

OpenFOAM: from fundamentals to postprocessing

Contents

Introduction to OpenFOAM

- a. Introduction to OpenFOAM releases and capabilities
- b. Instruction for installation and compilation from source
- c. Description of the test case: rotating swept axial fan
- d. Description of the typical structure of an OpenFOAM case

Simulation preprocessing

- a. Description of the mesh preprocessing steps
- b. Description of blockMesh utility
- c. Creation of periodic boundaries
- d. SnappyHexMesh utility and mesh automatization in OpenFOAM

Simulation setup

- a. Selection of the turbulence model, temporal scheme
- b. Selection of the boundary conditions
- c. Selection of the numerical schemes and convergence criteria
- d. Controlling saves, temporal steps and runtime functions

Simulation postprocessing

- a. A guide to function objects
- b. Creating a simple function object
- c. Sampling, time averaging and most relevant function for postprocessing
- d. Visualization of results: Paraview and ParaFOAM