CERTIFICATION OF APPROVAL

CFD Multiphase Modeling of Fluidized Bed using FLUENT© Software

by

Abd Rasyid Bin Hamzah

A project dissertation submitted to the
Chemical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(CHEMICAL ENGINEERING)

Approved by:

....

(DR KU ZILATI KU SHAARI)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
June 2009

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

A. Rasyid

____

(ABD RASYID B HAMZAH)

ABSTRACT

Basically this report discusses the preliminary research done and surface understanding about the chosen topic, **CFD Multiphase Modeling of Fluidized Bed using FLUENT© Software**. A fluidized bed is formed when a bulk quantity of a solid is forced to behave like a fluid; usually by the forced introduction of pressurized fluid, often a gas through the particulate medium. For this project, spouted bed was chosen as the main fluidized bed. The spouted bed is simulated using FLUENT software. Fluent basically is a computational fluid dynamics (CFD) computer program designed for modeling fluid flow and heat transfer in complex geometries. Simulation of fluidized bed is important in order to understand the particle behavior and effect of heat and mass transfer in the fluidized bed because in real working plant, the real fluidized bed is hard to be observed. A few variables were tested during this simulation: different in density of granule, air velocity, diameter size of granule and total simulation time. The conditions applied for this spouted bed are 3 m/s of air velocity, operated at 300.15 K of temperature and 101325 Pa of pressure. The 2-D spouted bed was modeled first in GAMBIT and all of the boundary condition required was selected and meshed. Then the meshed spouted bed was exported into FLUENT using double precision calculation to simulate the problem. As a conclusion, density of 750 kg/m^3 was selected to be the applicable density to be used because it formed a good fluidized bed among the others. Meanwhile when using 2000 kg/m^3 of granule density, granule diameter size of 0.0003 m was chosen since that size of granule gave a balance result, which is fluidized bed was formed at the middle level of spouted bed height. Longer iteration time does not represent more accurate data was produced. This was proven when by using 0.5 s of total simulation time was sufficient enough to get the required result.

ACKNOWLEDGEMENTS

Firstly, I would like to thank God for allowing me to finish this research successfully and provide me with extra strength and creative ideas. Next I would like to express my sincere appreciation and thankfulness to my Final Year Project Supervisor, Dr Ku Zilati Bt Ku Shaari for her inspirational guidance, motivational talk and supervised me in the way it should be throughout the course in order to complete the research and achieved my objective. She also wills to spend her precious time to answer my question although she was busy at that particular time.

I also would like to express my appreciation to postgraduate students, Nurul Huda Yunus and Nurul Huda Zailani for teaching, guiding, and help me to solve some problem related with FLUENT and GAMBIT software. Your cooperation is highly appreciated.

Next, to my parents who always gave their best supports and advice to me whenever I have problem, thank you for everything. Without you, all of this would not possible to achieve and your blessing in every way was what kept me going even when things were bleak.

Finally, to those who have been helping me indirectly or directly, thank you very much.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 BACKGROUND OF STUDY

A fluidized bed is formed when a bulk quantity of a solid is forced to behave like a fluid; usually by the forced introduction of pressurized fluid, often a gas through the particulate medium. This will result in the medium having new properties and characteristics such like normal fluids; such as ability to free-flow under gravity, or to be pumped using fluid type technologies, as the definition from GEA Process Engineering Inc. website [1]. Presents of pressurized fluid reduces density of the medium; without affecting its elemental nature. The resulting phenomenon is called fluidization. For this research paper, gas-solid fluidized bed is used as the main fluidized bed.

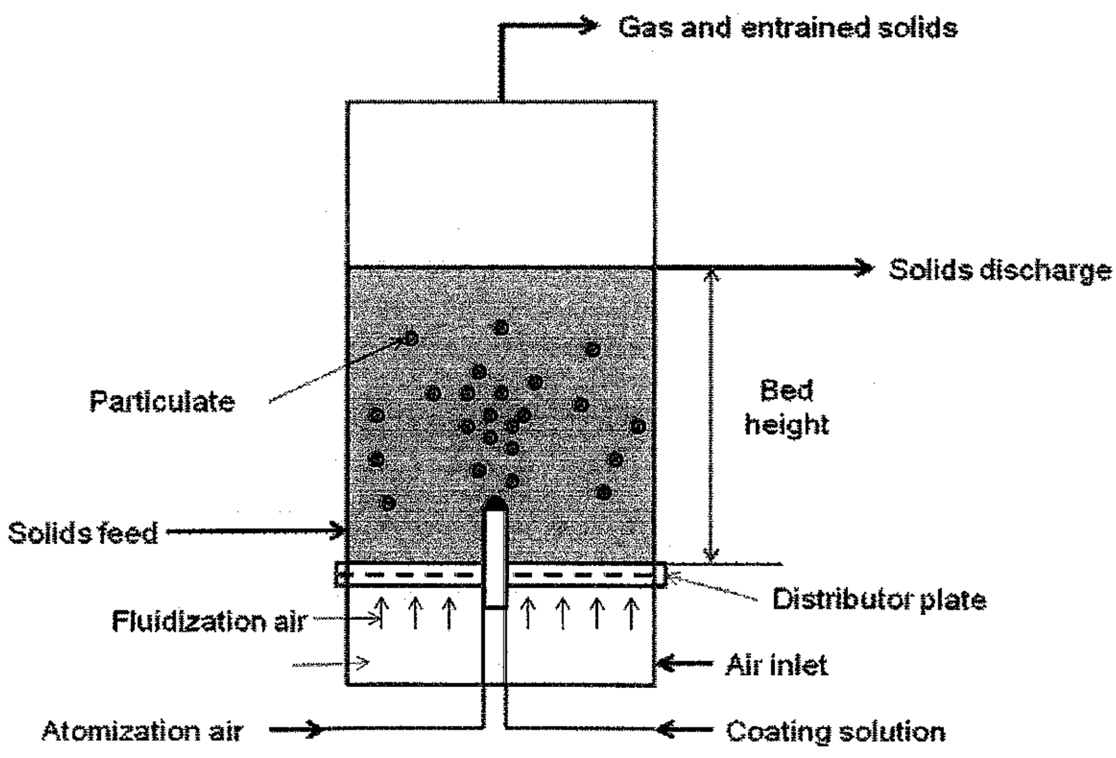


Figure 1.1 : Description of general fluidized bed

There are numerous advantages of fluidized beds, for example, Kari Myöhänen et al. [2] states that fluidization of particles enables controlled handling of solids; good solids mixing and large thermal inertia of solids produce nearly isothermal conditions; heat transfer from suspension to heat exchanger surfaces is enhanced by the solids; heat flow and reaction rates between gas and solids are high due to large gas particle contact area; constant movement of particles and the large interparticle forces enable operation close to melting temperature of solids. Therefore, fluidized beds are used for catalytic cracking, combustion and various other chemical and metallurgical processes.

1.0.1 Types of Fluidized Bed

a) Slugging bed is a fluid bed in which air bubbles occupy entire cross sections of the vessel and divide the bed into layers.
b) Boiling bed is a fluid bed in which the air or gas bubbles are approximately the same size as the solid particles.
c) Channeling bed is a fluid bed in which the air (or gas) forms channels in the bed through which most of the air passes.
d) Spouting bed is a fluid bed in which the air forms a single opening through which some particles flow and fall to the outside.

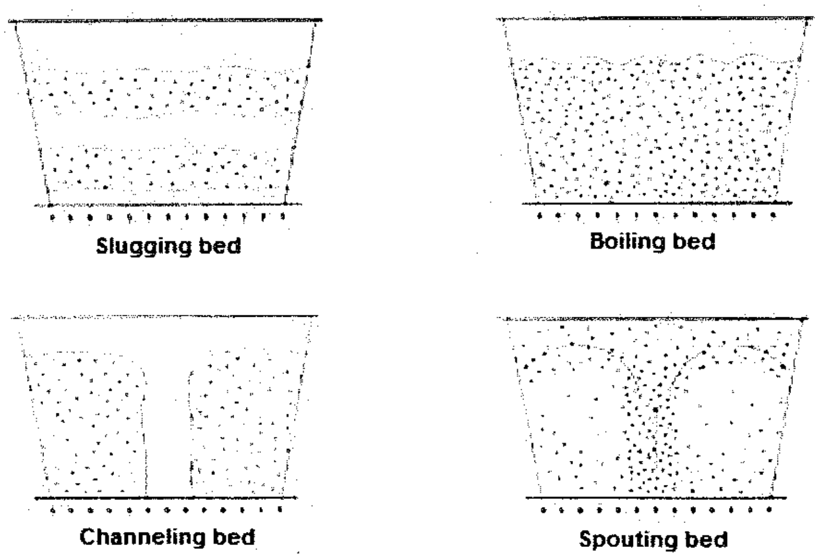


Figure 1.2: Different types of fluidized beds

1.0.2 Fluidized Beds Applications

There are many types of fluidized beds present in industrial plant; among of them are fluid catalytic cracking and circulating fluidized bed combustion.

Kari Myöhänen et al. [2] stated that the fluid catalytic cracking (FCC) is used for converting crude oil into a variety of higher-value light products. For this FCC, hot catalyst (650 – 700 °C) flows from the regenerator to the bottom of the riser and meets the liquid feed, which is vaporized. FCC particles used mean size of about 40 – 80 μm and particle density 1100 – 1700 kg/m^3, representing Geldart group A. Inside the riser, the feed vapor is cracked by the catalyst. Fluidization velocity in the riser is 6 – 28 m/s, increasing towards top due to molar expansion. The net solids flow is 400 – 1200 kg/m^{2}s. At the top of the riser, the catalyst is separated from reaction products and returned back to regenerator.

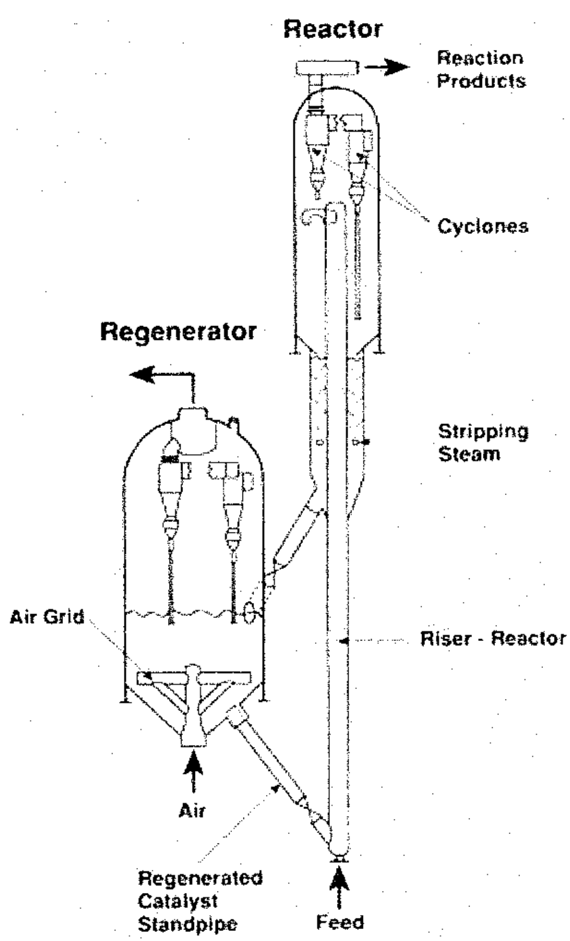


Figure 1.3 : Side-by-side FCC Unit (King, 1992)

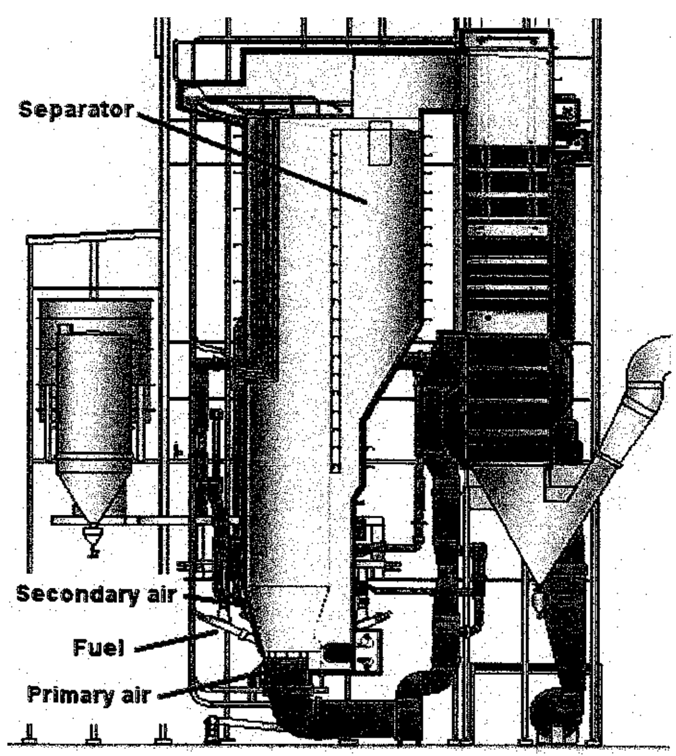


Figure 1.4 : CFB boiler (EC Chorzom Elcho, adapted from Foster Wheeler reference material).

