Summer Fellowship Report

On

Analysis of flow behavior when obstructed by moving vane

Submitted by

Vignesh.S.P

Under the guidance of

Prof.Kannan M. Moudgalya
Chemical Engineering Department
IIT Bombay

Vignesh.S.P

July 10, 2019

Acknowledgment

The success of this project required a lot of guidance and help from many people and I am extremely privileged to have got this along the completion of this project. I respect and thank Prof.Kannan M. Moudgalya, Chemical Engineering Department and Prof.Shivasubramanian Gopalakrishnan, Mechanical Engineering Department, for providing me an opportunity to do the project work in FOSSEE, IIT Bombay and giving me all the support and guidance which made me complete the project duly.

I'm extremely thankful to my mentors Mr. Sathish Kanniappan and Ms. Deepa Vedartham for their support and mentorship through out the course of completion of the project. I'm thankful to my mentors for providing equipment support which was very helpful.

I'm thankful and fortunate enough to get constant encouragement, support and guidance from all the personnel's from FOSSEE, IIT Bombay.

Abstract

This case study aims to numerically simulate the behavior of a water while the flow is being obstructed by a moving vane. The mesh motion of the vane is achieved by Overset Grid technique or Chimera Grid technique. This case study explores the functionality and capability of Overset Grid method for dynamic problems available in OpenFOAM.

Contents

List of Figures

List of Tables

Chapter 1

Introduction

The behavior of fluid in dynamic systems are very difficult to numerically simulate using traditional methods to handle dynamic systems such as mesh deformation, Arbitrary Moving Interface (AMI), and re-meshing. This case study explores the functionality of overset method in dynamic problems. The numerical simulation of water flow when obstructed by a moving vane at a constant velocity is studied here. The motion of the vane is controlled by overset grid method.

1.1 OpenFOAM

OpenFOAM (for "Open-source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics. OpenFOAM is freely available and open source, licensed under the GNU General Public Licence. The licence is designed to offer freedom, in particular it encourages users of the software to make modifications or developments. Overset Grid in OpenFOAM is available in versions released only after 2017. It is being actively developed and several solvers and validation has been done.

1.2 Overset Grid Method

Overset gridding refers to the use of multiple disconnected grids to discretize the flow domain. The component grids, which can be any size, type, or shape, need only overlap each other to completely cover the solution domain. Furthermore, a component grid resolving one geometric feature may intersect another geometric feature. As a final preprocessing step (or during the solution, in the case of moving bodies) composite grid assembly software determines which grid points lay outside the flow domain and grid-to-grid connectivity.

The overset approach also enables changing the geometry and grid system locally without requiring regeneration of other grids. This flexibility greatly simplifies design studies as geometry perturbations can easily be added to an existing design and grid system by gridding the new feature and possibly including grids to connect the new feature with the existing grids. Since the baseline grid system is not altered the changes in the flow are more reflective of the change in the geometry and not changes resulting from re-meshing the entire geometry.

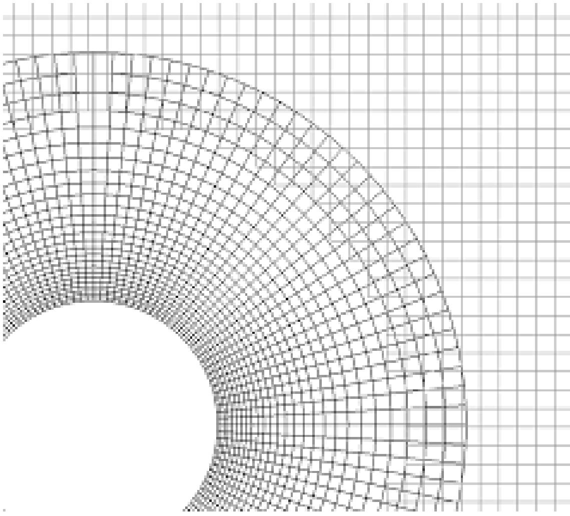

Figure 1.1: Overset Grid of Sphere

Chapter 2

Case Setup

2.1 Geometry and Mesh

The Geometry of the case study is a slice on a circular pipe of diameter of 0.1m and of length 1m. The vane is of thickness 0.01m and of height 0.1m. The mesh for the fluid domain is made using blockMesh utility with an empty boundary patch named oversetPatch to trigger overset interpolation. The vane is meshed separately and is merged with the mesh of the pipe using 'mergeMeshes' utility. The fluid domain in pipe is meshed coarsely with local size of 0.002m which resulted in 12500 cells and the vane is meshed finely with local size of 0.001m which resulted in 9000 cells.

