

# Computational Fluid Dynamics, An Overview

Prathmesh Rajendra Jagadale<sup>1</sup>, Alpesh Bhimji Chawdhary<sup>2</sup>

<sup>1</sup>Department of Mechanical Engineering, Amity School of Engineering and Technology, Amity University Mumbai, Maharashtra, India.

<sup>2</sup>Department of Mechanical Engineering, Saraswati College of Engineering, Kharghar, University of Mumbai, Maharashtra, India.

\*\*\*

**Abstract** - The use of CFD is increasing day by day since it can be used to analyze the dynamics of elements. The large simulations which were only possible in a controlled environment, are now possible on the computer. The experimentation of chemical reactions, heat transfer, explosions is not feasible to perform again and again as it involves considerable time and cost. Also, the simulation of large bodies like the A350 plane for which building a wind tunnel would not make sense. Hence, CFD has become more popular. This article provides an overview of computation fluid dynamics, explaining the differential equations that govern fluid flow. It explains why these equations cannot be solved directly and how they can be converted into simpler forms that can be solved by computers. This article also elaborates the process of CFD simulation thus pre-processing, solving and post-processing. Different types of meshing and their examples. Additionally, it explains all essential terms that one must understand before performing a CFD simulation.

**Key Words:** Meshing, Discretization, Navier's stokes equations, post-processing.

## 1. INTRODUCTION

Fluid dynamics is a study of fluid flow and behavior. A fluid can be liquid or gas. Fluid dynamics can be seen everywhere, including morning coffee, the commute to work, rain, rivers flowing, ships moving, and even rockets. In order to better understand the principle behind this, scientists and engineers studied the properties of fluids. Researchers have been studying fluid dynamics for decades. Studies in the past were mostly analytical or experimental. But it was difficult to study the large-scale problems and to resolve this issue CFD is used.

It is crucial to understand the science behind CFD before performing it on a computer.

CFD consists of five steps:

1. Geometry creation
2. Creating a mesh
3. Discretize the governing equation
4. Solution of the equation
5. Post-processing/Visualization of the result.

## 2. PRINCIPLE

Like any branch of physics, fluid mechanics is governed by some fundamental principles. These are the conservation of

mass, conservation of momentum and conservation of energy.

### 2. 1. The Conservation of Mass

Mass can neither be created nor destroyed. Imagine water flowing through a pipe to understand the equation better.

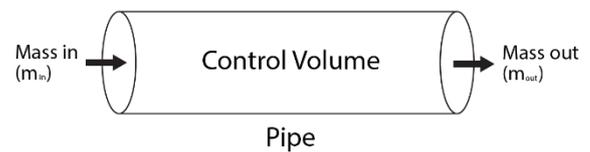


Figure1. control volume in the pipe

$$m_{in} = m_{out} \tag{1}$$

According to this principle, whatever fluid mass goes in has to come out. Mathematically that can be expressed in an integral equation,

$$\frac{d}{dt} \iiint_V \rho dv = 0 \tag{2}$$

Shrinking the domain to an Infinity similarly small control volume, the equation changes to a differential form.

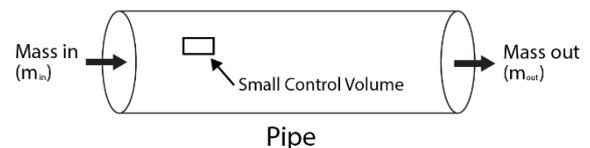


Figure 2. Small control volume in the pipe

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0 \tag{3}$$

For incompressible flow Density ( $\rho$ ) is constant:

$$\vec{\nabla} \cdot \vec{v} = 0 \tag{4}$$

### 2. 2. The conservation of momentum

It is an expression of Newton's second law of motion, which states that the rate of change of momentum of a fluid particle is equal to the sum of force acting on it.

