Investigation of Nano composite heat exchanger annular Pipeline flow using CFD analysis for Crude Oil and Water characteristics

M.Parthasarathy\textsuperscript{1}, Kadupu Rakesh Varma\textsuperscript{2}, Elumalai P.V\textsuperscript{2}

\textsuperscript{1}Vel Tech RangarajanDr.Sagunthala R & D Institute of Science and Technology, 400 Feet Outer Ring Road Avadi, Chennai, Tamil Nadu, India 600062.

\textsuperscript{2}Department of Mechanical Engineering, Aditya Engineering College, Surampalem, India.

Abstract. This study focuses on the dual-core annular process of crude oil and water. The crude oil contains non-Newtonian core behaviour features. The flow of heat transmission in pipeline locations is complicated. The crude oil has a high viscosity. As a result, the cooled water passes through the annular core. Water flow is performed by turbulence effects, and crude oil is performed by laminar flow effects. The non-Newtonian method's influence is investigated in terms of pipe angles and deformations. In the valve, the inlet velocity flow is set to 1.75 m/s. The flow is executed in downhill orientations in high-velocity locations. The simulation results show CFD variations of 3.5 m/s velocity, 2.33 MPa pressure, pressure gradients of 1.33MPa, and Reynold numbers of 1.38MPa. This study is being carried out for the horizontal pipeline. CFD solution solvers investigate the volumetric flow of fluid characteristics as well as temperature effects. The maximum heat energy has been lowered, resulting in flow directions that are one-of-a-kind.

Keywords: Composite Annular Core; Water; Crude Oil; Ansys; Velocity; Temperature; Pressure.

1. Introduction

The computational fluid dynamic technique helps in the solution of higher-order functional and numerical problems. In the iteration process, thermo mechanical numerical computations are difficult to solve. However, the CFD approach is used for numerical calculations with N of iterations. The mixing of fluid and oil particles in tube formation is determined by temperature variations and pressure distributions. The majority of CFD formations are based on solver iteration methods and graphical structures. The interaction between the hot and cold mixing processes is converted in two ways by the fluid pipe simulation method. Battisti et al. [1] explored the film distillation process by modifying the parameters of the CFD flow methods. Disruptions in distillation tubes interact with proper fluid transfers and structures. The greater the temperature changes, the more they are communicated from the entrance valve to the output valve. The velocity flow is generated by the thermal fluid behaviours and Nanoparticles from the input versions. Corre [2] constructed and characterised the core contact of two-phase fluid