CFD Modelling of Ultrasonic Wave Propagation in Non-Consolidated Porous Media

Wan Muhammad Hafidz Bin Wan Alias, Muhammad Shafiq bin Mat Shayuti

Faculty of Chemical Engineering, Universiti Teknologi Mara

Abstract— **Extraction of sand from oil sand using ultrasonic soundwaves and the effect of different frequency to liquid being exposed to sound wave by transducer is investigate in this paper. A cleaning reactor was designed with the ultrasound transducer installed at the bottom of the reactor. Flow behavior of liquid is investigate using CFD calculation and simulation by ASNYS FLUENT software. Liquid in the reactor is exposed with non-audible sound waves with frequency of 68 kHz which from transducer installed. Various frequency also being investigates which 20kHz and 150 kHz to find the comparison between each of them in term of flow behavior, pressure and cavitation field. The modeling results show the pressure and cavitation fields for different frequencies. Velocity vectors are also used to investigate the mixing of the liquid exposing the ultrasound waves. The modelling shows different flow behavior, pressure and cavitation field from each frequency used.**

Keywords— ***Cavitation field, CFD modelling, flow behavior, pressure field, sound wave frequency***

I. Introduction

Cleaning the produce sand or oily sludge is one of the major solid waste in petroleum industry. Removing the producing sand not only can save the maintenance cost, but it also can reduce solid waste volume and reducing pollutant to environment. Ultrasonic cleaning attracts main company in oil and gas industry due to the precision cleaning thus shown strong promise. Before the ultrasonic cleaning technologies been used, producing sand usually treated using physicochemical, froth flotation, pyrolysis, oxidative thermal treatment and others. Ultrasonic cleaning sensor used nowadays facing problem such as to find suitable frequency used to maximize the cavitation in the cleaning reactor.

The viable disposal of producing sand from the oil and gas industry during petroleum transportation, storage, and refinery process is an around the world issue [1]. The sand producing sand can be removed by using ultrasonic cleaning technology. Produced sand also known as oily sludge needs to be removed to ensure the equipment used throughout the process always in its high efficiency. Many studies shown that ultrasonic cleaning could enhance the recovery rate of crude oil from oily sludge or oil sand [2][3]. Separation of crude oil and producing sand using ultrasonic standing wave length have cause wide attention for the past decades due to its advantages of efficiency and simplicity [4]. The ultrasound wave propagates in sand producing medium, it creates compressions and rarefactions[1]. The pressure cycles apply a positive weight on the fluid by pushing the particles together and the rarefaction cycle applies a negative weight by pulling the sand molecules from each other.

Ultrasonic cleaning can provide high degree of cleaning that solely used sound as its medium. High frequency electronic generator created high frequency wave and transmitted through water in a tank or reactor. High frequency of sound wave which is non- audible create a gentle scrubbing effect within the fluid thus provided rapid and complete cleaning, remove contaminant from all surface area being contacted. The effectiveness of ultrasonic cleaning made it be used in many industries such as heavy industry, food industry, medical industry and others. The differences of ultrasonic cleaning with conventional cleaning methods is the tiny bubbles penetrate nearly every corner of the equipment and surface. The bubble expansion and collapse cycle are very short time.

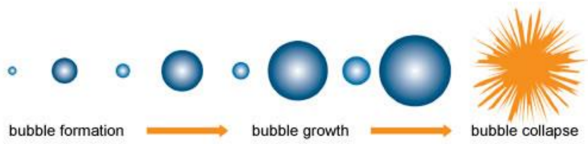

Figure 1: Expansion and collapse cycle of bubble in ultrasonic tank [19]

Cavitation is defined as formation of void within a liquid, and its subsequent behavior [5]. Cavitation bubbles are formed and grow when a liquid is put in known state of tension. Liquids, which unable to support shear stresses, can support compressive stresses, and for short periods, it also can support tensile stresses. Cavitation can also lead to erosion of tank while cleaning, but controlling the power and frequencies uses of the ultrasonic system. The power more spread out if the higher frequency used which can lead to more cleaning on the part being cleaned. Other than that, uses of higher frequency also produces smaller cavitation bubbles and can clean smaller particles.

