COMPUTATIONAL FLUID DYNAMICS AND ITS APPLICATIONS

M. Arokiasamy

[2]Department of Mathematics, Andhra Loyola College, Vijayawada, AP-520008, India
*corresponding author's Email Id: markarokiasamy@yahoo.co.in

Abstract

Transport phenomena includes heat transfer, fluid dynamics and diffusion. Itis the process in which mass, energy, charge, momentum and angular momentum transfer about in matter. All these phenomena share a very similar mathematical framework, and the parallels between them are exploited in the study of transport phenomena to draw deep mathematical connections that often provide very useful tools in the analysis of inter disciplinary fields in science. Computational Fluid Dynamics (CFD) is a methodology for obtaining a discrete solution of real world fluid flow problems using a high speed computers. CFD uses applied mathematics, physics and computationalsoftware to visualize how a gas or liquid flows and howthey affect objects as it flows past. It is based on basic laws of physics like Newton's law of motion, conservation of mass and first law of thermodynamics etc. usingNavier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related.Computational Fluid Dynamics provides simplified models with general 3D flow equations for both compressible and incompressible fluids with space time dynamics for both viscous and inviscid fluids having laminar and turbulent flow.Certain variables and property of fluids slow down the flow of fluids. Both viscous and inviscid flow has many applications in fluid dynamics.This paper discusses basics of computational fluid dynamics and some of their striking applications in real life situations.

Keywords

Transport phenomena; momentum; fluids; diffusion; viscosity; inviscid;computational fluid dynamics;

1.1. Introduction

Transport phenomena encompass all agents of physical change in the universe. They are considered to be the fundamental building blocks which developed the universe, and which is responsible for the success of all life on earth.In the known universe, systems and surroundings co-exist in dynamic equilibrium, from macro-scale to molecular-scale.The fundamental quantities of mass, momentum, energy and entropy are constantly being generated and consumed, and being exchanged between the system and its surroundings. Transport phenomena is the process in which mass, energy, charge, momentum and angular momentum transfer about in matter. It includes heat transfer, fluid dynamics and diffusion phenomena. Fluid mechanics is such an interdisciplinary topic that can penetrate to every domain of life starting from atmospheric application to flow of fluids in our bodies.

1.2. Fluids and their types

Fluids are a subset of the phases of matter. It includes liquids, gases and plasmasthat are capable of flowing. It has particles that easily move and change their relative position without a separation of the mass and so changes its shape at a steady rate when acted upon by a force tending to change its shape.Consider a hypothetical fluid having a zero viscosity ($\mu = 0$). Such a fluid is called an ideal fluid and the resulting motion is called as ideal or inviscid flow or super fluids. In an ideal flow, there is no existence of shear force because of vanishing viscosity.

$$\tau = \mu \frac{du}{dx} = 0 \text{ since } \mu = 0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots..(1)$$

All the fluids in reality have viscosity ($\mu > 0$) and hence they are termed as real fluid and their motion is known as viscous flow. Both viscous and inviscid flow has many applications in fluid dynamics.

1.3. Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics is a methodology for obtaining a discrete solution of real world fluid flow problems using a high speed computers. CFD uses applied mathematics, physics and computational software to visualize how a gas or liquid flows and how they affect objects as it flows past. It is based on basic laws of physics like Newton’s law of motion, conservation of mass and first law of thermodynamics etc. using Navier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. Computational Fluid Dynamics provides simplified models with general 3D flow equations for both compressible and incompressible fluids with space time dynamics for both viscous and inviscid fluids having laminar and turbulent flow. Measurement of the time a fluid takes to flow from one point to another (its distance) is called the fluid’s flow rate. Certain variables and property of fluids slow down the flow of fluids.

1.4.ConceptofComputationalFluidDynamics

Computational FluidDynamics is thesimulation of fluids engineering systems using modeling (mathematical physical problemformulation) and numericalmethods(discretization methods, solvers, numerical parameters,andgrid generations, etc.).The process is as shown in figure 1.

