CFD SIMULATION USING FLUENT TO DETERMINE THE HEAT TRANSFER COEFFICIENT OF A PACKED BED SYSTEM

LIM SING WEE

Report submitted in partial fulfillment for the award of the Degree of Bachelor in Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITY MALAYSIA PAHANG

JUNE 2012

EXAMINER APPROVAL DOCUMENT

UNIVERSITY MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I hereby certify my final year project entitled "CFD simulation using Fluent to determine the heat transfer coefficient of a packed bed system" is written by Lim Sing Wee. I have examined the final copy of this project and in my opinion this report is fully adequate in term of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering.

Mr. Tuan Mohammad Yussoff Shah bin Tuan Ya
Examiner Signature

SUPERVISOR DECLARATION

I hereby declare that I have read this report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature :
Supervisor : Professor Dr. K. V. Sharma
Date : 25 JUNE 2012

STUDENT DECLARATION

I declare that this report entitled CFD simulation using Fluent to determine the heat transfer coefficient of a bed packed system report is the result of my own research & learning except as cited in references. The report has not been accepted for any degree is not concurrently submitted in candidature for any other degree.

Signature : …….
Name : LIM SING WEE
Date : 25 JUNE 2012

ACKNOWLEDGEMENT

I would like to express my gratitude to my research supervisor Professor K. V. Sharma for his nonstop guidance, supervision and advice with solution for the problem that i have encountered. With my research supervisor effort putting on me, I was able to achieve the objective of my research. Although he always busy with his work, he can spend the time in completing this research.

I also wish to express my thankful to Mr. Fadzil bin Abdul Rahim for teach me the way to do the CFD analysis and check the problem and error from the simulation that I have encountered. He has contributed towards my understanding and thoughts. Without the continuous support from him, this thesis would not have been the same as presented here.

I would like to forward my appreciation to both of my parents who always give their support and motivation during the tough time throughout the duration of this study.

ABSTRACT

In this research, a packed bed is used in the heat exchanger will be considered. Particle to fluid heat transfer coefficient is a primal important when analyzing the performance of a heat exchanger. Basically, a bed packed consists of 44- spherical aluminium particles with proper arrangements is located inside a pipe wall will be considered in this research. A hot fluid is flowing through the packed bed in a solid pipe wall and from here, there will be a pressure drop though this system regarding to the fluid mechanic mechanism. Besides, temperature difference between fluid and solid will cause the convection process and from here heat transfer coefficient can be determined. After the results are taken from the numerical experiment, a comparison between the experimental results with past researcher's results can be done and together with the comparison of calculation using relevant formula.

ABSTRAK

Dalam kajian ini, penukar haba yang dilengkapi dengan system pembungkusan bahan akan dikaji. Zarah dengan pekali pemindahan haba bendalir adalah sangat penting apabila menganalisis prestasi penukar haba. Secara umunnya, system pembungkusan bahan dalam paip ini mengandungi 44- bebuli aluminum mengikut susunan yang betul akan dikaji. Cecair panas akan melalui system pembungkusan bahan ini dan dari sini akan berlaku perubahan tekanan dalam system ini mengikut mekanisma mekanik bendalir. Sementara itu, perbezaan suhu pada pepejal dan cecair dalam system ini akan menyebabkan proses olakan dan dari sini pekali pemindahan haba boleh ditentukan. Selepas keputusan diambil dari eksperimen, perbandingan di antara keputusan eksperimen dengan keputusan penyelidik lepas boleh dilakukan dan bersama-sama dengan perbandingan pengiraan menggunakan formula yang berkaitan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

D= effective particle diameter

s_v = specific surface of a particle

S_p = surface area of particle

V_p = volume of particle

ρ = fluid density

ε = dimensionless void fraction

μ = fluid viscosity

f_p = friction factor

Re = Reynold Number

Nu = Nusselt Number

St = Stranton number

C_7H_8 = toluene

Al_2O_3 = aluminium oxide

H = height of bed

ΔP = pressure drop

v = velocity of fluid

C_{nf} = specific heat of nanofluid

ρ_{nf} = density of nanofluid

K_{nf} = thermal conductivity of nanofluid

μ_{nf} = viscosity of nanofluid

Φ = percentage of concentration of nanomaterial in nanofluid

φ = amount of concentration nanomaterial in nanofluid (in percentage)

CHAPTER 1

INTRODUCTIONS

1.1 BACKGROUND OF STUDY

The convective coefficients of a packed bed heat exchanger are important in many process heat transfer equipment. As an example, within a heat exchanger the evaluation of temperature profile as well as the heat transfer rates of the bed packed is essential to control the performance of the heat exchanger. Hence, in this study is required to develop a packed bed heat exchanger model with suitable software for the estimation of heat transfer coefficients using water as the working fluid. Fluid may be heated from the wall while flowing through the packed bed system.

From here, by applying the theory of heat flow or the movement of thermal energy from place to place, heat is transferred in three methods that are conduction, convection and radiation. Conduction is heat transfer requires the physical contact of two objects. In the case of a wall, heat is conducted through the layers within the wall from the warmer side to the cooler side. Meanwhile, convection is heat transfer due to fluid or air flow. In here, heat is transferred from the wall of water is called as convection. For radiation, heat is transferred when surfaces exchange electromagnetic waves, such as light, infrared radiation, UV radiation or microwaves. Although radiation does not require any fluid medium or contact, but does require an air gap or other transparent medium between the surfaces exchanging radiation.

In such a packed bed operated under steady-state conditions, a difference in local temperature between the fluid and the particle may exist, but the overall solid and fluid temperature profiles are considered to be identical to each other. The temperature profiles in the bed are then predicted in terms of effective thermal conductivities and

wall heat transfer coefficients. An extensive review of the aforementioned can be found in Wakao and Kuguei, 1982.

A Computerized Fluid Dynamic (CFD) simulation is a most suitable strategy for the estimation of effective thermal conductivities as well as wall heat transfer coefficients. CFD is a tool uses numerical methods and algorithms to analyze systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The simulation of CFD is performed using the FLUENT software. CFD allows us to obtain a more accurate view of the fluid flow and heat transfer mechanisms present in packed bed heat exchanger.

Fluent is a computer program for modeling fluid flow and heat transfer in complex geometries. Fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Fluent also allows you to refine your grid based on the flow solution.

1.2 PROBLEM STATEMENT

The problems begin with a hot fluid is flowing through a hollow tube pipe with a packing material inside the pipe. The packing materials are spherical solid materials. The fluid is flowing through each spherical packing material through the column of the packing material. The energy of the hot fluid is transferred to the solid sphere through the convection process. The differences of temperature of the pipe wall and the fluid also make the energy transfer of fluid to the wall. Those energy transfers are simulated using the Fluent software in computational fluid dynamic (CFD). Although the simulated results are not as accurate as physical experiment results but simulated results are almost can be referred results. CFD simulations are relatively inexpensive because the cost of the powerful computer to simulate the design can be cheaper than the experimental solution. CFD simulations can be executed in a short period of time. Hence, the fastest and almost an accurate way to solve that problem or through the computer software and using CFD simulation.

1.3 OBJECTIVE

To determine convective heat transfer coefficient and pressure drop of a packed bed heat exchanger using Fluent software.

1.4 SCOPES

1. Develop the model of a packed bed system in Solidwork or any other commercial software available compatible with CFD package
2. Import the model and initialize for boundary conditions. Process/ Execute the model using the properties of water
3. The experimental data available for the water is to be validated with computational results for heat transfer coefficients
4. Evaluate the numerical heat transfer coefficient for nanofluid using the properties developed in the form of equations.

CHAPTER 2

LITERATURE REVIEWS

2.1 INTRODUCTIONS

The aim of this chapter is to provide the past research about the bed packed system and computational fluid dynamics (CFD) analysis in three dimension model. In order to understand more on this research and to achieve the objective of the research, reviewing back past research studies are needed to provide more useful information and point.

2.2 BED PACKED SYSTEM

Bed packed system is a hallow tube filled with fixed layer of small particles or packing material. The packing material can be any sizes and shapes but for this research, it is spherical aluminum particles and a fluid is flowing through the bed packed particles. The purpose of this system is to use for processes involving absorption, absorption of a solute, distillation, filtration and separation (Geamkoplis, p.125). One of the studies in this research is pressure drop. It is because pressure drop is important to determine the energy requirement to pump a fluid at any bed packed system. Besides, from the viewpoint of fluid dynamics, the most important cases are the pressure drop required for fluid to flow through the column at a specified flow rate.

In order to calculate this amount, we are dependent on the correlation of coefficient friction due to the Ergun. Ergun relates the flows and pressure drops to a Reynolds number and friction factor respectively. The Reynolds number for packed

beds, Re_p, depends upon the controlled variable U_{bs} and the system parameters ρ, ε, μ, and D and is defined as (Bird et al., 1996):

$$Re_p = \frac{D \times U_{bs} \times \rho}{\mu\,(1-\varepsilon)}$$

where, D is effective particle diameter = $\frac{6}{s_v}$, s_v is specific surface of a particle = S_p / V_p, S_p is surface area of particle and V_p is volume of particle. ρ is the fluid density, ε is the dimensionless void fraction defined as the volume of void space over the total volume of packing, and μ is the fluid viscosity.

The friction factor, f_p, in the Ergun equations for Reynolds's number range between 1 and 2500 are:

$$f_p = \frac{150}{Re_p} + 1.75$$

2.2.1` Relevant Studies

In "CFD studies on particle-to-fluid mass and heat transfer in packed beds: Free convection effects in supercritical fluids" by Guardo. A**,** Coussirat. M**,** Recasens. F**,** Larrayoz. M. A**,** Escaler. X have checked CFD capabilities for predicting particle-to-fluid mass/heat transfer coefficients when a supercritical fluid was used as a solvent in a packed bed reactor. First, numerical simulations are presented for validation model cases (forced convection at low pressure). Numerical simulation is done to model the mass transfer of mixed convection under high pressure, the analysis results was obtained was compared with experimental data previously issued by (Stüber et al., 1996), and to the heat transfer analogy proposed by (Guardo et al., 2006). Numerical results obtained presented in this study, to validate the idea that the modified correlation presented by (Guardo et al., 2006) can be used to describe the phenomenon of heat transfer in packed bed under mixed convection regime at high pressures. The boundary conditions in this study are as follow

Table 1: Boundary condition for carbon dioxide

Boundary condition	**Low pressure**	**High pressure**
Mass transfer simulations		
Circulating fluid	CO_2	
C_7H_8 concentration at inlet (mol/m^3)	0	
C_7H_8 concentration at particle surface (equilibria) (mol/m^3)	5.95	120–190
Pressure (Pa)	101325	$9–9.2\times10^6$
Mass flow at the inlet	–	0.015–0.100
Velocity at the inlet (m/s)	7.5×10^{-4}–7.5×10^{-1}	–
Heat transfer simulations		
Circulating fluid	Air	CO_2
Temperature at the inlet (K)	298	330
Temperature at particle surface (K)	423	340
Pressure (Pa)	101325	1×10^7
Mass flow at the inlet	–	0.013–0.132
Velocity at the inlet (m/s)	3×10^{-4}–7.5×10^{-1}	–

The fluid was taken to be incompressible, Newtonian, and in a laminar or turbulent flow regime. CO_2, air and toluene at standard conditions were chosen as the simulation fluids. Incompressible ideal gas law for density and viscosity were applied to the model for the production of these variables depends on temperature. For the high-pressure simulations, the fluid was taken to Newtonian, in laminar flow regime and with variable density. CO_2 and toluene, property that has been incorporated into the solver code using the (UDE) user-defined functions and user-defined formula (UDF) was chosen as a fluid simulation in this case, under high pressure.