In a Circulating Fluidized Bed Boiler (CFB) (Figure 1.4), it use fluidization velocity at about 5 m/s at full load. The solids are coarser and denser than in FCC, particle size 100 – 300 μm and particle density 1800 – 2600 kg/m^3 (Geldart group B). The fuels used in CFB combustion include coal, oil shale, petroleum coke, lignite, wood, biomass and different wastes. Typical furnace temperatures are 800 – 900 °C and net solids flow in furnace 10 – 100 kg/m^2s.

For this research, spouted bed was chosen as the main fluidized bed because less research was done on spouted bed compared to other types of fluidized bed and the implementation of fluidized bed in heat disinfection of grain shows the most cost efficient compared to typical Fluidized Bed and In-Bin systems [27].

1.0.3 What is Spouted Bed?

Duarte et al. [3] found that spouted bed is gas-particle contactors in which the gas is introduced through a single nozzle at the center of a conical or flat base of spouted beds. Above such a spouted bed the particles drop down in a fountain and move down the tube in an annular region, as from Gidaspow [4] findings.

Olazar et al. [5] showed that spouted bed using conical shapes has the properties of cylindrical conical base spouted beds, while also allowing for strong gas-solid contact since they operate stably in a wide range of gas flow rates. A high-velocity spout of gas penetrates the bed and carries the particle upward. The other sections of the charge move downwards at slow speed so that uniform circulation of particles is obtained.

Conical spouted bed is suitable for operating in a state of transition between the high voidage spouted beds and diluted spouted bed regimes. Usually it is use in application which requires handling of solids that cannot be handled in conventional spouted beds, such as solids with a wide particle distribution [6] or of irregular texture [7].

1.2 PROBLEM STATEMENT

The used of fluidized bed in the industrial plant has been increased over the past few years. Among of the fluidized bed function is for converting crude oil into a variety of higher-value light products, used as boiler and as a granulator to produced urea granule.

Among of the current problem are the fundamental research on fluidized bed is not parallel with industrial development [13] and most researchers lack of understanding about heat and mass transfer which is occurred inside the fluidized bed. Proper study about heat and mass transfer in spouted bed using related multiphase model must be done in order to understand what actually is happening inside fluidized bed by simulating the situation using Computational Fluid Dynamic (CFD) Software, such as for this case, using FLUENT© Software.

1.3 OBJECTIVE AND SCOPE OF STUDY

The objective of this research is to study on heat and mass transfer of solid-gas particles behavior in fluidized bed by modeling the fluidized bed using FLUENT© software. In order to achieve the objective, two problems will be studied, which are heat transfer and mass transfer involved in fluidized bed. Four variables will be tested to achieved the objective, that are velocity of air inlet, diameter size of granule, total simulation time and density of granule.

The scope of study for this research project are to do literature review of theory on Fluidized Bed, the theory behind Eulerian Multiphase Model in Fluent, and use FLUENT Software to simulate the heat and mass transfer behavior in Fluidized Bed.

CHAPTER 2
LITERATURE REVIEW

2.1 FLUIDIZED BED

The fluidization phenomenon can be visualized in an experiment as shown in Figure 2.1. Jacob [14] stated that a fluidized bed forms when a gas passes through the bed material upwards from below. The bed started to be fluidized at a certain velocity of the gas, which referred to as the minimum fluidization states (Figure 2.1, left). If the velocity of gas further increases, the particles will move faster and much of the gas in excess of the required amount for minimum fluidization passes through the bed material in form of ascending gas bubbles (Figure 2.1, center). Due to flow characteristics, the particles are mixed intensely and conditions are very good for many material and heat transfer processes. Following the bubble formation and movement, the particles below the bubbles move into the upper zones of the fluidized bed whereas they return downwards towards the gas inlet in other areas (Figure 2.1. right). Particles such as sand are poured into a tube equipped with a porous plate distributor, which is for a good quality of fluidization as according to Gidaspow [4], has a pressure drop of 10 to 20 percent of the weight of the bed.

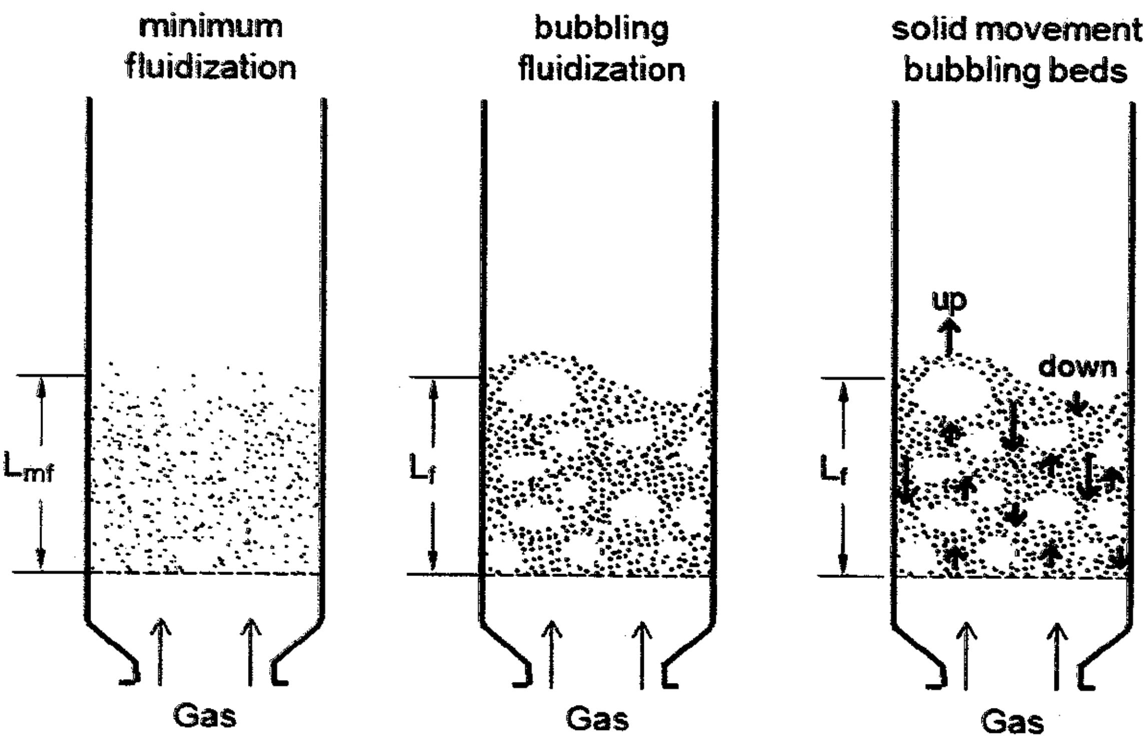


Figure 2.1 Fluidized bed principles

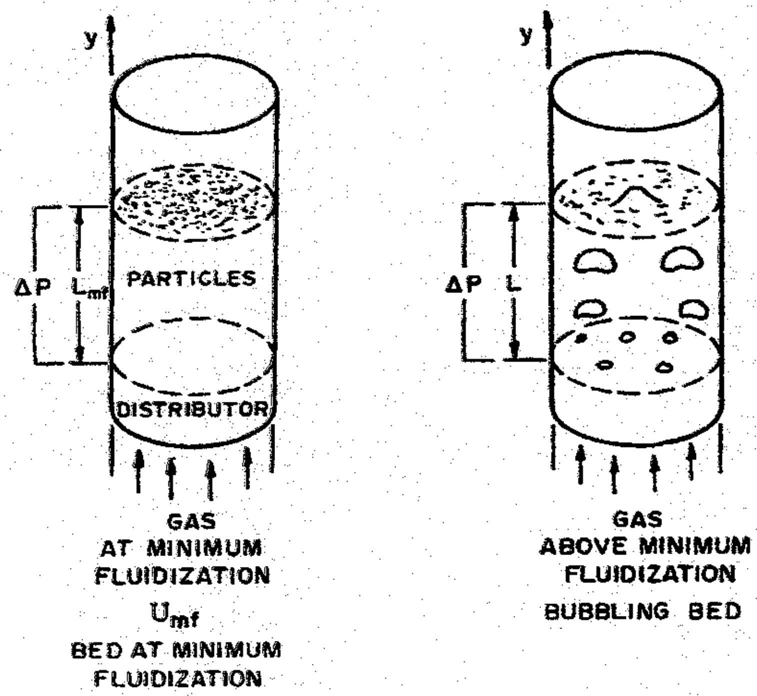


Figure 2.2: Pressure drop of fluidization bed

For a narrow tubes and a sufficiently deep bed, the bubbles coalesce and form a slug. These gas slugs then move up the bed in a fairly regular periodic motion. Large-size particles such as corn can be made to spout if the gas is brought in through a small central tube. When gas velocity is increased substantially above the terminal velocity, the fluidized bed, slugging bed, and spouted bed are all blown out of the tube. If the operation in the dense-phase is continuous, it is rapidly feeding solids into the bottom of the bed. The resulting flow regime is called recirculating fluidization, such as in Figure 2.3.

Research about heat transfer in fluidized beds has been started in the past. Numerical simulation becomes an additional tool to analyze the fluid dynamics and the heat transfer mechanism in multiphase flow. A few years back, the fluid dynamics of a bubbling fluidized bed has been calculated with an Eulerian approach, as according to Kuipers findings [15]. Their numerical setup consists of a two-dimensional isothermal fluidized bed filled with glass beads. On the basis of the conservation equations for both phases it is possible to predict instantaneous particle volume fractions, velocity distributions as well as the pressure field.

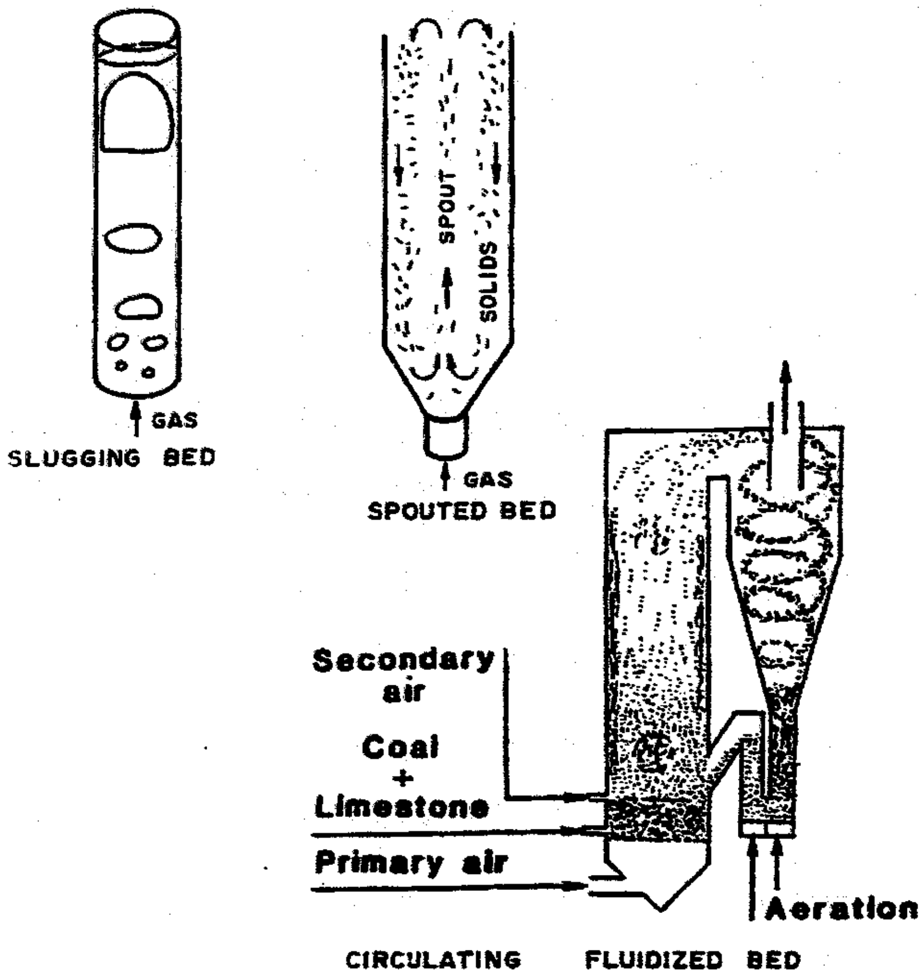


Figure 2.3 : Fluidization regime

2.2 MODELING OF SPOUTED BED

There are a few different approaches and methods for modeling of spouted bed. Based on M. Jacobs [14] study, a common method of calculation is using Computational Fluid Dynamic (CFD) modeling based on the finite volume method. Another method is Discrete Element Method (DEM) modeling, which is very good for the simulation of fluidized and spouted bed processes. DEM can do direct computation of particle to particle contacts and trajectories which allows more accurate simulations and for instance bubble shapes and particle flows through bubbles are better predicted, as according to Haidar et al. [16]. But DEM have a few disadvantages; very high requirement of computational power and limitations in the total number of particles which is currently limited to the order of 1 million.

