

Simulation of Backward Facing Step Flow using Immersed Boundary Method by FVM

Dennis Liang Chen Lin^{1,*}, Normayati Nordin¹, Ronny Yii Shi Chin²

¹ Flow Analysis, Simulation, and Turbulence Research Group (FASTREG),
Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, MALAYSIA

² GN Resound (Malaysia) Sdn Bhd, No.5, Jalan i-Park 1/1Kawasan Perindustrian i-Park,
Bandar Indahpura, Kulai, Johor, MALAYSIA

Received 02 October 2018;
Accepted 28 December 2018;
Available online 02 February
2019

Abstract: The ability to handle complex geometries with fluid-solid interactions has been one of the most challenging issues in the field of Computational Fluid Dynamics (CFD) as most of the engineering problems that exist nowadays are dealing with complex geometries. There are two methods that have been studied and developed for now which are the immersed boundary method (IBM) and the body fitted mesh method (BFM). This paper studies about the simulations on backward facing step flow using immersed boundary method by finite volume method. The two different approaches which are the IBM represented by structured grid and BFM represented by unstructured grid were being studied throughout the research. Although 2D backward facing step is considered as one of the simplest flows, it is still widely studied by a lot of researchers as it is a benchmark problem in the engineering field and the physics involved in the flow is complicated. Comparisons of the two methods which are the IBM and BFM are made in terms of the differences in total pressure, velocity profiles and acoustics power level of different Reynold number. The grid and mesh are generated by ANSYS Workbench. IBM is considered as a better method in handling complex geometries than the BFM and it is easier to construct a uniform grid generation. The parameters of the Re are 100, 500, 1000, 2500, 5000 and 10000. From the obtained results, it showed that the most active region of this flow is at the downstream near the step. The total pressure, velocity magnitude and acoustics power level are fluctuating in that region as it is the recirculation region.

Keywords: Immersed Boundary Method, Body Fitted Mesh, Backward Facing Step Flow

1. Introduction

Computer simulations of different random phenomena have become an indispensable tool in the modern scientific investigations. The role of computer in the implementation of a simulation model is defined. Simulation software was developed to overcome the deficiencies associated with modelling complex system especially in Computational Fluid Dynamics (CFD). Computational simulation software has evolved rapidly based on a general purposes programming language such as FORTRAN code, C language and MATLAB. Computer aided engineering (CAE) has recognized computational simulations to be one of the tools that is commonly and widely used in all the industries due to the advancement of technology. It is undeniable that computational simulation software is an important part of aerodynamics design process for automobiles. Thus, there are a lot of advantages in numerical simulation.

CFD deals with a lot of complex geometries and it has been a hot issue because majority of the problems existed in engineering are related to complex geometries. Immersed Boundary Method (IBM) and Body Fitted Mesh (BFM) method

are two different methods taken to simulate complex flow. IBM was introduced to study flow patterns around heart valves and has evolved generally useful method for problems of fluid structure interaction by Charles [1]. It is then used to deal with complex problems in engineering field with immersed rigid bodies by Lima [2]. In IBM, the Navier-Stokes equations are solved on a fixed background grid which can be unstructured and Cartesian grid. In the case of numerical method of IBM, it involves both the Eulerian and Lagrangian variables. Eulerian grid represents the total fluid domain which is stationary and regular whereas Lagrangian grid represent the immersed boundaries which are generally non-stationary and unstructured.

The study in flow separation is focused by industrial and fundamental. Backward facing step flow can determine the aerodynamic performances of aerofoil and automotive vehicles for example cars, trains and ships. It is a good passive control method that allow structural modifications through backward facing step flow as active control method is more expensive in cost. Laminar separation bubble is the primary effect when operating in low Reynolds number regime. The laminar

separation bubble strongly affects the forces of aerofoil which are the drag force and the lift force and aerodynamic performance. BFS flow problem is recognised as a benchmark in validating computational simulation algorithm in some numerical simulations.

2. Structure

The flow of fluids especially in complex geometries has been one of the main issues in Computational Fluid Dynamics due to most of the engineering problems involve complex geometries nowadays. The difficulty appears in the way of representing the body of the complex geometries, moving walls and its interaction with the fluid in the surrounding. There are few challenging problems in numerical simulations. The common problem is when handling the coupling of the structure deformations and fluid flow. In short, the most common different approaches used to simulate complex flow are unstructured grid method and IBM.

