ELEMENTS OF COMPUTATIONAL FLUID DYNAMICS

Chapter - 0

Pre-requisites:

a) Knowledge of computer programming.

b) Basic knowledge of Fluid Mechanics-1 and 2.

c) Basic knowledge of and Heat Transfer .

Syllabus

- Introduction to the Computational Fluid Dynamics (CFD).
- CFD modeling techniques.
- Finite Difference Methods.
- Special Topics.

Reference Books

✓Computational Fluid Flow and Heat Transfer by K. Muralidhar T. Sundararajan
✓Computational Fluid Dynamics and Heat Transfer by P. S. Ghoshdastidar
✓Computational Fluid Dynamics by John D. Anderson.
✓An Introduction to Computational Fluid Dynamics by H. Versteeg and W. Malalasekera

What is CFD?

Fluid dynamics is the science of fluid motions.
Fluid flows, in general, is studied in one of the three approaches:

✓Experimental fluid dynamics.

✓Theoretical fluid dynamics.

✓Numerically: Computational Fluid Dynamics.

Computational Fluid Dynamics is the science of predicting fluid flows, heat and mass transfer, chemical reaction kinetics, and other related physical phenomena by solving set of mathematical equations governing those processes by employing numerical methods.

CFD is the science of enumerating properties of fluid in motion.

What is CFD?

CFD provides a qualitative (and even sometimes quantitative) predictions of fluid flow (and heat transfer) by means of three ways:

I. Mathematical modeling (partial differential equations).

II. Numerical methods (discretization and solution techniques).

III. Software tools (solvers, pre- and post-processing utilities).

History of CFD

Lewis F. Richardson
(1881-1953)

- Earliest CFD work in 1910
- Used human computers
- Solved Laplace's equation using finite-difference methods
- Flow over cylinder (potential flow)

History of CFD

Richard Courant
(1888-1972)

Kurt O. Friedrichs
(1901-1982)

Hans Lewy
(1904-1988)

Landmark paper for hyperbolic equations (1928)

Courant, R., Friedrichs, K. and Lewy, H. (1928), "Uber die partiellen Differenzengleichungen der mathematischen Physik", Mathematische Annalon (in German), **100** (1): 32–74.

History of CFD

Von Neumann stability criteria for parabolic problems (1950)

John von Neumann
(1903-1957)

History of CFD

Harlow and Fromm (1963) computed unsteady vortex street using digital computer

Fromm, J.E. and Harlow, F.H. (1963), “Numerical solution of the problem of vortex street development”, Physics of Fluids, **6**: 975.

Harlow and Welch (1965) published a Scientific American article which ignited interest in modern CFD and the idea of computer experiments

Harlow, F.H. and Welch, J.E., (1965), “Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface”, Physics of Fluids, **8**: 2182-2189.

History of CFD

Professor **Brian Spalding** is a founding **father of CFD**

Boundary-layer codes developed in the 1960-1970's (GENMIX by Patankar and Spalding in 1972)

D. Brian Spalding
(1923-2016)

Solution techniques for incompressible flows published through the 1970's (SIMPLE family of algorithms by Patankar and Spalding)

Suhas V. Patankar
(born 1941)

History of CFD

Antony Jameson
(born 1934)

Jameson computed Euler flow over complete aircraft (published 14 journal papers in 1981)

Transonic Airfoil Calculations Using the Euler Equations, Proceedings of IMA Conference on Numerical Methods in Aeronautical Fluid Dynamics, Reading, March 1981, edited by P. L. Roe, Academic Press, 1982, pp. 289-308.

Viscid-Inviscid Interaction on Airfoils Using Euler and Inverse Boundary-Layer Equations (with D. Whitfield and W. Schmidt), US-German Data Exchange Meeting, DFVLR-AAVA, Göttingen, 1981.

Finite Volume Solution of the Euler Equations of Transonic Flow Over Airfoils and Wings Including Viscous Effects (with W. Schmidt, and D.Whitfield), AIAA Paper 81-1265, June 1981, J. Aircraft, Vol. 20, 1983, pp. 127-133.

Viscous Transonic Airfoil Flow Simulation (with J. Longo, and W. Schmidt), DGLR Paper 81-242, Stuttgart, November 1981, Zeitschrift Flugwissenschaften Weltraumforschung, Vol. 7, 1983, pp. 47-56.

History of CFD

Unstructured mesh methods developed in 1990's, Used for aerodynamic calculations in NASA

- Murthy, J.Y., Mathur, S. Periodic flow and heat transfer using unstructured meshes (1997) International Journal for Numerical Methods in Fluids, 25 (6), pp. 659-677.
- Mathur, S.R., Murthy, J.Y. A pressure-based method for unstructured meshes (1997) Numerical Heat Transfer, Part B: Fundamentals, 31 (2), pp. 195-215.
- Mathur, S.R., Murthy, J.Y. Pressure boundary conditions for incompressible flow using unstructured meshes (1997) Numerical Heat Transfer, Part B: Fundamentals, 32 (3), pp. 283-298.
- Murthy, J.Y., Mathur, S.R. Radiative heat transfer in axisymmetric geometries using an unstructured finite-volume method (1998) Numerical Heat Transfer, Part B: Fundamentals, 33 (4), pp. 397-416.
- Murthy, J.Y., Mathur, S.R. Computation of anisotropic conduction using unstructured meshes (1998) Journal of Heat Transfer, 120 (3), pp. 583-591.
- Murthy, J.Y., Mathur, S.R. Finite volume method for radiative heat transfer using unstructured meshes (1998) Journal of Thermophysics and Heat Transfer, 12 (3), pp. 313-321.

Jayathi Murthy
(born 1958)

Span of Computational Fluid Dynamics

- ✓ Computational fluid dynamics (CFD) is a subject that deals with solution of fluid dynamics and heat transfer problems using numerical methods that can be programmed on computer.
- ✓ CFD has emerged as an alternative method to experimental and theoretical method for solving the fluid mechanics and heat transfer problems.
- ✓ CFD is used in designing the products in aerospace, mechanical, chemical, metallurgical, electrical, electronics, and even food processing industries.
- ✓ Although CFD is not likely to replace experiments in near future, the trend is clearly toward computer-aided designs in industries the world over.

Applications of CFD

Contemporary CFD include all disciplines in which the flow of a fluid is taking place with or without heat and/or mass transfer. Some applications of CFD are:

- **Laminar and turbulent flow of air over an airplane wing:** It helps to determine the lift and drag of the aerofoil without doing actual wind tunnel experiments.
- **Study of the external flow of air over the body of a car:** CFD tools are used to reduce the drag over a car, thus, resulting in fuel economy.
- **Design of internal combustion engines:** It helps automotive engineers to better understand internal flow through the engine.
- **Manufacturing processes:** CFD modeling of materials processing applications such as metal cutting, welding, rolling, extrusion, solidification and melting, injection molding, and screw extrusion of plastic parts has greatly enhanced the quality of the end products.
- **Civil engineering applications:** Problems involving rheology of rivers, lakes, estuaries, etc.
- **Environmental engineering applications:** Simulation of heating, air-conditioning, and general air circulation through buildings
- **Naval architecture applications: A**nalysis of hydrodynamic problems associated with ships, submarines, torpedos, etc