Research Journal of Engineering Sciences ISSN 2278 – 9472
Vol. **6(3),** 48-51, March **(2017)** *Res. J. Engineering Sci.*

Review Paper

Application of CFD software for planning and design of civil engineering structures

Mahesh R. Nalamwar[1*], Dhananjay K. Parbat[2] and D.P. Singh[3]
[1]Civil Engineering Department, Jagdambha College of Engg. & Tech., Yavatmal, India
[2]Civil Engineering Department, Govt. Polytechnic, Nagpur, India
[3]Civil Engineering Department, KDK College of Engineering, Nagpur, India
mahesh.nalamwar@gmail.com

Available online at: www.isca.in, www.isca.me
Received 30th November 2016, revised 25th February 2017, accepted 14th March 2017

Abstract

Computational fluid dynamics (CFD) was used widely by every field of engineering for solving critical problems. Paper discussed the application of CFD software for planning and design of 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 town and contaminant migration in hospital. With proper planning of windows location, windows size and ventilators, natural flow of air can be maintained in residential and hospital building. CFD simulation result are very sensitive to the large number of modeling parameters set by the user. CFD Parameters like Size of computational domain, grid resolution, boundary layer property, turbulence model selected for output and convergence criteria implemented in software are responsible for proper simulation output. Application of Autodesk CFD simulation was demonstrated for wind simulation in building under fixed location, fixed size of windows with uniform wind velocity.

Keywords: Computational Fluid Dynamics (CFD), CFD parameters, air flow, CFD in Civil Engineering, Autodesk CFD.

Introduction

Computational fluid dynamics is a branch of fluid mechanics. CFD uses the numerical methods and algorithms to solve problems involve fluid flows. CFD has combined fluid mechanics which can simulate compressible and incompressible fluid flow behavior. With development in computational speed of computers, CFD application is increasing for simulation. Simulation requires very high speed computers. Depending on complications in mode, solver takes time. With development on new algorithm computation time are reducing. With the help of CFD simulation product development cost can be reduced. Studied was done on comparison of wind tunnel effect and validation with CFD results. Study results shows that the results of CFD software are quite near to practical scale model study. Using simulation various alternatives can be explore to achieve the optimum result.

Navier-Strokes equation was the basis for all CFD software. 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[1]. CFD results calculate and displays 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[2].

Application of CFD in Civil Engineering

In straight alluvial channel, meandering pattern formation was studied by using three dimensional CFD model. Finite volume method for numeric modeling using an unstructured hexahedral cells grid. For pressure computation and predict turbulence, k-ε model was used. Computation result shows the replication of meander characteristics, cross section profiles, downstream meander migration, meander wavelength and cute formation[3].

For modern civil transport study CFD sensitivity analysis was conducted. Condition for modeling range from attached flow to flow with substantial separation. For modeling, two different Navier-Strokes codes and different turbulent models are used. Results are compared with wind tunnel data at flight Reynolds number. Near buffet onset, CFD sensitivities result variations due to code, Spatial differencing method, grid density, turbulence model and aeroelstic shape. Drag, lift and moment curve results are compared with experimental results calculate using wind tunnel[4].

Numerical study of interaction of wind and building opens the new branch of engineering named Computational wind engineering. Using FLUENT 5 CFD software, wind effects on tall building was examined. The effect of flow condition around the building was also modeled. *k–ε* method was used for turbulence model and results are compared with measurements from wind tunnel study. Study results are helpful to understand the effect of development of construction of new towers in city. Because of construction of towers, wind pressure assumed on already constructed building may change. Two modeling method are compared and discussion on results are done[5].

Using FLOW-3D, a three dimensional CFD model was developed with portion of lake and two spillway. During the probable maximum flood, spillway performance was studied. To reduce computation time three different grid resolutions were studied. Sensitivity of model for change in grid resolution was observed in results. For the computational mesh, three levels of successive grid refinement were used to minimize computational time and to provide adequate resolution of the final model results[6].

Computational fluid dynamics simulations was used to study, addition of vertical baffle wall at the feed section in sedimentation tank. Purpose was to improve the solids settling in water treatment. Comparisons was done between standard and a baffle-equipped tank. As per study result, it is found that due to insertion of vertical baffle wall at inlet, settling of solids in potable water treatment plant was improved. Baffle wall decreases the inlet recirculation area and direct solid settlement towards the bottom of the tank with high velocity[7].

CFD modeling of a wastewater gas–liquid cross-flow reactor was done, taking into account mass transfer, hydrodynamics and biological reactions. Details of Transfer processes, kinetics model and assumptions made are discussed. Results obtained from bench scale rector were compared with simulation results. Measurement of Nitrate, COD, ammonium and oxygen concentrations along the length of the reactor was done and compared to the simulated profiles. For nitrate concentration and COD profile, results are with good agreement. Without adjusting any kinetics parameters, results are obtained[8].