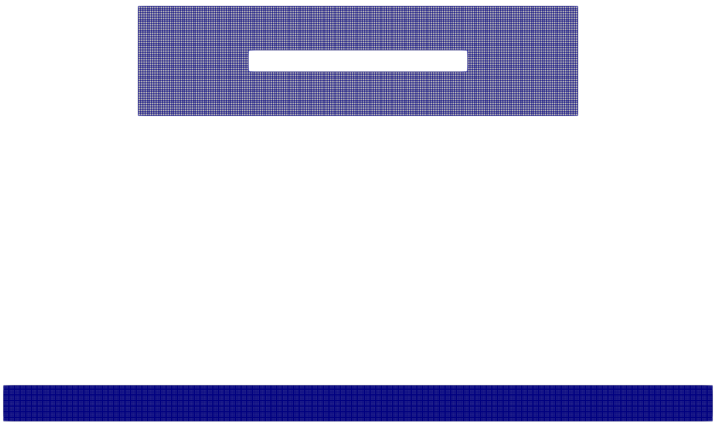

Figure 2.1: Vane Mesh and domain mesh separated

Figure 2.2: Overset grid of vane onto pipe mesh

2.2 Boundary conditions

The boundary conditions employed for the computational domain is given in table 2.1. The overset interpolation zones are setup using setFields with zoneID. The fluid domain in pipe is set to zoneID 0 and overset vane mesh is set to zoneID 1.

Boundary Name	U	p
walls	type uniformFixedValue; uniformValue (0 0 0);	type zeroGradient;
vane	type movingWallVelocity; value (0 0 0);	type zeroGradient;
inlet	type fixedValue; value uniform(1 0 0);	type zeroGradient;
outlet	type zeroGradient;	type fixedValue; value uniform 0;
sides and oversetPatch	type overset;	type overset;

Table 2.1: Boundary conditions for the case study.

2.3 Dynamic motion of vane

The dynamic motion of vane is an oscillating linear motion which is defined in dynamicMeshDict file inside constant folder. The details of the setup is as follows. The vane moves in y direction with an amplitude of 0.1m, the speed of the motion of the mesh is governed by the value omega.

The dynamicMeshDict is as follows,

```
dynamicFvMesh          dynamicOversetFvMesh;
solver                 multiSolidBodyMotionSolver;
multiSolidBodyMotionSolverCoeffs
{
    vaneZone
    {
        solidBodyMotionFunction  oscillatingLinearMotion;
        oscillatingLinearMotionCoeffs
        {
            amplitude    (0 -0.1 0);
            omega        2.0;
        }
    }
}
```

2.4 Solver and Simulation control

overPimpleDyMFoam solver is used for this simulation. The official definition for this solver is as follows:

Transient solver for incompressible, flow of Newtonian fluids on a moving mesh using the PIMPLE (merged PISO-SIMPLE) algorithm.

The transport properties for case is setup in transportProperties file in constant directory in which the value of density and kinematic viscosity for both air and water are given as well as surface tension value is also given.

Sl.No.	Description	Value
1	Kinematic viscosity of water	1e-06 m^2/s
2	Density of water	1000 kg/m^3

Table 2.2: Properties of Fluid

The important control values as given in controlDict is as follows

startTime 0;
endTime 1.5;
deltaT 0.00025;
writeControl adjustableRunTime;
writeInterval 0.005;
maxCo 1;
maxDeltaT 0.001;

Chapter 3

Result and analysis

3.1 Mesh Motion

The overset mesh of vane is coded to move in a linear oscillating pattern which can be seen in the figure 3.1 at different time steps of the simulation

