International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 09 | Sep2019 www.irjet.net p-ISSN: 2395-0072

Numerical investigation of the Forced Convection using Nano Fluid

Vishawanath[1], Dr Pravin V Honguntikar[2]

M.Tech Student[1] Prof & HOD Mechanical[2]
M.Tech in Thermal Power Engineering, Poojya Doddappa Appa College of Engineering Kalaburagi-585102

Abstract - The performance of heat exchangers especially for single phase flows can be enhanced by many augmentation techniques. One of the most popular method used is a passive heat transfer technique. Researchers have been quite active in the search of novel ways on heat transfer augmentation techniques using various types of passive techniques to increase heat transfer performances of heat exchanger. Computational Fluid Dynamics (CFD) simulations of heat transfer and friction factor analysis in a turbulent flow regime in semi-circle corrugated channels with Tio2 Fe3O4 Sio2-water nanofluid is presented in this paper. Simulations are carried out at Reynolds number range of 10000-30000, with nanoparticle volume fractions 0-6% and constant heat flux condition. The results for corrugated channels are examined and compared to those for straight channels. Results show that the Nusselt number increased with the increase of nanoparticle volume fraction and Reynolds number. The Nusselt number was found to increase as the nanoparticle diameter decreased. Maximum Nusselt number enhancement ratio 2.07 at Reynolds number 30,000 and volume fraction 6%.

Keywords: Cfd, Corrugated, Forced Convection, Nano Fluid.

1. Introduction

The sudden compression or expansion in the channels flow is very important design in many practical applications for cooling or heating systems. The forward-facing and backward-facing steps are significant applications in these types of flow. A number of heat transfer industrial applications through facing step channel have been included in energy systems equipment, electronic cooling systems, chemical processes, combustion chambers, turbine blades cooling, environmental control systems and high performance heat exchangers. Particularly, the drop in pressure and heat transfer enhancement in the reattaching flow area and inside the reverse flow area was great. For example, the low pressure drop and the high heat transfer augmentation obtained near the wall channel region whereas the low rate of heat transfer gain at the corner where the sudden change occurs starts in flow region. On the other hand, the corrugated channels have been used in a wide range of practical applications to enhance heat transfer. The augmentation of heat transfer in these channels is dependent on bulk fluid mixing and re-initiation of the thermal boundary layer. Through the author's knowledge, investigations concern on nanofluid flow over facing step and corrugated channels were still not entirely understood. The main objective of this review is to summarize the recent studies of heat transfer enhancement through facing step and corrugated channels. In addition, this review will provide a proposed new type of flow in corrugated facing step channels for future work.

2. Heat transfer through facing step geometry

There is one separated region in the flow geometry of backwardfacing step, which is developed by the step downstream. Likewise, the flow geometry of forward-facing step as well as the flow field that more complicated and one or two separated regions can develop to one upstream and the other downstream from the step, which depends on the ratio of the approaching flow of thick boundary layer to the forward-facing step height at the step. Flow over a backward-facing step generates recirculation zones and forms vortices due to the separation flow obtained from the adverse pressure gradients in the fluid flow . The phenomena of flow separation are found in different applications such as heat exchangers,

nuclear reactors, power plants, cooling devices, etc. In the past decades, a number of works have been performed on this phenomena and its effect on heat transfer rate . The heat transfer and flow characteristics of the conventional fluids such as air and water through the facing step channels have been studied by many researchers. Hattori and Nagano have studied boundary layer turbulent flow through a forward facing step. The separate regions have occurred in the step and in front of the forward facing step flow. Sparrow and Chuck performed a study of heat transfer and fluid flow through a backward facing step numerically. They implemented a numerical finite difference for studying the airflow phenomenon over a two-dimensional channel that was heated at constant temperature at the bottom wall from the foot of the step to the end of the channel. The low values of Nusselt number at the step were observed furthermore, it advances regularly and achieve an optimum heat transfer augmentation near the wall. Beyond the optimum point, Nusselt number decreases regularly with fully developed value. This behavior reflects the re-attachment, re-development and separation by the flow experience. Chiang et al conducted a topological study of a 3D backward facing step channel to improve the visualization of oil flow field. The size of the roof recirculation zone was observed to be dependent on Reynolds number. Saldana et al. reported the numerical results of simulating airflow through a forward facing step horizontal channel under three different Reynolds values. The expansion ratio (E =2) and the aspect ratio (A =4) of rectangular forward-facing step channel have been considered. In addition, a recirculation area was developed next to upstream and the bottom step wall. The separation of flow occurs early with increasing of Reynolds number. The flow through a facing step has simulated numerically . The results showed that the separation flow occurs with the increasing of Reynolds number.

2. Literature Review

Nanotechnology would be noted as the most important locomotive for the major industrial revolution of the present time. The poor performance of thermal conductivity of conventional fluids such as air, water, oil, and ethylene glycol mixture is the primary restriction to enhance the performance of heat exchangers [1]. Nanofluids are considered by suspending nanoparticles in conventional base liquids, and also the random motion process and dispersion structure of the suspended nanoparticles are the investigation fields of nanofluids [2]. Xuan and Li [3] have experimentally investigated the heat transfer and flow field of copper-water nanofluids flowing through a tube. They conclude their investment for a range of Re (10,000-25,000) and volume friction (0.3-2%). Yang et al [4] have investigated experimentally the convective heat transfer of graphite in oil nanofluid for laminar flow in a horizontal tube heat exchanger. Santra et al [5] showed that the heat transfer owing to laminar flow of copper-water nanofluid through two-dimensional channel with constant temperature walls and they conducted that the rate of heat transfer enhancement with the rising in flow Reynolds number, Re, as well as the increase in solid volume fraction. Kakac et al [6] revealed the nanofluid flow can be considered as a single-phase incompressible flow. Their investigation of the simplest approach for the single-phase assumption is the usage of the governing equations of pure fluid flow with taking the thermophysical properties of the nanofluid. Nield and Kuznetsov [7] made an analytical work of fully-developed laminar forced convection in a parallel-plate channel being concerned by a nanofluid which was subjected to uniform-flux boundary conditions, constant heat flux boundaries and constant temperature boundaries. Their model included the effects of Brownian motion and thermophoresis and they found that the combination of these effects caused decrease the Nusselt number. Selimefendigil and Oztop [8] implemented a numerical examination of laminar pulsating rectangular jet with nanofluids to investigate the effects of pulsating frequency, Reynolds number, and volume fraction on the nanofluid flow which can use heat transfer characteristics by using FLUENT finite volume-based code. From their results, in the pulsating flow case, the combined effect of pulsation and inclusion was not favorable for the rising of the stillness point. Manca et al [9] studied a numerical analysis on forced convection using Al_2O_3 nanoparticles in the water. They are considered the particle size is set equal to 38 nm, nanoparticle volume fractions from 0% to 4% and the flow regime is turbulent and Reynolds numbers are in the range 20,000-60,000.

