Analysis of the effect of flow position on homogeneity of lubrication in the blending circulation process using CFD

Syamsuri¹, Z Lillahulhaq¹ and Alim¹

¹ Department of Mechanical Engineering, Institut Teknologi Adhi Tama Surabaya, Arief Rahman Hakim St., No. 100, Surabaya 60117, Indonesia.

E-mail: Syamsuri@itats.ac.id

Abstract. Blending is the process of mixing of Base Oil and Additive to produce lubricating oil with the required specifications/characteristics. One of the determining factors to obtain the desired lubrication oil is the homogenization process. In this study, an analysis of the effect of the inlet position, circulation time, and flow velocity on the homogeneity of the resulting mixture was analyzed. The research method used is the computational fluid dynamic (CFD) simulation method with 2-dimensional (2D) fluent software. The flow that occurs in this condition is turbulent with the turbulent model used k-ε (epsilon). The results show that the upper inlet position within 10 minutes has gotten a homogeneous mixture compared to the lower inlet positions (30 minutes) and the center (20 minutes). Fluid flow velocity also affects the homogeneity of lubricating oil where the speed of 3 m/s within 10 minutes has obtained homogeneous results for all inlet positions compared to the velocity of 1.7 m/s and 2.5 m/s. In addition, the circulation time also affects the homogeneity level where for the lower inlet the circulation time needed to achieve homogeneity about 30 minutes. In conclusion, the longer the circulation time, the more homogeneous the mixture is obtained.

1. Introduction

Mixing in lubrication is the process of mixing base oil and additives to produce lubricating oil with the required specifications or characteristics. The homogenization process is a determining factor in obtaining the desired lubricating oil. The homogenization process has many methods, one of which is the circulation process. The circulation process is carried out by flowing a non-homogeneous semi-homogeneous mixture through the inlet and outlet. In this study, the analysis of the effect of the inlet position (bottom, center, and top), circulation time, and flow velocity on the resulting mixture homogeneity were analyzed.

The process of making lubricating oil starts from receiving raw materials from the base oil and additives from the refineries and exporting them through tanker or drum packaging. Then the raw materials are processed by mixing (blending) in accordance with applicable processing conditions. After the mixing and releasing process (off specs), lubricating oil is ready to be packaged in various types of packaging such as bottles (lithos), drums or bulk (bulk in IBC packages or skid tanks).

The most important process in making lubricating oil lies at several points that affect the quality of lubricating oil which is commonly called Total Quality Control (Q1 to Q8) and informed as follows:

Q1 => The process of checking the quality during the process of receiving raw materials.
Q2 => The process of checking the quality of raw materials after unloading.
Q3 => Quality inspection process at the time of stockpiling of raw materials.
Q4 => The process of checking the quality of the lubricating oil in the mixing process.
Q5 => The process of checking the quality of lubricating oil before the filling process.
Q6 => The process of checking the quality of the lubricating oil during the process of filling the lubricating oil in the package.
Q7 => The process of checking the quality of finished products at the time of stockpiling in warehouses and before sending them to consumers.
Q8 => The process of checking the quality of finished products in consumers by random sampling.

[1] In this discussion, the authors found that the process in Q4 where corrections were often found during the lubricating oil blending process is necessary. Mixing correction is also additional work in the mixing process for recirculation or the addition of raw materials in the form of base oil or additives to achieve the specified specifications. Mixing correction is normally caused by many factors and one of them is lubricating oil which is not homogeneous, so it needs to be recirculated.

Some of those studies used the CFD method and were successful. Research using the CFD method includes: Syamsuri et al., 2020 [2] conducted research on the effect of porous media on hydraulic jump characteristics by using smooth particle hydrodynamics. Another study that uses the CFD method is the SPH model for interacting of sloshing wave obstacle in shallow water tank was conducted by Syamsuri et al. in 2020 [3].

