

pyMeshFOAM

Description

The numerical tool presented here aims at performing meshes with free open-source mesh generation software. More particularly, the available scripts are dedicated to the meshing process of shapes within a box. The mesh of boundary layers is encompassed within the options. This numerical tool allows the simulation of fluid-structure interactions between fluid and deformable structures. Three open-source software codes are involved in such simulations: OpenFOAM, preCICE and CalculiX. This free mesh generation code is based on free tools such as blockMesh, snappyHexMesh, cfMesh and GMSH. The solid mesh interface matches automatically with the mesh of the fluid interface. Thus, there is no interpolation between the nodes of both meshes. It is however possible to use different interfaces since preCICE allows interpolations. It also enables the fluid simulation of a flow around a rigid structure (so only with OpenFOAM). The programming language of the scripts is Python.

The user settings are defined in both bash files `mesh_fluid` and `mesh_solid`. In 2D, the `mesh_solid` file does not exist. The meaning of each variable is detailed in these files. Some variables are defined in the Python files. The quality of the mesh might be improved with other values depending on the geometry. We advise users to have a look inside the Python codes.

Compatibility

The meshing tool runs with OpenFOAM-7, cfMesh and GMSH-3.0.6. The tutorial cases can be run with OpenFOAM-7, CalculiX-2.10/2.13/2.16, preCICE-1.6.1. This tool could also run with other versions.

Getting started

Two tutorials are provided. One should run the file `mesh_fluid` in the tutorial repositories and, in 3D, the file `mesh_solid` to achieve the meshing process of the solid. These files are in the repositories Fluid and Solid respectively for the 3D cases.

2D

This case is dedicated to the simulation of a flow around a 2D wing. Several ways of meshing are available: blockMesh and cfMesh. The boundary layer can be meshed with both but there are three options available with blockMesh. The first one corresponds to boundary layers that surround the wing. This needs to cut the trailing edge to make it smooth. In the second one, the boundary layer mesh does not surround the wing but continues towards the outlet boundary (the most robust way but the most time-consuming). The last one corresponds to the second one except that we break the

propagation of the boundary layer mesh. The quality of the mesh can decline but the mesh size is smaller.

3D

This case is dedicated to the simulation of a flow around a deformable wing in 3D. Note that the fluid-structure simulation can easily be limited to a fluid simulation with a rigid wing. The mesh of the fluid is achieved with cfMesh while the mesh of the boundary layer (if desired) is rather performed with snappyHexMesh. Indeed, the free version of cfMesh does not provide quality boundary layer meshes. It is worth noting that snappyHexMesh can also appear insufficient for some configurations. We foster users to play around with the key parameters in the boundary layer dictionary of snappyHexMesh. The solid structure is meshed with GMSH. The input file corresponds to the same mesh as the fluid interface of the solid. If the fluid mesh around the solid structure is very refined, the meshing process of the solid part can take some hours and GMSH can fail to make the mesh. If one does not necessarily want the solid nodes to be at the same location as the fluid nodes, we foster users to achieve two meshing processes: the desired mesh of the fluid part and another mesh of the fluid followed by the meshing process of the solid based on this second fluid mesh.