

## Software Lab:

### Main Objective:

Multiphase solvers in OpenFOAM are widely used in science and industry as they provide excellent results compared to experimental data.

This means that the pressure distribution around the ship can be determined more accurately with CFD than is possible through measurements in experimental tests. This allows the resulting forces on the ship's hull to be determined very well with a lateral flow.

In this study, a ship with different drift angles, speeds and water depths will be simulated and calculated with the multiphase solver.

The aim is to display the forces as a function of drift angle, speed and water depth parametrically in a table.

Task	Estimated duration
OpenFoam installation	Already done
familiarization and execution of tutorials: blockMesh, snappyHexMesh, DTCHull-tutorial in 3D flow around a ship in OpenFOAM (interFoam)	2 Week
familiarization and configuration of sampling post-process in OpenFOAM (point, line, forces output, wave pattern and surface sampling)	
Implement of real ship hull an generate meshes water depth 7.5m	-
running the case with different angles and one velocity RANS, URANS; reporting results	1 Week
running the case with diff. angles and diff. velocity RANS, URANS; reporting results	3 Weeks
water depth 5m, diff. angles and velocity RANS, URANS; reporting results	4 Weeks
preparing materials (contours, graphs, force plots, tables) and writing final report	4 Weeks
Summation:	14 Weeks

### Meeting Schedule:

#### Weekly Meeting with Stephan Schultz:

Day: Every Thursday, Time: 14:00, Location: Room I / 105

#### Progress Report Meetings with Prof. Kornev:

Frequency: Twice every month, Starting: May 2024.

### Contact information:

#### Prof Nikolai Kornev:

Raum: I / 09

Tel: +49 381 498 9550

nikolai.kornev@uni-rostock.de

#### Stephan Schultz:

Room: I / 105

Tel: +49 381 498 9426

stephan.schultz@uni-rostock.de