Computational fluid dynamics (CFD) was used to study the ventilation system of health care room. Simulation provides understanding of efficiency, reliability and adequacy of ventilation system. It also provides important suggestions for controlling energy consumption, patients comfort and air quality in room. In case study an actual hospital room was investigated to study the efficiency of ventilation, heating and air-conditioning plant. Considering different events of the patients like coughing or second breathing, three 3D models are prepared and studied. For simulating dispersal of bacteria-carrying droplets, particle tracing and diffusion model was developed[9].

Air circulation in model room was performed using k-ε Model and large-eddy simulation model. Experimental setup to study the air circulation in full scale model was very expensive. CFD approach is best solution for study of room airflow since it is low cost. It also allows flexibility to study the effect of various parameters on output results. It is found that experimental results and numerical simulation results, matches reasonably[10].

CFD Software's

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. Both category of software has different functionality available within software. Following are the few software used by the industry.

List of Some Commercial CFD Software: i. ANSYS - Fluent - Software used for faster optimize your product performance. With its physical modeling capability software models turbulence flow, heat transfer and reactions for industrial applications. Three transport equations related to k-epsilon model, the standard k-epsilon model, RNG k - ε model and the Realizable k - ε Model are available in software. Any type of fluid can be modeled in software. Software can be used by all branches of engineering for simulation. ii. IES VE – Microflo - Using CFD software undertake internal and external Air flow and heat transfer in and around buildings, taking into account boundary conditions such as internal energy sources, the effects of climate and HVAC systems. Calculate air temperature and airflow, occupant comfort temperatures. Detail natural and mixed mode ventilation strategies are available. Only standard k - ε turbulence model is available since use is specific for air circulation in building[11]. iii. 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. It includes 5 types of models. For turbulent flow study k- ε model was by researchers[12]. iv. 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 of the relevant physics. Software can be used to automatically drive the virtual product through operating scenarios and a range of design configurations. Eight different transformation equations are available in Star-CCM+ for choice of k - ε turbulence model[13]. v. 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. User Equation input option was provided to have full control over your CFD models[14].

List of Some Open source CFD Software: i. 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. With basic CFD solver, Open

FOAM solver includes Incompressible flow with RANS and LES capabilities, Buoyancy-driven flow solvers, Compressible flow solvers with RANS and LES capabilities, Solvers for conjugate heat transfer, Particle-tracking solvers, Direct Simulation Monte Carlo solvers etc. ii. FLUIDITY - An opens source, general purpose, multiphase computational fluid dynamics code capable of numerically solving the Navier-Stokes equation and accompanying field equations. It is parallelised using MPI and is capable of scaling to many thousands of processors. Equation can be arbitrary unstructured finite element meshes in one, two and three dimensions. Other features include the use of anisotropic adaptive mesh technology, and a user-friendly GUI, which can be used to calculate diagnostic fields, set prescribed fields or set user-defined boundary conditions. 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 (http://fluidityproject.github.io/). iii. 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[15]. iv. Palabos - Software is based on a modern approach called the lattice Boltzmann method. Through the innovative matrix-based interface, developing a new physical model has become simpler than ever. A software tool for classical CFD, complex physical interaction and particle-based models, Palabos offers a environment for fluid flow model simulations. (http://www.palabos.org/).

Case study of CFD for Indoor wind simulation

Simulation of wind flow inside the room was modeled to study the circulation of wind in all parts of room. Simulation helps in understanding the circulation of air inside room and to provide best comfort to occupant. As per interior design layout working area of occupant can be finalized and locations of windows or air duct can be planned to provide optimum air circulation to occupant. With proper planning of inlet and outlet location in rooms, optimization can be achieved in design of ventilation. Simulation study helps in design of mechanical or natural ventilation inside the building.

CFD simulation was done in AutodeskCFD® software. Model was prepared in Revit® software and imported in Autodesk CFD. Autodesk CFD doesn't have modeling tool. Simbuild® was developed by Autodesk to model the geometry and import in CFD for simulation. Room dimension are 4m x 4m. Inlet velocity of 1.6 m/s. Inlet window size was 1.2 x 1.2m. Velocity was measured at observation points. As per the interior layout in room and occupant working area model can be modified and effect of obstruction can be studied. Following result shows the simulation result wind flow inside the room.