Their results indicate that particle volume concentration provides to proliferate heat transfer enhancement even though the minimum power to pump the nanofluid must be increased. Also, heat transfer coefficient and pressure loss are investigated by using artificial roughness ducts with grid ribs [10] and semi attached rib–groove channels [11]. Peng et al [12] stated that the 45° V-shaped continuous ribs among different V-shaped ribs have the best thermal achievement. Promvonge et al [13] were carried a numerical analyze heat transfer characteristics in a square-duct with inline 60° V-shaped ribs placed on two opposite heated walls. It revealed the maximum thermal performance was around 1.8 for the rib with BR (rib height to duct diameter ratios) = 0.0725 where the heat transfer rate was about four times above the smooth duct at reduced Reynolds number. Choi [14] was the first who used the term nanofluids to refer to the fluids with suspended nanoparticles. Several researches have demonstrated that with low (1-5% by volume) nanoparticle concentrations, the thermal conductivity can be increased by about 20% [15- 16]. Xuan et al [16] experimentally studied and obtained thermal conductivity of copper-water nanofluid up to 7.5% of solid volume fraction. Several researches have studied heat transfer enhancement with nanofluid [17- 18]. Block and fin fitting in channel can be noted as control elements for rising or reducing of natural or forced convection heat transfer. Most of the investments on changing the flow pattern were performed using partitioning rectangular or square blockades [19-20]. Varol et al [21] studied the effects of fin placement on the bottom wall of a triangular enclosures filled with porous media. Heidary et al [22] have researched free convection and entropy generation in an inclined square cavity filled with a porous medium. Valinataj-Bahnemiri et al [23] investigated of two-dimensional laminar flow of nanofluids in a sinusoidal wavy channel with uniform temperature grooved walls. Tiwari et al [24] studied the heat transfer effect and fluid flow characteristics of nanofluids CeO_2 and Al_2O_3 flowing in a counter flow. The corrugated plate heat exchanger has been simulated, and the three dimensional temperature graphs and velocity fields have been provided through computational fluid dynamics. Darzi et al [25] experimentally studied to investigate the effect of nanoparticles on heat transfer and friction factor inside helically grooved tubes. Nanoparticles and helical grooves simultaneously augmented the heat transfers by factor of 3.2 for higher concentration of water-Al2O3 nanofluids. Navaei et al [26] investigated numerical simulations of turbulent forced convection heat transfer in a rib–groove duct exposed to uniform heat flux. Kareem et al [27] have researched spirally grooved tubes experimentally and numerically, they studied to analysis the effects of new spiral corrugation characteristics and showed the influence of the parameter called severity index on the total thermal performance. Ramadhan et al [28] investigated the fluid flow of turbulent heated flow inside a grooved tube and numerical analysis was used to find the heat transfers. Their study concerned fluid of air and grooved geometry for heat transfer enhancement. To the best of our knowledge a few papers in the literature has so far studied heat transfer in different corrugated wall ducts with nanofluid. Therefore, the present study aims to extend the investigation of the effects of corrugated shape, nanoparticle and Reynolds number on heat transfer and flow behavior. Numerical method is employed for flow in channels having different corrugated shapes such as circular, triangular, and trapezoidal under constant wall temperatures with nanoparticle volume fraction 0.5% and Reynolds number ranging from 10,000 to 20,000.

3. Methodology

In the Current Project Simulation of the Semi circular Corrugated Channel is Done with the Nano Fluids of the different Volumetric concentrations where Different nano Fluids is Used to solve the analysis to determine the best possible solution for this particular application when we define the Reynolds number of 30000 for nano fluids of volume concentrations for 1% 3% & 5% respectively with the nano fluids of Al2o3 Fe3o4 Tio2 and Sio3 to determine the Best possible nano particle and best possible Concentration in this simulation nano fluid concentration is determined and assumed as homogenous Mixture where the solid particle density is Mixed with the water and determined Theoretically

Density if the Nano Fluid is Determined by the Following Equation.

$$\rho_{nf} = (1-\varphi)\rho_{bf} + \varphi\rho_p$$

Specific Heat of the Nano Fluid is determined by the following equation.

$$c_{p,nf} = \frac{(1-\varphi)(\rho c_p)_{bf} + \varphi(\rho c_p)_p}{\rho_{nf}}$$

Viscosity of the Nano fluid is determined by following equation.

$$\mu_{nf} = (123\varphi^2 + 7.3\varphi + 1)\mu_{bf}$$

Thermal Conductivity of the Nano Fluid is Defined By following Equation.

$$k_{nf} = \left[\frac{k_s + 2k_w + 2(k_s - k_w)(1+\beta)^3\varphi}{k_s + 2k_w - (k_s - k_w)(1+\beta)^3\varphi}\right]k_w$$

3.1 INTRODUCTION OF CFD

Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction. CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods. These governing equations, which describe fluid flow, are the set of NavierStokes equation, continuity equation and any additional source terms, for example, porous medium or electric body force.

Since the advent of the digital computer, CFD, as a developing science, has received extensive attention throughout the international community. The attraction of the subject is two fold. Firstly, there is the desire to be able to model physical fluid phenomena that cannot be easily simulated or measured with a physical experiment, for example, weather systems. Secondly, there is desire to be able to investigate physical fluid systems more cost effectively and more rapidly than with experimental procedures.

