Hydrodynamic Analysis of a Wavy Channel using Finite Volume Approach

Navneet Kumar[1] Abhishek Shukla[2]
[1,2]Department of Mechanical Engineering
[1,2]SSTC(SSEC) Bhilai CG, India

Abstract— In this article comparative CFD analysis between a wavy channel and straight channel has been presented. For this purpose a two dimensional model of straight and wavy has been developed by using ANSYS geometrical module. The flow analysis has carried out in ANSYS Fluent where the governing equations have been solved by ANSYS Fluent module. The hydrodynamic characteristics of both wavy and straight channel have been examined and the purpose of using wavy channel in industrial application has been discussed in detailed. The obtained finite volume results have been validated with the result of published data and the results show good convergence. During comparison of hydrodynamic characteristic's wide range of Reynolds number has been considered i.e. from laminar to highly turbulent.

Key words: ANSYS, Baffles, Heat Transfer, Pressure Drop, Turbulence

I. Introduction

Wavy wall flows arise under a broad range of engineering applications and have, consequently, received considerable attention [1–2]. One of the most significant applications is the heat transfer enhancement in heat exchangers. The physical process of enhancing heat transfer in such application is to introduce some geometrical modifications on the wall in question in order to break the boundary layer that forms on the exchanger wall and replace it by a fresh fluid from the free stream flow [3]. In real applications, engineers are also interested in the additional pressure drop caused by such techniques. So, the best solution is that provides the least pressure drop and the largest heat transfer rate. Other parameters such as simplicity, manufacturability, maintenance, etc., are also important parameters in the design phase [3].

II. Literature

Valinataj et al. 2015 use artificial be colony approach in a sinusoidal wavy channel in order to optimize the heat transfer of twp phase modeling of a nano fluid. The effect of using nano-particles on thermal-hydraulic performance factor (j/f) has been examined which considers both heat transfer and hydrodynamics aspects.

Shyy Woei et al. 2015 examined the thermal performance of wavy sidewalls in rotating two-pass ribbed square channel. The Nusselt number and friction factor correlation has been developed along with the effect of Re, Ro, Bu.

Zachary et al. 2016 numerically investigate the heat transfer in wavy-walled channel. The heat transfer enhancement and thermal–hydraulic performance has been examined in the limelight of affect of geometry and driving pressure. They found that flow regimes and geometries increases the performance has compared with straight channel.

Khoshvaght-Aliabadi et al. 2016 numerically studied laminar convection of water and 1% vol. Al2O3–water nanofluid through the straight mini-channel (SMC) and wavy mini-channel (WMC) with various cross-section geometries.

RASHIDI et al. 2014 made an comparative numerical study of single and two-phase models of nano fluid heat transfer in wavy channel using CFD. The flow is examined for single phase and three different two-phase models and concludes the increasing the volume fraction of nano particles will enhance the heat transfer coefficient in the front and the middle of the wavy channel, but gradually decrease along the wavy channel.

Hafez et al. 2011 investigates the turbulent flow over a sinusoidal solid surface using two versions of the standard k–e turbulence model for examining the mainstream where periodic pressure gradient, successive acceleration and deceleration associated with multiple fluid flow separations and reattachments takes place. In order to verify the obtained results an exclusive comparison has been made between DNS and experimental.

Hang et al. 2005 Large eddy simulation (LES) has been applied to turbulent thermal fields in a channel having one wavy wall for Prandtl number = 0.7. Wall wave amplitude is changed in three steps. Increasing the wall wave amplitude, a flow separation bubble comes to appear and a separated turbulent shear layer develops above the separation bubble.

Stone and Vanka [6] perform transient state analysis for wavy channel consisting of 14 waves using a numerical scheme that solves the 2D energy equations. For wide range of Reynolds numbers Time-dependent simulations has been carried out. It is found that at low Reynolds number the flow is steady but on increasing Reynolds number the flow becomes unsteady which leads to increase in mixing between the wall fluids and core, thus increases the heat transfer rate.