The interaction between fluid and solid is promoted by a force term which has been added to the Navier-Stokes equation. There are various immersed boundary methods used to calculate the force term [5]. In IBM, the Navier Stokes equations are solved on a fixed regular grid which can be unstructured, curvilinear and Cartesian grid. Several modifications and reinforcement have been proposed after this. Eulerian grid represents the total fluid domain which is stationary and regular whereas Lagrangian grid represent the immersed boundaries which are generally non-stationary and unstructured shown in Fig. 1.

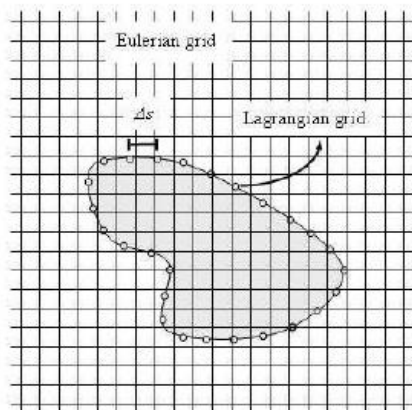


Fig. 1 - Lagrangian grid for the interface and Eulerian grid for the domain [6]

2.1 Backward Facing Step

The research of backward facing step flows developed an important branch in the fundamental of fluid mechanics. Flow geometry of the same is very important for investigating separated flows. Backward step flow is chosen as it eases the study of the reattachment process by minimizing the effect of the separation process. There are stronger interactions between the other separating and reattaching flow geometries [4]. The primary flow features of the backward facing step flow are illustrated in Fig. 2.

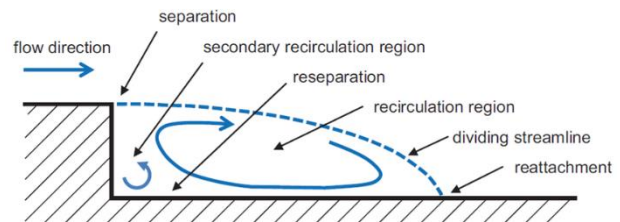


Fig. 2 - Graphical representation of gauge and absolute pressure [3]

3. Methodology

In this chapter, the method to generate a grid for flow over a BFS by using IBM and body fitted mesh method will be studied and mentioned to achieve the objectives of the research. This case study is a 2D problem. Simple fluid mechanics equations, for example the Navier Stokes equation is used to develop the numerical equation that is needed to generate the grid. The commercial software package that is used to run the simulations is ANSYS Workbench. Then, the readings and results will be obtained through the software. The numerical simulation is based on volume penalization method. The results that obtained from running the simulation is then analyzed by graphs and contours. The quantities that will be analyzed in this project are acoustics power level, velocity profiles and total pressure at selected locations whereas the parameters for Re is between 100 to 10000.

3.1. Governing Equations

The numerical simulation is carried out based on the incompressible Navier-Stokes equations. Periodic boundary conditions will be imposed on all flow variables except the pressure. The forcing term contains information about the presence of fluid submerged bodies [7]. That is, by proving a local momentum forcing, simulating the net momentum exchange between solid and fluid, various boundary conditions can be applied on their mutual interfaces. The only dimensionless number appearing in the governing equations is the Reynolds number. For the flow problem considered, the following definition is used for the Reynolds number.

3.2. Meshing

Ansys meshing is an automated intelligent product which helps to produce accurate meshes for physics solutions. It is an important step to create a mesh before running any engineering simulation. For this study case, there are two types of meshing which are structured and unstructured meshing. Structured meshing will represent the immersed boundary method whereas the unstructured meshing will represent the body fitted mesh method. Structured meshing is done by using the face meshing option where all the meshes will be in the same size of square in shape as shown in Fig. 3. Unstructured meshing is done by inserting the method of all triangles as shown in Fig. 4.



Fig. 3 - Immersed Boundary Grid

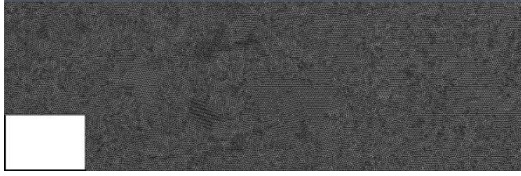


Fig. 4 - Body Fitted Mesh Grid

4. Result Outcomes/Contributions

The scatter graph for total pressure of different Reynold number along the horizontal line x is plotted as shown in Fig. 5.