Traditional restrictions in flow analysis and design limit the accuracy in solving and visualisation of the fluid-flow problems. This applies to both single and multiphase flows, and is particularly true of problems that are three dimensional in nature and involve turbulence, additional source terms, and/or heat and mass transfer. All these can be considered together in the application of CFD, a powerful technique that can help to overcome many restrictions inherent in traditional analysis.

FD is a method for solving complex fluid flow and heat transfer problems on a computer. CFD allows the study of problems that are too difficult to solve using classical techniques. The flow inside the ESP is complex and this can be analyzed using CFD tool, which provides an insight into the complex flow behavior.

3.2 CFD SIMULATIONS

The process of performing CFD simulation is split into three components:

- Pre processing
- Solving
- Post Processing

The preprocessor contains all the fluid flow inputs for a flow problem. It can be seen as a user friendly interface and a conversion of all the input into the solver in CFD program. At this stage, quite a lot of activities are carried out before the problem is being solved. These stages are listed below:

Geometry Definition -

The region of interests that is the computational domain which has to be defined.

Grid generation-

It is the process of dividing the domain into a number of smaller and non- overlapping sub-domains.

Physical and chemical properties-

The flow behavior in terms of physical and chemical characteristics are to be selected.

Fluid property Definition -

The fluid properties like density and viscosity are to be defined.

Boundary conditions-

All the necessary boundary conditions have to be specified on the cell zones. The solution of the flow problem such as temperature, velocity, pressure etc. Is defined at the nodes insides each cell. The accuracy of the CFD solution is governed by the number of cells in the grid and is dependent on the fineness of thegrid.

1) Geometry.
2) Mesh.
3) Setup.
4) Solution.

1) Geometry.

In the Current Simulation a pipe geometry is used simulate the Heat transfer in cfd for nanofluid the pipe is with the Thickness of 4.5mm and the length of 0.9m is drawn using the tools present in the design modeler.

Design Criteria.

The design of the Semi circular corrugated Channel is defined as per the Following Dimensions

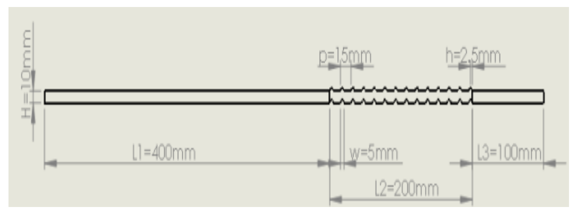


Fig 3.1 Dimensions of the Semi circular Corrugated Channel.

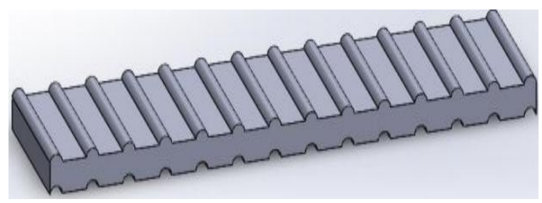


Fig 3.2 3D model of test section for the Simulaion

2) Mesh.

Generally the process of dividing the Total volume in to number of sub domains is called mesh to discretise the domain we use mesh window in the Ansys workbench to Discretize the Total pipe using Quad elements.

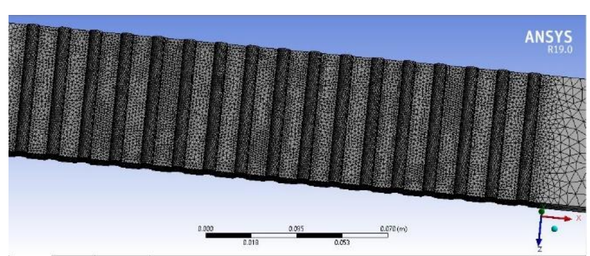


Figure 3.4: Meshed Rectangular channel in meshing module.

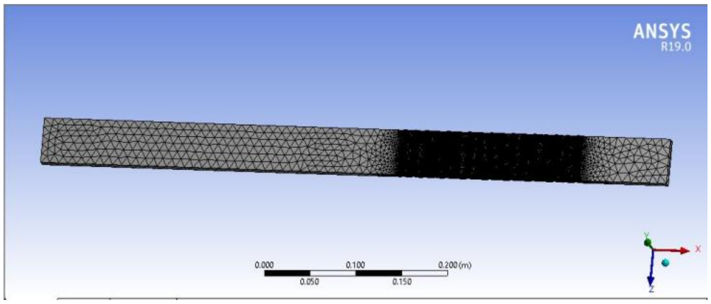


Figure 3.5: Closely sectioned View of meshed pipe.

Solver Preference	Fluent
Export Format	Standard
Export Preview Surface Mesh	No
Element Order	Linear
Sizing	
Quality	
Inflation	
Assembly Meshing	
Advanced	
Statistics	
Nodes	127591
Elements	640955

3) Setup.

Initially in the setup the Pressure based solver is used to define the type of problem along with the gravity Coming to the models The type of viscous flow is laminar to define the type of flow with energy equation on to simulate heat effects in the simulation.

Materials:

As the software database does not contain the Nano fluids by default we need to input the physical properties of the Fluid manually by calculating the Density Thermal Conductivity and Viscosity of the Fluid Manually by consider the liquid water properties and the Solid nanoparticle Pro

4. Results

The current Chapter Deals with the Results obtained from the Simultaion obtained from the Boundary conditions and Setup discussed in Chapter 3 where All the Cases is compared with each other with Respective results.

In this Chapter due to the space Constrain the Discussion of only Higher Volumetric Ratio is Presented.