The studies discussed here were made with 2-dimensional CFD simulations using the Euler-Euler approach. The calculations were calculated using CFD commercial software, FLUENT 6.3.26 meanwhile the computational mesh required is done using Gambit 2.2.30 software.

2.3 FLUENT© SOFTWARE

Nowadays, most company which used fluid in their application simulates their ideas in the computer first in order to optimize the productions rate. Simulation is used to fulfill the desire so that the fluid can be modeled and manufactured the equipment as precise as they can.

FLUENT software is among of the most software used in industrial field for modeling fluid flow and heat transfer in complex geometries. It has complete mesh flexibility in order to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. FLUENT supported various mesh types, including 2D triangular/quadrilateral, 3D tetrahedral / hexahedral / pyramid / wedge / polyhedral, and mixed (hybrid) meshes.

2.4 GAMBIT SOFTWARE

GAMBIT is FLUENT's geometry and mesh generation software. GAMBIT's single interface for geometry creation and meshing brings together most of FLUENT's pre-processing technologies in one environment. Advanced tools for journaling let user to edit and conveniently replay model building sessions for parametric studies. GAMBIT's combination of CAD interoperability, geometry cleanup, decomposition and meshing tools results in one of the easiest, fastest, and most straightforward pre-processing paths from CAD to quality CFD meshes [30].

2.5 CODES FOR MULTIPHASE MODELING

For this literature, Eulerian multiphase model is used in Fluent software. In Fluent, Granular temperature is an important factor in calculation of granular viscosities.

Differential equation for granular temperature is possible, but due to convergence problems, the simplified algebraic form is useful. For granular kinetic part of viscosity, Syamlal-O'Brien and Gidaspow models can be used, and for the frictional part, Schaeffer's expression is available. Lun et al. expressions can be used for granular bulk viscosity, solids pressure and radial distribution function.

For turbulence modeling Fluent offers, in multiphase cases, k-ε-models (Standard, RNG, realizable) and now also Reynolds Stress model. In these k-ε-models, turbulence can be modeled by 'mixture', 'dispersed' or 'k-ε per phase' models. In the dispersed model, k-ε-equations are solved for the primary phase and Tchen's theory is used for the dispersed phases. (Fluent manual 2001).

2.6 GOVERNING EQUATION

There are several approaches which have proved to be useful used for simulation of multiphase flows, especially for engineering purposes. Among of them are Eulerian and Lagrangian approach. Duarte et al. [3] stated that the Eulerian approach, which involves balances of forces at work upon the particle and requires considerable computational effort. Meanwhile the Lagrangian approach considers the dispersed phase as a continuous phase and is based on the Navier-Stokes equations applied to each phase, as according to Syamlal et al. [17] and Gidaspow et al. [18]. According to Duarte et al.[29], the Lagrangian model or discrete particle models, DEM, the two dimensional motion of each individual particle is directly calculated from the forces acting on them, accounting for the interaction between the particle and the gas phase. Whereas in Eulerian models, gas and solid phase are considered to be continuous and fully interpenetrating. Both phases are described in term of separate sets of conservation equations with appropriate interaction terms representing the coupling between phases.

Basically, Eulerian multiphase model used to model droplets or bubbles of secondary phase(s) dispersed in continuous fluid phase (primary phase). This multiphase model allow mixing and separation of different phases; solves momentum, enthalpy and continuity equations for each phase and tracks volume fractions; uses interphase drag coefficient; allows for heat and mass transfer between phases; and can solved turbulence equations for each phases [19].

In gas-solid fluidized beds, there are high particle concentrations which the particle interaction cannot be easily neglected. Furthermore, the solid phase has similar properties to a continuous fluid [20]. The gas-solid fluidized bed has large volume fraction of solids, thus the best alternative is usually the Eulerian approach, which different gas and solid phases are treated as interpenetrating continua and momentum and continuity equations are defined for each phase [2]. Compared to Lagrangian models, as stated by Duarte et.al [29], a large number of particle trajectories are needed in order to determine the average behavior of a system, thus lead to requirement of a extremely high computational power to calculate the model. Therefore, this article is using Eulerian approach for numerical simulation of fluidized beds. By using the kinetic theory of granular flows [14], the viscous forces and the solid pressure of the particle phase can be describe as a function of the so-called granular temperature. In principle, the kinetic theory of granular flows has been derived from the kinetic gas theory: While the thermodynamic temperature is a quantum of fluctuating energy of the molecules on the microscopic scale, the granular temperature expresses the macroscopic kinetic energy of the random particle motion. This leads to the following differential equations given in the Eulerian notation.

In FLUENT, mass balance for each phase can be written as:

Gas phase:

$$\frac{\partial \varepsilon_g \rho_g}{\partial t} + \nabla \cdot \left(\varepsilon_g \rho_g v_g \right) = 0$$

Solid Phase:

$$\frac{\partial \varepsilon_{s\alpha} \rho_s}{\partial t} + \nabla \cdot \left(\varepsilon_{s\alpha} \rho_s v_{s\alpha} \right) = 0$$

where ε, ρ and v are volume fraction, density and velocity, respectively. The assumption made is no mass transfer is allowed between the phases. Each computational shell is shared by interpenetrating phases, so that the sum overall volume is unity.

Van Wachem *et al.* [21] stated that in most of the multiphase model, they use the following set of equations for the continuum equations:

$$\frac{\partial \alpha_g}{\partial t} + \nabla \cdot (\alpha_g v_g) = 0 \qquad \textit{for gas}$$

$$\frac{\partial \alpha_s}{\partial t} + \nabla \cdot (\alpha_s v_s) = 0 \qquad \textit{for solids}$$

where α is volume fraction and 'g' and 's' denotes gas and solid phases. Moreover, $\alpha_g + \alpha_s = 1$, has to be satisfied.

Volume fraction represents the space occupied by each phase, and the laws of conservation of mass and momentum are satisfied by each phase individually. Volume fraction for gas, ε_g and volume fraction for solid phase, $\varepsilon_{s\alpha}$ must sum to one.

$$\varepsilon_g + \sum_{\alpha=1}^{2} \varepsilon_{s\alpha} = 1$$

For momentum balances, the change of momentum equalizes the net force on domain according to Newton's second law. Here are the descriptions about what is in the net force for gas-solid fluidized beds: viscous force τ_i, the body force $\varepsilon_i \rho_i g$, the solid pressure force ∇p_s^*, the static pressure force $\varepsilon_i \nabla p$ and the interphase force $\beta(\upsilon_i - \upsilon_j)$ which couples the gas and solid momentum equations by drag forces [22].

Gas phase:

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g \upsilon_g) + \nabla \cdot (\varepsilon_g \rho_g \upsilon_g \upsilon_g) = \nabla \cdot \tau_g - \varepsilon_g \nabla p + \varepsilon_g \rho_g g - \beta(\upsilon_g - \upsilon_s)$$

Solid phase:

$$\frac{\partial}{\partial t}(\varepsilon_s \rho_s \upsilon_s) + \nabla \cdot (\varepsilon_s \rho_s \upsilon_s \upsilon_s) = \nabla \cdot \tau_s - \nabla p_s^* - \varepsilon_s \nabla p + \varepsilon_s \rho_s g - \beta(\upsilon_s - \upsilon_g)$$

In order to calculate granular temperature, equation of the solids fluctuating energy can be written as in Boemer et al. [23]

$$\frac{3}{2}\left[\frac{\partial}{\partial t}(\varepsilon_s\rho_s\Theta_s)+\nabla\cdot(\varepsilon_s\rho_s\Theta_s)\upsilon_s\right]=\left(-p_s^*I+\tau_s\right):\nabla\upsilon_s+\nabla\cdot\left(k_\Theta^*\nabla\Theta_s\right)-\gamma_\Theta^*+\Phi_\Theta^*$$

The left hand side of the equation is the net change of fluctuating energy, equal to the sum of generation of fluctuating energy $\left(-p_s^*I+\tau_s\right):\nabla\upsilon_s$, the diffusion of fluctuating energy $\nabla\cdot\left(k_\Theta^*\nabla\Theta_s\right)$, the dissipation γ_Θ^* and the exchange of fluctuating energy between gas and solid phase Φ_Θ^*. More detailed discussion can be found in Boemer et al. [23].

The current version of the code uses an algebraic expression for granular temperature, Θ_m, obtained from the energy equation of Lun et al. (1984), by assuming that the granular energy is dissipated locally; neglecting the convection and diffusion contributions; and retaining only the generation and dissipation terms (Syamlal 1987c). The resulting algebraic granular energy equation is

$$\theta_m=\left[\frac{-K_{1m}\varepsilon_{sm}tr\left(\overline{\overline{D}}_{sm}\right)+\sqrt{K_{1m}^2tr^2\left(\overline{\overline{D}}_{sm}\right)\varepsilon_{sm}^2+4K_{4m}\varepsilon_{sm}\left(K_{2m}tr^2\left(\overline{\overline{D}}_{sm}\right)+2K_{3m}tr\left(\overline{\overline{D}}_{sm}^2\right)\right)}}{2\varepsilon_{sm}K_{4m}}\right]^2$$

where K_1,K_2,K_3 is the granular stress constant and $\overline{\overline{D}}$ is the rate of strain tensor and K_{4m} is given by

$$K_{4m}=\frac{12\left(1-e_{mm}^2\right)\rho_{sm}\,g_{0mm}}{d_{pm}\sqrt{\pi}}$$

According to Syamlal et al. [24], heat transfer between the fluid and solids is assumed to be a function of the temperature difference:

$$H_{gm}=-\gamma_{gm}\left(T_{sm}-T_g\right)$$

where γ_{gm} is the heat transfer coefficient between the fluid phase and the mth solids phase. By assuming that solids phases 2 to M are in thermal equilibrium, γ_{g2} is the sum of the heat transfer coefficients γ_{gm} for m=2 to M. γ_{gm} is determined from the heat transfer coefficient in the absence of mass transfer, γ_{gm}^{0} , corrected for interphase mass transfer by using the following:

$$\gamma_{gm} = \frac{C_{pg} R_{0m}}{exp\left(\frac{C_{pg} R_{0m}}{\gamma_{gm}^{0}}\right) - 1}$$

formula derived from film theory (Bird, Stewart, and Lightfoot (1960). p.658):

The heat transfer coefficient γ_{gm}^{0} is related to the particle Nusselt number Nu_m:

$$\gamma_{gm}^{0} = \frac{6\, k_g\, \varepsilon_{sm}\, Nu_m}{d_{pm}^{2}}$$

Where Nu_m is the Nusselt number for the individual particles constituting the m^{th} solids phase.

The Nusselt number is determined from one of the many correlation reported in the literature for calculating convective heat transfer calculations. The heat transfer can be expressed in term of Nusselt number.

$$Nu = \frac{hl}{k}$$

Where *Nu* is Nusselt number, *h* is convection heat-transfer coefficient (constant of proportionality)(W/m^2.K), *l* is characteristic dimension of the solid object (m), and *k* is thermal conductivity of the solid (W/m.k) [28].

CHAPTER 3
METHODOLOGY

Basically, there are two stages involved in order to simulate the spouted fluidized bed:

i) Design the spouted fluidized bed mesh using GAMBIT
ii) Export the mesh into FLUENT

3.1 DESIGN THE SPOUTED BED

Basically, this research is done with reference to Shah [25] after taking into consideration that some of the real scale spouted fluidized bed is almost the same with Shah's pilot scale of spouted bed. The spouted bed is in 2 dimensional has been redesign to be almost the same with Shah in order to make comparison since there is lack of research is done in studying about heat and mass transfer for spouted fluidized bed. Below is the summary of the spouted bed specification:

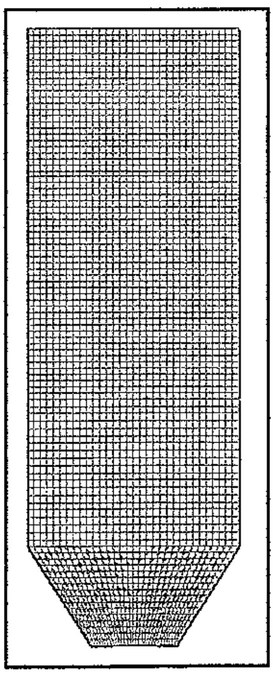


Figure 3.1 Mesh of spouted bed generated using GAMBIT

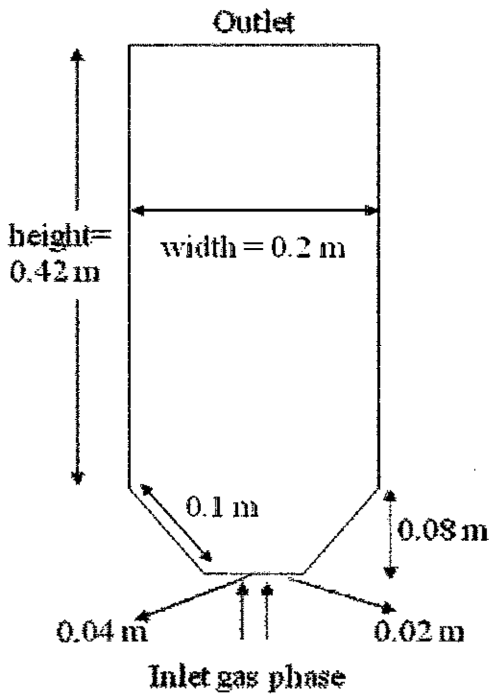


Figure 3.2 Dimension of spouted bed used in simulation

Figure 3.1 shows the generated mesh built using GAMBIT software. This mesh is a pilot scale size and has a total volume of 0.0952 m^3.