Frequency of the sound wave will affect the cavity size and cleaning efficiency. Low frequencies will generate large but few cavities with high cleaning power. Meanwhile high frequencies generate huge number of small cavities and have good cleaning capability [6]. Transducer is part of ultrasonic cleaning reactor that emits frequency and the wave hits the liquid surface thus resulting produced active cavitation zone [7]. Cavitation can be observed from the CFD simulation which is at the low-pressure field area. Large number of cavitation bubble are formed in front of the transducer can act as barrier to transfer acoustic energy and will dampen the power transmission to the reactor [7].

Computational fluid dynamic or CFD is mathematical and computational models of flowing fluid to describe and predicts fluid response. CFD analysis can examine the fluid flow in accordance with its physical properties such as velocity, pressure, temperature, density and velocity conducted. Prototypical experimental results from CFD makes it an efficient, cost-effectives, tool for predicting system response under a broad range of operating conditions. The reliability of the CFD analysis is highly depends on the whole structure of the process thus verification of the mathematical is extremely important to ensure accurate solving the problem. CFD analysis is an interdisciplinary topic between numerical analysis, fluid mechanics and computer science. CFD analysis can delivered better prediction in a short time, design better, economical and can leads to shorter time buildings the parts.

Computational Fluid Dynamic (CFD)model is used to in this research to evaluate the ultrasonic wave propagation. It used to describe the behavior of flow flowing in the non-consolidated porous media used in this research. Simulation flow of fluid, pressure and cavitation field can be achieved by using CFD software [8]. There are three benefits of using CFD software, which is insight where CFD software can give deeper insight of all design inside it and evaluate its performance. CFD software also can benefit by foresight, predicting what will happen to the equipment used and CFD software can benefit due to its efficiency, by evaluating the design, operators can save lots of money and equipment improvement can be done in minimal time. In this research, by using CFD software, a model will be developed to evaluate the ultrasonic wave propagation in non-consolidated porous media [8].

Flow behavior, pressure field and cavitation field to the liquid being exposed by ultrasonic sound wave with 68kHz frequency is the main objective of this project. The result obtain is compared with different frequency of 20kHz and 150kHz based on its flow behavior, pressure and cavitation field.

II. METHODOLOGY

A. *CFD Modelling*

CFD simulations can be divided in three main stages: pre-processing; simulation; and post-processing. The pre-processing includes: geometry definition, mesh generation, boundary conditions definition and solver parameters establishment. The simulation stage consists of using the constructed model for running the simulations. The post processing is the analysis of the results (transient flow)

The geometry of the simulation area is shown in Fig. 2. This apparatus includes a vessel with a height of 10.8 cm and a diameter of 7.2 cm, and ultrasonic transducer with a diameter of 3.0 cm. Ultrasonic sources (ultrasound transducers) with frequencies of 68kHz are installed at the bottom of the vessel and are activated by an electrical actuator.

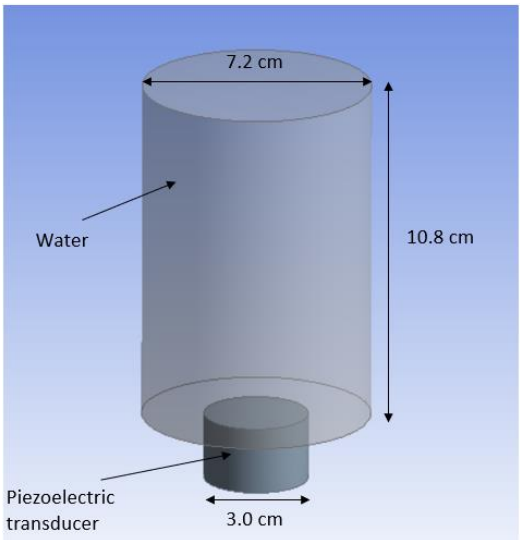

Figure 2: Schematic diagram and geometry of the apparatus.

The geometry was generated in ANSYS Fluent software and was meshed using the tetrahedral meshes. The ultrasound transducer was meshed using the small mesh size and the volume of the vessel was meshed as small as possible to show the proper behavior of the propagated wave inside the reactor.

The equations that describe flow properties and motion are numerically solved in each of the defined mesh cells. The final solution accuracy is strongly dependent on the number of cells in the mesh within computational domain. More accurate solution simulation can be achieved if larger number of cells generate in meshing. Boundary condition setup is established in the setup to established and describe the fluid flow physics.

