 ISBN:978-81-941253-2-7

APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS: A REVIEW

Manas Ranjan Padhi
Department of Mechanical Engineering
Centurion University of Technology and Management, Bhubaneswar, India

ABSTRACT
Computational Fluid Dynamics (CFD) is a simulation tool which is used to solve fluid flow and heat transfer problems.Mathematical models in terms of conservation laws represent the flow physics in terms of strongly coupled non-liner partial differential equations. This technology is applied in areas like cavitation prevention, aerospace engineering, HVAC engineering, electronics, manufacturing, automotive and way more. With the advent of powerful computers, CFD is used solve in many real life problems. The present work focuses on the practical applications of CFD technique in varies fields.

Keywords: CFD, Heat Transfer, Fluid Flow, Simulation, Application

INTRODUCTION

Computational Fluid Dynamics (CFD) evolved from a mere academic curiosity of the physicists and mathematics to an emerging technology for engineers. It uses powerful algorithm organising mathematics to simulate flow physics. CFD today is a very useful computer aided engineering design tool. It is used as an alternative in order to reduce number of experiments as well as minimizing cost and time for process designing.The uses of CFD are common in fields like automobile, aerospace, power plants, manufacturing, process plants and electronics. The recent applications of CFD In human body and biomedical fields have excited researchers.

There are three steps involved in CFD: pre-processing, processing and post processing. The pre-processing step involved creation of geometry, grid generation and selection of boundary conditions. In processing step, the mathematical equations are solved by using computers. There are three main numerical methods to solve the mathematical equations such as finite difference method, finite element method and finite volume method. Finite difference method is the ancient method and confined to simple cases of heat transfer and fluid flow problems. Both finite element and finite volume methods are more versatile than finite difference method. Finite volume method is the latest method and commercial CFD packages are based on this method. In post-processing step, the analysis of the results is carried out with versatile data visualisation tool.

Laohasurayodhin et al. [1] studied the effect of angle of inclination of fluid flow in a pipe by considering various viscous models through numerical simulation using CFD. Patel et al. [2] carried out their numerical investigation on the effect of pressure developed on various types of joints and fittings used in pipe through CFD technique. Kumar [3] used commercial CFD software called ANSYS for analysing fluid flow at various points in a pipe with an aim for determination of losses in head due to change in geometry of pipe. Nimadge and Chopade (2017) [4] numerically simulated the steady, incompressible fluid flow through a T-junction by CFD technique. They prepared experimental setup to obtain the reference data when fluid passes through T-junction of pipe. Motlagh et al. [5] presented an application of residual-based variational multiscale modeling methodology for computation of laminar and turbulent fluid flow through concentric annular pipe flows.

PRACTICAL APPLICATION OF COMPUTATIONAL FLUID DYNAMICS

HEAT EXCHANGER

Heat exchangers are used extensively in power plant, chemical industries and process industries.CFD has been used to simulate the fluid flow and temperature distribution of fluids in heat exchangers. It can also be employed for fluid flow maldistribution, fouling, pressure drop and thermal analysis in the design and optimization phase in various types of heat exchangers. With the help of a fluid flow simulation tool like CFD, the redesigning of heat exchangers can be carried out. CFD has also been used to study the effect of obstacles and holes inside heat exchangers to control temperature and resistance to flow. Khudheyer and Mahmoud [6] used CFD to investigate heat transfer and fluid flow simulations in a double role finned tube heat exchanger by using OPENFOAM, an open-source CFD code. They studied the heat transfer and pressure drop characteristics for Reynolds number ranging from 330 to 7000.While dealing with heat