Case 1 Semi Circular Corrugated Channel with Al203 0.05

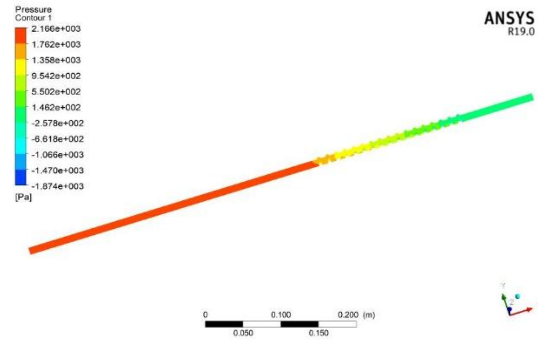


Fig 4.1 Pressure Distribution of Corrugated Channel with 0.05 AL2o3

The above Fig Represents the Pressure distribution of the semi circular Corrugated Channel with Al2o3 0.05 where left side of the picture is called Legend which shows the distribution scale of minimum to maximum in the picture where the blue colored region represents the minimum Pressure 14.62 pa and the Red colored region represents the maximum pressure 2166 pa.

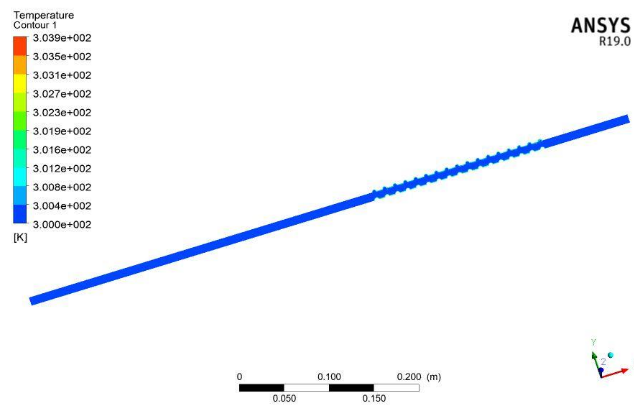


Fig 4.2 Temperature Distribution in semi circular corrugated channel with Al2o3 0.05

The above Fig represents the Temperature distribution of the Semi circular corrugated channel using Al2o3 0.05where left side is the legend with colored ranges in the contour Blue color region represents the minimum temperature and the Red Represents the maximum temperature due to the Heat flux given to the wall with 10Kw /m2 the maximum temperature Occuring at the Wall region the Nusselt number Calculated from the obtained values is 320.478.

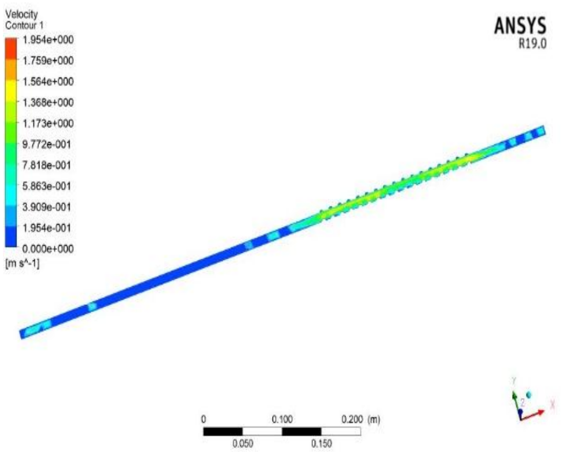
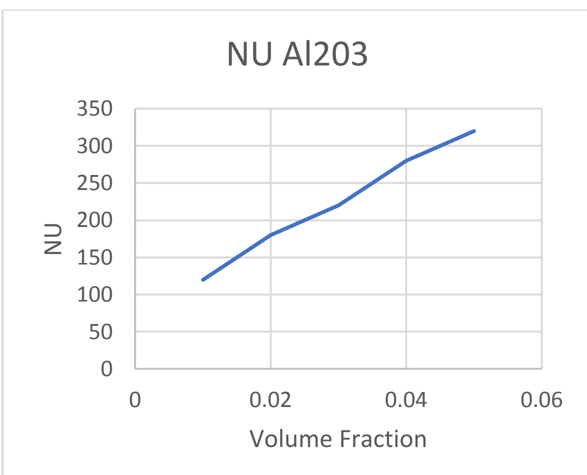


Fig 4.3 Velocity Distribution in Semi Circular Corrugated Channel with Al2o3 0.05.

The Above Figure Represents the Velocity Distribution of the semicircular corrugated channel with Al2o3 0. 05where left side is the legend with colored ranges in the contour Blue color region represents the minimum Velocity at wall region there is no velocity therefore 0 and the Red Represents the maximum Velocity at the Throat region of Corrugated Channel is 1.94m/s



Plot 4.1 Volume Fraction Vs NU of Aluminum oxide

The above Plot is Drawn between Variable Volume Fraction Vs Nusselt Number where in the Trend of the plot we can see the Significant increase in the Nusselt Number with increase in the volume fraction of nano particles this is due to the increase of thermal conductivity within the Fluid. Maximum Nusselt number at 0.05 is 320.478.

Case 2 Semi Circular Corrugated Channel with Fe_3O_4 0.05

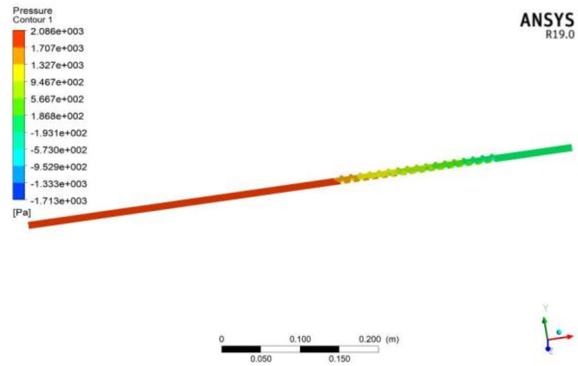


Fig 4.4 Pressure Distribution in Semi Circular Corrugated Channel with Fe_3O_4 0.05.

The above Fig Represents the Pressure distribution of the semi circular Corrugated Channel with Fe3o40.05 where left side of the picture is called Legend which shows the distribution scale of minimum to maximum in the picture where the blue colored region represents the minimum Pressure 18.68 pa and the Red colored region represents the maximum pressure 2086 pa.

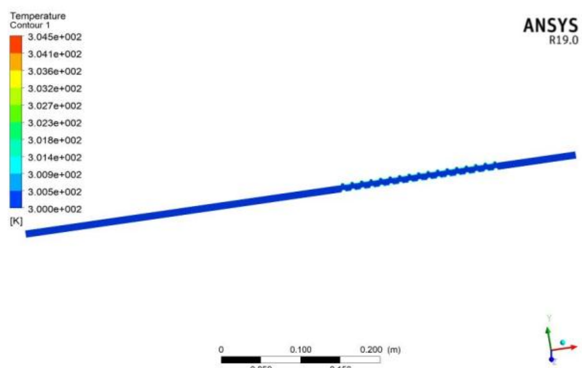


Fig 4.5 Temperature Distribution in Semi Circular Corrugated Channel with Fe_3O_4 0.05.

The above Fig represents the Temperature distribution of the Semi circular corrugated channel using Fe3o4 0.05where left side is the legend with colored ranges in the contour Blue color region represents the minimum temperature and the Red Represents the maximum temperature due to the Heat flux given to the wall with 10Kw /m2 the maximum temperature Occurring at the Wall region the Nusselt number Calculated from the obtained values is 332.25478.

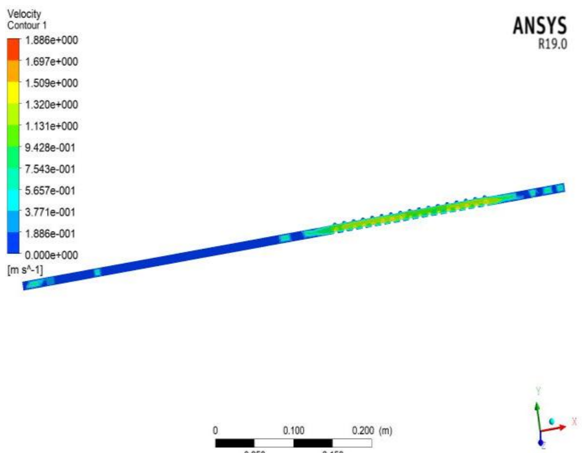
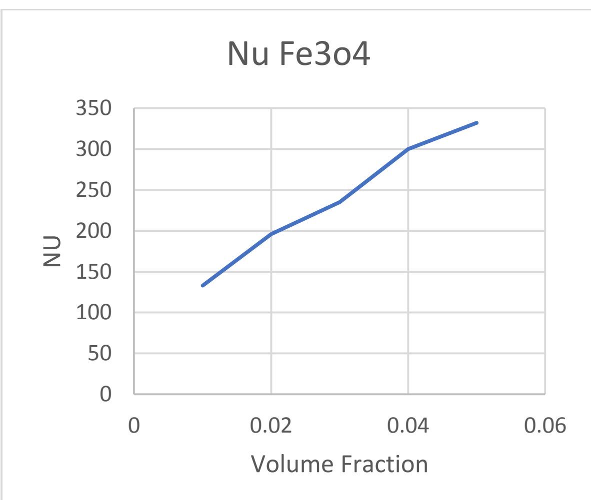


Fig 4.6 Velocity Distribution in Semi Circular Corrugated Channel with Fe_3O_4 0.05.

The Above Figure Represents the Velocity Distribution of the semicircular corrugated channel with Fe3o4 0. 05where left side is the legend with colored ranges in the contour Blue color region represents the minimum Velocity at wall region there is no velocity therefore 0 and the Red Represents the maximum Velocity at the Throat region of Corrugated Channel is 1.88m/s



Plot 4.2 Volume Fraction Vs NU of Ferric oxide

The above Plot is Drawn between Variable Volume Fraction Vs Nusselt Number where in the Trend of the plot we can see the Significant increase in the Nusselt Number with increase in the volume fraction of nano particles this is due to the increase of thermal conductivity within the Fluid. Maximum Nusselt number at 0.05 is 332.25478.

Case 3 Semi Circular Corrugated Channel with SiO_2 0.05

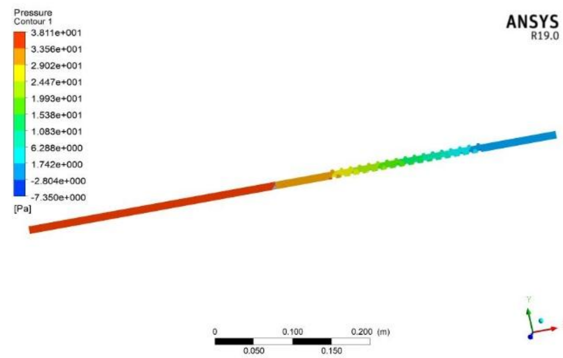


Fig 4.7 Pressure Distribution in Semi Circular Corrugated Channel with SiO_2 0.05.

The above Fig Represents the Pressure distribution of the semi circular Corrugated Channel with Sio2 0.05 where left side of the picture is called Legend which shows the distribution scale of minimum to maximum in the picture where the blue colored region represents the minimum Pressure 1.72 pa and the Red colored region represents the maximum pressure 38 pa.

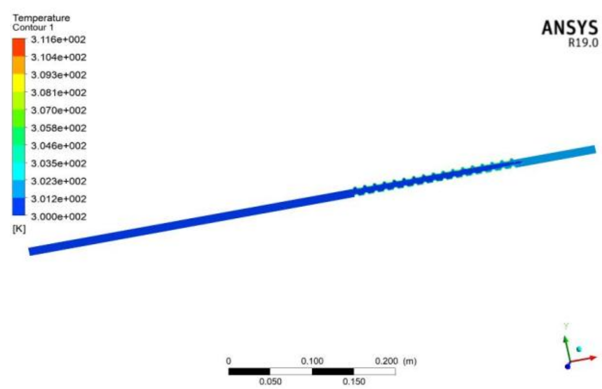


Fig 4.8 Temperature Distribution in Semi Circular Corrugated Channel with SiO_2 0.05.

The above Fig represents the Temperature distribution of the Semi circular corrugated channel using Sio2 0.05where left side is the legend with colored ranges in the contour Blue color region represents the minimum temperature and the Red Represents the maximum temperature due to the Heat flux given to the wall with 10Kw /m2 the maximum temperature Occuring at the Wall region the Nusselt number Calculated from the obtained values is 326.445.

