Proceedings of 2013 IAHR Congress

A Strategy for the Interaction between Hydraulic and Numerical Models

Angel N. Menendez,
Head of Program, INA (National Institute for Water); Professor, Dept. of Engineering, Univ. of Buenos Aires, Argentina. Email: angel.menendez@speedy.com.ar

Nicolas D. Badano,
Graduate Student, INA (National Institute for Water) and Dept. of Engineering, Univ. of Buenos Aires, Argentina. E-mail: nicolas.d.badano@gmail.com

Emilio A. Lecertua,
Research Assistant, INA (National Institute for Water); Teaching Assistant, Dept. of Engineering, Univ. of Buenos Aires. Argentina. E-mail: ealecer@yahoo.com.ar

ABSTRACT: A strategy for the design of hydraulic structures is proposed, consisting in a first stage where the flow in the physical model is numerically simulated in order to validate the numerical model, and in a second stage where the flow in the prototype is numerically simulated in order to extrapolate the results to this scale. This is illustrated for two problems with quite different levels of complexity, associated to the design of the Third Set of Locks of the Panama Canal: the determination of the time for water level equalization between chambers, and the calculation of the amplitude of free surface oscillations in the lock chambers.

KEY WORDS: Hydraulic models, Numerical models, Hydraulic design, Scale effects, Models interaction.

1 INTRODUCTION

The design of complex hydraulic structures requires its testing through hydraulic models. The main practical limitation of hydraulic models are the so called ‘scale effects’, i.e., the fact that only the primary physical mechanisms can be correctly represented, while the secondary ones are distorted. In particular, for free surface flows the gravitational driving forces – primary mechanism – must be correctly scaled in relation to inertia (Froude scaling), leading to an incorrect representation of viscous forces (no Reynolds scaling) – usually the leading secondary mechanism – as the fluid in the hydraulic model is the same as in the prototype (water). Though for most applications Reynolds number effects introduce only small quantitative deviations, which can be readily absorbed within the margin of safety assumed for design, this is not always the case. In fact, they can for example accumulate, in such a way that the effects compete with those arising from the primary mechanism. In those cases, being the Reynolds effects distorted in the hydraulic model, the observed response deviates from the one corresponding to the prototype, thus needing some empirical correction.

Numerical modeling is the appropriate tool to help solving in a rigorous way this type of difficulty. A sound numerical model should be able to correctly represent both the primary and secondary mechanisms, i.e., it is not subject to ‘scale effects’. Its main limitations might arise from insufficient resolution, or from inaccurate representation of turbulence effects. The first limitation could be overcome by reducing the spatial step of the numerical grid; the second one, by resorting to more elaborated theoretical approaches.

Based on these observations, the following strategy is proposed: (i) the flow in the hydraulic model is numerically simulated, i.e., the dimensions of the hydraulic model are used (thus accounting for the ‘spurious’ scale effects); this constitutes a way of validating the theoretical model; eventually, adjustments in the representation (higher resolution, more elaborated theoretical approaches) are introduced in order to improve the comparison; (ii) the flow in the prototype is numerically simulated, by introducing the dimensions of the prototype in the validated numerical model (i.e., distortion of secondary mechanisms is now avoided); this constitutes the adequate way of extrapolating the results to the prototype dimensions.

Two problems (with quite different levels of complexity) are presented as case studies in order to illustrate the proposed approach, both of them associated to the design of the Third Set of Locks of the Panama Canal (communicating the Atlantic and Pacific Oceans), for which the present authors were responsible: (a) the determination of the time for water level equalization between chambers, for which a one-dimensional numerical model was used; (b) the calculation of the amplitude of free surface oscillations in the lock chambers (which leads to increments in the hawser forces) due to close-to-resonance conditions under interaction with an oscillation in a flow partition component of the filling/emptying system (triggered by large turbulent eddies), for which a full three-dimensional numerical model – i.e., a CFD approach – was applied. More details are presented in Menendez and Badano (2012).

