Heat Transfer Analysis of Plain and Corrugated Channel Using Nano Fluids

Nirmala Devi E[1] | Naresh A[2] | Sudharshan Reddy G[3] | Shyam prakesh B[4] | Gnana Venkata Rajesh V[5]

[1] Associate Professor, Department of Mechanical Engineering, Godavari Institute of Engineering and Technology (A), Rajahmundry.
[2,3,4,5] UG Student, Department of Mechanical Engineering, Godavari Institute of Engineering and Technology (A), Rajahmundry.

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 Al2O3- 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 - Computational Fluid Dynamics, corrugated channels, nanofluid, nanoparticle, Reynolds number, Nusselt number.

Check for updates

DOI of the Article: https://doi.org/10.46501/GIETME30

Available online at: http://ijmtst.com/vol7si05.html

As per UGC guidelines an electronic bar code is provided to seure your paper

To Cite this Article: Nirmala Devi E; Naresh A; Sudharshan Reddy G; Shyam prakesh B and Gnana Venkata Rajesh V. Heat Transfer Analysis of Plain and Corrugated Channel Using Nano Fluids. *International Journal for Modern Trends in Science and Technology* 2021, *7*, pp. 180-197. https://doi.org/10.46501/GIETME30

Article Info. Received: 12 June 2021; Accepted: 16 July 2021; Published: 20 July 2021

I.INTRODUCTION

The enhancement of convection heat transfer is a very interesting topic for different kinds of industrial and engineering applications and can be improved passively by changing the flow geometry and boundary conditions or by enhancing the thermophysical properties of the fluid. The ability of a fluid medium to transfer a large amount of heat across a small temperature gradient enhances the efficiency of energy conversion, as well as improves the design and performance of heat exchangers. Therefore, research on enhancement technique in such channels has become very prominent. For this purpose, using nanofluids as a cooling fluids in corrugated channels instead of traditional fluids can enhance thermal conductivity of the base fluids and thereby a further improvement in thermal performance of heat exchangers with a more compact design. Various numerical and experimental studies on the forced convection flow of conventional fluids or nanofluids in channels exist in the literature. Sunden and Skoldheden experimentally studied the heat transfer and pressure drop in corrugated channels and smooth tubes. Sawyersnumerically and experimentally studied the effect of three dimensional hydrodynamics on the enhancement of heat transfer in corrugated channels for Reynolds numbers in the range of 9–149. Fabbri studied laminar convective heat transfer in a channel composed of smooth and corrugated walls. Vasudevaiah and Balamurugan studied theoretically the convective heat transfer in a corrugated microchannel. The transport equations of continuum theory were adopted under incompressible flow conditions. Wang and Chen applied a simple coordinate transformation method and the spline alternating direction implicit method for determining the heat transfer rates for flow through a sinusoidally curved converging–diverging channel. Gradeck et al. experimentally studied the effects of the hydrodynamic conditions on the enhancement of heat transfer for single phase flow in corrugated channels. Naphon conducted numerical and experimental study of forced convection heat transfer and flow developments in a channel with V-corrugated upper and lower plates in which all configuration peaks lie in an in-phase arrangement. The results show that wavy angle and channel height have significant effect on the temperature distribution and flow development. It was found that the sharp edge of the wavy plate (V-shaped) has a significant effect on the enhancement of heat transfer.The turbulent flow of nanofluids with different volume fractions of nanoparticles of the turbulent forced convection of nanofluid flow in triangular-corrugated channel over Reynolds number ranges of 1000- 5000. It is found that the average Nusselt number, pressure drop, heat transfer enhancement, thermal– hydraulic performance increase with increasing in the volume fraction of nanoparticles and with decreasing in the diameter of nanoparticles. In this paper, forced convection of nanofluids in semi-circular corrugated channels is numerically studied using finite volumeflowing through a two-dimensional duct under constant heat flux condition was simulated by Rostamani et al. [11]. Ahmed et al. investigated numerically method. The effects of Reynolds number, nanoparticles volume fraction ( ) and nanoparticle diameter (dp) on heat transfer and friction factor are presented and analysed. The aim of this research is to determine the optimum corrugation profile for heat transfer augmentation and to serve as reference for experimental work in the future.

STRUCTURE OF PAPER

The paper is organized as follows: In Section 1, the introduction of the paper is provided along with the structure, important terms, objectives and overall description. In Section 2 we discuss about related work. In Section 3 we discuss about Computational Fluid Dynamics. Section 4 tells us about the methodology and the process description. In Section 5we discuss about the simulation. In Section 6 we discuss about the results obtained, Lastly Section 7 concludes the paper with references.

OBJECTIVE

Heat transfer enhancement by modifying the surface of tubes is commonly practiced throughout the world. Grooves, dimples, flutes or corrugations are placed inside and outside the surface of tubes and channels for enhancement. In this article, a novel method for heat transfer enhancement by varying the spacing between the tubes is reported.

RELATED WORK

F. Selimefendigil, H. Oztop **(2017)** Numerical study of jet impingement cooling of a corrugated surface with water - SiO$_2$ nanofluid of different nanoparticle shapes was performed. The bottom wall is corrugated and kept at constant surface temperature while the jet is emerged from a rectangular slot with cold uniform temperature. The finite volume method is utilized to solve the governing equations. The effects of Reynolds number (between 100 and 500), corrugation amplitude (between 0 and 0.3), corrugation frequency (between 0 and 20), nanoparticle volume fraction (between 0 and 0.04) and nanoparticle shapes (spherical, blade, brick, cylindrical) on the fluid flow and heat transfer characteristics were studied. Stagnation point and average Nusselt number enhance with Reynolds number and solid particle volume fraction for both flat and corrugated surface configurations. An optimal value for the corrugation amplitude and frequency was found to maximize the average heat transfer at the highest value of Reynolds number. Among various nanoparticle shapes, cylindrical ones perform the best heat transfer characteristics in terms of stagnation and average Nusselt number values. At the highest solid volume concentration of the nanoparticles, heat transfer values are higher for a corrugated surface when compared to a flat surface case.

M. Ahmed, M. Yusoff, N. H. Shuaib **(2013)** Abstract In this article, laminar forced convection heat transfer of copper–water nanofluid in trapezoidal-corrugated channel has been numerically investigated. The two-dimensional governing continuity, momentum and energy equations in body-fitted coordinates are discretized using finite volume approach and solved iteratively using SIMPLE technique. In this study, the Reynolds number and nanoparticle volume fractions are in the ranges of 100–700 and 0–5%, respectively. The effect of geometrical parameters such as the amplitude and wavelength of the corrugated channel, nanoparticle volume fraction and Reynolds number on the velocity vectors, temperature contours, pressure drop and average Nusselt number have been presented and analyzed. The results show that the average Nusselt number enhances with increase in nanoparticles volume fraction and with the amplitude of corrugated channel but this enhancement accompanied by increases in pressure drop. In addition, as the wavelength of corrugated channel decreases, the average Nusselt number increases and the pressure drop decreases.

F. Selimefendigil, H. Oztop **(2014)** Abstract In this study, a numerical study of pulsating rectangular jet with nanofluids is presented. The aim of this work is to numerically investigate the effects of various parameters such as pulsating frequency, Reynolds number, nanoparticle volume fraction on the fluid flow and heat transfer characteristics. The unsteady Navier–Stokes and energy equations are solved with a commercial finite volume based code. It is observed in the steady case, adding nanoparticles increases the peak value of the Nusselt number at the stagnation point and spatial-averaged Nusselt number along the impingement plate. Heat transfer enhancement up to 18.8% is obtained for particle volume fraction of 6% at Reynolds number of 200. In the pulsating flow case, the combined effect of pulsation and inclusion of nanoparticles is not favorable for the augmentation of the stagnation point Nusselt number at Re = 200, φ = 1%, 3% and at Re = 400, φ = 1%, 3% when compared to the steady case.

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.Thesegoverning 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 developingscience, has receivedextensive attention throughout the international community. The attraction ofthe 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.

CFD 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.

METHODOLOGY

In the Current Project Simulation of the Smooth 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 CUO 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.

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

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 ofinterests 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.

SOLUTION

In the numerical solution technique, there are three different streams that form the basis of the solver. They are finite difference, finite element and finite volume methods.

The differences between them are the way in which the flow variables are approximated and the discretisation processes are done.