Similar research has been carried out to obtain a homogeneous mixing process including simulating a mixing tank by adding an agitator in the form of a propeller in the crude oil blending process [4], [5]. In addition, another investigation also conducts a mixing time study with the CFD approach for the homogenizer case with turbine blades [6][7]. Subsequently, research on this mixing process can also be carried out with a high mixing homogenizer reactor, with a stirrer, heating, and cooling controlled by stirring speed regulator and heat setting [8]. From these studies, the stirring system factor greatly affects the homogeneity of a mixture, so this problem is very important to be investigated. The purpose of this study was to determine how the influence of the position of the fluid flow inlet on the circulation process in the semi-finished product tank on the homogenization of lubricating oil.

2. Numerical methods

The equation describing the unsteady flow of the incompressible is:

\[ \nabla \cdot \mathbf{u} = 0 \]  

\[ \rho \frac{D \bar{u}_i}{Dt} = F_i - \frac{\partial P}{\partial x_i} + \mu \Delta \bar{u}_i - \rho \left( \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} \right) \]  

Where \( \rho \) is density, \( F \) is force, \( \mu \) is molecular viscosity, \( \nabla \) is vector operator, \( \bar{u} \) is a component of time average velocity, \( P \) is pressure, and \( \bar{u}' \) is a component of velocity of fluctuation and indexes \( i, j \) and \( k \) points to 1, 2 and 3 denoting the 3D components in the Cartesian coordinate system. Equations 1 & 2 are equations commonly used as Time Average Navier-Stokes Equation (TRANS), ANSYS FLUENT [9] has been used in this study with the \( k-\varepsilon \) turbulent model as proposed [10], [11] and the \( k-\varepsilon \) model have chosen after making comparisons with 3 models [12] and they found that the \( k-\varepsilon \) model is more accurate than other models and strong. This model was also used by Chern and Wang [13] where their findings were on ball valve flow rates. Some of these approaches have also been applied to study objects moving along a water tunnel flow as presented in [14]–[17]. The general equation of fluid motion in the form of TRANS as in equations 1 & 2 can be solved using various numerical schemes such as SIMPLE, SIMPLER, SIMPLEST, CTSSIMPLE, PISO or SIMPLE. Each has advantages and disadvantages [18]–[20] in this study using the SIMPLE (Semi Implicit Method for Pressure Linked Equation) scheme.
The computational domain has been prepared for the lubricating oil tank container in the mixing process. The simple construction of this tank with both the inlet and outlet sides is presented in Figure 3 & 4 below.

The discrete computational domain has been meshing using ANSYS software. In general, in this simulation, the author creates a mesh with a cheerful shape and uses 40 size intervals for all fields. The number of a mesh formed consists of 7650 triangular cells in zone 2 and 59770 triangular cells in zone 3 because the most important part is the lubricating oil tank itself. A general meshing can be seen in Figure 2. The types of boundary conditions that have been applied to the computational domain are shown in table 1.

| Criteria          | Value     |
|-------------------|-----------|
| Initial Condition |           |
| Gauge pressure    | 0 Pa      |
| x velocity        | 0 m/s     |
| y velocity        | 0 m/s     |
| Reynold numbers   | 7196.7    |
| Boundary Line     |           |
| Wall 1            | AO, OB, BC, CP, PH, HG, GF, FM, ME, ED, DA |
| Wall 2            | IJ, JK, KL, LI |
| Fan               | MN        |
3. Result and discussion

3.1. Validation

The simulation results that have been made need to be validated to determine the extent to which the simulation results are following the objectives to be achieved by the author. There are two methods for validating data from the simulation results as inform as follow:

- Data validation using study simulation results that have the same design.
- Data validation with Experiments, which compares data from simulation results with actual measurements from experimental results.

The simulation results that have been made need to be validated to determine the extent to which the simulation results are under the objectives to be achieved by the author. There are two methods for validating data from the simulation results, namely:

- Data validation using study simulation results that have the same design.
- Data validation with Experiments, which compares data from simulation results with actual measurements from experimental results.

Figure 5. Validation of pressure distribution

This validation is done by comparing the simulation results with the actual experiment at the bottom of the inlet position, with variations in sampling time at different circulation times, as a given time of 10 minutes, 20 minutes and 30 minutes. This validation aims to determine whether the simulation data is appropriate or reflects the actual process.

