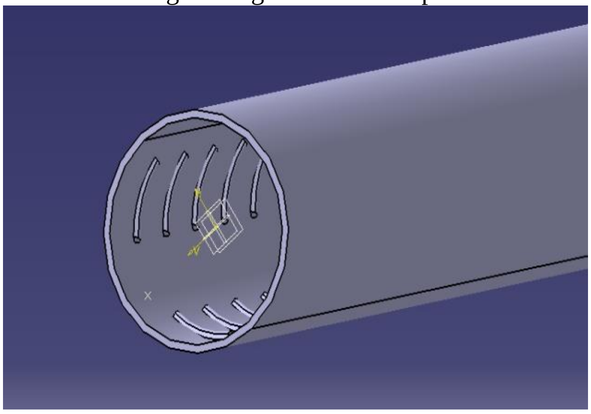

Fig. 4: deg orientation pipe

B. Ansys Imported Geometry Transferring View

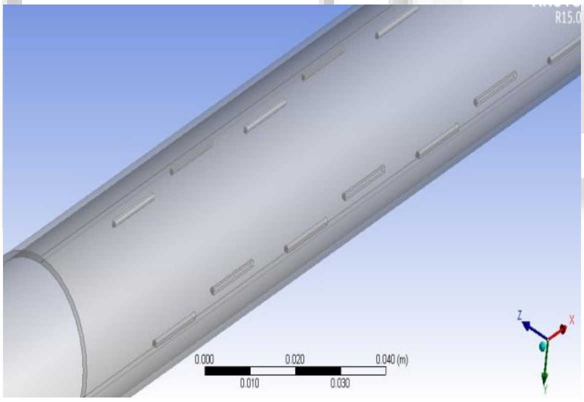

Fig. 5: deg orientation pipe

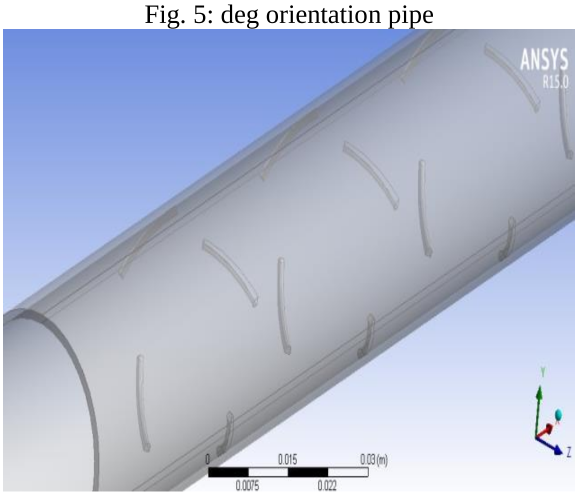

Fig. 6: deg orientation pipe

C. Making of Flow Domains for Different Orientations

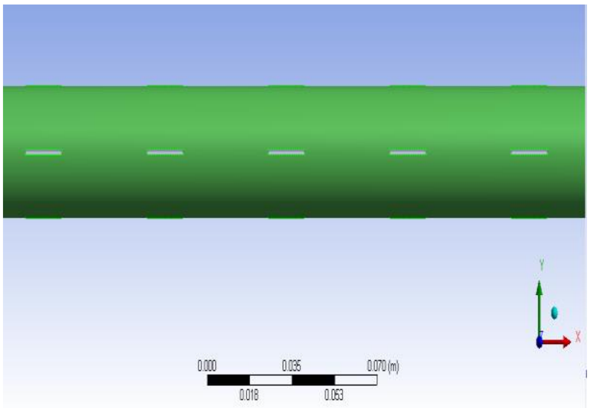

Fig. 7: deg orientation

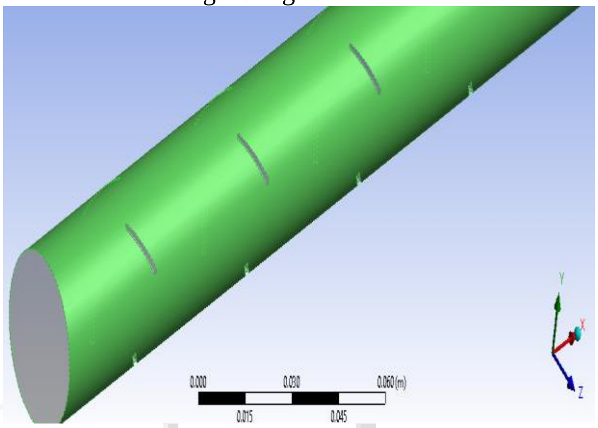

Fig. 8: deg orientation

D. Mesh Size of the All the Flow Domains

Orientation	smooth pipe	0 Deg	30 Deg	45 Deg	60 Deg	90 Deg
