Design of Packed Column for CO2 Absorption from NG at Reserves Using FLUENT

by

Haszlizah Binti Abd Hamit

Supervised by

Dr Murni Melati Ahmad

Dissertation submitted in partial fulfillment of

The requirements for the

Bachelor of Engineering (Hons)

(Chemical Engineering)

JUNE 2010

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Design of Packed Column for CO_2 Absorption from NG
at Reserves Using FLUENT

by

Haszlizah Binti Abd Hamit

Dissertation submitted in partial fulfillment of
the requirement for the
Bachelor of Engineering (Hons)
(Chemical Engineering)

Approved:

(Dr.Murni Melati Ahmad)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK

June 2010

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.

Haszlizah Binti Abd Hamit

ACKNOWLEDGEMENT

First and foremost, the author would like to thank the supervisor of this project, Dr Murni Melati Ahmad for the valuable guidance and advice. She inspired the author greatly to work in this project. Her willingness to motivate the author contributed tremendously to author project. To the Final Year Research Project Coordinator, Dr Khalik for providing the author with all the initial information required to begin the project.

To individual persons in from Research Department, thank you for assisting the author in completing this project.

Besides, the author would like to thank the authority of Universiti Teknologi PETRONAS (UTP) for providing a good environment and facilities to complete this project. Finally, an honorable mention goes to families and friends for their understandings and supports in completing this project. Without helps of the particular that mentioned above, the author would face many difficulties while doing this project.

ABSTRACT

Nowadays, the high composition of CO_2 at the gas reserves has incurred problem to the existing treatment system in the gas processing plant. As solution an additional treatment plant can be built at the gas reserves to reduce CO_2 composition before the gas enters the processing plant. The option to use a packed column in this additional treatment plant is investigated. This dissertation explain the background, problem statement, objective, methodology and the finding of the modeling simulation present computational fluid dynamics model built in FLUENT in order to simulate the CO_2 removal from high pressure natural gas using a specially designed solvent. The purpose for this simulation model is to study the behavior of mass diffusion of in the reactive absorption process with packing material as the contacting device in counter current absorption process. The packing area represent by a porous medium with 0.9 porosity. This research investigate the gas distribution throughout the column with 17ft packing height, at range of 1- 80 Bar operating pressure and the effect of liquid loading range from 50 – 150 m^3/m^2h to the decrease of CO2 content. In the study involving gas distribution, height of the column also increase from 14ft to 17ft in order to observe the effect to packing area to the gas distribution..

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURE

CHAPTER 1

INTRODUCTION

1.1. BACKGROUND NATURAL GAS AS AN ENERGY SOURCE

Natural gas is one of the important energy sources in the world after coal and oil. Since early 1970's the reserves have been increase five percent each year and now we have total of 152,000 milliard (10^9) cubic meter in 94 countries. Over sixty percent of these reserves are either in the Middle East or the Former Soviet Union. And most of the reserves are non-associated gas where the gas found separate from oil [1]. Known as the 'greenest' fossil fuel, natural gas is a source of energy that can relatively maintain a clean environment. In combustion process, NG produces almost no sulphur dioxide and low levels of nitrogen oxides, the main components of 'acid rain' and less carbon dioxide than coal or oil, which can cause the greenhouse effect. Eighty five percent of the of world gas production is consumed locally either by pipeline or as LNG and only fifteen percent are traded internationally [1]. Other than being used as a heating energy source for food processing, glass and ceramic industry, NG has huge demand for electricity production via co-generation. Roughly one third of total world gas production is use in power generation purposes. NG also being used as a raw material for fertilizer, hydrogen and plastic production [2]. Before being supply to the consumer, NG from the well are treated so it will be safe enough to be process. NG will be liquefied to become liquefied natural gas and liquefied petroleum gas .As a liquid it occupies just one over six hundredth of its gaseous volume making it economic in terms of both storage and transportation over vast distances.NG has been proven to be less harmful energy source compare to other fossil fuel. Refer to the Fossil Fuel Emission Levels by Energy Information Administration (EIA) [2]. Due to the awareness on preserving the environment; the demand of this fuel keeps on increasing. These high demands of NG urge this sector to explore more technology that can help on maintain the continuous supply of NG.

Fossil Fuel Emission Levels

- Pounds per Billion Btu of Energy Input

Pollutant	Natural Gas	Oil	Coal
Carbon Dioxide	117,000	164,000	208,000
Carbon Monoxide	40	33	208
Nitrogen Oxides	92	448	457
Sulfur Dioxide	1	1,122	2,591
Particulates	7	84	2,744
Mercury	0.000	0.007	0.016

Source: EIA - Natural Gas Issues and Trends 1998

Table 1 Fuel emission levels

1.2. CO_2 CONTENT IN NATURAL GAS

Even tough NG were stated as the cleanest fossil fuel among all, the acid gas content in the NG still need to be remove. CO_2 composition in NG has caused many problems both to unit operations and the pipeline. When combine with water, CO_2 will creates carbonic acid which is very corrosive which this can damage equipment in later process. High composition of CO_2 in Natural Gas will reduce the gross heating value. This will contribute to a lower price of the natural gas per unit volume. If to much CO_2 entrained to the liquefaction unit in LNG process, efficiency of MCHE (Main Cryogenic Heat Exchanger) will be reduce since the CO_2 freezing point is greater than the natural gas liquefaction temperature. CO_2 can solidify and block the tubing inside the MCHE. In order to meet the NG specification and the ease of further process as well as the transportation, CO_2 need to be removed. Low quality feed gas requires the removal of contaminants such as CO_2, H_2S or N_2 before the gas enters the pipelines in transportation system. The range of CO_2 content in Malaysian reservoir can be observed from the composition of Integrated Sarawak Offshore Gas Supply (Refer to Appendix; Table 9, Table 10 and Table 11). Normally natural gas process plant has their own Acid Gas Removal Unit, unfortunately it only design for some range concentration of acid gas and the unit cannot deal with the high concentration of acid gas. That is the reason why some reserves of natural gas are not being produced.

1.3. CO2 REMOVAL TECHNIQUES

Solvent based acid gas removal technologies were not the only option offered in NG industries, other option include membrane techniques and molecular gate. However, the solvent based acid gas removal technologies remains the most cost effective for deep removal of CO_2 for LNG production. Three basic types of liquid absorption processes are available:

Table 2 Type of liquid solvent used for absorption process [3]

Type of liquid	Description
Physical absorption processes	Physically absorbs CO2, H2S and organic sulphur components. Examples; Purisol and Selexol processes. Perform best mostly when partial pressure of the contaminants is high, the treated gas specification is moderate and large gas volumes have to be purified. Disadvantage; absorb significant quantities of hydrocarbons.
Chemical absorption processes	Chemically absorb the H_2S, CO_2 and to some extent COS. Organic sulphur components do not chemically react with the solvent. Examples; amine processes, using aqueous solutions of alkanol amines such as MEA, DEA, MDEA, DIPA and Flexsorb, and the carbonate processes, such as the

	Benfield process. Perform best mostly when contaminants are at relatively low partial pressure and have to be removed to very low concentrations. Advantages; minimum co-absorption of hydrocarbons Disadvantages; the process do not remove mercaptans down to low levels due to the low solubility of these components. Due to the chemical reaction between the solvent and CO_2 and H_2S, the regeneration energy requirements are higher than for a physical solvent.
Mixed solvents	A mixture of chemical and a physical solvent. Examples; Shell Sulfinol Process, which applies a mixture of sulfolane, water and DIPA (diisopropanolamine) or MDEA (methyldiethanolamine), Sulfinol-D and Sulfinol-M. The Flexsorb SE process also combines sulfolane and an amine and is in many ways similar way to Sulfinol. Advantages; simultaneously remove organic sulphur compounds and COS.

There are two types of flow arrangement that can be consider, countercurrent flow system and cross-flow system. Countercurrent are widely use since it has higher removal efficiency and able to minimize the amount of amine solution. Both operate similarly, the liquid fed from the top and flow downward throughout the packing material. But in cross-flow arrangement the contact of gas and liquid takes places in a horizontal profile [4] [5].

Basic concept on the treatment system are counter current process, untreated natural gas enters the column at the bottom of the column and the solvent (scrubbing liquid) enters the column from the top of the column. This allows the solvent to contact and absorb the CO_2 in the natural gas. (Refer to Figure 9 in the Appendices for the Simplified Process Diagram of Acid Gas Removal Unit) In order to optimal absorption efficiency, residence time and surface area need to be at most favorable state. In industrial application, several options has been implemented to meet this requirement, its include sieve tray column, contactor, spray column and packed column. These also called the contacting device, it is important to combine the technique and technology in order to design an absorption column that operate at given condition and at the same time archive the output target.

1.4. ISSUES WITH CURRENT TECHNOLOGIES

Most plant that we have nowadays can only treat gas with less than 10 % content of CO_2, which become a restriction for developments of sources that contain up to 60% CO_2. Other than that, economic and environmental forces have led the gas field with challenging treating requirement which at the same time affordable to be realize. Due to popular usage, demand on NG has increase from time to time. In order to meet the increasing demand, the NG developer has to take the risk on developing fields with higher levels of sulphur (H_2S, COS, organic sulphur components). One of the solutions that can be considered is higher pressure operation offshore treatment.

The focus now is to design packed column which functioning as the acid gas scrubber that has the capabilities to remove high concentration of CO_2 from the natural gas. In designing packed column, several important variables need to be consider, including the type and amount of contaminant to be removed, feed gas flow rate, temperature, molecular weight, humidity, selection of amine solution, presence of dust, allowable pressure drop for the system, effluent limitation in term of composition, temperature and entrain liquid, as well as the means for disposal of purge scrubbing liquid.

1.5. PROBLEM STATEMENT

Limitation of current AGRU shows that there is a need to design an offshore pretreatment unit that can reduce CO_2 content down to amount tolerable with the existed treatment unit in refinery system. Investigation on the removing of CO_2 from high pressure Natural Gas via modeling simulation approach and study the relation of the packed column design as well as its packing material, type of solvent and the effectiveness of the absorption process. The packing material will provide the necessary surface area and turbulence to archive the desired removal. Chemical reaction and occurrence of by-product will also be considered. Modify approach is used due to extensive and exhaustive range and operation that makes experimental work long and expensive.

1.6. SCOPE OF STUDY

In order to solve this problem, a treatment system needs to be set up at the reserve to reduce CO_2 composition before sending the gas to the process plant. The design need to have the ability to handle high pressure since feed is directly from the reservoir. However, this will double the cost since the treatment is significant contributor to the total cost of producing the gas [6].

In this study, the target is to reduce CO_2 composition from 50% to 20% approximately at 80 Barg inlet pressure at 30 degree Celsius temperature. The packing will provide necessary surface area and space time for the reaction absorption. The are a number of packing material available in the market such as; structured packing material e.g. Mallapak,INTALOX and FLEXIPAC, random packing material e.g. Nutter Ring, ceramic packing, metal and plastic random packing e.g. IMTP®, CASCADE MINI-RINGS®,β–ETA RING® and SNOWFLAKE [7]. Since this process involving chemical reaction, the presence of by product must be expected and fouling effect might occur. Due to that, design provision allowing frequent cleaning must be taken into account. Besides of that, the design will consist of the diameter of the column, optimal height, deep of the bed, position of the liquid distributor, arrangement of the packing material, type of the packing and the material of the packing itself [4][8].Upon completing this modeling project, the most favorable design of packed column will be produce as well as its economic evaluation.

1.7. OBJECTIVES

The main objectives are stated below:

1. To design a packed column for CO_2 removal unit in order to reduce CO_2 content from 50 % to 20% using a newly developed aminated solvent, that will be installed at the gas reservoir .
2. To study the behavior of mass transfer in the packed column and optimize the column design using FLUENT.
3. To identify the best operating parameter for the packed column.
4. To do the economic evaluation of the packed column.

CHAPTER 2

LITERATURE REVIEW

2.1. DESIGN OF PACKED COLUMN

Fundamentally, the removal performance of the CO_2 absorption process can be determined by the amount of gas- liquid contact provided by the column interior. The larger degree of contact, the better the absorption will be [9].In this work, mass transfer performance structured packing (Gempak 4A [1]in this case) were predicted modeling and compared with the experiment. Out of 23 simulation runs with different operating conditions (liquid temperature: 20-35°C; vapor temperature: 20-23°C), 21 runs show satisfactory prediction results in comparison with the experimental data. In this journal, I was found that only the effective mass transfer area causing the higher mass transfer energy. This experiment also proved that quality of initial liquid distribution also affects the mass transfer performance. Simulation done comparing two type of distributer, full distribution and 6H-distribution, has conclude that the full-distribution pattern generally provides better CO_2 absorption performance than the 6 H-distribution types. Rising of the feed temperature of the liquid solution from 20°C to 35°C also leads to a reduction in CO_2 exit gas content from 5.1 to 4.6%, since the increase in the liquid temperature increase effective mass transfer area [9]. As a rule of thumb, liquid temperature has to be at least 10°C higher than the vapor temperature.This is to prevent the vapor condensation.

---

[1] metal structured packing under KOCH-GLITSCH

Refering to the acticle by Cato Buch,2004, installation of washing tower might be nessesary in order to removes the remainder of the solvents that has been taken with the gases. In this acticle,structured packing material (Mallapak from Sulzer) used as top packing material and Random packing (Nutter Ring from Sulzer) are used as the middle packing. Random packing is cheaper but it gives a lower surface area which resulting a larger size of absorber and thus contribute to higher pressure loss [10].

