Simulation of Heat Transfer Enhancement in Corrugated Channel by Numerical Investigation

Rohan Kumar[1], Rajnish Kumar[2], Ashish Chaturvedi[3]

[1]MTech Schlor, [2,3]Assistant Professor

Me Department oriental College of Technology Bhopal, India

Abstract: In the current work, a numerical exploration is made on the flow characteristics and improvement of heat transfer in 2D channel with wavy wall covering a broad range of Reynolds numbers. For enhanced understanding, the numerical analysis is carried out by considering three different wall-geometry (triangular, sinusoidal, and trapezoidal corrugated wall). The outcomes are examined by drawing graph of the wall-Nusselt number along the channel length by varying the operating parameters like Reynolds number, amplitude of geometry, heat flux. The flow characteristics (such as pressure, temperature, velocity) deviation down the channel are also being examined to encapsulate the hydrodynamics. The two modes of boundary conditions employed are constant heat flux and constant wall temperature at the channel-wall, at the inlet velocity is completely stated, and atmospheric pressure is specified at outlet. The fluid used for the simulation is water. It is perceived that with the increase in the geometry amplitude and Reynolds number there is significant enhancement in heat transfer. It is obtained from the analysis that the heat transfer rate is maximum with triangular channel and the pressure drop is minimum with triangular channel.

Keywords-Heat transfer enhancement, corrugated wall, Wall-Nusselt number, trapezoidal, sinusoidal, triangular.

I. INTRODUCTION

Heat transfer augmentation with nominal pressure drop by any heat exchanging devices is extremely important phenomena within the thermal engineering. It has a bunch of application in Heat exchangers, method industries, Evaporators, Condenser, Thermal power plants, Air-conditioning systems etc. Overheating is the major problem associated with any power plants that triggers the failure of the system and the efficiency of the system is also reduced due to loss of heat in various forms. To overcome this problem effective cooling is required for which a heat exchanger is employed. A heat exchanger is a device that transfer heat from hot fluid to the cold fluid with maximum rate and minimuminvestment. Employing heat exchanger also improve the efficiency of the system for super-heater, feed hot-water heater, condenser, air pre-heater used in power plant is used to increase the efficiency of the system. The two major parameters associated with heat exchanger are heat transfer rate and the pressure drop across it (if it is high then additional power would need to pump the fluid). In a heat exchanger device heat transfer takes place mainly due to convection and from newton's law of cooling for convection heat transfer depend on surface area exposed and difference of wall temperature and fluid temperature. Since temperature difference can be varied only to certain limit, other ways to improve the heat transfer rate is by either varying heat transfer coefficient or to vary the area exposed in such a way that it has minimum pressure drop across it. A number of the ways to improve the warmth transfer rate are given below.

a. Active Method: This method is based on the forced convection that is an external devices like blowers, pumps, fan etc. are used to agitate the fluid. Due to which convective heat transfer coefficient increases.

b. Passive Method: This technique based on the surface treatment method without aid of any external power device. Various surface treatment like: imposing surface roughness on the wall, use of baffles or fins, changing the shape of the wall of pipe/channel (corrugated wall), etc are used.

c. Compound Method: This method is the amalgamation of the two methods discussed above.

This thesis is solely based on the passive method of heat transfer enhancement.

Corrugated Channel: The term corrugated means that wavy or uneven that's, it consists series of repetitive and parallel formed wall such as: curving, triangular grooves, square, trapezoidal etc. As a result of this uneven geometry, it creates the disruption within the flow and causes the reversal or recirculation of the flow. Recirculation regions at the wall boosts the blending of the fluid and diminish thermal boundary layer, which results in rise of the heat transfer rate.

II. METHODS

2.1. Methodology

The present work has been done by usingansys fluent-16. For this a steady state fully developed incompressible flow through 2-D corrugated channel is taken into account. The channel consists of two identical furrowed plates (in opposite phase) separated by a minimum distance of Hmin and maximum distance of Hmax. Constant heat flux or wall temperature boundary conditions with no slip and no penetration boundary condition are used at the channel-walls. Here water is taken as the working fluid. Three different geometry (sinusoidal, trapezoidal, and triangular) with varying amplitude that are considered is shown below.