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 09 | Sep2019 www.irjet.net p-ISSN: 2395-0072

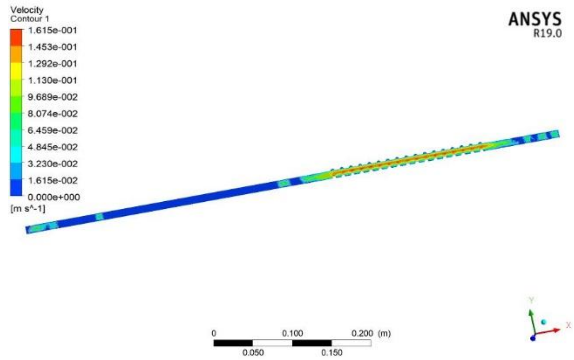
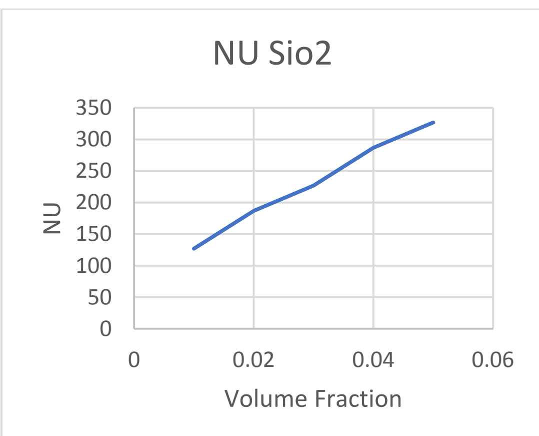


Fig 4.9 Velocity Distribution in Semi Circular Corrugated Channel with SiO_2 0.05.

The Above Figure Represents the Velocity Distribution of the semicircular corrugated channel with Sio2 0. 05where left side is the legend with colored ranges in the contour Blue color region represents the minimum Velocity at wall region there is no velocity therefore 0 and the Red Represents the maximum Velocity at the Throat region of Corrugated Channel is 1.615m/s



Plot 4.3 Volume Fraction Vs NU of Silicon dioxide

The above Plot is Drawn between Variable Volume Fraction Vs Nusselt Number where in the Trend of the plot we can see the Significant increase in the Nusselt Number with increase in the volume fraction of nano particles this is due to the increase of thermal conductivity within the Fluid. Maximum Nusselt number at 0.05 is 326.445.

Case 4 Semi Circular Corrugated Channel with TiO_2 0.05

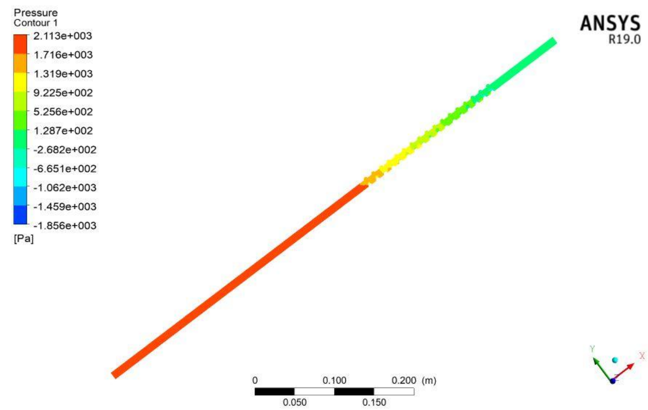


Fig 4.10 Pressure Distribution in Semi Circular Corrugated Channel with TiO_2 0.05.

The above Fig Represents the Pressure distribution of the semi circular Corrugated Channel with Tio20.05 were left side of the picture is called Legend which shows the distribution scale of minimum to maximum in the picture where the blue colored region represents the minimum Pressure 12.87 pa and the Red colored region represents the maximum pressure 2113 pa.

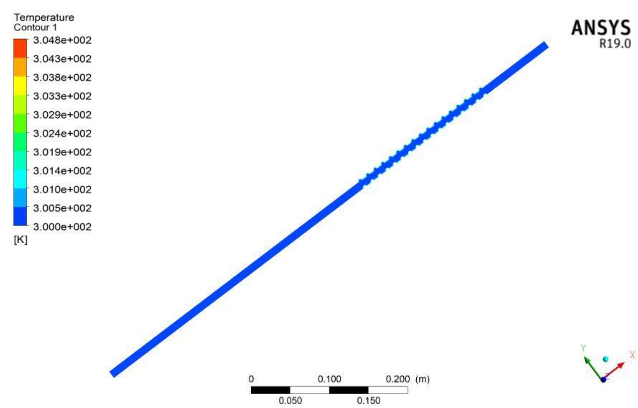


Fig 4.11 Temperature Distribution in Semi Circular Corrugated Channel with TiO_2 0.05.

The above Fig represents the Temperature distribution of the Semi circular corrugated channel using Tio2 0.05where left side is the legend with colored ranges in the contour Blue color region represents the minimum temperature and the Red Represents the maximum temperature due to the Heat flux given to the wall with 10Kw /m2 the maximum temperature Occuring at the Wall region the Nusselt number Calculated from the obtained values is 328.447.

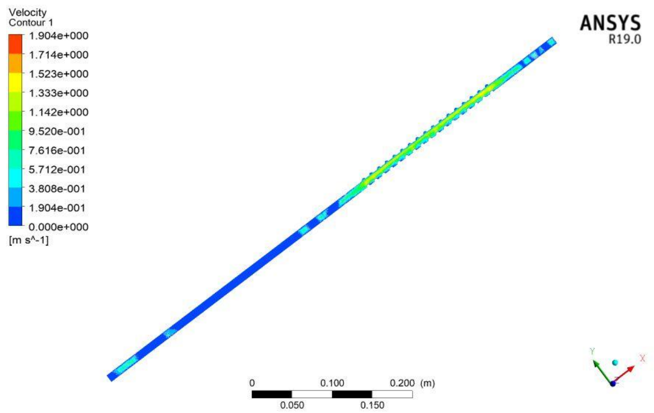
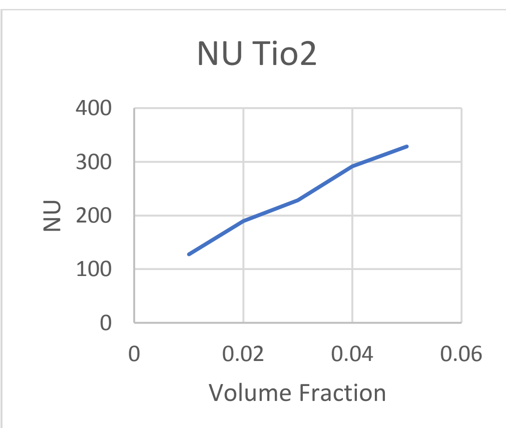


Fig 4.12 Velocity Distribution in Semi Circular Corrugated Channel with TiO_2 0.05.