2.2. Reactive Absorption

CO_2 absorption is controlled by diffusion with fast reaction in packed column [11]. Continuation of these researches, Manuel A. Pacheco et al in 1998 are emphasizing on the used of the fundamentals of rate-based modeling in reactive separation process, the authors purposely aimed at developing a better understanding of the rate processes present in reactive absorption system. The experiment applied to the selective absorption of H2S from the fuel gas containing CO_2 using aqueous Methyldiethanolamine. In this experiment, the diffusion coefficient of all reaction products and reactant except for CO_2 and H_2S were equal to diffusion coefficient of Methyldiethanolamine.

In this work, rate-based distillation module of Aspen Plus (RATEFRAC) was chosen to integrate the point model in order to explain the performance of the absorber. The result of mass transfer coefficient form Onda et al were used to predict liquid and gas side mass transfer coefficient in this research. Heat transfer coefficient for liquid and vapor applied in this routine were estimate from the Chilton-Colburn analogy and it has been proved that the liquid-side heat transfer coefficient does not any major effect on the temperature profile, where as the vapor-side heat transfer coefficient affect significantly with the predicted temperature profile. Point out also, as concentration CO_2 and H_2S decrease from the bottom to the top of the column, the interfacial mass of H_2S decrease as the driving force of the mass transfer decrease [12].

CO_2 absorption process involves chemical reaction at low pressure and high energy consumption. High absorption of CO_2 will result to lower energy consumption for solvent regeneration. Table below shows the reaction involve in CO_2 capture by amine solution [13].

Table 3 Process of CO2 solubility in solvent and chemical reaction involve.

Process of CO_2 solubility in solvent	Chemical reaction involve
Dissociation of water	$H_2O \Leftrightarrow OH^- + H^+$
First grade hydrolysis of CO_2	$CO_2 + H_2O \Leftrightarrow HCO_3^- + H^+$
Second grade hydrolysis of CO_2	$HCO_3^- \Leftrightarrow CO_3^{2-} + H^+$
Protonation of Amine	$AmineH^+ \Leftrightarrow Amine + H^+$
Formation of carbamate	$Amine + CO2 \Leftrightarrow AmineCOO^- + H^+$

Reactive absorption not necessarily requires elevated pressure and high solubility of absorbed component due to the chemical reaction. The equilibrium state can be shifted, causing the capacity of the solution increase. Reaction can be considered in bulk and in the film region [14]. In non-uniform packing and the creating turbulence, especially in the case of column-to-particle diameter ratios (= aspect ratios) lower than about 10, conventional plug-flow assumption is not valid [15].

2.3. Computational Fluid Dynamic (CFD)

Computational Fluid Dynamic is one division in Fluid Mechanics whereas CFD from FLUENT may refer to computational technology that shows dynamics interaction of gases and liquid with surface that defined by boundary condition. CFD can be used to do a computational model that represents a system that flow. Fluid flow physics and chemistry will be the necessary input to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. The software will provide you with images and data, which predict the performance of that design.

CFD has been used to predict the hydrodynamic in chemical process. In the journal written by Ludovic Raynal et al in 2005, CFD is used as a research and development tools. The applications of CFD in this work are on the bubble column simulation using Euler/Euler simulation and the gas-liquid flow along the structure packing using Volume of Fluid (VOF) simulation. For the bubble column the gas-liquid interaction are given by both drag law relationship and the bubble size. Mention here that bubble size variation may induce a big change in the hydrodynamic characteristic of the column. For the gas-liquid flow in the packed column, the structured packing which is made by corrugating metal sheet is arrange side by side with opposing channel direction. Mention also the structured packing can double up the surface, and high void fraction (90% at least).In this work, the flow is assume to be liquid film type and laminar flow. VOF model is used in this work because the gas and liquid flows do not interpenetrate therefore which is no formation of gas bubble and liquid droplet. This model enables the calculation done on the two phase flow that liquid and gas interface are clearly known. One found that the 1D model does not apply for a calculation involving complex geometry. The calculation was done in 2D geometry in order to determine the average value of the liquid film thickness along the packing. The result from this simulation validate with the experiment calculation [16].

The journal written by G.B. Liu et al in 2006 refers to a study of absorption in pilot scale and industrial-scale packed column by computational mass transfer. In this work, a multipart computational mass transfer model (CMT) is used for modeling the chemical absorption process by means of heat effect in packed columns. The feature of the model neglecting turbulent Schmidt number and not using the experimentally measured turbulent mass transfer diffusivity in order to predict the concentration and temperature as well as the velocity distributions. In this research, the model consists of the differential mass transfer equation with its auxiliary closing equations and together with formulations of computational fluid dynamics (CFD) and computational heat transfer (CHT). For experiment data, this research adopting data from Tontiwachwuthikul et al. in 1992. Experiment was conducted in a six equal-height sections packed column, with 1/2inch (~ 1.27 cm) ceramic Berl saddles with a total packing height of 6.55 m. It was absorption of CO2 from air by using aqueous MEA solution at total pressure of 103.15 kPa (103.15 bar) in a column of 0.1m ID. In order to analyze the concentration, the samples were taken at the inlet and outlet of each section. As for the industrial operation, data collected by Pintola et al (1993) from packed column using 2" stainless steel Pall rings, 1.9m in diameter was used. The packing material occupied three sections with total 14.1 m packing height. The operating pressure is between 5.39 (5.46 bars) and 7.60 atm (7.7 bars). However these parameters only reduce a little amount of CO2, from 2% to less than 100 ppm using MEA solvent. Here are some equations uses in CFD for this model;

The continuity equation:

$$\nabla \cdot (\rho h u) = M \qquad \textbf{Eq 1}$$

ρ = liquid density,
h = volume fraction of liquid phase based on pore space
u = *interstitial velocity vector*
M = source term of the continuity equation due to the chemical absorption of CO2 from gas phase, which is equal to the quantity of CO2 absorbed by the aqueous solutions per unit volume and unit time.

The *momentum equation:*

$$\nabla \cdot (\rho h u u) - \nabla \cdot (h\mu_{eff}(\nabla u + (\nabla u)^T - \tfrac{2}{3}\nabla \cdot u\mathrm{I})) = -h\nabla p + \mathbf{F}_{LG} + h(\mathbf{F}_{LS} + \rho \mathbf{g})$$

Eq 2

$$\mu_{eff} = \mu + \mu_t$$ **Eq 3**

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$ **Eq 4**

μ = molecular viscosity
μt = turbulent viscosity
μeff = effective viscosities of the liquid
I = unit tensor
FLG = interface drag force between gas phase and liquid phase,
FLS = flow resistance created by the random packing, which is considered to be the body force.

If turbulent viscosity μt is unknown, the equation can be solved simultaneously using standard k-ε model equations given below:

The turbulent kinetic energy k equation:

$$\nabla \cdot (\rho \square \mathbf{u} k) - \nabla \cdot \left[\square \left(\mu + \frac{\mu_t}{\sigma_k}\right) \nabla k\right] = h\mu_{eff} \nabla \mathbf{u} \cdot \left((\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \nabla \cdot \mathbf{u} \mathbf{I}\right) - \rho \square \varepsilon$$

Eq 5

The turbulent dissipation rate ε equation:

$$\nabla \cdot (\rho \square \mathbf{u} \varepsilon) - \nabla \cdot \left[\square \left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right) \nabla \varepsilon\right] = c_1 \square \mu_{eff} \nabla \mathbf{u} \cdot \left((\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \nabla \cdot \mathbf{u} \mathbf{I}\right) \frac{\varepsilon}{k} - c_2 \rho \square \frac{\varepsilon^2}{k}$$

Eq 6

The parameters which appear in k.ε model equation were shown below:

$$C_{\mu} = 0.09 , \sigma_k = 1.0 , \sigma_{\varepsilon} = 1.3 , c_1 = 1.44 \text{ and } c_2 = 1.92 \qquad \text{Eq 7}$$

L.raynal, F. Ben Rayana et al in 2009 were focusing on the simulation at small local scale and the simulation the large scale. This work had completed the latter study of Volume of Fluid (VOF) 2D simulation and had obtained original results for VOF 3D simulations dealing with wetting phenomena. In the small scale, CFD calculation on liquid film scale with the VOF approach, were used to determine effect of the surface wall texture and to catch the main characteristics of the liquid flow in the gas-liquid-solid interactions. The wetting phenomena study is important because of the determination of the wetting ratio of the effective interfacial area ae to the packing geometric area, aG. In this present work, three dimensional simulations are compared to experimental results. The studies consider a liquid stream flowing over an inclined plate which material can be altered. A parametric study has been done by changing separately the liquid flow rate QL, the static contact angle θ, the surface tension σ, the injector geometry and the slope of the plate α. Simulations at large scale, the effect of gas distributor on the flow field within the layers of packing are discussed. This research adapting the basic application of packed column using Mellapak 250 Y as the packing material. The gas and liquid flow counter currently at the industrial scale that is from inlets to outlets and in particular the gas distributor and the interactions are characterized by a length of the column as well as the packing material. In this study, each layer of packing are turned by each other with a 90° angle, and layer height of 21 cm. This work presenting a packed column with a liquid load which means the maximum value of CO2 absorption process of 50 m3 m-2 h-1 [17] , and a gas superficial velocity of 1.47 m/s.

Here are the explanations according to the scale study in these previous researches;

2.3.1. Small scale

2.3.1.1. *Wetting phenomena*

In small-scale the wetting phenomena were studied in order to determine the wetting ratio of the effective interfacial area, *ae* to the packing geometric area, *aG*. The geometric domain is 31 mm in length, 24 mm in width and 3 mm in height. The liquid injection is of rectangular shape of size 2*1 mm2 or 4*1 mm2. A meshing method is used with 15,000 to 45,000 initial cells depending on the injection size. It has been study that an increase in liquid flow rate of 100%, leads to an increase in wetted area of 70%. But in the other hand, an increase in contact angle θ from 24.5° to 67° leads to a reduced wetted area of 52%. Whereas only small effect is observed when the surface tension σ is multiplied by 2.2 or when the injector width decreased from 4 mm to 2 mm .it can be conclude that the liquid flow rate and the static contact angle controlling parameters for the wetting mechanism [18].

2.3.1.2. Characteristic of the liquid flow

This study is to determine the influence of the small-scale wall structure on the bi-dimensional liquid film flow. Resultant to a uniform fully developed laminar film over a smooth vertical wall with no gas interaction, no gas interaction, therefore the thickness of the liquid film is given by:

$$e = \left(\frac{3 q_L v_L}{g}\right)^{\frac{1}{3}}$$

Eq 8

The specific flow rate is given by the ratio of the liquid load, *QL* to the geometric area aG . [17]

$$q_L = \frac{Q_L}{a_G}$$

Eq 9

The liquid hold up is deduce from the liquid thickness by using,

$$h_L = e \times a_G$$

Eq 10

The simulations are to compare the liquid film characteristics, in terms of film thickness, *e*, and velocity at the interface, U*L,eff*, in values given by the commonly used laminar film model [17].

$$U_{L,eff} = \frac{2}{3}\left(\frac{gq_L^2}{3v_L}\right)^{\frac{1}{3}}$$

Eq 11

The averaged liquid velocity is given by the ratio of the specific liquid flow rate to the liquid film thickness [17].

$$\frac{3}{2}\overline{U}_L = \frac{3}{2}\left(\frac{gq_L^2}{3v_L}\right)^{\frac{1}{3}}$$

Eq 12

q_L= liquid flow rate per unit width of wetted surface.

v_L =liquid kinematic viscosity

g = acceleration of gravity

Q_L = liquid load of packing (For industrial conditions, the liquid load varies from approximately 10 to 100 $m^3m^{-2}h^{-1}$)

a_G = packing geometric area per unit volume.

The calculations provide the liquid hold up and the liquid velocity at the liquid film interface. VOF model was used in this simulation, this model able to capture the interface between two non-interpenetrating fluids in a steady calculation approach. The implicit scheme for the interface reconstruction and the HIRC module for VOF solver are used. In the work of Raynal et al in 2007,two geometry were used, first with a smooth wall and the other with texture sinusoidal like structure. Figure 1 shows a close view of the wall texture. For meshing purspose,8 to 10 grid point were used in this work,the number of cell is about 10,000 cells.

Next *the liquid hold up are obtain from the calculation of liquid* fraction in the central periodic element. For a result in this previus work, the small scale roughness of the wall give a better agreement with the experiment data even some discrepancy is still observed [17].When the liquid Reynolds numbers is vary from 40 to 1000, the recirculation zone grows as the Reynolds numbers increase. The present of these recirculation zone explain why the liquid hold up is greater in the case of rough walls. Therefore, liquid hold up can be conclude as a sum af a static hold up and a dynamic hold up.

