IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 5 Ver. III (Sep- Oct. 2014), PP 37-46
www.iosrjournals.org

Investigation on Design Optimization of Corrugated Surface Heat Exchangers

Dr. Mohammed Shekoor. T[1], Rajeesh. P[2]
[1](*Professor, Mechanical engineering, LBSCE Kasaragod, India*)
[2](*PG student, Mechanical engineering, LBSCE Kasaragod, India*)

Abstract: *Compact heat exchangers are one of the vital components in micro level turbo machinery engineering systems. The compact heat exchanger is used as recuperators or intercooler. Due to its high volume goodness and light weight, the cross-corrugated surface heat exchanger is a promising alternate for application to an advanced intercooled-cycle gas turbine engine. This paper presents a study of heat transfer and frictional characteristics of a corrugated compact heat exchanger. In the present study CFD is used to perform a quantitative assessment of the thermal performance of cross corrugated heat exchanger with various alterations such as different pitch-height ratio and different corrugation angle. The turbulence model is used for all the simulation. At the first stage with constant Reynolds number, and then with higher Reynolds number have taken into account. Found the variations of parameters with geometries. Finally select best suitable geometry of heat exchanger for the recuperators of micro turbine ranges from 5-100 kW.*
Keywords: *Corrugated surface, CFD, Heat exchanger, Recuperator*

I. Introdution

A heat exchanger is a device to transfer heat from a hot fluid to cold fluid across an impermeable wall. Fundamental of heat exchanger principle is to facilitate an efficient heat flow from hot fluid to cold fluid. This heat flow is a direct function of the temperature difference between the two fluids, the area where heat is transferred, and the conductive/convective properties of the fluid and the flow state. The common mode of generating electrical energy for remote area, for emergency power situations etc, was using diesel power plants. Recuperator is needed for improving efficiency of micro turbine above 35%. Recuperators are heat exchanger device, which should give the maximum heat transfer under such conditions. Further requirements on recuperators, of which low cost is identified above, are the following:

- High thermal effectiveness and low pressure losses. These affect the gas turbine cycle efficiency.
- High reliability and durability give low maintenance cost and long operational life time.
- Minimum weight and volume. The weight is directly proportional to material cost and a small volume of the unit makes the gas turbine packages easier to handle.

Heat exchangers are classified according to process function, compactness, flow directions, construction of geometries etc. We can broadly classify the compact type heat exchanger into two, plate-fin and surface (corrugated) compact type heat exchangers. London and shah give a good definition for the “compactness” of the HT surfaces[1], as one having a surface area density of more than $700m^2/m^3$. In this study we mainly look for the plate heat exchanger type and in which corrugated geometry gives the maximum effectiveness. They are more compact in nature. Lot of numerical, experimental and computational investigation has conducted by the researchers for designing the optimum heat transfer performance features. Recent trend is to make the heat transfer surface i.e. corrugated with good performance characteristics. Yue-Tzu Yang et al.[2] carried out a detailed numerical study of heat transfer characteristic of V corrugated upper and lower plates. They concluded that the increasing the angle of V corrugated Chanel, heat transfer performance become better, increasing of Reynolds number makes the fluid flow become more complex. Wenfeng Gao et al[3] conducted analytical and numerical studies on the thermal performance of cross corrugated plate. Compare plate surfaces with corrugated surface and obtain achievable efficiencies over the other surfaces. Experimental investigations are carried out by J.Stasiek.M.W. Collins et al [4,5,6] conduct experiments in corrugated surface investigate the dependence of heat transfer and pressure drop on Reynolds number and geometrical parameters, by making these vary in a systematic way and in sufficiently broad range. This was meant to lead to engineering correlations to be used for more general optimization studies, essentially involving energy conservation. The equivalent friction coefficient f was found to decrease with the Reynolds number roughly as $Re^{-1/2}$. The angle dependence of f was much larger than that of the Nusselt number. The friction coefficient increased with P/H**,** an asymptote is suggested by the experimental data for P/H, > 4**.**

Heat exchangers play a significant role in the operation of many systems such as power plants, process industries and heat recovery units[7]. Its inevitable need has necessitated work on efficient and reliable designs leading towards optimum share in the overall system performance. The Log Mean Temperature Difference

(LMTD) method and the number of heat transfer units (NTU) method have been used for heat exchanger design. These methods have some shortcomings associated with them i.e. iterative in nature and need of a prototype to implement the design. Due to these reasons, these methods are time consuming as well as expensive especially for large scale models. However, economical access to powerful micro processors has paved the way for evolvement of Computational Fluid Dynamics (CFD) during the design phase. Here after we are going to discuss about the application of computational fluid dynamics in the design of heat exchangers, it is good tool for optimizing the design with low cost. Some of the commercial CFD codes in use are FLUENT, CFX, STAR CD, FIDAP, ADINA, CFD2000, PHOENICS and others[8].

CFD is a science that can be helpful for studying fluid flow, heat transfer, chemical reactions etc by solving mathematical equations with the help of numerical analysis. It is equally helpful in designing a heat exchanger system from scratch as well as in troubleshooting/optimization by suggesting design modifications. CFD employs a very simple principle of resolving the entire system in small cells or grids and applying governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients, flow parameters and the like in a shorter time at a lower cost because of reduced required experimental work[9]. The results obtained with the CFD are of acceptable quality[10, 11]. In the current work, various problems encountered in the design of heat exchangers and their solutions with the help of CFD have been reviewed. This work can serve as a ready reference for application of CFD in design of various types of heat exchangers.