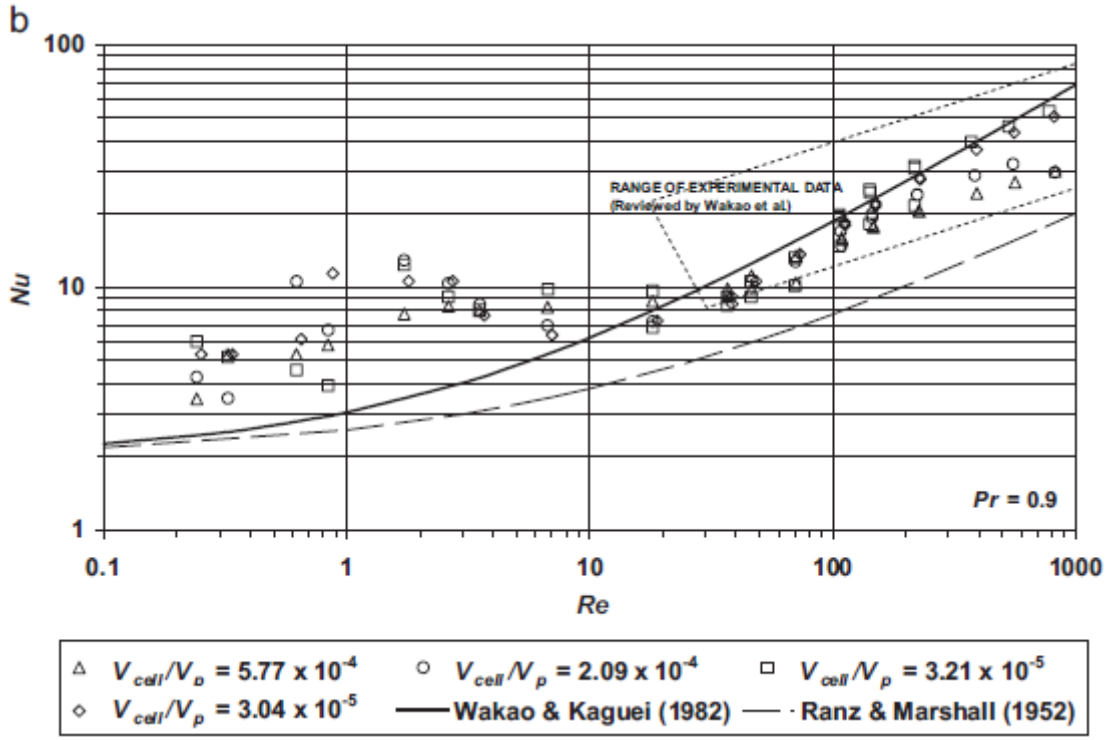

Figure 1: Graph Nusselt vs Reynold number for carbon dioxide

In the end, they find that at lower Reynolds numbers (*Re*<10), mass and heat transfer results obtained show that the fitting against Wakao's correlation is not good because of the CFD model in predicting *Nu* at low *Re* has stated by (Guardo et al., 2006). There is a discrepancy between the results for meshes tested in heat and mass transfer simulations for Re> 300, due to the fact that obtained at higher Re, turbulent transport term in the transport equation is important. Therefore, it is requires a denser mesh around the particles surface to capture the turbulence phenomena involved and the associated improvement in the boundary layer mixing (Guardo et al., 2006). The results obtained with the finer meshes fit better the prediction of (Wakao and Kaguei, 1982) in the turbulent flow zone (*Re*>300) related to a better determination of the vorticity energetic scales effects.

In "CFD study on particle-to-fluid heat transfer in fixed bed reactors: Convective heat transfer at low and high pressure" by Guardo. A**,** Coussirat. M**,** Recasens. F**,** Larrayoz. M.A**,** Escaler. X have review CFD capabilities for predicting particle-to-fluid heat transfer coefficients when a supercritical fluid is used as a solvent in a fixed bed reactor. At first, numerical simulations are presented for a validation

model case (single sphere model), and next is presenting convective particle-to-fluid heat transfer at low pressure. The results obtained are used to analyze mesh dependence of the numerical results at low flow velocities. Finally, mixed convection at high pressure is modeled and analyzed. Numerical results obtained are compared to accepted correlations and a CFD-based correlation for particle-to-fluid heat transfer at high pressure is presented. The boundary conditions are;

Table 2: Boundary condition for air and carbon dioxide

Boundary condition	**Low pressure**	**High pressure**
Circulating fluid	Air	CO_2
Temperature at the inlet, K	298	330
Temperature at packing surface, K	423	340
Pressure, Pa	101 325	1×10^{7}
Mass flow at the inlet, kg/m^{2}s	—	0.013–0.132
Velocity at the inlet, m/s	3×10^{-4}–7.5×10^{-1}	—

Model consists of a single sphere suspended in a box. In the CFD model the infinite fluid was limited in a box with a square flow inlet plane of seven sphere diameters and a length of 16 sphere diameters to keep the model reasonable in size (Nijemeisland, 2000). To discard the presence of wall effects on temperature and velocity profiles, models with flow inlet planes with sizes of 2, 3, 4, 6, 8 and 9 diameters were created. An unstructured tetrahedral mesh is built in the fluid region. No mesh is built in the sphere interior. The sphere in the box was designed with the same dimensions as the spheres used in the fixed bed model.

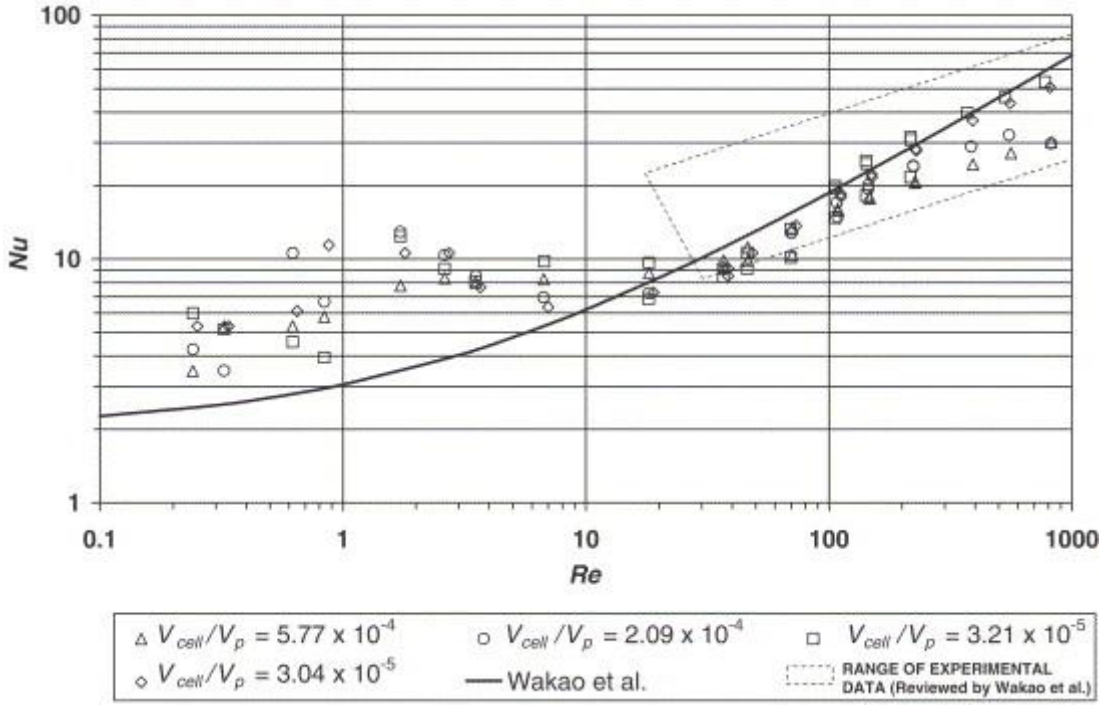

Figure 2: Nusselt number versus Reynold Number for air and carbon dioxide

From the results, they notice that in the laminar flow and transition zone (Re <300), the results not depending on the mesh density. At lower Reynolds numbers (Re <10) shows the results that the fitting against (Wakao, 1976) the correlation is not good. For a single velocity condition different meshes give results in a wide range of Nu and no relationship with mesh density can be determined.

There is a divergence between the results obtained for tested meshes for higher values of *Re*, due to the fact at higher *Re*, turbulent transport term in the transport equation need to consider. An accurate turbulence modeling requires a denser mesh around the particles surface in order to capture in a more suitable way the involved turbulence phenomena in the boundary layer (Guardo et al., 2005). A divergence in the results obtained with the low density meshes and the high density meshes can be seen for *Re*>300.

In "Heat and Flow Characteristics of Packed Bed by (Achenbach. E, 1995), mass transfer experiments with single spheres are preferably conducted according to the method of naphthalene sublimation in air. The majority of the present experiments were conducted using a bed diameter of D = 0.983 m and a bed height of H = 0.84 m. To eliminate wall effects, the core wall was structured such that a regular orientation of the

spheres adjacent to the wall was avoided. The sphere diameter was d = 0.06 m. The void fraction was experimentally determined to be 0.387. The heat transfer experiments were carried out by applying the method of the electrically heated single sphere in an unheated packing. The test spheres were manufactured from copper, the surface being highly polished and covered with a silver layer to keep the contribution of thermal radiation low.

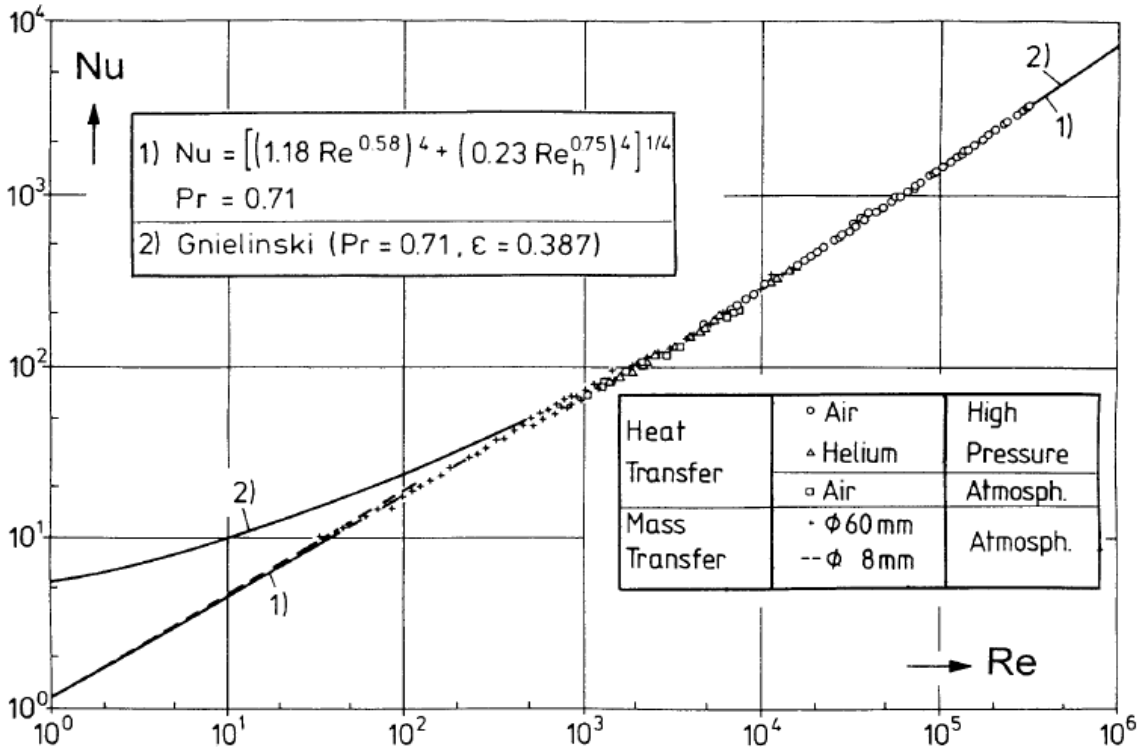

Figure 3: Convective particle to fluid heat transfer

2.2.2 Pressure drop along the bed packed

The increasing of pressure drop along the bed pack is due to the wake region is created by each solid particle in the bed packed when water flow pass through it. Wake region is a recirculating flow immediately behind a moving or stationary solid body and it is caused by the flow of surrounding fluid around the body. The pressure is a maximum at the stagnation point (first point contact of fluid and spherical particles) and gradually decreases along the front half of the spherical particles of bed packed. The pressure starts to increase in the rear half of the spherical particles of bed packed and the particle now experiences an adverse pressure gradient. Consequently, the flow

separates from the surface and creating a highly turbulent region behind the spherical particles of bed packed called the wake region. The pressure inside the wake region remains low as the flow separates and a net pressure force (pressure drag) is produced.

2.2.3 Pressure drop of bed packed in different fluid flow velocity

The increasing pressure drop in different velocity is because of increasing of velocity is directly proportional to increasing of pressure drop value in pressure drop formula.

Pressure drop can be calculated using the following formula:

$$\Delta P = \frac{v^2 \times f \times L \times \rho}{2D_{pipe}}$$

Where;

ΔP = pressure drop in Pascals (Pa)

v = velocity in metres per second (m/sec)

f = friction factor

L = length of pipe in metres (m)

ρ = density of the fluid in kilograms per cubic metre

D_{pipe} = inside diameter of pipe in metres (m)

Hence, when the velocity of the fluid is increasing, the pressure drop values are also increasing.

2.2.4 Heat transfer coefficient in different fluid flow velocity

All the graph of Nusselt number versus Reynold number in different fluid properties must shows the increasing pattern. It is because due to the convection process between fluid and the bed packed. Convection is the mode of energy transfer between solid surface and the adjecent fluid in motion and it involves combined effect of conduction and fluid motion. Hence, the faster the fluid motion, the greater the convection heat transfer and the higher the Nusselt number will be.

2.3 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations. CFD provides numerical approximation to the equations that govern fluid motion. These equations are then discretized to produce a numerical analogue of the equations.

All CFD codes contain three main elements:

1. A pre-processor, which is used to input the problem geometry, generate the grid, define the flow parameter and the boundary conditions to the code.
2. A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. There are four different methods used as a flow solver: (i) finite difference method; (ii) finite element method, (iii) finite volume method, and (iv) spectral method.
3. A post-processor, which is used to massage the data and show the results in graphical and easy to read format.