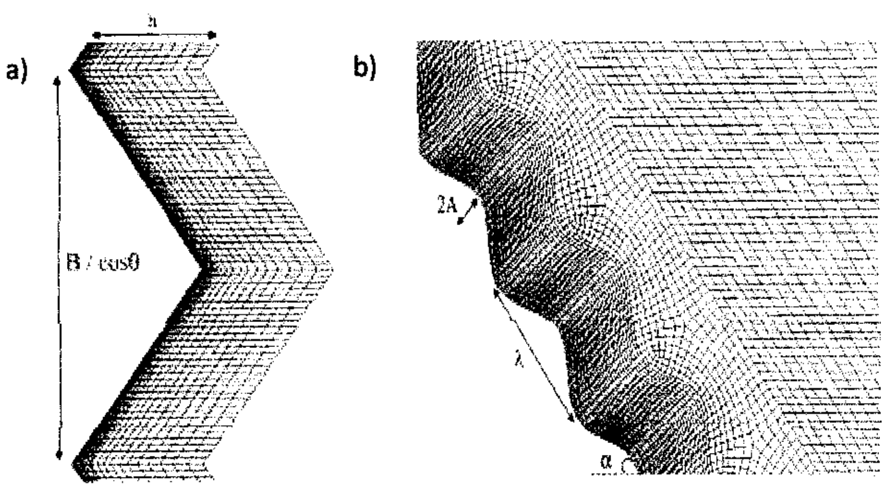


Figure 1 Close up view of the wall texture a) The smooth wall. b) The wall with texture

2.3.2. Intermediate scale

The calculation involve smallest periodic geometry characteristic of a packing, in work done by Raynal and Royon-Lebeaud in 2007, the smallest periodic of Mellapak 250Y were taken in consideration. The computational area correspond to the volume between the two opposite smooth metal sheet. With a lineic pressure drop, DP/L, imposing along the z direction,flow are condsider to periodic in the z and and y direction. Calculation run in a steady mode until the constant surface averaged gas in z component velocity is obtain. The required mass flux are about 1000 iterations and the value of residual are less than 10exp-5. The computational domain contains 150,000 tetrahedral cells. In order to study the viscous flow model influence, laminar and standard *k-ε* turbulent models have been used. Only simulations with gas flow are considered in these complex three dimensional geometry VOF simulations [17].

In order to take indirectly the liquid influence into account, boundary conditions have been adapted in two ways. First, the velocity obtained from the calculations, *UCFD*, has to be corrected by the liquid hold-up, since part of the volume should be occupied by the liquid. In actuality, a given value of pressure drop would be reached at a lower gas velocity liquid flow rather than without any liquid flow. Second, modification is done in the walls boundary condition. Commonly accepted by referring to Stichlmair et al in 1989 and Suess et al in 1992, below the loading point, the liquid hold up is not affected by the gas flow. Also from the previous work done by Sidi-Boumedine et al in 2005, observe that from the tomography measurement across the structural packing that there is almost no axial evolution of the liquid flow distribution [19]. This conclude that the main liquid velocity component is in the vertical direction and the velocity already obtain in the at the small scale which known as the liquid velocity at the interface. From this, the relationship of pressure drop coefficient in the vertical direction, *Kz* and the gas-liquid flow characteristic can be determine by [17];

$$K = \frac{\Delta P}{\frac{1}{2}\rho_G U_G^2} \times \frac{1}{L} = K(Q_G, Q_L, \nu_L), K_Z = \frac{\Delta P}{\frac{1}{2}\rho_G U_G^2} \times \frac{1}{L} = K_Z(Q_G, Q_L, \nu_L)$$

Eq 13

It has been observe that in this meso scale, the best validation data with the experiment data is obtained with the simulation run assuming a laminar flow. Calculation assuming turbulent flow gives higher pressure drop. In this particular laminar flow, the gas Reynolds numbers cover a wide range from 400 up to 2 exp 4 where the gas flow of the Reynolds numbers,*ReG* is obtain by;

$$Re_G = \frac{\rho_G V_{SG} / \sin(\theta) \frac{4}{a_G}}{\varepsilon(1 - h_L)\mu_G}$$

Eq 14

4/aG =hydraulic diameter being four time of the hydraulic radius.

ε = packing porosity (here = 0.95)

2.3.3. Large scale

At the large scale, packed bed consider as a porous bed which pressure drop characteristics are given by results obtained at meso-scale the previous results obtained at small scale. From previous studies, changing the packing disposition could have an effect on the gas flow distribution at packed bed inlet for a given gas distributor [17]. The work done by L.Raynal et al in 2009, there is no change done in the packing position, the differences only observe on the type of distributor.

Three different gas distributors where examine in this journal;

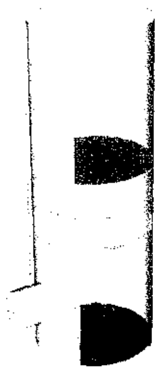
a) No distributor,
b) Vertical pipe distributor,
c) Vertical pipe with impact of plate distributor.

It is observed that the pipe distributor gives bad results even when equipped with a baffle. On the opposite, it is the simplest geometry, which is the case with no distributor gives best results.

These measurements indicates that, when possible, experimental measurements or industrial operation should first be run at maximum liquid load before being *set to nominal conditions to simulate as much as possible pre-wetted* conditions [18]. From this journal, it has been observed that geometry with no distributor give the best result of vertical velocity countor. It can be conclude that vertical pipe and a baffle might not suitable to be used in this particular design study.

Beside of that, in the journal written by M.Wehrli et al which discussing about the influence of vapor feed design to the flow distribution has considering various type of distributor. The type of distributors discussed includes standard inlet (no distributor), orifice baffle, vapor horn, schoepentoeter, and tubular distributor. The influence of these distributors was observed in CFD model. The study is most focus on the open space between the entry and the packing. The packing is model as a porous body that has a desired resistance factor which has the acceptable pressure drop [22]. The authors neglect the influence of the liquid and temperature variation in this model. The vapor flow is assumed to be incompressible and turbulence effect is taken into account by using standard k-ε turbulence model. This study also shows that if the geometry is symmetrical, the flow will not necessarily be symmetry. Diameter of geometry used in this study is 1m and the typical computational grid of some hundred thousand up to 1.5 million finite volume cells is applied to capture geometrical details of the feed system. Simple geometries are modeled by structured multiblock grids. For more complex inlet devices, the grids are handling by automatic grid generator using unstructured tetra grid. Below is the figure of the computational domain and the boundary condition in this study [20].

Table 4 Boundaries and boundary conditions (the colors refer to Figure 2)

<table>
<tr><th>Boundary</th><th>Position</th><th>Boundary Condition (BC)</th></tr>
<tr><td rowspan="4">Vapor inlet (Green)
Vapor outlet (Yellow)
Sump (Blue)
Walls (White)
</td><td>Cross section trough nozzle, some diameters upstream</td><td>Uniform velocity profile, typical turbulence intensity and length scale.</td></tr>
<tr><td>Cross section through the column, some space above the packing bed</td><td>Free outlet</td></tr>
<tr><td>Liquid surface considered flat</td><td>Symmetry, no shear</td></tr>
<tr><td>Column wall, nozzle wall</td><td>Adiabatic for mass and energy, log-law for turbulence</td></tr>
</table>

In order to have a clear comparison between the influences of the type distributor to the distribution flow, the results were the results are simplified as in Table 5 in the Appendix.

CHAPTER 3

METHODOLOGY

3.1. Research Methodology

The methodology presented in Figure 5 shows the execution phases involve in Final Year Project (FYP) titled Design of Packed Column for CO_2 Absorption from NG at Reserves using FLUENT. This Project is divided into two division, FYP 1 and FYP 2.For FYP 1, part one and part two in the flowchart have been done. The purpose of FYP 1 is mainly to develop strong knowledge on this particular topic. The learning process are mostly done by research and literature review. In FYP 2, the work execution will be focus on finding the suitable case study and develop the simulation. In this part, optimization of the design will be done in order to make sure it economical enough to be applied in the industries. The entire activities are summarized in the Gantt chart attached in the Appendix. Further explanation of the execution phases are described in the subsection.

3.1.1. *Part 1: Preliminary Research*

The research starts on the related journal about the CO2 reactive absorption then about the treatment system using absorption. Develop knowledge on packed absorption column and its industrial application. The research continued and focuses on the CFD information material and the software application. This part gives the author strong basic understanding by collecting information on design of the packed column, CO_2 reactive absorption and operating parameter.

3.1.2. Part 2: Screening, Investigate and Identify

Extract and analyze the information (article or journal)

3.1.2.1. Strategy for variable value selection

Study the packed bed as one porous medium that is characterizes by the pressure drop obtains in the intermediate scale. [17] This study is focus on identifying the optimum design (geometry of the column such as diameter, height, liquid distributor and etc) at the acceptable pressure drop and certain flow rate.

The methodology on determine the optimum design of packed column in GAMBIT and CFD is a circulation process as shown in Figure 5 below, this shows that there a lot of space on improving and modification on to get the best design.

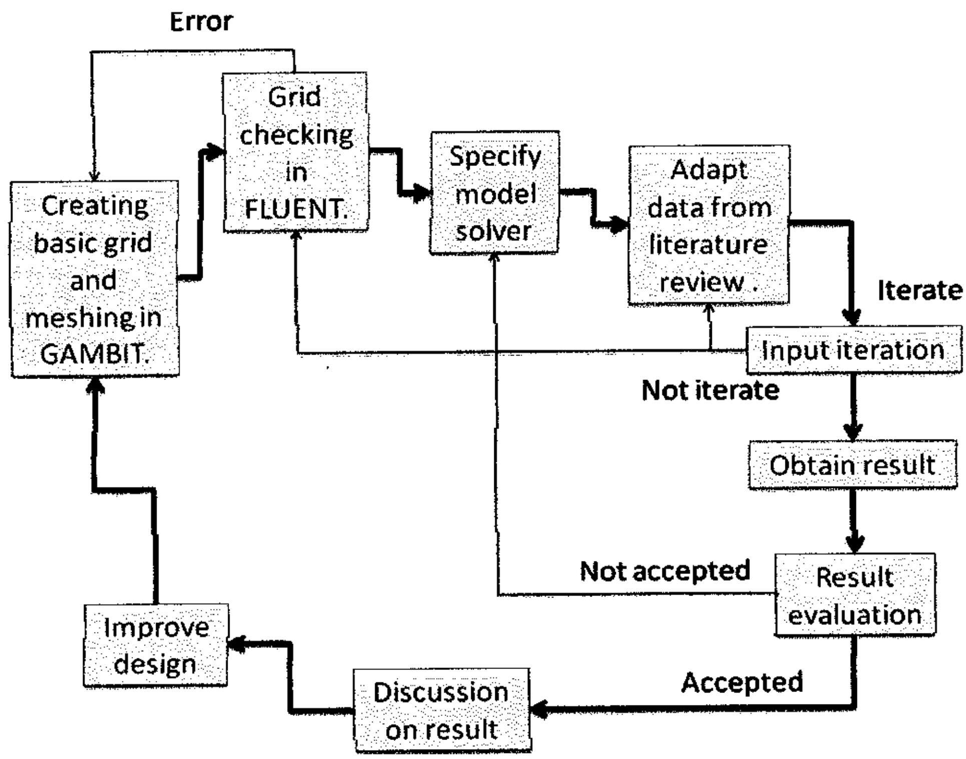


Figure 2 Flowchart on the simulation process in developing the column design.

In this study the simulation involved:

a. *Selecting height and radius of the column. (Considering only the gas flow)*
    i. By heuristic, the most efficient geometry of packed area is height of the packing are twice of the diameter. [4]Height and diameter adapted are with correlation factor of H/D = 0.5 obtained from JEAGER where the radius is 7 ft (2.13 m) and the height is 14ft (4.27 m).[4] From a hollow cylinder, the design improved base on result obtain.
b. Gas loading selection
    i. Two value of inlet gas velocity are used in order to observe the effect on the velocity vector. The velocity are 22.6 m/s, from previous research done by M.Wehrli et al and 1.546 m/s from JEAGER .[20][4]

3.1.3. Part 3: Selection of input Variables and Column Geometry

Feed gas flow rate, pressure and temperature, feed gas composition, and packed column characteristic.

Table 5 Value and variable selected in the study.

Variable	Value
Height of packing area	14 ft – 17 ft
Gas inlet velocity	22.6 m/s – 1.546 m/s

3.1.3.1. Inlet Properties

Properties of the inlet were taken from the Case Study: MLNG TIGA Module 8 Absorption Tray Column 8U91101 [24] (Kiong, 1998). The mentioned case study used is mole sieve tray column which is differ from our simulation i.e.

packed column, however outlet pressure and density of liquid inlet as well as the gas inlet composition can be use as the basis of simulation.

a. Operating parameter (Case Study)
    i. Pressure : 59 Barg
    ii. Temperature :
        1. Top : 45°C
        2. Bottom : 65°C
b. Gas inlet
    i. Pressure : 59 Barg
    ii. Temperature : 21°C
        1. Inlet stream of the gas will be the Natural Gas with 50 percent concentration of CO2
c. Liquid inlet
    i. Pressure : 59 Barg
    ii. Temperature : 30°C
        1. The liquid inlet will be the solvent.

Composition	Percentage
Alkaline aminated resin (AAR)	20-30
Polyhydric alcohol (PA)	10-15
Isopropyl alcohol (IPA)	10-15
Water	40-60

Table 6 Composition of the special solvent.

In the simulation model, liquid inlet is specified with viscosity of 0.001 kg/m-s since most of composition is water. In amine solvent, water is functioning as viscosity control. [24]Gas outlet which is the leaner NG from the column outlet is send to the refinery for further treatment

d. Liquid outlet.