$$\vec{F} = \frac{d\vec{p}}{dt} ; \quad \text{if } \frac{dm}{dt} = 0$$

$$\vec{F} = m \cdot \vec{a} \tag{5}$$

The momentum equation can be expressed in an integral form,

$$\frac{\partial}{\partial t} \int_V \rho \vec{v} dv + \oint_S (\rho \vec{v} \cdot d\vec{s})v = - \oint_S p ds + \oint_V \rho f dv + F_{viscous} \tag{6}$$

The momentum equation also can be expressed in a differential form,

$$\rho \frac{D\vec{v}}{Dt} = -\nabla \cdot p + \rho \cdot f + F_{viscous} \tag{7}$$

$$\rho \frac{D\vec{v}}{Dt} = \text{Similar to } m \cdot \vec{a}$$

The differential form is more intuitive to understand as a mass represented by a density, which is balanced by the force or (the right-hand side : The pressure force, the body force and the viscous forces.) The differential form of the momentum equation is an exact representation of Newton's second law of momentum.

The differential form is more intuitive to understand as a mass represented by a density, which is balanced by the force or (the right-hand side : The pressure force, the body force and the viscous forces.) The differential form of the momentum equation is an exact representation of Newton's second law of momentum.

A third principle which is the conservation of energy. The energy conservation principle is similar to the momentum equation but also has a separate variable of temperature. This is only used when the density and the temperature of fluids vary.

### 2. 3. Conservation of energy

The rate of change of energy of a fluid particles equals the rate of heat added and the net rate of work done by the fluid particle.(first law of thermodynamics)

The mass and momentum equations together called Navier stokes equations that describe any type of fluid flow.

Mass Conservation:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \cdot \vec{v}) = 0 \tag{8}$$

Momentum conservation:

$$\rho \frac{D\vec{v}}{Dt} = -\nabla \cdot P + \rho \cdot \vec{f} + \vec{F}_{viscous} \tag{9}$$

Both eq. (9) and (10) are partial differential equations.

The momentum conservation equation, the term on the left-hand side can be rewritten as,

$$\frac{\partial v}{\partial t} + v \cdot \nabla v \tag{10}$$

There are also some non-linearities in this equation. The mass and momentum equations are highly coupled. To solve any type of fluid flow both equations are required and cannot be solved one without the other. It is nearly impossible to obtain an analytical solution for the Navier-Stokes equations since they are Partial Differential Equations, which are nonlinear and highly coupled. Therefore, an analytical solution to the full set of Navier-Stokes equations in three dimensions is not possible and computational fluid dynamics has to be employed.

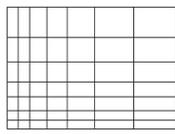
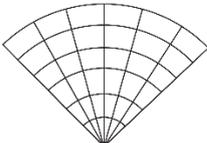
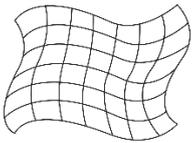
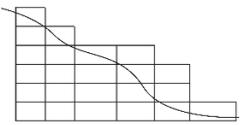
### 3. Geometry Creation:

To solve a partial differential equation, boundary conditions are required. The boundary could be at the ends of the domain that has to be solved at or at the surface of the body that has to be computed. Navier-Stokes equations are not ordinary equations. There haven't been any proofs yet that there is a smooth solution to the 3-dimensional problem with initial conditions. To overcome this problem, computational fluid dynamics is being used.

### 4. Meshing:

Meshing is important step of CFD. An improper meshing will lead to incorrect results. Meshing is primarily divided into two types, one structured mesh and unstructured mesh. Mesh can be structured as per the requirements and unstructured mesh is used in complex geometries. Conformal mesh is used when one part is stator, and one part is rotor mating at same points. When flux transfer takes place over non overlapping zones the Non conformal meshing is used. Adaptive mesh is used when the finer mesh is required at particular location but not the entire domain.