When exchanging heat with air, the main thermal resistance is located on the airside of the heat exchanger .To improve the heat transfer rate, the heat transfer surface area is increased by adding finned plates. The heat transfer characteristics of flow through such corrugated channels are quite different than for parallel plate channels. In a corrugated channel, the main flow direction is parallel to the channel axis, but the local flow direction is always changed due to channel waviness. However, such gains in heat transfer are invariably accompanied by increased pressure drop penalty. There are different geometries are available in corrugated plate such as cross corrugated surface, corrugated undulated, cross wavy surfaces etc.

The cross wavy CW and especially the cross corrugated CC surfaces show superior performance over the others giving a small volume and weight of the heat transfer matrix, but as the CC surface is well documented in the literature and probably is easier to manufacture with small passage dimensions this should be the first choice for further studies by recuperator manufacturers. The design results show that a cross corrugated CC surface has the best potential for use in compact recuperators of the future as it will have smaller volume than any recuperator in operation today[12].

II. Problem Definition

2.1Physical model: - Geometry of the cross corrugated surface

The need of compact heat transfer equipment, contributed to development of the study about variety geometrical structure of corrugated duct flow and heat transfer. Corrugated passage characteristics of the fluid flow largely depend on the flow in the region of separation and reattachment vortex current. The intensity of the heat transfer and the size of the heat transfer area depend on the continuous destruction and rebuilding of the boundary layer so that the fluid produces rotation, the unstable vortex induced by shear layer as well as the formation of the hydraulic diameter smaller channel, etc. Pressure loss on a wall depends largely on the mainstream in the direction of the gradient wall rather than vertical gradient of secondary flow, a lower pressure loss cause a larger heat exchanger result in paying more attention to use vortex motion to enhanced heat exchanger. In this paper, intensification of heat exchange in Cross corrugated passage is to use same fluid in the vicinity of composition cross-flow section, the interaction between two beams of fluid cause vortex, and the motion of vortex becoming unstable free shear layer to strengthen the fluid disturbance.

The geometrical features of the three dimensional corrugated flow channel, as shown in Fig.2.1, are described by the height(H), pitch(p), plate thickness(t) and corrugation angle (θ) plays significant role as indicated by London an Shah[1].

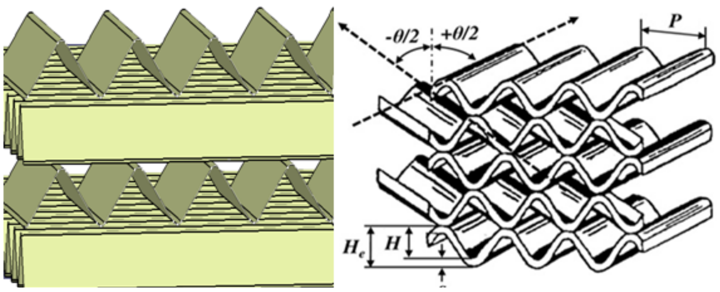

Fig.2.1. Corrugation geometry with geometrical parameters

The performance of a corrugated surface found by fabricate heat exchanger cores and conduct experiments over reasonable ranges of all the geometric variables, it is prohibitively expensive and time consuming. In contrast, it is relatively easy and cost-effective to carryout parametric study through numerical simulation and derives performance characteristics of the surface.

Our problem is to design the geometrical parameters with optimum performance of the heat exchangers. Mainly looking to different corrugation angle and pitch to height ratio. CFD simulation are carried out on following corrugation geometries, CC represent cross corrugation and values mention the pitch to height ratio.

Table 2.1 Corrugation geometries for numerical simulation

Geometry (All dimensions are in mm)	A –CC 2.2	B – CC 3.06	C – CC 4
Pitch, P	25	29.5	32
Height, H_e	12	10.4	8.8
Height, H	11	9.6	8
P/H	2.22	3.06	4
Hydraulic diameter, D_h	15.4	15.4	15.4

Many authors have carried out their research work on this corrugation geometries, but the results are little bit different. Here we are looking to optimize the design of the cross corrugated surface with different corrugation angles such as 30^0, 45^0, 60^0 and 75^0.

- With same input parameter such as inlet fluid temperature, mass flow rate of the fluid, same corrugated material and same fluid (air - for cold side and hot side).
- The problem is a counter flow heat exchanger with Newtonian fluid.
- Steady flow, incompressible, one dimensional heat transfer problem in 3-Dimensional corrugated geometry.

2.2Mathematical model

For fully developed steady flow state three dimensional mass conservation, under the conditions of ignoring the force of volume, governing equations can be written as,

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial uj}{\partial xj} = 0 \tag{2.1}$$

Momentum equation:

$$\frac{\partial ui}{\partial t} + \frac{\partial uiuj}{\partial xj} = -\frac{1}{\rho}\frac{\partial p}{\partial xi} + \frac{\partial(\nu\frac{\partial ui}{\partial xj})}{\partial xj} \tag{2.2}$$

Energy equation;

$$\frac{\partial T}{\partial t} + \frac{\partial ujT}{\partial x} = \frac{\partial}{\partial xj}\left(\frac{\nu}{Pr}\frac{\partial T}{\partial xj}\right) \tag{2.3}$$

Two equation turbulent models are used for the analysis, in which k-epsilon turbulent model have selected. In the two-equation models, we develop two PDEs: one for the turbulent kinetic energy and one for the turbulent dissipation rate.

$$\left.\begin{aligned} \frac{\partial \rho K}{\partial t} + \frac{\partial \rho U_j K}{\partial x_j} = -\rho\overline{u_i u_j}\frac{\partial \rho U_i}{\partial x_j} \quad & \ldots\ldots\ldots(P_k) \\ -\frac{\partial}{\partial x_j}\left[\frac{1}{2}\rho\overline{u_i^2 u_j} + \overline{pu_j} - \mu\frac{\partial K}{\partial x_j}\right] \quad & \ldots\ldots\ldots(D_k) \\ -\mu\overline{\frac{\partial U_j}{\partial x_j}\frac{\partial U_j}{\partial x_j}} \quad & \ldots\ldots\ldots(\varepsilon_k) \end{aligned}\right\} \tag{2.4}$$

The PDE for the turbulent kinetic energy is already given by above "Equation 2.4" however, the expression for the turbulent or eddy viscosity is different. So, the idea is to express the turbulent viscosity as a function of K and ε and then derive PDEs for K and ε.

$$\nu_t \ \acute{\alpha}\ ul = k^{\frac{1}{2}}\left(\frac{k^{\frac{3}{2}}}{\varepsilon}\right) \tag{2.5}$$

$$\nu_t = C_\mu\frac{k^2}{\varepsilon} \tag{2.6}$$

$$k_t = \nu_t\frac{c_p}{Pr} \tag{2.7}$$

The equation for the turbulent kinetic energy is shown above equation, now we are looking to turbulent dissipation (ε, is the dissipation rate of k.), instead of that we develop an independent PDE for its transport. We obtain,

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho U_j \epsilon}{\partial x} = \frac{\partial}{\partial x}\left[\left(\mu + \frac{\mu_t}{\sigma}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1} P_k \frac{\varepsilon}{K} - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{K} \tag{2.8}$$

The constraints are obtained from bench mark problems.

III. Solution Methodology

3.1 Pre- processor: - Geometry creation and meshing

The first step of any design process is preprocessing. Preprocessing means the complete process just before giving to the analysis part, this may include creation of geometry, meshing, assigning boundaries types etc. Corrugated geometry created by using CATIA V5R20. The system created using boundary layer forming method, i.e. first we form the boundary of the surface, then it turn to 3-Dimensional surface using the tool extrude.

After completing the geometry, it was exported for meshing job in another software Gambit 2.3.16. The geometries have lot of open space, this have to close for the meshing job. So virtual edges have created first then faces last virtual volume have created. Mainly there are three main volumes such as heat transfer, cold fluid and hot fluid, in which there are 23 faces.

The mesh finalization is done before analyzing the geometry. Different mesh configurations starting with very coarse to very fine are taken at a particular input parameter. The grid optimization or grid independence has to take into account. The grid independence test carried out for the same geometry with same input parameter and the graphs are plotted as the number of elements versus hot fluid outlet temperature and cold fluid out let temperature as shown in figure, Fig.3.3.

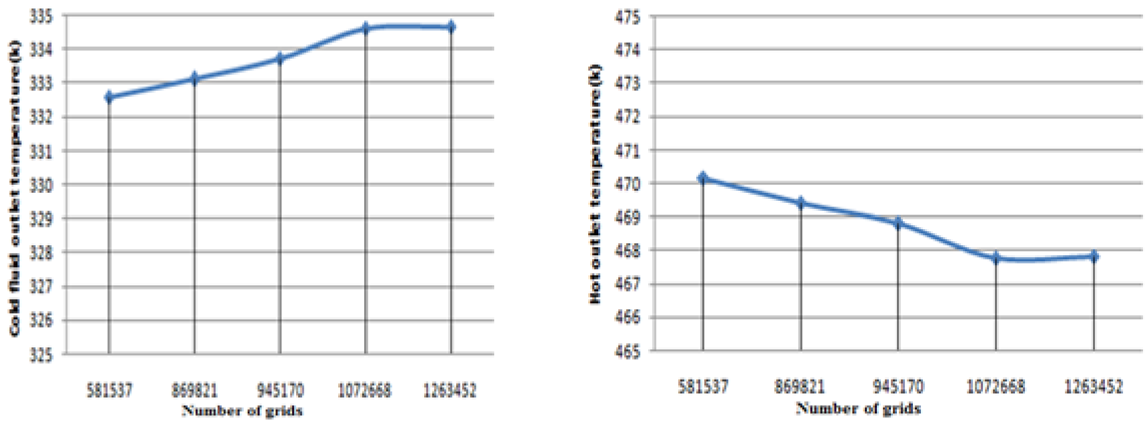

Fig 3.1.Grid independency graph

The figure Fig 3.1 shows, after 1072668 cells there is not much variations in the hot and cold fluid outlet temperature. Hence, the minimum grid size greater than 1072668 cells maintained throughout the analysis.

3.2 Analysis

This is the second step in the hierarchy of the design process, in which the problem has to solve using CFD code. For doing the calculation, we export the meshed geometry to analyzing software Fluent6.3.26. First check meshes of the given geometry, for example in the geometry CC-4 -45^0, FLUENT count **1072668** cell. Problem defining in Fluent is,

Model Settings	
Space	3D
Time	Steady
Viscous	Standard k-epsilon turbulence model
Discretization Scheme	
Pressure	Standard
Momentum	First Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind
Energy	First Order Upwind

Including all these, boundary conditions also mentioned in the fluent. Inlet velocity of both cold and hot fluid, inlet temperature of both fluid, and pressure out let of both the fluids have discussed.

