International Journal of Inventions in Engineering & Science Technology http://www.ijiest.in

(IJIEST) 2017, Vol. No. 3, Jan-Dec **e-ISSN: 2454-9584; p-ISSN: 2454-8111**

APPLICATIONS OF CFD IN GRID GENERATION AFTER MODELING

Dr Arun Prakash Singh

Associate Professor Department of Physics, Hindu College Moradabad

ABSTRACT

Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction . CFD is commonly accepted as referring to the broad topic encompassing the numerical solution by computational methods, of the governing equations which describe fluid flow, the set of Navier-Stokes equations, continuity and any additional conservation equations, e.g.- energy or species concentrations.

INTRODUCTION

Development of computational fluid dynamics has now reached a state of maturity, which enables it to stimulate the flow of past practical car design configurations. It can thus play a very useful role in car design and development. CFD codes can be used to determine different loads on car and thus provide necessary input for the structural design. They can also be used to study and optimize the flow patterns and thus help to design car in efficient shape.

What do we mean when we speak of simulating a fluid flow on a computer? In simplest terms, the computer solves a series of well-known equations that are used to compute, for any point in space near an object, the velocity and pressure of the fluid flowing around that object .

WHAT IS COMPUTATIONAL FLUID DYNAMICS (C.F.D.)?

Before we answer this question let's recall a quote by 'James Clerk Maxwell', "All the mathematical sciences are founded on relation between physical laws and laws of numbers, so the aim of exact science is to reduce the problem of nature to the determination of quantities by operations with numbers."

Similarly the physical aspect of any fluid flow are governed by three fundamental principles namely-

1. **Mass is conserved(continuity equation)**
2. **Newton's second law (Navier stokes equation)**
3. **Energy is conserved (energy equations)**

These fundamental physical principles when expressed in terms of mathematical equations while making very few assumptions lead to a set of nonlinear partial differential equations (P.D.E.). C.F.D. is the art of replacing these PDE's with discrete algebraic form which in turn are solved by numerical methods to obtain numbers for the flow field values at discrete points in time and/or space. The end product of CFD is indeed a collection of numbers or values.

In the previous generation, there was a two way approach of designing. The theoretical approach immediately followed by experimental analysis. However, the above approach was time consuming and error prone.

CFD acts as a connecting link between the theoretical design and experimental analysis and has considerably reduced the time required of vehicle development process (VDP) as well as the frequent experimental analysis. Over the years CFD has matured to a point where it now has the capability to supplant both experimental and theoretical approaches to some specific problems.

This indeed has been possible because of high speed digital computers since CFD generally require the repetitive manipulations of many thousands, even millions of numbers, a task that is humanly impossible without the aid of computers. Therefore advances in CFD and its applications to problems of more and more detail and sophistication are intimately related to advances on computer hardware, especially in regard to storage and execution speed. So if you ever wondered what is driving the development of super computers, think of the CFD community.

All CFD codes contain three main elements:

(i) **A pre-processor:** which is used to input the problem geometry, generate the grid, define the flow parameter and the boundary conditions to the code.

(ii) **A flow solver:** which is used to solve the governing equations of the flow subject to the conditions provided. There are three different methods used as a flow solver:

a) Finite difference method,
b) Finite element method,
c) Finite volume method.

(iii) **A post-processor:** which is used to massage the data and show the results in graphical and easy to read format.

CFD Components

- Human being
- Scientific knowledge expressed mathematically
- Software which expresses problem in scientific terms
- Hardware which performs calculations dictated by the software
- Human being who inspects and interprets their results

Steps in CFD

1. Grid Generation
2. Disceretization
3. Simulation
4. Computation using (super) computer
5. Post processing

GRID GENERATION:

In CFD the first step after modeling is Grid Generation.

The domain i.e. the flow area under consideration is divided into some shapes like triangles or quadrilaterals and assumption is made that flow properties are constant cell to cell. The way that such a grid is determined is called Grid Generation.