Nodes	24367	195064	190138	190009	202634	162690
Elements	19776	1004060	986341	986514	1054538	834213
Skewness	0.23	0.23	0.24	0.23	0.23	0.24
aspect ratio	1.63	1.87	1.87	1.87	1.86	1.87

Table 1: Mesh size of all flow domains

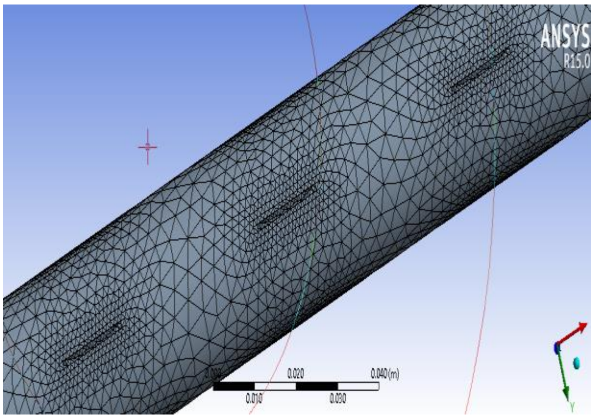

Fig. 9: deg orientation pipe

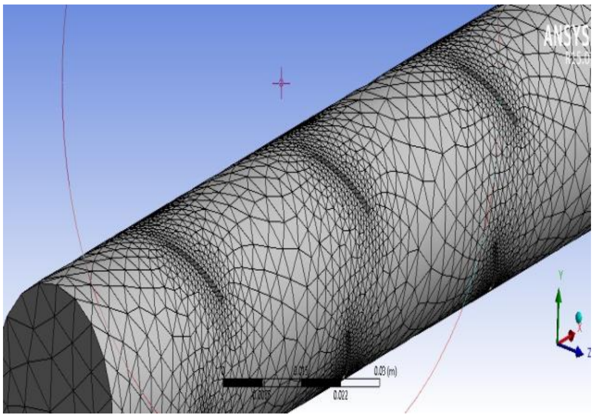
Fig. 10: deg orientation pipe

E. Post Processing

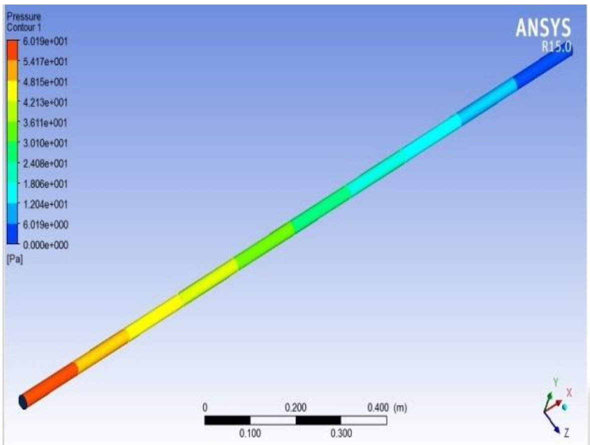

Fig. 11: Pressure Contour

When the fluid flows through pipe this shows the Contour flow result.