3.2.1 Convergence criteria

Convergence criteria test to show the level of convergence and beyond which result are not much varied. The convergence criterion of 10^{-4} for the continuity equation and k-epsilon equation and convergence criterion of 10^{-6} for the energy equation is adopted for the entire analysis. The convergence criterion with respect to the number of iteration is shown in Fig.3.2

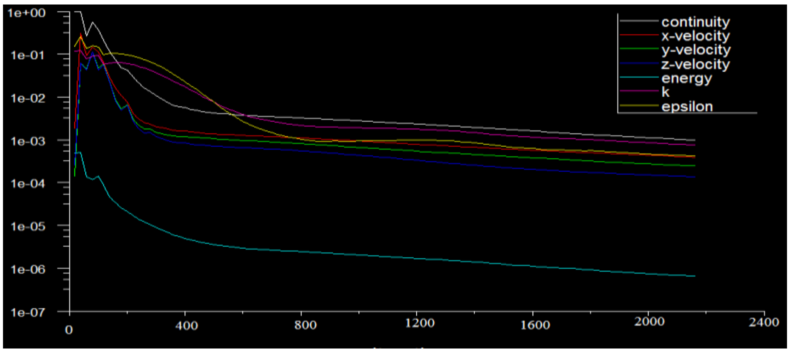

Fig 3.2 Convergence graph

3.2.2 Model validation

The CFD data of subject heat exchanger, a corrugated plate has been analyzed and compared with available published data. The results are found in good agreement with lots of experimental data conducted in various conditions

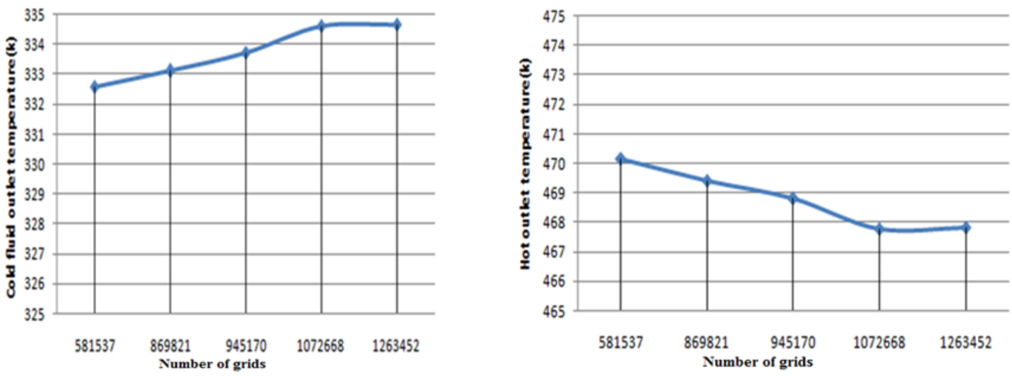

Fig 3.1.Grid independency graph

For the validation of numerical analysis conducted in the present study, two of the corrugated fins taken from the J. Stasiek. et al.[4,5] have been analyzed for the following cases. Validate our result with the experimental result of the M.Ciofalo , J. Stasiek. and m. W. Collins.(1996,1998,2000). They conducted experiments on corrugated surfaces with different pitch –height ratio and different Reynolds number.

For a comparison of experimental results and other literature data, it is important to ensure that all variables that affect heat transfer characteristics in corrugated plate heat exchangers are taken into account. Some of these variables are geometric parameters, such as, corrugation angle, axial length of cycle, width, length. The CFD results of corrugated plates are compared with M.Ciofalo , J. Stasiek as shown in Fig.3.4 a and b. The CFD results are in good agreement with experimental results given by M.Ciofalo , J. [Stasiek4,5] for the low and high Reynolds number region. The very small variations are found in the values. However, some more variations are observed in some region with low p/h ratio when compared with FLUENT data. Giving exact reasons for the variations of these factors may not be possible due to involvement of so many parameters.

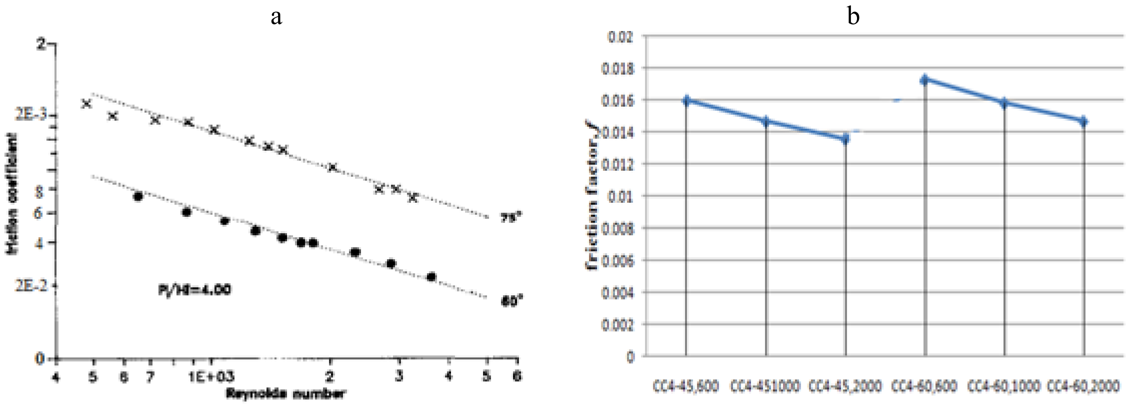

Fig.3.4. Validation Graph -2

The conclusion from the validation of the result obtained from the numerical simulation, the numarical calculations are almost similar to the prevously obtained result experimental and analytical study of various researchers in allover the world.