2.3.1 History of CFD

In England, Lewis Fry Richardson (1881-1953) developed the first numerical weather prediction system CFD approximation in 1922 when he divided physical space into grid cells and used the finite difference approximations of Bjerknes's "primitive differential equations". His own attempt to calculate weather for a single eight-hour period took six weeks and ended in failure. His model's enormous calculation requirements led Richardson to propose a solution he called the "forecast-factory". The "factory" would have involved filling a vast stadium with 64,000 people. Each one, armed with a mechanical calculator, would perform part of the flow calculation. A leader in the centre, using coloured signal lights and telegraph communication, would coordinate the forecast. What he was proposing would have been a very rudimentary CFD calculation. The earliest numerical solution for flow pass a cylinder is in 1933 that is by Thom. A, publishing ‘The Flow Past Circular Cylinders at Low Speeds’, Proc. Royal Society, A141, pp. 651-666, London, 1933.

During the 1960s, the theoretical division of NASA at Los Alamos in the U.S. contributed many numerical methods that are still in use in CFD today, such as the following methods: Particle-In-Cell (PIC), Marker-and-Cell (MAC), Vorticity- Stream function methods, Arbitrary Lagrangian-Eulerian (ALE) methods, and the ubiquitous k - e turbulence model. In the 1970s, a group working under D. Brian Spalding, at Imperial College, London, developed Parabolic flow codes (GENMIX), Vorticity-Stream function based codes, the SIMPLE algorithm and the TEACH code, as well as the form of the k - e equations that are used today (Spalding & Launder, 1972). They went on to develop Upwind differencing, 'Eddy break-up' and 'presumed PDF' combustion models. Another event of CFD industry was in 1980 when Suhas V. Patankar published “Numerical Heat Transfer and Fluid Flow", probably the most influential book on CFD to date, and the one that spawned a thousand CFD codes.

It was in the early 1980s that commercial CFD codes came into the open market place in a big way. The use of commercial CFD software started to become accepted by major companies around the world rather than their continuing to develop in-house CFD codes. Commercial CFD software is therefore based on sets of very complex non-linear mathematical expressions that define the fundamental equations of fluid flow, heat and materials transport. These equations are solved iteratively using complex computer algorithms embedded within CFD software. The net effect of such software is to allow the user to computationally model any flow field provided the geometry of the object being modelled is known, the physics and chemistry are identified, and some initial flow conditions are prescribed.

CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modeling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTIONS

Basically, the flow of this research is as shown as figure 1. When there is an error in this simulation, the research has to go backward to any cases before the simulation takes place. In this research, geometry model of this bed packed is looked simple but it requires lots of steps to build it. From the overall, a proper geometrical model will define a good result in simulation.

3.2 FLOW CHART

3.2.1 Overall Research Methodology

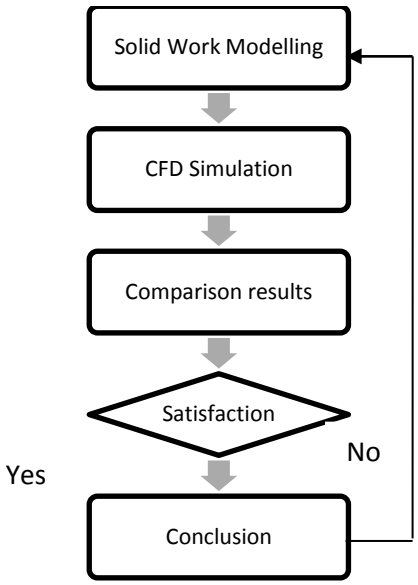

Figure 4: Overall process of research

3.2.2 Steps of CFD analysis

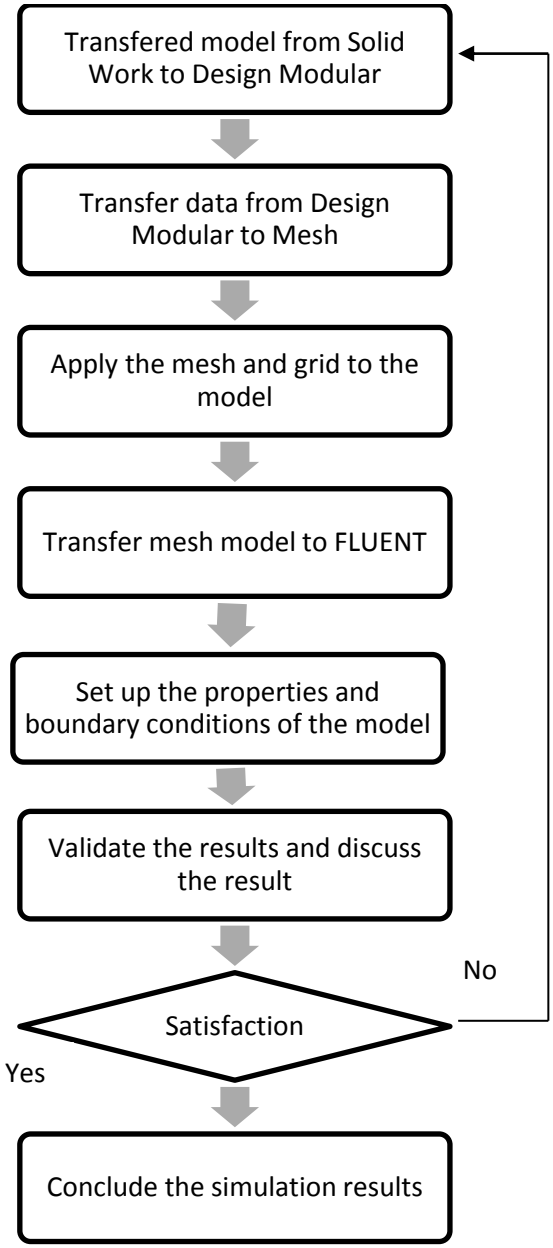

Figure 5: Steps of CFD simulations

3.3 NUMERICAL EXPERIMENT SETUP

3.3.1 Geometrical Modelling

Geometrical modeling is the most difficult task in this bed packed simulation. It is because the CFD is needed all in solid model to define all the properties of the model. For example the liquid volumes of the water in this geometry model also need a solid volume to represent water in CFD analyzing. A technique used for constructing geometry will ensure the feasibility of generating the good mesh to apply in the phenomena involved in the problem. The differential between analysis in Solidwork and CFD are the Solidwork just need a solid surface area to close all the liquid volume and the Solidwork can define the vacuum inside the closed area as liquid properties but in CFD will still define the vacuum area as a vacuum in the analysis.

Hence, for this bed packed constructions are needed a solid pipe wall volume together with 44 solid sphere particle pack inside the solid pipe wall. The construction consists of 11 solid sphere particle for each plane (as shown in figure 3), a total of 36 solid sphere particles are attached to the inner wall of the solid pipe. The next construction is a solid water volume together with 44 vacuum sphere cavity, same size, same arrangement and same coordinate with the solid sphere inside the solid pipe. The ratio of the solid sphere particle and the pipe wall inner diameter are 0.95: 3.9238. The solid sphere must be arranged in circle packing and the entire solid sphere particle must not overlap each other. Finally, assemble or mate those two models together Solidwork (as shown in figure 4).

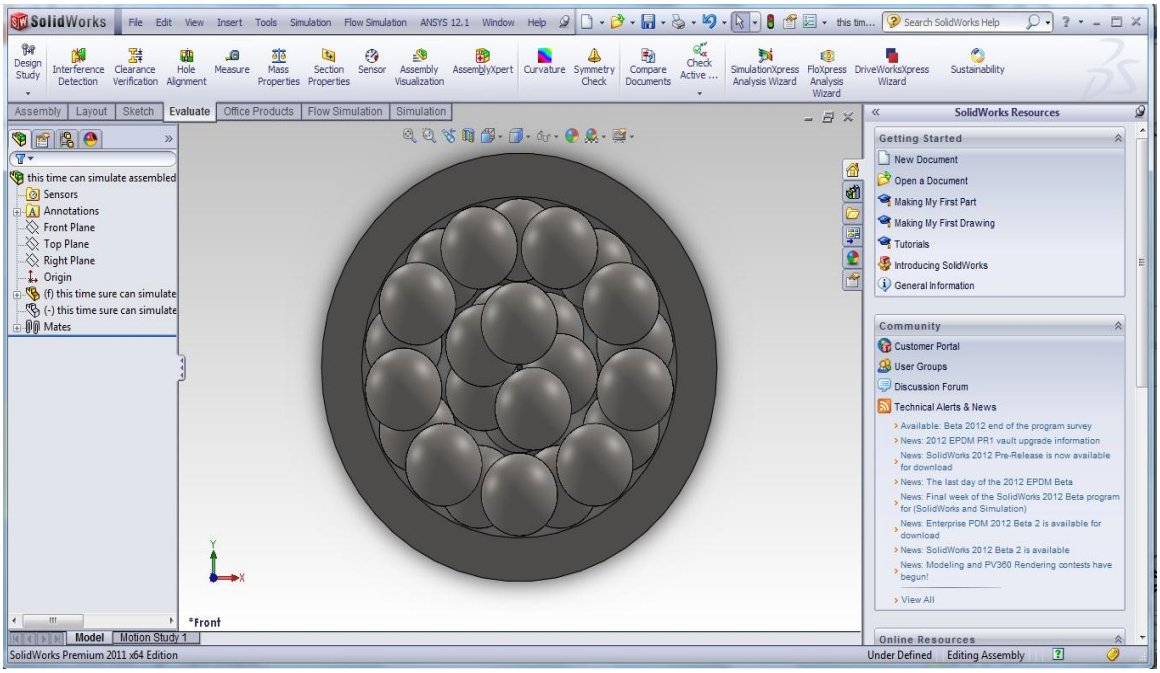

Figure 6: Front view of bed packed model

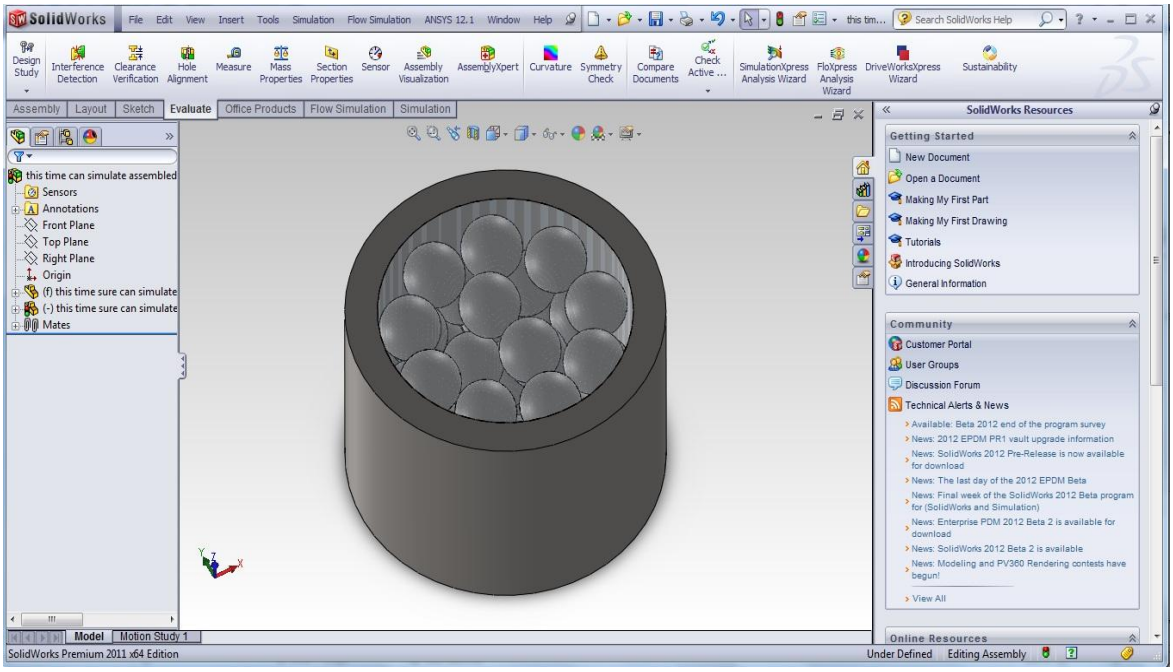

Figure 7: Solidwork isometric view of bed packed model

3.3.2 Design Modular

After finishing the Solidwork modeling, export the Solidwork model file in the ACIS file (. sat). When export the file, the option of saving the file must set to unit centimeter and the version of the file must set to version 12 and below. Import the ACIS file in Design Modular in Workbench. After import, generate the model in Design Modular by right click the model and select the generate button. After the generating process in Design Modular, each part of the model will be divided out with a different name. For this bed packed case, there will be 46 parts generating out of the Design Modular. The total of 46 parts are 1 will represent solid pipe wall, 44 will represents solid sphere particle and last one will represent solid water and the user should identify all the part of the model (as shown in figure 5).