Different types of mesh are shown in the table below:

	
Cartesian mesh (structured)	Polar mesh (Structured)
	
Curvilinear mesh (structured)	Immersed Boundary Methods (structured)

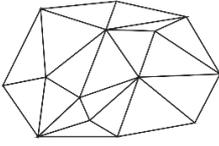
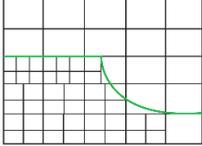
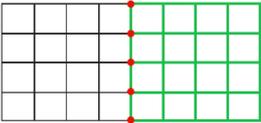
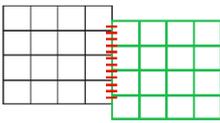
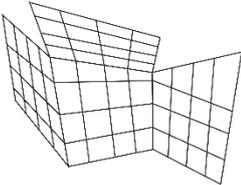
	
Unstructured mesh	Adaptive mesh (Structured)
	
Conformal mesh (structured)	Non conformal mesh (Structured)
	
Block structured mesh (locally structured, globally unstructured)	

Table 1. Types of meshing

**5. Discretization**

The process of breaking down a large volume of fluid into smaller volumes or elements.

Take a simple analogy to better understand discretization. Consider a curve with two points. A straight line will result from those two points, and it does not resemble the curve. Consider taking three points, and although slightly better, it still isn't exactly the curve; four points, five points, it keeps improving as shown in table below.

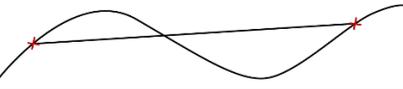
Curve	
2 Points	
3 points	
4 points	
5 points	
Large number of points	

Table 2. Discretization process

It is possible to reproduce an exact curve by increasing the points to infinity. Unless the problem is discretized optimally, one cannot expect good results.

**6. Solution**

Consider a rod where one end is at 0 kelvin as shown in figure 3. In order to determine the temperature distribution along the rod, one needs to know the differential equations that govern physics.

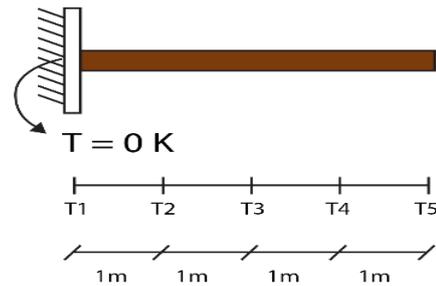


Figure 3. Rod with 0k temperature at one end

Here, let's consider  $Dt/Dx=1$  as the differential equation controlling temperature distribution along the rod with  $T1=0$  as a boundary condition. To solve this problem, which is a continuous one, discretize the rod into 4 different elements.

$Dt/dx$  is the change in length and  $dt$  is  $T2 - T1$  and  $dx$  is the gap between the two ends of the domain which is 1m. ( $T$  = Temperature,  $x$  = Spatial coordinate)

$$\frac{dT}{dx} = \frac{T1 - T2}{1} = 1 \quad T2 = 1$$

$$\frac{dT}{dx} = \frac{T3 - T2}{1} = 1 \quad T3 - T2 = 1$$

$$\frac{dT}{dx} = \frac{T4 - T3}{1} = 1 \quad T4 - T3 = 1$$

$$\frac{dT}{dx} = \frac{T5 - T4}{1} = 1 \quad T5 - T4 = 1$$

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ 0 & -1 & 1 & 0 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} T2 \\ T3 \\ T4 \\ T5 \end{bmatrix} = \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1 \end{bmatrix}$$

By solving these equations for 4 elements, a set of linear algebraic equations is obtained. Converted a differential equation into linear algebraic equations. CFD is very similar to this, but these differential equations are converted to algebraic equations using three main methods which are the finite difference method, the finite volume method and the finite element method. All these methods are a way to approximate differential equations.

Once the domain has been discretized and the linear algebraic equations are formed, the next step is solving. The solution is done in steps or iterations,[12] and if the meshing is done properly and the computer is able to solve them without any problem, a converged solution is obtained. Convergence means the results have come as close as possible to the real solution of the differential equations.

### 7. Post Processing

Now, the final stage of CFD. Even if a solution was obtained after processing, it might not be the best solution for the problem. Hence always verify the solution's validity and accuracy. First is a visual inspection. Does the solution seem physical or non-physical?

The process of post-processing computational fluid dynamics (CFD) data is crucial to ensure that the right conclusions can be drawn from the models, and that product designs are as effective as possible. Since CFD simulations often produce large output files, it is difficult to create effective visualizations.

