ANALYSIS OF FLOW FLUID MIXING CRUDE OIL STORAGE TANK WITH CFD

Name : Fachruddin Ali
NRP : 4208 100 026
Department : Teknik Sistem Perkapalan
Lecturer Advisor : Irfan Syarif Arief, ST.MT.
Ir. Tony Bambang Musriyadi, PGD.

Abstract
This study relates to the influence of fluid flow on the mixing crude oil in a stirred tank is so influential in the time required in mixing to avoid precipitation (mixing time). The complexity of the flow to be part of global change flow patterns from one type to another type, alternately in a large scale. Because of this, can provide a significant effect on the performance of mixing. In this study using simulations based on Computational Fluid Dynamics (CFD) using the model of Large Eddy Simulation (LES) with multiphase flow modeling using mixture models. The study was conducted in a cylindrical tank with flat bottom (flat bottomed cylindrical tanks) with a diameter of 10 m and a Pitch Blade Turbine (PBT) with a diameter of 3 m which is equipped with a baffle width of each - each baffle of 1 / 12 H wall-mounted symmetrically in the vertical direction. Simulations performed by the method of unsteady with 27 variations (3 variable blade angle 30 °, 45 ° and 60 °, 3 variable number of baffles with the use of 0, 2 and 4 and 3 variable speed rotating impeller 150,200 and 250 rpm) and used the number of iterations as much as 60 times for every 10 time step. From the simulation results, to get the mixing time with achieve homogenity, then the approach contained in the Use of Number of Baffle as much as 4 to 60 ° Angle impeller (150 rpm).

Keywords: Pitched Blade Turbine, LES, Mixture Model, Fluid Flow, Mixing Time.