

README

Author: Thomas Henkel<thomas.henkel@leibniz-ipht.de>

Created: 2026-03-19

Case: stepEmuV2ScaledFixedInletPressureTemplate

SubDir: /

OpenFoamSolver: interFoam

OpenFoamVersion: OpenFoam2406

Usage

For running the simulation customize the settings and run scripts, initialize the openFoam environment by running the *openfoam2406* script from the shell and proceed with the target run script.

Technical Hint: When starting the simulation the interface geometries are very coarse and must slowly converge to the real geometries and target contact angles. This must be forced by running the first stage of the simulation at very low time step size which can be limited in the system/controlDict dictionary by setting the maxDeltaT value to 1E-10 and increase it successively after some iterations.

Folders and Files

FileName	Note
0.orig	Original data for the time step 0 with the initial values
Allclean	The Allclean Script of the simulation
Allrun	The Allrun Script of the simulaiton
Allrun-parallel	The Allrun script for a parallel run
AllToVTK	Conversion of the Simulation data to VTK legacy format
constant	The constant subfolder of the OpenFoam case
ParametersAndScaling	The folder holing the utilized parameters and scaling rules
Readme.txt	This File
system	The system subfolder of the OpenFoam case
VTK_RESULTS_SELECTED	A selection of VTK Datasets from the published simulation run.
postprocessing	paraview state files for postprocessing