2 EQUALIZATION TIMES

The equalization time, named as Filling/Emptying (F/E) time during the study, is a key parameter in establishing the system performance of a Lock Complex, as it has a direct impact on the vessel throughput, measured as the number of vessels passing through the system per day. Contractual requirements existed for the design of the Third Set of Locks of the Panama Canal, imposing maximum allowable F/E times for different scenarios. Minimization of these F/E times, through the reduction of local head losses, was the main strategy used during the design optimization process. Consequently, scale effects affecting these F/E times were carefully studied.

The Third Set of Locks of the Panama Canal, presently under construction, comprises twin lock complexes located near each ocean. Each complex has three lock chambers in series. Each lock chamber has three side pools, called Water Saving Basins (WSB). These WSBs store part of the water used during the equalization operations, that otherwise would be flushed downstream towards the ocean. This stored water is then utilized to refill part of the lock, allowing a reduction of freshwater consumption during dry hydrological seasons. In order to allow an even filling or emptying of the chambers, thus minimizing longitudinal water surface slopes and thus hawser forces, water enters or exits each lock chamber through 20 ports located on each lateral wall. They are connected to secondary culverts which, in turn, connect to the main culvert at the midpoint of each chamber, through a carefully designed hydraulic component called Central Connection (CC) (Figure 1). The two branches of the CC are called ‘U’ and ‘S’ branches, in reference to the trajectory followed by the incoming flow.

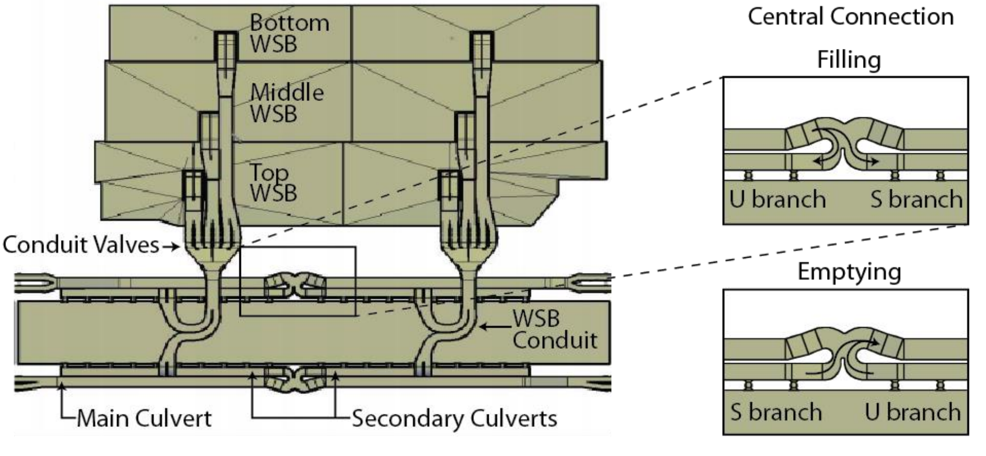

Figure 1 Detail of the F/E system and Central Connection

The hydrodynamic model of the F/E system was built using Flowmaster V7 (http://www.flowmaster.com/), a commercial code which solves one-dimensional transient flow over a network of conduits. For convenience, the whole F/E system was divided into sub models, one for each type of operation.

The Third Set of Locks has been subject to physical modeling at the Compagnie Nationale du Rhône (CNR), Lyon, France. The physical model was built at a 1/30 scale. Extensive tests were made for various normal and special operations, measuring water levels, discharges, pressures and water slopes in the chambers.

The flow in the hydraulic model was numerically simulated. Real physical dimensions of the physical model components were used. Local head loss coefficients for the special hydraulic components were obtained through steady CFD modeling. The results obtained with the numerical model showed a very good agreement with physical model measurements, for different operations and conditions. As an illustration, Figure 2 shows the comparison for a typical Lock to Lock operation (with results scaled up to prototype dimensions).

