Numerical study of a Taylor bubble rising in stagnant water

Seungchul Shin(skytls5108@naver.com)
University of Strathclyde and Pusan National University

Abstract

Multiphase flows in occur in various fields including Oil & Gas industry, nuclear systems, chemical processes. The oil and gas industry is the most important application of multiphase flow. There are different kinds of multiphase flows known as “flow pattern/flow regime” depending on the flow rate and geometry of the system in two-phase gas-liquid flow. Among them, gas-liquid slug flow in pipe is one of the complex flow pattern in two phase flow. In vertical pipes, the bulk of the gas which is trapped inside large bullet-shaped (Taylor) bubbles that moves upwards causes flow intermittence.

Flow intermittence is one of the main reasons why the offshore oil and gas systems encounter severe corrosion, structural instability and a poor reservoir management. Therefore, the prediction of the appropriate the hydrodynamic characteristic is essential for successful pipeline operation and optimization. In this article, velocity characteristic of the flow field around a Taylor bubble rising in stagnant water in a vertical pipe is measured by using numerical simulation software STAR CCM and VOF method to know Instantaneous velocity field around the bubble and the mean velocity field in front of the bubble, in the liquid film, and in the wake region by ensemble-averaging the instantaneous velocity fields measured around 100 different bubbles

Keywords: *numerical simulation, multiphase flow, flow regime, VOF method*

Biography

MSc student at University of Strathclyde.
Graduated at Pukyeong National University in 2018
Studied at Wayne State University as an Exchange student in 2017