

Lecture 40

IMPORTANT ASPECTS OF REAL LIFE CFD ANALYSIS

40.1 INTRODUCTION

CFD analysis provides an approximate solution of the underlying flow problem. An analyst must be aware of the approximations made at different stages (modelling as well as simulation stages) and their impact on the solution accuracy. Ideally, the following aspects should be addressed:

- A quantitative bound on the solution accuracy should be obtained as part of the simulation process.
- Numerical solution must be verified, and validated with available experimental data if possible.

In this lecture, we look at some of these aspects. Further, CFD analysis of real life problems invariably requires modelling of complex geometries and solution of large scale algebraic systems which cannot be solved on a serial workstation. We would outline some discretization methods for complex geometries and solution techniques for parallel computing.

40.2 VERIFICATION AND VALIDATION

CFD simulation represents an approximation to a real world problem. Approximations are involved in each stage of numerical simulation process: in mathematical modelling, boundary conditions, discretization and computer solution of the discretized system of equations. Errors in the solutions must be identified and quantified for use of CFD results in engineering analysis and design. This quantification process should be done in two steps: (a) verification and (b) validation (AIAA, 1998).

40.2.1 Verification

Verification stands for quantitative estimation of the closeness to the numerical simulation results to the exact solution of the mathematical model. The verification process requires comparison of the computational solution with known analytical solutions OR high-accuracy benchmark solutions. CFD simulation of any flow problem must be verified as follows:

- **Grid independence test:** Perform a careful grid independence test in spatial as well as temporal domain by systematically refining grid size and time step used in CFD simulation. This step would yield a grid-independent solution.
- **Bench-mark comparison:** To establish the accuracy of the CFD solution, compare it with analytical solution (if available) or high-accuracy benchmark solution (which may have been obtained using some other numerical scheme and a high-resolution grid).

Note that the verification process provides a quantitative measure of accuracy of CFD solution with respect to the mathematical model used in CFD analysis.

40.2.2 Validation

Validation is the process of determining the closeness of the approximate numerical simulation to actual real world problem (AIAA, 1998). CFD simulations should ideally be validated with experimental measurements performed on the real system (or its physical model). In real life applications, it may not be possible to obtain detailed experimental data for the physical system. In such cases, validation is normally done by comparing the numerical simulation with experimental data obtained for a sub-system. For further details on the validation process, refer AIAA(1998) and Versteeg and Malalasekera (2007) .

40.3 METHODS FOR COMPLEX GEOMETRIES

Practical flow problems invariably involve complex geometries. For these problems, generation of the suitable grid(s) for CFD simulation is the most demanding aspect of CFD modelling. The type of grid is dictated by the numerical method used for numerical simulation as well as requirements of the flow physics (e.g. need to provide fine mesh close to the wall to resolve boundary layer).

40.3.1 Unstructured grid techniques

With finite volume and finite element based CFD analysis of flow in complex geometries can be modelled using unstructured grids. The following guidelines should be helpful in generation of a good quality grid for CFD analysis:

- Instead of attempting to generate volume mesh in the entire geometry, decompose the complex problem domain in many small sub-domains. Mesh each sub-domain separately.
- For a properly graded grid, first mesh the edges, then the surfaces of a sub-volume. Thereafter, generate the volume mesh.
- Check quality of the mesh before using it in CFD simulation.
- Care should be exercised in merging the grids of the subdomain. One of the most common errors is creation of an artificial wall boundary between two sub-domains while merging. This must be taken care of.
- In finite volume analysis, hexahedral finite volumes are usually preferred over tetrahedral (due to better accuracy obtained in interpolation and integration with the former). Hence, tetrahedral and wedge elements should be confined to regions too complex to mesh using hexahedral elements.

40.3.2 Structured grid methods

We have three options with structured grids for complex geometries:

1. **Cartesian grids** in which use a stair-case approximation for the curved parts of the complex domains.
2. **Body-fitted structured grids** which are based on mapping of the complex physical domain onto a rectangular computational domain. These require transformation of governing equations in curvilinear co-ordinates.
3. **Block-structured Cartesian grids** are based on the division of the problem domain into smaller sub-domain with each sub-domain having a different Cartesian grid. Thus, finer grids can be used in regions near solid boundaries providing a closer approximation of the problem geometry.

Initially, use of stair-case approximation for the curved parts of the complex domains was only option with Cartesian grids. However, recent developments in immersed boundary

methods (IBM) have brought the Cartesian grids back in favour for arbitrarily complex geometries. IBMs permit a much better representation of actual geometries of problem domain while still employing a Cartesian grid (which may be block-structured). For further details, please see Peskin (2002), Mittal and Iaccarino(2005).

40.4 PARALLEL IMPLEMENTATION

Discretization of a complex real life problem domain (say flow around an aircraft or an automobile) usually requires a very large number of grid nodes/elements (order of a few millions or even billions). The resulting system of equations of such a large order cannot be solved on serial workstations, and one must use a parallel cluster which could be

- A shared memory machine (wherein all processors share a large random access memory), or
- A distributed memory cluster where each processor (or node) has its own dedicated memory.

Parallel implementation of a CFD code on each type of machine would be somewhat different. However, in either context, the bigger problem must be broken into smaller sub-problems which can be solved in parallel on a set of processors. Two popular approaches for this break-up are

- **Domain decomposition:** Divide the problem domain into a set of non-overlapping subdomains, generate grid on each subdomain, and proceed with the solution process. Each processor (or processor core) is assigned a subset of these subdomains. This approach can be used with any discretization scheme.
- **Grid partitioning:** Use graph-partitioning tools (such as METIS and JOSTLE) to divide the overall discretized problem into a set of sub-grids.

In parallel implementation, care must be taken to balance the computing load on each processor. Further, iterative schemes which can be easily parallelized have to be chosen for solution of the discrete algebraic system.

REFERENCES/FURTHER READING

AIAA (1998). *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*, AIAA Guide G-077-1998.

Chung, T. J. (2010). *Computational Fluid Dynamics*. 2nd Ed., Cambridge University Press, Cambridge, UK.

Ferziger, J. H. And Perić, M. (2003). *Computational Methods for Fluid Dynamics*. Springer.

Mittal, R. and Iaccarino, G. (2005). Immersed Boundary Methods, *Annual Review of Fluid Mechanics*, vol. 37, pp. 239–261.

Peskin, C. S. (2002). The immersed boundary method, *Acta Numerica*, 11, pp. 1–39.

Versteeg, H. K. and Malalasekera, W. M. G. (2007). *Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Second Edition (Indian Reprint) Pearson Education.