Grid generation is by far the most critical process of CFD analysis apart from selecting the flow equations. The type of grid you choose for a given problem can make or break the numerical solution.

Any successful grid structure should have following essential properties:

1. The grid lines of the same family in the physical plane should not cross each other i.e. no two cells must overlap.
2. The grid distribution must be smooth.
3. The grid system must be capable of resolving all the scales i.e. we should be able to control the grid point distribution.
4. The grid points should be clustered close to the body.
5. Also for fast convergence of any iterative schemes employed for the solution of linear algebraic equation, we must have a cell aspect ratio that is not too large.
6. Cells must be convex in shape.

TYPES OF GRIDS:

Basically there are two types of grids

- Structured grid
- Unstructured grid

Structured grid: In structured three dimensional grids, one can associate with each computational cell an ordered triple of indices (i,j,k) where each indices varies over a fixed range, independently over the value of other indices that varies by +/- 1. Thus if N_i, N_j, N_k are number of cells in i,j,k, index directions respectively, than the number of cells in entire mesh is $N_i*N_j*N_k$.

Types of structured grid

a) O-type grid
b) C-type grid
c) H-type grid

O-type grid: Here one of the grid lines resembles the general shape of the letter O; such grid are typically used for multiple connected body as shown in fig.1.

C-type grid: Here one of the grid lines resembles the letter C as shown in fig 2. C-grids are preferred when one is interested in investigating the wake of a body with fine grids in the wake. In O-grid, a large number of points are present ahead of the body where hardly anything happens. This is specifically avoided in C-type grid.

H-type grid: Here the grid lines form a lattice network similar to the stream function contour lines and the velocity potential contour lines of the irrotational potential flow. Such grids have the good attributes of a C-type grid. They are mostly used in internal flows where the fluid is moving without solid walls.

Figure 3. shows H-type grid for a fluid flow in a pipe. Note the denser grid spacing in the contracted section of the duct. This was specifically done to capture the phenomenon of shockwave, which was likely to occur in that region, in more detail. Figure 4. shows the flow results of H-grade simulation.

Unstructured grid: In contrast to structure grid which reflects some type of consistent geometrical regularity, unstructured grid consists of grid points placed in flow field in a very irregular fashion.

There is nothing about finite volume method (FVM) that demands a structured mesh; it can be applied to mesh cells of any arbitrary shape. This has given rise to the use of unstructured meshes. Unstructured mesh allows for maximum flexibility in matching cells with boundary surfaces and for putting cells where you want them. Unstructured grids require more information to be stored and recovered than structured grids.

A popular type of unstructured grid consists of tetrahedral elements as shown in fig. 5. and fig. 6.

