ISSN 2395-1621

CFD - Introduction and Application

#1Prof.P.P.Awate ,#2Prof. N.V.Hargude

1pankajawatepp@gmail.com
2nvhargude@gmail.com

#12Associate Professor PVPIT Budhgaon

ABSTRACT

The paper deals with brief knowledge about Computational fluid dynamics (CFD) by introduction, Methodology, advantages and live applications of CFD. CFD is nothing but analysis of flowing fluid under various parameters like temperature, pressure, velocity etc. CFDs were originally developed in the early 1990s in London. It is used in various field like engineering, medical, agricultural etc. In the method of CFD first we have to mesh it and then by applying boundary conditions it can be analysed. Meshing can be done in either triangular or quadrilateral shapes. Its main advantage is to reduce the cost of analysis of flowing fluid. It can be used to reduce noise in gas turbine, analysis of velocity of flowing fluid in the boiler, analysis of air resistance to running vehicle. There are different commercial CFD codes are used for analysis.In case of nozzle parameters of flowing fluid, velocity and pressure are analysed by using CFD method, in which at outlet of converging nozzle, velocity increases and pressure decreses.CFD has good time efficiency. CFD has become in indispensable tool in design, development and evaluation of new industrial equipment and process like elbow pipe, boiler, blast deflector, automobile, gas turbine etc. CFD having ease of access and low capital requirement which can cause lead to over-trading.

ARTICLE INFO

Article History :

Received:2nd November 2015

Received in revised form:

4th November 2015

Accepted : 5th November 2015

Keywords - Methodology, advantages and live applications etc.

I.INTRODUCTION

Computational fluid dynami**cs**, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.CFD uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier-Stokes and allied equations) for predefined geometries and boundary conditions. The result is a wealth of predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

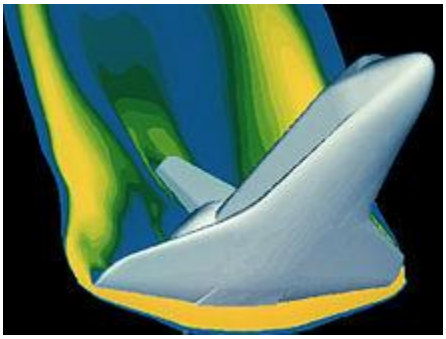
II. BACKGROUND AND HISTORY

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow.These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.Historically, methods were first developed to solve the linearized potential equations.

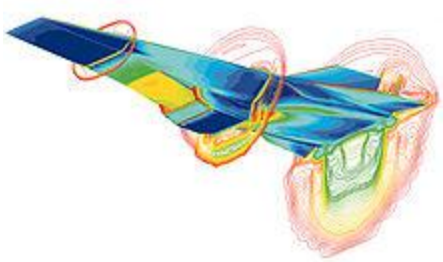
Two-dimensional (2D) methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s.

One of the earliest type of calculations resembling modern CFD are those by Lewis Fry Richardson, in the sense that these calculations used finite differences and divided the physical space in cells. Although they failed dramatically, these calculations, together with Richardson's book "Weather prediction by numerical process", set the basis for modern CFD and numerical meteorology. In fact, early CFD calculations during the 1940s using ENIAC used methods close to those in Richardson's 1922 book.

The computer power available paced development of three-dimensional methods. Probably the first work using computers to model fluid flow, as governed by the Navier-Stokes equations, was performed at Los Alamos National Lab, in the T3 group. This group was led by Francis H. Harlow, who is widely considered as one of the pioneers of CFD. From 1957 to late 1960s, this group developed a variety of numerical methods to simulate transient two-dimensional fluid flows, such as Particle-in-cell method (Harlow, 1957),Fluid-in-cell method (Gentry, Martin and Daly, 1966), Vorticity stream function method (Jake Fromm, 1963), and Marker-and-cell method (Harlow and Welch, 1965). Fromm's vorticity-stream-function method for 2D, transient, incompressible flow was the first treatment of strongly contorting incompressible flows in the world.