The second check is to ensure that mass is conserved globally because all the problems are solved are approximations. Make sure that physically it makes sense, whatever mass is entering the domain has to leave.

Post-processing can also provide information on the instantaneous values of all variables at certain points in a domain.[12]

### 8. Discussion

The Navier's Stokes equations are very robust they can solve any type of fluid flow ranging from slow viscous creeping flow, non-Newtonian flows and also turbulent flows. Turbulent flows are everywhere in nature and engineering. Turbulence involves a lot of scales from tiny scales of motion to scales of motion that are of the order of the domain. The range of scales is dependent on a non-dimensional number called the Reynolds number.

To solve all these scales the domain has to be discretized very finely. This type of solving is called a direct numerical simulation where the Navier's Stokes equations are solved completely. The expected computational time of direct numerical simulation is the order of Reynolds number to the power of 3.

In real-life applications like an aircraft which operates at the order of Reynolds number to the power of 5, it is almost impossible to perform a Direct Numerical Simulation study on it. To overcome this, techniques such as Reynolds Averaged Navier Stokes (RANS), and Large Eddy Simulations (LES) has been implemented.

RANS equations are time averaged equations of fluid flow. In this method all instantaneous quantities like velocity are decomposed into their time-averaged and fluctuating quantities.

consider, the momentum equations that are already described.

$$\rho \frac{D\vec{v}}{Dt} = -\nabla \cdot \vec{p} + \rho \cdot \vec{f} + \vec{F}_{viscous} \quad (13)$$

By applying time averaging and decompose these instantaneous quantities into its average and fluctuating component, the Reynolds averaged momentum equation is obtained.

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[ -\bar{p} \delta_{ij} + \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{\rho u_i u_j} \right] \quad (14)$$

the term  $(\overline{\rho u_i u_j})$  from above equation that is nonlinear and also known as Reynold's stress term.

In order to come up with models for this Reynolds stress term, several techniques have been developed, including the zero-equation model, k-omega, SST (shear stress transport) model, and the K-epsilon model. Large eddy simulations (LES) is another technique like RANS used to model fluid flow. In this technique the effect of the smaller scales is filtered out by applying a low pass filter. Applying a low pass filter to Navier-Stokes equations, equations, gives rise to the equation,

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + 2\nu \frac{\partial}{\partial x_j} S_{ij} \quad (15)$$

Which is then solved to obtain results. This is done because solving these smaller scales is computationally very intensive. The behaviour of these smaller scales is almost universal that means they are independent of the external flow as a result of which these smaller scales are modelled and the larger scales which are highly dependent on the external flow are resolved.

That is a brief description of CFD. As mentioned, it has steps like discretization, solving and post process in addition to which CFD is performed like DNS, RANS and LES.

### 9. Conclusion

Computational fluid dynamics (CFD) has become a popular solution to problems involving fluid flow, heat transfer, mass transfer, chemical reactions, or a combination of these. In the past most of these studies were either Analytical or experimental. Experimental work can be interesting and fun until it involves something dangerous, like studying the exhaust of a rocket, or something even impossible to fit a full-size aeroplane A380 in a wind tunnel. That is why Fluid dynamics is not easy to analyse; however, modern computers make it easier and faster. Solving fluid problems with computers become more and more popular and affordable.

