

Using CFD in Hydraulic Structures

Mehmet Cihan Aydin (Corresponding author)
Bitlis Eren University, Civil Engineering Department, Bitlis, Turkey
E-mail: mcaydin@gmail.com

Ercan Isik
Bitlis Eren University, Civil Engineering Department, Bitlis, Turkey

Abstract

The theoretical background of the fluid mechanics and hydraulics had been almost completed in the eighteenth nineteenth centuries. Since that day, the physical hydraulic models have helped development of hydraulic structures empirically. Although the advanced analytical and numerical methods have been developed together with fluid mechanics theory, the use of these methods has not been possible in practice until recent years due to computation difficulties. However, with the developing in computer technology and numerical solutions methods, Computational Fluid Dynamics (CFD) techniques have been used widely in the hydraulics such as other relevant scopes, especially since the beginning of the 21th century. In this study, the using of CFD in the hydraulic structures was explained with some examples and the advantage and disadvantage of this method were discussed.

Keywords: CFD, Fluid Dynamics, Hydraulic structures, Turbulence modeling

1. Introduction

The first known theoretical contribution in fluid mechanics was done Archimedes (285-212 B.C.) who formulated the laws buoyancy. Leonardo da Vinci (1452-1519) who was an excellent experimentalist derived the equation of conservation of mass in one-dimensional steady flow. Isaac Newton (1642-1727) found out the law of viscosity of the linear fluids now called newtonian. Leonhard Euler (1707-1783) developed both differential equations of motion and their integrated form, now called Bernoulli equation. Since perfect-fluid assumption have very limited application in practice, engineers began to reject what they regarded as a totally unrealistic theory and developed the science of *hydraulics*. Then the scientists carried out their study based almost entirely on experiments. At the end of the nineteenth century, the using of combination of experimental hydraulics and theoretical hydrodynamics began with developing model laws and dimensional analysis by William Froude (1810-1879) and his son Robert (1846-1924), Lord Rayleigh (1842-1919) and Osborne Reynolds (1842-1912). Nevier (1785-1836) and Stokes (1819-1903) derived an equation set of fluid motion called Navier-Stokes equations adding newtonian viscous term. But, applying these equations to the most of real flow was very difficult. In the first part of 20th century, a lot of study performed on boundary layer theory and turbulence modelling. In 1904, a German engineering Ludving Prandtl (1875-1953) pointed out that fluid flow with small viscosity, such as water flow, can be divided into a thin viscous layer called as *boundary layer* near wall, patched onto a nearly inviscid outer layer, where Euler and Bernoulli equations apply. Boundary-layer theory proposed by Prandtl has been a most important toll in progressing of modern fluid flow analysis. Theodore von Karman (1975-1953) analyzed what is now known as the von Karman vortex street. In 1922, the firs numerical weather prediction system was developed by Lewis Fry Richardson (1881-1953). The earliest numerical solution was done for flow past a cylinder by Thom (1933). Kawaguti (1953) calculated a solution for flow around a cylinder by using a mechanical desk calculator, working 20 hours per week for 18 months. During the 1960s and 1970s, many numerical methods which are still in use today had been developed. (White 2003; Cengel and Cimbala, 2015; Bakker, 2006).

Previously, CFD was only carried out by academics and researchers. After 1980s many commercial CFD codes were released. Recently, CFD analyses have been applied many of areas such as aerodynamics, aerospace, automotive industries. The evolution of CFD codes is rising very rapidly in the twenty-first century. From the beginning of this century, a few advanced 2D and 3D CFD programs have been emerged, i.e. FLUENT, CFX, Polyflow, FIDAP, FLOW-3D. Fluent and Flow-3D which is the most used in the worldwide is a general purpose finite volume program. FLUNET is using structured and

unstructured non-orthogonal 3D grid while FLOW-3D is only using a 3D orthogonal grid. These codes present very good options for multiphase flow which enables simulation of free surface flows.
In this study, the main technics and procedures of CFD for hydraulic structures are presented to encourage the using of CFD in civil engineering. The advantage and disadvantage of CFD with some examples are discussed in the last of the study.