A computer simulation of high velocity air flow around the Space Shuttle during re-entry.



A simulation of the Hyper-X scramjet vehicle in operation at Mach-7

Basic Procedure for CFD Analysis-

- During preprocessing
  - The geometry (physical bounds) of the problem is defined.
  - The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.
  - The physical modeling is defined – for example, the equations of motion + enthalpy + radiation + species conservation
  - Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.
- The simulation is started and the equations are solved iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

III.METHODOLOGY

Following Methods are using for CFD Analysis

Discretization methods -The stability of the selected discretisation is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretisation handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces.

Some of the discretisation methods being used are:

Finite volume method - The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion).

In the finite volume method, the governing partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes. This discretization guarantees the conservation of fluxes through a particular control volume. The finite volume equation yields governing equations in the form,

$$\frac{\partial}{\partial t}\iiint Q\,dV + \iint F\,d\mathbf{A} = 0,$$

where Q is the vector of conserved variables, F is the vector of fluxes (see Euler equations or Navier–Stokes equations), V is the volume of the control volume element, and $\mathbf{A}$ is the surface area of the control volume element.

Finite element method- The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. However, the FEM formulation requires special care to ensure a conservative solution. The FEM formulation has been adapted for use with fluid dynamics governing equations. Although FEM must be carefully formulated to be conservative, it is much more stable than the finite volume approach.[41] However, FEM can require more memory and has slower solution times than the FVM.

In this method, a weighted residual equation is formed:

$$R_i = \iiint W_i Q\,dV^e$$

Where R_i is the equation residual at an element vertex i, Q is the conservation equation expressed on an element basis, W_i is the weight factor, and V^e is the volume of the element.

Finite difference method - The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

where Q is the vector of conserved variables, and F, G, and H are the fluxes in the x, y, and z directions respectively.

Spectral element method- Spectral element method is a finite element type method. It requires the mathematical problem (the partial differential equation) to be cast in a weak formulation. This is typically done by multiplying the differential equation by an arbitrary test function and integrating over the whole domain. Purely mathematically, the test functions are completely arbitrary - they belong to an infinitely dimensional function space. Clearly an infinitely dimensional function space cannot be represented on a discrete spectral element mesh. And this is where the spectral element discretization begins.

Advantages

A key advantage of CFD is that it is a very compelling, non-intrusive, virtual modeling technique with powerful visualization capabilities, and engineers can evaluate the performance of a wide range of HVAC/IAQ system configurations on the computer without the time, expense, and disruption required to make actual changes onsite.

CFD has seen dramatic growth over the last several decades. This technology has widely been applied to various engineering applications such as automobile and aircraft design, weather science, civil engineering, and oceanography. Today, the HVAC/IAQ industry is one of the fields that has initiated utilizing CFD techniques widely and rigorously in its design.

IV.PRACTICAL ADVANTAGES OF EMPLOYING CFD

The many reasons CFD is being widely used today are as follows:

- CFD predicts performance before modifying or installing systems:

- Without modifying and/or installing actual systems or a prototype, CFD can predict which design changes are most crucial to enhance performance.
- CFD provides exact and detailed information about HVAC design parameters:
- The advances in HVAC/IAQ technology require broader and more detailed information about the flow within an occupied zone, and CFD meets this goal better than any other method, (i.e., theoretical or experimental methods).

CFD Saves Cost and Time:

CFD costs much less than experiments because physical modifications are not necessary. (Note that the cost and time for physical changes/modifications increase almost exponentially as the size of the system increases).

CFD is Reliable:

The numerical schemes and methods upon which CFD is based are improving rapidly, so CFD results are increasingly reliable. CFD is a dependable tool for design and analyses.

Where Can CFD be Utilized?

In validation/optimization of HVAC design parameters:

CFD data can be utilized to validate various design parameters such as the location and number of diffusers and exhausts, and temperature and flow rate (CFM) of supplied air to meet design criteria. For example, CFD simulation helps design verification of the following systems: natural ventilation systems, displacement ventilation systems, raised floor system, atrium smoke system, etc.

