A COMPARISON BETWEEN CONVENTIONAL MESH DRIVEN AND MESHLESS CFD METHODS TO STUDY FLUID FLOWS

¹Sumathi K and ²Divya V. V*

¹Associate Professor, Department of Mathematics, PSGR Krishnammal College for Women ²Assistant Professor, Department of Mathematics with Computer Applications, PSG College of Arts and Sciences

ABSTRACT

Most fluid flow studies make use of the standard computational fluid dynamics approaches of finite volume methods to estimate the transient flow characteristics of different types of fluids. These methods while very accurate in estimating the response of the flowing fluids under various conditions, are a little tedious and involve a lengthy process of creating a mesh before estimation experiments can be conducted on the fluids. The mesh creation becomes more accurate with an increase in the number of compartments, which means more points are used. Also creating a three dimensional mesh, the time involved in setting it up and running for a solution are both very lengthy processes. This paper explores a newer method of executing the same process without the use of a mesh and compares the accuracy of the results

Keywords: CFD, mesh, meshless, boundary conditions, Navier stokes equation, euler's equations, numerical mathematics

INTRODUCTION

Fluid flows were traditionally studied either analytically or using experimental settings for over two centuries. However with the advent of better computing facilities and algorithms, the numerical angle of studying fluid flows began to get more popular starting from the early 1970s (Blazek, 2015). These techniques have been collectively called computational fluid dynamics(CFD) and symbolise the combination of physics and numerical mathematics.

Computational fluid dynamics is a branch of fluid dynamics that is very useful in simulating and studying fluid flows, heat flows, mass transfer, etc in places where real experiments would be too expensive or time consuming to perform. The current fluid dynamics algorithm addresses the numerical process introduced by Claude-Louis Navier and Gabriel Stokes into the Euler equations which are universally now referred to as Navier Stokes equations (Jeong & Seong, 2014).

Hu (2012) adds that the science of CFD enables us to predict fluid flows using governing laws of fluid flow. These laws include the conservation of mass, momentum and energy. This method involves the solving of the partial differential equations governing the three laws by making suitable assumptions like boundary and initial conditions of the fluid flowing. However, if the conditions are not exactly modelled, then the results are likely to give wrong results which do not tally with experimental values. The key to this approach is in converting partial differential equations to algebraic equations. There are multiple methods to derive these algebraic equations from the partial differential equations. The most common equations used in this type of study are the Navier stokes equotations, though Lattice Boltzman equations too are considered. But both these are derived from the Euler equations.

CFD saves a lot of time and money to run flow simulation experiments in the virtual world to give very useful results related to the decisions that need to be made in relation to the fluid flows. However CFD has become possible only with the development of newer and more powerful computing machines and software.

Characteristics of Conventional CFD Methods

The conventional CFD methods can be classified into Finite difference, finite element, finite volume and boundary element methods (Garg and Pant, 2018). All these methods had one thing in common, the division of the area into subdomains which were fixed in nature. Each subdomain was classified using a mesh created with fixed points. The larger the number of points, the greater is the accuracy of the simulation. This mesh created could be done in both two dimensions and three dimensions (Peiró & Sherwin, 2005).

The oldest of the methods was finite difference methods. Richardson, (1911) was the first to devise a numerical method of solving the euler equations by dividing space into grid cells and then using the finite difference approximations of Bjerknes primitive finite differential equations to predict the weather. This led to the development of the Finite difference method. This effort evolved later by multiple efforts to finally become a

International Journal of Research in Science and Technology

Volume 9, Issue 2: April - June, 2022

very accurate method of simulating fluid flows. From the early 1980s the first commercial software programs that execute computational fluid dynamic experiments appeared in the marketplace (Fluent, 2002).

Later the finite element method and the finite volume method were conceived (Peiró, & Sherwin, 2005). Today mostly the finite element method and the finite volume methods are used for fluid simulation experiments. The finite difference method has lost favour among researchers due to its inherent difficulty in conceptualising certain problems. The Finite element method was based on the Galerkin method of solving the partial differential equations derived by restating the conservation equations. This is a very accurate method of simulating in both solids and fluids, but takes a long time to both create and to run. The biggest drawback in these methods was the long model development time and long test run times. Ansys software pioneered the finite element method. But later with the merger of Ansys with Fluent, another method started gaining more popularity for fluid flow simulations. On account of its greater speed of execution and testing, the finite volume methods(Jeong & Seong, 2014) have become the de facto method to use for fluid flow simulations even though finite element methods can give similar accuracy in answers. But the finite volume approach is not so sensitive to changes in mesh characteristics like the finite element methods.

