Use of Large Eddy Simulations Model to Simulate the Flow of Heavy Oil-Water-Air through Pipe

Authors : Salim Al Jadidi, Shian Gao, Shivananda Moolya

Abstract : Computational Fluid Dynamic (CFD) technique coupled with Sub-Grid-Scale (SGS) model is used to study the flow behavior of heavy oil-water-air flow in a horizontal pipe by adapting ANSYS Fluent CFD software. The technique suitable for the transport of water-lubricated heavy viscous oil in a horizontal pipe is the Core Annular flow (CAF) technique. The present study focuses on the numerical study of CAF adapting Large Eddy Simulations (LES). The basic objective of the present study is to gain a basic knowledge of the flow behavior of heavy oil using turbulent CAF through a conventional horizontal pipe. This work also focuses on the success and applicability of LES. The simulation of heavy oil-water-air three-phase flow and two-phase flow of heavy oil–water in a conventional horizontal pipe is performed using ANSYS Fluent 16.2 software. The influence of three-phase heavy oil-water air flow in a selected pipe is affected by gravity. It is also observed from the result that the air phase and the variation in the temperature impact the behavior of the annular stream and pressure drop. Some results obtained during the study are validated with the results gained from part of the literature experiments and simulations, and the results show reasonably good agreement between the studies.

Keywords : computational fluid dynamics, gravity, heavy viscous oil, three-phase flow
Conference Title : ICCFD 2023 : International Conference on Computational Fluid Dynamics
Conference Location : Prague, Czechia
Conference Dates : March 20-21, 2023