Overview

Over the past three decades, experimental evidence has shown that the injection of micro-bubbles (diameter range from 1 to 10 µm) at a large volume fraction (up to 70%) into a liquid turbulent boundary layer over a flat plate or an axisymmetrical body can reduce the skin friction by as much as 80% from the value without bubble injection. The application of this discovery can be very useful to optimize the operation of current navy fleets and future ship design. However, the basic physical mechanisms responsible for the reduction are not fully understood (Elghobashi 2004 and Xu et al 2002). The experimental findings regarding the physical mechanism are somehow controversial (Hassan et al 2004 and Kwaamura et al 2004). Thus, development of a highly accurate numerical model is needed to assist in better understanding of the physical mechanism involved.

Numerical Model

The main objective of this research is to study the interaction between the micro-bubbles and the turbulent structure in order to have a good insight into the physical mechanisms responsible for the drag reduction. This is done firstly by reviewing the existing multi-phase flow models; secondly by developing an accurate numerical model for bubbly turbulent boundary layer flow; and thirdly validation of the same against available experimental data. Also Large Eddy Simulation (LES) model is introduced to account for accurate study of two-phase flow turbulence.

This project aims to deliver a better understanding of the two-phase bubble-liquid interaction, the basic cause of drag reduction and also the factors affecting the same, thereby opening avenues for further research into the implementation of drag reduction using Micro-bubbles, as the cause of drag reduction cannot be easily studied through experiments due to inherent bottlenecks in experimental methods.