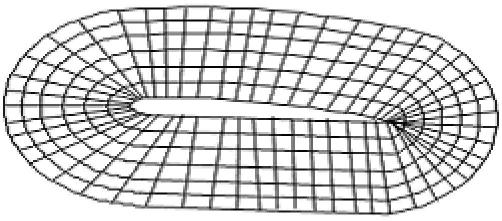

Figure 1. O-type grid around an airfoil

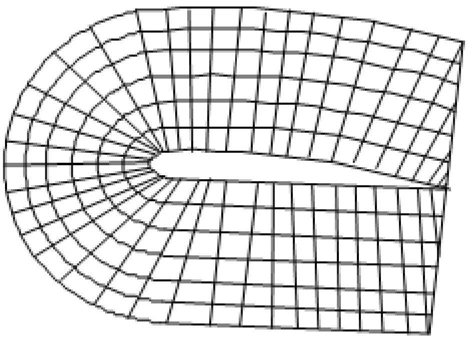

Figure 2. C-type grid around an airfoil

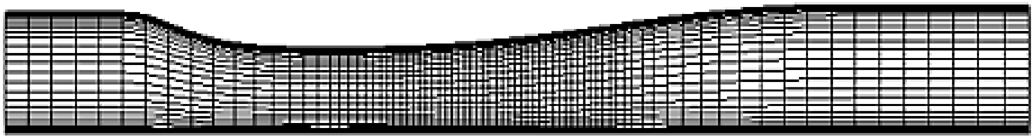

Figure 3. H-type grid of a duct

Figure 4. flow results of H-grid simulation

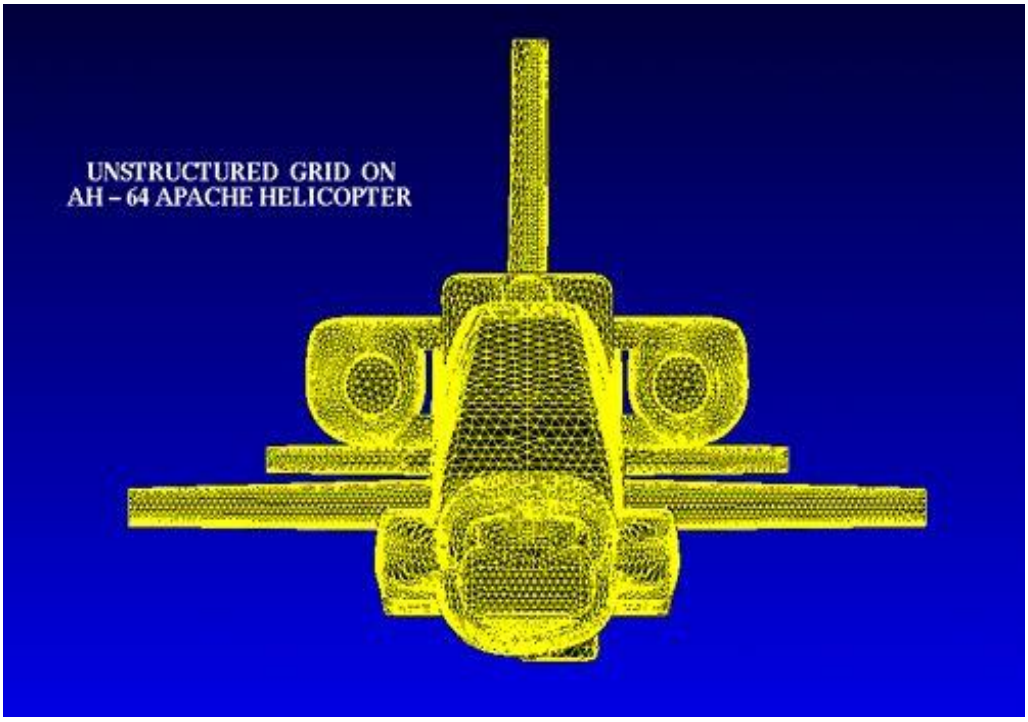

Figure 5. Unstructured grid of an Apache Helicopter

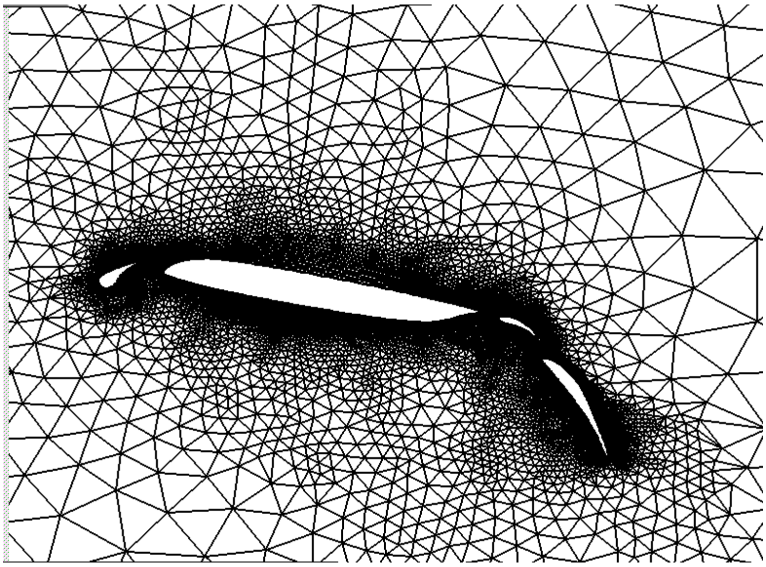

Figure 6. An unstructured grid around a multi element airfoil

CFD SOFTWARES:

The following table shows short description used for various steps in CFD.