The finite volume methods use an integral form of the partial differential equations derived from the conservation equations. The value of the conservation equations are averaged across the volume of the study. In the first step the domain is divided into finite volumes or cells and then the equations are solved for each domain. The accuracy of finite volume methods when compared to finite element methods and experiments by Ahamed et al(1998) who validated the accuracy of this approach as against the two other methods.

The visualisation of the two approaches is shown in the figure below



Finite volume mesh

Experiments, however accurate, still miss out on plenty of variables due to lack of resolution and equipment. This can be overcome by appropriate computational fluid dynamics methods.

The mesh used can be of various shapes and sizes like tetrahedron, hexahedron, polyhedron, prism, etc (Jeong & Seong, 2014) Among these various mesh types, the hexahedron is the most accurate, but the most common one in use is the tetrahedron, on account of its ease of use, though it is less accurate (Biswas & Strawn, 1998)

The general procedure used in mesh approach to these simulations was suggested by Garg and Pant (2018)

- 1. Generate the mesh
- 2. Set the conditions
- 3. Discretise the equations
- 4. Solve the algebraic equations

However the standard methods have the huge problem of creating sufficiently large mesh cells to ensure sufficient accuracy in the study. The run also takes a very long time to complete. Another huge problem in the mesh approach is that most of the cells do not figure in the study at all, but are recorded as observations in any case. This makes a bulk of the recorded values worthless from the point of view of the final results of the simulation.

International Journal of Research in Science and Technology

Volume 9, Issue 2: April - June, 2022

Characteristics of Non Mesh CFD Methods

In cases where mesh is difficult to model like bends, nano pipes, compressed meshes, etc conventional methods tend to give errors in estimated values at the end of simulation (Garg and Pant 2018). Added to the extra time needed to model and run the simulation models, makes researchers look for more convenient alternatives. This resulted in experiments with meshless methods. The first meshless method tried out was smoothed particle hydrodynamics (Gingold & Monaghan, 1977). Till date this is the most applied method in meshless approaches to fluid simulations. This was followed by the development of the Diffuse element method and Dissipative particle dynamics in 1992. Among the plethora of methods developed are the Finite pointset method (Tiwari, & Kuhnert,2003), finite mass method (2000), finite point method, smoothed point interpolation method, repeated replacement method (Liu, 2009) and generalised strain meshfree formulation (Oliveira & Portela, 2016)

The procedure used for these simulations according to Garg and Pant (2018) is

- 1. Set the conditions (Note absence of setting the mesh)
- 2. Discretise the equations
- 3. Solve the equations

The method can be classified into three forms, strong, weak and strong-weak (Garg & Pant, 2018). The various methods under strong form where the problem is modelled using partial differential equations are

- 1. Smoothed particle hydrodynamics
- 2. Reproducing kernel particle
- 3. Generalised finite difference method
- 4. Finite point method
- 5. Radial point collation method

The weak form collection which is an integral statement of a differential equation includes(Garg and Pant, 2018)

- 1. Element free Galerkin method
- 2. Meshless local petrov Glerkin method
- 3. Partition of unity
- 4. Point interpolation method
- 5. H-p clouds method

The strong weak form uses both the strong form and the local weak form to discretise the system equations of the same problem. It is a combination of the two methods mentioned above. Different nodes use different equations. The boundary nodes use the local weak form, while other nodes use the strong form (Garg and Pant, 2018).

The general steps to solving these problems are different from the conventional CFD procedure in the first few steps as no mesh is used here

In all these cases, the simulation is extremely simple and quick to formulate and runs at a much faster speed when compared to the conventional mesh based techniques. The accuracy of the results too are comparable to the conventional methods used.