2. Basics of CFD

Computational fluid dynamics (CFD) is a simulation method of fluid flow, head and mass transfer, chemical reactions and other similar fluid patterns by solving mathematical equations with numerical processes. Since beginning of the twenty-first century, the using of CFD in the engineering and industrial applications has risen with advancing of numerical of computational techniques. Because conventional method in the hydraulics is the use of experimental physical models, the researchers have often approached with suspicion to the results of numerical models. Therefore, it is used to using the numerical models together with physical model. Although the numerical models have some errors and uncertainties such as discretization, convergence (iteration), geometry, boundary condition, initial condition, parameter and user uncertainties and errors; the physical models also have some disadvantages such as the scale effects, the measuring instruments and users' errors, the obtaining undetailed results, more time and cost consumptions. But however, the both methods have many important advantages. A verified numerical method with some experimental data or a case study can be used to analyze hydraulic problems more detailed from experimental study.

3. Discretization Methods

Some of the discretization methods of Navier-Stokes equations being used are finite volume, finite element, finite difference, spectral element and boundary element methods. The finite volume method is general method used in CFD software. This method has an advantage in memory usage and iteration convergence especially for large models and high turbulent flows. In the finite volume method, the governing equations which are typically Navier-Stokes equations and the mass and energy conservation equations are solved for each finite volume cells throughout the domain.
The turbulent multiphase flows are very complex. The flows over hydraulic structures are often free surface flows that are turbulent and multiphase flows. The numerical simulations of free surface flows are considerable sophisticated due to their surface opening atmosphere, turbulence and momentum equations. The volume of fluid (VOF) method can be best choice to estimate the free surface and to solve their governing equations of the flow. The momentum equation is dependent on the volume fraction of all phases in the VOF method.
The momentum equation:

$$\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial P_s}{\partial x_j} + \frac{\partial}{\partial x_i}\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \rho g_j + F_j \tag{1}$$

Volume fraction equation:

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \nabla.\left(\alpha_q \rho_q \vec{v}_q\right) = S_{\alpha_q} + \sum_{p=1}^{n}\left(\dot{m}_{pq} - \dot{m}_{qp}\right)\right] \tag{2}$$

In these equations; ρ is fluid density, u is velocity vectors, μ is dynamic viscosity, P_s is pressure and F is a body force; $\dot{m}_{qp}$ and $\dot{m}_{pq}$ are mass transfer between both phases (from q to p and from p to q respectively), α_q is q^{th} fluid's volume fraction in a cell. The term of S_{α_q} is zero by default, α_q is between 0 and 1 whether cell is empty or full, the subscripts of p and q represent phases of fluid. (ANSYS-FLUENT, 2012)

4. Turbulence Models

The choosing of turbulence models is very important in respect to the effect on numerical results. So, some different turbulence models should be compared, and the most suitable of which can be selected for relevant problem.

4.1. Direct Numerical Simulation (DSN)

Direct numerical simulation (DSN) is used for resolving the entire range of turbulent length scales. This method requires extremely fine grids (3D), and is extremely expensive in terms of computational cost and time consumption especially for high Reynolds numbers. Therefore, this model is not applicable at CFD solutions of turbulent flows with high Reynolds numbers.

4.2. Large Eddy Simulation (LES)

A lower level of DSN is large eddy simulation (LES). This model allows the largest and most important turbulent scales to be resolved, and smallest scale of turbulent is ignored. Although LES uses much less computational resources than DSN, it is also too hard to use DSN in the present computation technology. This method requires greater computational resources than RANS methods.

4.3. Detached Eddy Simulation (DES)

Detached eddy simulation (DES) is a modification of a RANS model in which it uses fine grid enough for LES calculation in region near to solid walls. When the turbulent length scale overcomes the finite cell dimension, this domain is resolved by the LES method. Thereby the computational cost is decreased. DSN is first formulated for Spalard-Allmaras model (Spalart 1997), but it can be applied to other RANS models. Therefore, while Spalart-Allmaras model based DES acts as LES with a wall model, DES based on other models (like two equation models) behave as a hybrid RANS-LES model. The some other models for DES are the realizable k-ε and SST k-ω models.