STEPS IN CFD	SOFTWARES USED
Geometric acquisition	ICEM-CFD, CATIA, I-I-DEAS,AUTO-CAD OCAD, etc
Grid generation	ICEM-CFD, GRIDGEN, PATRAN, CFD-GEOM**,** PHOENICS, etc
Post processing	ICEM-CFD, PATRAN, EN-SIGHT, etc

OTHER APPLICATIONS:

Automotive:	**Aerospace and Defense:**	**Power Generation:-**
➢ Aerodynamics	➢ Anti Icing	➢ Burners
➢ Valves	➢ Combustors	➢ Combustors
➢ Filters	➢ Seals	➢ Hydropower
➢ Fan design	➢ Tanks	➢ Turbo machinery
➢ pumps	➢ Instrumentation	➢ Silencers

ADVANTAGES:

Some of the advantages of CFD over experiment are:

- Ability to simulate realistic systems. For example CFD may discover significant flow features that otherwise couldn’t be uncovered with physical experiments.
- The cost of CFD is less compared to the cost of performing experiments.
- Parametric studies are easily performed with CFD. A large number of design concepts can be carried out, before the prototype of the design is ever built.
- Detailed local information as well as surface information is obtained from CFD simulation.

LIMITATIONS:

- The input data may involve too much of guess work or imprecision.
- The available computer power for high numerical accuracy may sometimes become hurdle in ongoing research.
- The reliability is greater
  1. -for laminar flow rather than turbulent ones.
  2. -for single-phase flows rather than multiphase flows.
  3. -for chemically inert rather than chemically reactive materials.

REFERENCES:

1. John D. Anderson , Jr., "Computational Fluid Dynamics.", McGraw-Hill, New York, 1995 pp- 3,23-27 , 168-211.
2. Tapan K. Sengupta, "Fundamentals of Computational Fluid Dynamics"., University Press Pvt. Ltd., pp 1-2,123-124.
3. Computational fluid dynamics for Engineer.-by Hoffmann K.A , Edition:- Yr 1989
4. Fluid Power Magazine -April June 2002
5. Dale A. Anderson (1984), Computational Fluid Dynamics and Heat Transfer, Hemisphere
6. Publishing corporation Washington, New York, London.
7. Bouainouche M. et al. *Int. J. Numer. Methods Heat Fluid Flow*. 1997. **7**. 548–564p.
8. Craft T.J. et al. *Int. J. Heat Mass Transfer*. 1993. 36. 2685–2697p.
9. Dianat M. et al. *The Pennsylvania State University, University Park,* 1995.
10. Craft T.J. et al. *Int. J. Heat Fluid Flow.* 1996. **17.** 108–115p.
11. Neil Zuckerman. et al. *Journal of Heat Transfer*. 2005. 127.
12. Chuang S.H. *Int. J. Numer. Methods Fluids.* 2000. 33. 475–498p.
13. Craft T.J. et al. *Int. J. Heat Mass Transfer.* 1993. **36**. 2685–2697p.
14. Merci B. et al. *Int. J. Numer. Methods Heat Fluid Flow.* 2003. **13**. 110–132p.
15. Papageorgakis G.C. et al.*Numer. Heat Transfer*. 1999. 35. 1–22p.
16. Hosseinalipour S.M. et al.*Numer. Heat Transfer Part A.* 1995. 28. 647–666p.
17. Mujumdar A.S. et al. *Appl. Therm. Eng.* 2005. 25. 31–44p.
18. Gibson M.M. et al. *Int. J. Heat Fluid Flow.* 1997. 18. 80–87p.
19. Wilcox D.C. *DCW Industries.* 2002.