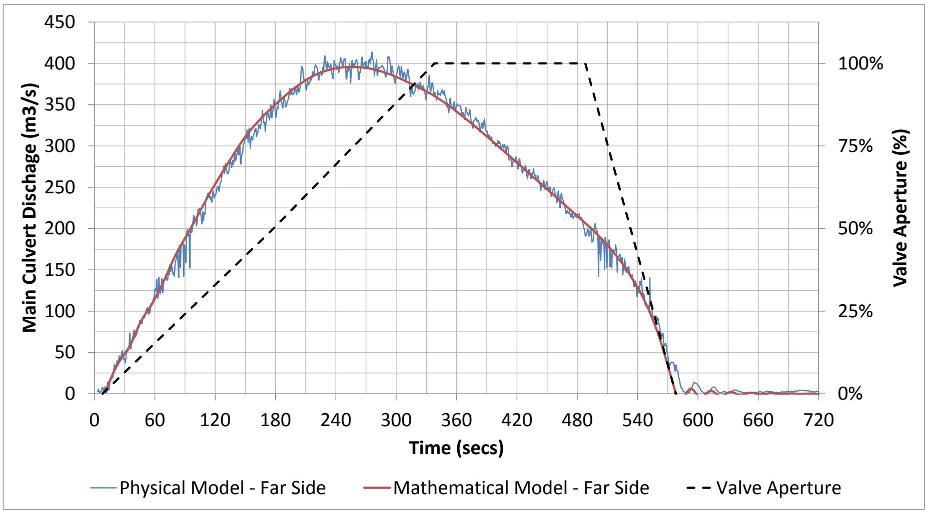

Figure 2 Comparison of physical and numerical models for the discharge in the Main Culvert.

Practical knowledge exists about the discrepancies between F/E times as measured in a physical model and those effectively occurring at the prototype. For instance, USACE manual on hydraulic design of navigation locks (2006) states that a prototype lock filling-and-emptying system is normally more efficient than predicted by its model; it explicitly suggests different empirical quantitative corrections for F/E times, depending on the size of the locks.

The alternative, rigorous strategy proposed in the present paper is to numerically simulate the flow in the prototype. This means using the physical dimensions of the prototype, the corresponding local head loss coefficients for the special hydraulic components, and the roughness height for concrete. The numerical model contemplates the variation of frictional losses with the Reynolds number. Hence, it allows to be used in order to extrapolate the physical model results to those expected for the prototype, overcoming the distortion introduced by scale effects in the physical model results.

For the Panama Canal Third Set of Lock, the validated 1D model was scaled up to prototype dimensions. Variations in local head loss coefficients, indicated by 3D models, were also introduced, though they led to relatively little effects. On the contrary, friction losses decreased significantly. For a typical Lock to Lock operation with maximum initial head difference, F/E times showed a 10% decrease (61 seconds) relative to the physical model (Figure 3).