4.4. Reynolds-averaged Navier-Stokes (RANS)

The one of the oldest turbulence modelling approach is Reynolds-averaged Navier-Stokes (RANS) equations. In this method, the governing equation with new apparent stresses called as Reynolds stresses is solved. There are two main approaches for RANS models: The first is Boussinesq hypothesis which use an algebraic equation for Reynolds stresses, solving transport equations for determining the turbulent kinetic energy and dissipation (k-ε). The model include Models include k-ε (Launder and Spalding, 1974), Mixing Length Model (*Prandtl*) and Zero Equation Model (Wilcox, Devid, 2006). The second approach is Reynolds stress model (RSM) which solves transport equations for Reynolds stresses. This method is much more costly for computational effort in order to solve several transport equations for all Reynolds stresses. The standard, RNG and realizable k-ε models present similar form with transport equations for k and ε, except from following major differences: i. the method of calculating turbulent viscosity, ii. the turbulent Prandtl numbers governing the turbulent diffusion of k and ε, iii. the generation and destruction terms in the ε equation. While the standard k-ε including two-equations is based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε), the RNG k-ε model is derived using a statistical technique called renormalization group theory. The realizable k-ε model, as distinct from the standard k-ε model, contains an alternative turbulent viscosity formulation, and it uses a modified transport equation for dissipation rate (ε) (Ansys-Fluent, 2012).

5. Numerical Uncertainties and Verification

The most of numerical uncertainties can be defined as iterative convergence, grid type and size, selection of turbulence model. For iterative convergence, it is generally stated that the normalized residuals for each equation in the numerical solutions must be drop under at least 10^{-3}. Especially for time-dependent problems, the convergence should be ensuring at the every time steps. For grid convergence, the independence from the grid sizes of numerical results should be showed. For this, the changes of a key parameter with grid size, which is important for your problems, can be examined. More sophisticated methods such as Grid Convergence Index (GCI) method in literature should be used to reveal grid effects. Generally, the hexahedral meshes are more adequate for same cell numbers. But, the solution domain cannot always be meshed by hexahedral. Therefore the grid skewness must be controlled. Mesh density should be high enough to ensure the flow features for all fields, and if it is important for your problems, the boundary layers adjacent to walls should be defined by high resolutions mesh normal to wall. The skewness of a cell can be defined as follows:

$$Skewness = \frac{optimal\_cell\_size - cell\_size}{optimal\_cell\_size} \tag{3}$$

The range of skewness is 0 (best quality) to 1(worst quality). It is offered that the skeness should not exceed 0.85 for hexhedral, quad and triangular, 90 for tetrahedral cells.

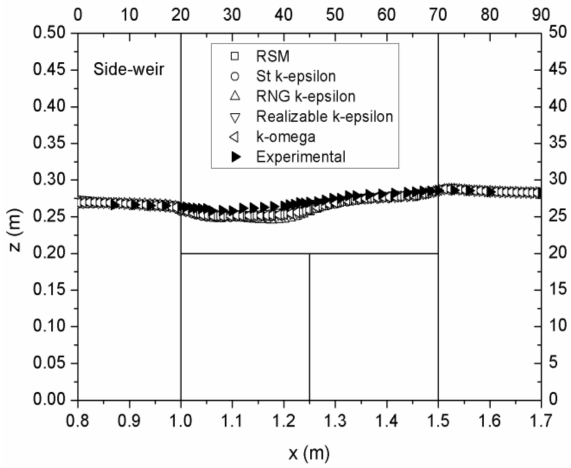
Table 1. Cell quality and their skewness

Skewness	0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.99	0.99-1.00
Quality of Cell	Excellent	Good	Acceptable	Poor	Sliver	Degenerate

The defining of numerical uncertainties mentioned above is not enough to verify numerical analysis. The results of numerical solutions should be calibrated by real prototype or experimental observation, with a benchmark or case study. Benchmark solutions may be either analytical solutions or highly accurate numerical solutions. So the validation and verification of the numerical analyses can be ensured.

6. Examples of Hydraulics Application

Aydin (2012) modelled successfully free surface flow over a triangular side weir by using FLUENT with VOF method and different turbulence models. The results show that while the all turbulence models give similar free surface profiles with experimental data (Fig. 1), the RSM turbulence model gives the best results in terms of vortex occurrence in the triangular side weir (Fig. 2). The RSM model also gave the best simulation of the waves on the surface. The velocity vectors and pathlines at the free surface flow over the side weir with RSM turbulence model are illustrated in Fig. 3. Aydın and Emiroglu (2013) investigated the discharge capacity of this side weir by using CFD, and they obtained good results also in respect to discharges of the side weir. Another CFD simulation was given in Fig. 4 which shoved the free surface flow over two cycles labyrinth side weir. These study show that the CFD can be a good tool to analyze free surface flows such as over hydraulic structures, provided that it is used in accordance.



