International Journal of Current Engineering and Technology

E-ISSN 2277 – 4106, P-ISSN 2347 – 5161
Available at http://inpressco.com/category/ijcet

National Conference on Recent Advances in Civil Engineering Infrastructure(rACEi-2021)

Application of Computational Fluid Dynamics in Civil Engineering-A Review

Asma Sultana[1], Qamar Sultana[2]

1. Asst. Prof., Dept. of Civil Engg, Muffakham Jah College of Engineering & Technology
2. Assoc. Prof., Dept. of Civil Engg, Muffakham Jah College of Engineering & Technology

Received 05 Aug 2021, Accepted 10 Aug 2021, Available online 15 Aug 2021, **Special Issue-9 (Aug 2021)**

Abstract

Computational fluid dynamics (CFD) was used widely by every field of engineering for solving critical problems that entail the phenomena such as flow of fluids, heat transfer, mass transfer and chemical reaction. In many industrial and engineering processes this tool is used for investigating and predicting flow behavior.CFD techniques have been widely applied to the field of water Conservancy Engineering, Environmental Engineering, Structural Engineering, and Industrial Engineering with the continuous development in the field of computers. This paper summarizes the theoretical basisand computational process of CFD,Numerical calculation methods. Finally, CFD technology in the Civil engineering related applications are summarized. It is hoped that this review will help researchers in the field of Civil engineering.

Keywords: *Computational Fluid Dynamics, Civil Engineering, Fluid Flow Problems.*

1. Introduction

Computational fluid dynamics is a branch of fluid mechanics. CFD utilize the numerical methods and algorithms to solve problems involved fluid flows. CFD has combined fluid mechanics which can simulate compressible and incompressible fluid flow behavior. With continuous improvement in computational speed of computers, CFD technique is increasingly used for simulation. Simulation needs very high-speed computers. Depending on complications in mode, solver takes time. With development of new algorithm, computation time is reducing. Product development cost can be reduced with the help of CFD simulation. Using simulation, various alternatives can be explored to achieve the optimum result.

Navier-Strokes equation was the basis for all CFD softwares. Single phase fluid flow was defined by Navier-Stokes equation. Navier-Stokes equation can be simplified in order to yield the Euler's equations with removal of term that describe viscous actions. CFD results calculate and display fluid properties like distribution of velocity, temperature, contaminant concentration, pressure and other fluid properties. CFD output results help design engineers to improve in model and communicate result in graphical form effectively.

The continuing development of cheaper and more powerful computing has driven computational fluid dynamics (CFD) to the point where it is now routinely used to predict internal conditions and increasingly to examine external flows. The method is extremely attractive to design engineers as it allows great flexibility in testing alternative designs and can in many cases be performed more quickly by their in-house specialists than physical testing, which requires an external consultant.

2. Methodology of CFD

CFD is a Computational Fluid dynamic which enables us to understand the dynamics of things that flow.CFD uses the basic equations of flow namely Mass conservation equation, momentum conservation equation, energy conservation equation used in numerical simulation of the flow. CFD uses some computational techniques to find the complex problems of fluid mechanics and discrete numerical solution. Numerical solution is a discrete approximation of the calculation method, which generally includes pre-processing stage, obtaining the flow solution stage and post-processing stage.

In the first stage that is pre-processing stage the input of a flow problem to a CFD package is taken up. This stage mainly consists of the definition of the geometry of the problem of interest, mesh generation, specification of physical properties of the fluid and appropriate boundary conditions. An unknown flow variable such as velocity, pressure and temperature is solved at nodes inside each cell. In the second stage the simulation is started and the equations are solved iteratively as a steady-state or transient and solutions of the governing equations for the unknown flow variable are obtained. Finally, a postprocessor is used for the analysis and visualization of the resulting solution.

3. Numerical calculation method

For the last few decades, varieties of numerical solutions had emerged. The main difference lies in the regional discrete approach, the equation of discrete and algebraic equations for solution. The numerical methods used in CFD numerical solution are finite difference method, finite volume method and finite element method. The most widely solver method for CFD is the finite volume method.

Finite Difference Method:

Finite difference method (FDM) is the earliest method of computer numerical simulation. The method divides the solution region into a separation grid and uses a finite number of grid nodes instead of a continuous solution area. The finite difference method is used to discretize the derivative of the control equation with the difference of the function value on the grid node to establish the algebraic equations with unknown values on the grid nodes. The method is an approximate numerical solution that directly turns the differential problem into algebraic problem.

Finite volume method:

Finite Volume Method (FVM), also known as the control volume method, is a commonly used method for spatial discretization. It is mainly from the equations of conservation type, and make integral in its control volume, and further solve the conservation equation in integral form. These all-control differential equations have a common form; this form of consistency is the basis of a common solution.

