

Report: Automation Tutorial for Simulation Case Generation in OpenFOAM

Miguel Rosas

March 7, 2024

1 Introduction

This report describes a Bash script designed to automate the generation and execution of multiple simulation cases in the OpenFOAM software. The script takes an integer as input, representing the number of cases to generate and execute. The main functions of the script and how they can be used and modified are detailed below.

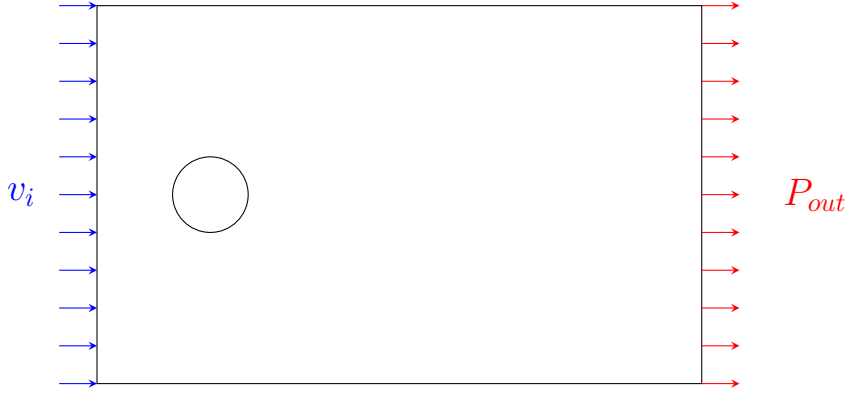


Figure 1: Tutorial schematic

2 Script Operation: `start_simulation.sh`

The script consists of the following parts:

1. **Definition of Variables:** The script starts by defining the desired Reynolds numbers for each simulation (Re_1 to Re_6), a viscosity coefficient (ν), and the duct diameter d . (Lines 4-12)
2. **Calculation of Coefficient k :** Using the `bc` command (allows working with decimals in bash), the value of k is calculated as the division of ν by d . (Line 14)
3. **Calculation of Velocities:** Velocities v_1 to v_6 are calculated by multiplying each Reynolds number by the value of k . (Lines 18-23)
4. **Creation of Velocity List:** A list is created with the calculated velocity values. (Line 26)
5. **Argument Verification:** The script checks if the number of cases is provided as an argument. If not provided, it displays a usage message and exits. (Lines 29-32)
6. **Case Generation Loop:** For each case, the script performs the following actions:

- (a) Creates a folder for the case. (Lines 40-43)
- (b) Copies files and folders from a base case and appends the case number to some files. (Lines 46-62)
- (c) Replaces the value of \$v in the initial velocity file U of the case with the corresponding Reynolds number value. (Lines 65-66)
- (d) Generates a geometry file. (Lines 69-70)
- (e) Converts the mesh created to OpenFOAM format. (Line 73)
- (f) Makes adjustments to the mesh file. (Lines 76-83)
- (g) Executes the simulation in parallel with `mpirun` and then reconstructs the case in VTK format. (Lines 85-89)
- (h) Performs post-processing operations with ParaView. (Lines 93-95)
- (i) Moves the results to the main folder and deletes the case folder, keeping only initial conditions, results in VTK format, and animations. (Lines 97-128)

3 Script Usage

Before running the script, it is necessary to give it permissions. To do this, write in the console:

```
$ chmod +x ./start_simulation.sh
```

To use the script, it must be executed from the bash command line, ensuring that the OpenFOAM commands are active, providing the number of cases as an argument. For example:

```
$ ./start_simulation.sh 5
```

This will generate and execute 5 simulation cases for Re_1 , Re_2 , ..., Re_5 .

4 Conclusions

The script provides an efficient way to generate and execute multiple simulation cases in OpenFOAM, which can be useful for parametric studies and sensitivity analyses in numerical simulations. The modular structure of the script allows for easy modification and adaptation to different case studies.