3.1.1 Geometry Modeling

In order to model the spouted bed, the specification of spouted bed was measured based on literature review. This was included with air inlet velocity, pressure used inside the spouted bed and operating temperature of air inlet and inside the spouted bed temperature. In this stage, it is important to ensure that the line is properly connected so that error can be eliminated in the simulation. Figure 3.3 shows the complete geometry model of spouted fluidized bed.

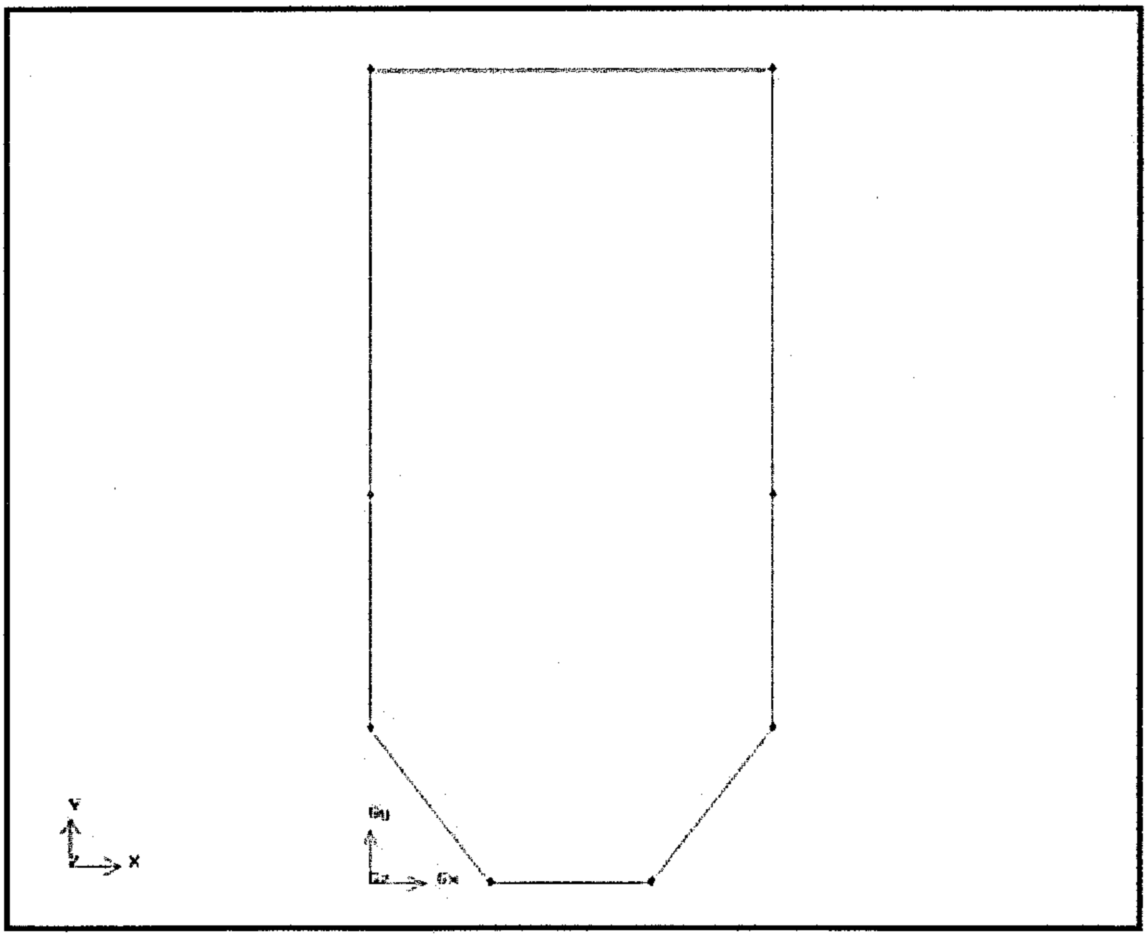


Figure 3.3: Complete geometry model of spouted fluidized bed

3.1.2 Geometry Meshing

Geometry meshing is a process to setup a grid system for computational purposes, which in other words to calculate what is inside the grid to be used in FLUENT. During this stage, volume of this spouted bed will be mesh according to their respective part: mesh edge, face and volume.

Function of mesh edge is to control the density of the mesh at specific region meanwhile mesh face is to give view the mesh edge effect. Mesh volume is the final step of meshing which give FLUENT ability to calculate in each mesh grid.

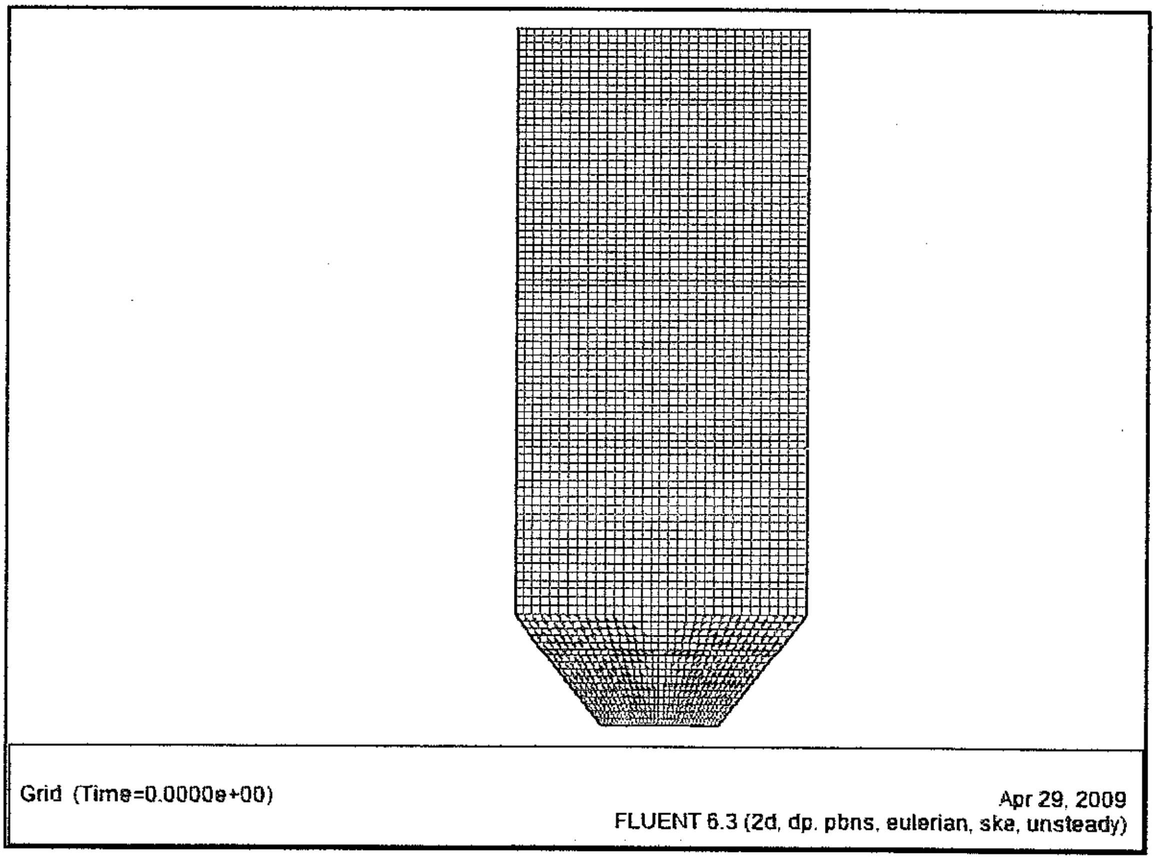


Figure 3.4: Complete mesh of spouted fluidized bed

3.1.3 Defining Boundary Types

Determining the boundary types of each edge of spouted bed is important for FLUENT which gives the information for solver to solve the mesh problem. In GAMBIT, solver defined is FLUENT 5 or 6. The boundary involves are velocity inlet, pressure outlet, and wall. The boundary condition is determine by trial and error to see whether it fits or not with the calculations in FLUENT.

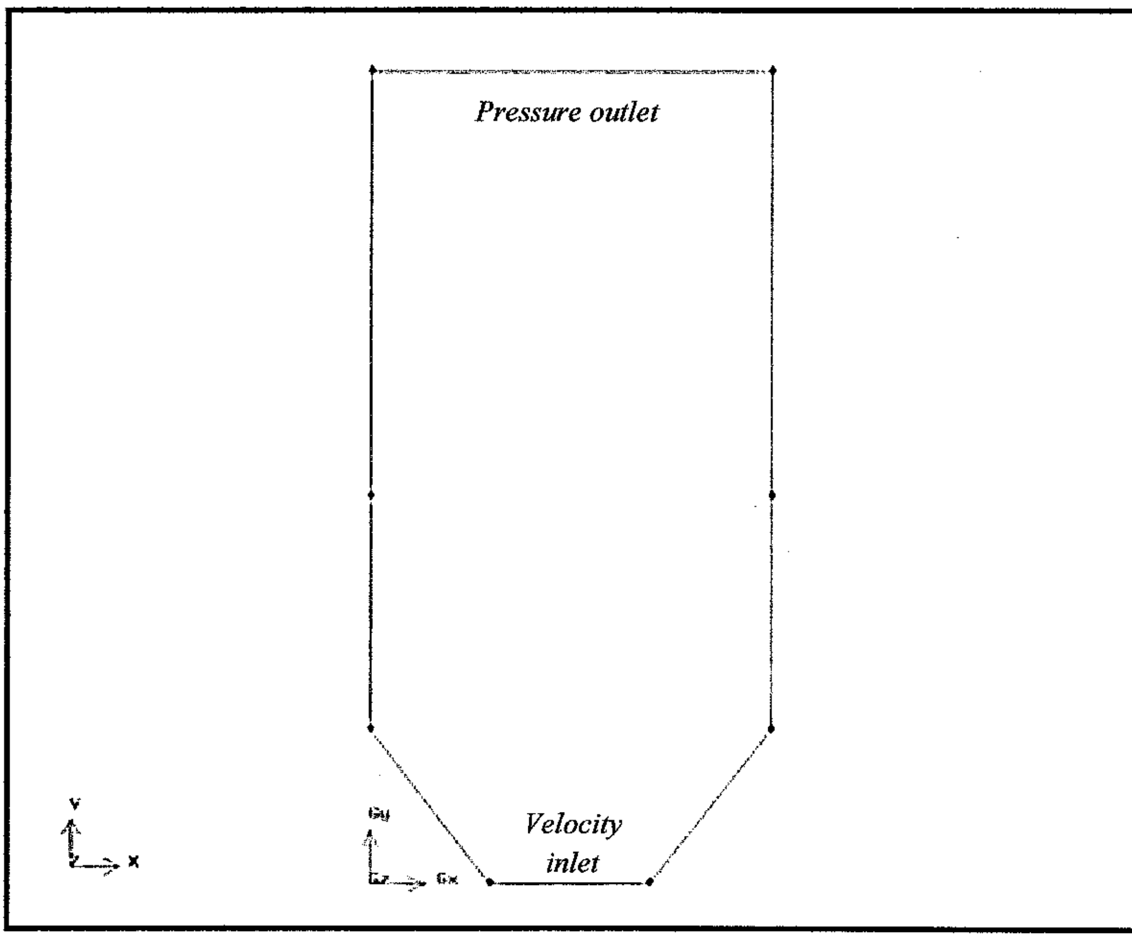


Figure 3.5: Boundary types of spouted bed

3.2 EXPORT THE MESH INTO FLUENT

The completed mesh is exported into FLUENT to simulate the problem. The mesh is a 2-dimensional and calculated using double precision. Below are the basic procedures on how to solve the problem using FLUENT:

1. Export the mesh into FLUENT.
2. Set up the problem in FLUENT (using 2D double precision)
    a. Read the mesh
    b. Check and display the grid
    c. Define solver properties (unsteady, 2D, 1^{st} order implicit)
    d. Define Eulerian multiphase model to be used (2 phases)
    e. Define the viscous model (using k-epsilon model, with mixture as k-epsilon Multiphase model)
    f. Define materials to be used (2000 kg/m^3 per granule)
    g. Define phases to be used (air and granule)
    h. Define the energy equation
    i. Define operating condition (using 101.325 kPa, gravitational of -9.81 m/s^2 at y-axis, operating temperature of 300.15 K)
    j. Define boundary condition (velocity inlet of 3 m/s, inlet temperature of 300.15 K)
3. Solve the equation
    a. Solve control solution
    b. Set initial guess (initialize)
    c. Set convergence criteria (time step size : 0.005, number of time steps : 100)
    d. Iterate up to specified time (0.5 s)
4. Analyze and visualize the result

3.3 APPLIED CONDITIONS IN FLUENT

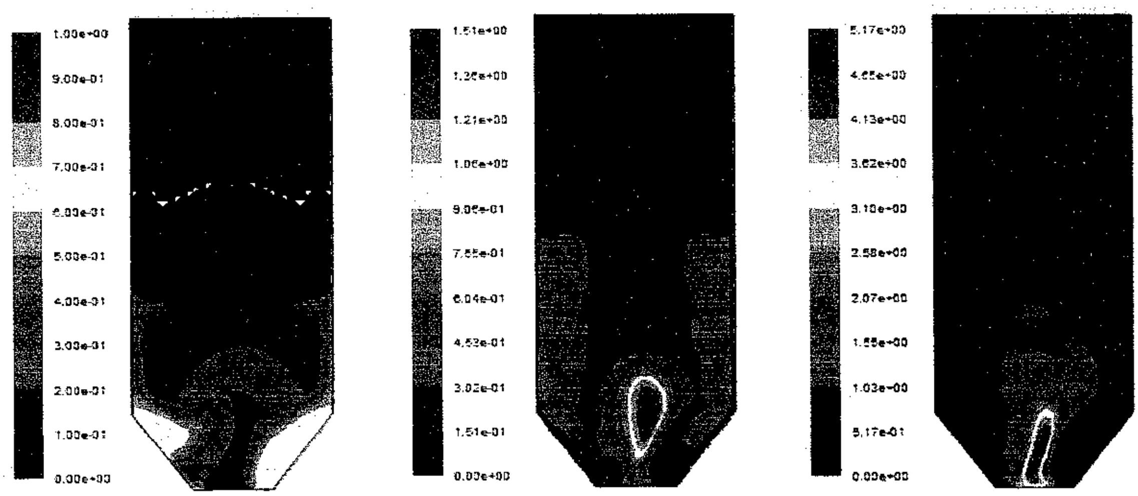
Applied conditions as a basis of simulation used is spouted bed height: 0.5 m, width: 0.2 m, solid particles with diameter: $d_s = 3\text{x}10^{-4}$ m, ρ_b: 2000 kg/m^3) have been used. Air inlet at the bottom of the fluidized bed is supplied at 3 m/s with temperature of 300.15 K is used. The operating pressure inside the spouted bed is fixed at 101325 Pa. The literature used time step size of 0.005 with 100 time steps which equivalent to 0.5 s of total simulation time and using Eulerian model to simulate the applied condition since Eulerian model is highly used multiphase model in gas-solids flow. Energy equation is used in this simulation to see the temperature profile of granule. Variables tested are velocity (10 m/s, 7 m/s, 3 m/s and 1 m/s), granular size (0.03 m, 0.003 m, 0.0003 m, and 0.00003 m), density and total iteration time.

It is assumed that the granule used in this simulation has sphere in shape, no heat loss to the outside of the spouted bed, and the granule will not stick to the wall.

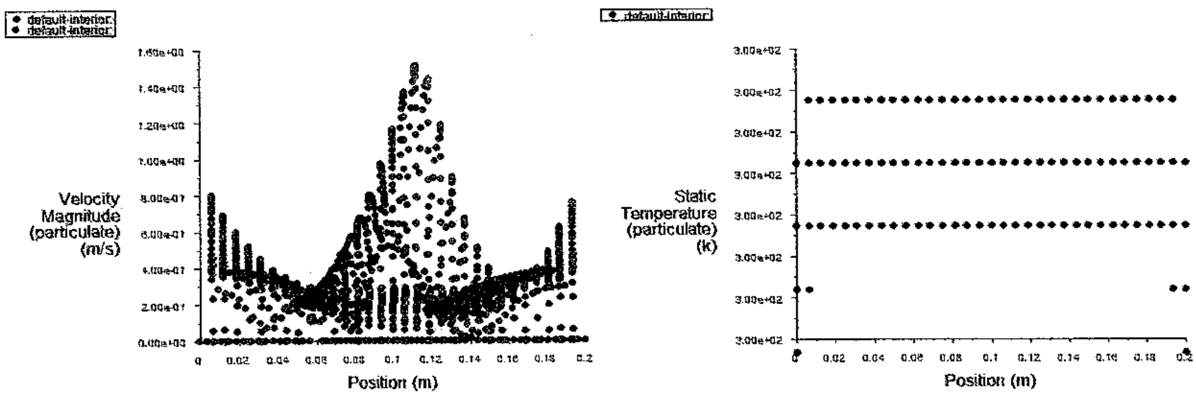
Since there is lacking in study of heat and mass transfer using spouted fluidized bed, this research is quite new and the result is discuss based on theoretical findings and referring only to a few journal.

4.0 RESULTS AND DISCUSSIONS

In results and discussion, the result is compared with the controlled simulation which is done by Shah [25] with time step size of 0.005, number of time steps of 100, inlet air speed of 3 m/s and using granule size of 0.0003 m. Assumption is made that the operating temperature of spouted bed is at 300.15 K in order to do heat transfer simulations. Figures below are the generated result based on Shah [25] data.

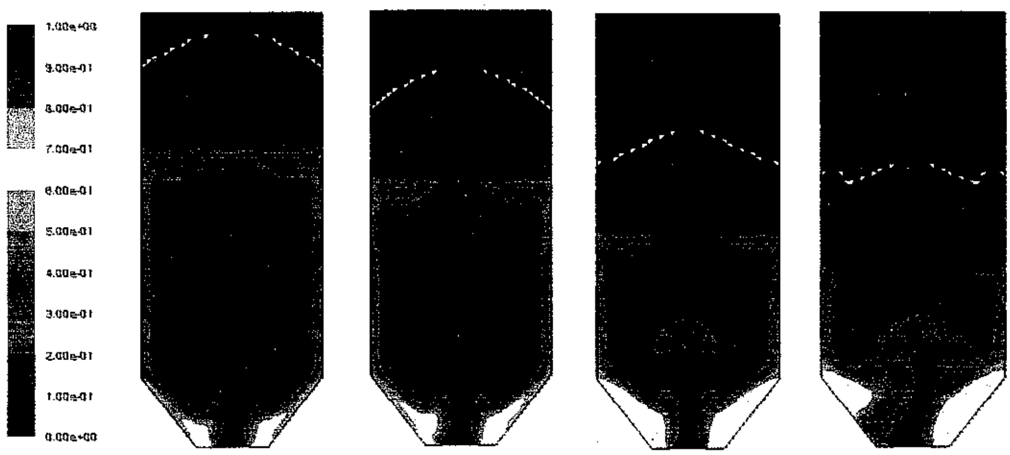


Volume fraction of granule phase | Velocity magnitude of granule phase | Velocity magnitude of air phase



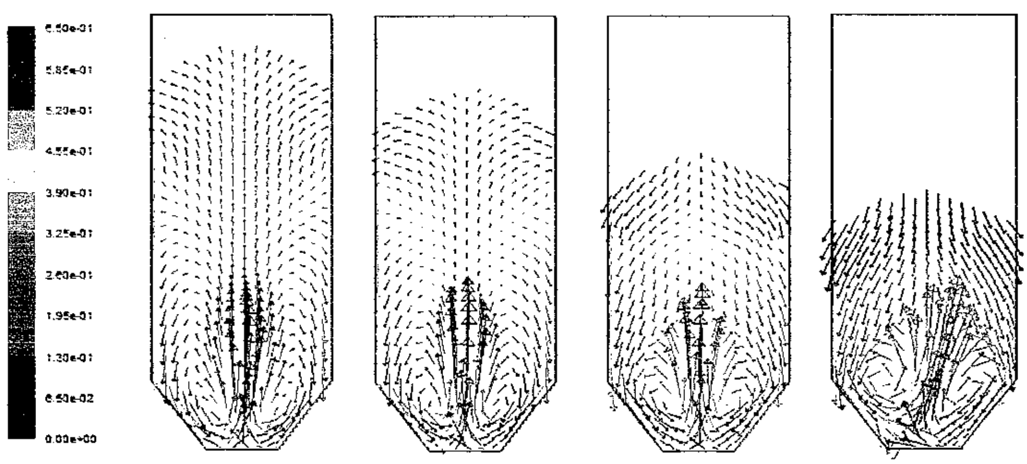
Temperature profile of granule phase | Temperature gradient of granule phase

Figure 4.1 Generated result based on the literature data



750 kg/m^3 1040 kg/m^3 1600 kg/m^3 2000 kg/m^3

Figure 4.2 : Contours of volume fraction of granule



750 kg/m^3 1040 kg/m^3 1600 kg/m^3 2000 kg/m^3

Figure 4.3: Vector of granule velocity coloured by volume fraction

Based on Figure 4.2, simulation is done using granular with various densities to see the effects: 750 kg/m^3 (based on average size of industrial granule size), 1040 kg/m^3 (based on Jacob [14]), 1600 kg/m^3 (based on Almuttahar [26]), and 2000 kg/m^3 (based on Srujal [25]). The trend on the spouted bed shows increase in density of granule will lead to decrease in bed height. This is because the addition weight per volume of granule is not proportional with the air inlet velocity, which is maintained at 3 m/s. At granule density of 750 kg/m^3 and 1040 kg/m^3, it seems that the formation of fluidized bed is happening quite aggressively at more than half of the total spouted bed height, with no granule observed at particular moment at the core of fluidization bed. Compared to higher density of granule usage (1600 kg/m^3 and 2000 kg/m^3), the formation of fluidized bed is happening at half or less than the total spouted bed height with all granule fill up the space at half of the spouted height and below. This is parallel as in the work done by Duarte et. al [29] who stated that gas interstitial velocity on the axis, decreases with longitudinal position on the axis, with this decrease being more pronounced as the solid density increases. He also stated that bed voidage on the axis of the spout and fountain decreases with bed level and bed voidage in the fountain core decreases radially from the axis to the core-periphery interface, which parallel with the findings on the density test of this research.

Meanwhile Figure 4.3 has shown the movement of granule inside the spouted bed. The movement of granule is basically start with granule move up from the inlet at the bottom of spouted bed, up to a certain height where the force of air inlet is no longer enough to support the weight of granule in a fluidize state before fall down and build a fountain shape of granule. The less density of granule, the higher fountain of granule formed, thus lead to a good heat and mass transfer process between the air and the granule.

In the next simulation, heat is supplied to study the heat transfer of fluidized bed.

