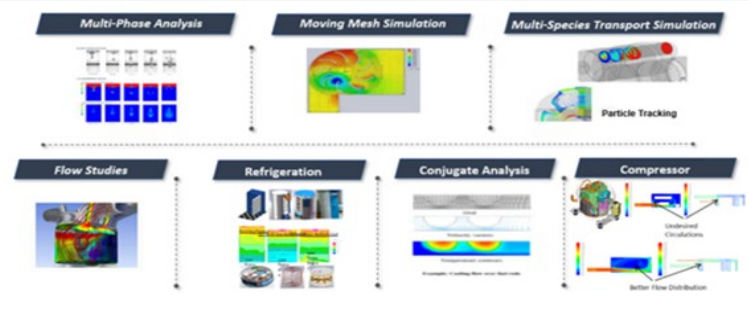
CFD Technologies used in Industries, Problems and Encounter

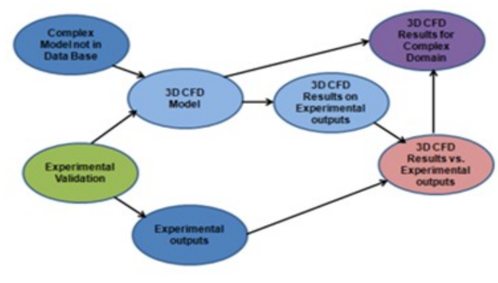
Mr. Sushilkumar Jogdhankar
Sr. Team Leader, CFD Division,
Onward Technologies Ltd. Pune

Computational Fluid Dynamics is the analysis of the systems involving fluid flow, heat transfer and associated phenomenon such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial applications areas. Some examples are: aerodynamics of aircrafts and vehicles, hydrodynamics of ships, combustion, turbo machinery, electrical and electronic engineering, and chemical process engineering, external and internal environment of buildings, marine engineering, environmental engineering, hydrology and oceanography, metrology, biomedical engineering etc. from the 1960s onwards, the aerospace industry has integrated CFD technique into design, R & D and manufacture of aircrafts and jet engines. More recently the methods have been applied to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Furthermore, motor manufactures now routinely predict drag forces, under bonnet airflow and the in-car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

The ultimate aim of development in the CFD field is to provide a capability comparable to other CAE (Computer-Aided Engineering) tools such as stress analysis codes. Upsurge of interest and CFD is poised to make an entry into the wider industrial community in the 1990s.



CFD Overview



CFD Validation to prediction

In solving fluid flow problems we need aware that the underlying physics is complex and the results generated by a CFD code are at best as good as the physics and chemistry embedded in it and at worst as good as its operator. The user of the code must have skills in a number of areas. Prior to setting up and running a CFD simulation, there is a stage of identification and formulation of the flow pattern in terms of physical and chemical phenomena that need to be considered. Flow inside a CFD solution domain is driven by the boundary conditions.

In a sense the process of solving a field problem (e.g. a fluid flow) is nothing more than the extrapolation of a set of data defined and a boundary contour or surface into the domain interior. It is, therefore, of paramount importance that we supply physically realistic, well-posed boundary conditions, otherwise sever difficulties are encountered in obtaining solution. The single most cause of rapid divergence of CFD simulation is the inappropriate selection of boundary conditions. A good understanding of the numerical solution algorithm is also crucial.

Three mathematical concepts are useful in determining the success or otherwise of such algorithm: convergence, consistency and stability. Convergence is the property of a numerical method to produce a solution with approaches the exact solution as the grid spacing; control volume size or element size is reduced to zero. Consistent numerical schemes produce systems of algebraic equations, which can be demonstrated to be equivalent to the original governing equation, as the grid spacing tends to zero. Stability is associated with damping of errors as the numerical methods proceeds. If a technique is not stable even round off errors in the initial data can cause wild oscillations or divergence. Round off errors would swamp the solution long before a grid spacing of zero is actually reached.

Engineers need CFD codes that produce physically realistic results with good accuracy in simulations with finite (sometimes quiet coarse) grids. The bounded ness property is akin to stability and requires that in a linear problem without source the solution be bounded by the maximum and minimum boundary values of the flow variables.

Bounded ness can be achieved by placing restrictions on the magnitude and signs of the coefficient of the algebraic equations. Although flow problems are non-linear it is important to study the bounded ness of a finite volume scheme for closely related. Specification of the domain geometry and grid design is the main tasks at the input stage. The two aspects that characterize good results are convergence of the iterative process and grid independence. Progress towards a converged solution can be greatly assisted by careful selection of the settings of various relaxation factors and acceleration devices and they are problem dependent.

At the end of a simulation the must take a judgment whether the results are "good enough" .It is impossible to assess the validity of the models of physics and chemistry embedded in a program as complex as a CFD code or the accuracy of its final results by any means other than comparison with experimental test work. Anyone wishing to use CFD in a serious way must realize that it is no substitute for experimental, but a very powerful additional problem-solving tool. Validation of a CFD code requires highly detailed information concerning the boundary conditions of a problem and generates a large volume of results.

To validate these in a meaningful way it is necessary to produce experimental data of similar scope. This may involve programs such as velocity measurement with hot-wire or laser anemometer. Sometimes the facilities to perform experimental work may not (yet) exist in which case the CFD users must rely on

- Previous experience
- Comparisons with analytical solutions of similar but simpler flow
- Comparisons with high quality data from closely related problems reported in the literature.

Excellent sources of the last type of information can be found in transactions of ASME (in particular the Journal of Fluid Engineering, Journal of Engineering for Gas Turbines and Power and Journal of Heat Transfer), AIAA Journal, Journal of Fluid Mechanics and Proceedings of the Imeche.It is clear that there are guidelines for good operating practice, which can assist the user of a CFD code; the main ingredients for success in CFD are experience and a thorough understanding of the physics of fluid and the fundamentals of the numerical algorithms.