

CFD Investigation on Fluid Flow Analysis in Fluid Separator

Techno-Societal 2018 pp 131-141 | Cite as

- Pranita Shinde (1)
- Pramod Kothmire (1)
- Yogesh Bhalerao (1)

1. School of Mechanical and Civil Engineering, MIT Academy of Engineering, , Alandi, Pune, India

Conference paper

First Online: 07 November 2019

- 393 Downloads

Abstract

The analysis of fully developed flow in the two fluid separators is an important issue in the industry such as production, processing, and petrochemical. The role of the two fluid separator is to separate two different fluid by using an appropriate mechanism without changing the quality. In this study, we have reviewed different mechanisms of two fluid separations such as gravity sedimentation, centrifugation, and electro kinetics, etc. The current work focuses on the design aspect of a fluid separator with respect to geometry and thermal design. CFD has been used to simulate flow in a fluid separator and its results have been verified experimentally. Flow rates used in the simulation have different values in interval 0.1 LPM. The study shows the best performance of fluid separator with respect to shape and flow rates. The given work helps to co-relate various design of separator in the industry with laboratory separators.