The Above Figure Represents the Velocity Distribution of the semicircular corrugated channel with Tio2 0. 05where left side is the legend with colored ranges in the contour Blue color region represents the minimum Velocity at wall region there is no velocity therefore 0 and the Red Represents the maximum Velocity at the Throat region of Corrugated Channel is 1.90m/s



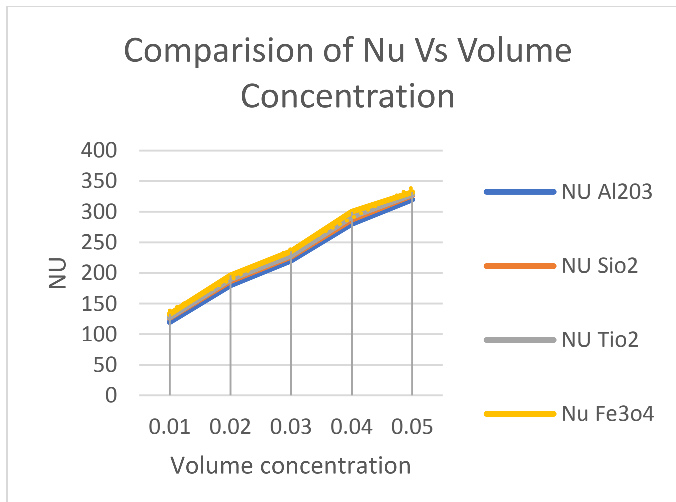
Plot 4.4 Volume Fraction Vs NU of Titanium dioxide

The above Plot is Drawn between Variable Volume Fraction Vs Nusselt Number where in the Trend of the plot we can see the Significant increase in the Nusselt Number with increase in the volume fraction of nano particles this is due to the increase of thermal conductivity within the Fluid. Maximum Nusselt number at 0.05 is 328.447

5. Conclusions

Numerical simulations of turbulent forced convection heat transfer in a semi-circular corrugated channel subjected to uniform heat flux were carried out. The computations were performed for a symmetrical semi-circular corrugated channel with varying Volume Concentrations (1% ≤ Ø ≤5%), For Different Nano fluids of Tio2 Sio2 Fe3o4 Al2o3 The results of numerical solution showed that Nu increase with increasing the Ø . The results of the present study are consistent with the results presented by [7], [8], [9], [10], [11] and [12]. Finally, higher Nusselt number enhancement ratio which indicates the optimum configuration is Fe3o4 and volume fraction 5%. Based on the above results, the use of nanofluids in semi-circular corrugated channel is a suitable method to achieve a good enhancement in the performance of many thermal devices as a passive method.

The Below Graph Represents the Behaviour of Nu with various Nano fluid Concentrations from the graph we can say that For Fe3o4 5% we observe High Nusselt Number



REFERENCES

[1] Khanafer, K., Vafai, K., & Lightstone, M. (2003). Buoyancy-driven heat transfer enhancement in a twodimensional enclosure utilizing nanofluids. International Journal of Heat and Mass Transfer, 46(19), 3639-3653.

[2] Xuan, Y., Li, Q., & Hu, W. (2003). Aggregation structure and thermal conductivity of nanofluids. AIChE Journal, 49(4), 1038-1043.

[3] Xuan, Y., & Li, Q. (2003). Investigation on convective heat transfer and flow features of nanofluids. Journal of Heat transfer, 125(1), 151-155.

[4] Yang, Y., Zhang, Z. G., Grulke, E. A., Anderson, W. B., & Wu, G. (2005). Heat transfer properties of nanoparticle-in-fluid dispersions (nanofluids) in laminar flow. International Journal of Heat and Mass Transfer, 48(6), 1107-1116.

[5] Santra, A. K., Sen, S., & Chakraborty, N. (2009). Study of heat transfer due to laminar flow of copper–water nanofluid through two isothermally heated parallel plates. International Journal of Thermal Sciences, 48(2), 391- 400.

[6] Kakaç, S., & Pramuanjaroenkij, A. (2009). Review of convective heat transfer enhancement with nanofluids. International Journal of Heat and Mass Transfer, 52(13-14), 3187-3196.

[7] Nield, D., & Kuznetsov, A. (2014). Forced convection in a parallel-plate channel occupied by a nanofluid or a porous medium saturated by a nanofluid. International Journal of Heat and Mass Transfer, 70, 430-433.

[8] Selimefendigil, F., & Öztop, H. F. (2014). Pulsating nanofluids jet impingement cooling of a heated horizontal surface. International Journal of Heat and Mass Transfer, 69, 54-65.

[9] Manca, O., Nardini, S., & Ricci, D. (2012). A numerical study of nanofluid forced convection in ribbed channels. Applied Thermal Engineering, 37, 280-292.

[10] Karmare, S., & Tikekar, A. (2007). Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs. International Journal of Heat and Mass Transfer, 50(21-22), 4342-4351.

[11] Liu, H., & Wang, J. (2011). Numerical investigation on synthetical performances of fluid flow and heat transfer of semiattached rib-channels. International Journal of Heat and Mass Transfer, 54(1-3), 575-583.

[12] Peng, W., Jiang, P.-X., Wang, Y.-P., & Wei, B.-Y. (2011). Experimental and numerical investigation of convection heat transfer in channels with different types of ribs. Applied Thermal Engineering, 31(14-15), 2702- 2708.