IV. Result And Discussion

The post processing was carried out in FLUENT. The main factors affecting the heat transfer and flow characteristics of the CC heat exchanger surface are the pitch-over-height ratios P/H and the corrugation β (9). Using commercial software CFD preprocessor Catia and Gambit to build the three-dimensional model and divide the unstructured grid mesh. The division of the grid is non-uniform: the grid mesh near the inlets and outlets is more intensive than that in the middle part, so that to solve the high temperature and velocity gradient near the wall effectively[13, 14]. The simulations have carried out by almost constant Reynolds number ranges from 600-700 (in corrugation geometries, Re > 150, become turbulent). Therefore k-ε turbulent model is used for the simulation.

In present study, we analyzed nine different geometries with constant Reynolds number we found some characteristics of the heat transfer surfaces. In the first graph it shown that the variation of temperature with different geometrical parameters. The graph actually showed that variation in the maximum temperature of cold fluid and the minimum temperature of the hot fluid.

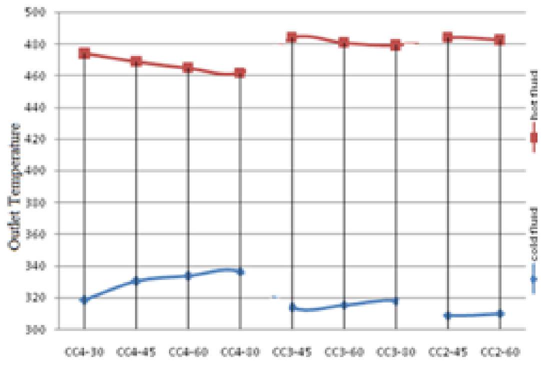

Fig.4.1. Outlet Temperature V/S different geometries

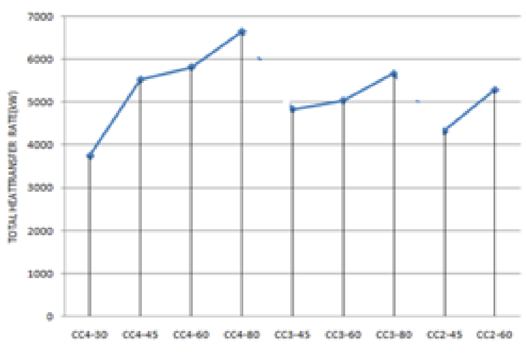

Fig4.2 Total heat transfer V/S different geometries

We can observe that the best geometry is CC4-80, because it got very high cold fluid temperature and very low hot fluid temperature which means in CC4-80 has maximum transfer of heat between cold fluid and hot fluid. The reason for obtaining maximum heat transfer in this geometry is due to the availability of maximum transfer surface area, fluid moves in a large angle zigzag motion. If considering other geometries CC4-45 and CC4-60 also have better performance. From CC4-45 to CC4-80 not a steep growth, considerable changes obtained. In the Fig.4.2 shows the heat transfer rate of the surface in between the fluid. Maximum heat transfer rate obtained in CC4-80.

For considering a heat transfer problem including conduction and convection, dimensionless parameter Nusselt number considered as the important design parameter. If the value of Nusselt number is unity, then it is pure conduction. Higher values means the heat transfer is enhanced by convection. A large Nusselt number means very efficient convection, for example, turbulent pipe flow yields a Nusselt number of order 100 to 1000.

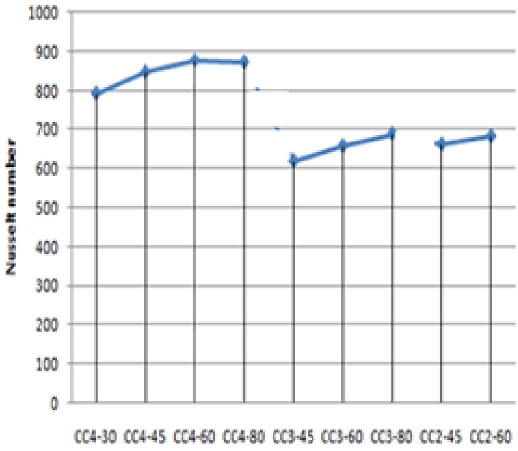

Fig.4.3. Nusselt number V/S different geometries

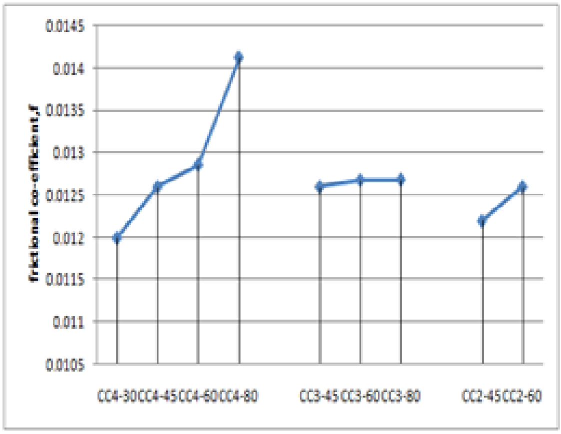

Fig.4.4. Friction factor V/S different geometries

Another important design parameter for heat transfer equipment is the pressure loss/head loss of the flowing fluid. Pressure drop is defined as the difference in pressure between two points of a fluid carrying network. Pressure drop occurs when frictional forces, caused by the resistance to flow, act on a fluid as it flows through the gap between the plates. When the flow is turbulent, the relationship becomes more complex. The equation for pressure loss calculation is,

$$\Delta P = \frac{G^2}{2g\rho_i}\left[f\frac{L}{r_h}\rho_i + \text{constants}\right] \tag{4.1}$$