    Fat solvent which will be send to the regenerator column (desorption column)

3.1.3.2. Process requirement

- High pressure gas inlet, approximately 80 Barg.
- Reduce 50 percent of CO_2 concentration to 20 percent concentration.
- The operating temperature is 30 °C (303 K)

3.1.3.3. Multiphase model (gas-liquid flow)

The type of model determination is depending on the characteristics of the flow itself. One has to know the flow regime that presence in the process. Regimes that can occur in gas- liquid flow are;

a. Bubbly flow: discrete gaseous or fluid bubbles in a continuous fluid
b. Droplet flow: discrete fluid droplets in a continuous gas
c. Slug flow: large bubbles in a continuous fluid
d. Stratified/free-surface flow: immiscible fluids separated by a clearly-defined interface

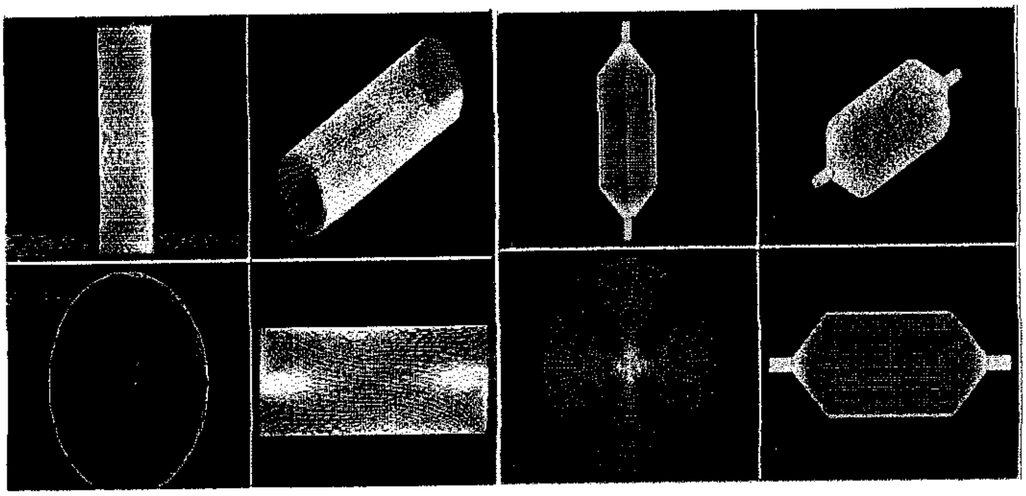
In FLUENT software there are several multiphase models that can be chosen to solve the simulation. The model includes:

a. Discrete Phase Model (DPM)
b. Mixture multiphase Model
c. Volume of Fluid Model (VOF)
d. Eulerian Multiphase Flow Model

At the first stage of the study, pressure-based solver is used since it only involve gas phase. At the second stage of the study where by both gas and liquid phase are consider in order to capture CO_2 concentration a Mixture multiphase model are used. Even tough this high pressure operating condition might give better agreement with turbulent models, laminar model are because this involve complex three-dimensional flow, therefore turbulent model is not recommended.[17]

3.1.4. Part 4: Geometry selection

Information gathered in the literature review were used as a guideline in the creating the geometry design. Since the sizing will be depend on the capacity of the column, design are more focusing on the shape of column, vapor distributor, liquid distributor, and the packing used. An issue that has to be keeping in mind in designing the geometry is the high pressure feed.



a) b)

Figure 3 Basic design of the column generated using GAMBIT, a) hollow cylinder that represent packing area, b) geometry of the column.

Next, the type of internal a taken into consideration by referring to their performance from the previous research. Internal of the column include the type of contactor, distributor, mesh and vortex breaker. For the time being, only two internal are currently take as a major part, which is the packing and the vapor distributor.

In the current industries, there are variety of packing can be found to be proven giving a good performance on providing desired active surface area. In this study, there are two type of packing that had been analyzed to become the contactor internal for this absorption column. The packing are random packing and structured packing. Studies on this packing had been done by Jian Chen and Weiyang Fei from Tsinghua University, Beijing. Their studies are focus on the ring packing which called pall ring and structured packing. They had observed that corrugated surface of structured packing had tremendous effect on the liquid flow.

Other literature review also support result obtains by Jian and his co worker on this finding. Kevin Bennett and Mark Pilling from Sulzer Chemtech also agreed that the structured packing implementation is the best option for high pressure absorption [21]. Since it has been proven that structured packing are predicted to handle high pressure absorption process better than random packing, structured packing are being used in this current study. However, industries nowadays offer various design of structured packing. In this particular study, structured packing is chosen by the capacity of the liquid hold-up. The researches on this part are still on going.

The most critical part when considering high pressure feed is the inlet of the column. Expected high velocity can damage the column as well as the internal of the column. It is essential for the inlet to have the designs that manage to handle high velocity and distribute the feed uniformly for the sake of both column and internal life span. Various options have been found from the previous research and one type of distributor that gives excellent result is vapor horn [17][20] The technical design of this distributor give an advantage for it to distribute high velocity vapor uniformly without damaging other internal part of the column. The design of vapor horn develops a swirl forcing to the stream and creating a high secondary velocity component. These characteristics induced flow path length and then balance the flow field.

3.1.5. Part 5: Investigation on the effect of gas velocity,height,pressure and liquid loading using simulation.

Analyze the result whether the design meet the purpose as absorption column that manage to handle high pressure inlet and meet the outlet requirement. The simulation using FLUENT is in progress.

a. Effect of operating pressure.

    ii. Effects of operating pressure are observed in this study. The pressure starts from 1bar, 10 bar, 30 bar, 50 bar and 80 bar. The contour in the packing are observe in axial and radial direction.

b. Mass transfer between gas and liquid

    iii. The ability of the column to provide a contacting area for the mass transfer to occur is observed by the concentration CO2 profile throughout the column for 10 bar operating pressure and 80 bar operating pressure.

3.1.6. Part 6: Possible Design Optimization

Simulate the design with a more suitable internal part that can improve the relevancy and performance of the absorption column.

The methodology use in the overall research is simplified in this flowchart:

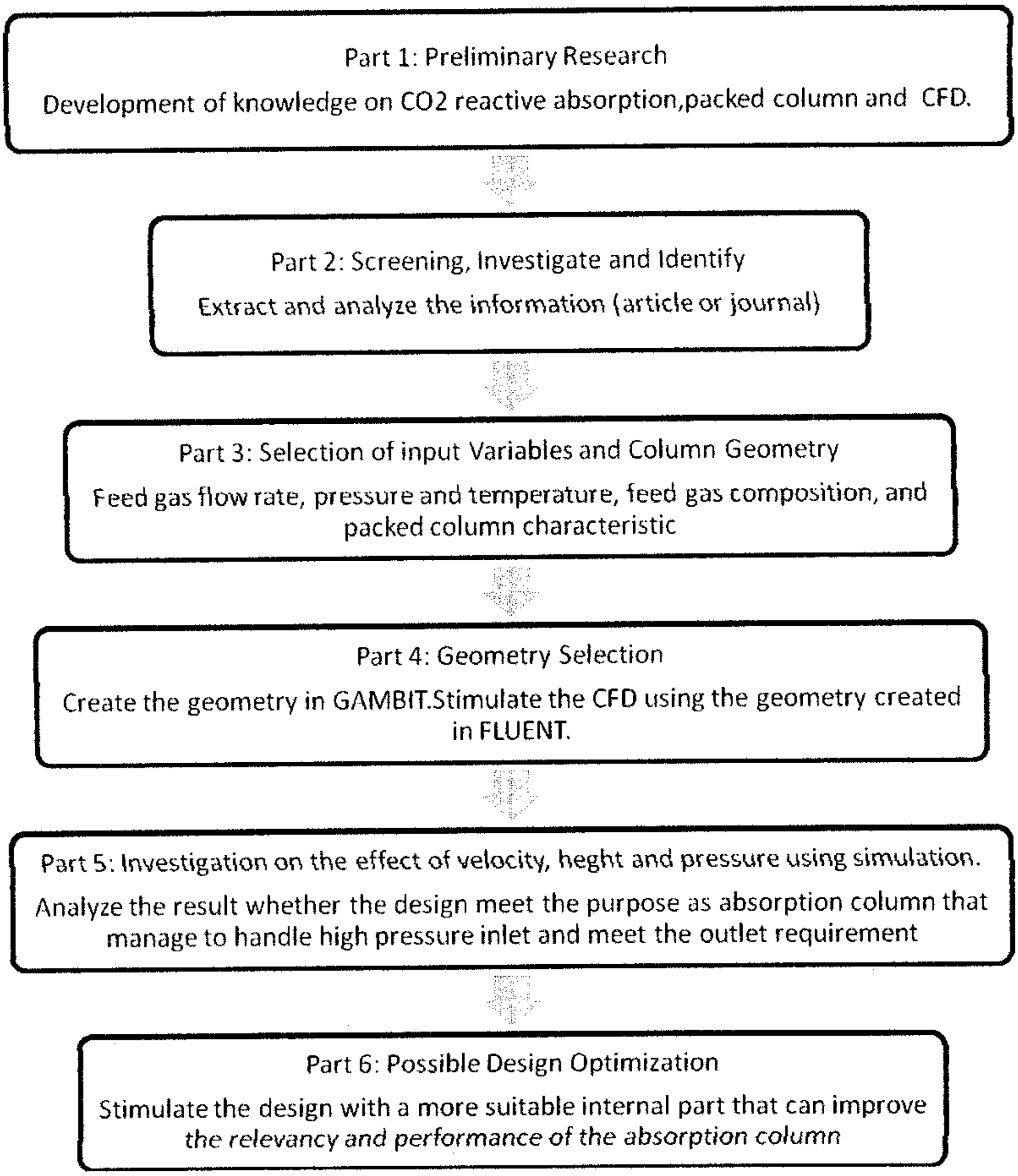


Figure 4 Flowchart of the Research Methodology

CHAPTER 4

RESULT AND DISCUSSION

4.1. GEOMETRY CREATION IN GAMBIT

The designs are developed step by step, starting from the hollow cylinder and then developed as a column with scaled diameter and height. Where as the distance from the nozzle to the sump and the nozzle from the packing are adapted by ratio equation from M.Werli et al. [24] Table 7 below shows the correlation used on geometrical and physical parameters in creating the column in GAMBIT (refer to Figure), the value are taken from previous work by M.Wehrli et al. Value for H and S for this study is H=0.38 and S=0.34. [24]

Table 7 Dimensionless parameter used in physical parameter of the column.

Definition	Description
H= *H/D*	Normalize clearance nozzle - packing
S= *Hs/D*	Normalize clearance nozzle – sump

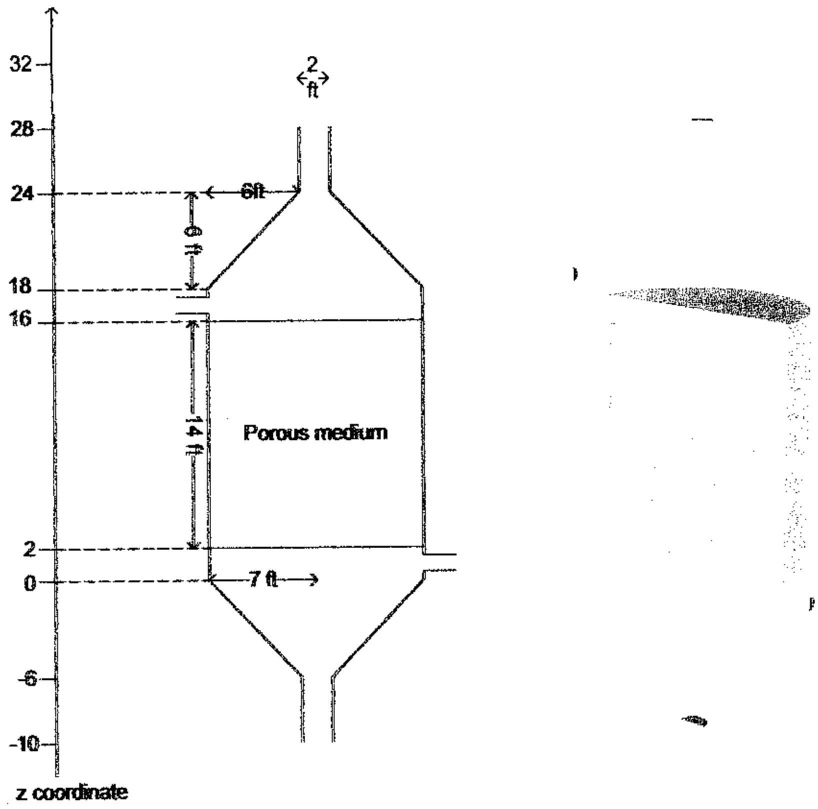
D= diameter of the column

H= distance between nozzle and the packing

Hs= distance between the nozzle and the sump.

As for initial simulation, pressure based solver are used in order to investigate the uniformity of gas inlet in this packed column. These part of simulation only considering gas flow only since the liquid influence is indirectly taken into account via pressure drop laws in the porous zone that represent the packing area.The specification of the porous zone are adapted from study of meso scale done by previous research with laminar flow and 0.95 packing porosity.[17]

Figure 5 Initial design develops in GAMBIT with z coordinate and figure with a colored boundary.



4.1.1. Boundary Condition

The boundaries condition of flow in packed column is specify as Table 8.The inlet both for both gas and liquid are specify as velocity inlet. Simulation on the gas phase boundary condition, the velocity gas at range of 22.6 m/s to 1.542 m/s are specified. After the volume fraction as are specify in this simulation gas, for the gas phase flow simulation, and then both gas and liquid for later study on the both phase. Then both outlets are specified as pressure outlet since the solver chosen for initial iteration is Pressure-based model.

Then the column wall are specify as a nonslip boundary condition. Then to include the packing area in the column, continuum type of packing are that are presented as a fluid.

Table 8 Boundary condition of the geometry. (the color map are referring to Figure 5)

Color coordination	**Boundary** ***Position***	**Boundary Condition (BC)**
Inlet (Blue)	Gas inlet (bottom column) Liquid inlet (Top column)	Velocity inlet
Outlet (Red)	Gas outlet (Top column) *Liquid outlet (Bottom column)*	Pressure outlet (Pressured –based *model)*
Porous medium (yellow and green)	Porous medium with 0.9 porosity (Middle column)	Symmetry, no shear
Walls (White)	Column wall, nozzle wall	Non –slip wall Adiabatic for mass and energy, and turbulence flow

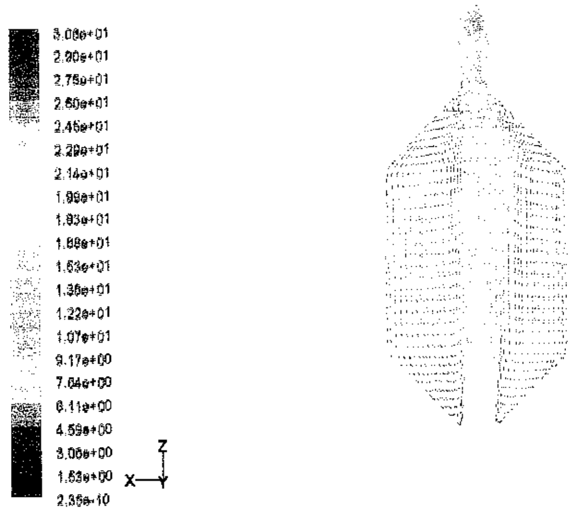
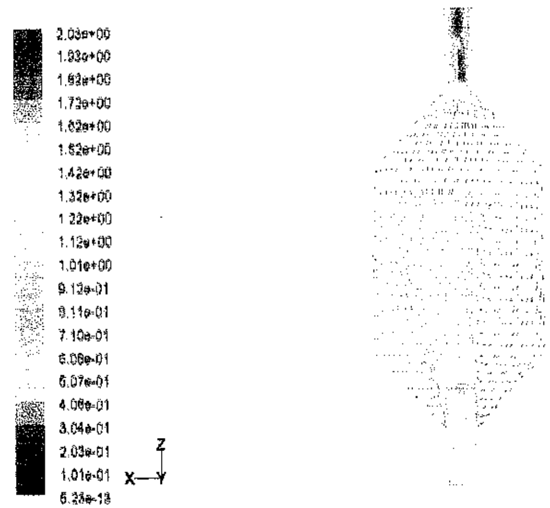
4.2. RESULTS AND DISCUSSION

Comparison of velocity profile and pressure profile at gas inlet velocity of 22.6 m/s and 1.546 m/s.

Table 9 Constant parameter use in simulation invertigate the effect of gas velocity.

Parameter	Dimension
Height of packing area(ft)	14
Diameter of column(ft)	7
Pressure (bar)	1

Table 10 Color map of velocity vector of velocity magnitude at axial direction (y=0) at inlet gas velocity of 22.6 m/s and 1.546 m/s at constant pressure of 1 bar

Velocity	22.6 m/s	1.546 m/s
Velocity vector of velocity magnitude in axial direction where y=0		
Observation	Circulation velocity vector at the packing area	More uniform velocity vector

The simulation continues by using velocity 1.546 m/s. The simulations are then continued with 17 ft height packing are and operation condition are varied by increasing the operating pressure. The velocity and pressure profile are show as below. The velocity and pressure profile are observe as the pressure increase.

Table 11 Parameter used as an input in order to investigate the effect of operating pressure.

Parameter	Dimension
Height of packing area(ft)	17
Diameter of column(ft)	7
Inlet velocity (m/s)	1.524

Table 12 Velocity vector of velocity magnitude in axial direction (y=0) for 1- 30 bar operating pressure at constant velocity of 1.546 m/s

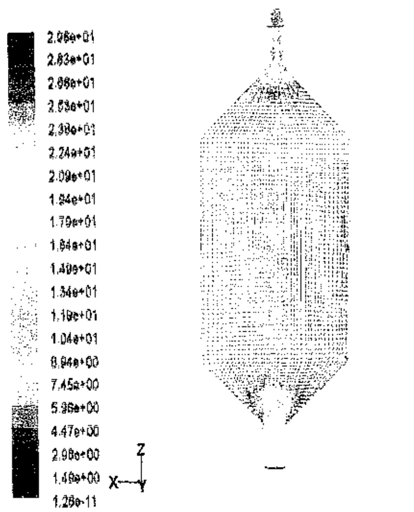
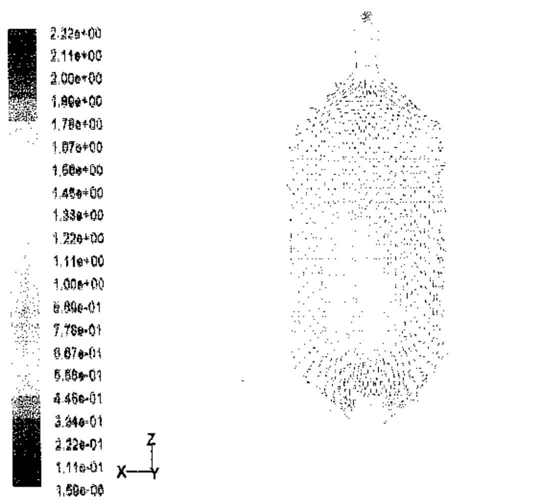
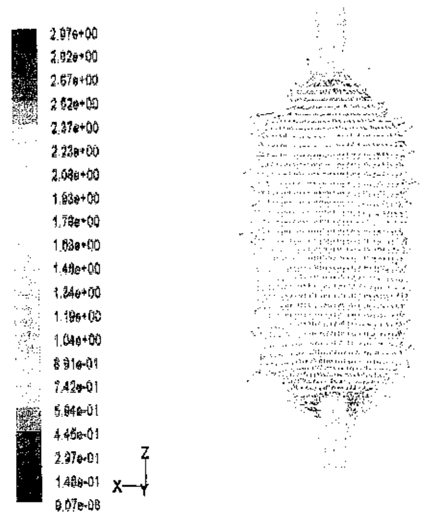
Pressure	1 bar	10 bar	30 bar
Velocity vector of velocity magnitude in axial direction y=0.			
Observation	Minimal circulation flow	Circulation flow at the early packing area.	Increase circulation flow at the packing area

Table 13 Velocity vector of velocity magnitude in axial direction (y=0) for 50 and 80 bar operating pressure at constant velocity of 1.546 m/s

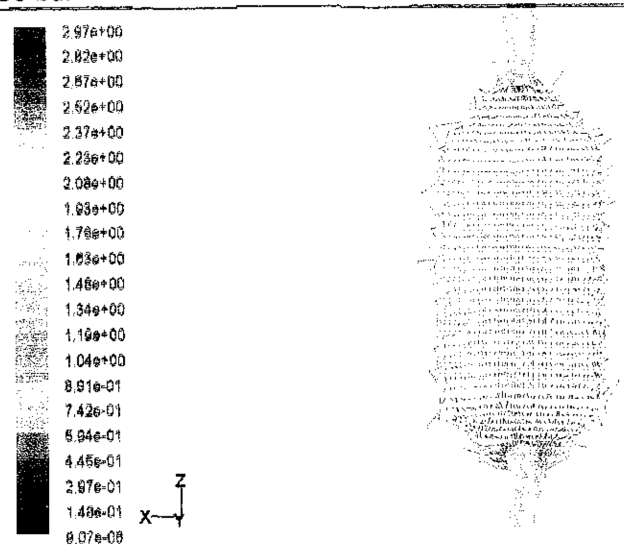
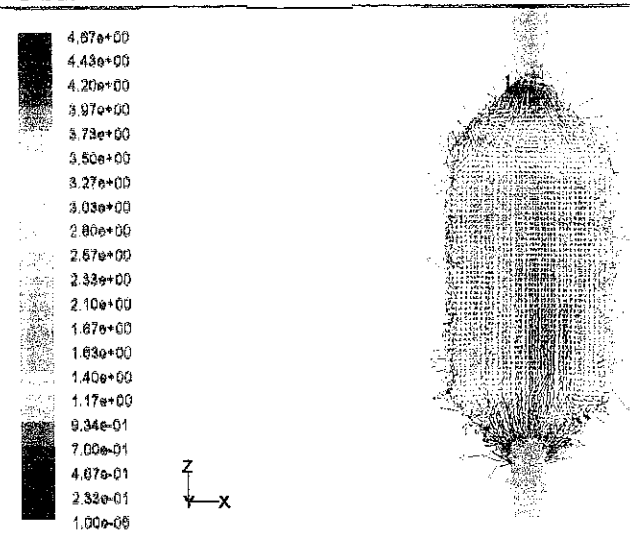
Pressure	50 bar	80 bar
Velocity vector of velocity magnitude in axial direction y=0.		
Observation	Non-uniform vector shown on the wall column.	Increase non-uniform vector on the wall column

Table 14 Velocity magnitude profile for 1 - 30 bar with constant inlet gas velocity of 1.546 m/s

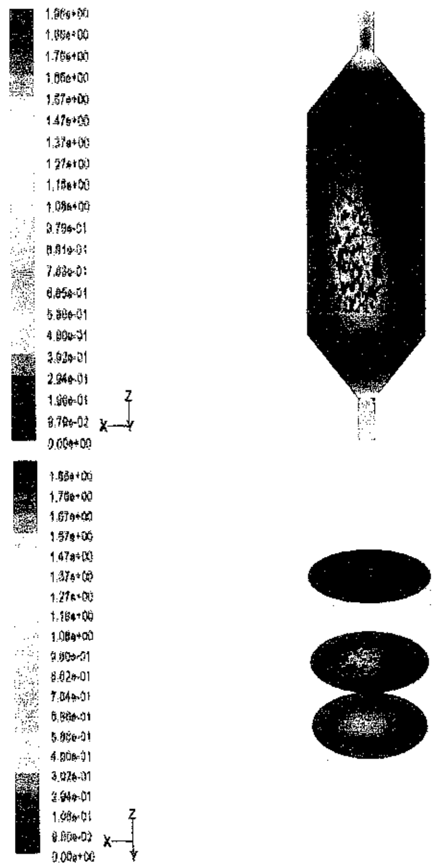
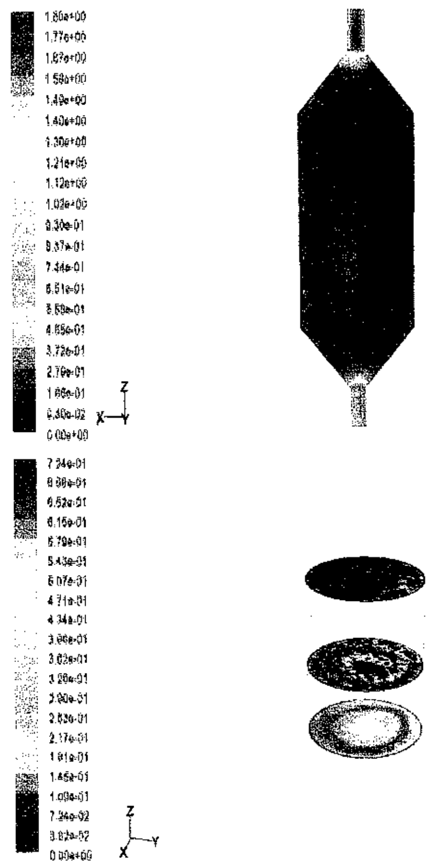
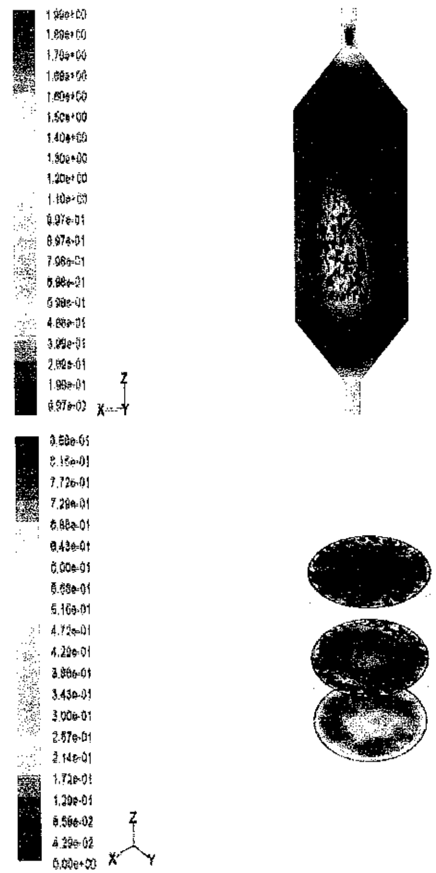
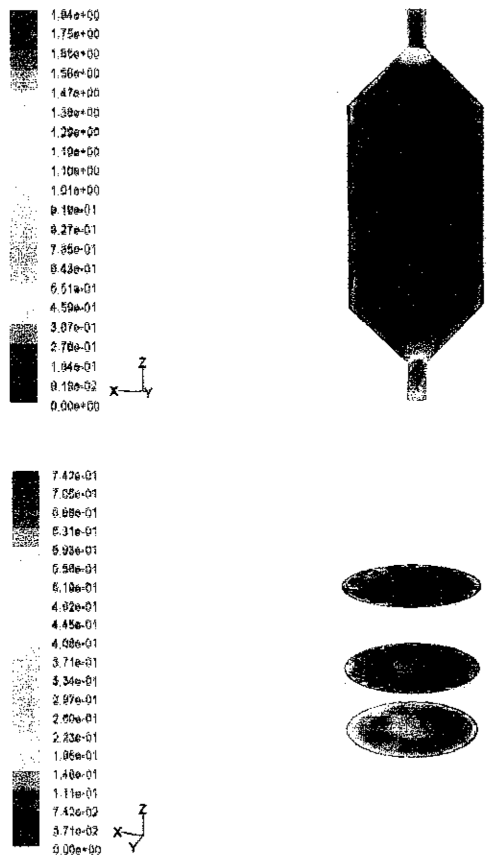
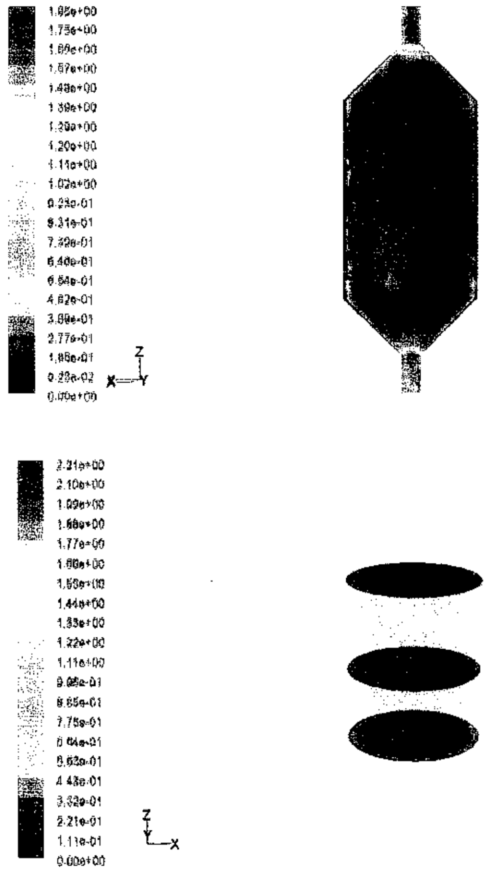
Pressure	1bar	10 bar	30 bar
Color map of velocity magnitude at axial plane (y=0) and at radial plane (porous-in z=2,middle porous z=9.5, porous-out z=19)			
Observation	Velocity are higher in the middle of the packing	High velocity in the middle of the packing.	Velocity magnitude higher at wall and in the middle of the packing.