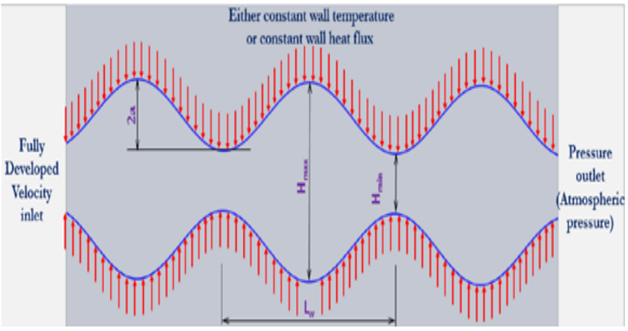

Fig 1. Sinusoidal geometry

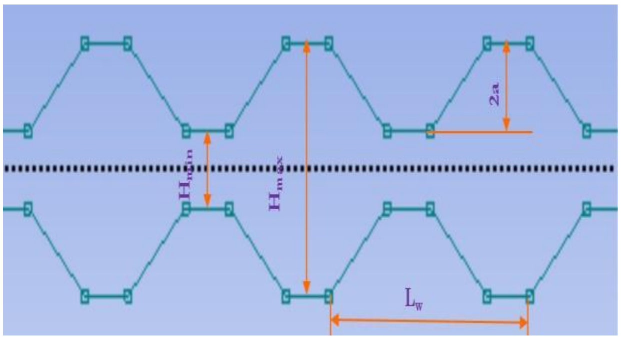

Fig 2. Trapezoidal geometry

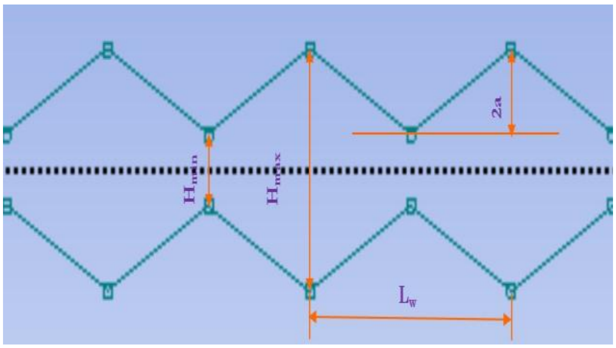

Fig 3 Triangular geometry

Table 1. Overall specifications of the channel with totally different amplitudes

Amplitude (a) in mm	Wavelength (L_w) in mm	Minimum space (H_{min}) in mm	Maximum space (H_{max}) in mm
1.75	28	6	13
3.50	28	6	20
7.00	28	6	34

The problem is solved using two boundary condition i.e constant heat flux (50, 100, 150, 200 kW/m2) and constant wall temperature (365, 367, 369.25 K) for both laminar as well as turbulent flow. The condition of laminar flow is taken for the Reynolds number 500, 1000, 1500, and 1900. For turbulent flow Reynolds number taken is 5000 and 7000.

2.2. Procedure

• Creation of geometry: The above shown geometry are drawn in ansys fluent-16 workbench. The properties for which are kept in 2-D. and with the help of "surface from sketch" option the drawn geometry is made to surface.

• Meshing: once the geometry is made, we have to mesh it. For this purpose a tetrahedron meshing is employed. To capture the wall effect as well as to save the computation time, finer grid are selected near wall and coarser at the middle portion of the channel. The meshing of all geometry are shown below:

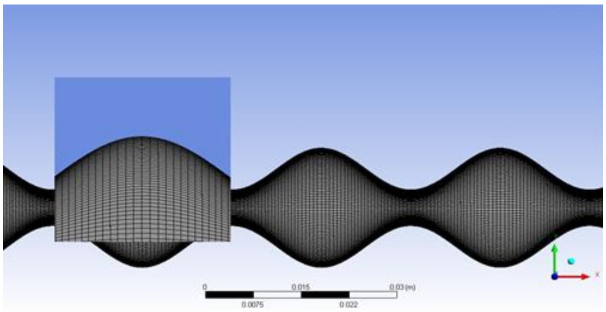

Fig 4. Pattern of grid for sinusoidal corrugated channel

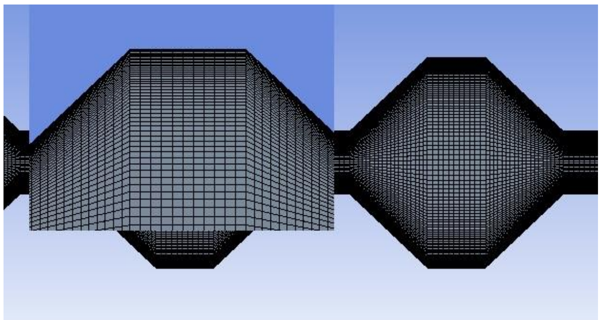

Fig 5. Pattern of grid for trapezoidal corrugated channel

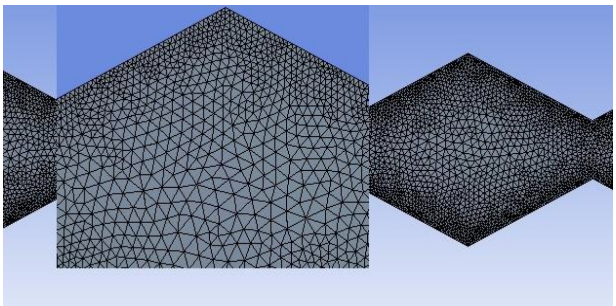

Fig 6. Pattern of grid for triangular corrugated channel

• Boundary condition: The boundary conditions applied at the channel wall is either constant heat flux (q = constant) or constant wall temperature (Twall = Constant) with no slip (Ux = 0) and no penetration (Uy = 0) and water inlet temperature is fixed to 300 K (Tin, watert = 300 K). At the inlet, ‘Velocity inlet ‘boundary condition is used and the fully developed velocity-profile is specified by using user defined function. At the channel outlet ‘pressure outlet ‘boundary condition (Pout = Patm) is used and 1 atm is fixed at the outlet.

• For discretization of the equation least square cell based method is used. For coupling pressure and velocity the simple algorithm is incorporated. The energy and the momentum equation was discretize by using the Second order upwind scheme. The relaxation factors for controlling the parameters were kept default as shown in the problem initialize using standard initialization method and computed the solution from all zone.

III. RESULT AND DISCUSSION

Results on the geometry considered shows that surface Nusselt number increases as we increase the Reynolds number, heat flux, wall temperature and amplitude of geometry. Various graph has been plotted for surface nusselt number along the length as shown below:

1. Effect of Reynolds number on surface Nusselt number:

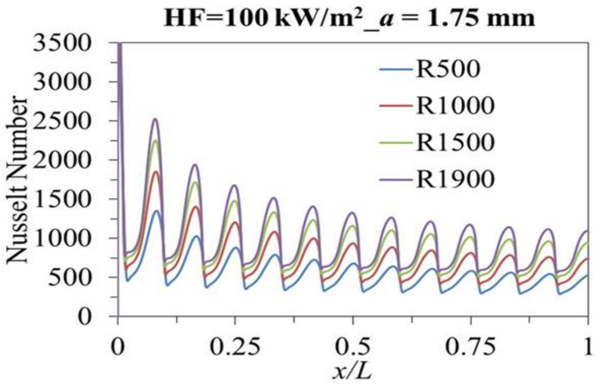

Fig 7. For Sinusoidal Geometry (laminar flow).

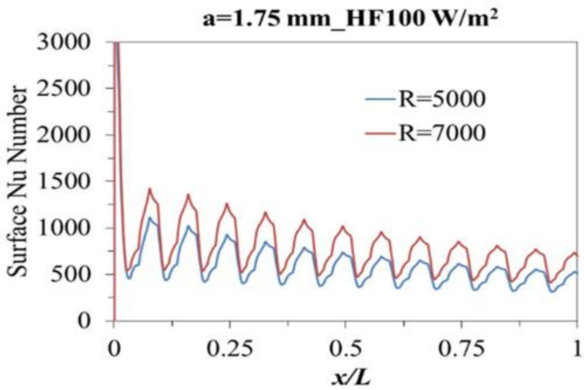

Fig 8. For Triangular Geometry (turbulent flow)