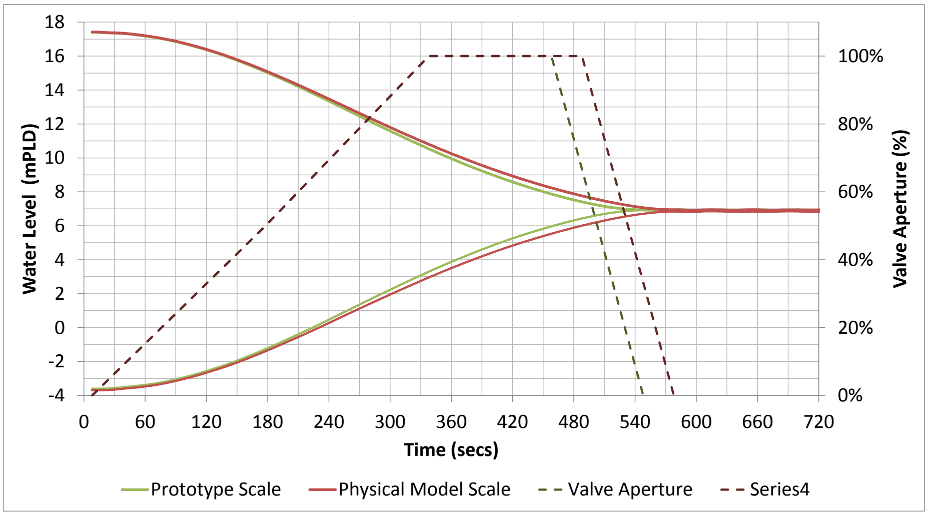

Figure 3 Comparison of physical model scale and prototype scale numerical models for water levels in the chambers.

3 FREE SURFACE OSCILLATIONS

Free surface oscillations in the lock chambers leads to forces in the hawsers. Based on results from the physical model constructed during the development of the conceptual design, a correlation was found between these forces and the free surface slope in the absence of the vessel. Hence, the free surface slope was used as an indicator for the hawser forces. As a design restriction, a maximum value of 0.14 ‰ was contractually established for the longitudinal water surface slope.

Free surface oscillations in the lock chambers are triggered by asymmetries both in the flow distribution among ports, and in the geometry of the chambers. A 2D (vertically averaged) hydrodynamic model, based on code HIDROBID II developed at INA (Menéndez, 1990), was used to simulate the surface waves. It was driven by the inflow from the ports, specified as boundary conditions through time series for each one of them, that were obtained with the 1D model described in Section 2. Figure 4 shows the comparison between the calculated longitudinal free surface slope and the recorded one at the physical model, for a case with a relatively low initial head difference between the Lower Chamber and the Ocean. The agreement is considered as very good, taking into account that the numerical model does not include the resolution of turbulent scales (which introduce a smaller-amplitude, higher-frequency oscillation riding on the basic oscillation).

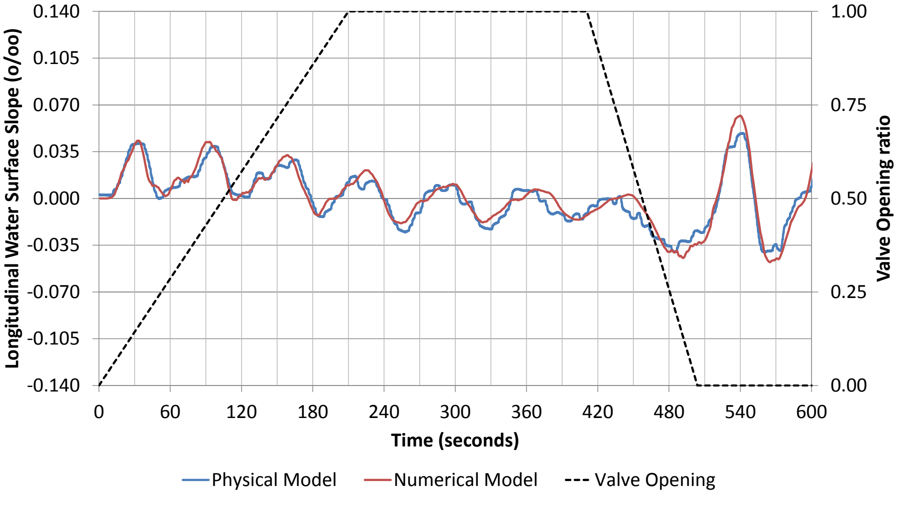

Figure 4 Longitudinal water surface slope using 1D model input. Low initial head difference.