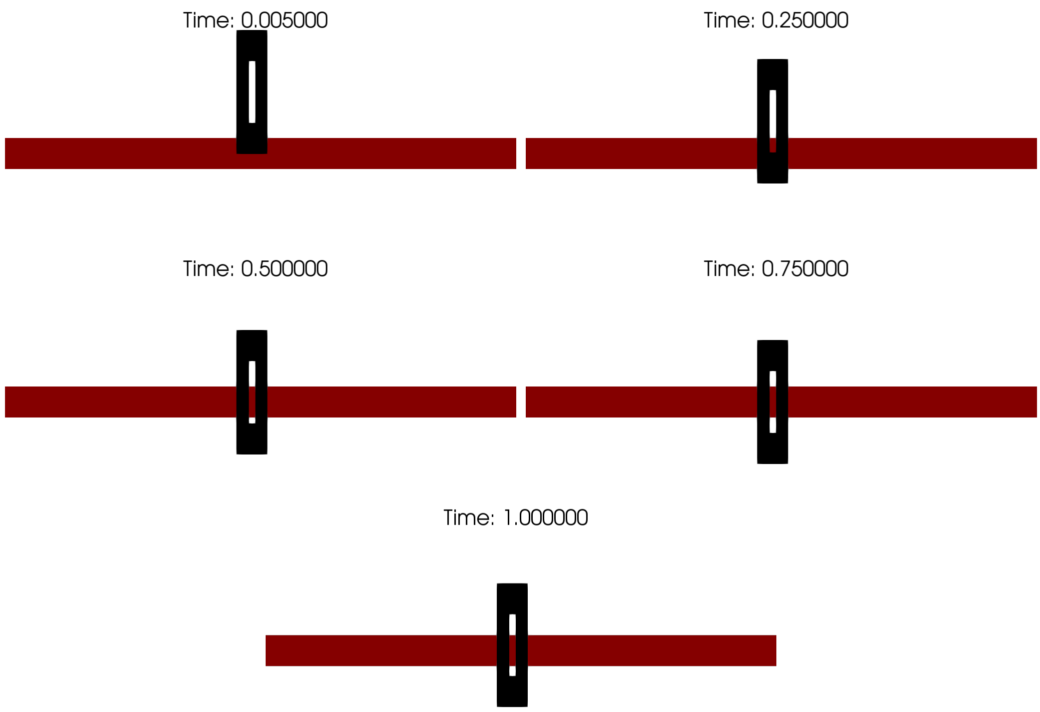

Figure 3.1: Mesh motion

3.2 Pressure Field

The pressure field from the simulation are plotted against the length of the pipe for different time steps can be seen in the figure 3.2. The pressure difference on the inlet side and the outlet side of the pipe during closing will rapidly increase as high as 60 Pa.

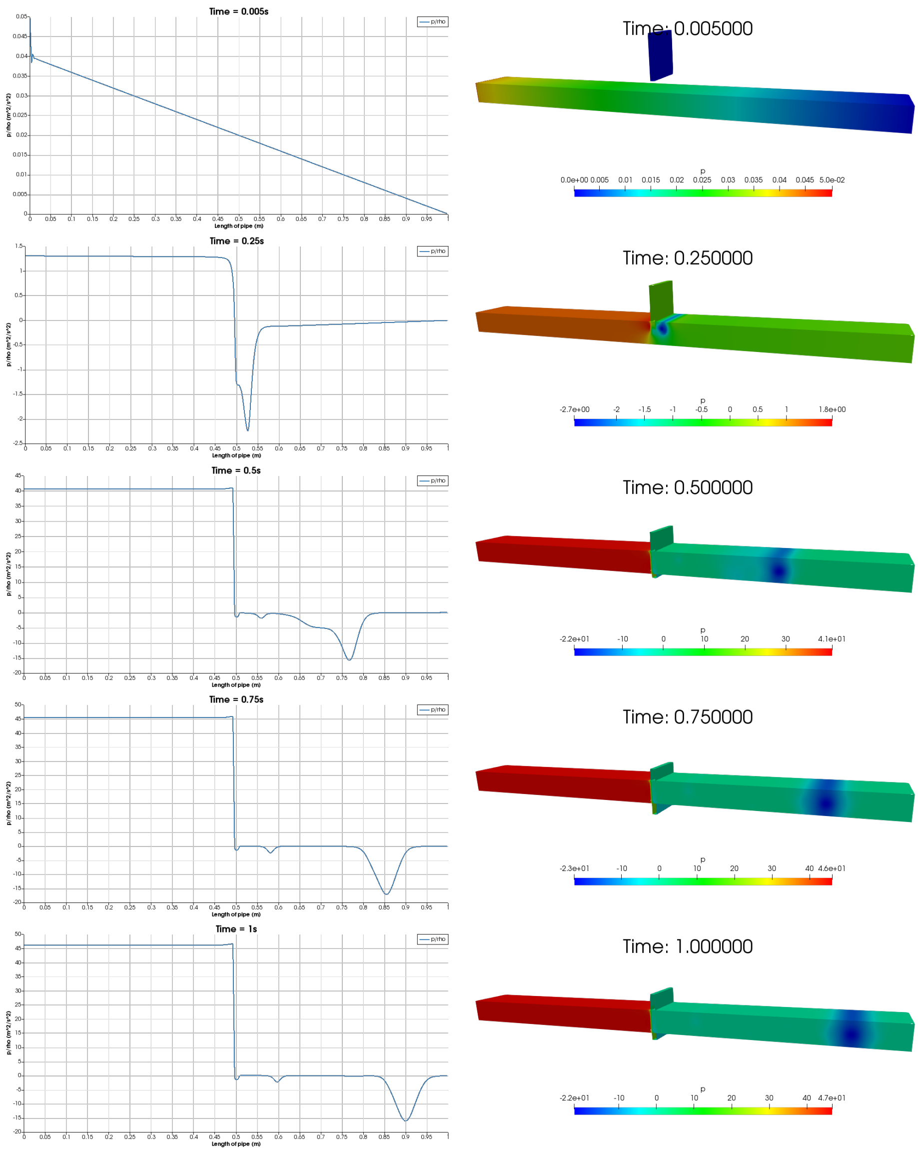

Figure 3.2: Pressure Plot and contours for different time steps

Chapter 4

Conclusion

This case study explores the overset grid methodology for handling dynamic systems in CFD in a standard solver for incompressible flow. The setup and simulation of the case is explained in the report and the results are also viewed.

Reference

- OpenFOAM - Overset grids wiki from cfd-online.com
- Youtube playlist on Overset grid by Wolf Dynamics
- Official Overset Guide by ESI-OpenCFD