[13] Promvonge, P., Changcharoen, W., Kwankaomeng, S., & Thianpong, C. (2011). Numerical heat transfer study of turbulent square-duct flow through inline V-shaped discrete ribs. International Communications in Heat and Mass Transfer, 38(10), 1392-1399.

[14] Choi, S. U., & Eastman, J. A. (1995). Enhancing thermal conductivity of fluids with nanoparticles. (No. ANL/MSD/CP--84938; CONF-951135--29). Argonne National Lab., IL (United States).

[15] Lee, S., Choi, S. S., Li, S. A., and, & Eastman, J. A. (1999). Measuring thermal conductivity of fluids containing oxide nanoparticles. Journal of Heat transfer, 121(2), 280-289. Journal of Thermal Engineering, Research Article, Vol. 4, No. 3, pp. 1984-1997, April, 2018 1996

[16] Xuan, Y., & Li, Q. (2000). Heat transfer enhancement of nanofluids. International Journal of heat and fluid flow, 21(1), 58-64.

[17] Das, S. K., Choi, S. U., Yu, W., & Pradeep, T. (2007). Nanofluids: science and technology: John Wiley & Sons.

[18] Ding, Y., Chen, H., Wang, L., Yang, C. Y., He, Y., Yang, W., ... & Huo, R. (2007). Heat transfer intensification using nanofluids. KONA Powder and Particle Journal, 25, 23-38.

[19] Bilgen, E. (2005). Natural convection in cavities with a thin fin on the hot wall. International Journal of Heat and Mass Transfer, 48(17), 3493-3505.

[20] Hasnaoui, M., Bilgen, E., & Vasseur, P. (1991). Natural convection above an array of open cavities heated from below. Numerical Heat Transfer, 18(4), 463-482.

[21] Varol, Y., Oztop, H. F., & Varol, A. (2007). Natural convection in porous triangular enclosures with a solid adiabatic fin attached to the horizontal wall. International Communications in Heat and Mass Transfer, 34(1), 19- 27.

[22] Heidary, H., Pirmohammadi, M., & Davoudi, M. (2012). Control of free convection and entropy generation in inclined porous media. Heat Transfer Engineering, 33(6), 565-573.

[23] Valinataj-Bahnemiri, P., Ramiar, A., Manavi, S., & Mozaffari, A. (2015). 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, 18(4), 727-737.

[24] Tiwari, A. K., Ghosh, P., Sarkar, J., Dahiya, H., & Parekh, J. (2014). Numerical investigation of heat transfer and fluid flow in plate heat exchanger using nanofluids. International Journal of Thermal Sciences, 85, 93-103.

[25] Darzi, A. A. R., Farhadi, M., & Sedighi, K. (2014). Experimental investigation of convective heat transfer and friction factor of Al2O3/water nanofluid in helically corrugated tube. Experimental Thermal and Fluid Science, 57, 188-199.

[26] Navaei, A., Mohammed, H., Munisamy, K., Yarmand, H., & Gharehkhani, S. (2015). Heat transfer enhancement of turbulent nanofluid flow over various types of internally corrugated channels. Powder Technology, 286, 332-341.

[27] Kareem, Z. S., Abdullah, S., Lazim, T. M., Jaafar, M. M., & Wahid, A. F. A. (2015). Heat transfer enhancement in three-start spirally corrugated tube: Experimental and numerical study. Chemical Engineering Science, 134, 746-757.

[28] Ramadhan, A. A., Al Anii, Y. T., & Shareef, A. J. (2013). Groove geometry effects on turbulent heat transfer and fluid flow. Heat and Mass Transfer, 49(2), 185-195

[29] Sharma, K., Sundar, L. S., & Sarma, P. (2009). Estimation of heat transfer coefficient and friction factor in the transition flow with low volume concentration of Al2O3 nanofluid flowing in a circular tube and with twisted tape insert. International Communications in Heat and Mass Transfer, 36(5), 503-507.

[30] Shahi, M., Mahmoudi, A. H., & Talebi, F. (2011). A numerical investigation of conjugated-natural convection heat transfer enhancement of a nanofluid in an annular tube driven by inner heat generating solid cylinder. International Communications in Heat and Mass Transfer, 38(4), 533-542.

[31] Patankar, S. (1980). Numerical heat transfer and fluid flow. CRC press.

[32] Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. AIAA journal, 26(11), 1299-1310.

[33] Vanaki, S. M., Mohammed, H., Abdollahi, A., & Wahid, M. (2014). Effect of nanoparticle shapes on the heat transfer enhancement in a wavy channel with different phase shifts. Journal of Molecular Liquids, 196, 32- 42.

[34] Weihing, P., Younis, B., & Weigand, B. (2014). Heat transfer enhancement in a ribbed channel: Development of turbulence closures. International Journal of Heat and Mass Transfer, 76, 509-522.

[35] Ağra, Ö., Demir, H., Atayılmaz, Ş. Ö., Kantaş, F., & Dalkılıç, A. S. (2011). Numerical investigation of heat transfer and pressure drop in enhanced tubes. International Communications in Heat and Mass Transfer, 38(10), 1384-1391.

[36] Sahin, B., Gültekin, G. G., Manay, E., & Karagoz, S. (2013). Experimental investigation of heat transfer and pressure drop characteristics of Al2O3–water nanofluid. Experimental Thermal and Fluid Science, 50, 21-28.

[37] Petukhov, B. (1970). Heat transfer and friction in turbulent pipe flow with variable physical properties Advances in heat transfer (Vol. 6, pp. 503-564): Elsevier. Journal of Thermal Engineering, Research Article, Vol. 4, No. 3, pp. 1984-1997, April, 2018 1997 [38] Mohamad, A. (2015). Myth about nano-fluid heat transfer enhancement. International Journal of Heat and Mass Transfer, 86, 397-403.

Name: VISHAWANATH

Branch:M.TECH IN THERMAL POWER ENGINEERING

Guide name:"Dr PRAVIN V HONGUNTIKAR"prof and Head of the Mechanical Department Poojya Doddappa Appa College of Engineering Kalaburagi-585102