The equation shows that the pressure loss mainly depends on the parameter called friction factor (f). From the Colburn analogy, friction factor f is equal to two times Colburn factor.

$$j = f/2 \tag{4.2}$$

Heat transfer rate is the important parameter for the design of heat exchangers. So according to this consideration CC4-80 is the best suitable geometry from our study. But pressure loss is another important parameter for design heat exchanger geometry. The pressure graph showed that the maximum pressure loss in the geometry CC4-80 and CC3-80. So these two geometries should not take into account for further processing. Here after we take only two geometries –CC4-45and CC4-60.

In CC4-45 and CC4-60 at Reynolds number 600 shows the maximum cold fluid temperature and minimum hot fluid temperature after CFD simulation. This means velocity of fluid inversely proportional to the transfer of heat energy.

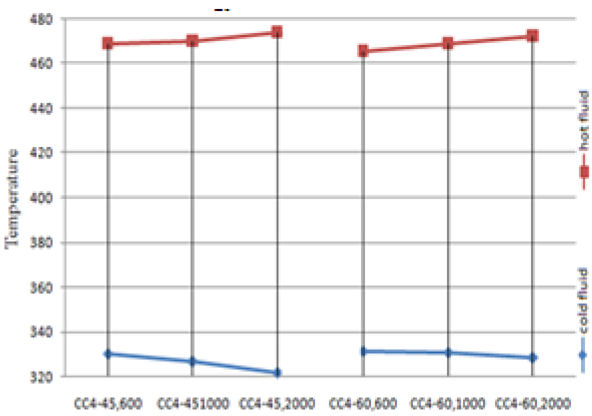

Fig.4.5. Outlet Temperature V/S different geometries

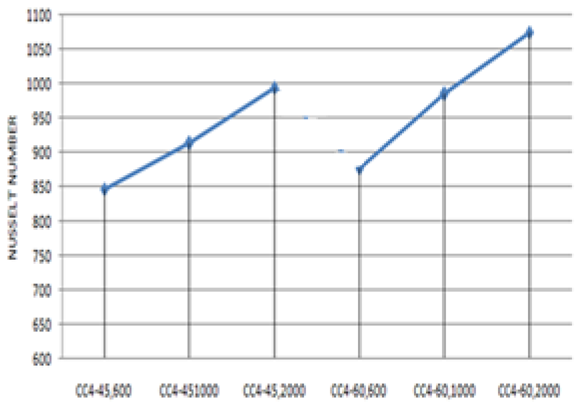

Fig.4.6. Nusselt number V/S different geometries

Due to the reduction of heat transfer Nusselt number also decreases. Heat transfer co-efficient is maximum at high Reynolds number and corrugation angle 60. This is because the Nusselt number is directly proportional to Reynolds number, where h is proportional to the Nusselt number.

The pressure drop is mainly due to the frictional effect and other flow restrictions. Here we can see the pressure drop is higher in the geometry CC4-60 at Reynolds number 2000, reason is due to the higher turbulence dissipation rate of kinetic energy,

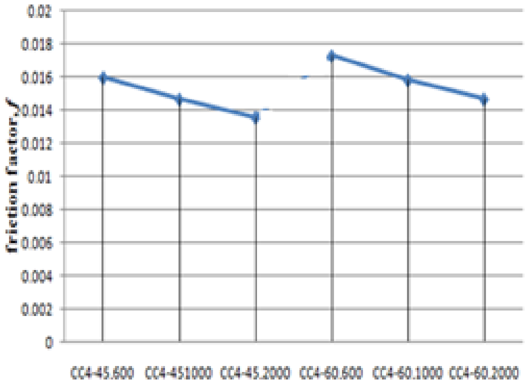

Fig.4.7. Friction factor V/S different geometries

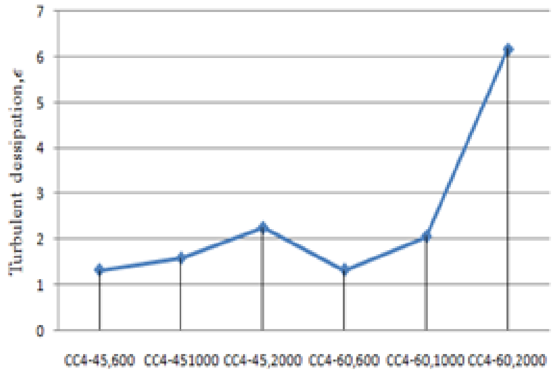

Fig.4.8. Turbulent dessipation V/S different geometries

From these discussions we found that the best geometries with pitch-height ratio four and the corrugation angle 45 or 60. Analyzing these geometries with different Reynolds number, we found that the geometry with angle 45 is the best suitable design for the varying velocity flow field heat exchangers. The reason is that the heat transfer rate of CC4-45 is almost equal to the geometry CC4-60. And the pressure drop characteristics is much better than CC4-60 with different velocities. Much more detailed study of the geometry CC4-45 as discussed in the following sections.