Finite Difference Element (FDM)

FDM describes the unknown flow variables of the flow problem by means of point samples at node point's of a grid coordinate. By FDM, the Taylor's expansion is usually used to generate finite differences approximation.

Finite Element Method (FEM)

FEM uses the simple piecewise functions valid on elements to describe the local variations of unknown flow variables .Governing equation is precisely satisfied by the exact solution of flow variables. In FEM, residuals are used to measure the errors.

Finite Volume Method (FVM)

FVM was originally developed as a special finite difference formulation. The commercial CFD code packages using the FVM approaches are PHOENICS, FLUENT, FLOW 3D an d STARCD. Basically, the numerical algorithm in these CFD commercial packages involve s the formal integration of the governing equation over all the finite control volume, the discretisation process involves the substitution of a variety of FDM types to approximate t he integration equation of the flow problem, and the solution is obtained by iterative method. Discretisation in the solver involves the approaches to solve the numerical integration of the flow problem. Usually, two different approaches are made, one at a time.

Explicit approach

Usually, this is the most useful approach that makes sense. It is relatively simple to setup and program. The limitation is that for a given t and x, the time must be less than some limitimposed by stability constraints. In some cases, t must be very small to maintain the stability, and consequently long running time is required for the calculation over a given time interval t.

Implicit approach

For this approach, the stability can be maintained over a large value of t and fewer time steps are required for making calculation resulting in less computer time. But it is complicated to set up and program. The computer time per time step is much larger than the explicit approach due to the matrix manipulation, which is required for each time step. This approach is very accurate to follow the exact transients, i.e., the time variations of the independent variables.

Post-Processing

The CFD package provides the data visualization tools tovisualize the results ofthe flow problem. This includes vectors plots, domain geometry and grid display, line and shaded counter plots, particle tracking etc. Recent facilities are aided with animation for dynamic result display and they also have data export facilities for further manipulation external to the code.

Determining the convergence, whether the solution is consistent and stable for all range of flow variables, is important.

Convergence is a property of a numerical method to produce a solution that approaches the exact solution by which the grid spacing and control volume size are reduced to a specific value or to zero value.

Consistency is to produce the system of algebraic equations that can beequivalent to the original governing equation.

Stability associates with the damping of errors as a numerical method proceeds. If a technique chosen is not stable, even the round off error in the initial data can lead towild oscillations or divergence.

GOVERNINGEQUATIONS

CONSERVATION LAW

Navier Stokes equations are the governing equations of Computational Fluid Dynamics. It is based on the conservation law of physical properties of fluid. The Principle conservational law is the change of properties, for example mass, energy, and momentum, in an object is decided by the input and output.

For example, the change of mass in the object is as follows

$\frac{dM}{dt} = \dot{m}_{in} - \dot{m}$ (1)

If $\dot{m}_{in}$-$\dot{m}$=0, we have (2)

$\frac{dM}{dt} = 0$ (3)

Which means M⊚ ©CONST?

Navier - Stokes Equations:

Applying the mass, momentum and energy conservation, we can derive the continuity equation, momentum equation and energy equation as follows.

(i) Continuity Equation

$$\frac{D\rho}{Dt} + \rho = 0 \quad (4)$$

(ii) Momentum equation

$$\underbrace{\rho\frac{\partial U_j}{\partial t}}_{I} + \underbrace{\rho U_i\frac{\partial U_j}{\partial x_i}}_{II} = -\underbrace{\frac{\partial P}{\partial x_j}}_{III} - \underbrace{\frac{\partial \tau_{ij}}{\partial x_i}}_{IV} + \underbrace{\rho g_j}_{V} \quad (5)$$

Where,

$$\tau_{ij} = -\mu\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3}\delta_{ij}\mu\frac{\partial U_k}{\partial x_k} \quad (6)$$

I: Local change with time

II: Momentum convection

III: Surface force

IV: Molecular-dependent momentum exchange (diffusion)

V: Mass force

(iii) *Equation*

$$\underbrace{\rho c_\mu\frac{\partial T}{\partial t}}_{I} + \underbrace{\rho c_\mu U_i\frac{\partial T}{\partial x_i}}_{II} = \underbrace{-P\frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\lambda\frac{\partial^2 T}{\partial x_i^2}}_{IV} - \underbrace{\tau_{ij}\frac{\partial U_j}{\partial x_i}}_{V} \quad (7)$$

I: Local energy change with time

II: Convective term

III: Pressure work

IV: Heat flux (diffusion)

V: Irreversible transfer of mechanical energy into heat

If the fluid is incompressible: we can simplify the continuity equation and momentum equation as follows.

Continuity Equation

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (8)$$

Momentum Equation

$$\rho\frac{\partial U_j}{\partial t} + \rho U_i\frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} - \mu\frac{\partial^2 U_j}{\partial x_i^2} + \rho g_j \quad (9)$$

General Form of Navier-StokesEquation

To simplify the Navier -Stokes equations, we can rewrite them as the general form.

$$\frac{\partial(\rho\Phi)}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho U_i\Phi - \Gamma_\Phi\frac{\partial\Phi}{\partial x_i}\right) = q_\Phi \quad (10)$$

When ⊚ ⊚ *Uj, T*

We can respectively get continuity equation, momentum equation and energy equationThe procedure or methodology obtained to achieve the results is as follows.

SIMULATION

Generally in the procedure to complete the simulation Specific simulation techniques are obtained

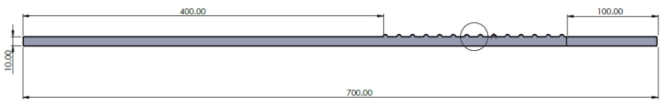
1) Geometry.
2) Mesh.
3) Setup.
4) Solution.

I. GEOMETRY.

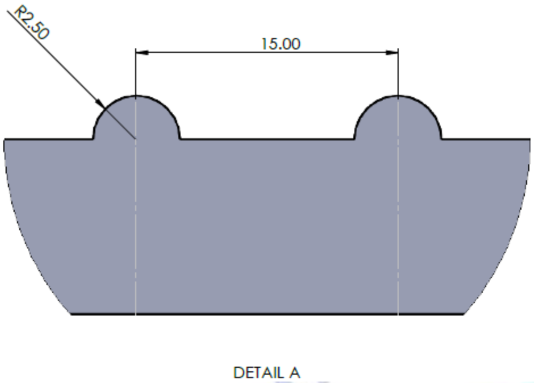
In the Current Simulation a pipe geometry is used simulate the Heat transfer in CFD for nanofluid the pipe is with the Thickness of 4.5mm and the length of 0.9m is drawn using the tools present in the design modeler.

Design Criteria.

The design of the Smooth Corrugated Channel is defined as per the Following Dimensions.



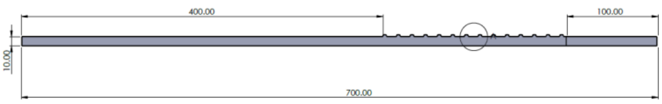
(a)



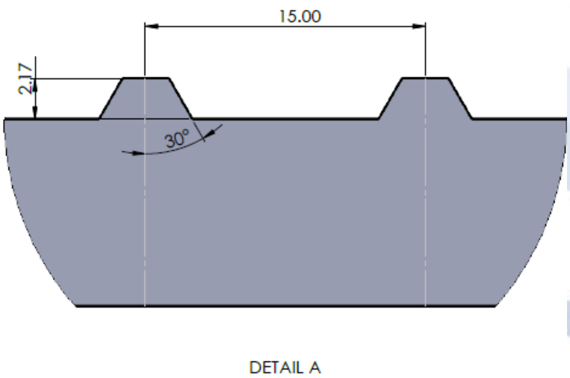
(b)

Fig 5.1.*Dimensions of the Smooth Corrugated Channel.*

The design of the Rough Corrugated Channel is defined as per the Following Dimensions.



