

General instructions

December 24, 2021

The following instructions underline the main details and organization of the tool at the current stage and version.

Future developments could add or reduce needed software or packages required by the tool.

1 Overview

The tool is organised two main three main folders:

- **1 - Grid & CFD:** this folder allows a single CFD run from the input data provided by the user, either viscous or inviscid. There is also the possibility to simply generate the grid file. Specific details are reported within PDF **README_1**.
- **2 - Optimisation:** this folder executes an optimisation of a single or multi-element configuration provided by the user in input. Specific details are reported within PDF **README_2**.
- **3 - Post Processing.** Here you can find various Python3 script to further analyse your CFD results, like convergence history, polar plots etc.

For each folder, the user can define its own inputs (like geometry, grid details, CFD characteristics optimisation parameters etc.).

2 Requirements

2.1 Software

The software/languages mandatory required for a generic run are listed below.

- **Python3** for the entire computational procedure.
- **Gmsh** for grid generation.
- **SU2** for CFD computation and preliminary studies.

Additionally, an useful but not mandatory post-processing software could be **ParaView**.

2.1.1 Python3 packages

The tool requires the following Python3 packages:

- numpy
- math
- copy
- sys
- time
- subprocess
- matplotlib
- pandas
- random
- scipy
- multiprocessing
- shutil
- os
- pyDOE