

# Multipase simulation using OpenFOAM for calculating ship hull

Rishabh Sharma

August 4, 2022

## Abstract

This is a comprehensive study to calculate the Ship resistance when sailing in water. The CFD tool used is OpenFOAM which is an opensource and very robust software. The study focuses on the mesh generation, fields initialization and comparison between different solvers and finite volume solutions. All the verification of the resistance calculation is done through the ITTC 1987 empirical formula. Utilites like *blockMesh*, *snappyHexMesh* and the *interFoam* solver are used. Also a comparison between the accuracy of different divergence and time Schemes is explained.