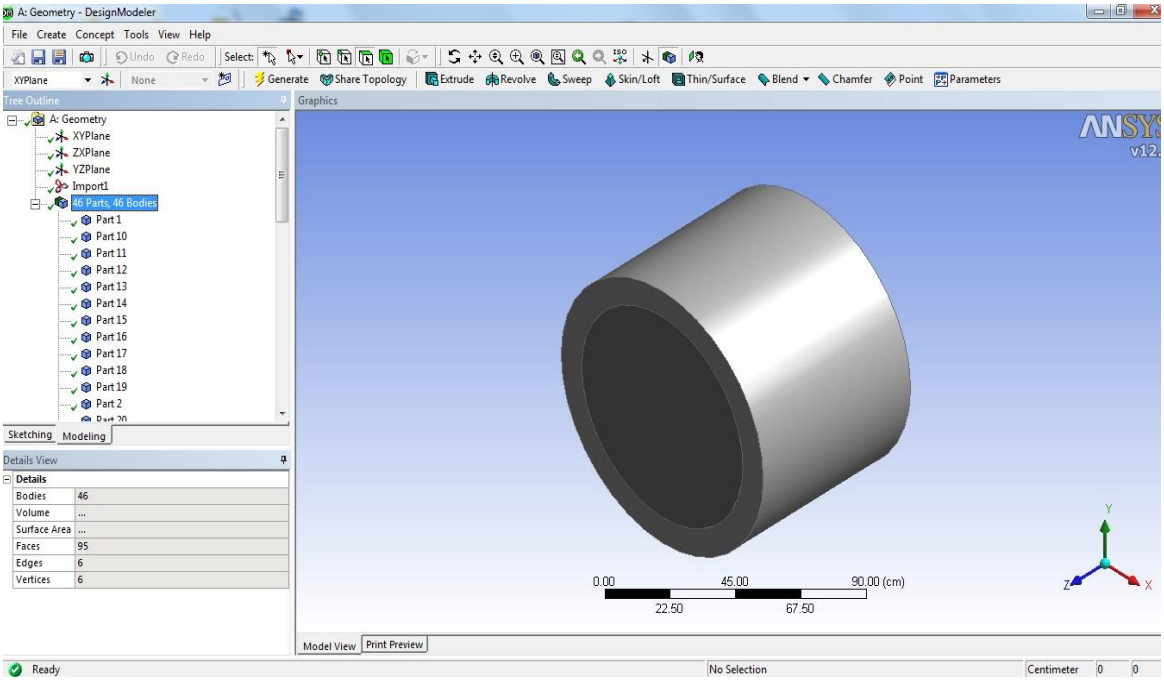

Figure 8: 46 parts in Design Modular of bed packed model

3.3.3 Mesh

Next, the model will go into meshing. The mesh establishes the accuracy of the simulation which has to be chosen with enough detail to describe the simulation processes accurately and with a degree of coarseness that enables solution within an acceptable amount of time. In meshing, this model will use CFD mesh as physical reference and solver reference is Fluent. The meshing advance size function is set as proximity and curvature because proximity is number of mesh elements employed in the gap between two geometry entities and curvature is the angle between normal for adjacent mesh elements. In this model, proximity will set up the mesh volume of the solid pipe wall and curvature mesh will set up the liquid volume together with cavity and all solid sphere particles inside the pipe. Then, generate the mesh to the model and the generation of the mesh is automatically generated by the programme. After applying mesh on the model, inspect the model and looks for the bad meshing surfaces (as shown in figure 6). The bad meshing surface is needed to sweep again to gain smaller and smoother mesh. After all the mesh surfaces and volume is smooth, rename all the surface of the model in different name.

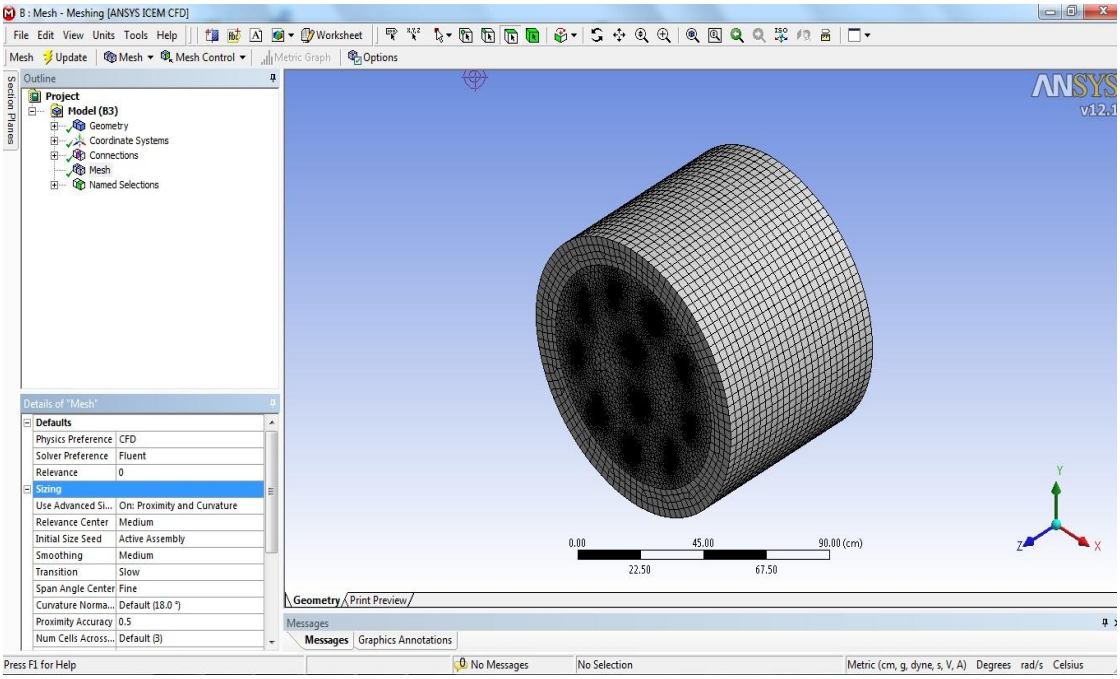

Figure 9: Meshing medium size bed packed model

3.3.4 Fluent

After finishing meshing, just update the workbench and connect the mesh together with Fluent. From here, the user can generate the mesh model in FLUENT. The user is needed to complete the entire task at the left side of the model. The first task is a general inspection of the model. The user is needed to complete the scale that is defining all the view scale in centimeter. When FLUENT reads a mesh file (. msh), all physical dimensions are assumed to be in units of meters. It is because of the model was not built in meters and then it must be scaled. Next is the user can let the programme to check the mesh quality of the model by hitting the check and report quality. The most important part of this mesh analysis is the maximum skewness of the mesh should not exceed the FLUENT tolerance. The next user can set the solver type to apply to the model. There are two kinds of solvers available in FLUENT:-

1. Pressure-based solver
2. Density-based coupled solver (DBCS)

The pressure-based solvers take momentum and pressure (or pressure correction) as the primary variables. Hence, to calculate the pressure drop of the model, it is more suitable to use pressure based solver.

Next is set up the models in problem setup. Change the laminar and energy and let other as default. After that, edit the material information in the problem setup by adding the water liquid properties and aluminium properties. This material general reference for the Fluent system to identify the properties needed to insert in the model.
In cell zone condition, the user is needed to insert the properties of the material in the model. Each part of the model is needed to set the property either aluminium or water liquid properties.

The parts with the aluminium property's motion type are needed to set as stationary that means the volume is not moving. The last part with the water liquid property motion type is needed to set as a moving reference frame to make the flow moving in the mesh. Boundary conditions are for the user to set up the condition of each part of the model. Defining boundary conditions involve identifying the boundary locations and supplying information on the boundaries. For this model, there are three things needed to be set that is a water inlet area, water outlet area and the pipe outer

surface area. The water inlet surface is needed to set as velocity inlet because of need to apply the temperature and the mass flow rate. The water outlet surface is needed to set as pressure- outlet and other is defined by the fluent system. The pipe outer surface is needed to set the temperature only and all other is set as default by fluent.

Table 3: Boundary conditions for water as fluid properties

Boundary Conditions	Value
Circulating fluid	Water
Packing Material	Aluminium
Fluid temperature inlet, K	343
Wall temperature, K	303
Fluid velocity inlet, m/s	0.2- 0.5
Pressure, Pa	101325

In mesh interference, the user is needed to interface all the cavity surface in the water to the solid particle surface and the water surface with the pipe surface. This mesh interference is to let the fluent system identify the joining area of two different volume properties. The final step in problem setup is to set the reference value that is velocity inlet boundary conditions and the projected volume analysis. The projected volume in this model is the water volume.

The solution method is using the pressure- velocity coupling for this model. Pressure- velocity coupling refers to the numerical algorithm which uses a combination of continuity and momentum equations to derive an equation for pressure (or pressure correction) when using the pressure-based solver. The pressure- velocity coupling in this model is using the SIMPLEC scheme, the gradient is set to least squares cell based, pressure is set to second order, the momentum is set to second order upwind and energy is set to second order upwind. SIMPLE-Consistent (SIMPLEC) is used to allow faster convergence for simple problems (e.g., laminar flows with no physical models employed). The gradients of solution variables at cell centres can be determined using a Least-Squares Cell-Based to recommend to polyhedral meshes It has the same accuracy and properties as Node-based Gradients Interpolation schemes for the convection term

Second-Order Upwind is using larger stencils for 2nd order accuracy, essential with tri/tet mesh or when flow is not aligned with the grid; convergence may be slower.

Finally, set up the number of literation of the analysis and for this model it is set to 700 to 1000 iterations.

3.4 APPARATUS

The simulation was run on ASUS Intel core i5, physical memory 8GB workstation and still not enough memory to run for the fine mesh model. The graphic of the workstation is NVIDIA GT 430. 2GB and for each simulation time taken is in ranged from 6 hours to 8 hours and the simulation range is 700 to 1000 iterations. Second order upwind discretization is selected to compute the field variables. The pressure- velocity coupling algorithm was the SIMPLEC scheme.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTIONS

A computational simulation using FLUENT has been done for the bed packed system with 3 different types of fluid flow that is air, water and different concentration of aluminum oxide in water. All the fluid properties are run in FLUENT with a same inlet temperature of 70℃ and in atmospheric pressure. From the simulation, pressure drop coefficient and heat transfer coefficient can be defined by FLUENT software and comparison between the results from a simulation with the numerical calculation and previous researcher has to be done. After finishing the comparison, the same numerical experiment can be done with nanofluid properties.

4.2 Numerical experiment results

4.2.1 Air

Table 4: Simulation results for air

Velocity, *m/s*	**Reynold Number**	**Pressure Drop (Pascal)**	**Heat Transfer coefficient, *W/m ²k***	**Nusselt Number**
0.2	67.32	0.0674	4.93	14.58
0.3	110.13	0.090	6.26	18.54
0.4	153.16	0.178	11.84	35.08
0.5	197.61	0.267	20.59	60.99

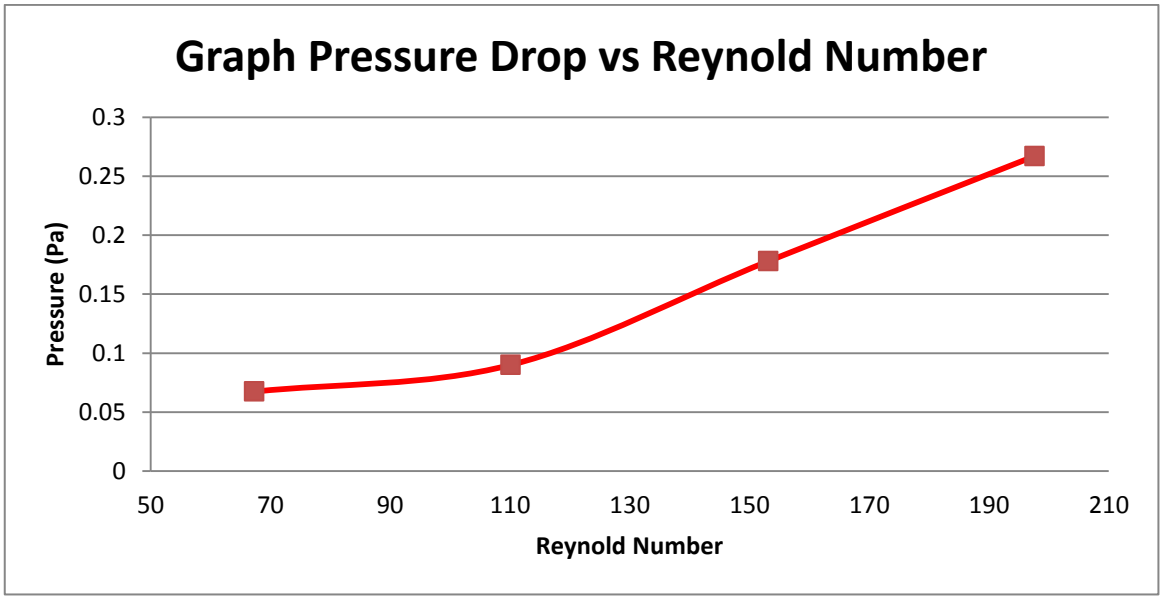

Figure 10: Graph Pressure Drop versus Reynold Number of air

Figure 10 shows the pressure drop versus the Reynold Number from the air as circulating fluid. The graph shows a linear pattern of pressure drop with a velocity of air.