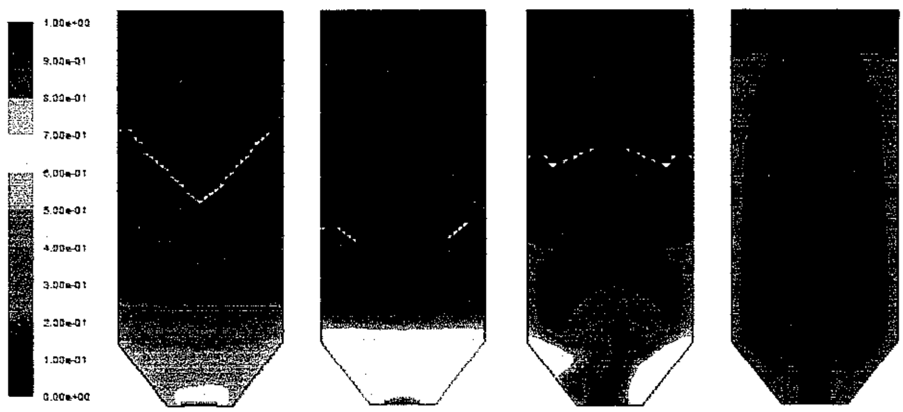
In order to meet the objectives, heat and mass transfer simulation are done in three different tests; that are simulation for:

Table 4.1: Variables tested in FLUENT

Heat Transfer	**Mass Transfer**
Different in granule diameter sizes (0.03m, 0.003m, 0.0003m, and 0.00003m)	Different in granule diameter sizes (0.03m, 0.003m, 0.0003m, and 0.00003m)
Different in air inlet velocities (10 m/s, 7 m/s, 3 m/s and 1 m/s) with air inlet temperature of 373.15 K	Different in air inlet velocities (10 m/s, 7 m/s, 3 m/s and 1 m/s)
Different in total simulation time (0.1s, 0.5s, 15s, and 30s)	Different in total simulation time (0.1s, 0.5s, 15s, and 30s)

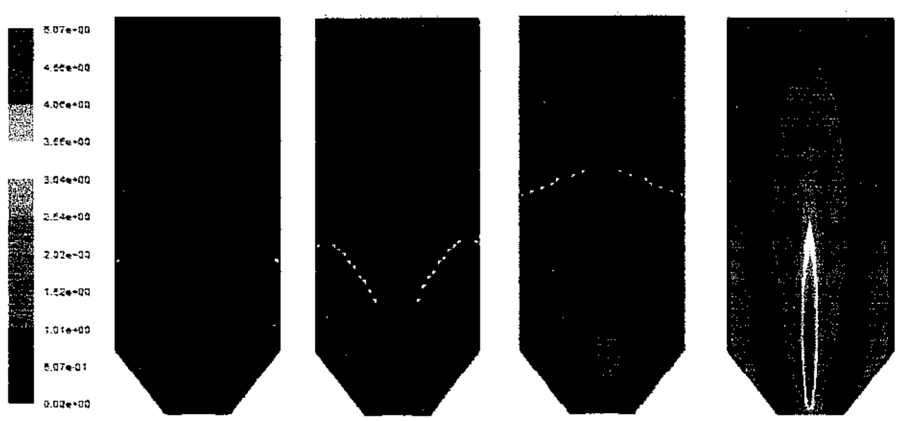
4.1 MASS TRANSFER - Simulation for different diameter sizes of granule (0.03m, 0.003m, 0.0003m, 0.00003m)

4.1.1 For Granule Phase



0.03 m 0.003 m 0.0003 m 0.00003 m

Figure 4.4: Contour of volume fraction for granule phase



0.03 m 0.003 m 0.0003 m 0.00003 m

Figure 4.5: Contour of Velocity Magnitude (in m/s) for granule phase

Based on Figure 4.4, with 3 m/s of air inlet speed and 2000 kg/m^3, granule inside the first two fluidized bed (0.03m and 0.003m) are fluidizing not very well. This is due to the diameter size of granule is too large for the given air speed to be fluidized In other words, the force created by air inlet is not enough to set granule in motion. Compared to granule with diameter size of 0.0003 m, the bed is fluidizing well by looking at the granule accommodating half of spouted bed with some of the granule sediment at the bottom wall of spouted bed which formed high volume fraction of granule phase. Meanwhile spouted bed with 0.00003 m size of granular diameter not forming any bed since the size of granule is too small and too light.

For Figure 4.5, spouted bed with granule diameter of 0.00003m in size over flown due to it very fine size of granule, thus prevent from creating a good fluidized bed. Mass transfer between air and granule will not occur efficiently due to less residence time for granule to be contacted with air (for dryer fluidized bed) or to be agglomerate accordingly (for granulation case).

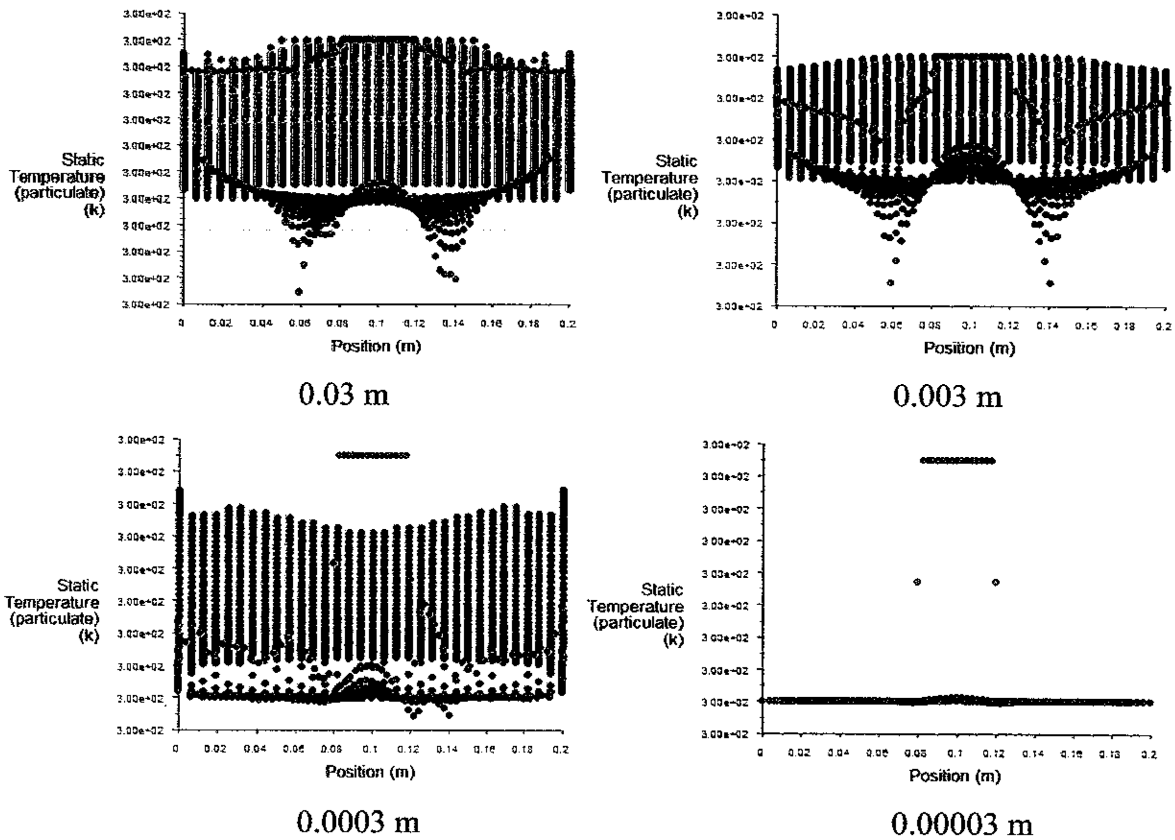


Figure 4.5: Temperature profile for granule phase

From Figure 4.5, only very small temperature difference of granule is observed due to no heat supplied at the air inlet of spouted bed for all sizes of granule diameter although there are pattern exists for different sizes of granule. For spouted bed with granular size of 0.00003 m, the granule's temperature is almost the same with each other since the size of granule is very small.

4.1.2 For Air Phase

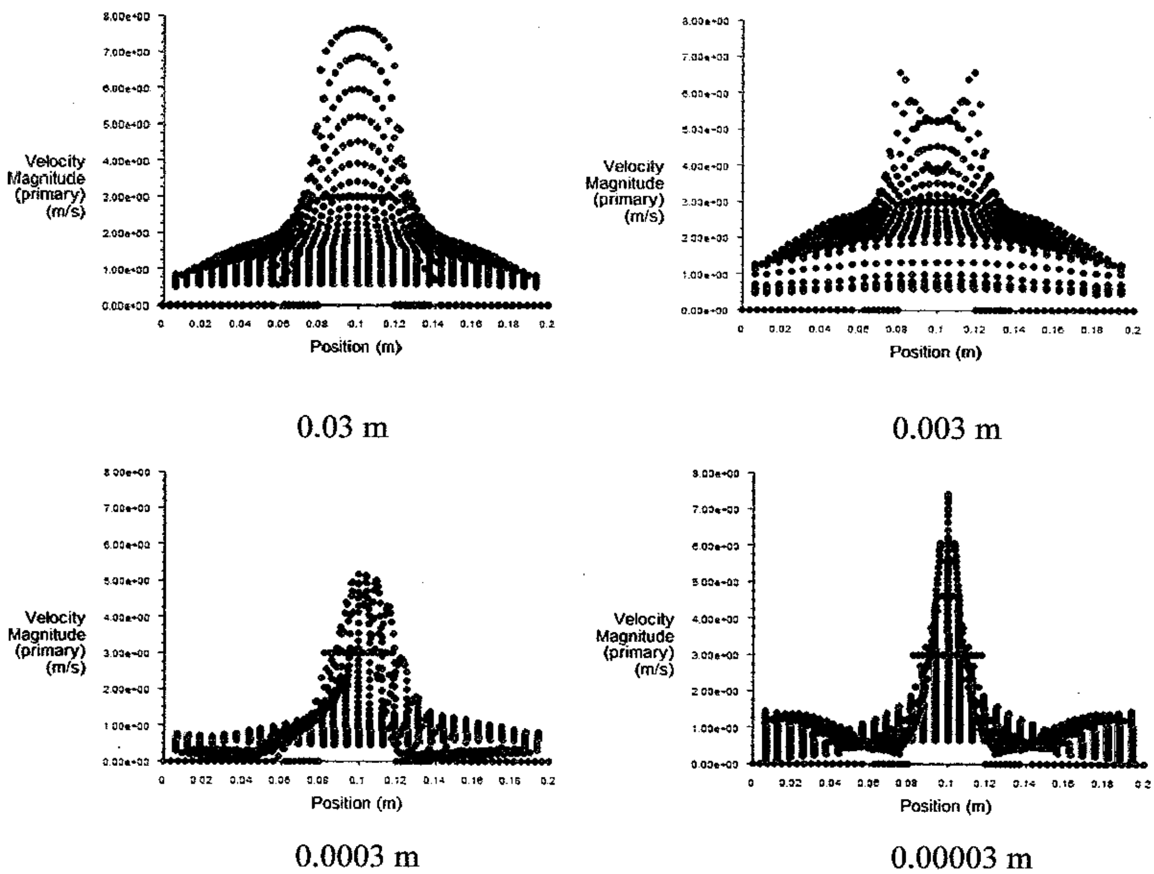


Figure 4.6: Velocity profile for air phase

For air phase, only velocity profile is discussed since the temperature is almost constant in each spouted bed (no heat supplied at the inlet of spouted bed). From Figure 4.4, the velocity is dropping and rising in ascending the granule size because for large diameter size of granule (0.03 m and 0.003 m) creating high resistance of air flow inside spouted bed, thus decrease the fluidized bed efficiency. For small diameter of granule, the air resistance is minimize thus allow the air to pass through the granule easily. The red and black colours of graph represents to indicate the granule observed is come from top and bottom partition of spouted bed.

4.2 MASS TRANSFER - Simulation for different speed of air inlet velocity (10 m/s, 7 m/s, 3 m/s, 1 m/s)

4.2.1 For Granule Phase

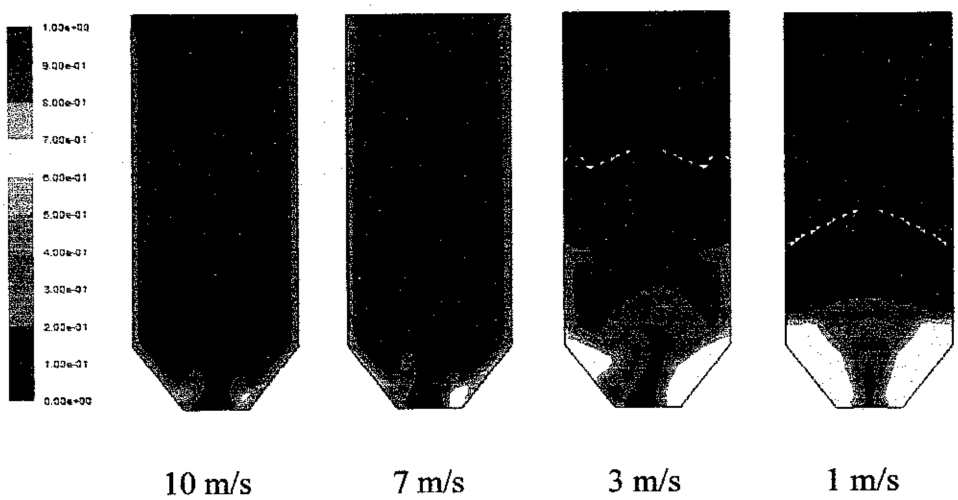


Figure 4.7 Contour of volume fraction for granule phase

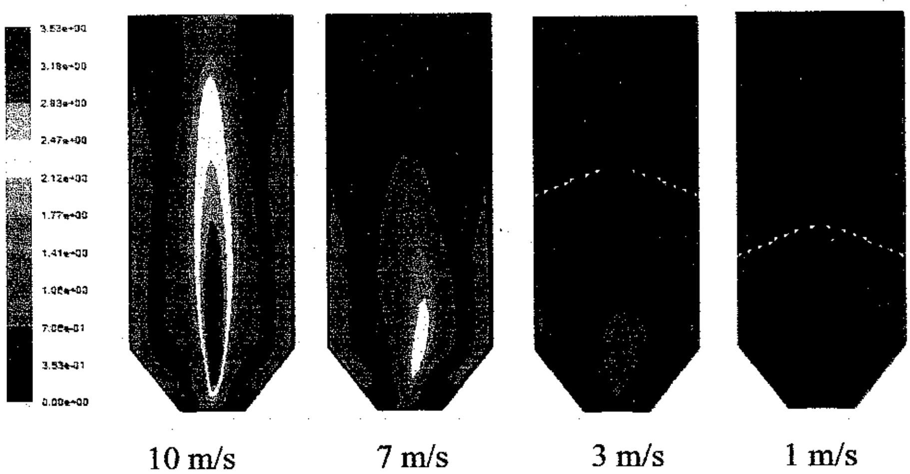


Figure 4.8: Contour of velocity magnitude for granule phase

Based on Figure 4.7, using air inlet speed of 10 m/s and 7 m/s are not practical since no fluidization bed is formed because of the air inlet velocity is too fast for granule size of 0.0003 m in diameter. Furthermore, most of the granule fluidized at the center of fluidized bed and fall downward very near to the spouted bed wall and only circulate back when the granule reached at the bottom of fluidized bed. Thus it is proven that using high speed of air will not lead towards proper mixing between air and granule. The velocity magnitude is shown extremely fast in Figure

4.8 for spouted bed of 10 m/s and 7 m/s with granule shoot up towards the top of spouted bed, which means the granule will have insufficient residence time to react with air. Meanwhile, the fluidized bed of granule is formed when using air velocity of 3 m/s for inlet and proven in Figure 4.7. Thus at this speed, mixing will be done between air and granule with enough residence time because the granule shoot up in the center of spouted bed until half of the spouted bed height before fall downward to circulate back, thus creating a good fountain shape of granule fluidization. This statement is supported by looking at spouted bed for 3 m/s in Figure 4.8 which sufficient amount of air inlet speed is used here. Besides, using 1 m/s of air inlet speed could not make the granule fluidized because the diameter size is too large for the supplied air speed.