III. Mathematical Modelling

For the fluid flow through pipe, duct and channel the conventional governing equations are the Navier–Stokes equations can be written in the most useful form for the development of the finite volume method:

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu\, grad u) + S_{Mx} \quad (1)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu\, grad v) + S_{My} \quad (2)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu\, grad w) + S_{Mz} \quad (3)$$

Governing equations of the flow of a compressible Newtonian fluid

A. Continuity

$$\frac{\partial \rho}{\partial x} + div(\rho u) = 0$$

1) x-momentum

$$\frac{\partial(\rho u)}{\partial x} + div(\rho u u) = -\frac{\partial p}{\partial x} + div(\mu\, grad u) + S_{Mx} \quad (4)$$

2) y-momentum

$$\frac{\partial(\rho v)}{\partial y} + div(\rho v u) = -\frac{\partial p}{\partial y} + div(\mu\, grad v) + S_{My} \quad (5)$$

3) z-momentum

$$\frac{\partial(\rho w)}{\partial z} + div(\rho w u) = -\frac{\partial p}{\partial z} + div(\mu\, grad w) + S_{Mz} \quad (6)$$

B. Energy

$$\frac{\partial(\rho i)}{\partial t} + div(\rho i u) = -p\, div u + div(k\, grad T) + \Phi + S_i \quad (7)$$

Using various correlation FEV results are been compared analytically

$$h_f = f \frac{L V^2}{D_h 2g}$$

Where,

f is the friction factor for fully developed laminar flow

L: length of the channel, duct, pipe

V: mean velocity of the flow

d: diameter of the pipe

f is the friction factor for fully developed laminar flow:

$$f = \frac{64}{Re} \text{ (For Re<2000)} \quad Re = \frac{\rho u_{avg} d}{\mu}$$

C_f is the skin friction coefficient or Fanning's friction factor.

For Hagen-Poiseuille flow

$$C_f = \tau_{wall} / \frac{1}{2} \rho u_{avg}^2 = \frac{16}{Re}$$

For turbulent flow

$$\frac{1}{\sqrt{f}} = 1.74 - 2.0 \log_{10}\left[\frac{\varepsilon_p}{R} + \frac{18.7}{Re\sqrt{f}}\right]$$ Moody's Chart

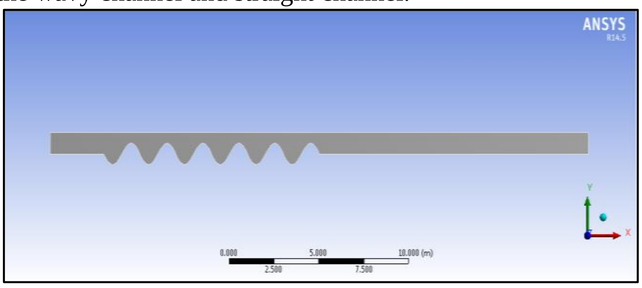
R: radius of the channel, duct, pipe

ε_p: degree of roughness (for smooth channel, duct, pipe, $\varepsilon_{p=0}$)

$Re \to \infty$: Completely rough channel, duct, pipe.

IV. METHODOLOGY

The ANSYS 14.5 finite element program was used for analyzing flow in wavy channel and straight channel. For this purpose, the key points were first created and then line and spline segments were formed. The lines were combined to create an area. Finally, this area was extruded a We modeled the wavy channel and straight channel.



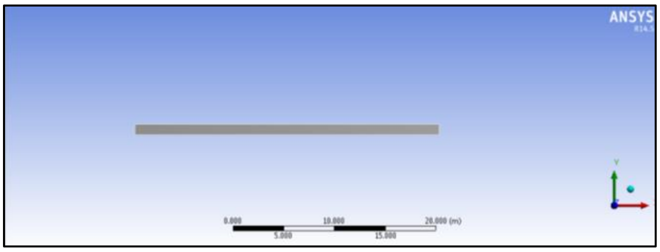


Fig. 1: Model Geometry

A 20-node three-dimensional structural solid element was selected to model the wavy channel. The wavy channel and straight channel was discretized into 6529 elements with 5990 nodes. The wavy channel boundary conditions can also be provided in mesh section through naming the portion of modeled Pipe i.e Inlet, Outlet, Top wall, Bottom Wall, wavy.

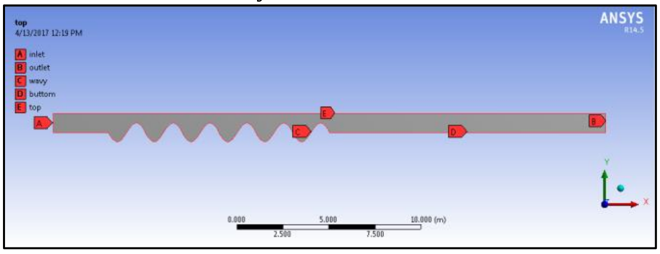


Fig. 2: Boundary Condition

V. RESULT AND DISCUSSION

The governing equations of the problem were solved, numerically, using a Element method, and finite Volume method (FVM) used in order to calculate the Hydrodynamic characteristics of a wavy channel and straight channel. As a result of a grid independence study, a grid size of 106 was found to model accurately the Hydrodynamic performance characteristics are described in the corresponding results.

The accuracy of the computational model was verified by comparing results from the present study with those obtained by Cokljat and S.E. Kim [11] and kuzan [10], Experimental, Analytical and FVM results.

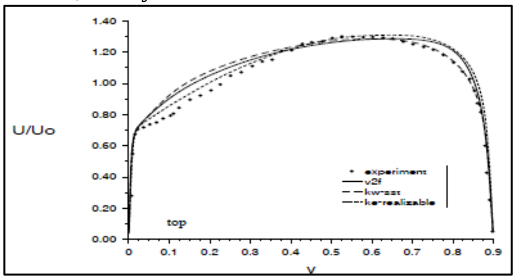


Fig. 3: Velocity profile of wavy channel [10]

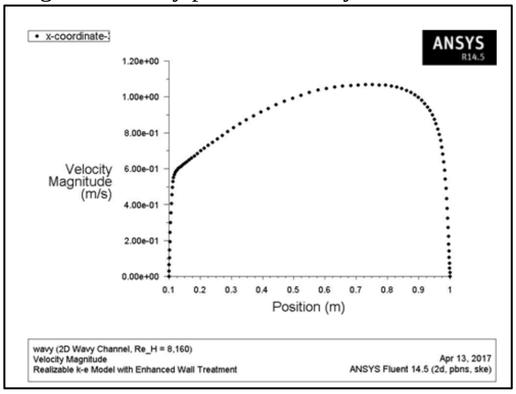


Fig. 4: Validation of velocity profile of wavy channel

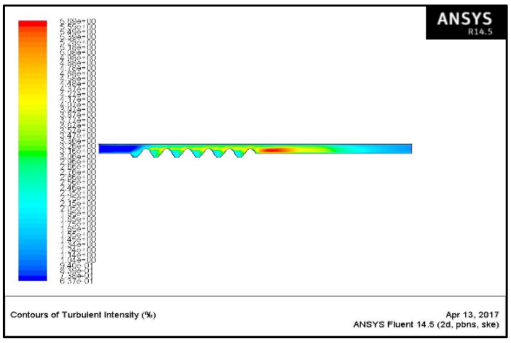


Fig. 5: Contour plot of turbulent intensity across wavy channel (a=2.5)

Figure 5 demonstrates Contour plot of turbulent intensity across wavy channel (a=2.5). It has been observed that the turbulent intensity increases as the wavy texture begins significant variation has been on observed in the turbulence intensity due to presence of wavy texture in the channel length.

Due to presence of wavy the turbulence increases because of boundary layer separation. Higher the wave amplitude the tendency of formation of turbulence is more. In figure red portion shows the high level of turbulence while the low portion shows the low level of turbulence region.

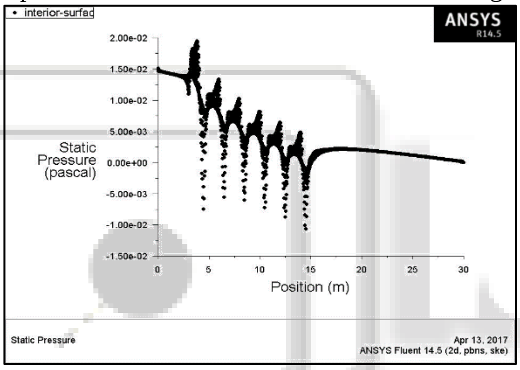


Fig. 6: Pressure drop across wavy channel at high Reynolds number

Figure 6 illustrates the pressure drop across wavy channel at high Reynolds number. It has been observed that the pressure across the wavy channel continuously goes on decreasing. It is interesting to know that across the wavy pattern the variation in pressure has been seen in term of increase and decrease.

It has also been seen that for wavy channel having high amplitude the variation across the wavy texture is quite high as compared to wavy channel with low amplitude.