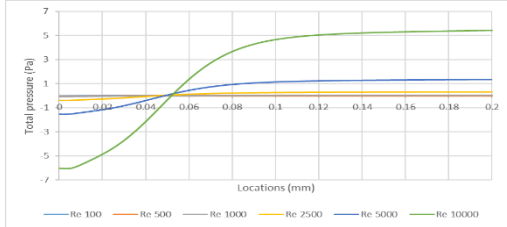


Fig. 5 - Total pressure of different Reynold numbers at specified locations for IBM

It has clearly shown that the total pressure varies along the specified locations. For Re 100, the difference in total pressure is hardly to be noticeable as the difference is very small compared to Re 10000. The total pressure before 50mm is in negative values because it falls in the recirculation region. The flow separation separates the region into two boundary layers. Flow separation is a boundary layer separation and it occurs when there is a change in the velocity of the fluid in either direction or magnitude which is too large for the fluid to adhere to the solid surface. Therefore, the total pressure is negative value in the recirculation region and it will be in positive value when it exits the recirculation region from the separation flow and enters another normal boundary layer.

A graph of total pressure at unstructured grid of different Reynold number at specified locations is plotted in Figure 6. Comparisons are then made between the graph of total pressure from IBM and graph of total pressure from BFM. The total pressure decreases as the Reynold number increases for 0mm and increases when the Reynold number increases for 50mm and above.

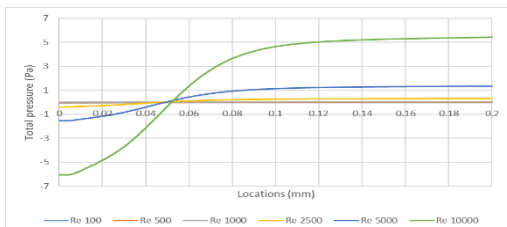


Fig. 6 - Total pressure of different Reynold numbers at specified locations for BFM

Based on the comparison of Fig. 5 and Fig. 6, the graphs of total pressure of different Reynold numbers at different specified locations for IBM and BFM are almost the same. There is just a slight difference in values of the total pressure which cannot be observed through the graphs. The main factor that cause the total pressure for both the methods to be almost the same is because of the sizes of the grids generated. Therefore, both the IBM and BFM have not much difference when it comes to analysing the total pressure for small sizes of grids.

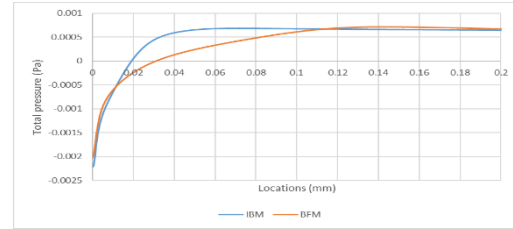


Fig. 7 - Total pressure of Re 100 at specified locations for IBM and BFM

Fig. 7 has shown the graph of total pressure of Re 100 against locations for IBM and BFM. By comparing both the IBM and BFM, it can be clearly observed that IBM increased sharply in the beginning to a point at 0.05m then almost stabilised throughout it. As for the graph of BFM, the graph increases steadily from the beginning till the end. According to studies done before this, the graph of total pressure against specified locations after the step should increase sharply in the beginning as it's the recirculation region in the case of the 2D BFS flow. When the total pressure near the step is lower, causing the recirculation phenomenon to be happened. Therefore, it can be concluded that for studying the case of 2D BFS flow, it is the best to use IBM as most of the studies show that IBM is a better method compared to the BFM.

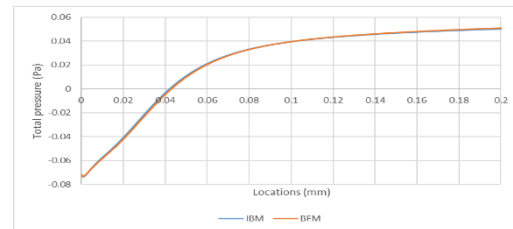


Fig. 8 - Total pressure of Re 1000 at different locations for IBM and BFM

As for higher Re study case, a Re 1000 simulation is carried out to study about the total pressure graphs too. From Figure 8, both IBM and BFM increased steadily throughout the end of the point. Both the methods did not have much difference in terms of the readings of total pressure. Therefore, it can be concluded that for case study of higher Re which is Re 1000 and above, either the IBM or BFM can be used in the model of 2D BFS flow.