Table 15 Velocity magnitude profile for 50 and 80 bar with constant inlet gas velocity of 1.546 m/s

Pressure	50 bar	80 bar
Color map of velocity magnitude at axial plane (y=0) and at radial plane (porous-in z=2, middle porous z=9.5, porous-out z=19)		
Observation	Velocity relatively low throughout the packing area and higher at the column wall.	Velocity magnitude increases at the column wall

4.2.1. Profile of velocity vector and velocity magnitude contour

The important cause for the liquid spreading is the unstable turbulent flow; therefore it is important to obtain uniform turbulent flow, before introducing the liquid inlet. In the simulation with packing height of 14 ft, as shown in Table 10, the gas velocity vector profile at 1.546 m/s shows a more uniform flow at the packing area compare to the profile at 22.6 m/s. Next, when the height of the packing area increases to 17ft, more uniform distribution can be observe. This shows that the height of packing area as well as the velocity on the gas phase has significant effect on the gas flow distribution. Having the ability to capture radial and axial variation in the flow the model can predict the efficiency based on detail local condition. At 10 Bar it can be observe that strongly heterogeneous at the early of the packing zone and uniform at end of the packing. In the other hand, at 80 Bar operating pressure, almost homogeneous velocity magnitude with increase of velocity magnitude at the wall area. However, the improvement on the gas distribution can be solve by considering the type of gas distributor Referring back to work done by M.Wehrli et al in the study of gas inlet distributor, vapor horn might be the most suitable gas distributor for this current study.For future work, the height of packing has to be increase in order to improve gas distribution in the column.

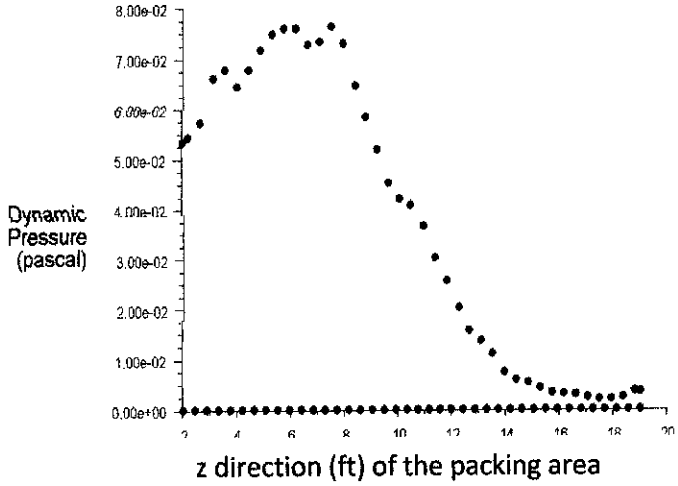


Figure 6 Plot of dynamic pressure profile throughout the packing area at 30 bar

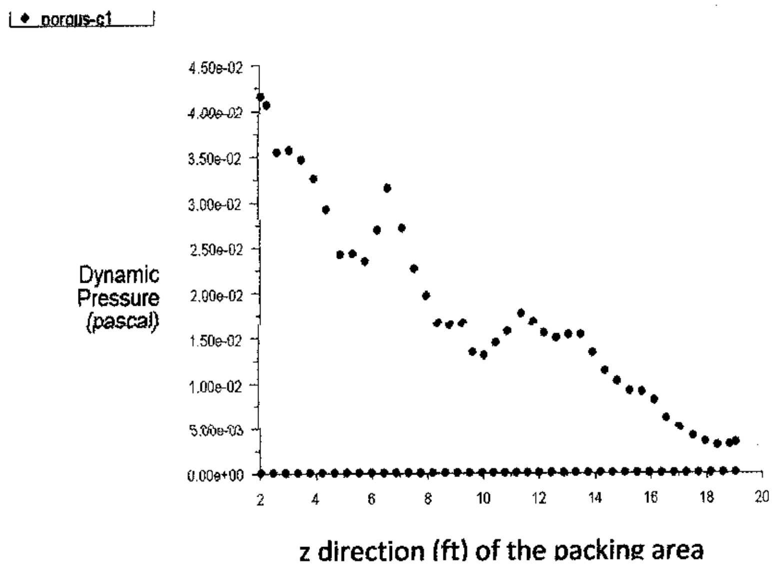


Figure 7 Plot of dynamic pressure profile throughout the packing area at 50 bar

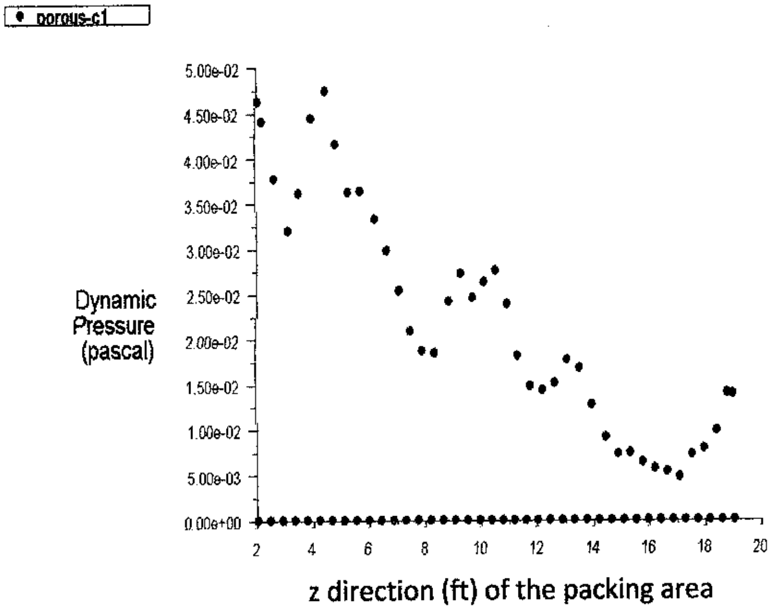


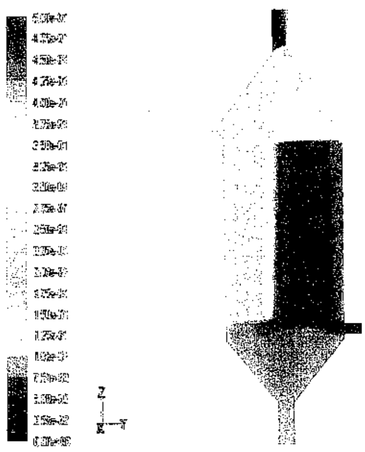
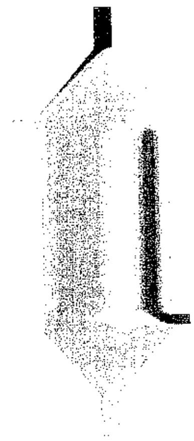
Figure 8 Plot of dynamic pressure profile throughout the packing area at 50 bar

4.2.2. Dynamic pressure drop

The pressure drop is one of the important design parameter. The dynamic pressure, along with the static pressure and the pressure due to elevation, is used in Bernoulli's principle as an energy balance. Present graph of dynamic pressure throughout the column shows a decrease of pressure as the gas flow upward the packing area. However, it can be observe that as the operating pressure of the simulation increased from 50 bar to 80 bar, the graph shows oscillating plot with not much different in pressure drop range. It can be concluded that pressure drop is not affected in high operating pressure. In order to validate this conclusion, an extended work has to be done on determining the pressure drop at higher operating pressure. In addition to that, more accurate result on pressure drop can be obtain if the flow rate of liquid phase and flow rate of gas (F-factor) are taken into consideration in this study.

The concentration of CO_2 in this simulation indicate by the mass fraction profile throughout the column. The investigating of the effect of operating pressure are done by using mixture solver at 1 Bar and 80 Bar operating pressure, gas velocity is 1.546 m/s and liquid loading is 50 m3/m2/h.[17] The result are shown below.

Table 16 Contour shows of concentration on simulation at 1 and 80 bar with mixture multiphase model.

Pressure	1 bar	80 bar
Color map of mass fraction of CO_2 throughout the column. **Temperature, gas inlet velocity, liquid loading is constant for both simulations.**		
Observation	Two regime of CO_2 mass fraction can be observe from the profile.	Contour of mass fraction of CO_2 varies only at the gas outlet.

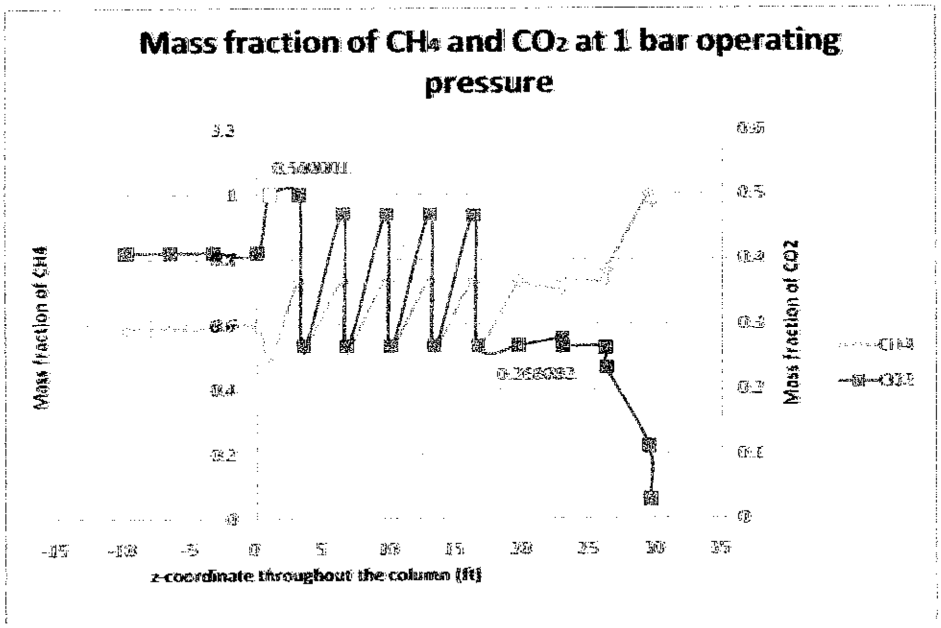


Figure 9 Graph shows the mass fraction of CO2 and CH4 at 1 Bar operating pressure.