(a)



(b)

Fig 5.2.*Dimensions of the Smooth Corrugated Channel.*

II. 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.

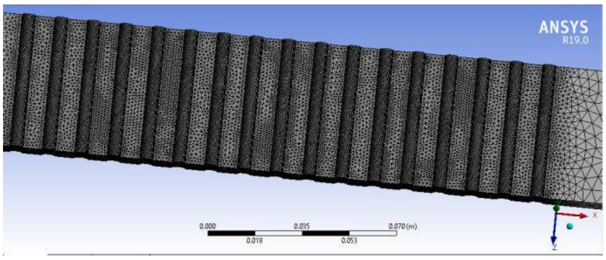


Fig5.3.*Meshed Rectangular channel in meshing module.*

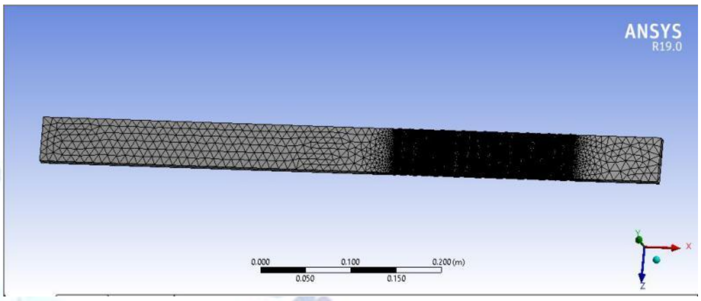


Fig 5.4. *Closely sectioned View of meshed pipe.*

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

Fig 5.5. *Statistics.*

III. 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.

Density(bf)	Specific Heat (bf)	Thermal Conductivity (bf)	Viscosity (bf)
998.2	4182	0.597	9.98E-04

Nano Particle	Density (P)	Specific Heat (P)	Thermal Conductivity (P)
Sio2	2400	853.3496	35

Sample Calculation

1. Density of nanofluid

Pnf= (1-0.01)*998.2+0.01*2400=1012.218kg/m3

2. Specific

HeatCp=[(1-0.01)*(998.2*4182)+0.01(2400*837.5)]/1012.218=4102.698

3. Viscosity

M={(123*0.01*0.01)+7.3*0.01+1}*9.98e-4=1.08e-3

4. Thermal Conductivity

K=37.10981/35.27819=0.62799

The Above Calculated data is inputted into the Physical Properties of fluid for 1% 3% 5% of all the Nano Fluids in the Simulation.

Boundary Conditions

Inlet Velocity Re=30000

UHF(uniform Heat Flux (Q W/m^{2}K)) =10KW/m^{2}K

RESULT AND DISCUSSION

The current Chapter Deals with the Results obtained from the Simulation 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 Smooth Corrugated Channel With CUO 0.05 for Re=10000

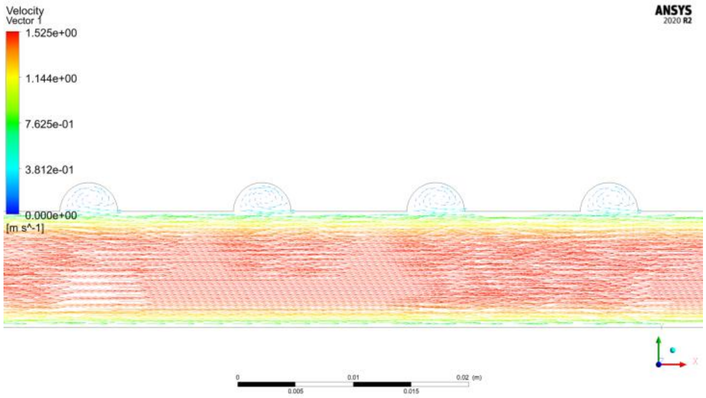
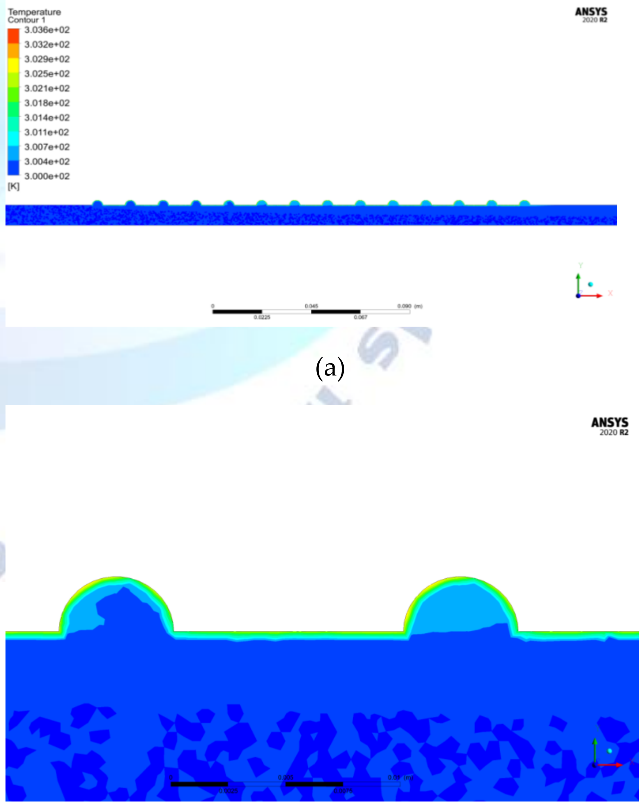


Fig 6.1.*Flow Distribution of Corrugated Channel With 0.05 CUO for Re=10000*

The above Fig Represents the Flow Distribution of the Smooth Corrugated Channel with CUO 0.05 for Re=10000 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 velocity is 0.3812 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 1.525 at the Middle of the Pipe.



(b)

Fig 6.2.*Temperature Distribution in Smooth Corrugated channel with CUO 0.05 for Re=10000*

The above Fig represents the Temperature distribution for Reynolds number of 10000 of the Smooth Corrugated channel using CUO 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 231.360.

Case 2 Smooth Corrugated Channel With CUO 0.05 for Re=15000

Case 2 Smooth Corrugated Channel With CUO 0.05 for Re=15000

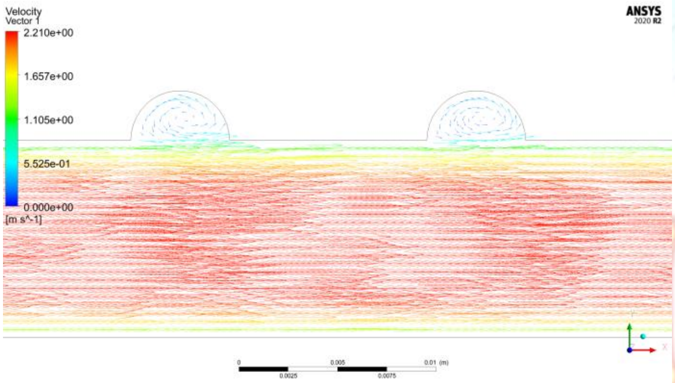
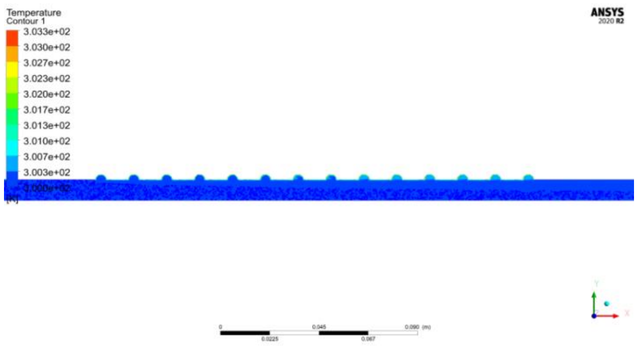
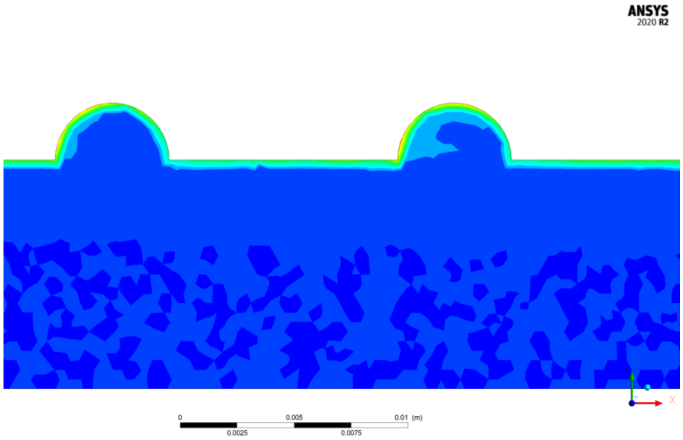


Fig 6.3. *Flow Distribution of Corrugated Channel With 0.05 CUO for Re=15000*

The above Fig Represents the Flow Distribution of the Smooth Corrugated Channel with CUO 0.05 for Re=15000 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 velocity is 0.525 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity at the Middle of the Pipe.



