

NUMERICAL INVESTIGATION OF WAVY-ANNULAR AIR-WATER TWO-PHASE FLOW IN A SMALL HORIZONTAL PIPE

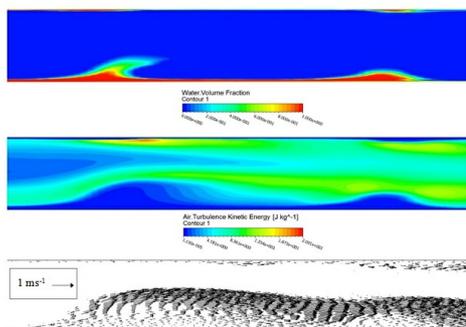
ANTONINO ARINI (SCHOOL OF ENGINEERING, UNIVERSITY OF LINCOLN); JUN YAO (SCHOOL OF ENGINEERING, UNIVERSITY OF LINCOLN); TIMOTHY GORDON (SCHOOL OF ENGINEERING, UNIVERSITY OF LINCOLN); YUFENG YAO (DEPARTMENT OF ENGINEERING DESIGN AND MATHEMATICS, UNIVERSITY OF THE WEST OF ENGLAND)

This study has been carried out using Computational Fluid Dynamics (CFD) to investigate the annular two phase flow behavior that flow pattern has some desirable features for renewable energy system application. The circumferential distribution of the liquid film in a small diameter horizontal pipe (ID = 8.8mm) is non-uniform owing to gravitational force, where liquid film is thicker at the bottom than that at the top. In a wavy-annular flow regime, large amplitude waves formed at the gas-liquid interface has played a significant role to replenish the top liquid film.

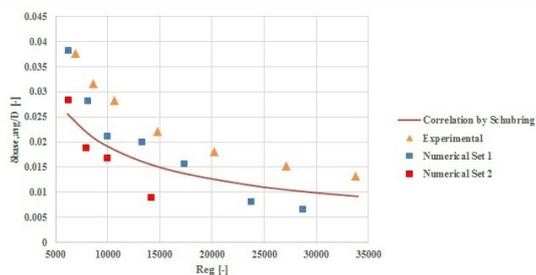
CFD study was conducted to analyse the interfacial gas and liquid flow structure with the change of air and water superficial velocity. The flow wave characteristics in wavy-annular region were able to be determined using ANSYS Fluent and ad hoc User Defined Function (UDF), where the top and bottom liquid film coverage through the entire pipe by penetrating the liquid interfacial waves was predicted (see in fig.1). The boundary condition was chosen, according to experimental test data given by Shedd, et al.

A transition layer is determined to characterize air and water interfacial region using water volume of fraction (VOF). The large amplitude interfacial waves were tracked at the downstream of flow. The wave transverse time with wave velocity and wave peak height was evaluated by correlations (see in full paper). The maximum height of wave decreases with increasing gas and liquid wave velocities. The pressure drop was calculated via time-average to account for the oscillations caused by wavy penetration in the flow direction. The numerical results were compared with the available experimental data, and a qualitative agreement of results was found (see in fig 2).

CFD results has demonstrated that the current two-phase flow model **using UDF code** is capable to **predict** the annular liquid film interfacial structures in horizontal pipe. Although some discrepancies were found between the numerical results and experimental data in 2D simulations, there was **an** interesting flow **behavior** observed. CFD simulation has confirmed that the interfacial waves are the source of entrainment to constrain liquid film flowing into the gas stream. The liquid film formation may follow the mechanism replenishing the liquid **droplets** to the pipe upper region via liquid deposition process.



water film distribution (top), turbulent kinetic energy of air (middle) and water velocity vector.



The numerical and experimental dimensionless base film thickness comparison.