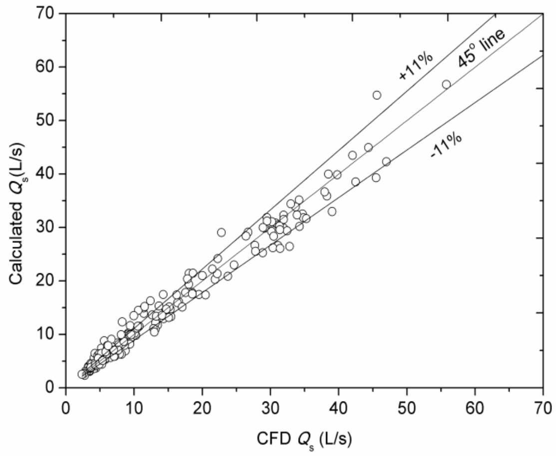


Fig. 1.a) Free surface levels over a triangular side weir for different turbulence models and experimental (Aydın 2012), b) Comparison discharges obtained by CFD with empirical results. (Aydin and Emiroglu 2013)

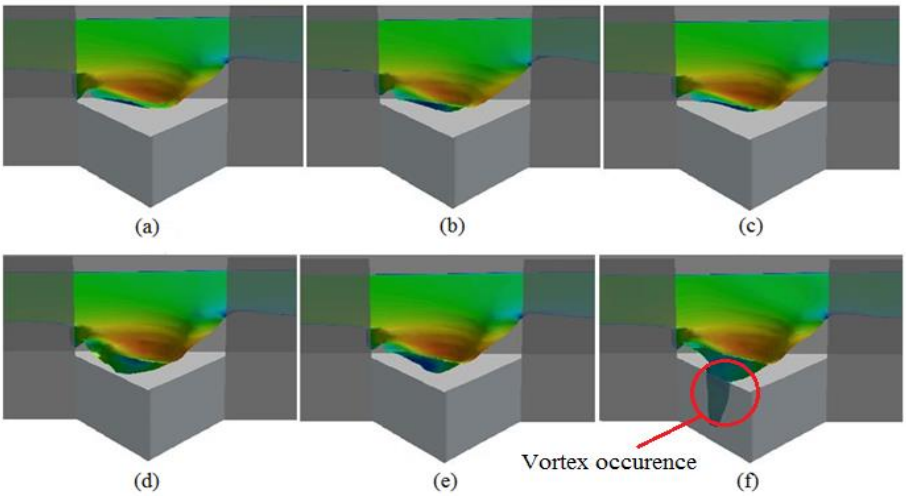


Fig.2. Vortex occurrence in a triangular labyrinth side weir for different turbulence models: a) Spalart-Allmaras, b) k-omega, c) Standard k-epsilon, d) RNG k-epsilon, e) Realizable k-epsilon, f) RSM model (Aydin 2012)

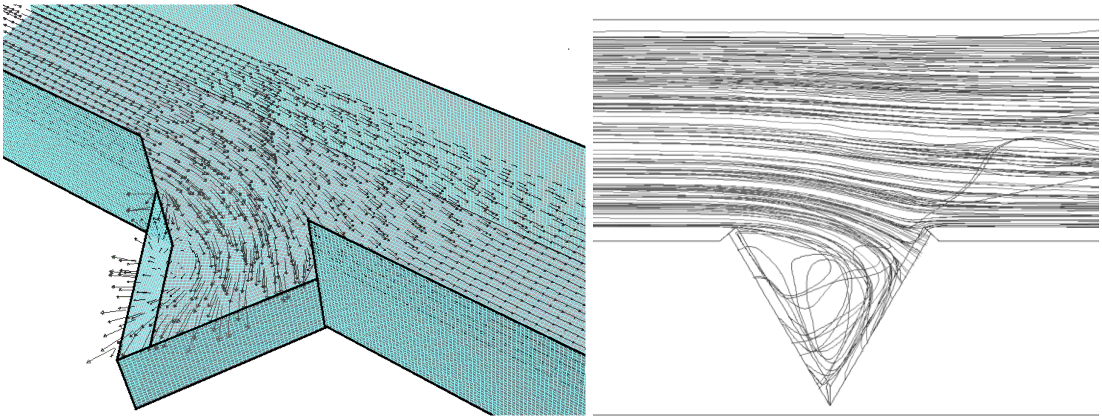


Fig 3. CFD views of side weir: velocity vectors and pathlines over the side weir flow.

Fig. 4. Views of free surface flows over a labyrinth side weir: (a) CFD, (b) Experimental.