(a)



(b)

Fig 6.4. *Temperature Distribution in Smooth Corrugated channel with CUO 0.05 for Re=15000*

The above Fig represents the Temperature distribution for Reynolds number of 15000 of the Smooth Corrugated channel using CUO 0.05where left side is the le-[=gend 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 284.3443.

Case 3 Smooth Corrugated Channel With CUO 0.05 for Re=20000

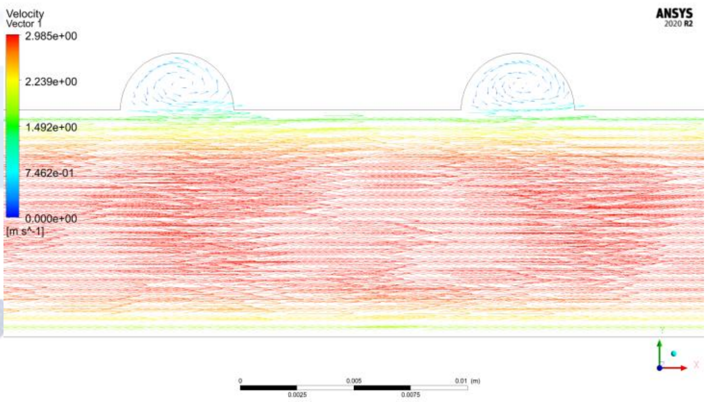
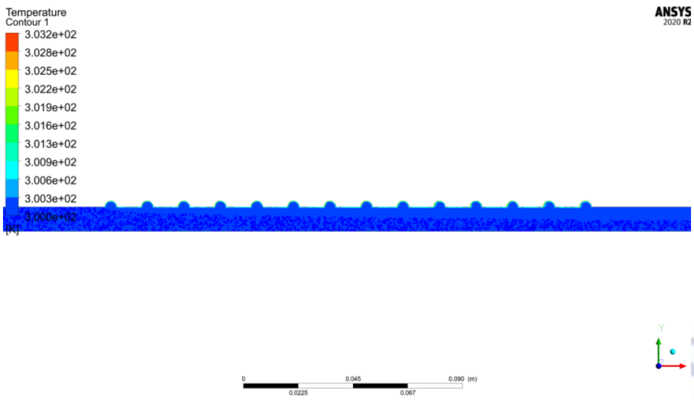
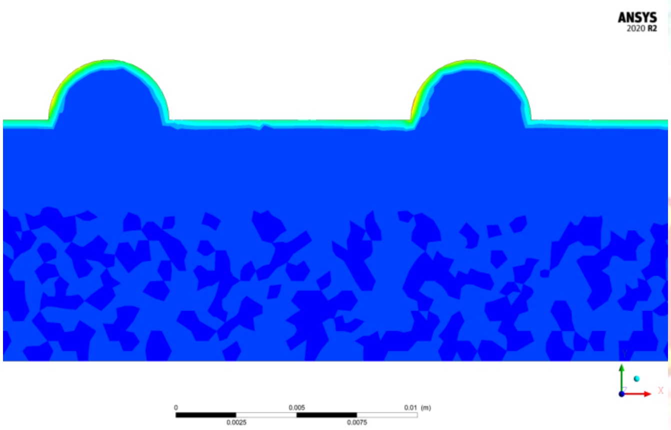


Fig 6.5. *Flow Distribution of Corrugated Channel With 0.05 CUO for Re=20000*

The above Fig Represents the Flow Distribution of the Smooth Corrugated Channel with CUO 0.05 for Re=20000 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 velocity is 0.7462 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 2.985 m/s at the Middle of the Pipe.



(a)



(b)

Fig 6.6. *Temperature Distribution in Smooth Corrugated channel with CUO 0.05 for Re=20000*

The above Fig represents the Temperature distribution for Reynolds number of 20000 of the Smooth Corrugated channel using CUO 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 345.4719.

Case 4 Smooth Corrugated Channel With CUO 0.05 for Re=25000

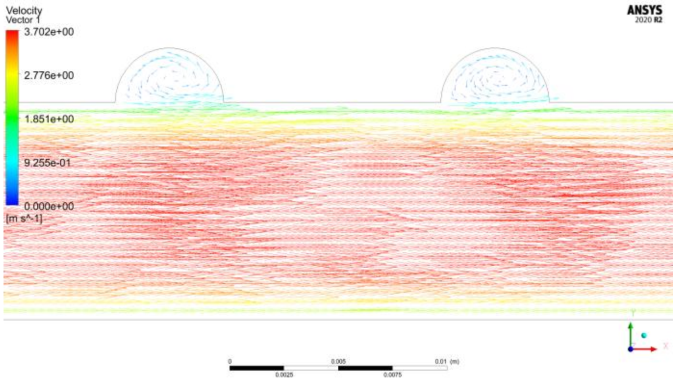
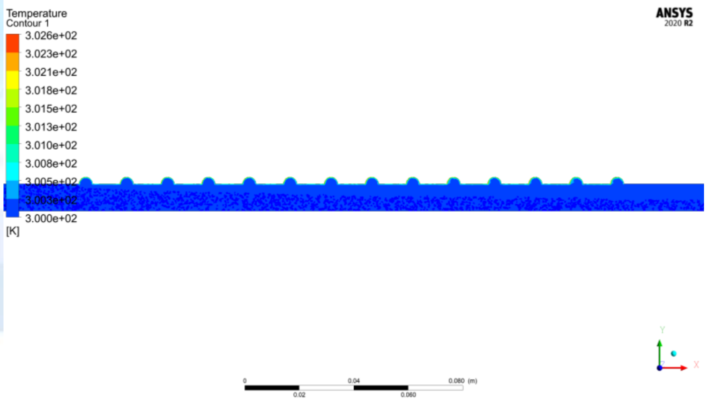
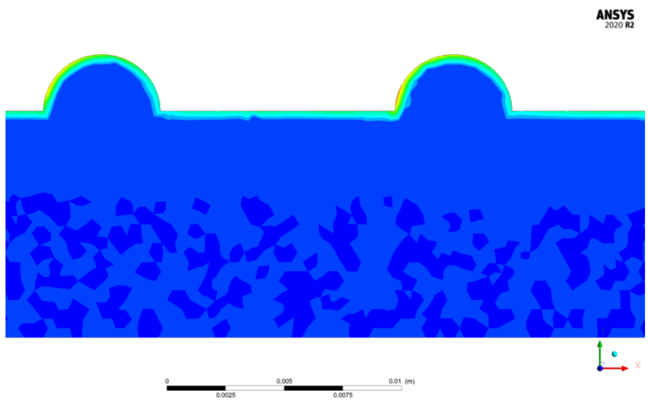


Fig 6.7. *Flow Distribution of Corrugated Channel With 0.05 CUO for Re=25000*

The above Fig Represents the Flow Distribution of the Smooth Corrugated Channel with CUO 0.05 for Re=25000 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 velocity is 0.9255 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 3.702 m/s at the Middle of the Pipe.



(a)



(b)

Fig 6.8.*Temperature Distribution in Smooth Corrugated channel with CUO 0.05 for Re=25000*

The above Fig represents the Temperature distribution for Reynolds number of 25000 of the Smooth Corrugated channel using CUO 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 406.6383.