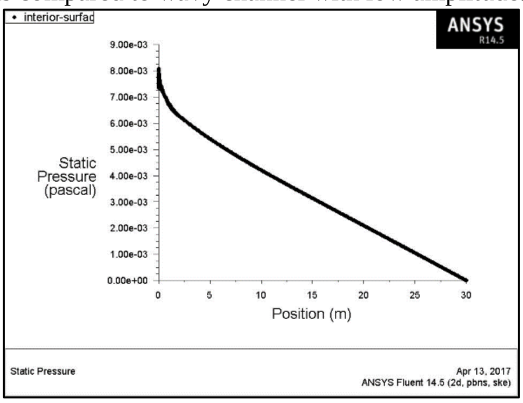


Fig. 7: Pressure drop across straight channel at low Reynolds number

Figure 6 shows the pressure drop across the wavy channel at different Reynolds number. It has been observed that the pressure significantly decreases throughout the channel length. It is interesting to know that after 3m the wavy texture in channel begin and the variation in pressure drop can be seen in zig-zag pattern this is due to separation of viscous sub layer. While in straight channel this drop in pressure is linear throughout the channel length, which can be evident from figure 7.

VI. Conclusions

- In comparison with straight surface wavy surface creates much turbulence at higher Reynolds number.
- The pressure drop can be reduced by implementing wavy surface instead of flat surface.
- Increasing the number of wave in the channel increases the rate of turbulence intensity.
- The separation of shear layer and the formation of vortex at near-wall stream, play an important role for the heat and momentum transfer near the wavy surface.
- Increasing wavy amplitude the better heat transfer can be achieved.

References

[1] Vijay KG; Maji PK (1988) Laminar flow and heat transfer in a periodically converging diverging channel. Int J Numer Meth Fluids 8: 579–597
[2] Patel VC; Tyndall Chon J; Yoon JY (1991) Turbulent flow in a channel with a wavy wall. J Fluids Eng 113: 579–586
[3] Wang G; Vanka SP (1995) Convective heat transfer in periodic wavy passages. Int J Heat Mass Transfer 38(17): 3219–3230
[4] P. Valinataj-Bahnemiri, A. Ramiar, S.A. Manavi, A. Mozaffari, Heat transfer optimization of two phase modeling of nanofluid in a sinusoidal wavy channel using Artificial Bee Colony technique, Engineering Science and Technology, an International Journal, Volume 18, Issue 4, December 2015, Pages 727-737
[5] Shyy Woei Chang, Tong-Miin Liou, Jing Yan Gao, Thermal performance of rotating two-pass ribbed square channel with wavy sidewalls, Experimental Thermal and Fluid Science, Volume 68, November 2015, Pages 412-434
[6] M. Khoshvaght-Aliabadi, S.E. Hosseini Rad, F. Hormozi, Al2O3–water nanofluid inside wavy mini-channel with different cross-sections Journal of the Taiwan Institute of Chemical Engineers, Volume 58, January 2016, Pages 8-18
[7] M. M. RASHIDI, A. HOSSEINI1, I.POP, S. KUMAR, N. FREIDOONIMEHR, Comparative numerical study of single and two-phase models of nanofluid heat transfer in wavy channel, Appl. Math. Mech. -Engl. Ed., 35(7), 831–848 (2014)
[8] K.A. Hafez, O.A. Elsamni, K.Y. Zakaria, Numerical investigation of the fully developed turbulent flow over a moving wavy wall using k–e turbulence model, Alexandria Engineering Journal (2011) 50, 145–162
[9] Hang Seok Choi , Kenjiro Suzuki, Large eddy simulation of turbulent flow and heat transfer in a channel with one

wavy wall, International Journal of Heat and Fluid Flow 26 (2005) 681–694

[10] K stone and Vanka, Numerical Study of Developing Flow and Heat Transfer in a Wavy Passage, Journal of Fluids Engineering, 1999, 121, 713-719

[11] Kuzan, J. D., “Velocity Measurements for Turbulent Separated and Near-Separated Flows over Solid Waves,” Ph.D. Thesis, Department of Chemical Engineering, varsity of Illinois Stanford University, 1986)

[12] D. Cokljat and S.E. Kim, A Comparative Assessment of the V2f Model for recirculating Flows, 41st Aerospace Sciences Meeting and Exhibit 6-9 January 2003, Reno, Nevada.