However, the 2D model completely fails to correctly predict the longitudinal free surface slope for higher initial head differences, as observed in Figure 5 for a Lock to Lock operation with an initial head difference of 21 m. More specifically, the recorded oscillation indicates a quite more irregular response, with much higher amplitude than the one calculated with the numerical model. This indicates that turbulence scales are exerting a significant influence, so a more elaborated theoretical approach is needed. Hence, 3D modeling of the combination Central Connection + Secondary Culvert + Ports + Lock Chamber was undertaken using a Large-Eddy Simulation (LES) approach (Sagaut, 2001).

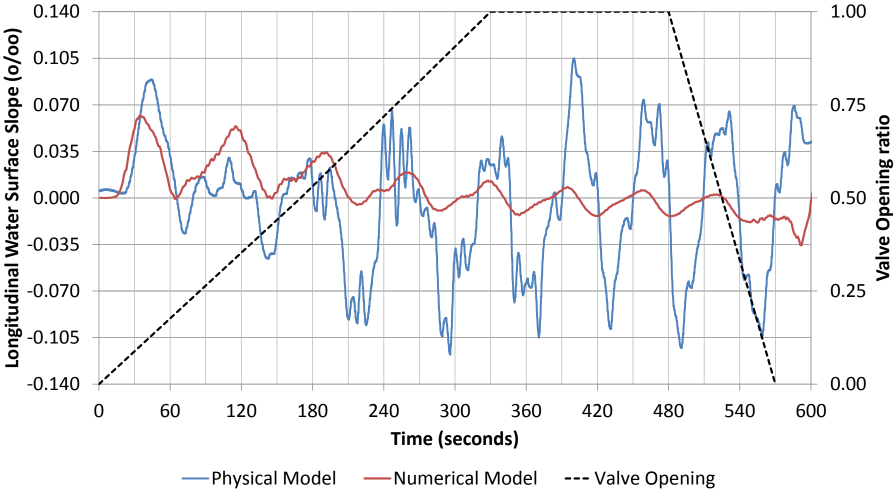

Figure 5 Longitudinal water surface slope using 1D model input. High initial head difference.

As sub-grid scale (SGS) model for the LES approach, a sub-grid kinetic energy equation eddy viscosity model was used (Sagaut, 2001). Deardorff's method was selected to define the filter cutoff length (Sagaut, 2001). A wall model was considered to treat the boundary conditions at solid borders; Spalding law-of-the-wall was selected for the velocity (White, 1974), while a zero normal gradient condition was taken for the remaining variables. At the inflow boundary, in addition to the ensemble-averaged velocity (which arises from the 1D model), the amplitude of the stochastic components were provided (Sagaut, 2001): 4% for the longitudinal component, and 1.3% for the transversal one, values associated to a fully developed flow, very appropriate for the present problem; additionally, a weighted average of the previous and present generated stochastic components was imposed in order to add some temporal correlation; for the turbulent kinetic energy, a zero normal gradient was taken. For the free surface at the Chamber, the rigid-lid approximation was used, where uniform pressure was imposed, together with zero normal gradient conditions for the remaining quantities. The model was implemented using OpenFOAM (Open Field Operation And Manipulation), an open source toolbox for the development of customizable numerical solvers and utilities for the solution of continuum mechanics problems (Weller et al., 1998). The model solves the integral form of the conservation equations using a finite volume, cell centered approach in the spirit of Rhie and Chow (1983). PISO (Pressure Implicit with Splitting of Operators) algorithm is used for time marching (Ferziger & Peric, 2001).

The computational mesh was composed by 1.5 million elements. Special considerations were made for the mesh near the wall, as the center of the first cell has to lie within a distance range to the wall to rigurously apply the logaritmic velocity profile as boundary condition (Sagaut, 2001). Typical computing times for stabilization with a steady discharge, in a Core i7 PC running 8 parallel processes, were 3 to 8 days. When complete hydroghaphs were simulated (of approximately 550 secs), 15 to 30 days of computing time were required. By parallelizing the simulation using more than one PC, computing times were reduced, though non-linearly.