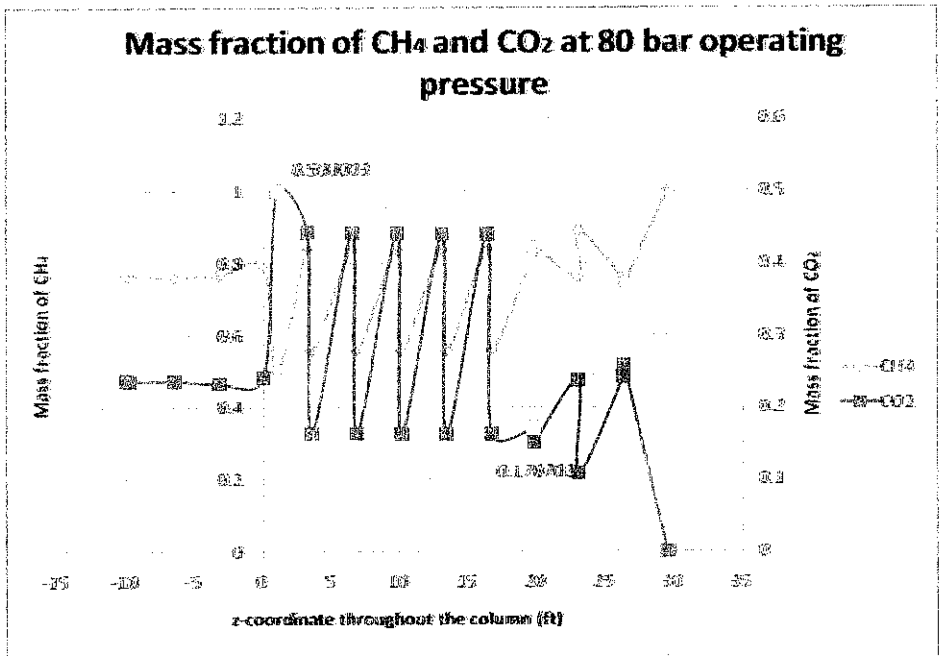
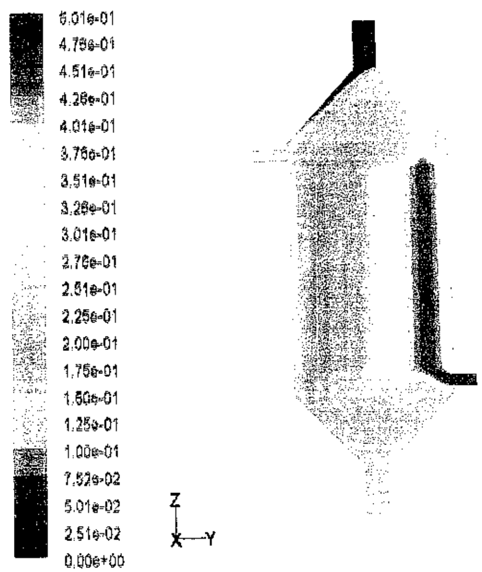
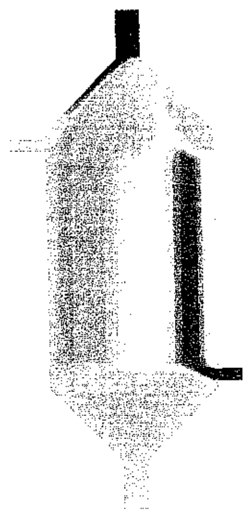
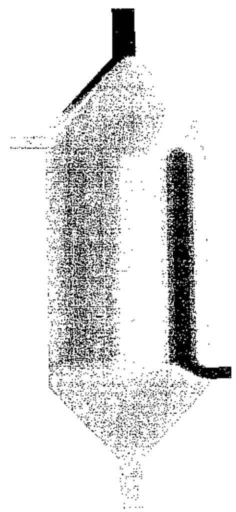


Figure 10 Graph shows the mass fraction of CO2 and CH4 at 80 Bar operating pressure.

Table 17 Contour of mass fraction of CO_2 at 0.01389 m/s , 0.02778 m/s and 0.04167 m/s liquid inlet velocity.

Liquid velocity (m/s)	0.01389 (50 m3 m-2 h-1)	0.02778 (100 m3 m-2 h-1)	0.04167 (150 m3 m-2 h-1)
Contour of mass fraction of CO2 in gas phase.			

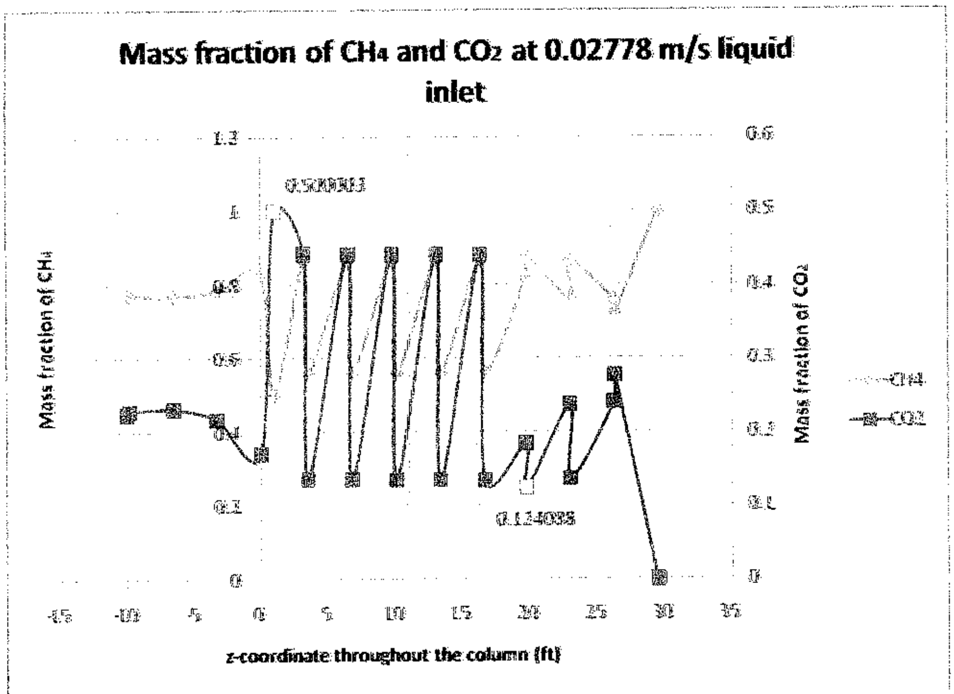


Figure 11 Graph shows the mass fraction of CO2 and CH4 at 80 Bar operating pressure with 0.02778 m/s liquid inlet velocity.

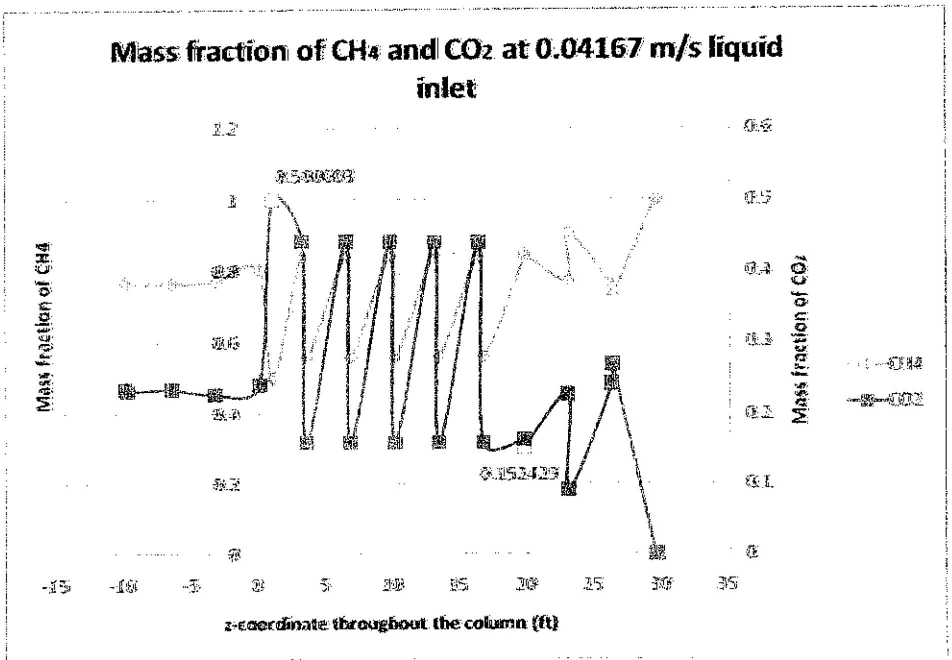


Figure 12 Graph shows the mass fraction of CO2 and CH4 at 80 Bar operating pressure with 0.04167m/s liquid inlet velocity.

4.2.3. Concentration

In this research, concentration of CO_2 is specified to be 50% at the inlet gas. CFD has ability to analyst the interface mass transfers parameters and the mass dispersion coefficient in this packed bed. However, the closure model that has the drag force, body forces, interface mass transfer, phase dispersion and species dispersion for this specific special solvent and feed composition has to be determined. If the process of the whole treatment system is consider, the CO_2 concentration can be handle by number of amine circulation rate. However, the study on mass transfer are done for 1 and 80 bar operating pressure; the mass fraction profile in 1 bar operating pressure can be observe from Table 16 where the mass fraction of CO_2 are changing from 0.5 to 0 trough out the column . From the contour profile, two region profile with different mass fraction can be observed.Refering to the nature of liquid in the cocurrent flow of gas and liquid the liquid flow has the tendency to flow at wall nearby.[26] Refering to both Figure 9 and 10, the reduction of CO2 mass fraction are greater in 80 bar operating pressure.the point of mass fraction are taken at z=1ft and z=20ft, these are the point where the gas inlet and liquid inlet are situated. Other way to investigate accuracy of the result is to compare the trend of separation power of the structured packing. Figure 13 shows the gas loading versus pressure drop.

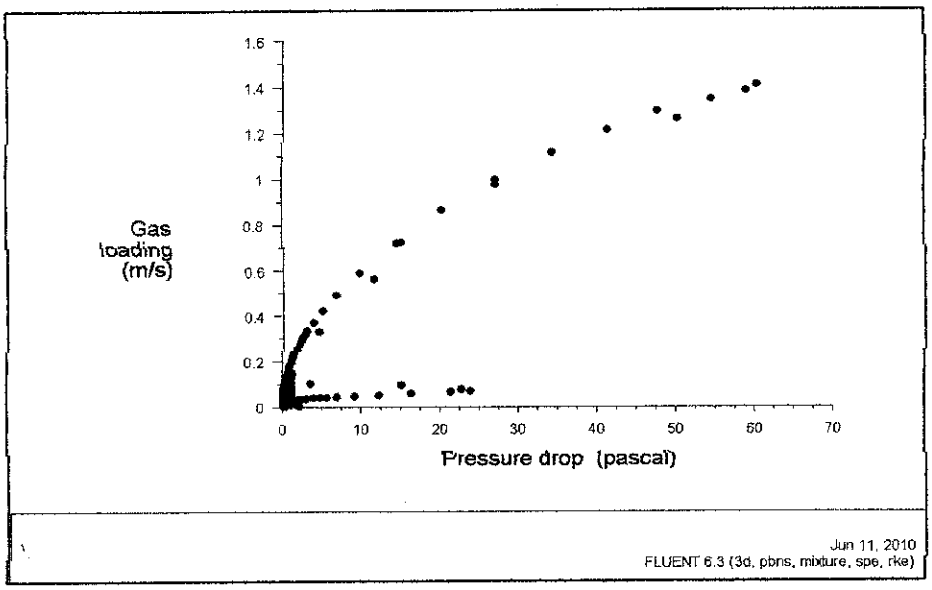


Figure 13 Separation power at 100 m3/m2 liquid loading, 80 bar operating pressure and 1.546 m/s gas velocity.

The trend of the separation power is similar to the trend obtain in the study investigating the performance of structured packing (Mellapak plus 252 Y) which is done in meso scale.[27]

4.3. RECOMMENDATION

In two phase simulation of countercurrent process, the predicted result can be improve by specifying the liquid phase temperature at least 10°C higher than the gas phase temperature because the vapor may condense entrained at the liquid outlet. The accuracy of CFD in solving design problem is depending on the mesh generated in the associated software for the geometry and the chosen solver in FLUENT. In this simulation GAMBIT are used in mesh generation process. Therefore in order to have accurate result on CFD, mesh created has to be suitable with shape of the design otherwise CFD will not be able to iterate accurate result for the geometry design. As for that, study on the meshing itself are recommended in order to get accurate result and speed up the design process since design process involving many alteration before optimum design can be achieve. Next, in future study, higher order discretization solver need to be consider for improved accuracy. Furthermore, in case of problem iteration involving mass transfer, two dimensional simulations will be more possible to iterate since three dimensional flow are consider as complex model when its involve two phase problem.[24]

CHAPTER 5

ECONOMIC EVALUATION ON THE COLUMN

5.1. COST OF CO_2 ABSORPTION PACKED COLUMN

Cost correlation based on the book Chemical Process Equipment Selection and Design (Couper J.R. et al)

D:	7	ft
L:	41	ft
Cp:	7.65	
f1:	1.7	(stainless steel 304)
W:	200000	lb
Vp:	2616.9	m3

$V_p = Volume\ of\ packing$

$C_p = Cost\ of\ packing\ \$/cuft$

- $C_b = 1.218\exp[6.629 + 0.1826 (\ln W) + 0.02297 (\ln W)^2]$
  $Cb = \$262314$

- $C_{p1} = 300D^{0.7396}L^{0.7068}$
  $C_{p1} = \$17461$

Total cost of packed column

$TOTAL\ COST, C = 1.218[f_1C_b + V_pC_p + C_{p1}]$

$TOTAL\ COST, C = 1.218[(1.7)(262314) + (2616.9)(7.65) + (17461)]$

Total Cost, $\boldsymbol{C}$ = **\$ 588 798.35** = RM 1 902 270

CHAPTER 6

CONCLUSION

In designing packed absorption column, the strategy of calculation approach using CFD the design optimization can be done by looking at the packing area as one porous medium, this is the part were fluid distribution, height, radius, and other column characteristic are analyze. Structured packing was use as a base case since mention that it has higher geometric surface [16][23]. The focus is on optimizing design of the packed column to be able to operate at high operating pressure. With porosity specified as 0.9, investigations are done on the effect of gas velocity, height of packing area and operating pressure to the gas distribution and also the effect of operating pressure and liquid loading to the CO_2 mass fraction reduction. The study with 17 ft height of packing area with porosity specified as 0.9 ,can be conclude that best result of gas distribution achieve at 1.546 m/s at 80 bar and higher reduction CO2 mass fraction can be observe at 100 m^3/m^2h liquid loading. In this study also prove that high operating pressure favor absorption process.[8][24]Target on decreasing value of CO_2 concentration throughout the column are achieve, yet further revision has to be done in order prove the finding.. As a conclusion, packing in the packed column generally gives a uniform flow for gas liquid interaction in high pressure environment and further research has to be done to increase the absorption efficiency.