4.1. CFD predictions

4.2.1. Flow distribution

The flow distribution of complex 3D corrugated plate geometry with angle 45 degree as shown in Fig4.9. The flow path of the fluid on the heat transfer surface is shown in Fig.4.13and Fig.4.14. The arrows of the fluid show the zigzag motion of the fluid which helps heat exchanger to transfer maximum amount of energy. Actually this motion provides maximum contact between adjacent fluid layers, this helps for convective transfer of heat.

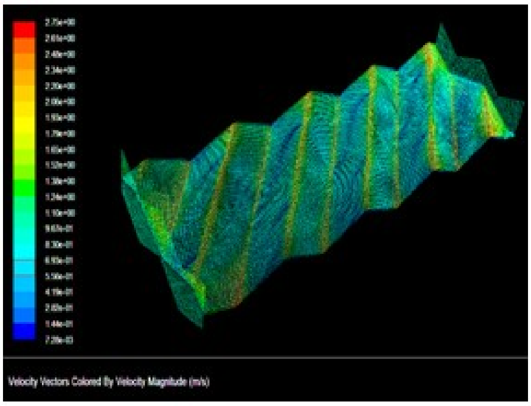

Fig.4.9. Flow distribution over the corrugated surface

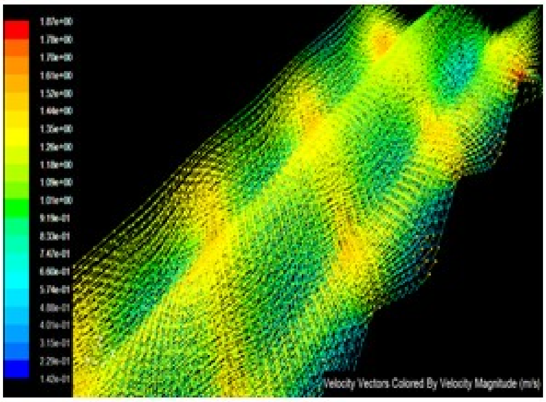

Fig.4.10. Flow distribution on CC4-45 geometry

The angle of view has been especially chosen so as to clearly show the effect that results from the shear between the fluid traveling along one of the troughs have on the fluid in the other trough. In the fluid flow a secondary flow fields are come to play. It is the whirling motions in the corrugated part of the heat exchanger. Which increase heat transfer rate and also increase the pressure drop due to the formation of eddies. The fig shows the flow path and also some eddy motions in the flow.

A. **B.**

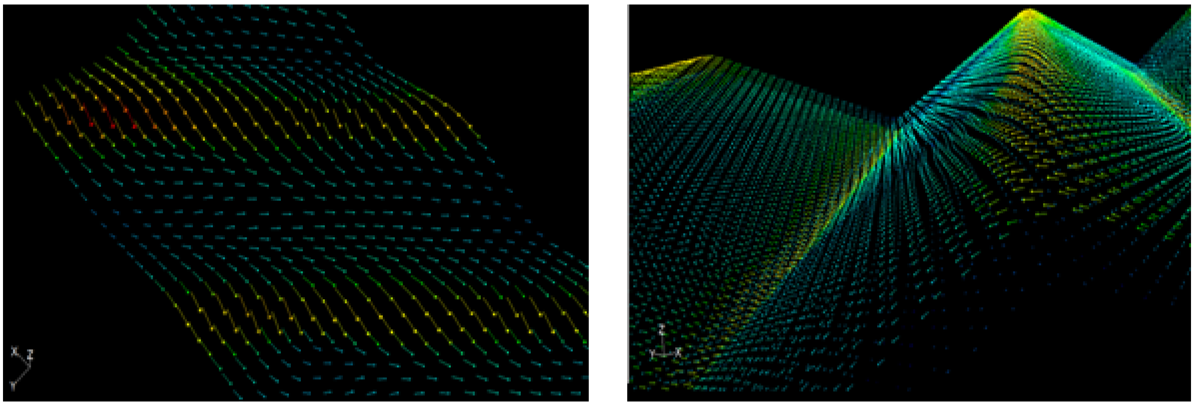

Fig.4.11. Flow distribution of a fluid on the heat transfer surface

4.2.2. Pressure and temperature distribution

Pressure distribution on the surface of computational domain is illustrated in Fig.4.12. At the inlet of the corrugation geometry with higher pressure shown in red color, after moves on the surface heat transfer pressure get reduced. This is due to frictional effect.