In this study, numerical simulation is based on the computational fluid dynamics (CFD). ANSYS FLUENT [8] software is used to describe the behavior of the fluid flow using the Navier-Stokes and $\kappa - \varepsilon$ turbulent equations. Distribution of pressure, velocity, and vapor volume fraction were calculated by solving the conservation of mass, momentum and energy equations. In this model no slip velocity was considered.

The transducer is set as pressure inlet boundary condition in this project, pressure outlet is defined by top wall and other boundaries is defined were set as wall. In this project, pressure-velocity coupling is calculated by simple algorithm.

The pressure inlet value calculated when the transducer is active to stimulate the distribution of sound pressure and was set as boundary condition of transducer according to the following equation:

$$P_u = P_a \cos\left[\omega(t + z/C)\right] \tag{1}$$

$$P_a = \sqrt{2I\rho C} \tag{2}$$

$$I = P_a / A \tag{3}$$

Where:
Pu (Pa) = Sound pressure
Pa (Pa) = The amplitude of the sound pressure
ω (2πf) = Angular frequency (rad/s)
t (s) =Time
z (m) = Space coordinate
C (m/s) = Sound speed in water (1480 m/s)
I (W/m^2) = Sound intensity
Pus (W) = Ultrasonic power
A (m^2) = Ultrasonic transducer area
ρ (kg/m^3) = Density water (997 kg/m^3)

In this part of modeling, Eq. (1) is used as pressure inlet which is set for the boundary condition of transducer and constant pressure inlet boundary represent this condition in the CFD model. Top wall is defined as pressure outlet and remain open and others is defined as wall. The wall is considered as ‘non-slip condition’.

κ-ε standard turbulence model is used as viscous model. For this kind of simulation including the wave motion in a liquid, the time step of steady-state is assume zero (0)

III. RESULTS AND DISCUSSION

CFD results are presented in 2D pictures of the vertical cut on the domain under study (y=0). In this part of modeling, Eq. (1) is used as pressure inlet which is set for the boundary condition of transducer.

A. *Evaluation of the effect of ultrasonic frequency on the cavitation field*

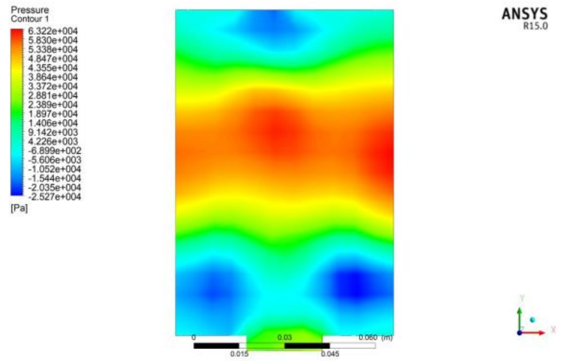

Figure 3: Pressure field for 24kHz

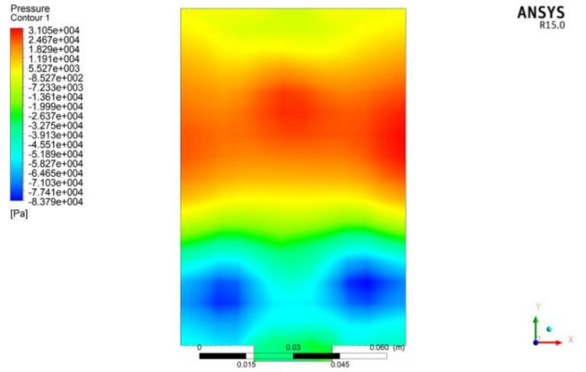

Figure 4: Pressure field for 68kHz

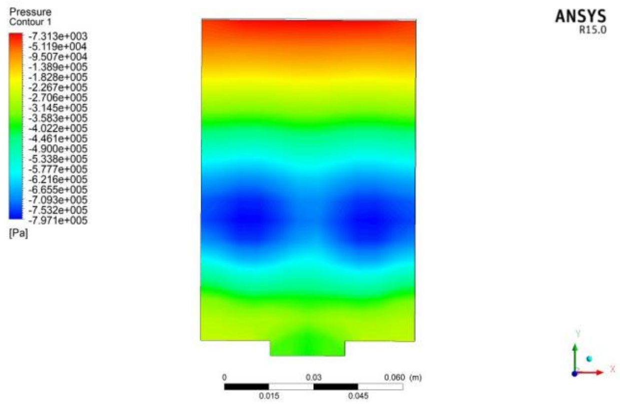

Figure 4: Pressure field for 150kHz

These figures above show how the wave will propagate when transducer is active. Low pressure regions showed are potentially the active sites for cavitation to take place [9].