## REFERENCES

- [1] Md Lokman Hosain, Rebei Bel Fdhila,, Literature Review of Accelerated CFD Simulation Methods towards Online Application, Energy Procedia, Volume 75, 2015, Pages 3307-3314, ISSN 1876-6102, <https://doi.org/10.1016/j.egypro.2015.07.714>.
- [2] H. Ayremlouzadeh, J. Ghafouri, Computational Fluid Dynamics Simulation and Experimental Validation of Hydraulic Performance of a Vertical Suspended API Pump, International Journal of Engineering (IJE), TRANSACTIONS B: Applications Vol. 29, No. 11, (November 2016) 1612-1619  
DOI: 10.5829/idosi.ije.2016.29.11b.16
- [3] K. Azizi, M. Keshavarz Moraveji, Computational Fluid Dynamic- Two Fluid Model Study of Gas-Solid Heat Transfer in a Riser with Various Inclination Angles, International Journal of Engineering (IJE), TRANSACTIONS A: Basics Vol. 30, No. 4, (April 2017) 464-472  
DOI: 10.5829/idosi.ije.2017.30.04a.02
- [4] Ashory, M., Talebi, F., Roohi Ghadikolaei, H. A Computational Study about the Effect of Turbines Pitched Blade Attack Angle on the Power Consumption of a Stirred Tank. International Journal of Engineering, 2018; 31(1): 65-70.
- [5] W. Lin, H. Wang, Y. Zhang, Comparison of Two Computational Microstructure Models for Predicting Effective Transverse Elastic Properties of Unidirectional Fiber Reinforced Composites, International Journal of Engineering (IJE), IJE TRANSACTIONS C: Aspects Vol. 31, No. 9, (September 2018) 1498-1504  
DOI: 10.5829/ije.2018.31.09c.04
- [6] Chun-Jiang Zhao, Jia-Wei Han, Xin-Ting Yang, Jian-Ping Qian, Bei-Lei Fan, A review of computational fluid dynamics for forced-air cooling process, Applied Energy, Volume 168, 2016, Pages 314-331, ISSN 0306-2619, <https://doi.org/10.1016/j.apenergy.2016.01.101>
- [7] Tomás Norton, Da-Wen Sun, Computational fluid dynamics (CFD) – an effective and efficient design and analysis tool for the food industry: A review, Trends in Food Science & Technology, Volume 17, Issue 11, 2006, Pages 600-620, ISSN 0924-2244, <https://doi.org/10.1016/j.tifs.2006.05.004>.
- [8] N.J. Smale, J. Moureh, G. Cortella, A review of numerical models of airflow in refrigerated food applications, International Journal of Refrigeration, Volume 29, Issue 6, 2006, Pages 911-930, ISSN 0140-7007, <https://doi.org/10.1016/j.ijrefrig.2006.03.019>.
- [9] Anderson, John D., Jr., Fundamentals of Aerodynamics, 2nd Edition McGraw-Hill, New York, 1991.
- [10] Mokhtarifar, N., Saffaraval, F., Saffar-Avval, M., Mansoori, Z. and Siamie, A., "Experimental modeling of gas–solid heat transfer in a pipe with various inclination angles", Heat Transfer Engineering, Vol. 36, No. 1, (2015), 113-122.
- [11] Fenjan, S.A., Bonakdari, H., Gholami, A. and Akhtari, A., "Flow variables prediction using experimental, computational fluid dynamic and artificial neural network models in a sharp bend", International Journal of Engineering-Transactions A: Basics, Vol. 29, No. 1, (2016), 14.
- [12] Ayremlouzadeh, H. and Ghafouri, J., "Computational fluid dynamics simulation and experimental validation of hydraulic performance of a vertical suspended api pump", Intenational Journal of Engineering, Vol. 29, No. 11, (2016), 1612-1619.

## NOMENCLATURE

$min$	Mass entering
$mout$	Mass Coming out
$\rho$	Density (Kg/m <sup>2</sup> )
$dv$	Volume (m <sup>3</sup> )
$\rho dv$	Mass in the control Volume
$F_{viscous}$	Force due to viscosity of the fluid
$\nabla(\rho v)$	net mass flow through the small control volume
$p$	Pressure
$f$	Body forces
$T$	Temperature (K)
$X$	Spatial coordinate
$\nabla$	Divergence
$V$	Velocity
$t$	Time (s)

## BIOGRAPHIES



### Prathmesh Rajendra Jagadale

Department of Mechanical Engineering, Amity School of Engineering and Technology, Amity University Mumbai, Maharashtra, India.



### Alpesh Bhimji Chawdhary

Department of Mechanical Engineering, Saraswati College of Engineering, Kharghar, University of Mumbai, Maharashtra, India.