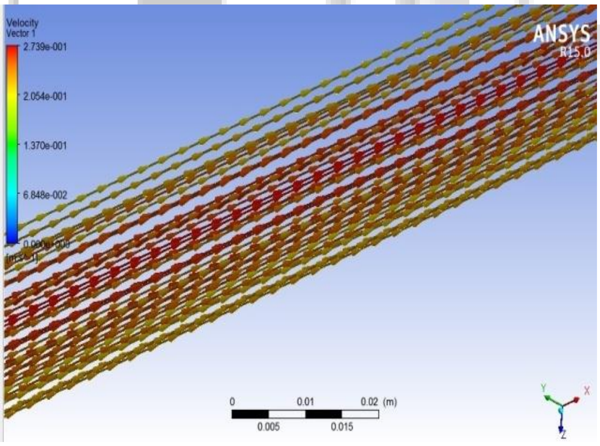

Fig. 12: Velocity vector

V. RESULT AND DISCUSSION

A. Laminar Flow Result

	HEAT TRANSFER COEFFIENT				
Orientation	Flow in LPM	1	1.5	2	2.5
	smooth pipe	277.163	303.263	405.632	419.961
	0 Deg	709.858	864.288	922.714	1022.010
	30 Deg	640.243	774.048	918.890	1019.792
	45 Deg	709.858	770.956	915.076	1013.153
	60 Deg	709.858	857.600	922.714	997.764
	90 Deg	640.243	770.956	909.374	991.211

Table 2: Laminar Flow Result

B. Turbulent Flow Result

	HEAT TRANSFER COEFFIENT				
	Flow in LPM	4.5	5	6	7
Orientation	smooth pipe	761.365	845.962	855.722	961.680
	0 Deg	1067.126	1189.095	1213.052	1348.520
	30 Deg	1051.880	1192.498	1209.211	1440.410
	45 Deg	1073.248	1189.095	1213.052	1390.640
	60 Deg	1073.248	1185.695	1213.052	1433.190
	90 Deg	1076.315	1192.498	1213.052	1428.690

Table 3: Turbulent Flow Result

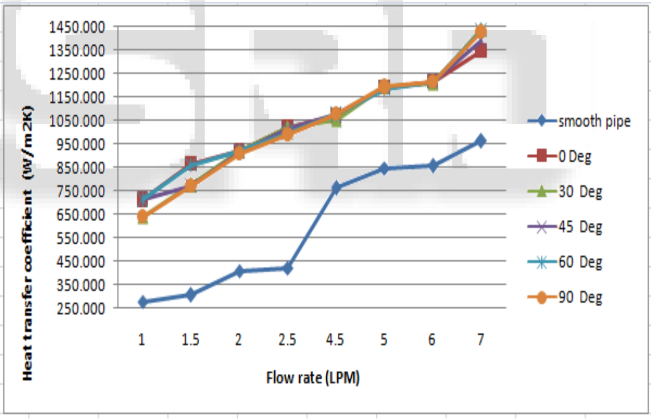

Fig. 12: Graph 1 Heat transfer coefficient Vs Flow rates (Laminar and Turbulent Flow)

C. Nusselt number Vs Reynolds number

Nusselt No.	smooth pipe	12.934	14.152	18.930	19.598	35.530	39.478	39.934	44.879
	0 Deg	33.127	40.333	43.060	47.694	49.799	55.491	56.609	62.931
	30 Deg	29.878	36.122	42.882	47.590	49.088	55.650	56.430	67.219
	45 Deg	33.127	35.978	42.704	47.280	50.085	55.491	56.609	64.897
	60 Deg	33.127	40.021	43.060	46.562	50.085	55.332	56.609	66.883
	90 Deg	29.878	35.978	42.437	46.257	50.228	55.650	56.609	66.673
Re		773.98	1161	1548	1935	3482.9	3869.9	4643.9	5411