Finite element method:

The finite element method (FEM) is used in structural analysis of solids and is also applicable to fluids. It is based on the variation principle and the weighted margin method. The basic solution idea is to divide the calculation area into a finite number of non-overlapping units. In each unit, select some of the appropriate nodes as the interpolation point for the function. Then, the variables in the differential equation are rewritten as a linear expression (shape function) consisting of the node value of each variable or its derivative and the selected interpolation function. Finally, the differential equation is discretized and solved by means of the variation principle or the weighted margin method. Different finite element methods are used to form different weight functions and interpolation functions.

4.Softwares for CFD

Computational fluid dynamics (CFD) was used by civil engineers to model practical problems on field. CFD software available in market are of two types. Some software are commercial license software and others are Open-source software. Category of software has different functionality available within software. Following are a few softwares used by the industry.

Commercial CFD Software:

1. ANSYS - Fluent - Software used for faster optimization. With its physical modeling capability, software models turbulence flow, heat transfer and reactions for industrial applications. Any type of fluid can be modeled in software. Software can be used by all branches of engineering for simulation.

2. IES VE – Microflow -Using CFD software undertake internal and external air flow and heat transfer in and around buildings, considering boundary conditions such as internal energy sources, the effects of climate and HVAC systems, calculates air temperature and airflow, occupant comfort temperatures. Detail natural and mixed mode ventilation strategies are available. Simulations of air circulation in building are done.

3. Autodesk CFD2016 – Autodesk CFD software was used for thermal simulation tools and computational fluid dynamics. Software uses the CFD Design Study Environment with a solver to predict product optimize designs performance. Software is useful for validating product behavior before manufacturing.

4. Star – CCM+ - Software works in multidisciplinary engineering simulation like “aerodynamics”, “hydrodynamics”, “heat transfer” and “solid mechanics”. In multidisciplinary engineering for real-world performance of product, simulation can accurately capture all the relevant physics. Software can be used to automatically drive the virtual product through operating scenarios and a range of design configurations.

5. COMSOL – Numerical simulation platform for computational fluid dynamics (CFD) that accurately describes your engineering designs and fluid flow processes. Using the CFD Module, you can model most aspects of fluid flow, including non-isothermal, compressible, non-Newtonian, porous media flows and multiphase, all in the laminar and turbulent flow regimes.

Open source CFD Software:

1. Open FOAM - Open FOAM is industries free open-source software for computational fluid dynamics.

It is owned by Open FOAM Foundation and developed by peoples working in CFD research. Under GNU General public license, software code is released as free and open-source software.

2. FLUIDITY - An opens source, general purpose, multiphase computational fluid dynamics code capable of numerically solving the Navier- Stokes's equation and accompanying field equations. Finite element/control volume method is used which allows arbitrary movement of the mesh with time dependent problems. Finite element/control volume method allow mesh resolution to increase or decrease locally according to the current simulated state. Wide range of element choices is available in software.

3. SU2 - The SU2 software suite is an open-source collection of C++ based software tools for performing Partial Differential Equation analysis and solution of PDE-constrained optimization problems. Software is designed for aerodynamic shape optimization in mind but is extensible to treat arbitrary sets of governing equations such as elasticity, potential flow, electrodynamics, chemically reacting flows, and others.

5. Application of CFD in Civil Engineering

Considered one of the oldest branches of engineering, civil engineering is a broad discipline that incorporates many different and important aspects of engineering, including structural, fluid and soil mechanics. These form the key inputs in the planning and design of manmade structures as diverse as water supply systems, buildings, power plants, bridges and tunnels. Civil engineers use simulation software to guide design and construction as well as to solve a wide range of projects in: Building, Environmental engineering, Geotechnical engineering, Hydraulic engineering, Materials science, Structural engineering, Transportation engineering, and Wind engineering.

1. Building Construction: In building constructionfield, CFD can be used for numerical analysis in the innovative design of intelligent buildings, scoring an HVAC goal for hockey spectators and design ventilation systems for sports arenas. By using comprehensive multi-physics solutions from ANSYS, truly innovative buildings can be designed faster. New alternative materials can be investigated and their impact on the structure of the construction and comfort of building occupants together with their safety can be analyzed before construction is begun. The simulation-driven building design, cost effective and environmentally friendly solutions can be combined with original and innovative architecture.

2. Geotechnical Engineering: In Geotechnical Engineering,CFD can be used for flow through granular dam filters and modeling of particle migration and finite elements analysis on weak foundation of oil storage tank.

3. Structural Engineering: In Structural Engineering, CFDcan be used for structural analysis, helps predict vibration for pumping platforms, supporting the oil and gas industry, longevity and safety of drilling derricks and substructures are increased through stress analysis, two-way interface between thermal solid elements and structural beam and shell elements, non-linear analysis of shear dominant pre-stressed concrete beams.

4. Transportation Engineering: In TransportationEngineering, CFD can be used for bridging the gaploadingcapacity, simulation for temporary bridges, assists inproviding effective disaster relief, ventilatinggiant railway tunnels, improving underground safetyand ventilation for metro train.

