# Introduction To Computational Fluid Dynamics

Stephen Roberts

July 9, 2014

## 1   Introduction

Computational fluid dynamics, commonly abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

## 2   1 Dimensional Explicit Schemes

### 2.1   Advection