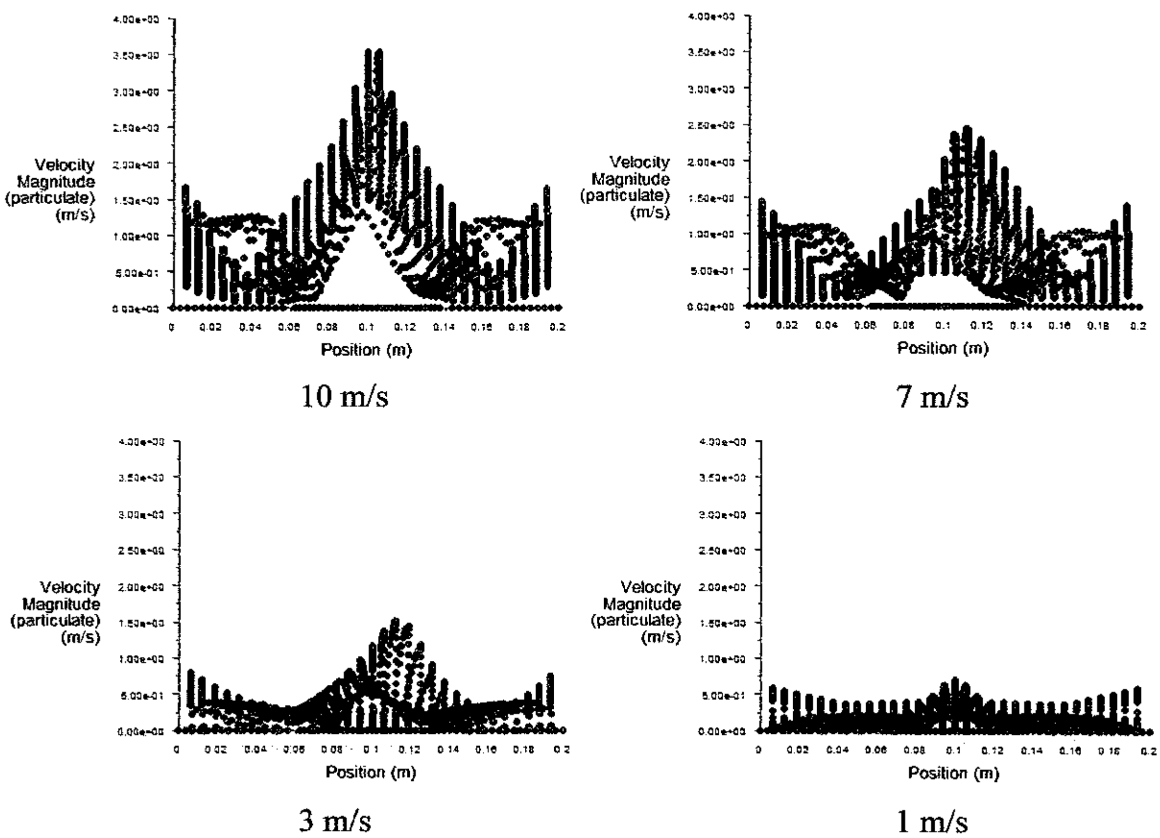


Figure 4.9: Velocity profile for granule phase

Based on Figure 4.9, there is a hole at the center of the graph for air inlet velocity of 10 m/s and 7 m/s, which indicates that all of the granule is fluidized and circulate at high speed. Compared to 3 m/s and 1 m/s of air inlet velocity, some of the granule has 0 m/s which mean some granules are not moving at that particular moment to allow heat transfer to occur between air and granule phase.

4.2.2 For Air Phase

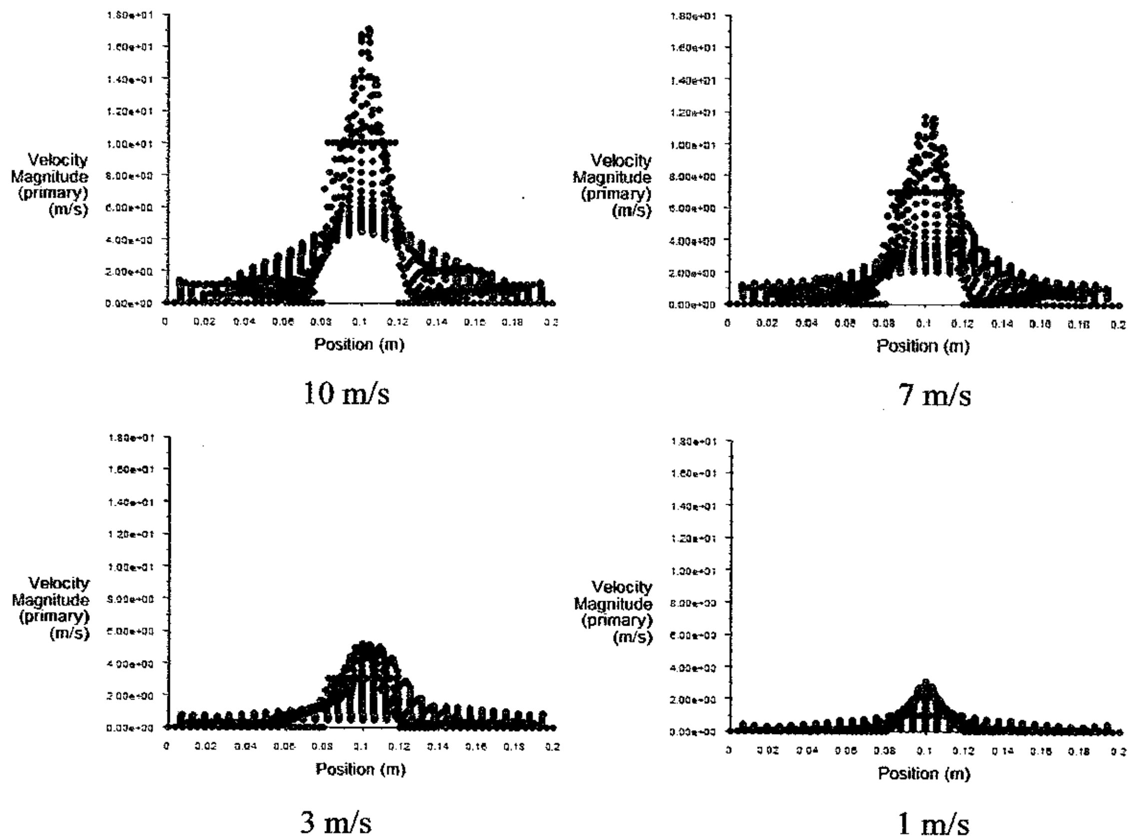


Figure 4.10: Velocity profile for air phase

Based on Figure 4.10, air is moving very fast for spouted bed with velocity of 10 m/s and 7 m/s of air inlet. The graph shown the air speed is moving faster than the air inlet velocity due to venture effect that is resistance of air exists when air is moving from spacious region into a narrow region. In narrow region, the air speed is increase due to all of the air is force to move through a small region from a big region. The situation is same for 3 m/s and 1 m/s of air inlet speed.

4.3 MASS TRANSFER - Simulation for different total simulation time (0.1s, 0.5s, 15s, 30s)

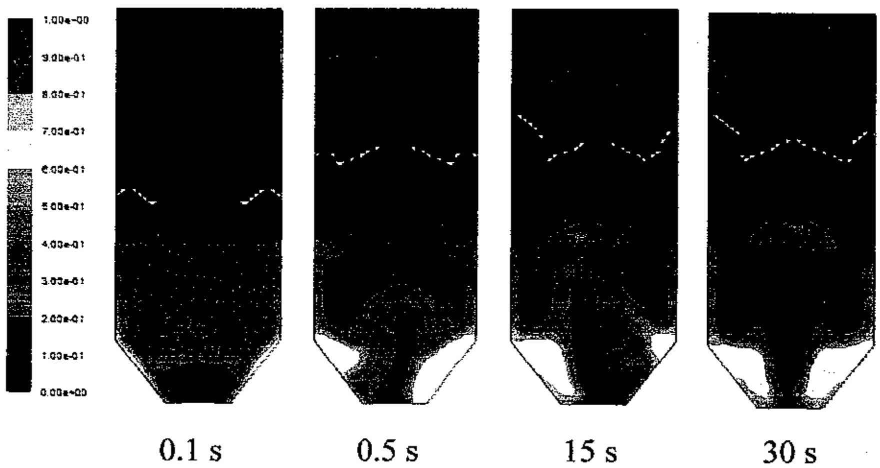


Figure 4.11: Contour of volume fraction for granule phase

This iteration is done on a computer with specification of Intel Core 2 Duo 1.86GHz with 6 gigabyte of RAM on Windows Vista 64 bit Service Pack 1. Based on Figure 4.9, using 0.1s of total simulation time (TST) is not practical since the granule is not yet mixing for that particular time although it takes about 1-2 minutes to complete. Using 0.5s of TST will takes about 5-6 minutes to complete, and the fluidized bed is already formed for this particular time, thus suitable to be used for rigorous test of simulation. For 15s and 30s of TST, this iteration will takes about 2.5-3 hours and 5-6 hours to complete. The result shown is nearly the same with 0.5s of TST, thus this is not practical to be used for repetitious test of variables in Fluent.