5.Wind Engineering: In Wind Engineering CFDcan be used for flow modeling. For damage caused by wind, harnessing natural energy, multiple simulation tools are used as a cost-effective way to design reliable offshore wind turbines.

6.Hydraulic Engineering: In Hydraulic EngineeringCFD can be used for unearthing municipal water system, numerical modeling of non-stationary free surface flow in embankment dams, hydro-electric power plant intakes, flow through granular dam filters and modeling of particle migration, research on choice of dam foundation elevation in hydropower engineering. During the probable maximum flood, spillway performance was studied.

7.Environmental Engineering:In Environmental EngineeringCFD can be used for harnessing natural energy. Multiple simulation tools are used as a cost-effective way to design reliable offshore wind turbines, blending solar panels with roof profiles - simulation guides the design of innovative solar panel frames, reducing molding time, material, and cost.CFD can be applied for prediction of clogging locations in building drainage system.

8. Irrigation Engineering: Irrigation systems are designed to transport water from a source through a network of pipes to irrigate crops. Flow velocities, friction losses, and minor losses are some of the flow parameters that must be accurately predicted to achieve proper operational conditions and acceptable uniformity of water application over the irrigated area. Computational fluid dynamic techniques can be useful for simulating flow parameters such as friction losses and local head losses.

Conclusions

Computational fluid dynamics (CFD) can be used by civil engineers for design of various civil engineering structures. In civil engineering, CFD can be used for study of liquid flow in environmental structures, sloshing effect of liquid in tanks, natural ventilation, force ventilation, thermal comfort study, stack effect, planning of building in towns and contaminant migration in hospitals. With proper planning of windows location, windows size and ventilators, natural flow of air can be maintained in residential and hospital buildings. Researchers have used CFD software for modeling in environmental structures like wastewater gas–liquid crossflow reactor, sedimentation tank flow design, other hydraulic structures, spillway design, turbine, pumps design, thermal comfort study, wind circulation study inside and outside the building, effect of development on air circulation in city and many more. With availability of advance high-speed computers and validated CFD software's, experimental study can be shifted to simulation study. It is required for civil engineers to utilize the power of CFD to solve and invents new technology in construction industry.

References

JD Anderson. Computational Fluid Dynamics: The Basics with Applications [M]. Beijing: Tsinghua University Press, 2002.

Yao R T, Guo D P. Computational Fluid Mechanics Foundation and STAR-CD Engineering Application [M]. Beijing: National Defense Industry Press, 2015: 143-144.

Gessler D. (2005). CFD Modeling of Spillway Performance. *Impacts of Global Climate Change*, 1-10. doi:10.1061/40792(173)398.

SWADDIWUDHIPONG S. and KHAN M.S. (2002) Dynamic response of wind-excited building using CFD. *JSoundVib*., 253(4), 735-754. doi:10.1006/jsvi.2000.3508.

Goula A.M., Kostoglou M., Karapantsios T.D. and Zouboulis A.I. (2008). A CFD methodology for the design of sedimentation tanks in potable water treatment: Case study: The influence of a feed flow control baffle. *Chem Eng J*., 140(1), 110-121. doi:10.1016/j.cej.2007.09.022.

Le Moullec Y., Gentric C., PotierO. and Leclerc J.P.(2010). CFD simulation of the hydrodynamics and reactions in an activated sludge channel reactor of wastewater treatment. *Chem Eng Sci*., 65(1), 492-498. doi:10.1016/j.ces.2009.03.021.

Thool S.B. and Sinha S.L. (2014). Simulation of Room Airflow using CFD and Validation with Experimental Results. 6(5), 192-202

Autodesk (2016). Computational Fluid Dynamics Software. Autodesk CFD. http://www.autodesk.com/ products/cfd/overview. 1/11/2016.

STAR-CCM+ (2016). Discover better designs, Faster. http://mdx.plm. automation.siemens.com/star-ccm-plus.1/11/2016.

CFD Software (2016). Creating Computational Fluid DynamicsSimulations. https://www.comsol.com/cfdmodule. 1/11/2016.

Economon T.D., Palacios F., Copeland S.R., Lukaczyk T.W. and Alonso J.J. (2015). SU2: An Open-Source Suite for Multiphysics Simulation and Design. *AIAA J*., 54(3), 828-846. doi:10.2514/1.J053813.

Date Anil W., *Introduction to Computational Fluid Dynamics*, Cambridge University Press, New York, 2005.

HervéMorvan, *CFD Applied to Flood Flow*, School of Civil Engineering, The University of Nottingham, UK, 2005.

Miguel Francisco and António Gameiro Lopes, and Vítor Costa, *Optimization of a sanitary discharge valve using ANSYS CFX*, Department of Civil Engineering Mecanica, Univ.Coimbra, Portugal, 2006.