Note that the rigid-lid approximation implies that the free surface oscillations are not solved by the 3D model; this was done in order to avoid extremely high computing times. Instead, the 3D model provided the time series of the flow discharge for each port, which were used to drive the 2D model of the chamber.

Figure 6 shows the longitudinal water surface slope obtained with the numerical model and its comparison with the results from the physical model, for the high initial head difference case. It is observed that the numerical simulations is now able to capture the high amplitude oscillations, indicating that large eddies must be responsible for this amplification phenomenon.

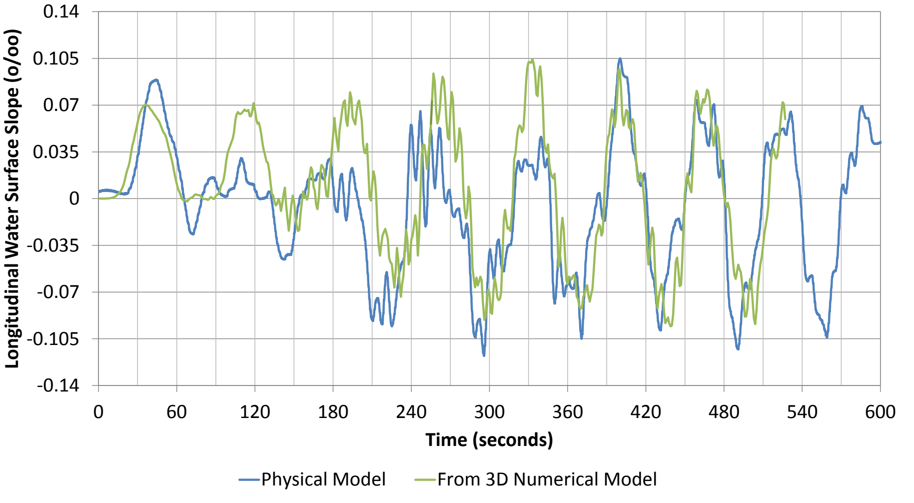

Figure 6 Longitudinal water surface slope using 3D-LES model input. High initial head difference.

A close analysis of the problem indicate that the discharges through the U and S branches of the CC present oscillations coherently out-of-phase. The dominant period of oscillation spans from 40 to 80 seconds, a range which includes the period of free surface oscillations in the Chamber (around 70 seconds), indicating that conditions close to resonance are achieved, thus resulting in an amplification of the free surface oscillation, which is the observed effect on the water surface slope. This dominant period is associated to the largest, energy-containing eddies (the ones resolved with the LES approach). More details are presented in Menendez and Badano (2012).

Next, the flow in the prototype was simulated in order to determine the behavior at that scale. Figure 7 shows the evolution of the longitudinal water surface slope arising from the results of the 3D model. It is compared with the numerical results for the physical model; the ones arising from the 1D modeling approach (no 3D LES model) are also represented, as a reference. Note that the prototype response is significantly noisier than the physical model response, as it includes a higher range of turbulent frequencies. It is observed that, though the amplification effect manifest in the prototype (the amplitude of oscillation is higher than the one predicted by the 1D model), its amplitude is definitely smaller than the one for the physical model. It is speculated that this should be due to differences in the energy spectrum: the larger eddies of the prototype would contain less energy than the corresponding ones in the physical model. It is concluded that, for this problem, scale effects tend to increase the amplification effects.