Table 4: Nussel number VS Reynolds number

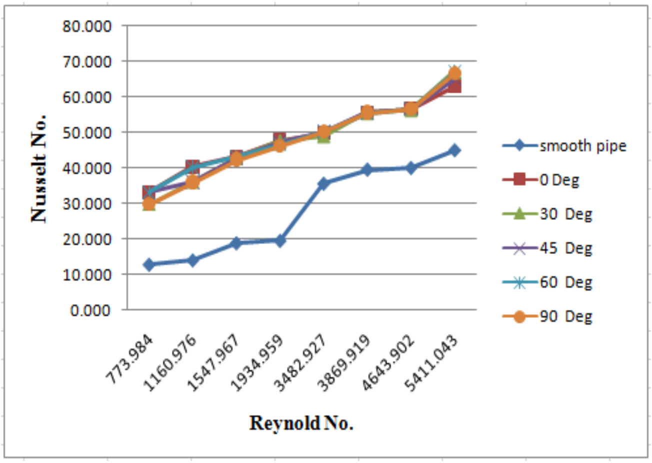

Fig. 13: Graph 2 Nusselt number vs Reynolds number

VI. CONCLUSION

1) The Simulation analysis is done with the help of Computational Fluid Dynamics (CFD) –FLUENT -15 and the results obtained from this compared with the results obtained from the Dittus bolter correlation values.
2) From graph it is analysed that the Nusselt number and the pressure drop increases with increase in Reynolds number in laminar as well as turbulent flow conditions. Nusselt number depends on the flow domain geometry, hence it calculated for different flow domains.
3) The overall enhancement of the heat transfer is 33.5 % achieved as compared with smooth pipe result.
4) In the analysis it is found that the 30 deg orientation pipe has the higher heat transfer coefficient with minimum pressure drop and other orientations having the higher pressure drop with minimum heat transfer coefficient.
5) The results of the CFD analysis, experiment analysis and the results are obtained from the correlation are validated.
6) Simply by using the internal ribs of 1 mm diameter with15 mm in length the thermal performance of the heat exchanger increases. This will help to minimise the size of the heat exchanger equipment.

REFERENCES

[1] M.N.Nazri, Tholudin M.Lazim “Corrugation profile effect of heat transfer enhancement of laminar flow region”, international conference on Mechanical and industrial engineering. Feb 8-9 2015.
[2] Ventsislav Zimparov “Enhancement of heat transfer by a combination of a signal start spirally corrugated tubes with twisted tape” Elsevier, vol 25(2002)pp. 535-546.
[3] M.Faizal “Experimental studies on a corrugated plat heat exchanger for small temperature difference application”Vol.36, (2012) pp.242-248.
[4] S.V. karmare, “Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs” Science-Direct, Heat and mass transfer Vol. 50 (2007) pp.4342-4351.
[5] Aicha chorak “Numerical evaluation of heat transfer in corrugated heat exchanger” (2014)pp.1-6.
[6] Paisarn Naphon “Effect of corrugated plate in an in-phase arrangement of heat transfer and flow developments” science direct Vol 51(2008) pp.3963-3971.
[7] Yang Dong “pressure drop, heat transfer and performance of single phase turbulent floe in spirally corrugated tubes” Elsevier, Vol 24(2001)pp. 131-138.
[8] Wen-Tao Ji “Summery and evaluation on single phase heat transfer enhancement techniques of liquid laminar and turbulent pipe flow” Science Direct, Vol 88(2015) pp. 735-754.
[9] Khalid M. Saqr “Numerical study of heat transfer augmentation in pipes with internal discontinues longitudinal fins” IJMME Vol 4 (2009) pp. 1, 62-69.
[10] M.A. Ahmad “Effect of geometrical parameters on flow and heat transfer characteristics in trapezoidal-corrugated channel using nanofluid” Science Direct, Vol 42 (2013) pp. 69-74.