Case 5 Smooth Corrugated Channel With CUO 0.05 for Re=30000

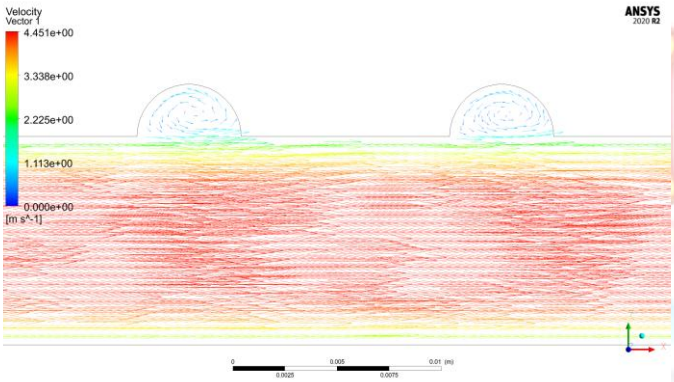
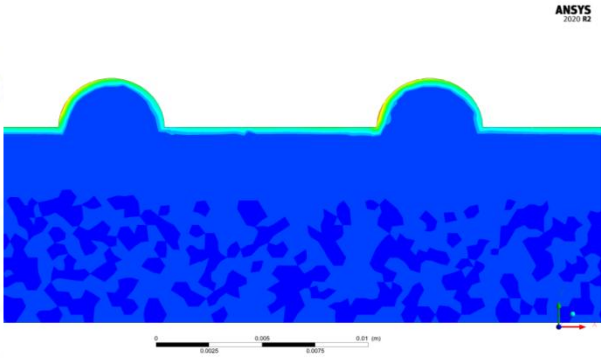


Fig 6.9.*Flow Distribution of Corrugated Channel With 0.05 CUO for Re=30000*

The above Fig Represents the Flow Distribution of the Smooth Corrugated Channel with CUO 0.05 for Re=30000 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 velocity is 1.113 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 4.451 m/s at the Middle of the Pipe.

(a)



(b)

Fig 6.10.*Temperature Distribution in Smooth Corrugated channel with CUO 0.05 for Re=30000*

The above Fig represents the Temperature distribution for Reynolds number of 30000 of the Smooth Corrugated channel using CUO 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 469.8296.

Case 5 Smooth Corrugated Channel With CUO 0.05 for Re=35000

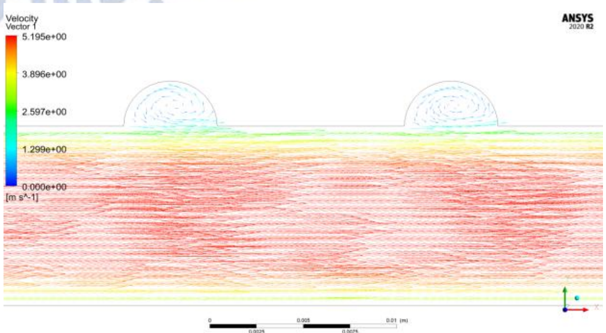
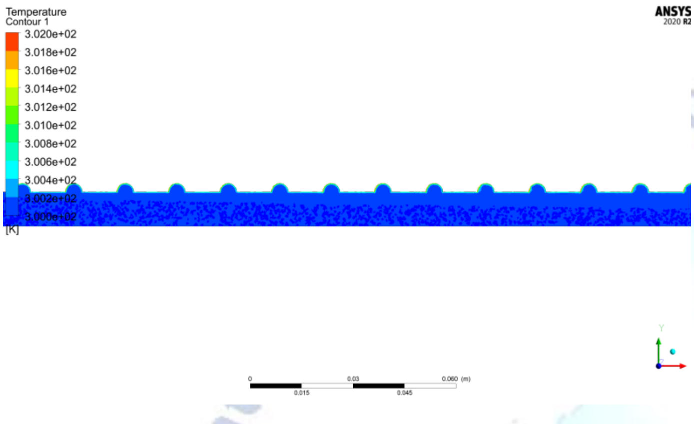
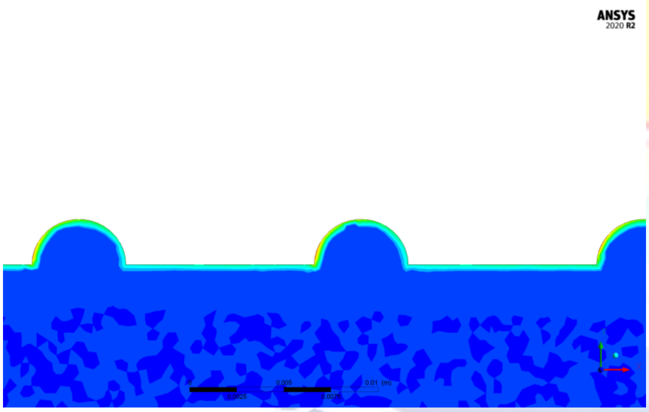


Fig 6.11.*Flow Distribution of Corrugated Channel With 0.05 CUO for Re=35000*

The above Fig Represents the Flow Distribution of the Smooth Corrugated Channel with CUO 0.05 for Re=30000 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 velocity is 1.299 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 5.195 m/s at the Middle of the Pipe.



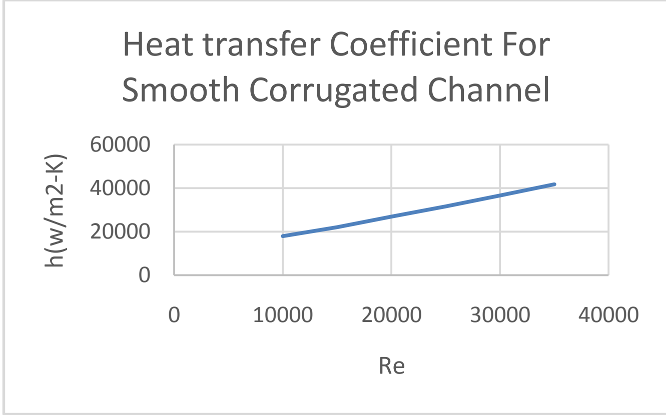
(a)



(b)

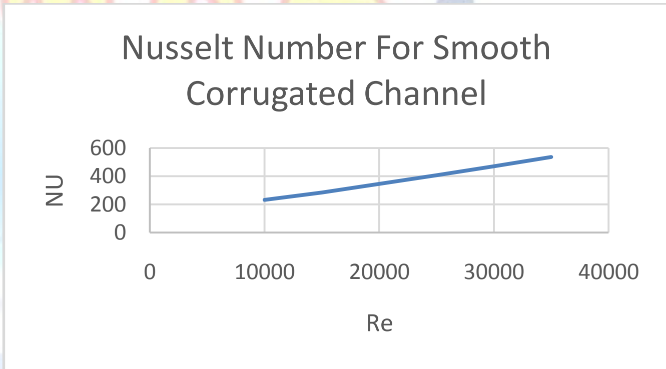
Fig 6.12. *Temperature Distribution in Smooth Corrugated channel with CUO 0.05 for Re=35000*

The above Fig represents the Temperature distribution for Reynolds number of 35000 of the Smooth Corrugated channel using CUO 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 535.1359.



Plot 6.1. *Re Vs Heat transfer Coefficient for Smooth corrugated Tube.*

The above Plot is Drawn between Variable Reynolds Number Vs Heat Transfer Coefficient where in the Trend of the plot we can see the Significant increase in the Heat transfer coefficient with increase in the Reynolds Number of nano particles this is due to the increase of Mass Flow Rate within the Fluid. Maximum Heat transfer Coefficient at 0.05 Volume Fraction With CUO is 41705.07009 w/m2-K at Re=35000.



Plot 6.2. *Re Vs Nusselt Number for Smooth corrugated Tube.*

The above Plot is Drawn between Variable Reynolds number Vs Nusselt Number where in the Trend of the plot we can see the Significant increase in the Nusselt Number with increase in the Reynolds number of nano particles this is due to the increase of Mass flow within the Fluid. Maximum Nusselt number at 0.05 is 535.1359.