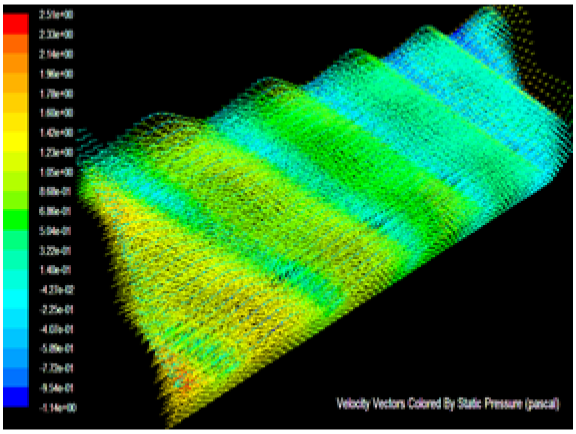

Fig.4.12. Pressure variation of fluid on H.T surface

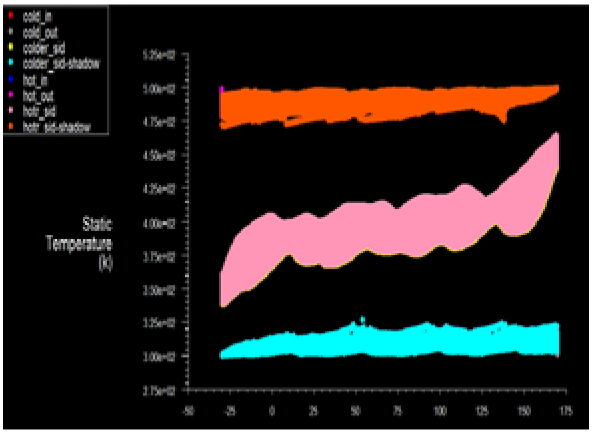

Fig.4.13. Temperature distribution over the H.T surface

The graph between static temperature and position of the heat exchanger as showed in fig.4.13, which characterizes the temperature distribution of cold fluid and hot fluid through the heat transfer surface. The upper portion shows the hot fluid with inlet temperature 500^0k and outlet of hot fluid with varying with temperature. Similarly in lower portion of the graph shows cold fluid with inlet temperature as 300^0k. In the middle portion shows the heat transfer surface, pink color for the surface nearer to the hot fluid and yellow color shows the colder side of the heat exchanger.

V. Conclusion

This paper reports the successful outcome of predictive study of the problem of understanding the complex local flow and heat transfer patterns in the crossed-corrugated geometry, which are widely used in the recuperators, intercoolers in the micro level turbo machineries. With CFD prediction, in the present problem, we showed the relations between the various parameters. The effects of corrugation angle, geometry and Reynolds number have all been investigated. This is with a view to optimizing the design, as used in the regenerators; however, the results are of interest to compact heat exchanger design generally.

In the present study my contribution is to find the reason for the high performance of the heat transfer surface with other geometry. The result is due to the flow distribution of the cold and hot fluid. In which the hot and cold fluid are flows over the corrugated heat transfer plate with zigzag motion. Actually this is due to the arrangement of corrugated plates. This zigzag motion allows the contact of fluids with heat exchanging surface and increase surface area contact. The close observation in fluid flow founds another flow region, which is the secondary motion of flow due to turbulence. The frequency of swirling motion increases with increase of the Reynolds number. And also observe that the turbulent flow occurs only when the Reynolds number should be greater than 200.

References

[1] London AL, Shah RK, Offset rectangular plate fin surfaces- heat transfer and flow friction characteristics. ASME J Eng Power 90, 1968,218–228.

[2] Yue-Tzu yang, Numerical simulation of fluid flow and heat transfer characteristics in channel with V corrugated plates, Springer, Heat mass transfer 46, 2010, 437-445.

[3] Wenfen Gao, Wenxian Lin,Tao Liu, Analytical and experimental studies on the thermal performance of cross-corrugated and flat-plate solar air heater, Elsevier, Applied energy, 84, 2007, 425-441

[4] J. Stasiek, Experimental studies of heat transfer and fluid flow across corrugated-undulated heat exchanger surface, Int. J. Heat Mass Transfer 41 (6–7), 1998, 899–914.

[5] J. Stasiek, Investigation of flow and heat transfer in corrugated passagesses-1. Experimental results, Elsevier, Int. J. Heat Mass Transfer, vol.39, 1., 1996, PP149-164.

[6] M.Ciofalo, J.stasiek, Investigation of flow and heat transfer in cross- corrugated-undulated plate heat exchangers, Springer, Heat Mass Transfer, 36,2000, 449-462.

[7] E. Utriainen, B. Sunden, Evaluation of the cross corrugated and some other candidate heat transfer surfaces for micro turbine recuperators, Trans. ASME 124, 2002, 550–560.

[8] V. Kumar, S. Saini, M. Sharma, K.D.P. Nigam, Pressure drop and heat transfer study in tube-in-tube helical heat exchanger, Chem. Eng. Sci. 61, 2006, 4403-4416.

[9] B. Sundén, Computational fluid dynamics in research and design of heat exchangers, Heat Transf Eng. 28 (11),2007, 898-910.

[10] A.G. Kanaris, A.A. Mouza, S.V. Paras, Flow and heat transfer prediction in a corrugated plate heat exchanger using a CFD code, Chem. Eng. Technol. 8, 2006, 923-930.

[11] K. Kritikos, C. Albanakis, D. Missirlis, Z. Vlahostergios, A. Goulas, P. Storm, Investigation of the thermal efficiency of a staggered elliptic-tube heat exchanger for aero engine applications, Appl. Therm Eng. 30,2010, 134-142.

[12] J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, Experimental and CFD estimation of heat transfer in helically coiled heat exchangers, Chem. Eng. Res. Des 86,2008, 221-232.

[13] E. Utriainen, B. Sunden, Evaluation of the Cross Corrugated and Some Other Candidate Heat Transfer Surfaces for Micro turbine Recuperators, ASME, Journal of Engineering for Gas Turbines and Power JULY, vol. 124,2002, 550- 560.

[14] Bengt Sunden, High temperature heat exchangers. proceedings of fifth international conference on enhance, compact and Ultra-Compact. Engineering Conf. International, Hoboken, NJ, USA, 2005, pp.226-238.