Figure 2 inform the comparison of the validation results between the actual measurements. The result illustrates that the validation results are slightly different from one and another. The results of the experiment on circulation for 10 minutes, 20 minutes and 30 minutes respectively obtained the average measurement for 3 experiments as the mixed kinematic viscosity of 27.59 cSt, 30.32 cSt and 30.9 cSt in order, while for the simulation results obtained 27.54 cSt, 30.75 cSt and 31.43 cSt, respectively. As a matter of fact, the simulation agreed on the result of the experiment.

3.2. Effect of Inlet Position on Mixed Homogeneity

From the fluent simulation results, the circulation process at the lower and center inlet positions with the same speed of 1.7 m/s takes longer to achieve the desired homogeneity. The inlet with the above position has reached homogeneity in 10 minutes, where for the lower and center side has not yet reached the same homogeneity in that time. This explanation is in accordance with figures 6, 7 and 8 in the image below for the details.
Figure 6. Lower Inlet positioned within 10 minutes process with a flow rate of 1.7 m/s

Figure 7. Center Inlet positioned within 10 minutes process with a flow rate of 1.7 m/s

Figure 8. Upper Inlet positioned within 10 minutes process with a flow rate of 1.7 m/s

Table 2. The effect of variations in inlet position and tank time on fluid viscosity at a speed of 1.7 m/s.

| Inlet Position | Time | Simulation result (cSt) | Std Specification |
|----------------|------|-------------------------|-------------------|
|                |      | Min | Max   |                        |
| Lower inlet    | 10 min | 27,54 | 35,31 | 29 – 35,2 |
|                | 20 min | 30,75 | 32,86 | 29 – 35,2 |
|                | 30 min | 31,43 | 32,16 | 29 – 35,2 |
| Middle inlet   | 10 min | 24,66 | 35,94 | 29 – 35,2 |
|                | 20 min | 33,51 | 34,24 | 29 – 35,2 |
|                | 30 min | 33,80 | 34,06 | 29 – 35,2 |
| Upper inlet    | 10 min | 34,02 | 35,10 | 29 – 35,2 |
|                | 20 min | 34,19 | 34,58 | 29 – 35,2 |
|                | 30 min | 34,21 | 34,48 | 29 – 35,2 |

3.3. Effect of Circulation Time and Inlet Position on Mixed Homogeneity

At this point, the comparison between the effect of the circulation time and the homogeneity of the mixture at the same inlet position is explained. From the simulation results, we get the data as in table 3. The comparison of homogeneity in the flow with a speed of 1.7 m/s with variations in the location of the inlet at various time variations of the mixture is shown in Figures 9, 10, and 11. Based on the simulations that have been carried out, Figures 9 and 10 show that in tanks with lower and center inlet it may take up to 20 minutes for the flow to reach a predetermined minimum homogeneity level. Whereas in Figure 11 the flow with an upper opening at a speed of 1.7 shows the homogeneity conditions above the lower limit. Fluid flow in the tank with the upper inlet position is easier to achieve
homogeneous conditions compared to other inlet positions. In addition, Figures 9, 10, and 11 show that the longer the fluid circulation time will increase the level of flow homogeneity.

**Figure 9.** Comparison of the homogeneity at various variations in the velocity of the lower inlet conditions in 1.7 m/s velocity

**Figure 10.** Comparison of the homogeneity at various variations in the velocity of the center inlet conditions in 1.7 m/s velocity

**Figure 11.** Comparison of the homogeneity at various variations in the velocity of the upper inlet conditions in 1.7 m/s velocity

### 3.4. Effect of Flow Speed and Inlet Position on Mixing Homogeneity

In this discussion, the authors compare the homogeneity levels of the mixtures at different circulation rates. The varying flow rates are given 2.5 m/s and 3 m/s. From the simulation results by varying the velocity at each inlet position, the data for viscosity is as shown in Table 4. The table showed that when the fluid circulates for 20 minutes, the viscosity of the fluid tends to be stable, where the viscosity of the fluid does not change drastically due to changes in fluid flow velocity at various inlet positions. Further, at a given speed of 2.5 m/s, the fluid with various variations in inlet positions also implied a viscosity value which tends to be stable.