Another CFD simulations of a siphon side weir assemble to an open channel are illustrated in Fig. 5 and 6. The free surface of CFD model was obtained similar to that of experimental as shown Fig.5. The velocity profiles of the both CFD and experimental along flow depth are shown in Fig. 6. It is seen that the CFD results is considerable agree with experimental and real flow conditions. The simulation was performed by VOF method, RSM turbulence model with 200.000 hexahedral cells (Aydin et al. 2015).

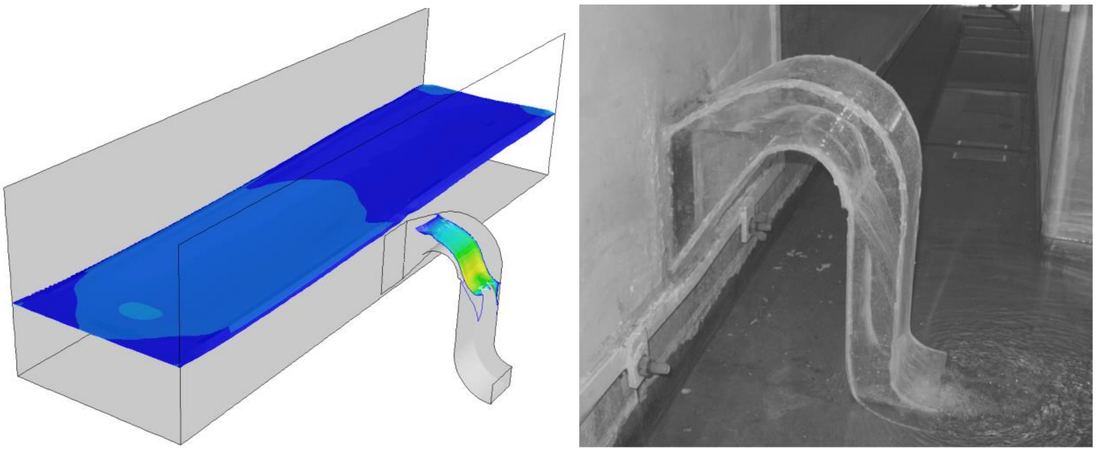


Fig 5. CFD simulation of a siphon side weir.

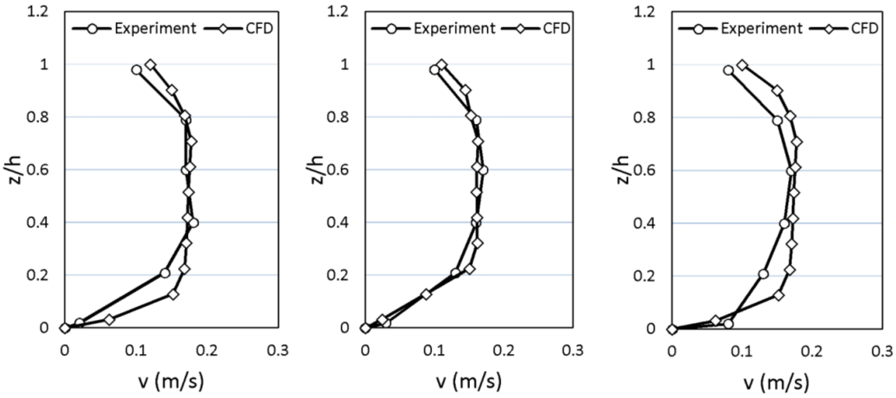


Fig. 6. Velocity profiles along flow depth at the main channel centerline (Aydin et al. 2015)

The spillway aerator of a large dam was simulated by using VOF method and k-epsilon turbulence model with 1782000 mixed cells (Fig. 7). The CFD modelling of spillway aerator is very complex due to their air-water mixing flows. But, since the physical modeling of this type structures is required quite cost and effort, the well validated numerical models can be used in the initial stage of a project, or further.