For 24kHz and 68kHz, liquid pressure is low near the bottom of the reactor above the transducer. Then, from mid to top of the reactor, the pressure is more uniform for 24kHz and vary in pressure distribution for 68kHz. So, most of the cavitation activity occur at the bottom area of the reactor.

For 150kHz, it is observed that the cavitation area occurs in lower range and more uniform compare with the other two frequency. Observation can be made from the result obtain, at low frequency, 24kHz, the cavitation field is formed near the bottom of the reactor and expand the area of cavitation for 68kHz. Most uniform layer of cavitation field can be seen when sound wave from 150kHz is active.

Besides that, the area of cavitation field decreases with increasing the frequency of sound wave used. Using high frequency, although the area of cavitation is small, but the bubble of cavitation created is smaller compare to other frequency thus effective cleaning can be observe [10].

B. Evaluation of the effect of ultrasonic frequency on the flow behavior

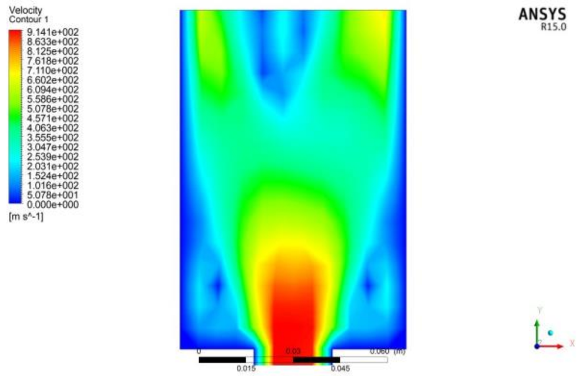

Figure 5: Flow distribution for 20kHz

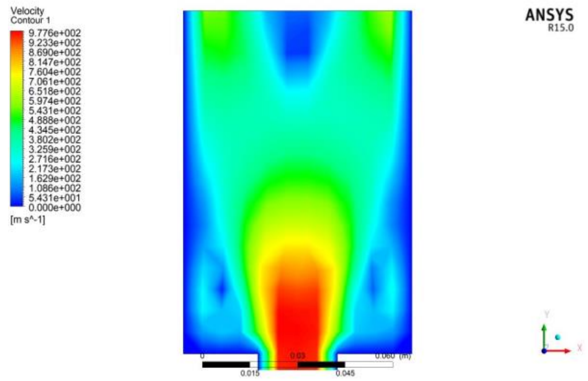

Figure 6: Flow distribution for 68kHz

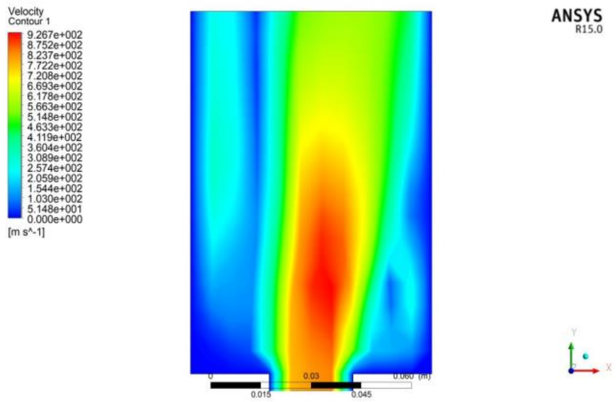

Figure 7: Flow distribution for 150kHz

Velocity vectors of different ultrasound frequencies are compared in Fig. 7. Maximum flow velocity is observed in the region above the ultrasonic transducer. As shown in Fig. 5, in low frequency most of the velocity changes occur near the ultrasound transducer and it does not have any changes on the other regions of the vessel. So, the main mixing is observed in the regions near the bottom. But, in the higher frequencies, almost, all the fluid will have high velocity and the violent mixing will be generated in all part of the vessel.