**Table 3.** The effect of variations in inlet position, circulation time and tank fluid velocity on fluid viscosity

| Inlet Position | Time | 1.7 m/s (cSt) | 2.5 m/s (cSt) | 3 m/s (cSt) | Lower boundary | Upper boundary |
|----------------|------|---------------|---------------|-------------|----------------|---------------|
| Lower inlet    | 10 min | 27.54         | 32.99         | 29.91       | 29             | 35.2          |
|                | 20 min | 30.75         | 30.32         | 30.49       | 29             | 35.2          |
|                | 30 min | 31.43         | 30.98         | 30.96       | 29             | 35.2          |
| Middle inlet   | 10 min | 24.66         | 33.04         | 32.55       | 29             | 35.2          |
|                | 20 min | 33.51         | 33.67         | 33.36       | 29             | 35.2          |
|                | 30 min | 33.80         | 33.67         | 33.38       | 29             | 35.2          |
| Upper inlet    | 10 min | 34.02         | 32.99         | 32.94       | 29             | 35.2          |
|                | 20 min | 34.19         | 34.20         | 34.01       | 29             | 35.2          |
|                | 30 min | 34.21         | 34.62         | 34.41       | 29             | 35.2          |
Figure 12. Flow of circulation pattern in the center inlet position at a speed of 1.7 m/s at a time of (a) 10 minutes (b) 20 minutes (c) 30 minutes, speed 2.5 at a time (d) 10 minutes (e) 20 minutes (f) 30 minutes and a speed of 3 m/s at (g) 10 minutes (h) 20 minutes (i) 30 minutes.

4. Conclusion

From the results of data simulations carried out to determine the phenomenon and characteristics of pressure distribution and speed of the decompression tank with Oxygen fluid, it can be concluded that: The validation results indicate to have the same trend between the simulation method and the experimental method, where the simulation results for the lower inlet with a speed of velocity 1.7 m/s are obtained for circulation 10 minutes, 20 minutes and 30 minutes in order, 27.54 cSt while the actual 27.25 cSt, 30.75 cSt while the actual is 29.82 cSt and 31.43 cSt while the actual is 31.36 cSt, respectively. The position of the inlet gives the indication as the variable influence. It is shown by the data of 10 minutes circulation within a speed of velocity ratio 1.7 m/s which positioned at the upper side can give
a homogeneous effect to the mixture, compared to the lower or center inlet. The circulation process with more than 30 minutes period, resulting in a more homogeneous mixture than the short circulation time. The flow rate also affects the homogeneity process where the flow rate of 3 m/s in 10 minutes circulation as the example is improved to produce a homogeneous mixture compared to the flow rate of 2.5 m/s or 1.7 m/s.

5. References

[1] N. Gotongroyong, “QUALITY MANUAL PT. PERTAMINA (Persero.” Accessed: Oct. 03, 2020. [Online]. Available: https://www.academia.edu/7329997/QUALITY_MANUAL_PT_PERTAMINA_Persero.

[2] Syamsuri, M. J. Chern, and N. Vaziri, “Effect of Porous Media on Hydraulic Jump Characteristics by Using Smooth Particle Hydrodynamics Method,” Int. J. Civ. Eng., vol. 18, no. 3, pp. 367–379, Mar. 2020, doi: 10.1007/s40999-019-00465-8.

[3] Syamsuri, Ming-Jyh Chern, Nima Vaziri. 2020. SPH model for interaction of sloshing wave with obstacle in shallow water tank, Journal of King Saud University - Engineering Sciences, https://doi.org/10.1016/j.jksues.2020.07.009.

[4] R. K. Grenville, J. J. Giacomelli, G. J. VanOmmeren, C. F. Hastings, and M. J. Walters, “Blending in above ground storage tanks with side-entering agitators,” Chem. Eng. Res. Des., vol. 137, pp. 395–402, Sep. 2018, doi: 10.1016/j.cherd.2018.07.017.

[5] F. Habla, H. Marschall, O. Hinrichsen, L. Dietsche, H. Jasak, and J. L. Favero, “Numerical simulation of viscoelastic two-phase flows using openFOAM®,” Chem. Eng. Sci., vol. 66, no. 22, pp. 5487–5496, Nov. 2011, doi: 10.1016/j.ces.2011.06.076.