DISCUSSIONS AND CONCLUSION

Zhang et al (2003) have studied a conventional method (finite element method) and a mesh free simulation method (Reproducing kernel particle method) and shown that the results in both cases are extremely accurate when compared to analytical methods and much simpler to arrive at results of the simulation. Considering the dual bench mark of conventional meshed methods and the gold standard analytical methods, mesh free methods are a quick and easy method to adopt if the conditions of the problem are carefully studied and modelled. The modelling time of mesh free methods is typically 50% or less than with conventional mesh methods. This makes simulating complex problems much easier and requires less technical expertise to model. However, being a disparate set of methods, a number of approaches have come up and the number of software available to run the simulations is restricted to older tried and tested methods like smoothed particle hydrodynamics and finite pointset methods. Most other methods do not have commercial software available and are restricted to

ISSN 2394 - 9554

Volume 9, Issue 2: April - June, 2022

algorithms and code written by research groups which are not very user friendly like the conventional mesh methods. The use of mesh free methods also far outnumbers the mesh methods due to their flexibility and the ease of formulation. Mixed modelling is another advantage in mesh free methods allowing a researcher to choose which method to use in which sections of the simulation study (Liu, 2016). Sakai et al(2020) have used mesh free methods to simulate multiphase fluid flows with a high degree of accuracy.

REFERENCES

- 1. Ahmad, N., Rappaz, J., Desbiolles, J. L., Jalanti, T., Rappaz, M., Combeau, H., Stomp, C. (1998). Numerical simulation of macrosegregation: a comparison between finite volume method and finite element method predictions and a confrontation with experiments. Metallurgical and Materials Transactions A, 29(2), 617-630.
- 2. Biswas, R., & Strawn, R. C. (1998). Tetrahedral and hexahedral mesh adaptation for CFD problems. Applied Numerical Mathematics, 26(1-2), 135-151.
- 3. Blazek, J. (2015). Computational fluid dynamics: principles and applications. Butterworth-Heinemann.
- 4. Fluent, I. (2002). Fluent 14.5 user guide. Fluent Inc., Lebanon. NH-03766.
- 5. Gingold, R. A. and Monaghan, J. J. [1977] "Smoothed particle hydrodynamics-theory and application to non-spherical stars," Mon. Not. R. Astron. Soc. 181, 375–389.
- 6. Hu, H. H. (2012). Computational fluid dynamics. In Fluid mechanics (pp. 421-472). Academic Press.
- Jeong, W., & Seong, J. (2014). Comparison of effects on technical variances of computational fluid dynamics (CFD) software based on finite element and finite volume methods. International Journal of Mechanical Sciences, 78, 19-26.
- 8. Liu, G. R. (2009). Meshfree methods: moving beyond the finite element method. CRC press.
- 9. Oliveira, T.; Portela, A. (December 2016). "Weak-form collocation A local meshless method in linear elasticity". Engineering Analysis with Boundary Elements.
- 10. Peiró, J., & Sherwin, S. (2005). Finite difference, finite element and finite volume methods for partial differential equations. In Handbook of materials modeling (pp. 2415-2446). Springer, Dordrecht.
- Richardson, L. F. (1911). IX. The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam. Philosophical Transactions of the Royal Society of London. Series A, Containing Papers of a Mathematical or Physical Character, 210(459-470), 307-357.
- 12. Sahil Garg & Mohit Pant, (2018), Meshfree Methods: A Comprehensive Review of Applications, International Journal of Computational Methods Vol. 15, No. 3
- 13. Sakai, M., Mori, Y., Sun, X., & Takabatake, K. (2020). Recent progress on mesh-free particle methods for simulations of multi-phase flows: A review. KONA Powder and Particle Journal, 37, 132-144.
- 14. Sayma, A. (2009). Computational fluid dynamics. Bookboon.
- 15. Tiwari, S., & Kuhnert, J. (2003). Finite pointset method based on the projection method for simulations of the incompressible Navier-Stokes equations. In Meshfree methods for partial differential equations (pp. 373-387). Springer, Berlin, Heidelberg.
- 16. Zhang, L. T., Wagner, G. J., & Liu, W. K. (2003). Modelling and simulation of fluid structure interaction by meshfree and FEM. Communications in Numerical Methods in Engineering, 19(8), 615-621.