Case 7 Rough Corrugated Channel With CUO 0.05 for Re=10000

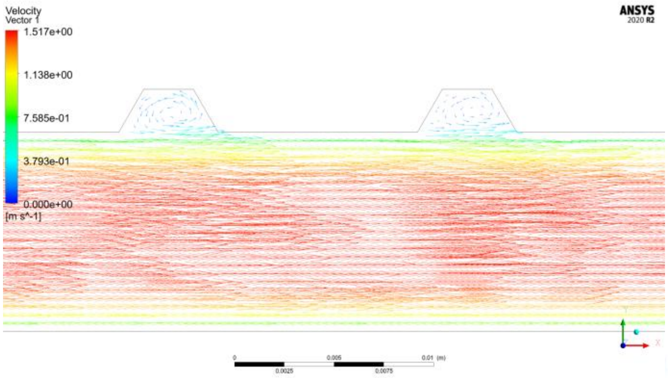
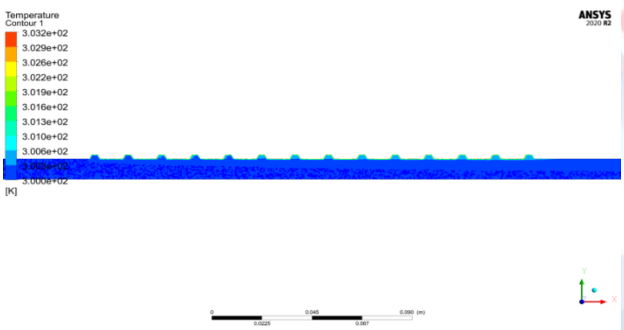
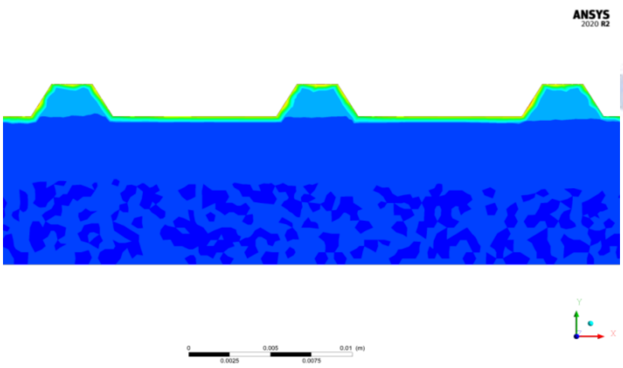


Fig 6.13.*Flow Distribution of Rough Corrugated Channel With 0.05 CUO for Re=10000*

The above Fig Represents the Flow Distribution of the Rough Corrugated Channel with CUO 0.05 for Re=10000 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 velocity is 0.3793 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 1.517 at the Middle of the Pipe.



(a)



(b)

Fig 6.14.*Temperature Distribution in Rough Corrugated Channel with CUO 0.05 for Re=10000*

The above Fig represents the Temperature distribution for Reynolds number of 10000 of the Rough Corrugated Channel using CUO 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 246.955.

Case 8 Rough Corrugated Channel With CUO 0.05 for Re=15000

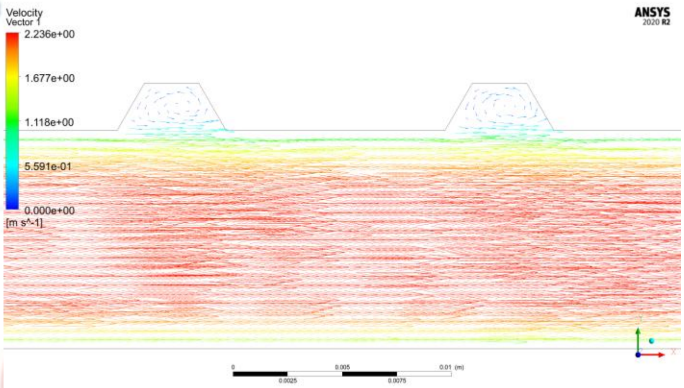
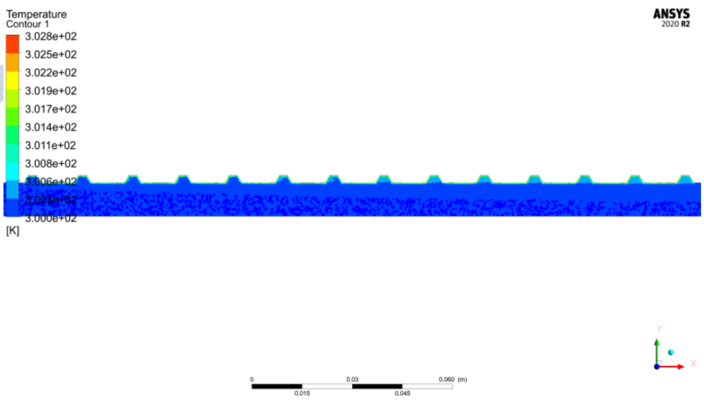
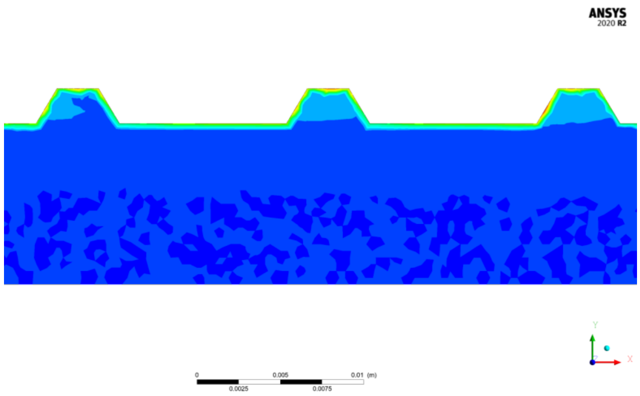


Fig 6.15.*Flow Distribution of Rough Corrugated Channel With 0.05 CUO for Re=15000*

The above Fig Represents the Flow Distribution of the Rough Corrugated Channel with CUO 0.05 for Re=15000 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 velocity is 0.5591 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 2.236 m/s at the Middle of the Pipe.



(a)



(b)

Fig 6.16 *Temperature Distribution in Rough Corrugated Channel with CUO 0.05 for Re=15000*

The above Fig represents the Temperature distribution for Reynolds number of 15000 of the Rough Corrugated Channel using CUO 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 308.6594.

Case 9 Rough Corrugated Channel With CUO 0.05 for Re=20000

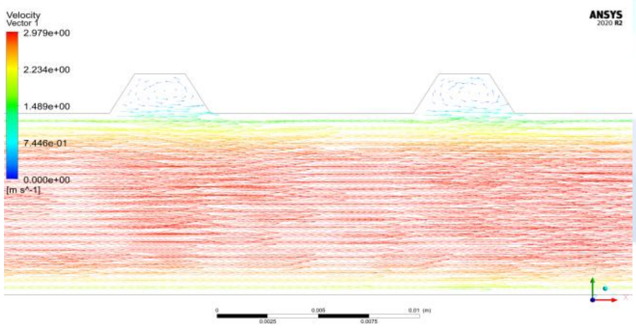
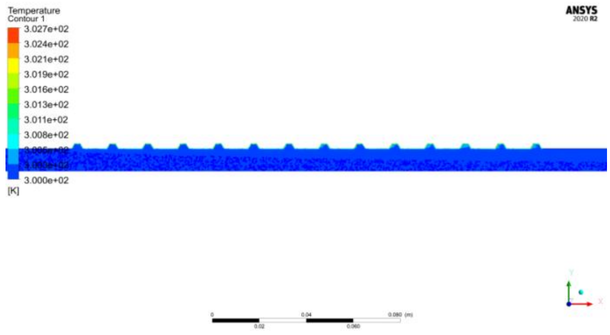
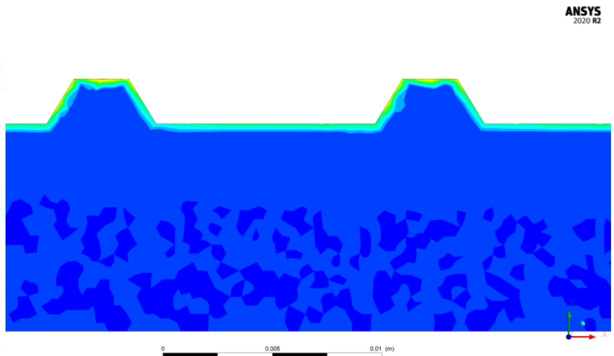


Fig 6.17. *Flow Distribution of Rough Corrugated Channel With 0.05 CUO for Re=20000*

The above Fig Represents the Flow Distribution of the Rough Corrugated Channel with CUO 0.05 for Re=20000 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 velocity is 0.7446 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 2.979m/s at the Middle of the Pipe.



(a)



(b)

Fig 6.18. *Temperature Distribution in Rough Corrugated Channel with CUO 0.05 for Re=25000*

The above Fig represents the Temperature distribution for Reynolds number of 20000 of the Rough Corrugated Channel using CUO 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 377.3485.