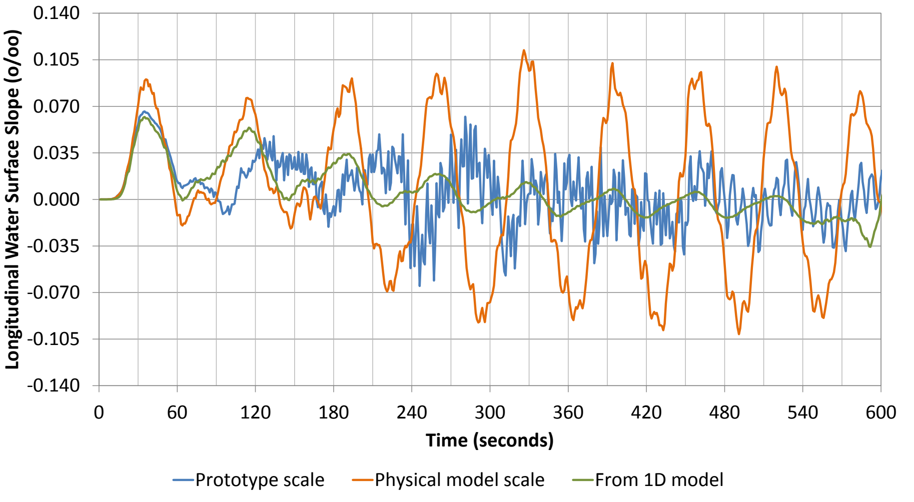

Figure 7 Prototype response. Longitudinal water surface slope.

4 CONCLUSIONS

The proposed strategy for the design of hydraulic structures, consisting in a first stage where the flow in the physical model is numerically simulated in order to validate the numerical model, and in a second stage where the flow in the prototype is numerically simulated in order to extrapolate the results to this scale, has been shown to be effective in correcting for the scale effects present in the physical model.

This has been illustrated for the particular case of the design of the Third Set of Locks of the Panama Canal, for two problems with quite different levels of complexity. The first problem was the determination of the time for water level equalization between chambers, using a one-dimensional numerical model. Friction losses are shown to be over-represented in the physical model, leading to larger equalization times. Differences of the order of 10% are calculated for a case with maximum initial head difference.

The second problem was the calculation of the amplitude of free surface oscillations in the lock chambers, due to close-to-resonance conditions, under interaction with an oscillation in a flow partition component of the filling/emptying system, using a full three-dimensional numerical model with a LES approach. Differences in the energy spectrum lead to a significant amplification of the amplitude of oscillation in the physical model.

ACKNOWLEDGEMENT

The hydraulic team for the Panamá Canal Project worked under the supervision of Nicolás Badano (MWH). The smooth interaction with the responsible for the physical model, Sébastien Roux, from CNR, was fundamental in order to achieve the goals of the study.

References

Ferziger, J. H., and Peric, M., 2001. Computational Methods for Fluid Dynamics. Springer-Verlag, 3rd Ed., 2001. ISBN 3-540-42074-6, New York, USA.

Menéndez, A. N., 1990, Sistema HIDROBID II para simular corrientes en cuencos. Revista internacional de métodos numéricos para cálculo y diseño en ingeniería, Vol. 6, 1, pp 25-36, ISSN 0213-1315.

Menéndez, A.N., and Badano, N.D., 2011, Interaction between Hydraulic and Numerical Models for the Design of Hydraulic Structures. In "Hydrodynamics – Optimizing Methods and Tools", H.E. Schulz, A.L. Andrade Simoes, R. Jahara Lobosco (Eds.), InTech.

Rhie, C.M., and Chow, W.L., 1983, A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. AIAA Journal, 21, 1525-1532.

Sagaut, P., 2001, Large Eddy Simulation for Incompressible Flows, Springer-Verlag, ISBN 3-540-67890-5, New York, USA.

USACE, 2006, Engineering and Design - Hydraulic Design of Navigation Locks. United States Army Corps of Engineers EM 1110-2-1604.

Weller, H.G., Tabor, G., Jasak, H., and Fureby, C., 1998. A tensorial approach to computational continuum mechanics using object orientated techniques. Computers in Physics, 12(6):620 - 631, 1998.

White, F.M., 1974, Viscous Fluid Flow, McGraw-Hill, ISBN 0-07-069710-8, USA.