[6] M. Jahoda, Moštěk, A. Kukuková, and V. Machoň, “CFD modelling of liquid homogenization in stirred tanks with one and two impellers using large eddy simulation,” in Chemical Engineering Research and Design, Jan. 2007, vol. 85, no. 5 A, pp. 616–625, doi: 10.1205/cherd06183.

[7] G. Harikrishnan, P. Ebin, A. J.-I. J. of C. E. and, and undefined 2014, “CFD simulation of subcooled flow boiling using openFOAM.”

[8] A. Ghanem, T. Lemenand, D. Della Valle, and H. Peerhossaini, “Static mixers: Mechanisms, applications, and characterization methods - A review,” Chem. Eng. Res. Des., vol. 92, no. 2, pp. 205–228, Feb. 2014, doi: 10.1016/j.cherd.2013.07.013.

[9] A. K. Sahu, K. A. Joshi, V. Raghavan, and A. S. Rangwala, “Comprehensive numerical modeling of ignition of coal dust layers in different configurations,” Proc. Combust. Inst., vol. 35, no. 2, pp. 2355–2362, Jan. 2015, doi: 10.1016/j.proci.2014.06.128.

[10] G. Montante, D. Horn, and A. Paglianti, “Gas-liquid flow and bubble size distribution in stirred tanks,” Chem. Eng. Sci., vol. 63, no. 8, pp. 2107–2118, Apr. 2008, doi: 10.1016/j.ces.2008.01.005.

[11] Y. Zhang et al., “Effect of impeller on sinking and floating behavior of suspending particle materials in stirred tank: A computational fluid dynamics and factorial design study,” Adv. Powder Technol., vol. 28, no. 4, pp. 1159–1169, Apr. 2017, doi: 10.1016/j.apt.2017.02.002.

[12] A. S. Mahdi Al-Obaidi and W. A. Otten, “AERODYNAMIC ANALYSIS OF PERSONAL VEHICLE SIDE MIRROR,” 2016.

[13] M. J. Chern and C. C. Wang, “Control of volumetric flow-rate of ball valve using V-port,” J. Fluids Eng. Trans. ASME, vol. 126, no. 3, pp. 471–481, May 2004, doi: 10.1115/1.1760536.

[14] A. Lavrov, J. M. Rodrigues, J. F. M. Gadelho, and C. Guedes Soares, “Calculation of hydrodynamic coefficients of ship sections in roll motion using Navier-Stokes equations,” Ocean Eng., vol. 133, pp. 36–46, Mar. 2017, doi: 10.1016/j.oceaneng.2017.01.027.

[15] Z. Driss, O. Mlayeh, D. Driss, M. Maaloul, and M. S. Abid, “Numerical simulation and experimental validation of the turbulent flow around a small incurved Savonius wind rotor,” Energy, vol. 74, no. 8, pp. 506–517, Sep. 2014, doi: 10.1016/j.energy.2014.07.016.
[16] B. Deng and J. Wang, “Saturated-unsaturated groundwater modeling using 3D Richards equation with a coordinate transform of nonorthogonal grids,” Appl. Math. Model., vol. 50, pp. 39–52, Oct. 2017, doi: 10.1016/j.apm.2017.05.021.

[17] Z. Lillahulhaq and V. S. Djanali, “Unsteady simulations of Savonius and Icewind turbine blade design using fluid-structure interaction method,” 2019, doi: 10.1063/1.5138264.

[18] P. G. Tucker and P. G. Tucker, “Solver Algorithms and General Solution Procedures,” in Computation of Unsteady Internal Flows, Springer US, 2001, pp. 79–116.

[19] D. Sun, J. Xu, and P. Ding, “Performance analyses of IDEAL algorithm on highly skewed grid system,” Adv. Mech. Eng., vol. 2014, Feb. 2014, doi: 10.1155/2014/813510.

[20] M. M. Rahman, “Avoiding under-relaxations in SIMPLE algorithm,” https://doi.org/10.1080/10407790.2020.1787043