Case 10 Rough Corrugated Channel With CUO 0.05 for Re=25000

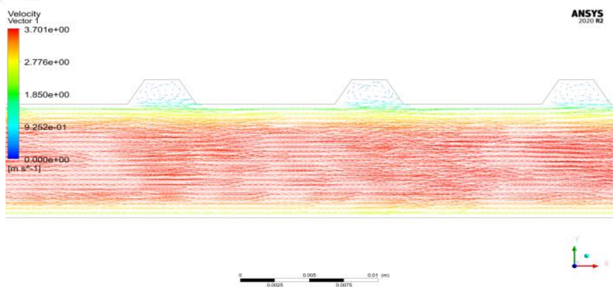
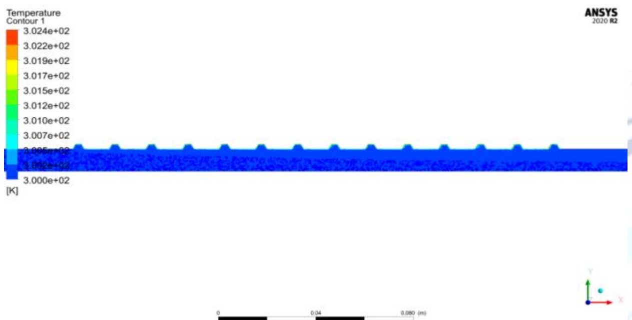
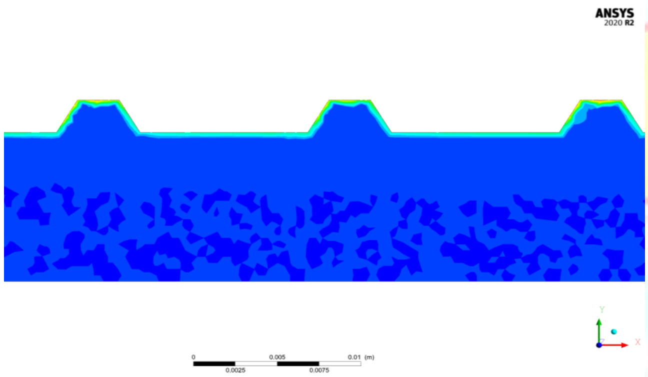


Fig 6.19. *Flow Distribution of Rough Corrugated Channel With 0.05 CUO for Re=25000*

The above Fig Represents the Flow Distribution of the Rough Corrugated Channel with CUO 0.05 for Re=25000 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 velocity is 0.9252 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 3.701 m/s at the Middle of the Pipe.



(a)



(b)

Fig 6.20.*Temperature Distribution in Rough Corrugated Channel with CUO 0.05 for Re=25000*

The above Fig represents the Temperature distribution for Reynolds number of 25000 of the Rough Corrugated Channel using CUO 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 441.405.

Case 11 Rough Corrugated Channel With CUO 0.05 for Re=30000

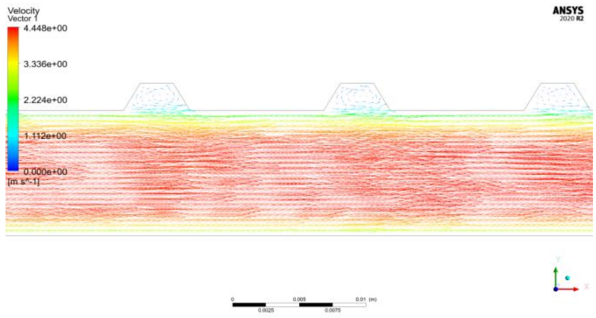
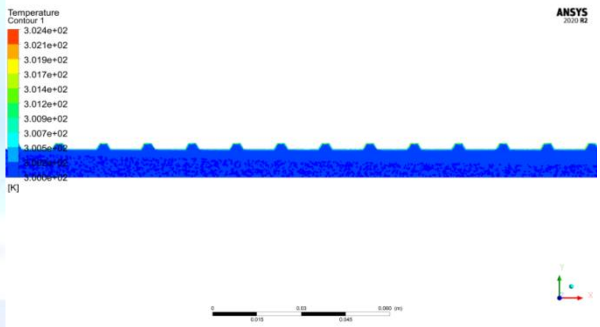
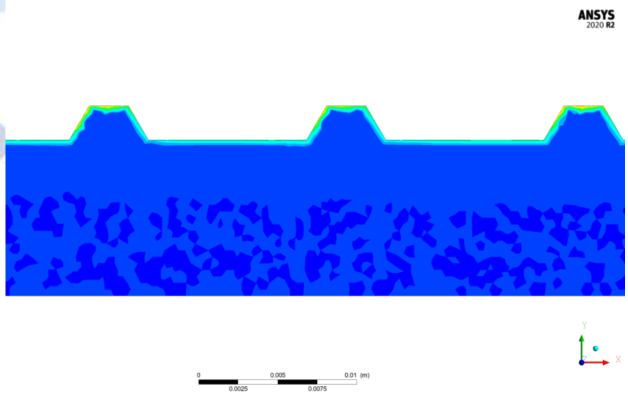


Fig 6.21.*Flow Distribution of Corrugated Channel With 0.05 CUO for Re=30000*

The above Fig Represents the Flow Distribution of the Rough Corrugated Channel with CUO 0.05 for Re=30000 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 velocity is 1.112 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 4.448 m/s at the Middle of the Pipe.



(a)



(b)

Fig 6.22.*Temperature Distribution in Rough Corrugated Channel with CUO 0.05 for Re=30000*

The above Fig represents the Temperature distribution for Reynolds number of 30000 of the Rough Corrugated Channel using CUO 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 404.638.

Case 12 Rough Corrugated Channel With CUO 0.05 for Re=35000

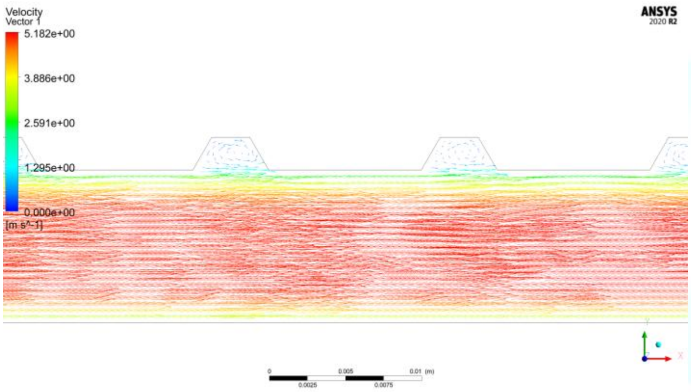
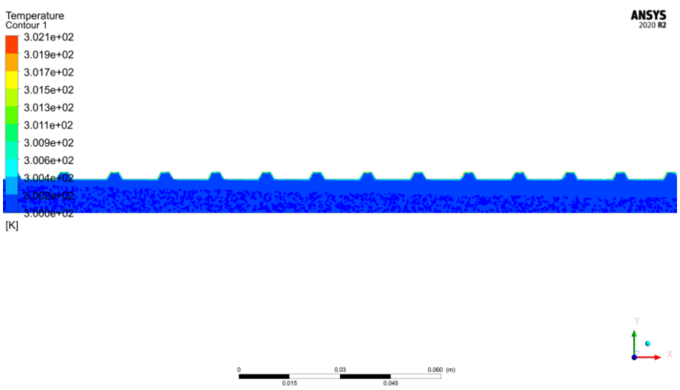
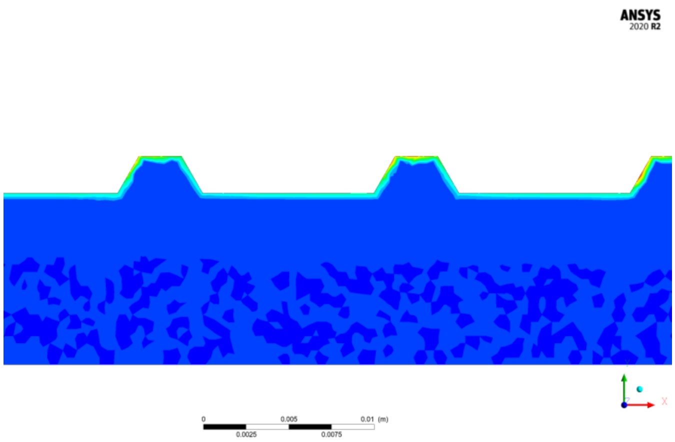


Fig 6.23. *Flow Distribution of Corrugated Channel With 0.05 CUO for Re=35000*

The above Fig Represents the Flow Distribution of the Rough Corrugated Channel with CUO 0.05 for Re=30000 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 velocity is 1.295 m/s in the smooth Corrugated Region and the Red colored region represents the maximum Velocity of 5.182 m/s at the Middle of the Pipe.



