

README

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

Created: 2026-03-19

Case: stepEmuV2ScaledFixedInletPressureTemplate

SubDir: constant/triSurface

OpenFoamSolver: interFoam

OpenFoamVersion: OpenFoam2406

The original geometry source files are in the ParametersAndScaling Subfolder

Files

FileName	Note
channel.eMesh	The eMesh, generated by open foam / snappyHexMesh.
channel.stl	the utilized input stl geometry.(A copy of stepEmuV3Scaled.stl)
Readme.txt	This File.
stepEmuV3Scaled.msh	The original gmesh mesh of the DG_UNIT
stepEmuV3Scaled.stl	The original stl file of the DG_UNIT