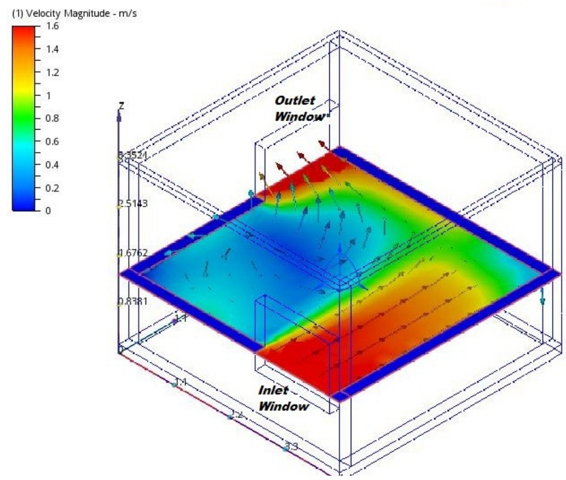

Figure-1: Room Model and simulation result.

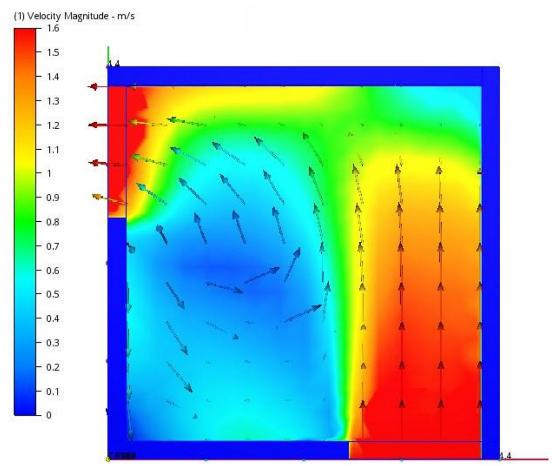

Figure-2: 2D Simulation Result.

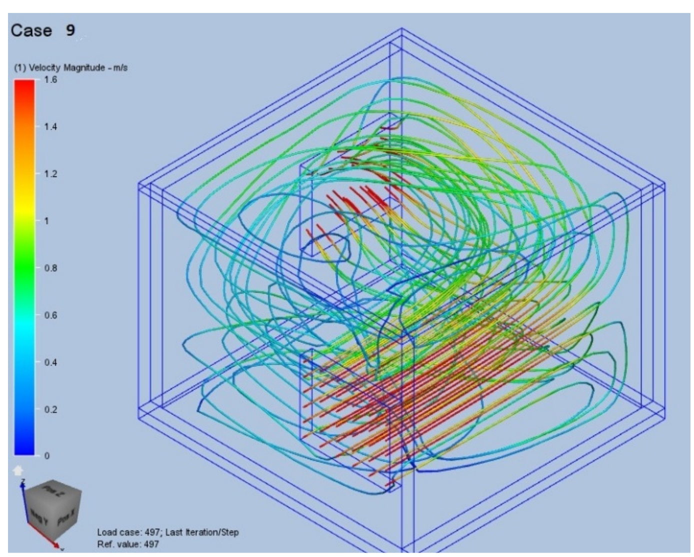

Figure-3: Trace path of air Circulation.

Conclusion

Computational fluid dynamics (CFD) can be used by civil engineers for design of various civil engineering structures. Researchers have used CFD software for modeling in environmental structures like wastewater gas–liquid cross-flow reactor, Sedimentation tank flow design, other Hydraulic structures, spillway design, turbine, pumps design, Thermal comfort study, Wind circulation study inside the building 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

1. Pavlovic D., Todorovic M., Jovanovic M. and Milosavljevic P. (2013). Comparison of Commercial CFD Software Packages. *ICAICTSEE.*

2. Li N. (2015). Comparison between three different CFD software and numerical simulation of an ambulance hall.

3. Olsen N.R.B. (2003). Three-Dimensional CFD Modeling of Self-Forming Meandering Channel. *J Hydraul Eng*., 129(5), 366-372. doi:10.1061/(ASCE)0733-9429.

4. Rumsey Christopher L., Allison Dennis O., Biedron Robert T., Buning Pieter G., Gainer Thomas G., Morrison Joseph H., Rivers S. Melissa, Mysko Stephen J. and Witkowski David P. (2001). CFD Sensitivity Analysis of a Modern Civil Transport Near Buffet-Onset Conditions.

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

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

7. 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.

8. Le Moullec Y., Gentric C., Potier O. 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.

9. Balocco C. (2011). Hospital ventilation simulation for the study of potential exposure to contaminants. *Build Simul*., 4(1), 5-20. doi:10.1007/s12273-011-0019-6.

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

11. IES V.E. (2016). MICROFLO Undertake internal or external air flow and thermal studies using Computation Fluid Dynamics (CDF) software. https://www.iesve.com/software/ve-for-engineers/module/MicroFlo/463. 1/11/2016.

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

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

14. CFD Software (2016). Creating Computational Fluid Dynamics Simulations. https://www.comsol.com/cfd-module. 1/11/2016.

15. 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.