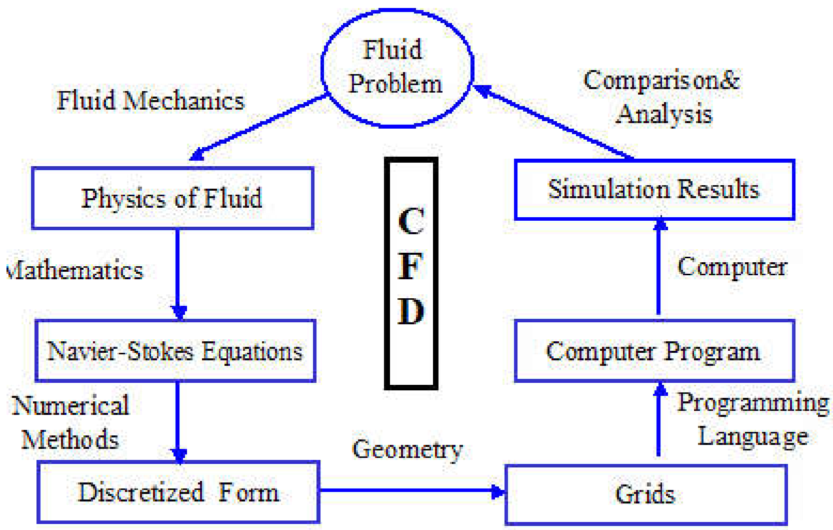


Figure 1 Processof Computational Fluid Dynamics

The knowledge of physical properties of fluidandtheir description usingmathematicalequationsis essential to solve a fluid problem.Navier-Stokes Equationsare thegoverning equationsof CFD. The translators are numericaldiscretization methods such as Finite Difference, Finite Element, FiniteVolumemethods. The process of CFD is to divide the whole problemdomainintomanysmallpartsbecausediscretizationisbasedonthem.Simple programmesare to be writtenusing Fortran and C tosolve them.Normally the programs arerun onworkstations or supercomputers.At the end, simulation results can be obtained and canbe comparedand analysed with experimentsand the real problem. Ifthe results arenot sufficient to solve the problem, then the process has to be repeateduntil satisfied solution is found.

1.5.ImportanceofComputationalFluidDynamics

Therearethreemethodsinstudy ofFluid: theory analysis,experimentand simulation(CFD).Asa new method, CFD has many advantagescomparedto experiments. It provides better predictions in a short time so that the innovators can design efficient new products based on simulations instead of “build & test” and so it is more cost effective, repeatable, and environmentally very safe and can get into market faster.

Simulation of physical fluid phenomena like water flow around ships and air flow around the wings of airplanes, explosion, radiation and pollution during hazards, weather predictions, stellar evolution etc. which are very difficultfor experiments can be modelled

1.6.PhysicsofFluid

Fluidisboth liquidandgas.Forexample,waterandair.Fluidhasmanyimportantpropertiessuchas velocity, pressure,temperature, density and viscosity etc.Thedensity(□)ofafluidisitsmassperunitvolume.Ifthedensityoffluidisconstant(orthe changeisverysmall),the fluidisincompressiblefluid.Ifthedensityoffluidisnotconstant, thefluidiscompressiblefluid.Normally,waterandaircan be treatedasincompressiblefluid. If the fluid is incompressible, the equations for this type of fluidcan be simplified as follows

$$\rho = \frac{M}{V}\left[\frac{kg}{m^3}\right] \text{---------------------} (2)$$

The viscosity (3) is an internal property of a fluid that offers resistance to flow. For example, to stirwater is much easier than to stir honey because the viscosity of water is much smaller than honey.

$$\mu = \left[\frac{Ns}{m^3}\right] = \left[Poise\right] \text{---------------------}(3)$$

2.Navier-StokesEquations

2.1.ConservationLaw

Navier-Stokesequations arethegoverning equationsof ComputationalFluid Dynamics. It is based ontheconservationlawofphysicalpropertiesoffluid.Theprincipleofconservationallawisthe changeofproperties,forexamplemass,energy,andmomentum with respect to time, andisdecidedbythe input andoutput.For example, the changeof mass in theobject is as follows:

$$\frac{dM}{dt} = \dot{m}_{in} - \dot{m}_{out} \text{---------------------}(4)$$

If $\dot{m}_{in} - \dot{m}_{out} = 0$, then $\frac{dM}{dt}=0$ which means M = a constant.---------------------(5)

2.2.Navier-StokesEquation

Applyingthemass,momentumandenergyconservation,wecanderivethecontinuity equation, momentumequation and energyequationas follows.