In modification/improvement of malfunctioning HVAC systems:

The system with suggested modifications can be simulated computationally without actual physical modifications to the existing systems. The information from CFD reveals what modification satisfies the design criteria.

Comparisons between alternative systems:

Under some circumstances, there may be several different options for designing HVAC systems for a space (for example, mixing ventilation or displacement ventilation). Computer simulation data can provide crucial information to find the best possible system.

An engineering investigation:

CFD analysis of temperature, velocity and chemical concentration distributions can help engineers understand the problem correctly and provide ideas for the best resolution.

Examples of CFD Applications for HVAC Systems

- General office/room simulations
- Contaminant/species simulations
- Fume hood design
- Copy machine rooms (VOC)
- Contamination control chemical lab design
- Industrial ventilation design
- Smoking lounges
- External building flows
- Problem solving simulations
- AHU mixing enhancement investigation
- Fire and smoke management
- Building atria fire simulations
- Warehouse fire simulations
- Educational facilities
- Libraries
- Classrooms
- Swimming pool ventilation
- Medical facilities (operating rooms)
- Clean room simulations
- Animal and plant environments
- Enclosed vehicular facilities
- Halls, stadiums, arenas, and places of assembly
- Computer cluster rooms

Commercial CFD codes:

- Fluent – CFD software
- Polyflow – for viscous / viscoelastic flows
- MixSim – for rapid mixing tank simulations
- I-Deas – 3D solid modeler

V.CONCLUSION

CFD is a mature and reliable tool for evaluating many industrial processes. However, appropriate knowledge and experience are very essential for obtaining reliable results.

Also Delaunay method is used for implementation, Various two dimensional complex geometries were mapped using the implemented code, Quality of mesh obtained is good, Good time efficiency and Reduces the development cost to new products.

REFERENCES

[1] Milne-Thomson, L.M. (1973). Theoretical Aerodynamics.Dover Publications.ISBN 0-486-61980-X.

[2]Richardson, L. F.; Chapman, S. (1965).Weather prediction by numerical process.Dover Publications.

[3]Hunt (1998). "Lewis Fry Richardson and his contributions to mathematics, meteorology, and models of conflict".Annual Review of Fluid Mechanics **30**.Bibcode:1998AnRFM..30D..13H. doi:10.1146/annurev.fluid.30.1.0.

[4]"The Legacy of Group T-3". Retrieved March 13, 2013.[5]Harlow, F. H. (2004). "Fluid dynamics in Group T-3 Los Alamos National Laboratory:(LA-UR-03-3852)". Journal of Computational Physics (Elsevier) **195** (2): 414–433. Bibcode:2004JCoPh.195..414H. doi:10.1016/j.jcp.2003.09.031.

[6]F.H. Harlow (1955). "A Machine Calculation Method for Hydrodynamic Problems". Los Alamos Scientific Laboratory report LAMS-1956.

[7]Gentry, R. A., Martin, R. E., Daly, J. B. (1966)."An Eulerian differencing method for unsteady compressible flow problems".Journal of Computational Physics**1** (1): 87–118. Bibcode:1966JCoPh...1...87G. doi:10.1016/0021-9991(66)90014-3.

[8]Fromm, J. E.; F. H. Harlow (1963)."Numerical solution of the problem of vortex street development".Physics of Fluids**6**: 975. Bibcode:1963PhFl....6..975F. doi:10.1063/1.1706854.

[9]Harlow, F. H.; J. E. Welch (1965)."Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface" (PDF).Physics of Fluids**8**: 2182–2189. Bibcode:1965PhFl....8.2182H. doi:10.1063/1.1761178.

[10]Hess, J.L.; A.M.O. Smith (1967)."Calculation of Potential Flow About Arbitrary Bodies". Progress in Aerospace Sciences **8**: 1–138. Bibcode:1967PrAeS...8....1H. doi:10.1016/0376-0421(67)90003-6.

