

README

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

Created: 2026-03-19

Case: stepEmuV2ScaledFixedInletPressureTemplate

SubDir: /postprocessing

OpenFoamSolver: interFoam

OpenFoamVersion: OpenFoam2406

Usage

Folder and templates for postprocessing and rendering of the simulation results. Launch *paraFoam* after running *openfoam2406* and load the state file *postprocessing-state.pvsm* from the menu */File/LoadState*. Maybe that some of the paths need to be customized according the location of your simulation on the file system of your computer.

Folders and Files

FileName	Note
postprocessing-state.pvsm	The paraview state file for visualizing the simulation results.
pp-state_Screenshot.png	The screenshot of the para view showing the simulation data.
Readme.txt	This file