4.4 HEAT TRANSFER - Simulation for different sizes of granule (0.03m, 0.003m, 0.0003m, 0.00003m) with air preheat

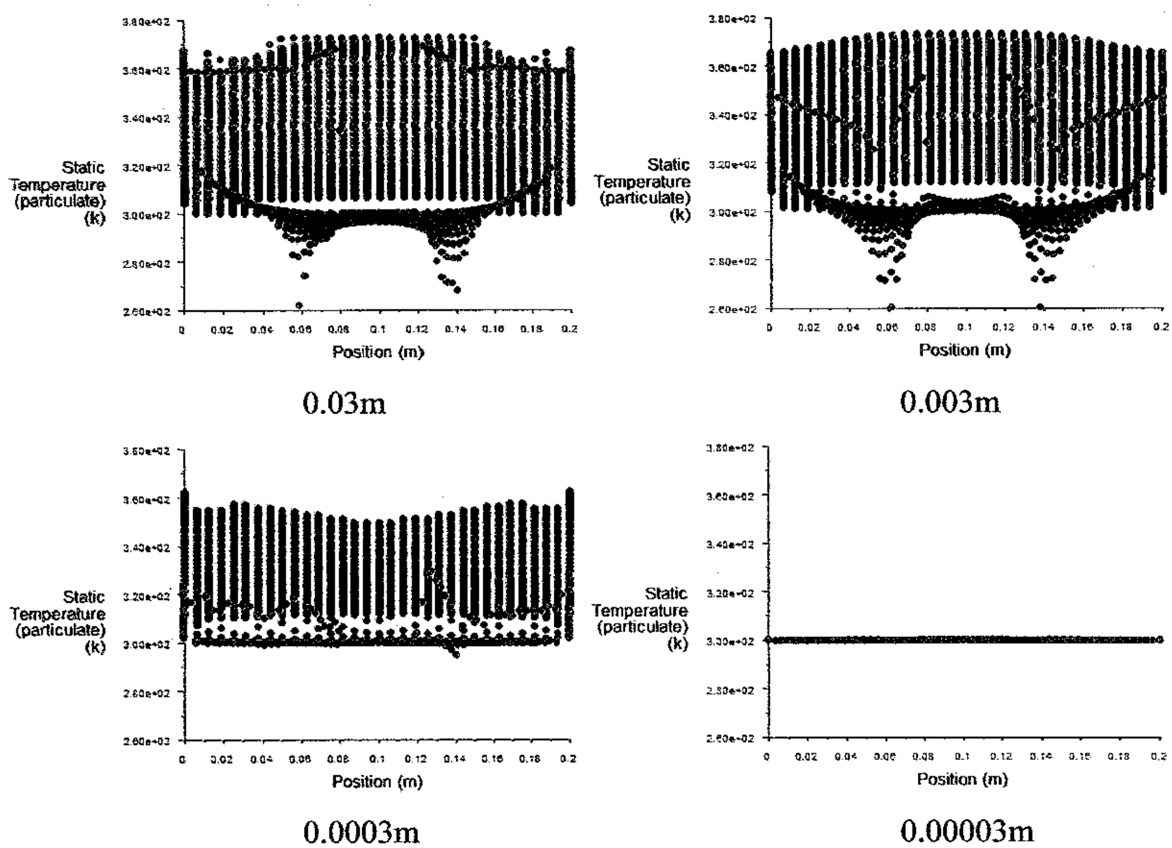


Figure 4.12: Temperature profile for granule phase

For heat transfer section, volume fraction and velocity profile are not discussed here since the effect of air inlet preheat to 373.15 K of temperature compared to 300.15 K of operating temperature gives the same results. For Figure 4.10, the black and red dots represents granule from top and bottom region of spouted bed. For spouted bed with granule size of 0.03m and 0.003m, most of the granule is heated up by the heat supplied together with air inlet but some of the granule is not heated, due to the large size of granule makes the air hard to reach, thus heat transfer is inefficient for this particular size of granule. For spouted bed with 0.0003m size in diameter, the entire granule is heated up by air by observing the minimum temperature of 295 K of granule compared to 260 K for the first two spouted bed of granule. Meanwhile for the smallest size of granule, no temperature difference is observed due to very small size of granule which contribute large heat transfer area and the given air velocity for that particular size of granule makes granule over-fluidized by air, thus makes the granule's temperature is the same with the operating temperature.

4.5 HEAT TRANSFER - Simulation for different speed of air inlet velocity (10 m/s, 7 m/s, 3 m/s, 1 m/s) with air preheat

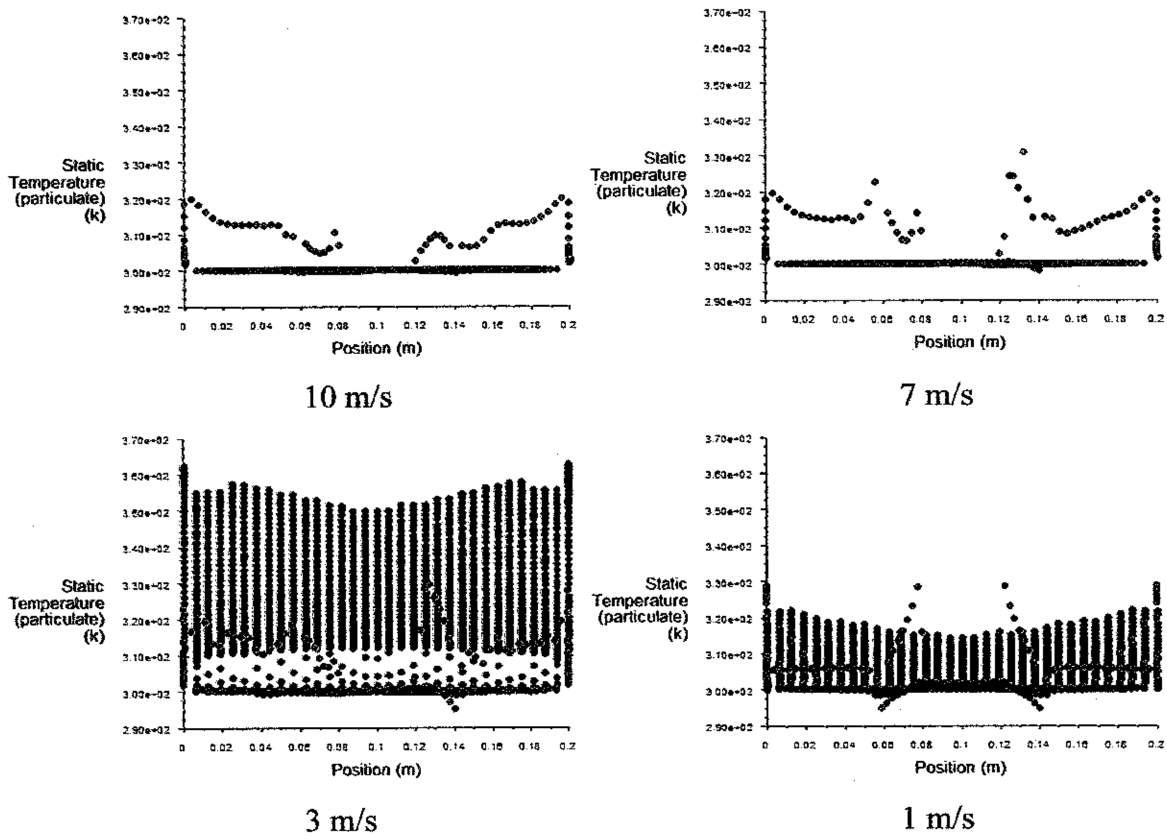


Figure 4.13: Temperature profile for granule phase

Same goes to simulation with different speed of air inlet, referring to Figure 4.13, heat distribution using air speed of 10 m/s and 7 m/s is not good by observing the temperature profile of graph. This is because some granule is heated with air but some is not; due to usage of high velocity of air thus lead to insufficient heat transfer and residence time between air and granule. Compared to usage of 3 m/s of air inlet, the heat distribution is occurred efficiently with maximum temperature of 363 K for granule, close to the temperature of air inlet. This is because the air speed is suitable for the size of granule chosen (0.0003 m) and has enough residence time for heat transfer to occur. Usage of 1 m/s of air inlet speed is not good since the maximum temperature of granule is still far away from the air inlet temperature.

5.0 CONCLUSIONS AND RECOMMENDATIONS

CONCLUSIONS

Based on the literature review and simulation result, it can be concluded that by using Eulerian multiphase model to simulate gas-solids flow in spouted fluidized bed produced result according to literatures (Myohanen et al. [2], Shah [25]). From all of the result above, by using total simulation time of 0.5 s and number of time step size of 0.005 will produce good formation of fluidized bed when using 750 kg/m^3 of granule density. The other results which use granule density of 2000 kg/m^3 and granule diameter size of 0.0003 m produce good formation of granule fountain which represent good granule circulation inside the spouted bed and also sufficient residence time for heat and mass transfer to occur. 0.5s of total simulation time proved to be sufficient enough to produce good results of varies condition.

RECOMMENDATIONS

In the next simulation, the range tested should be narrow down since the value obtain from this research is almost perfect. Furthermore, the simulation should be done with high computational power of computer in order to minimize the consuming time and error which usually occurred during the simulation calculation.

6.0 REFERENCE

[1] Fluidized Bed http://www.niroinc.com/pharma_systems/airflow_batch_fluid_bed.asp

[2] Kari Myöhänen, Vesa Tanskanen, Timo Hyppänen, Riitta Kyrki-Rajamäki. *CFD Modelling Of Fluidized Bed Systems*. Lappeenranta University of Technology

[3] C.R. Duarte, et al.(2008). *Numerical simulation and experimental study of fluid-particle flows in a spouted bed.* Powder Technology. Doi:10.1016/j.powtec.2008.04.077

[4] Dimitri Gidaspow (1993). *Multiphase Flow and Fluidization: Continuum and Kinetic Theory Descriptions*. California: Academic Press.

[5] M. Olazar, M.J. Sa Jose, A.T. Aguayo, J.M. Arandes, J. Bilbao. (1992). *Stable operation conditions for gas-solid contact regimes in conical spouted beds.* Ind. Eng. Chem. Res. 31. 1784-1791

[6] M.J. San José, M. Olazar, F.J. Peñas, J.M. Arandes, J. Bilbao. (1995). *Correlation for calculation of the gas dispersion coefficient in conical spouted beds.* Chem. Eng. Sci. 50. 2161–2172.

[7] M. Olazar, M.J. San José, A.T. Aguayo, J.M. Arandes and J. Bilbao. (1994). *Hydrodynamics of Nearly Flat Base Spouted Beds.* The Chemical Engineering Journal and the Biochemical Engineering Journal, 55. 27–37.

[8] Multiphase Flow. http://en.wikipedia.org/wiki/Multiphase_flow

[9] André Bakker (2002-2005). *Multiphase Flows – Applied Computational Fluid Dynamic.* Fluent Inc.

[10] C. Crowe, M. Sommerfeld, Y. Tsuji.(1998). *Multiphase flows with droplets and particles*. CRC Press.

[11] K.B. Mathur, N. Epstein. (1974). *Spouted Beds*. Academic Press Inc, LTD., New York. 304 pp.

[12] KuZilati KuShaari, Preetanshu Pandey, Yongxin Song, Richard Turton. (2006). *Monte Carlo simulations to determine coating uniformity in a Wurster fluidized bed coating process*. Powder Technology. 81-90

[13] Zhao Hui Wang, Guohua Chen. (1999). *Heat and mass transfer in batch fluidized-bed drying of porous particles*. Chemical Engineering Science 55 (2000). 1857-1869

[14] M. Jacob, ProCell technology: *Modelling and application, Powder Technology* (2008), doi:10.1016/j.powtec.2008.04.035

[15] Kuipers, J. A. M., Prins, W., & van Swaaij, W. P. M. (1992). *Numerical calculation of wall-to-bed heat-transfer coefficients in gas-fluidized beds*. American Institute of Chemical Engineers Journal, 38. 1079-1091

[16] A.H. Haidar, B. Matthews, *Future trend for computational fluid dynamics in the process industry, Third International Conference on CFD in the Minerals and Process Industries CSIRO,* Melbourne, 2003.

[17] M. Syamlal, T.J. O' Brien. (1989). *Computer simulation of bubbles in a fluidized bed.* AIChE Symp. Ser. 85. 22-31

[18] D. Gidaspow, R. Bezburuah, J. Ding. (1992). *Hydrodynamics of circulating fluidized beds, kinetic theory approach in fluidization.* Proceedings of the 7th Engineering Foundation Conference on Fluidization. pp.75-82

[19] Andre Bakker (2002-2006). *Eulerian Flow Modeling: Applied Computational Fluid Dynamics*. Fluent Inc. (2002)

[20] A. Schmidt, U. Renz. (1999). *Eulerian computation of heat transfer in fluidized beds*. Chemical Engineering Science 54. 5515-5522

[21] Van Wachem B.G.M., J.C. Schouten, R. Krishna, C.M. van den Bleek (2000). *Overview of CFD Models for Laminar Gas-Solid Systems.* AMIF-ESF Workshop "Computing methods for two-phase flow"

[22] Achim Schmidt, U. R. (2000). *Numerical Prediction of Heat Transfer in Fluidized Beds by a Kinetic Theory of Granular Flows.* Int. J. Therm. Sci. 871-885

[23] Boemer A.,Qi H., Renz. U, (1997). *Eulerian Simulation of Bubble Formation at a Jet In A Two-Dimensional Fluidized Bed.* Int. J. Multiphase Flow 23 (5) 927-944

[24] Madhava Syamlal, William Rogers, Thomas J. O'Brien (1993). *MFIX Documentation Theory Guide – Technical Note*. U.S. Department of Energy

[25] Srujal Shah.(2008). *Theory and Simulation of Dispersed-Phase Multiphase Flows.* Lappeenranta University of Technology.

[26] Adnan Almuttahar, Fariborz Taghipour. (2008). *Computational fluid dynamics of a circulating fluidized bed under various fluidization conditions.* Chemical Engineering Science 63. 1696 – 1709

[27] Stored Grain Research Laboratory (SGRL). http://sgrl.csiro.au/storage/heat/heat_grain.html

[28] CFD Online http://www.cfd-online.com/Wiki/Nusselt_number

[29] C.R. Duarte, V. V. Murata and M. A. S. Barozo (2008). *Experimental and Numerical Study of Spouted Bed Fluid Dynamics*. Brazillian Journal of Chemical Engineering, 25. 95-107.

[30] FLUENT Website. www.**fluent**.com/software/**gambit**/index.htm