ContinuityEquation

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial U_i}{\partial x_i} = 0 \text{---------------------}(6)$$

Momentum Equation

$$\rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = \frac{\partial P}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_j \text{---------------------}(7)$$

Where

$$\tau_{ij} = -\mu\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3} + \delta_{ij}\mu \frac{\partial U_k}{\partial x_k} \text{---------------------}(8)$$

In equation (7), $\rho \frac{\partial U_j}{\partial t}$ is thelocal change of momentumwithtime, $\rho U_i \frac{\partial U_j}{\partial x_i}$ is themomentumconvection, $\frac{\partial P}{\partial x_j}$ is the Surface force, $\frac{\partial \tau_{ij}}{\partial x_i}$ is the molecular-dependent momentumexchange (diffusion) and ρg_j is themass force.Here ρ is the density,u is the flow velocity, $\frac{\partial}{\partial x_j}$ is the divergence, t is time, τ is the deviatoric stress tensor, which has order two, and g represents body accelerations acting on the continuum, for example gravity, inertial accelerations, electrostatic accelerations, and so on.

EnergyEquation

$$\rho c_\mu \frac{\partial T}{\partial t} + \rho c_\mu U_i \frac{\partial T}{\partial x_i} = -P \frac{\partial U_i}{\partial x_i} + \lambda \frac{\partial^2 T}{\partial x_i^2} - \tau_{ij} \frac{\partial U_j}{\partial x_i} \text{-----------(9)}$$

Here, $\rho c_\mu \frac{\partial T}{\partial t}$ is the local energy changewith time, $\rho c_\mu U_i \frac{\partial T}{\partial x_i}$ is the convectiveterm $P \frac{\partial U_i}{\partial x_i}$ is the pressure work, $\lambda \frac{\partial^2 T}{\partial x_i^2}$ is the Heat flux (diffusion) and $\tau_{ij} \frac{\partial U_j}{\partial x_i}$ is the irreversible transfer of mechanical energy into heat.

Ifthefluididcompressible,wecansimplifythecontinuityequationandmomentumequationas follows.

ContinuityEquation

$$\frac{\partial U_i}{\partial x_i} = 0 \text{-----------------------------(10)}$$

MomentumEquation

$$\rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} + \mu \frac{\partial^2 U_j}{\partial x_i^2} + \rho g_j \text{-----------(11)}$$

2.3.GeneralFormofNavier-StokesEquation

To simplify the Navier-Stokes equations, we can rewrite themas the generalform as

$$\frac{\partial(\rho\Phi)}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho U_i \Phi - \Gamma_\Phi \frac{\partial \Phi}{\partial x_i}\right) = q\Phi \text{----------------------(12)}$$

When Φ □ □ □, U_j,T,wecanrespectivelygetcontinuityequation,momentumequationandenergyequation.

3.FiniteVolumeMethod

TheNavier-Stokesequationsareanalyticalequations.Humancanunderstandandsolvethem,butif we wantto solve themby computer,we have to transfer theminto discretized form.This processis discretization.The typical discretization methods are finite difference, finiteelementand finite volume methods. Here we introduce finite volumemethod.

3.1.TheApproachofFiniteVolumeMethod

IntegratethegeneralformofNavier-StokesequationoveracontrolvolumeandapplyGauss Theory

$$\int_V \frac{\partial}{\partial x_i}\Phi dV = \int_S \Phi \cdot n_i dS$$ --------------------------(13)

We can get the integralform of Navier-Stoke equation

$$\int_V \frac{\partial(\rho\Phi)}{\partial t} dV + \int_S \left(\rho U_i \Phi - \Gamma \frac{\partial \Phi}{\partial x_i}\right) \cdot n_i dS = \int_V q_\Phi dV$$ -----------(14)

Toapproximatethevolumeintegral,wecanmultiplythevolumeandthevalueatthecenterof thecontrolvolume.Forexample,wehavea2Ddomainasfig2.Toapproximatethemassand momentumof control volume P, we have

$$m = \int_{V_i} \rho dV \approx \rho_P V, \quad mu = \int_{V_i} \rho_i u_i dV \approx \rho_P u_P V$$ -----------(15)

To approximate the surface integral, for example pressure force, we have

$$\int_{S_i} P dS \approx \sum_k P_k S_k \quad k = n, s, e, w$$ -----------------(16)

NormallywestoreourvariablesatthecenterofcontrolvolumesoweneedtointerpolatethemtogetP$_k$, which are located at thesurface ofcontrol volume.

Typically,wehavetwotypesofinterpolations,oneisupwindinterpolation,andtheotheroneis central interpolation.