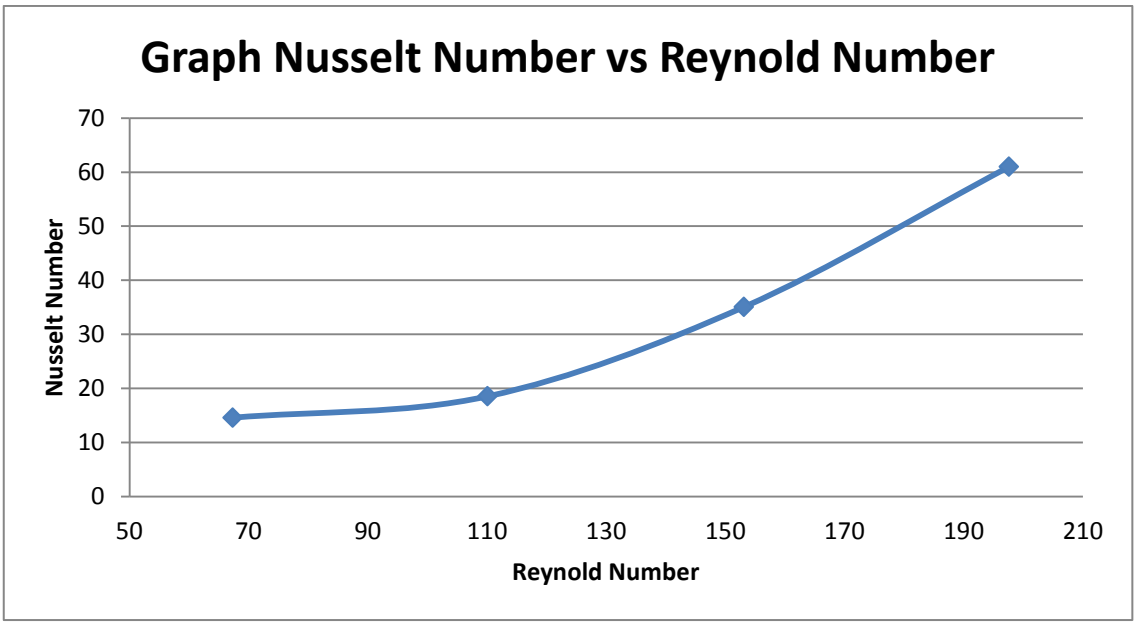

Figure 11: Graph Nusselt Number versus Reynold Number of air

The figure 11 shows the graph Nusselt number versus Reynold number for air as circulating fluid. The graph shows a constant increasing of Nusselt Number with Reynold number.

4.2.2 Water

Table 5: Simulation results for water

Velocity, m/s	Reynolds number	Pressure Drop, *Pa*	Heat transfer coefficient, W/m ²k	Nusselt Number
0.2	1235	50	736.52	88.00
0.3	1874	100	803.18	95.96
0.4	2514	200	805.10	96.17
0.5	3121	400	808.54	96.61

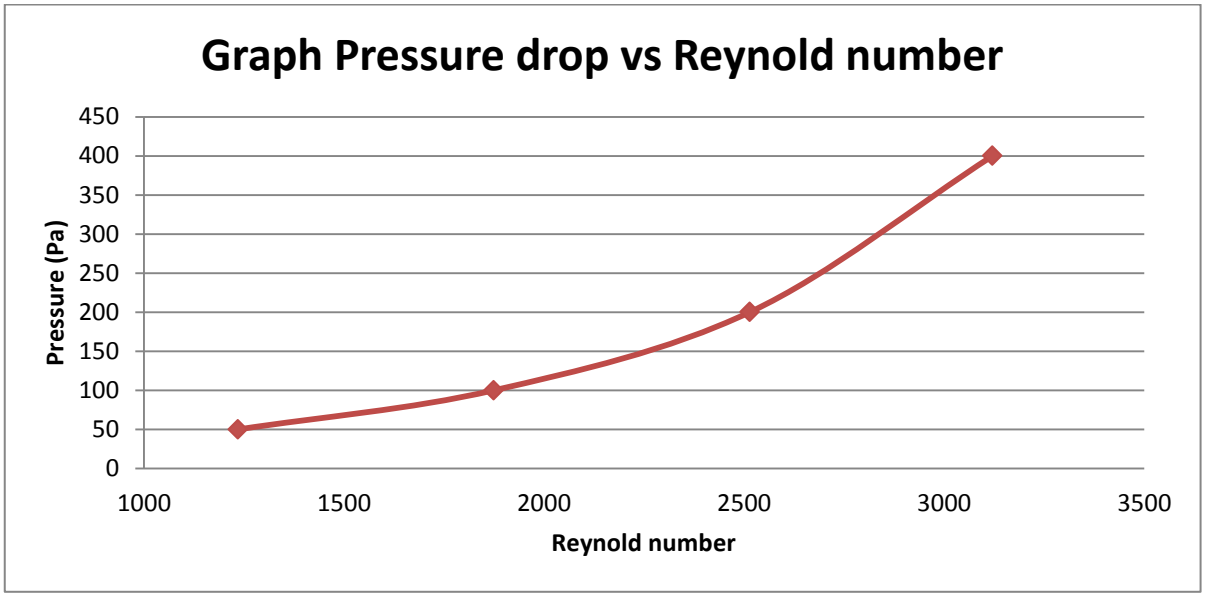

Figure 12: Graph Pressure Drop versus Reynold Number of water

Figure 12 represents the pressure drop along the bed packed system versus the Reynolds number. From the graph above, it shows the steady increasing of pressure drop with the Reynolds number.

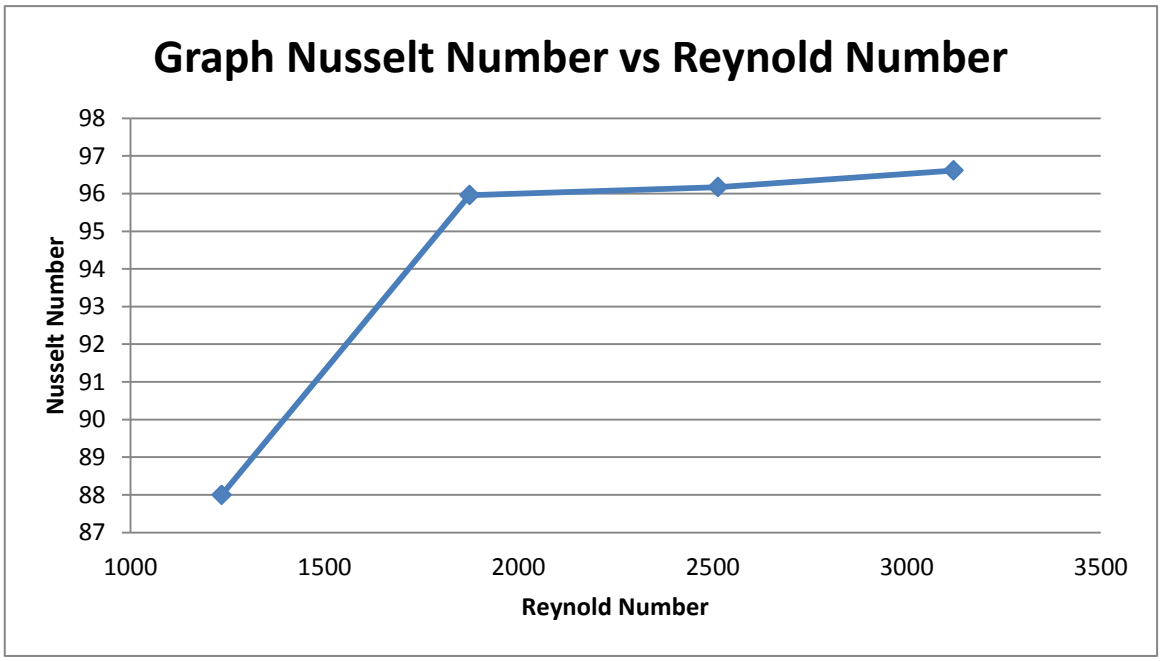

Figure 13: Graph Nusselt Number versus Reynold Number of water

The figure 13 represents the Nusselt number versus Reynolds number. The graph shows the drastically increasing of the heat transfer coefficient at low speed of flow and it continues with almost constant increasing of heat transfer coefficient when it reaches turbulence flow.

4.2.3 Nanofluid: Aluminium Oxide (Al_2O_3) + Water

Table 6: Simulation results for different concentration of aluminum oxide in water

Concentration of Al_2O_3	**Velocity, *m/s***	**Reynold Number**	**Pressure Drop, *Pa***	**Heat Transfer Coefficient, *W/m ²k***	**Nusselt Number**
2%	0.2	2658.08	25.00	242.19	21.53
	0.3	4017.38	66.67	638.01	56.71
	0.4	5344.07	112.50	798.55	70.98
	0.5	6671.03	158.33	849.79	73.53

4%	0.2	2246.39	25.00	130.82	11.32
	0.3	3398.16	58.30	729.90	63.13
	0.4	4505.20	100.00	816.37	70.60
	0.5	5648.93	200.00	893.40	77.27

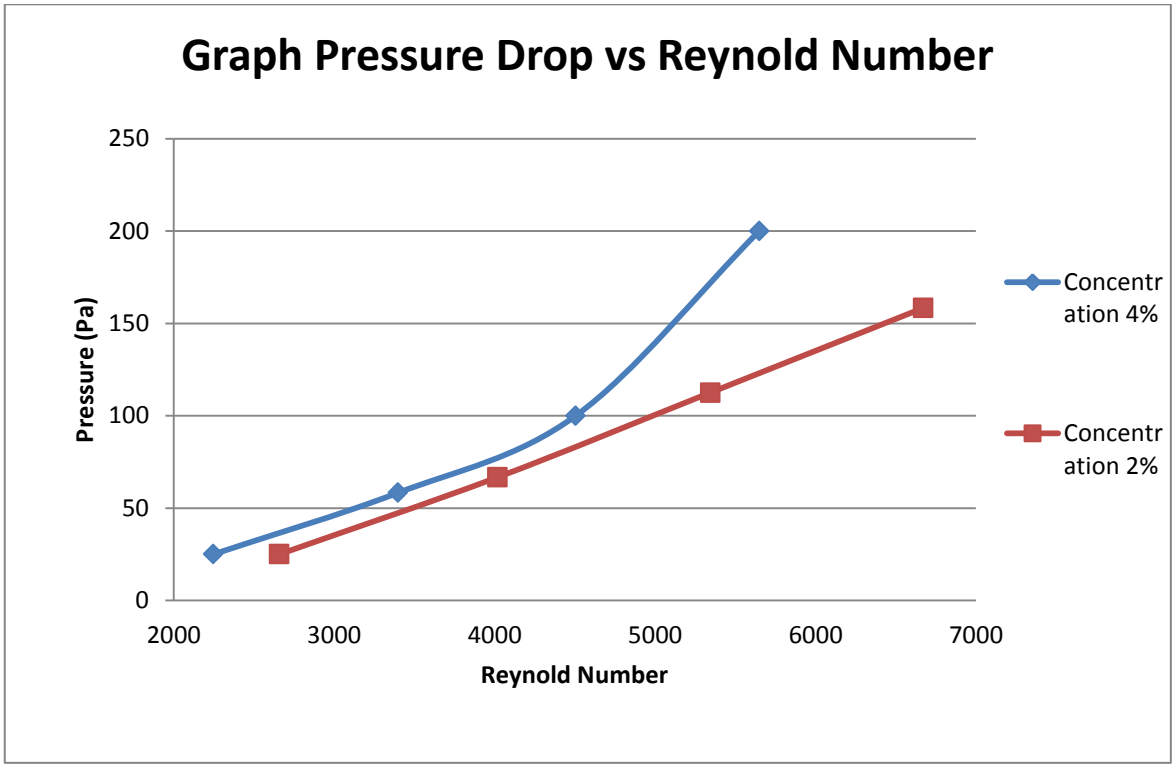

Figure 14: Graph Pressure Drop versus Reynold number for different concentration of aluminum oxide in water

The figure 14 shows the pressure drop versus different Reynold number and different concentration of aluminum oxide with water. From the graph, it shows the concentration of 4% aluminum oxide is higher than concentration of 2% aluminum oxide. Besides, when 4% concentration of aluminum oxide achieves *Re* = 4500, the rate of increasing pressure drop is higher than concentration of 2% aluminum oxide. The 2% concentration of aluminum oxide shows a constant increase of pressure drop over the Reynold number changes.

