SIMPLE implementation and study of velocity field around a 2D geometry

PI: Tommaso Bocchietti, Department of Mechanical & Mechatronics Engineering, University of Waterloo
Contact: Tommaso Bocchietti, tbocchie@uwaterloo.ca
Date: February 19, 2024

Abstract Personal project proposal for the course ME663 - Computational Fluid Dynamics. Implementing SIMPLE algorithm to solve the steady state Navier-Stokes equations for a 2D problem and analyze the velocity field around a 2D geometry.

Keywords CFD, SIMPLE, Velocity field analysis, 2D problem, External flow

1 Introduction

This project has been chosen to further explore the content of the course ME663 - Computational Fluid Dynamics and to apply the knowledge acquired trying to finally answer a personal question of interest.

Given that for the first assignment we have successfully implemented a 2D solver for the steady state Navier-Stokes equations using the SCGS algorithm, we would like to further explore the topic of CFD solver by implementing in the same codebase the SIMPLE algorithm and compare the results of the two.

Time and capabilities / resources permitting, the second part of the project will be to analyze the velocity field around a 2D geometry (external flow), using the SIMPLE algorithm just implemented.

This study, is related to the author's interest in the problem of: *Investigating the distribution of flow velocity around a pillar during windy conditions to enhance personal comfort by determining the optimal position that reduces wind exposure.*

Up to now, we think that the implementation of the SIMPLE algorithm would be enough to be considered a personal project itself. However, we believe that the addition of a complete velocity field analysis would make the project more interesting and challenging.

2 Methods

The project will be divided into two main parts: the implementation of the SIMPLE algorithm and the analysis of the velocity field around a 2D geometry.

2.1 SIMPLE algorithm

The SIMPLE algorithm will be implemented starting from the same codebase used for the first assignment, which already contains the SCGS algorithm. The codebase has been written in `C` and from the beginning has been designed to be modular and easily extendable with new algorithms and features.

The SIMPLE algorithm will be implemented following the guidelines provided in the course notes and at first will be able to just solve the steady state Navier-Stokes equations for a 2D problem (u, v, and p).

The code will be tested using the same test cases used for the SCGS algorithm, and the results will be compared with the ones obtained with the SCGS algorithm.

2.2 Velocity field analysis

The velocity field analysis will be performed using the same codebase and the SIMPLE algorithm.

We will use our code to answer the question of interest to us: *Investigating the distribution of flow velocity around a pillar during windy conditions to enhance personal comfort by determining the optimal position that reduces wind exposure.*

The problem will make use of multiple simplifying assumptions, so to be simple enough to be modeled within our 2D solver, but at the same time complex enough to analyze a real world problem.

As a real world scenario where our analysis could be useful, we can think of a person waiting for a bus or a train on a windy day at a station with only a pillar that can be used as a shield. At the end of the project, we will be able to provide a map of the flow velocity distribution around the pillar, based on the wind speed and direction. Eventually, multiple simple geometry will be tested (e.g. a circular pillar, a square pillar, etc.) and the results will be compared.

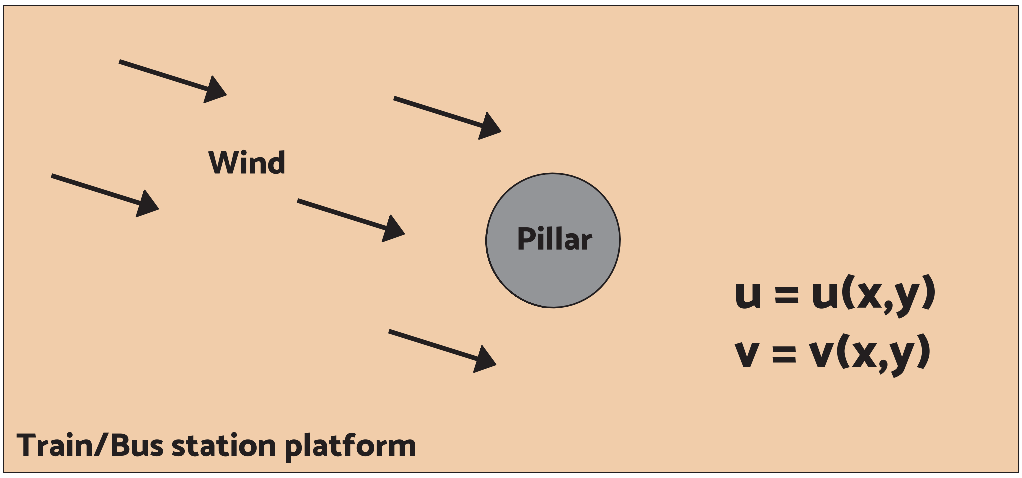

Figure 1: Problem drawing and real world scenario.

3 Expected Results

The main expected result of the project is the implementation of the SIMPLE algorithm. We will report the mathematical derivation of the algorithm, the implementation details, and the results obtained for the Lid Driven Cavity problem in the same PDF report used also for the first assignment.

The second expected (and optional) result is the analysis of the velocity flow field around a 2D geometry. We will report the results obtained considering different problem geometries and wind parameters (speed & direction) in a separate PDF report.

Up to now, we don't have the competences to use a commercial CFD software to compare our results with. We would greatly appreciate it if the professor could provide us with some results obtained from such software for comparative analysis, if feasible.

In the end, we expect to have a better understanding of the CFD solvers at least for 2D steady-state problems, and to be able to have a higher self-consciousness about their implementation and their limitations.