2. Effect of heat flux on surface Nusselt number:

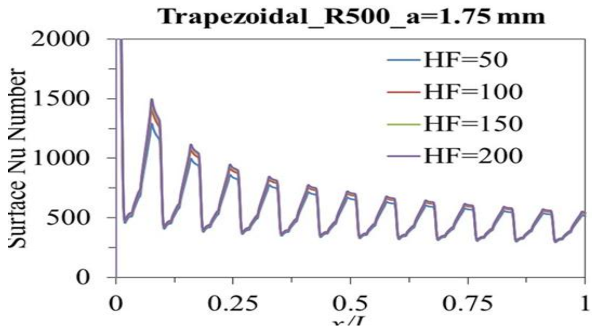

Fig 9. For trapezoidal profile (laminar flow)

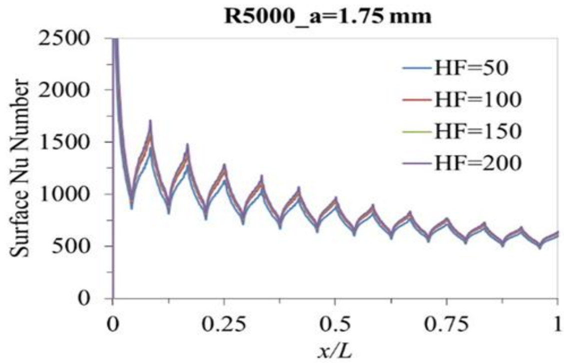

Fig 10. For triangular profile (turbulent flow

3. Effect of amplitude on Nusselt number:

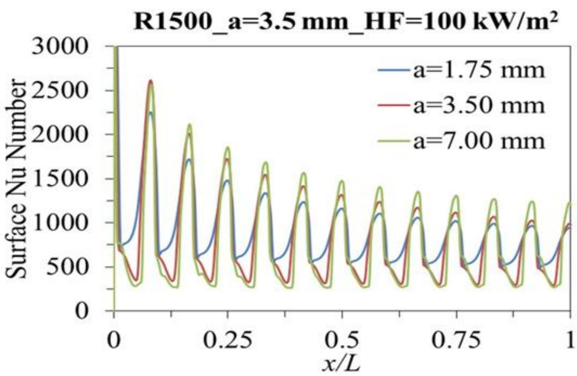

Fig 11. For sinusoidal profile (laminar flow).