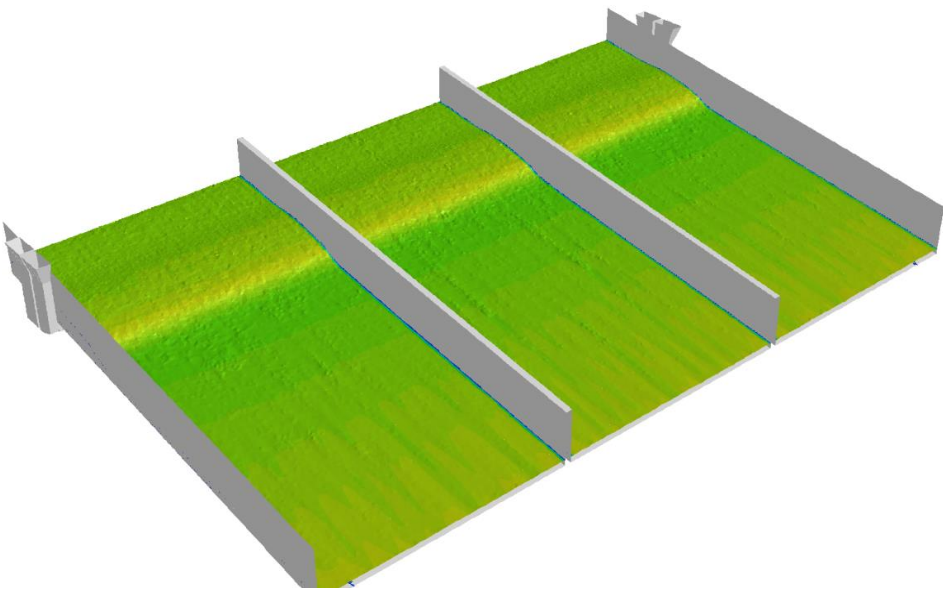


Fig. 7. CFD simulation of spillway aerator of a large dam (Ilisu Dam Spillway in Turkey).

7. Conclusions

There are numerous applications of CFD in various fields, e.g. industrial process, aerodynamics of vehicles, electronic and mechanical engineering. Because the design of hydraulic structures in especially civil engineering traditionally based on physical laboratory models, the using of numerical model in this area has been approached with suspicion for a long time. Although the theoretic fundamentals of fluid mechanics were obtained long time ego, the CFD technics have been widely used in design of hydraulic structures since the beginning of 21th century.

There are many advantage of CFD usage. For example, CFD simulations are relatively cheaper, faster and more flexible than physical experiments and tests. The some specific results which cannot be observed in experiments are possible by using the CFD analyses. CFD solutions provide the ability to theoretically simulate any physical condition. On the other hand, some disadvantages of CFD as follows: Numerical iteration introduces many numerical errors such as discretization and round-off errors. The mesh quality and sensitivity also play important role upon solution accuracy. Therefore the numerical errors should be well observed and revealed, and the mesh quality is should be paid attention. Additionally, the CFD solutions need to be verified and calibrated with real physical models or prototypes. So, the accuracy of CFD solutions can base on the physical models results.

Consequently, it can be stated that if the CFD simulations are used properly, it can be a good choice for designing and analyzing of hydraulic structures as in other applications.

References

ANSYS-FLUENT. 2012. Fluent Theory Guide, ANSYS Help System, ANSYS Inc.

Aydin M.C. 2012. CFD simulation of free-surface flow over triangular labyrinth side weir, Advances in Engineering Software, 45(2012): 159-166.

Aydin M.C., Emiroglu M.E. 2013. Determination of capacity of labyrinth side weir by CFD. Flow Measurement and Instrumentation, 29(2013):1-8.

Aydin M.C., Ozturk M, Yucel A. (2015). Experimental and numerical investigation of self-priming siphon side weir on a straight open channe. Flow Measurement and Instrumentation, 45(2015): 140-150.

Bakker A. 2006. Applied Computational Fluid Dynamics, Lecture1-Introduction to CFD. In www.bakker.org.

Cengel Y.A., Cimbala J.M. 2015. Akışkanlar Mekaniği Temelleri ve Uygulamaları, Palme Yayıncılık, 984s.

Launder, B. E.; D. B. Spalding (1974). "The Numerical Computation of Turbulent Flows". Computer Methods in Applied Mechanics and Engineering 3 (2): 269–289.

Spalart, P.R. (August 1997). Comments on the feasibility of LES for wing and on a hybrid RANS/LES approach. 1st ASOSR CONFERENCE on DNS/LES. Arlington, TX.

Thom A. 'The Flow Past Circular Cylinders at Low Speeds', Proc. Royal Society, A141, pp. 651-666, London, 1933

White F.M. 2003. Fluid Mechanics, fifty edition, International Edition, McCraw-Hill. 866p.

Wilcox, David C. (2006). Turbulence Modeling for CFD (3 ed.). DCW Industries, Inc. ISBN 978-1-928729-08-2.