BIBLIOGRAPHY

[1] *Brunei LNG Sdn Bhd.* (2008-2010). Retrieved January 23, 2010, from LNG Worldwide: http://www.blng.com.bn/worldwide_index.htm

[2] *Natural Gas and The Enviroment* . (2004). Retrieved December 2, 2009, from NaturalGas.org: http://www.naturalgas.org/environment/naturalgas.asp

[3] J.M. Klinkenbijl, M. H. (1999). *Gas Pre-Treatment and their Impact on Liquefaction.* Amsterdam: Shell International Oil Products,Research & Technology Centre Amsterdam.

[4] JAEGER Product Inc. (-). Srubbing Pollutant from Vent Stream. *Gas srubbing* , -.

[5] Subramaniam, R. (n.d.). *Flow through Packed Beds and Fluidized Beds.* Retrieved October 30, 2009, from www.clarkson.edu web site : http://web2.clarkson.edu/projects/subramanian/ch301/notes/packfluidbed.pdf

[6] GTI. (n.d.). Retrieved from http://www.gastechnology.org: http://www.gastechnology.org/webroot/app/xn/xd.aspx?it=enweb&xd=1researchcap%5C1_8gasificationandgasprocessing%5Cnaturalgascleanupsystems.xml

[7] KOCH-GLITSCH. (2003). INTALOX Packed Tower Systems, Structured Packing. *The Fundamental Driving Force, in Mass Transfer Technology* . United States of America: KOCH-GLITSCH.

[8] T.K Sherwood, R. P. (1952). *Absorption and Extraction.* . Newyork: McGraw-Hill.

[9] Adisorn Aroonwilas, P. T. (1999). Mechanistic model for prediction of structured packing mass transfer. *Chemical Engineering Science* , 3651-3663.

[10] Buch, C. (2004, December 15). *Zero Emission Resource Organization* . Retrieved September 1, 2009, from ZERO Web site: http://www.zero.no/

[11] R.M. Counce, J. P. (1986). Designing packed-tower wet scrubbers:emphasis on nitrogen oxides,. In N.P. Cheremisinoff (Ed.), *Handbook of Heat and Mass Transfer, vol. 2* (pp. 953-966.). Houston: Gulf Publ Comp Book Division.

[12] Manuel A. Pacheco, G. T. (1998). Rate-Based Modeling of Reactive Absorption of CO2 and H2S into Aqueous Methyldiethanolamine. *Ind. Eng. Chem. Res.* , 4107-4117.

[13] Jian CHEN, W. F. (2006). *Study on solvent and process simulation for CO2 absorption.* Beijing: State Key Laboratory of Chemical Engineering,Tsinghua University.

[14] C. Noeres, E. K. (2001). Modelling of reactive separation processes: reactive absorption and. *Chemical Engineering and Processing* , 157-178.

[15] G.B. Liua, K. X. (2006). Simulations of chemical absorption in pilot-scale and industrial-scale packed. *Chemical Engineering Science* , 6511 – 6529.

[16] Raynal, L. (2005). Use of CFD for application in the oil and gas industry. *Ecoulements diphasiques* , 75-78.

[17] Royon-Lebeaud, L. R. (2007). A multiscale approach for CFD calculations of gas-liquid flow within large size column equipped with structured packing. *Chemical Engneering Science* , 7169-7204.

[18] L.Raynal, F. R.-L. (2009). Use of CFD for CO2 absorber optimum design: from local scale to large industrial scale. *Energy Procedia* , 917-924.

[19] Raynal, S.-B. a. (2005). Influence of the viscosity on the liquid hold-up in trickle-bed reactors with structured packings. *Catalysis Today* , 673-679.

[20] M.Wehrli, S. R. (n.d.). Influence of Vapor Feed Design on The Flow Distribution. *Sulzer Chemtech* .

[21] Kevin Bennett, M. P. (2003). Efficiency Benefits of High Performance. *Texas Technology Showcase 2003 Separation & Distillation Technology Session.* Texas: Sulzer Chemtech.

[22] L.Spiegel, P. a. (1992). Hold-up of Mellapak structured packings. *Chem Eng Process* , 119-124.

[23] Smith, R. (2005). *Chemical Process ; Design and Integration* . University of Manchester: Wiley.

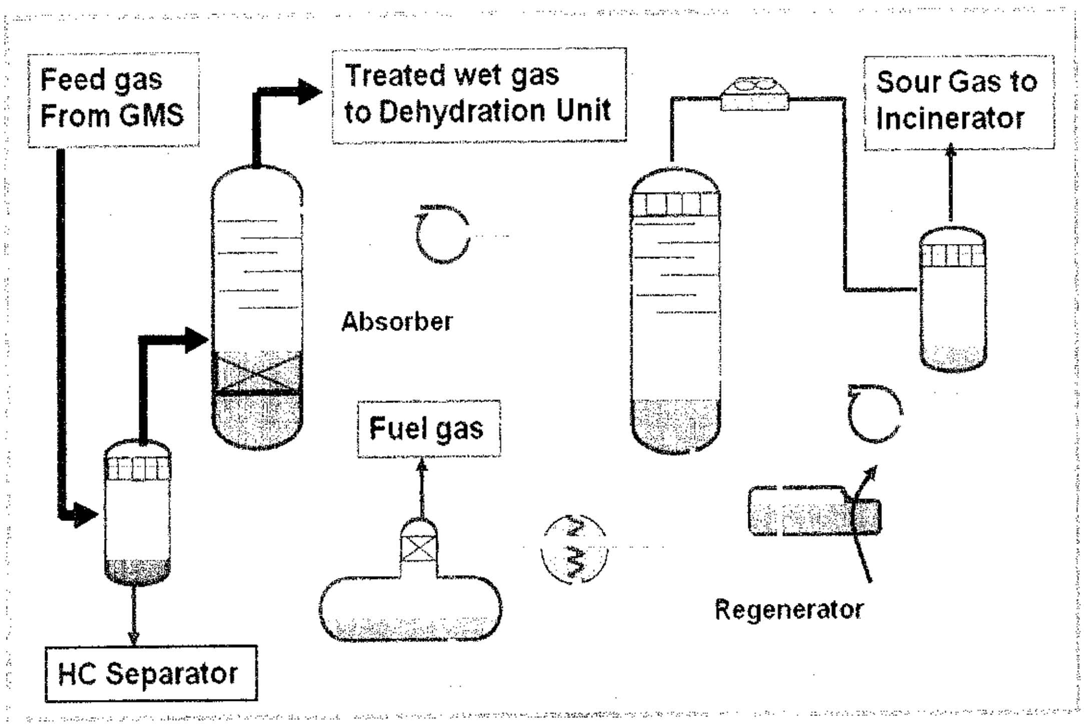
[24] Kiong, C. T. (1998). *MLNG Process Overview.* Bintulu: MLNG HRD Centre.

[25] Masudi, A. F. (2008). *Performance Study Of Membrane Unit For C02 Removal From Natural Gas.* Ipoh: Univesity Technology Of PETRONAS.

[26] F.H. Yin, C. S. (Alberta). CFD Modeling of Mass Transfer Process in Randomly Packed Distillation Coumns. *Chemical and Materials Engineering* , 1369-1380.

APPENDIX 1

Figure 14 Simplified Flow Diagram of Acid Gas Removal Unit.



APPENDIX 2

Table 18 Integrated Sarawak Offshore Gas Supply Composition

	BN	BY	D35	E11	E11W	E11SC	E8	F13SC
GHV(Btu/scf)	1240	1145	1270	946	985	1069	1147	1147
CO2(%mole)	1.46	1.73	1.73	7.35	1.00	1.00	1.73	1.00
N2	0.45	0.36	0.36	1.52	1.19	1.19	1.31	1.19
C1	83.01	88.46	88.46	85.35	92.79	92.79	85.74	92.79
C2	6.49	4.17	4.17	2.55	2.65	2.65	5.10	2.65
C3	4.78	2.29	2.29	1.47	1.42	1.42	3.44	1.42
iC4	0.98	0.46	0.46	0.35	0.29	0.29	0.73	0.29
nC4	1.33	0.51	0.51	0.37	0.32	0.32	0.94	0.32
iC5	0.46	0.12	0.12	0.23	0.12	0.12	0.38	0.12
nC5	0.36	0.19	0.19	0.15	0.08	0.08	0.28	0.08
C6+	0.66	1.73	1.73	0.68	0.14	0.14	0.35	0.14
H2S{PPM}	0	0	0	14	0.2	0.2	10	0.2
Flow(mmscfd)	90	10	15	250	100	130	620	200

Table 19 Integrated Sarawak Offshore Gas Supply Composition

	F13E	F13W	F6	F23	M1	M3	M3S	B11
GHV(Btu/scf)	834	946	1184	1168	1099	1170	1087	950
CO2(%mole)	17.36	13.49	1.63	2.25	3.03	7.55	7.35	7.00
N2	2.43	2.39	0.65	0.84	0.30	0.59	0.55	1.18
C1	72.45	79.83	88.71	88.68	86.39	80.73	76.66	86.73
C2	1.99	2.24	4.23	3.59	5.13	5.29	5.34	2.47
C3	1.02	1.15	2.81	2.69	3.00	3.56	4.00	1.44
iC4	0.23	0.27	0.67	0.69	0.74	0.89	1.19	0.35
nC4	0.23	0.27	0.60	0.57	0.66	0.72	1.08	0.37
iC5	0.10	0.12	0.23	0.21	0.23	0.27	0.68	0.14
nC5	0.06	0.08	0.14	0.12	0.14	0.14	0.32	0.18
C6+	0.08	0.20	0.34	0.36	0.37	0.14	0.58	0.15
H2S{PPM}	10	10	8	15	10	0	90	1200

Table 20 Integrated Sarawak Offshore Gas Supply Composition

	B12	PC4	JN	HL	SERAI	SADERI	G7
GHV(Btu/scf)	1065	1168	1117	1260	1155	1068	1016
CO2(%mole)	4.40	5.47	3.03	1.04	3.73	5.27	9.46
N2	0.87	1.58	0.30	0.28	0.23	0.22	0.28
C1	89.93	84.73	86.39	87.25	81.12	85.40	81.34
C2	2.48	2.76	5.13	3.75	7.29	5.17	5.35
C3	1.17	1.86	3.00	4.47	4.72	2.19	1.81
iC4	0.26	0.45	0.74	0.95	1.23	0.60	0.44
nC4	0.41	0.19	0.66	1.20	0.93	0.54	0.36
iC5	0.35	0.13	0.23	0.37	0.28	0.26	0.18
nC5	0.11	0.15	0.14	0.26	0.15	0.16	0.10
C6+	0.02	0.06	0.37	0.44	0.32	0.16	0.16
H2S{PPM}	65	0	10	0	0	5	0
Flow(mmscfd)	200	150	760	290	100	300	100

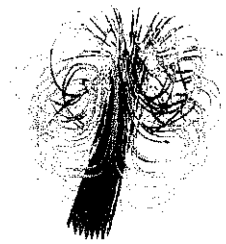
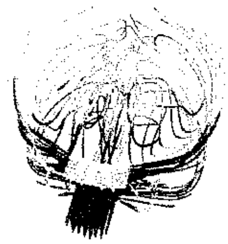
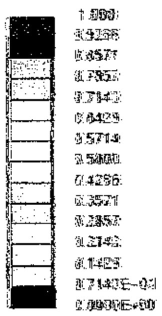
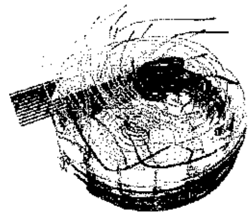
APPENDIX 3

FYP	FYP 1														FYP 2													
WEEKS	1	2	3	4	5	6	7	8	9	10	11	12	13	14	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Project Familiarization																												
Literature review (Research)																												
Preliminary report																												
Software familiarization																												
Interim report																												
Presentation FYP 1																												
Generate design in GAMBIT																												
Run simulation in FLUENT																												
Cost estimation of the design																												
Poster presentation																												
Dissertation report																												

Figure 15 Gantt Chart of Final Year Project

APPENDIX 4

Table 21 Comparison results between distributors.

Type of inlet	Geometry	Distribution of vertical velocity ob horizontal plane	Streamline (colored with local speed)
Standard inlet ***Kp*=0.82**			
Orifice baffle ***Kp*=0.37**			
Vapor horn ***Kp*=0.25**		  **vertical velocity in m/s**	 **Red is the highest velocity (4 m/s and more), blue is the lowest (0 m/s).**