IV. CONCLUSION

In this paper, CFD simulations of the effect of ultrasound frequency on the flow behavior of the liquid imposing the ultrasound waves are studied using CFD. The ultrasonic transducers with different frequencies of 20 and 68 kHz and 150 MHz are used in the simulations. The contours of total pressure field and flow distribution are compared for these frequencies. The simulation results show that the pressure field which lower pressure area indicate the cavitation activity can occur are more uniform in the highest frequency, so that the cavitation bubbles will have very small size comparing two other frequencies. For investigating the effect of mixing on the fluid, flow behavior are also compared and result shows that 150kHz have violent mixing throughout the reactor meanwhile for the 20kHz and 68kHz, the mixing occur near the bottom of the reactor.

The experiment can be further analyzing in the future to investigate the location of the transducer inside the reactor, which location will give more cavitation area throughout the reactor. Width of reactor also can be analyze in the future to find suitable width of reactor to get high efficiency of separation of oil sand when using certain frequency.

Acknowledgment

Above all, I wish to thank to Allah for the wisdom, strength and faith and good health in order to finish this research. The completion of this study could not have been possible without the expertise of Sir Shafiq Bin Mohd Shayuti, my thesis adviser as his guidance and supervision largely aided me to complete this project. Also, not to forget, Dr Faizal Bin Mohamed, as I can assume my second supervisor which also guide me throughout this project.

I would also like to express my appreciation also goes to my colleagues and friends for helping me with this project.

Finally, this thesis is dedicated to my parents for the endless support and motivation given during the whole duration of writing this thesis. Alhamdulillah

References

[1] Zhao, X., Zhang, X., Liu, L., Fan, L., & Ge, D. (2017). Effect of ultrasonic reactor and auxiliary stirring on oil removal from oily sludge. Environmental Technology (United Kingdom), 38(24), 3109–3114. https://doi.org/10.1080/09593330.2017.1290146

[2] Khanal SK, Grewell D, Sung S, et al. Ultrasound applications in wastewater sludge pretreatment: a review. Crit Rev Environ Sci Technol. 2007;37(4):277–313.

[3] Akin B, Khanal SK, Sung S, et al. Ultrasound pre-treatment of waste activated sludge. Water Sci Technol. 2006;6 (6):35–42.

[4] Luo, X., Cao, J., Yan, H., Gong, H., Yin, H., & He, L. (2018). Study on separation characteristics of water in oil (W/O) emulsion under ultrasonic standing wave field. *Chemical Engineering and Processing: Process Intensification*, *123*(May 2017), 214–220. https://doi.org/10.1016/j.cep.2017.11.014

[5] Verhaagen, B., & Fernández Rivas, D. (2016). Measuring cavitation and its cleaning effect. *Ultrasonics Sonochemistry*, *29*, 619–628. https://doi.org/10.1016/j.ultsonch.2015.03.009

[6] Nguyen, D. D., Ngo, H. H., Yoon, Y. S., Chang, S. W., & Bui, H. H. (2016). A new approach involving a multi transducer ultrasonic system for cleaning turbine engines' oil filters under practical conditions. Ultrasonics, 71, 256–263. https://doi.org/10.1016/j.ultras.2016.07.001

[7] Mason, T. J. (2016). Ultrasonic cleaning: An historical perspective. Ultrasonics Sonochemistry, 29, 519–523. https://doi.org/10.1016/j.ultsonch.2015.05.004

[8] Ramli, E. M. N. bin. (2007). CFD simulation in the OIl and Gas Industry, (June), 2. Retrieved from https://www.google.com/url?sa=t&rct=j&q=&esrc=s&source=web&cd=3&cad=rja&uact=8&ved=0ahUKEwid6Y2HhezQAhUIp48KHR_nA80QFggmMAI&url=http%3A%2F%2Fdspace.unimap.edu.my%2Fdspace%2Fbitstream%2F123456789%2F13807%2F1%2F026-028CFD%2520Simulation.pdf&usg=AFQjCNGL9R_Q

[9] Liao, K., Ge, J., Zhang, G., & Fu, L. (2017). Multiple Frequency Ultrasounds Assisted Oil Sands Separation Technology. *Archives of Acoustics*, *42*(4), 767–773. http://doi.org/10.1515/aoa-2017-0080

[10] Nguyen, D. D., Ngo, H. H., Yoon, Y. S., Chang, S. W., & Bui, H. H. (2016). A new approach involving a multi transducer ultrasonic system for cleaning turbine engines' oil filters under practical conditions. Ultrasonics, 71, 256–263. https://doi.org/10.1016/j.ultras.2016.07.001

[11]