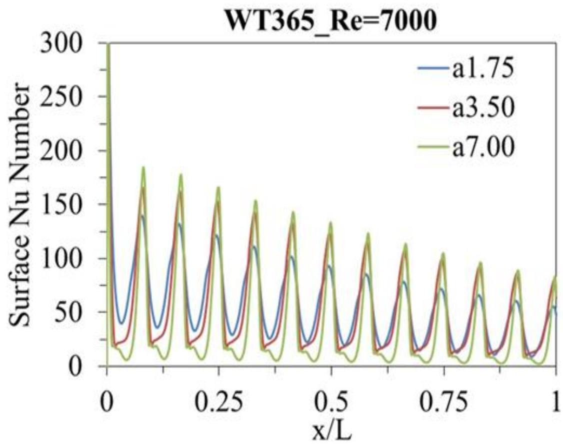

Fig 12. For sinusoidal profile

4. Effect of geometry profile:

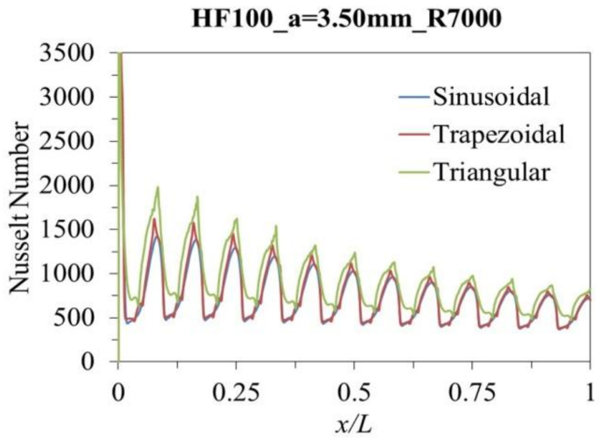

Fig 13. Comparison between geometry

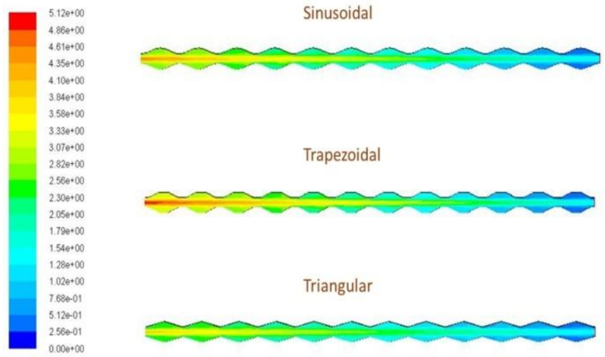

Fig 14. Effect of the geometry profile on the total pressure. Wall heat flux=200 kW/m2, a=1.75 mm and Re = 500

IV. CONCLUSION

The conclusion drawn from the simulation are listed below:

1) Heat transfer enhancement is maximum for triangular corrugated channel followed by trapezoidal and triangular channel for laminar flow.

2) In case of turbulent flow, the triangular corrugated channel shows the better heat transfer rate than others two corrugated channel.

3) Pressure loss is maximum in case of trapezoidal corrugated channel followed by sinusoidal and triangular channel for laminar flow.

4) For higher Reynolds number the surface-Nusselt number is higher.

5) On increasing the amplitude of wavy wall, the Nusselt number increases.

6) With increment of Re, outlet temperature decreases.

V. ACKNOWLEDGMENT

We are using the conceptual knowledge to do some useful work for the benefit of our society. This work is being done in the guidance of Assistant professor Mr. Rajnish Kumar who showed confidence in me and showed me different approach to achieve my goal. I am thankful to work with him.

REFERENCES

[1] Islamoglu, Y., Parmaksizoglu, C., 2003. The effect of channel height on the enhanced heat transfer characteristics in a corrugated heat exchanger channel. Applied Thermal Engineering 23, 979-987.

[2] Jafari, M., Farhadi, M., Sedighi, K., 2013. Pulsating flow effects on convection heat transfer in a corrugated channel: A LBM approach. International Communications in Heat and Mass Transfer 45, 146-154.

[3] Ahmed, M. A., Yusoff, M. Z., Ng, K. C.,Shuaib, N. H., 2014. Effect of corrugation profile on the thermal–hydraulic performance of corrugated channels using CuO– water nanofluid. Case Studies in Thermal Engineering 4, 65-75. Mohammed, H. A., Abed, A. M., Wahid, M. A., 2013. The effects of geometrical parameters of a corrugated channel with in out-of-phase arrangement. International Communications in Heat and Mass Transfer 40, 47-57.

[4] Paisarn, N., 2007. Laminar convective heat transfer and pressure drop in the corrugated channels. International communications in heat and mass transfer 34, 62-71.

[5] Pehlivan, H., Taymaz, I., İslamoğlu, Y., 2013. Experimental study of forced convective heat transfer in a different arranged corrugated channel. International Communications in Heat and Mass Transfer 46, 106-111.

[6] Pethkool, S., Eiamsa-ard, S., Kwankaomeng, S., Promvonge, S., 2011. Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube. International Communications in Heat and Mass Transfer 38, 340-347.

[7] Umavathi, J. C., Shekar, M., 2014. Mixed convective flow of immiscible fluids in a vertical corrugated channel with traveling thermal waves. Journal of King Saud University-Engineering Sciences 26, 49-68.

[8] Ozbolat, V., Tokgoz, M., Sahin, B., 2013. Flow Characteristics and Heat Transfer Enhancement in 2D Corrugated Channels. International Journal of Mechanical, Aerospace, Industrial, and Mechatronics Engg. 7, 1011-1015.

[9] Yin, J., Yang, G., Li, Y. 2012. The effects of wavy plate phase shift on flow and heat transfer characteristics in corrugated channel. Energy Procedia 14, 1566-1573.