U_P U_E

U_e

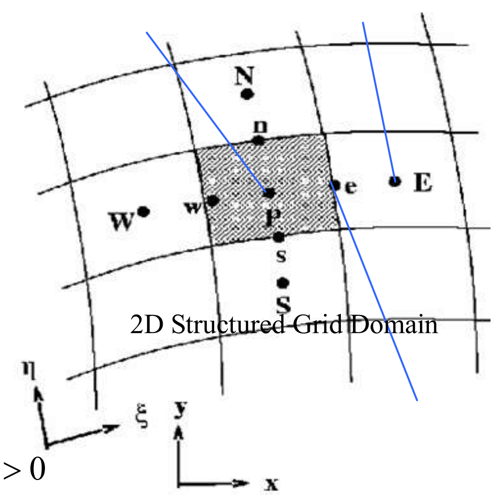


Figure 2 2D Structured Grid Domain

UpwindInterpolation

$$U_e = \begin{cases} U_P & \text{if } (\vec{U}\cdot\vec{n})_e > 0 \\ U_E & \text{if } (\vec{U}\cdot\vec{n})_e < 0 \end{cases}$$

CentralInterpolation

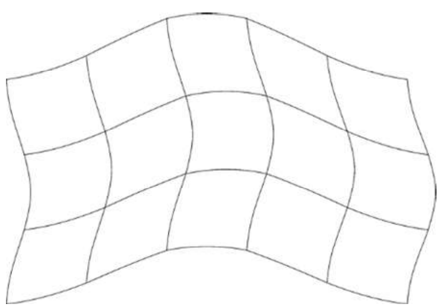
$$U_e = U_E \lambda_e + U_P \left(1 - \lambda_e\right) \qquad \lambda_e = \frac{x_e - x_P}{x_E - x_P}$$

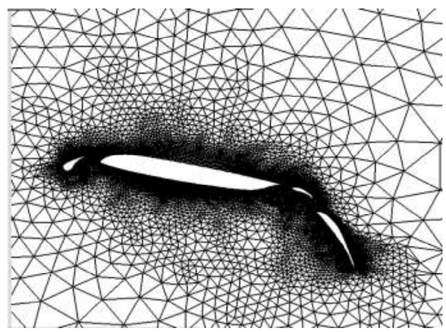
3.2. Conservation of Finite Volume Method

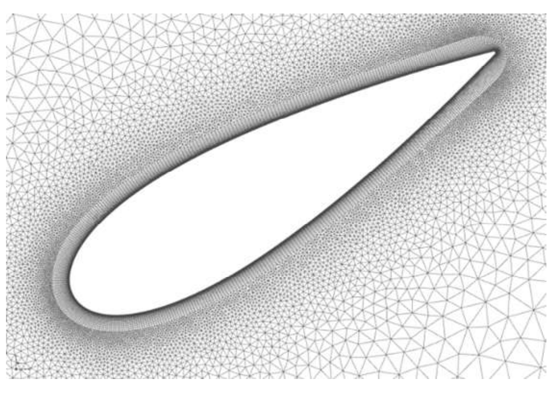
Conservation of mass, momentum and energy canbe manually controlled by the use of finite difference and finite element approach to discretized Navier-Stokes equation. One can easily find out that whether the Navier-Stokes equation is satisfied in every control volume or not using finite volume method. If so,then automatically it will be satisfied for the whole domain. In other words, if the conservation is satisfied in every control volume, it will be automatically satisfied in whole domain.This is the reason for preferring finite volume method in computational fluid dynamics.

4.Grids

There are three types ofgrids: structured grids, unstructured gridsandblock structured grids.Thesimplest one is structured grid(fig 3).Thistype of grids,allnodes have thesame number of elementsaround it.We candescribe and store themeasily. But this type of grid is only for the simple domain.







Unstructured grid is used for a complex domain.Fig 4 is the complex structure of airfoil.The flownearthe objectis very important and is very complex and need very fine grid at this region. Far away fromthe airfoil, the flow is comparably simple,and a coarse gridcan be used. Generally, unstructured grid issuitable forallgeometries andis very popular in CFD.The disadvantage isthat it ismore difficult todescribe and store them because thedata structure is irregular.Block structure grid is a compromising of structured and unstructured grid. So, it is wise to divide the domain into several blocks, and then use different structured grids in different blocks.

5.BoundaryConditions

Boundary conditionsare very essential in order to solve the system of equations.The typical boundary conditions in CFDare No-slip boundary condition,axisymmetricboundary condition, inlet, outlet boundary condition and periodic boundary condition.Forexample,fig5isapipe,thefluidflowsfromlefttoright.Theinletis atleftside,so that one cansetthevelocitymanually and the outlet is at theright side anduseboundaryconditionto keep all the properties constant, whichmeans all the gradients arezero.At the wall of pipe, set the velocity to be zero. This is no-slip boundary condition.At the center of pipe, use axisymmetric boundary condition.