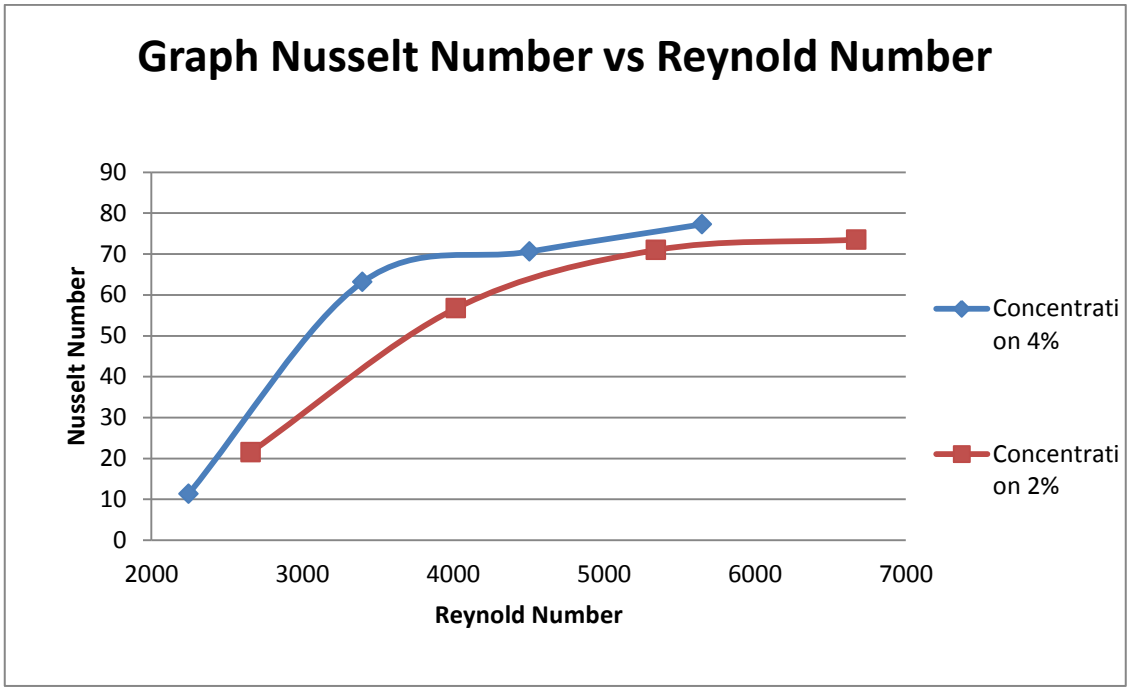

Figure 15: Graph Nusselt Number versus Reynold number for different concentration aluminum oxide in water

The figure 15 shows the Nusselt number versus Reynold number for different concentration of aluminium oxide with water. The 4% concentration of aluminium oxide shows the higher heat transfer coefficient than 2% concentration of aluminium oxide. The 4% concentration of aluminium oxide comes to the almost steady increase of heat transfer coefficient from *Re* = 3800 and 2% concentration of aluminium oxide is from *Re* = 5000.

4.3 CALCULATION RESULTS

4.3.1 Calculation for air

Table 7: Calculated Nusselt Number of air

Velocity, *m/s*	**Reynold Number**	**Simulated Nusselt Number**	**BEEK Nusselt Number**	**UPADHYAY Nusselt Number**
0.2	67.32	14.58	6.26	7.30422

0.3	110.13	18.54	9.05	11.94911
0.4	153.16	35.08	11.63	16.61786
0.5	197.61	60.99	14.13	21.44069

4.3.2 Calculation for water

Table 8: Calculated pressure drop of water

Velocity, m/s	**Reynold Number, Re**	**Simulated pressure drop, *Pa***	**ERGUN pressure drop, *Pa***	**BEEK pressure drop, *Pa***	**ERGUN friction factor**	**BEEK friction factor**
0.2	1235	50	34.1234	93.01578	1.871	5.100093
0.3	1874	100	75.0952	207.0979	1.830	5.046783
0.4	2514	200	132.0435	366.2605	1.810	5.020553
0.5	3121	400	204.9502	570.5794	1.798	5.005616

4.3.3 Differences between calculated values and the simulation result values

In the calculation, the value of pressure drops from ERGUN and BEEK equations in different velocities are increasing higher and faster than the simulation result values. It is because the ERGUN and BEEK equations are for a particle research and will not be same with situations for the bed packed. The simulation results are based on the bed packed and not for a single particle system. But the trend for the calculated values and simulations value are same that is increasing almost constantly.
The same situation comes with the calculated Nusselt number using BEEK and UPADHYAY equations with the result values of the simulation.

4.3.4 Calculation of nanofluid properties, Aluminum Oxide (Al_2O_3) + Water

4.3.4.1 Properties of Nanomaterial

Table 9: Nanomaterial properties

Nanoparticle	**Size, *m***	**Thermal Conductivity, *W/m. K***	**Density, *kg/m^3***	**Specific heat, *J/ kg. K***
Al_2O_3	30nm	36	3880	773

Source: Pak and Cho, 1998

4.3.4.2 Properties of Water

Table 10: Water properties

Fluid	**Thermal Conductivity, *W/m. K***	**Density, *kg/m^3***	**Specific heat, *J/ kg. K***	**Viscosity, *kg/ms***
Water	0.663	977.66	4189.13	0.00040657

4.3.4.3 Properties of Nanofluid

Table 11: Different concentration of aluminum oxide in water properties

Concentration	**Thermal Conductivity, *W/m. K***	**Density, *kg/m^3***	**Specific heat, *J/ kg. K***	**Viscosity, *kg/ms***
2%	0.80659	1036.5	3930.13	0.000490
4%	0.82895	1095	3699	0.000611

4.4 Comparison of numerical experiment with relevant research

For air, by comparing the air from the simulation with the figure 3 from the journal, Heat and Flow Characteristics of Packed Bed (Achenbach. E, 1995) the Nusselt number versus Reynold number for both graphs has the same trend. The difference in the value is because of the different sizes of the particle.

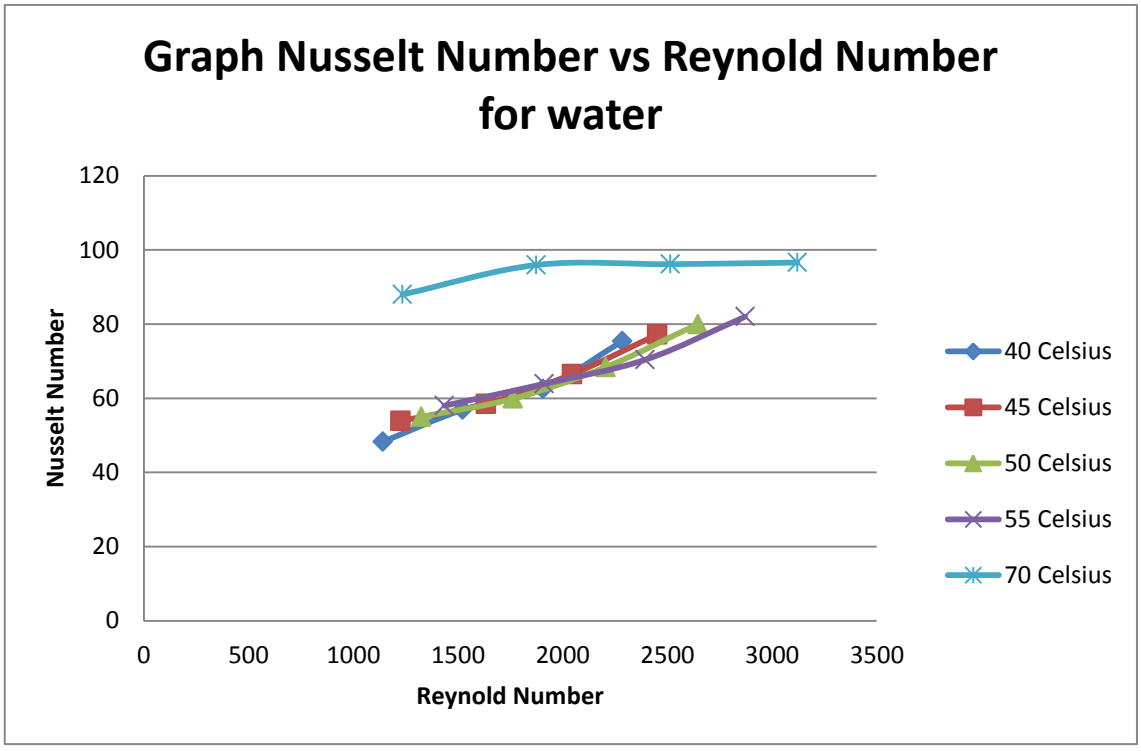

Figure 16: Graph Nusselt Number versus Reynold Number for different temperature conducted.

Figure 16 illustrates the Nusselt Number versus Reynold Number in water as fluid recirculation. In figure 16, from 40 Celsius graph to 55 Celsius graph is from the Journal "Experimental Study on Heat Transfer Coefficient and Friction Factor of Al_2O_3 Nanofluid in Packed Bed Column" (Rao.G.S et al, 2011). The 70 Celsius graph is from the simulated results. The simulated result is compared with 50 Celsius because the average heat in the system is 50 degree Celsius (summation of fluid temperature, 70 Celsius and wall temperature, 30 Celsius divide by 2). Others temperature of the reference is compared with the 50 Celsius of the same reference. Based on figure 16, from 40 Celsius to 55 Celsius graphs are increasing with velocity increases and the increasing rate is beginning to constant. The numerical experimental graph that is 70 Celsius is increasing but almost constantly increasing. The Nusselt number values of simulated results are significantly higher than the experimental journal because of the different sizes of the particles of packed bed and the flow rate. The flow rate of the experimental journal is from 150 to 300 liters per hour meanwhile the simulated flow rate is just 35 to 87 liters per hour. Therefore, the numerical experiment values can be accepted by the experiment.

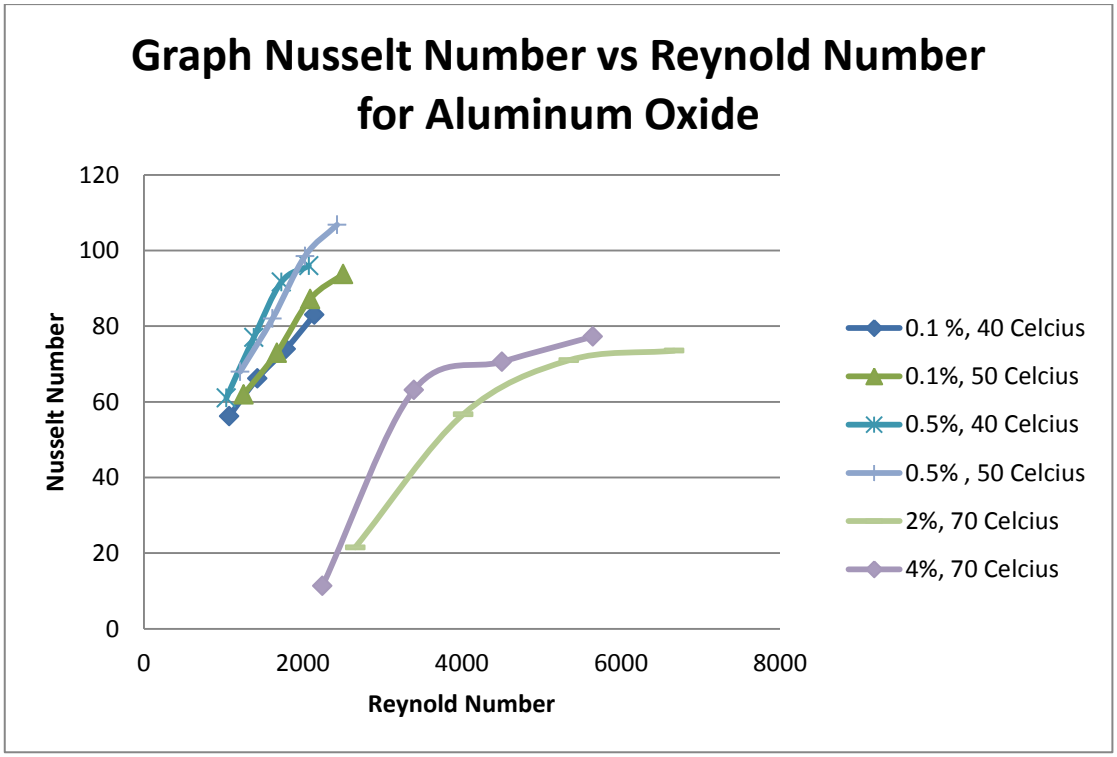

Figure 17: Graph Nusselt Number versus Reynold Number for different concentration of Aluminum Oxide and different temperature conducted.

Figure 17 shows the graph of Nusselt Number versus Reynold Number for different concentration of Aluminum Oxide and different temperature conducted in nanofluid as fluid recirculation. From the figure 17, all the 40 Celsius graph to 50 Celsius graph is from the Journal "Experimental Study on Heat Transfer Coefficient and Friction Factor of Al_2O_3 Nanofluid in Packed Bed Column" by G. Srinivasa Rao, K.V. Sharma, S.P. Chary, R.A.Bakar, M.M. Rahman, K. Kadrgama and M.M. Noor. The 70 Celsius graph is from the simulated results.