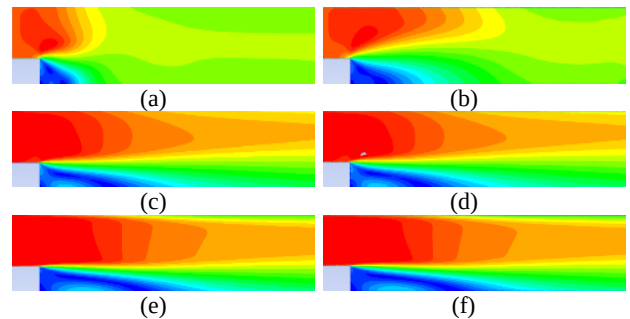


Fig. 9 - Velocity magnitude of backward facing step for IBM and BFM of Re 100, 1000 and 10000

Fig. 9(a) and Fig. 9(b) show the velocity magnitude of IBM and BFM of Re 100. The air velocity travels faster in BFM than IBM. When the Reynold number had been increased, the velocity magnitude also increased over the flow in the backward facing step model. The velocity magnitude is increasing and most active at the inlet which is at the upper stream of the step. Fig. 9(e) and Fig. 9(f) were taken from the simulation of Re 10000 for IBM

and BFM which are turbulent flows and they had the highest velocity magnitude of 5.0 m/s near the inlet of the flow and decreasing across the flow and lowest magnitude at the downstream near the step.

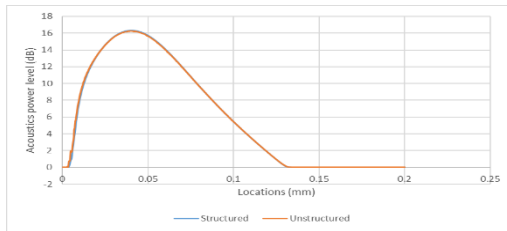


Fig. 10 - Acoustics power level of Re 10000 at specified locations for IBM and BFM

It can be observed from Fig. 10 that the acoustically most active region was at 0.04m away from the step which is the peak of the sound level and it started to decrease all the way till the point at 0.13m and acoustics power level can no longer to be recorded. Even though there were simulations of lower Re range from 100 to 5000 for both the IBM and BFM were carried out, the end results could not capture any reading for Reynold number in between that range. One of the assumptions made from this failure of capturing the acoustics power level was because of the simulation used is not suitable in studying the acoustics field for lower Reynold number.

There are other types of simulations which are more suitable or have the better ability in capturing the acoustics level of the flow that had been studied by other researchers. Direct numerical simulation (DNS) is one of the simulations which has the better ability in capturing acoustics sound level but the requirements are so large that this method is not a practical approach for simple case like two dimensional backward facing step flow. Besides that, there is another direct method which is the large eddy simulation (LES) that can be used to study acoustics sound level. This is a cheaper method compared to the DNS and it allows for more complex flow configuration at higher Reynolds numbers to be resolved but it is still computationally demanding.

5. Conclusions

Throughout the simulations done in the study, it was found that immersed boundary method had validated the relevant fluid mechanics bench mark problem of the 2D BFS. The linear interpolation scheme is used to satisfy the no slip condition on the IBM and the continuity for the cell containing the immersed

boundary, which is numerically stable regardless of the relative position between the grid and the immersed boundary. The first objective of the project was achieved. It was to generate a grid for flow over BFS using IBM and BFM. Immersed boundary method was represented by the structured grid whereas the body fitted mesh method was represented by the unstructured grid method. For this study case, there are two types of meshing which are structured and unstructured meshing. When it came to the comparison of the total pressure of same Re between the IBM and BFM, it had been noticed that for low Re, there were differences when comparing the total pressure of Re 100. Lastly, acoustics power level of both the IBM and BFM were captured using the model in ANSYS Workbench software. For Re of 10000 and lower, the simulations did not manage to read any acoustics power level emitted from the backward facing step flow. When it came to higher Re which was Re 10000, the model managed to capture the acoustics power level of the flow.

Acknowledgement

The authors thank the Ministry of Higher Education, MALAYSIA (MOHE) and Universiti Tun Hussein Onn Malaysia (UTHM) for the award of sponsorship under the student development programme.

References

- [1] C.S Peskin, 1972 (2007). Flow patterns around heart valves: a numerical method, Journal of Computational Physics, vol.10, pp 252-271
- [2] Silva, A. L. F. de L. e, Silva, A. R. da, & Silveira Neto, A. da. (2007). Numerical simulation of two-dimensional complex flows. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 29(4), 379–387.
- [3] Scharnowski, S. (2013). Investigation of turbulent shear flows with high resolution PIV methods.
- [4] Armaly, B.F, DURST, F. & KOTTKE, V. 1980 Momentum, heat and mass transfer in backward facing step flows. Sonderforschungsbereich 80.
- [5] Lee & Choi (1998). Experimental and Numerical Investigation of 2D Backward Facing Step Flow.
- [6] R. Mittal, and G. Iaccarino, (2005). Immersed Boundary Methods, Annual Review of Fluid Mechanics, vol 37, pp.239-261
- [7] Kim J & Moin. P (1985). Application of a fractional step method to incompressible Navier Stokes equations. Journal of Computational Physics 5