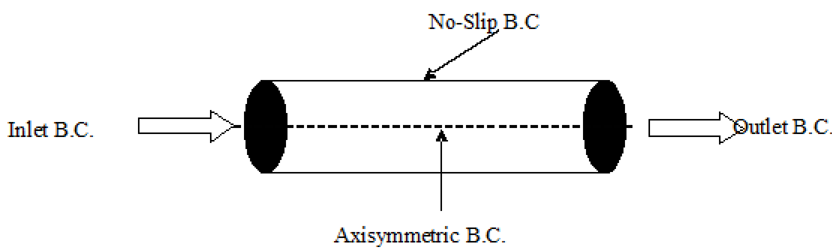


Figure 5 Boundary Conditions of Pipe Flow

6. Properties

The Navier–Stokes equations are nonlinear partial differential equations in the general case and so remain in almost every real situation. In some cases, such as one-dimensional flow and Stokes flow (or creeping flow), the equations can be simplified to linear equations. The nonlinearity makes most problems difficult or impossible to solve and the equations model is the main contributor to the turbulence. The nonlinearity is due to convective acceleration, which is associated with the change in velocity over position. Hence, any convective flow, whether turbulent or not, will involve nonlinearity. An example of convective but laminar (nonturbulent) flow would be the passage of a viscous fluid (for example, oil) through a small converging nozzle. Such flows, whether exactly solvable or not, can often be thoroughly studied and understood.

7.ApplicationofComputationalFluidDynamics

As CFDhas somany advantages, it is already generally used in industry such as flow and heat transfer in industrial processes (boilers, heat exchangers, combustion equipment, pumps, blowers, piping, etc.). Aerodynamics of ground vehicles, aircraft, missiles in aerospace, automotive, biomedicine, chemical processing,chemical vapor deposition (CVD) for integrated circuit manufacturing, heat transfer for electronics packaging applications, heat ventilation air condition like heating, and cooling flows in buildings,hydraulics, film coating, thermoforming in material processing applications,flow and heat transfer in propulsion and power generation systems, sportsand marine etc.

8. CONCLUSION

The Navier–Stokes equations together with supplemental equations (for example, conservation of mass) and well formulated boundary conditions seem to model fluid motion accurately and agree well with real world observations. The Navier–Stokes equations can be applied only for the fluid being studied is a continuum (it is infinitely divisible and not composed of particles such as atoms or molecules), and is not moving at relativistic velocities. At very small scales or under extreme conditions, real fluids made out of discrete molecules will produce results different from the continuous fluids modeled by the Navier–Stokes equations.For example, capillarity of internal layers in fluids appears for flow with high gradients. Another limitation is simply the complicated nature of the equations. Time-tested formulations exist for common fluid families, but the application of the Navier–Stokes equations to less common families tends to result in very complicated formulations and often to open research problems. For this reason, these equations are usually rewritten for Newtonian fluids where the viscosity model is linear.

References

[1] Smits, Alexander J. (2014), *A Physical Introduction to Fluid Mechanics*, Wiley, ISBN 0-47-1253499.

[2] E., Stewart, Warren; N., Lightfoot, Edwin (2007-01-01). Transport phenomena. Wiley. ISBN 9780470115398. OCLC 762715172.

[3] White, Frank M. (2006), Viscous Fluid Flow, *McGraw-Hill*, *ISBN 0-07-124493-X*

[4]Polyanin, A. D.; Kutepov, A. M.; Vyazmin, A. V.; Kazenin, D. A. (2002), Hydrodynamics, Mass and Heat Transfer in Chemical Engineering, Taylor & Francis, London, ISBN 0-415-27237-8

[5] Acheson, D. J. (1990), Elementary Fluid Dynamics, Oxford Applied Mathematics and Computing Science Series, Oxford University Press, ISBN 0-19-859679-0

[6] V. Girault and P.A. Raviart. Finite Element Methods for Navier–Stokes Equations: Theory And Algorithms. Springer Series in Computational Mathematics. Springer-Verlag, 1986.

[7] Roger Temam (1984): "Navier-Stokes Equations: Theory and Numerical Analysis", ACM Chelsea Publishing, ISBN 978-0-8218-2737-6.