Simulation of Erosion Rate in a Reducer for Liquid Solid Flow System using Computational Fluid Dynamics (CFD)

ABSTRACT

This research aims to simulate the influences of flow parameters such as particles size, stream velocities, and outlet reducer diameter on the erosion rate for a reducer in light crude oil ($C_{19}H_{30}$)-solid (sand) flow system. A commercially accessible ANSYS Fluent 2020 R1 (Academic Version)-computational fluid dynamics (CFD) was applied to numerically simulate the erosion rate in the reducer. Three separate models were used in the CFD approach called as a continuous flow modelling, Lagrangian particle tracking, and empirical erosion equation. The simulated parameters covered 100 - 500 µm particles size, 3 - 7 m/s stream velocities and 0.0762 - 0.1778 m outlet reducer diameter. It was found that the maximum erosion rate increased with the increasing size of the particles and stream velocities and decreased with the increasing of the outlet reducer diameter. For all the simulated parameters, the location of maximum erosion rate was found to be at the outlet location of the reducer except for the reducer with the diameter larger than 0.1270 m whereby it is located at the inlet location of reducer.