Based on figure 17, the increasing of concentration of aluminium oxide will decrease the heat transfer coefficient and increase the Reynold number. Both from the previous researcher's graph value and simulated graph value are seems to have the same trend of the graph that is formed of the curve with linear increasing at the beginning and almost constantly increasing when reach the end of the graph. Hence, the numerical experimental values can be accepted in the experiment.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATION

5.1 CONCLUSIONS

From the numerical experiment results for nanofluid shows the same trend of the graph with the experimental results with the journal of "Experimental Study on Heat Transfer Coefficient and Friction Factor of Al_2O_3 Nanofluid in Packed Bed Column". The trend of the Nusselt number graph should have linearly increasing at the beginning and constantly increasing when reaches the end of the graph for the nanofluid.

Basically, all the numerical experiment graphs show the same trend with the pass researcher's results. The same trend of results does not mean the results of this simulation are correct but it shows that the way to perform the numerical experiment is correct. Different values of heat transfer coefficient and pressure drop is due to the size of the model and the technique of getting the results such as for this simulation is based on bed packed and for some researcher is based on a particle.

CFD proves to be useful estimating the heat transfer coefficient between the particle and the fluid and also for calculation of pressure drop along the bed in packed bed equipment. It was possible to model a realistic case of a packed bed of spheres including contact points within the surfaces involved in the geometry.

5.2 RECOMMENDATION

The recommendation is suggested to further studies of CFD simulations by using different packing material instead of aluminium. With the same boundary condition of the model, determination of heat transfer coefficient and pressure drop can be applied and comparing with this aluminium results.

REFERENCES

Achenbach. E, 1995. E. Achenbach Heat and Flow Characteristics of Packed Beds. Elsevier B.V company

Bird, R. Byron, 1996. Transport Phenomena. Madison, Wisconsin: John Wiley & Sons

Geankoplis, 2003. Christie J., Transport Process and Unit Operations. 4th ed., New Jersey. Prentice Hall

Guardo et al, 2005. Influence of the turbulence model in CFD modeling of wall-to-fluid heat transfer in packed beds

Guardo et al, 2006. CFD study on particle-to-fluid heat transfer in fixed bed reactors: convective heat transfer at low and high pressure

Kravchenko, V.S. 1966. Empirical Equation Derived of Density Of Heavy Water for Temperature Dependence. *Atomnaya Energiya*

Nijemeisland, 2000. M.Sc. Thesis. Worcester Polytechnic Institute. Worcester, MA, USA.

Pak, B.C. & Cho, Y.I. 1998. Hydrodynamic and heat transfer study of dispersed fluids with submicron metallic oxide particles. Experimental Heat Transfer

Rao G.S et al, 2011. Experimental Study on Heat Transfer Coefficient and Friction Factor of Al_2O_3 Nanofluid in Packed Bed Column. Journal of Mechanical Engineering and Sciences (JMES). FKM, University Malaysia Pahang

Stüber et al, 1996. Supercritical fluid extraction of packed beds: external mass transfer in upflow and downflow operation

Wakao, 1976. Particle-to-fluid transfer coefficients and fluid diffusivities at low flow rate in packed beds

Wakao and Kaguei, 1982. Heat and Mass Transfer in Packed Beds. McGraw-Hill, New York

APPENDIX A

Sample of calculations for water pressure drop coefficient

Using ERGUN equation, Velocity = 0.2*m/s*

Friction factor, f_p

$f_p = \frac{150}{\mathrm{Re}} + 1.75$; for 1 < Re < 2500

$f_p = \frac{150}{1235} + 1.75 = 1.871$

Pressure drop, *ΔP*,

$$\Delta P = 1.871 \times \frac{0.07169153\mathrm{m}}{0.07848\mathrm{m}} \times \frac{\frac{998.2\mathrm{kg}}{\mathrm{m}^3} \times (\frac{0.2\mathrm{m}}{\mathrm{s}})^2}{2}$$

$= 34.1$

Using BEEK equation, velocity = 0.2*m/s*

Friction factor, f_p

$f_p = \frac{1-\varepsilon}{\varepsilon^3} \times (1.75 + 150 \times \frac{1-\varepsilon}{Re})$;

$f_p = \frac{1-0.054435}{0.054435^3} \times (1.75 + 150 \times \frac{1-0.054435}{1235})$

$= 5.100093$

Pressure drop, ΔP

$$\Delta P = 5.100093 \times \frac{0.07169153\mathrm{m}}{0.07848\mathrm{m}} \times \frac{\frac{998.2kg}{m^3} \times (\frac{0.2m}{s})^2}{2}$$

$= 93.01578$

Sample of calculation for air Nusselt number,

Using BEEK equation, velocity = 0.3*m/s*

Nusselt number, Nu_1

$Nu_1 = 0.203 \times \mathrm{Re}^{0.33} \times Pr^{0.33} + 0.22 \times Re^{0.8} \times Pr^{0.4}$;

$Nu_1 = 0.203 \times 110.13^{0.33} \times 0.7^{0.33} + 0.22 \times 110.13^{0.8} \times 0.7^{0.4}$

$= 9.05$

Using UPADHYAY equation, velocity = 0.3*m/s*

Stanton number,

$St = \frac{1}{\varepsilon} \times 0.455 \times Re^{-0.4}$;

$St = \frac{1}{0.54435} \times 0.455 \times 110.13^{-0.4}$

$= 40.3$

Nusselt Number, Nu_2

$Nu_2 = St \times Re \times Pr$

$Nu_2 = 40.39 \times 110.13 \times 0.7$

$= 11.95$

Sample of calculation for nanofluid properties: aluminum oxide (4%) + water

Specific Heat;

$C_{nf} = \phi C_p + (1-\phi) C_w$

$C_{nanofluid}$ = 0.04 (773 *J/kg.K*) + (1 – 0.04) (4189.13 *J/kg.K*)

= 3699 *J/kg.K*

Density;

$\rho_{nf} = \phi \rho_p + (1-\phi) \rho_w$

$\rho_{nanofluid}$ = 0.04 (3880 kg/m^3) + (1 – 0.04) (977.66 kg/m^3)

= 1095 kg/m^3

Thermal Conductivity;

$$K_{nf} = K_w \left[0.8938 \left(1+\frac{\varphi}{100}\right)^{1.37} \left(1+\frac{T_{nf}}{70}\right)^{0.2777} \left(1+\frac{d_p}{150}\right)^{-0.0336} \left(\frac{\alpha_p}{\alpha_w}\right)^{0.01737} \right]$$

$K_{nanofluid}$ = 0.663*W/m.K* [0.8938 $(1 + 0.04)^{1.37}$ $(1 + \frac{70}{70})^{0.2777}$ $(1+ \frac{30}{150})^{-0.0336}$ $(\frac{1.2 \times 10^{-5}}{1.619 \times 10^{-7}})^{0.01737}$]

= 0.8289 *W/m.K*

Viscosity;

$$\frac{\mu_{nf}}{\mu_w} = C_1 \left\{ \left(1+\frac{\varphi}{100}\right)^{11.3} \left(1+\frac{T_{nf}}{70}\right)^{-0.038} \left(1+\frac{d_p}{170}\right)^{-0.061} \right\}$$

$C_1 = 1.0$ for metal and metal oxide nanofluids in base liquid

$\mu_{nanofluid}$ = 0.00040657 *kg/ms* [$(1 + 0.04)^{11.3}$ $(1 + \frac{70}{70})^{-0.038}$ $(1 + \frac{30}{170})^{-0.0061}$]

= 0.000611*kg/ms*

APPENDIX B

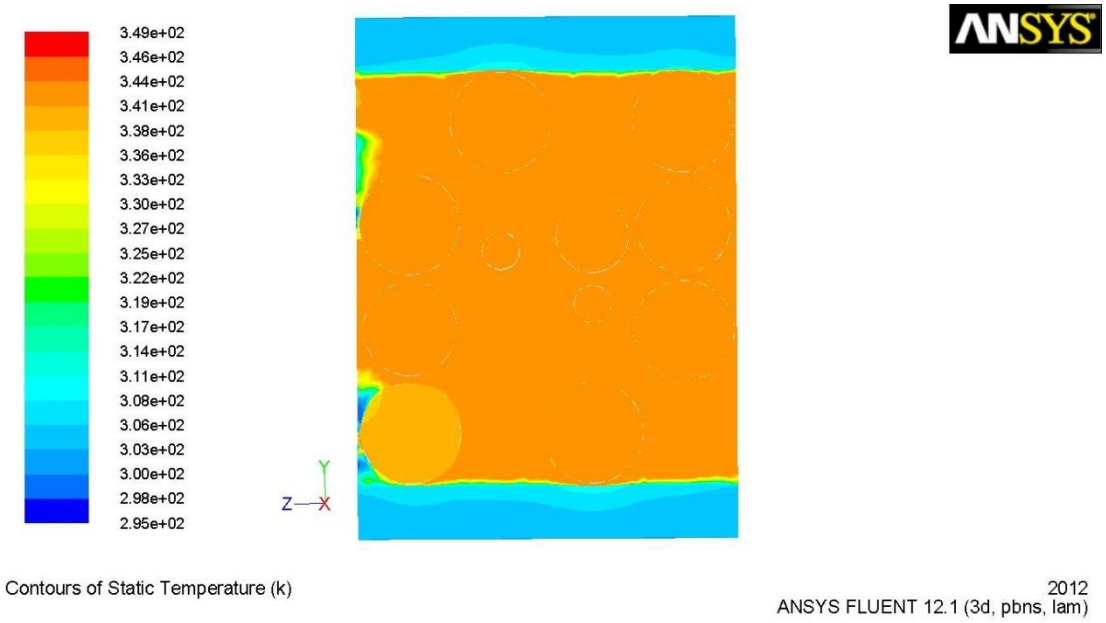

Figure 18: Temperature contour for water at Re = 2514

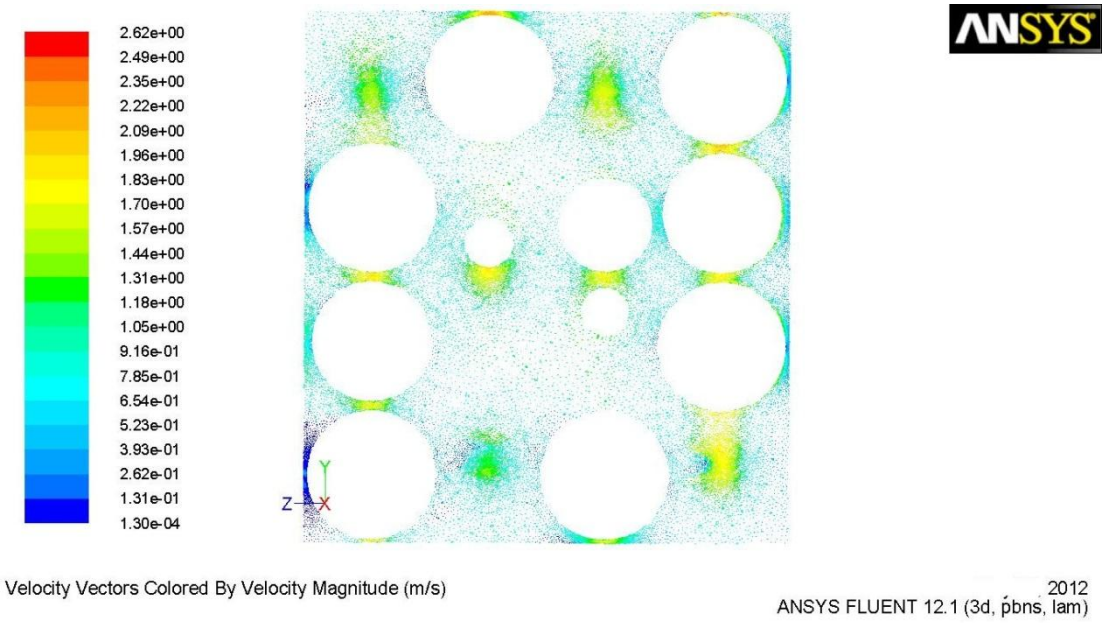

Figure 19: Velocity vector of water at Re = 2514

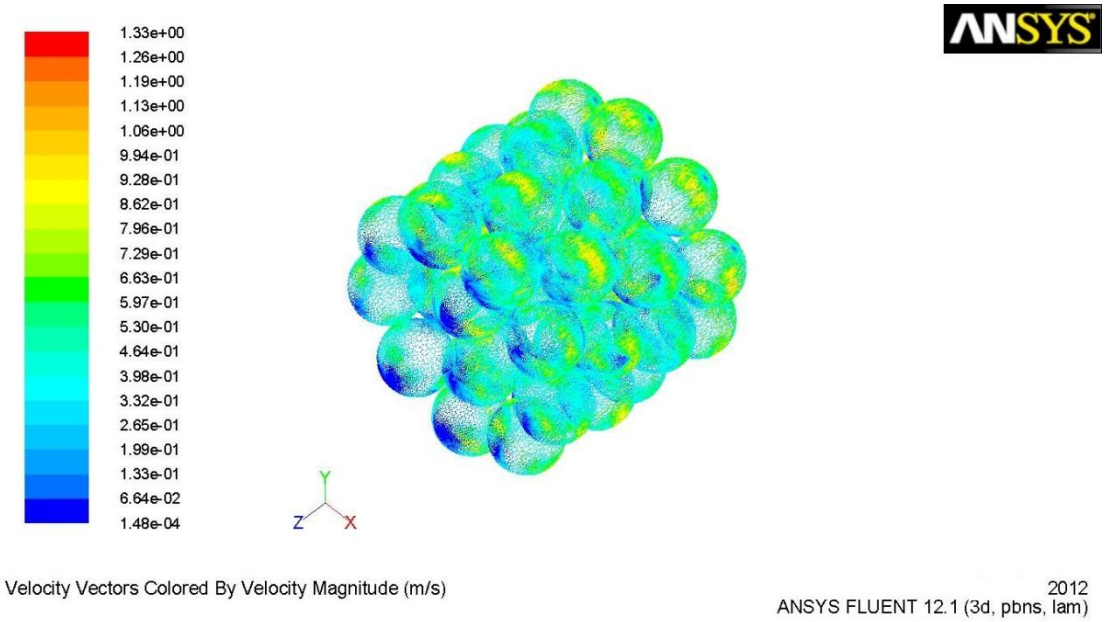

Figure 20: Velocity vector of water at Re = 2514, near the wall surface.

