Comparison of Staggered and Collocated grids for solving Navier-Stokes Equations

M. J. Ketabdari, H. Saghi, H. Rezaei

1- Associate Professor, Faculty of Marine Technology, Amirkabir University of Technology
2- PhD student of Hydraulics, Department of Civil Engineering, Ferdowsi University
3- PhD student of Hydrodynamic, Faculty of Marine Technology, Amirkabir University of Technology

ketabdar@aut.ac.ir
Has.saghi@stu-mail.um.ac.ir
hrezaei@aut.ac.ir

Abstract

For fluid flow the governing equations can be solved using numerical methods. Bases of these equations are Momentum and Continuity leading to Navier-Stokes equations. There are several methods such as Simple, PISO and Fractional Step for solving these equations. In solution procedure, it needs to decide where to store the velocity components. To evaluate this, staggered and collocated grids can be used. On staggered grid, the velocity components are stored at the cell face and the scalar variables such as pressure are stored at the central nodes. However on collocated grids, all parameters are defined at the same location at the central nodes. The advantage of staggered grids is more accurate pressure gradient estimation. In collocated grids method, solving the equations is simpler as all parameters can be stored at the same location. In this paper a comparison between collocated and staggered grids for solving Navier-Stokes equations is presented. The results were validated using standard tests such as led driven cavity, channel and backward facing step. Discussion is made on accuracy of these methods to estimate horizontal and vertical velocity profiles.

Keywords: Navier-Stokes Equations, Staggered grid, Collocated grid, Backward facing step, PISO

1. Introduction

For evaluation of behavior of fluid flow in a physical phenomenon, the Navier-Stokes equations can be solved using numerical methods. These governing equations are basically valid for laminar flow. However, the turbulent flow can be analysed using a time and spatial averaged form of these equations. Turbulence models are divided to classical, large eddy simulation (LES) and Direct Numerical Solution (DNS) models. The classical models use the Reynolds Average Navier Stokes (RANS) equations. Complication of computational procedure for unsteady turbulent flow simulations is due to the very large number of grid points to be resolved for a wide range of length scales. In DNS all scales are resolved both spatially and temporally. In LES approach, the equations are filtered to separate large eddies from small ones. In these models it is assumed that the largest eddies interact strongly with the mean flow and contain most of the energy. Therefore the main effects of turbulence can be modeled by this approach accurately.

2. Governing equations and solution algorithm

In this section, the governing equations and the solution algorithm is presented.

2.1. Governing equations

The fluid flow is assumed to be incompressible. Therefore the governing equations can be written as: