MODELING HYDRAULIC STRUCTURES WITH COMPUTATIONAL FLUID DYNAMICS

Volkan Kiricci[1]*, Ahmet Ozan Celik[1]

[1] Dept. of Civil Eng., Anadolu University, 2 Eylul Campus, Eskisehir 26555, Turkey

Abstract

Computational Fluid Dynamics (CFD) is a method which combines fluid dynamics theory and computing power. CFD computations depend on fluid flow formulations to obtain numerical solutions for fluid flow problems. In some cases, the complexities in fluid behaviour mostly due to complex geometries (flow domains) make the solutions based on empirical formulation almost impossible. This highly influences the design procedure of hydraulic structures. However, using the advances in computer technology, CFD applications have become more effective and widespread among the branches of hydraulic engineering in facing the complex flow challenges.

Generally, design of hydraulic structures consists of empirical simplified assumptions and physical model experiments. Certain assumptions which are derived from empirical studies are mostly not appropriate for specific case problems in hydraulic structures. And the physical modelling is a quite problematic, expensive and time consuming process. In some cases, physical modelling can be useless for the geometries in which the necessary measurements of flow parameters are too difficult or not feasible. The obvious remedy is CFD method to cope with disadvantages of these classical design methodologies. Recent studies show that the accuracy levels of CFD analysis are approaching to that of physical model solutions provided that benchmark physical model tests are performed.

In this study we present a few case studies where CFD method is utilized either to design or investigate hydraulic structure projects. The method used, its limitations and the reliability of the results are discussed. Possible use of CFD method in hydraulic engineering applications is elaborated.

Key words

Computational fluid dynamics; numerical modelling; hydraulic structures; civil engineering

To cite this paper: *Kiricci, V., Celik, A. O. (2014). Modeling hydraulic structures with computational fluid dynamics, In conference proceedings of People, Buildings and Environment 2014, an international scientific conference, Kroměříž, Czech Republic, pp. 585-591, ISSN: 1805-6784.*

*Corresponding author: Tel.: +90-222-3213550-6613
E-mail address: vkiricci@anadolu.edu.tr

1 INTRODUCTION

Development in CFD method with advanced techniques and new approaches have made it a powerful tool for reliable and accurate modelling in many engineering applications, including aerospace, mechanical and atmospheric engineering fields. CFD, however, is not a very commonly applied method in civil engineering compared to the other fields but recent studies show that CFD solutions provide quite accurate results for design or rehabilitation of hydraulic structures and associated fluid dynamic problems [1,2]. CFD method can be particularly very useful in certain conditions where it is difficult to create physical models and in complex geometries where the measurement of required data is almost impossible. On the other hand, CFD is capable of solving two and three phase flow, that is, free surface flow and /or mixture with moderate to high turbulence levels in three dimensions where the empirical approaches are not sufficient.

This study presents the typical application of CFD in hydraulic engineering, its limitations and future direction, employing a few case studies in an effort to show details. In the next chapter we introduce the basics of CFD method.

2 COMPUTATIONAL FLUID DYNAMICS

CFD method basically depends on finite elements theory. Sets of differential equations which govern the conservation of momentum/energy and mass are utilized in discretised forms. Equations of motion of fluid are converted to algebraic expressions and integrated forms with the first order derivatives. Solution domain is divided in to finite number of cells by a numerical grid (mesh) and aforementioned equations are solved for dependent variables at each node of grid according to defined boundary conditions of the solution (flow) domain. It is also possible to run transient (time dependent) simulations with the desired time steps [3,4]. With these features, CFD method, given the experience and fluid flow knowledge, proves to be a powerful tool for resolving complex flows. Readers can refer to references 1-4 for examples of successful CFD applications of hydraulic engineering with physical model test validations.

The point to be emphasized here is the physical accuracy, stability and consistency of solutions of the numeric model. Along with the experience, solution must satisfy certain requirements to avoid truncation errors, round off errors and convergence problems. Reducing the mesh size might provide higher resolution solutions where the sudden changes occur in flow parameters (or smaller scale flow structures are relevant) but it also causes a significant increase in the computational load. Therefore, appropriate conditions must be applied in accordance with the essential hydraulic knowledge and requirements of models to optimize the computational time while maintaining the accuracy.

Methodology of CFD generally consists of 5 successive stages;

1. Creating (drawing) the geometry of the domain in which the flow occurs
2. Generating a numeric grid (meshing) with-in the domain
3. Defining the set up (boundary conditions, required initial values)
4. Solution (solver)
5. Results (visualization and evaluation of results)

In this study several CFD examples of hydraulic structures are analysed using a commercial solver (ANSYS CFX). CFD steps as well as results are discussed within the concept of method strengths and weaknesses.

3 CFD APPLICATIONS OF HYRAULIC STRUCTURES

Below we present simulations of a spillway (air/water), a river mouth (fresh and saline water mix including air water interphase) and an industrial chimney (smoke-air mix). All three cases are typical environmental flow conditions each posing different engineering problems and also CFD challenges. There are several successful applications in the literature where similar geometry and set-up conditions were studied [5]. We note that the models described here are not supported by physical model results though the simplicity in the fluid domains and the flow conditions similar to those in the literature are maintained in order to further comment on the method used here rather than the accuracy of the findings.

In many numerical model applications involving open channel flow, the main challenge is to accurately predict the behaviour of the free surface. This study does not aim to elaborate on the water/air interphase problems. However, the challenge can be faced by certain methods, such as volume fraction, provided by CFX. We show in Fig1 a, b and c respectively the CAD model, mesh and the set-up stages of spillway model with two channels and energy dissipaters. This flow domain was created with realistic geometries and inlet/outlet flow conditions with about 790,000 mesh cells (Fig 1d).

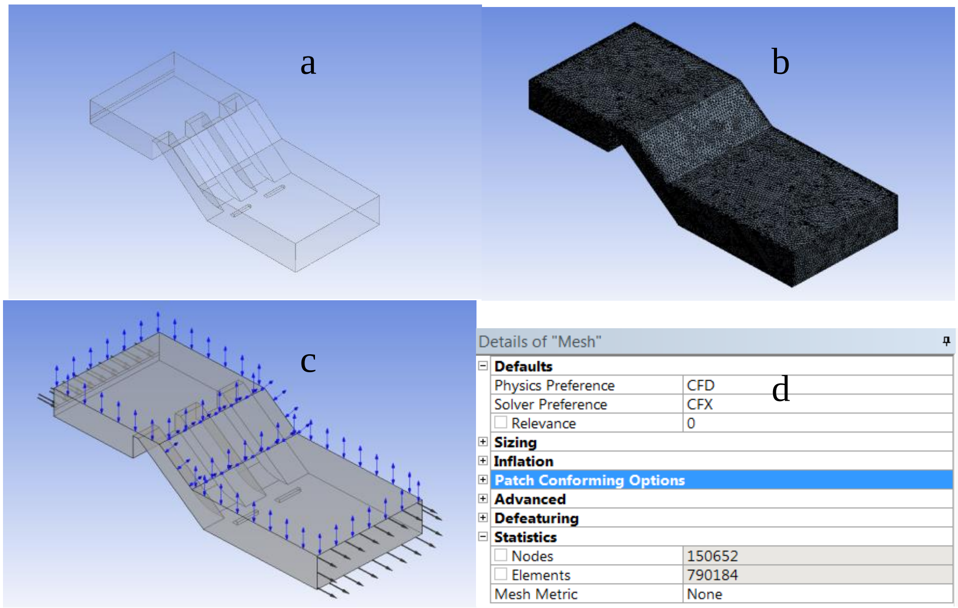


Fig. 1: Spillway model. a) CAD model, b) meshing, c) initial and boundary conditions (set-up), d) mesh details.

Here the mesh quality is quantified using a specific parameter called the mesh skewness. This parameter represents the convergence of each cell element to equilaterally sized ideal cells.

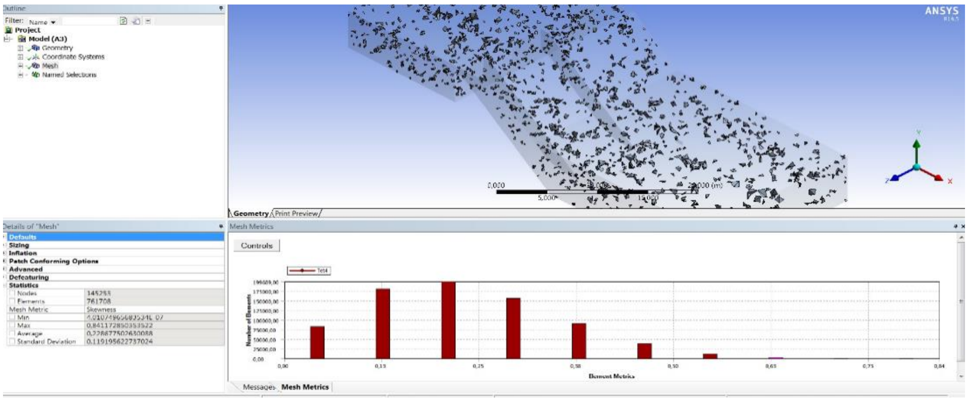


Fig. 2: Mesh quality and skewness parameter (mesh elements with high skewness value)

Similar procedures about the CAD and mesh stages were applied for all the models presented in this study.

The advantage of building such a numerical model is that the flood condition (discharge capacity) of the system can be tested using various discharge values. In addition, the cavitation check can be made at the channel bottom. Also the dynamic forces applied by the fluid on the solid boundaries can be detected. This simulation was time dependent and appropriate turbulence model was used for close. The results are presented in Fig. 3. The progression of a flood wave into the system can be monitored from the three successive images given in Figs 3a, b and c. The continuous contour plot of pressures acting on the solid boundary is obtained and given in Fig. 3d.

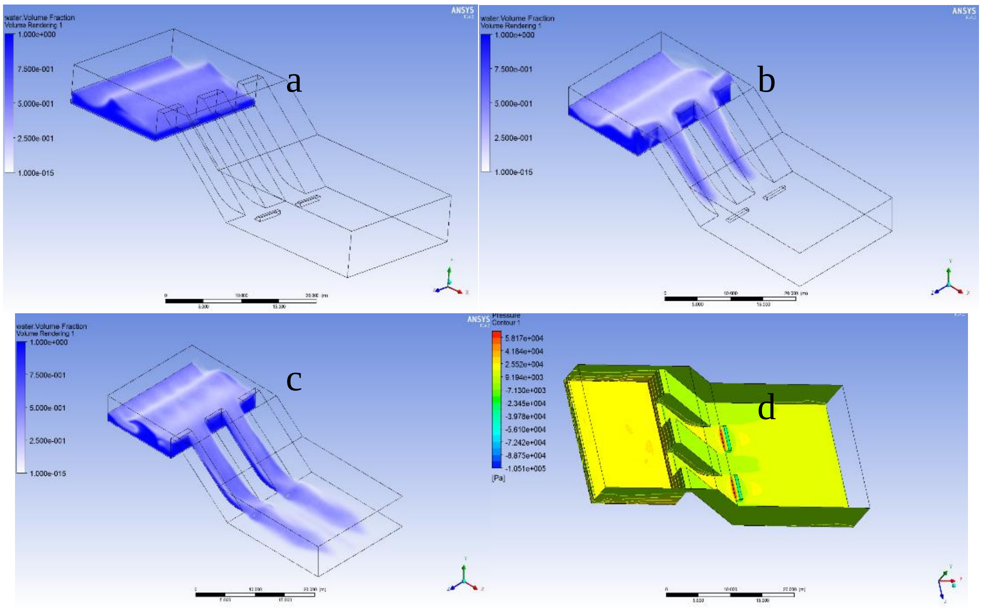


Fig. 3: Spillway model results. a) Flood at t= 2 s., b) flood at t= 4 s., c) flood at t = 6s. (set-up), d) pressure contour plot at the wall showing the steady state conditions. The volume rendering in a, b and c represent water volume fraction to detect the free surface.

In such a CFD model, the main problem is the outlet condition. A steady state condition has to be reached in the run while the inlet and outlet conditions are dominated by the initially defined boundary conditions. This problem is often encountered and possible solutions include a preliminary run to determine the actual inlet and/or outlet flow and pressure conditions. The prediction of free surface at the outlet for more complex geometries, in our opinion where the flow is non-uniform, has to be investigated carefully.

The spillway example shown here, when used cautiously, can be a very useful tool at the design stage to investigate/detect failures. Optimization in the geometry is also possible via various runs for which the better system performance can be monitored.

Here we introduce a much simpler model with single phase flow, a penstock section. The result from such a model is given in Fig. 4. We will not discuss the result here extensively as the model, because of its nature (pressurized flow), is straightforward and in many cases doesn't require physical model validation. In this model the stresses and pressures on the pipe can be checked for various steady and unsteady flow conditions.

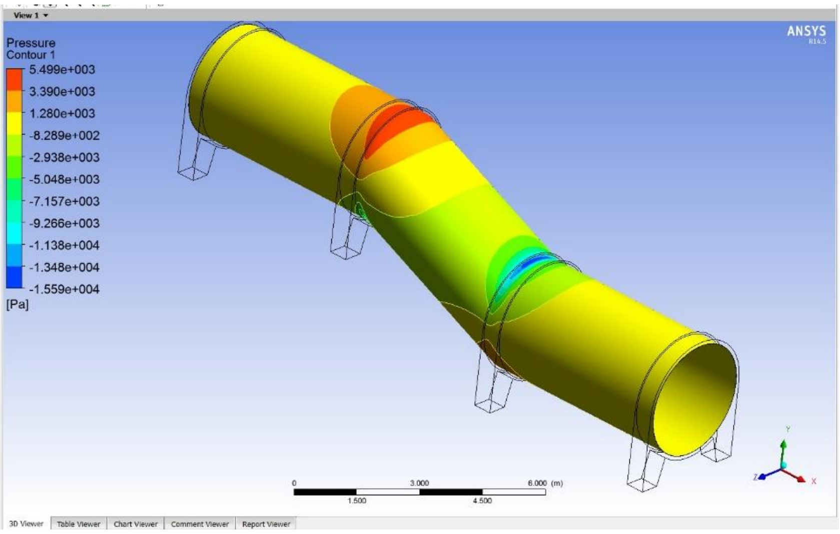
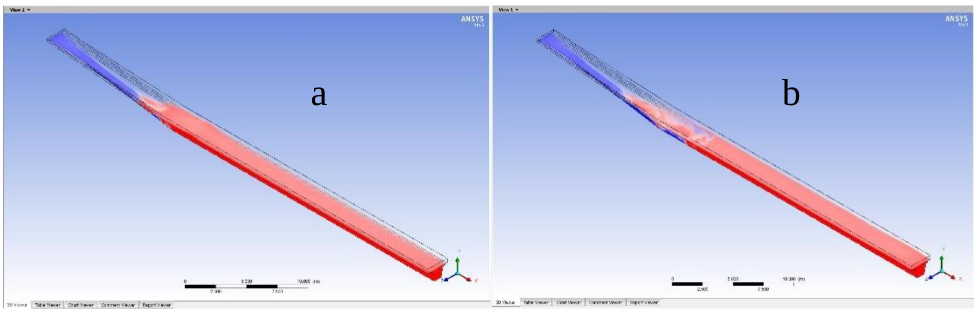


Fig. 4: Contour plot of pressure acting on a penstock section.

Below we continue with the examples of more complex environmental flows. Next case is where we studied a river mouth. Fresh water is discharged into the saline water and the main problem here being investigated is the behaviour of the fresh water and the changes in the water level. This is a typical case where the flood prevention and wall resistance is of concern. The model is required as the empirical approaches cannot explain the phenomenon described here. In Figs. 5a, b and c we show the sequential fresh water movement into the saline water and the mix. The visual results are satisfactory while the experimental validation is still needed as explained below.



c

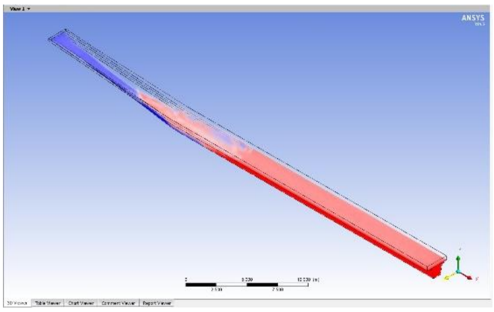


Fig. 5: Volume fractions of fresh (blue) and saline (red) water in a river mouth. a, b and c modules show successive time steps from the simulation.

As far as the CFD model set-up is concerned, such a study is the most challenging and demanding case. By that we mean, the parameters regarding the mix in a liquid/liquid mix case like this requires experiments to obtain important parameters such as the mass flux and characteristic length scale over which the mix occurs on the average between the given two different kinds of liquids [6]. Yet another difficulty is that this is a multi-phase solution where the influence of air (as well as the movement of air) also needs to be reflected and solved. Our results are from a preliminary run. We note that an accurate liquid/liquid mix model requires further investigation as the mixing agent, namely the turbulence and its level is case specific which causes variations in the set-up parameters for different geometry and flow conditions.

The final case presented is a chimney where the interaction of air (with wind flow) and smoke (CO_2 in our simulation) rising from the chimney is investigated Fig. 6.

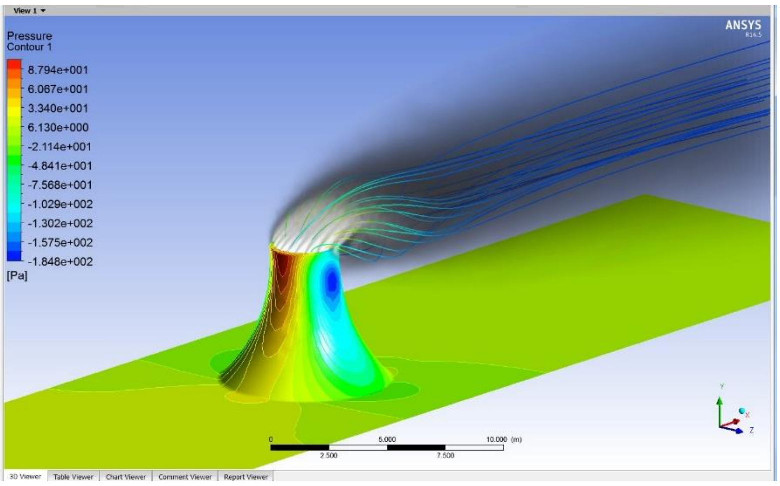


Fig. 6: Pressure contours on the solid walls of a chimney and the streamlines of smoke flow.

The problem of mix is present here again while the fluids being mixed due to turbulence is air and CO_2. Therefore the same statements as above apply here regarding the accuracy of such a model. However, the steady state solutions where the mean flow parameters are concerned can accurately be detected using appropriate mix parameters available in the literature [7].

4 CONCLUSIONS

In this study we showed that the CFD method can be used in many civil engineering applications. We state again that CFD modelling requires a very solid knowledge of fluid mechanics. In addition, the experimental validations of certain cases/parameters involving complex flow environments such as multiphase flow cases with mixture is needed to provide benchmark solutions for researchers and designers. But the promising developments in commercial software, when used carefully, seem to provide a very powerful tool to face challenges the environmental flows bring in right hands.

REFERENCES

[1] Liaqat, A. K., Edward, A. W., Mizan, R. (2005). A 3D CFD model analysis of the hydraulics of an outfall structure at a power plant. *Journal of Hydroinformatics*, **7**(4), pp. 283-290.

[2] Hoseini, S. H., Jahromi, S. H. M., Vahid, M. S. R. (2013). Determination of Discharge Coefficient of Rectangular Broad-Crested Side Weir in Trapezoidal Channel by CFD. *International Journal of Hydraulic Engineering*, **2**(4), pp. 64-70.

[3] Andersson, B., Andersson, R., Hakansson, L., Mortensen, M., Sudiyo, R., vanWachem, B., Hellstrom, L. (2012). *Computational Fluid Dynamics for Engineers*. 1st ed. Cambridge University Press. ISBN 978-1-107-01895-2.

[4] Lomax, H., Pulliam, T. H., Zingg, D. W. (2002). *Fundamentals of Computational Fluid Dynamics*. Springer-Verlag.

[5] Li, S., Cain, S., Wosnik, M., Miller, C., Kocahan, H., Wyckoff, R. (2011). Numerical Modeling of Probable Maximum Flood Flowing through a System of Spillways. *Journal of Hydraulic Engineering*, **137**, pp. 66-74.

[6] Lagumbay, R. S. (2006). Modeling and Simulation of Multiphase/Multicomponent Flows. *PhD Thesis, University of Colorado.*

[7] Ismail, M. H. S., Hussain S. A., Shavandi, M. A., Haddadian, Z. (2012). Simulation of Hot Gas Desulfurization Using Liquid Tin in Scrubber. *Research Journal in Engineering and Applied Sciences*, **1**(4), pp. 258-265.