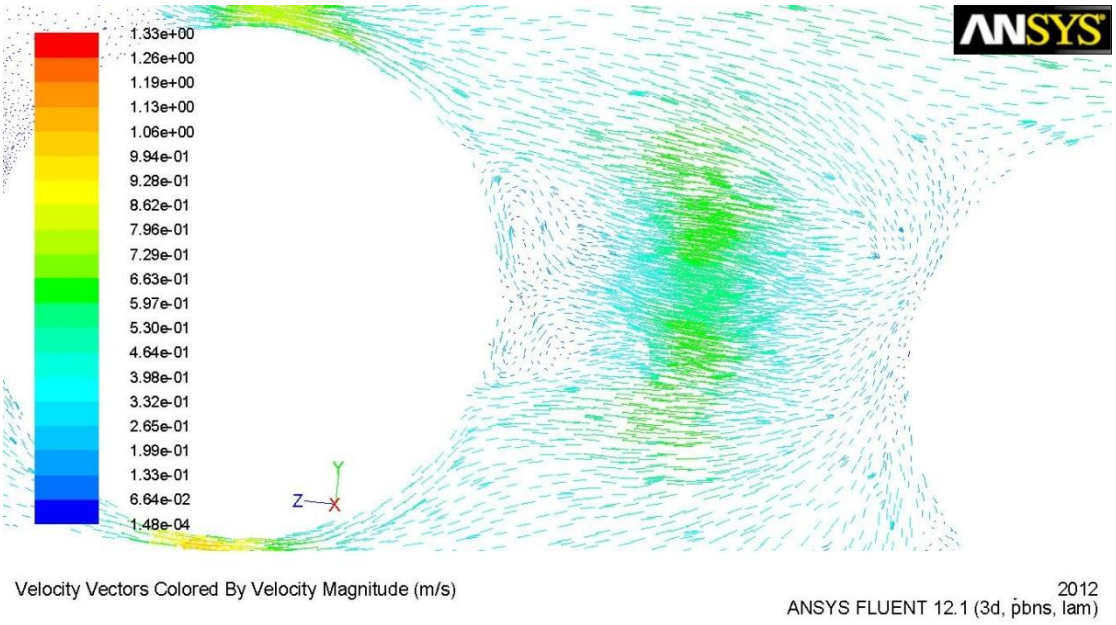

Figure 21: Wake region for water at Re = 1235

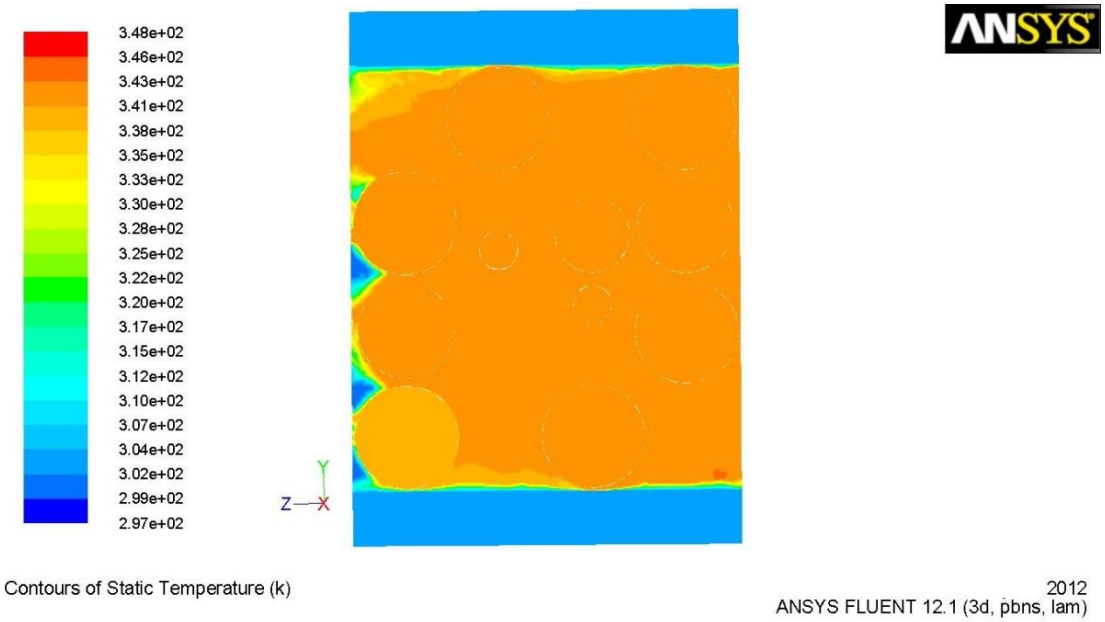

Figure 22: Temperature contour for air at Re = 153.16

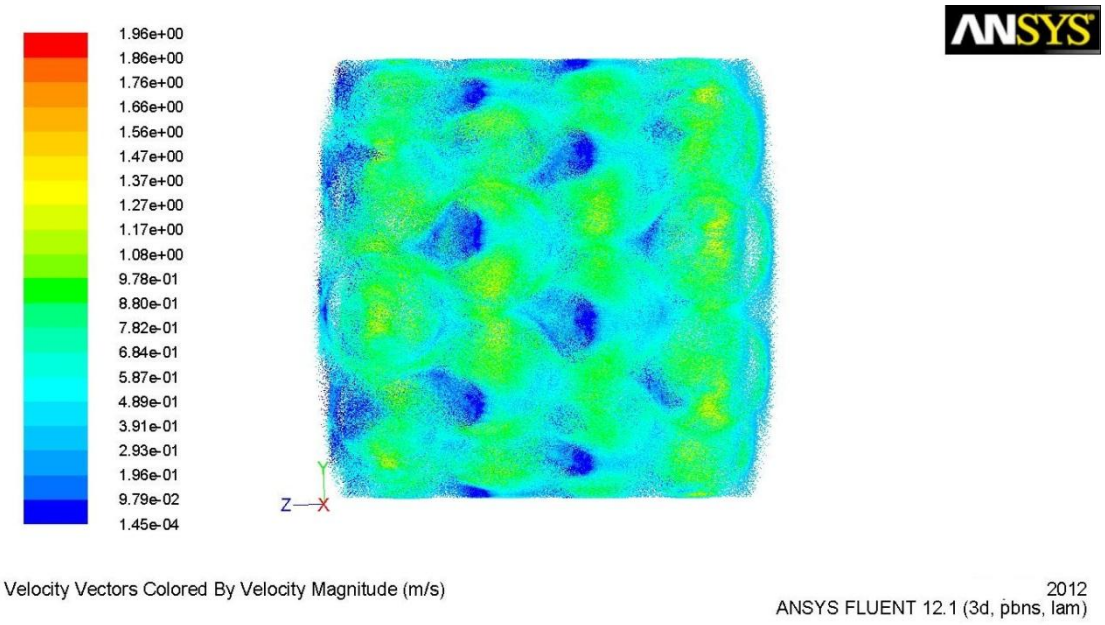

Figure 23: Velocity vector for air at Re = 110.13, near the wall surface

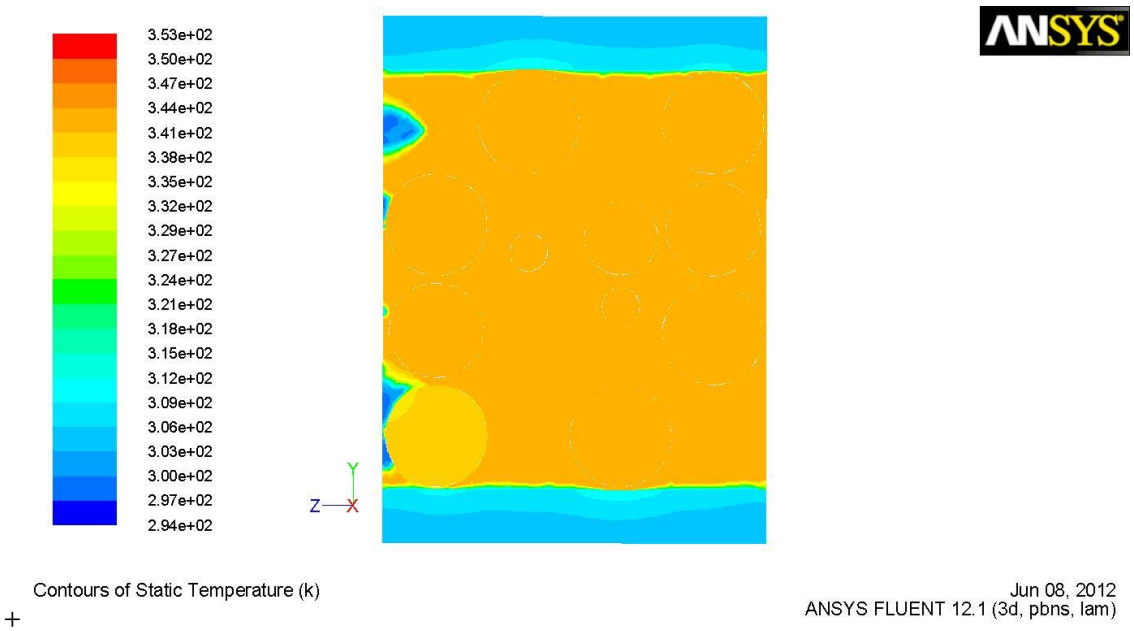

Figure 24: Temperature contour for 2% concentration of aluminum oxide in water at Re = 2658

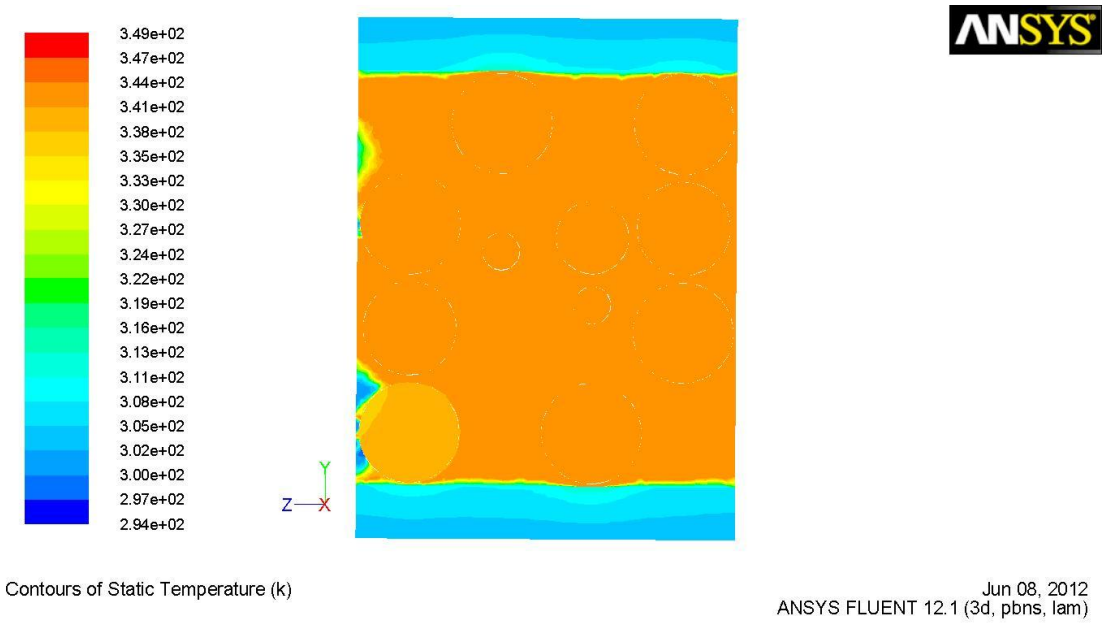

Figure 25: Temperature contour for 4% concentration of aluminum oxide in water at Re = 4505