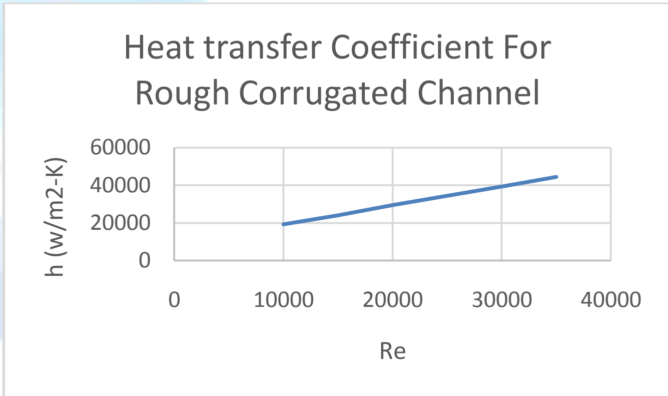
(a)



(b)

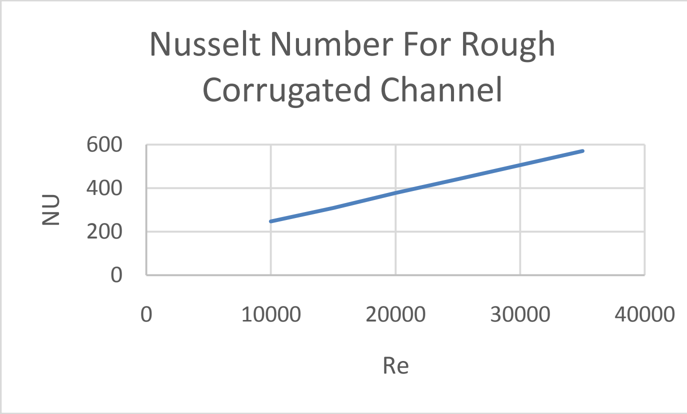
Fig 6.24. *Temperature Distribution in Rough Corrugated Channel with CUO 0.05 for Re=35000*

The above Fig represents the Temperature distribution for Reynolds number of 35000 of the Rough Corrugated Channel using CUO 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 569.956.



Plot 6.3. *Re Vs Heat transfer Coefficient for Smooth corrugated Tube*

The above Plot is Drawn between Variable Reynolds Number Vs Heat Transfer Coefficient where in the Trend of the plot we can see the Significant increase in the Heat transfer coefficient with increase in the Reynolds Number of nano particles this is due to the increase of Mass Flow Rate within the Fluid. Maximum Heat transfer Coefficient at 0.05 Volume Fraction With CUO is 44418.190 w/m2-K at Re=35000.



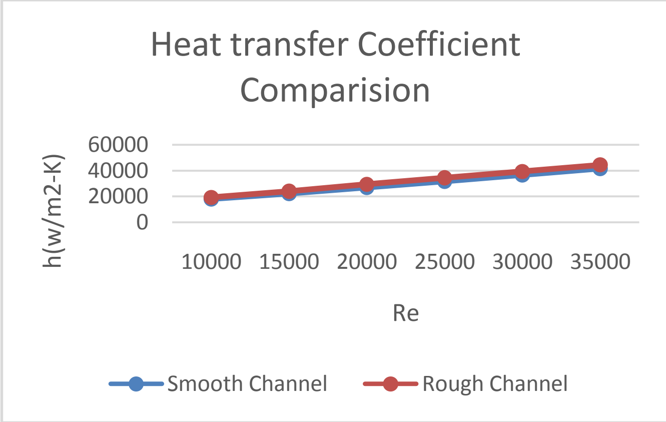
Plot 6.4.*Re Vs Nusselt Number for Smooth corrugated Tube.*

The above Plot is Drawn between Variable Reynolds number Vs Nusselt Number where in the Trend of the plot we can see the Significant increase in the Nusselt Number with increase in the Reynolds number of nano particles this is due to the increase of Mass flow within the Fluid. Maximum Nusselt number at 0.05 is 569.95.

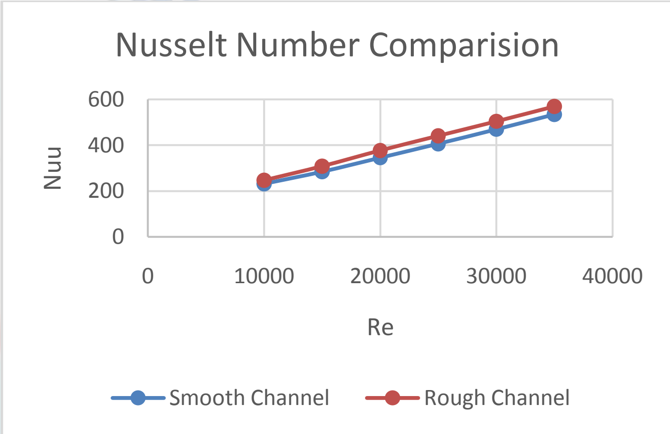
CONCLUSION

Numerical simulations of turbulent forced convection heat transfer in a Smooth and Rough corrugated channel subjected to uniform heat flux were carried out. The computations were performed for a symmetrical Smooth and Rough corrugated channel with varying Reynolds Numbers For CUO Nano Fluid The results of numerical solution showed that Nu increase with increasing the Re . 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 CUO and volume fraction 5%. Based on the above results, the use of nanofluids in smooth and Rough 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 Reynolds numbers from the graph we can say that Rough Channel with CUO 5% we observe High Nusselt Number.



Plot 7.1.*Re VS heat transfer coefficient Comparison for Smooth and Rough Corrugated Channels*



Plot 7.2.*Re VS Nusselt Comparison for Smooth and Rough Corrugated Channels*

REFERENCES

[1] Sundén B, Sköldheden T. Heat transfer and pressure drop in a new type of corrugated channels. International communications in heat and mass transfer. 1985 Sep 1;12(5):559-66.

[2] Sawyers DR, Sen M, Chang HC. Heat transfer enhancement in three-dimensional corrugated channel flow. International Journal of Heat and Mass Transfer. 1998 Nov 1;41(22):3559-73.

[3] Fabbri G. Heat transfer optimization in corrugated wall channels. International Journal of Heat and Mass Transfer. 2000 Dec 1;43(23):4299-310.

[4] Vasudevaiah M, Balamurugan K. Heat transfer of rarefied gases in a corrugated microchannel. International journal of thermal sciences. 2001 May 1;40(5):454-68.

[5] Wang CC, Chen CK. Forced convection in a wavy-wall channel. International Journal of Heat and Mass Transfer. 2002 Jun 30;45(12):2587-95.

[6] Gradeck M, Hoareau B, Lebouche M. Local analysis of

heat transfer inside corrugated channel. International journal of heat and mass transfer. 2005 May 31;48(10):1909-15.
[7] Naphon P. Laminar convective heat transfer and pressure drop in the corrugated channels. International communications in heat and mass transfer. 2007 Jan 31;34(1):62-71.
[8] Naphon P. Heat transfer characteristics and pressure drop in channel with V corrugated upper and lower plates. Energy conversion and management. 2007 May 31;48(5):1516-24.
[9] Naphon P. Effect of corrugated plates in an in-phase arrangement on the heat transfer and flow developments. International Journal of Heat and Mass Transfer. 2008 Jul 15;51(15):3963-71.
[10] Naphon P. Effect of wavy plate geometry configurations on the temperature and flow distributions. International Communications in Heat and Mass Transfer. 2009 Nov 30;36(9):942-6.
[11] Rostamani M, Hosseinizadeh SF, Gorji M, Khodadadi JM. Numerical study of turbulent forced convection flow of nanofluids in a long horizontal duct considering variable properties. International Communications in Heat and Mass Transfer. 2010 Dec 31;37(10):1426-31.
[12] Ahmed MA, Yusoff MZ, Ng KC, Shuaib NH. Numerical investigations on the turbulent forced convection of nanofluids flow in a triangular-corrugated channel. Case Studies in Thermal Engineering. 2015 Sep 30;6:212-25.
[13] Jaisankar S, Radhakrishnan TK, Sheeba KN. Experimental studies on heat transfer and friction factor characteristics of forced circulation solar water heater system fitted with helical twisted tapes. Solar Energy. 2009 Nov 30;83(11):1943-52