CFD FOR HYDRAULIC DESIGN OF THE PANAMA CANAL THIRD SET OF LOCKS

Angel N. Menéndez[a,b], Nicolás D. Badano[a,b], Emilio A. Lecertúa[a,b], Martín Sabarots Gerbec [a,b], Fernando Re[b] and Mariano Re[a,b]

[a]*Laboratorio de Hidráulica, INA, Aut. Ezeiza-Cañuelas Tramo J.Newbery km 1,620, Ezeiza, Prov. Buenos Aires, Argentina, angel.menendez@speedy.com.ar, http://laboratorios.fi.uba.ar/lmm*

[b]*Laboratorio de Modelación Matemática (LaMM), Facultad de Ingeniería, Universidad de Buenos Aires, Av. Las Heras 2214, C1127AAR Buenos Aires, Argentina, anmenendez@gmail.com*

Keywords: Hydraulic design, scale effects, OpenFOAM, turbulence modeling.

Abstract. Within a general modeling framework for the hydraulic design of the filling/emptying system of the Third Set of Locks of Panamá Canal, the application of CFD, via OpenFOAM, is presented and discussed. Strategies for mesh generation and adoption of convergence criteria are explained. Validation tests, based on comparisons with experimental data, are described. The new role of CFD, providing the preliminary hydraulic design of special components, to be tested in a physical model, is stressed. The ability of numerical model to explain scale effects in physical models is clearly shown, leading to a new paradigm of extrapolation to prototype from the numerical model. The performance of OpenFOAM, as an efficient and accurate tool for hydraulic design with CFD, is demonstrated.

1 INTRODUCTION

The process of expansion of the Panama Canal is under way. A third set of locks will be added to the existing two, at each extreme (Pacific and Atlantic) of the Canal, to allow the passage of Post-Panamax vessels (Figure 1). The filling/emptying (F/E) system for the third set of locks constitutes the key element to maximize the rate of vessel passage, while minimizing the freshwater loss to the oceans.

Figure 1: Project for Third Set of Locks

CICP ('Consorcio Consultores Internacionales del Canal de Panamá'), in charge of the engineering studies for GUPC ('Consorcio Grupo Unidos por el Canal'), asked INA to build a modeling system to help in the hydraulic design of the F/E system. The modeling system includes: (i) a 1D (section averaged) model of the whole F/E system, to determine operation times, flow velocity at the conduits, rate of change of the water level at the lock chambers, water pressures, and operation policy for the valves; (ii) a 2D (vertically integrated) model of the lock chambers, fed by 1D model results, to verify the longitudinal and transversal water surface slopes, indicative of the hawser forces; (iii) a 0D (volume integrated) model of the lock system, with some inputs defined by the 1D model results, to determine the vessel throughput, the water consumption, and the maximum operative water level differences (which influence the structural design); (iv) 3D models of the F/E system components, to determine the 'local' head losses (i.e., the energy transfer towards eddies and secondary currents) to be used in the 1D model. The 3D models truly constitute CFD applied to hydraulic design.

In this paper, besides general references to the modeling system, the 3D model approach is presented and discussed.

2 MODELING SYSTEM

Hydraulic problems deal with quite different spatial (and temporal) scales. Then, they are not 'simply' Fluids Dynamics problems which can be treated with CFD tools. An efficient approach, compatible with engineering deadlines, requires including only the most relevant spatial scales, and the associated physical mechanisms, when modeling each problem. Links among the different models are established through driving forces and boundary conditions.

The modeling system developed for the present study is schematized in Figure 2. The core of the modeling system is a 1D hydrodynamic model of the F/E hydraulic system. It is based on commercial software Flowmaster V7 (*http://*www.flowmaster.com/). The estimation of the values for the head loss coefficients, which parameterize the local energy losses at the different hydraulic components, is made using data from bibliography, from a preliminary physical model, or from 3D models of those components.

The 1D model provides, in particular, the hydrographs at the ports, through which water is fed to or extracted from the lock chambers. They are used as boundary conditions for the 2D model of the chambers, in order to simulate the resulting oscillation of the free surface. The 2D model is based on numerical code HIDROBID, developed at INA (Menéndez, 1990).

Software ESCLUSA was specially developed to build the 0D model. It uses lockage times provided by the 1D model, in addition to other non-hydraulic operation times.

The 3D models of the system components are described in the next section.

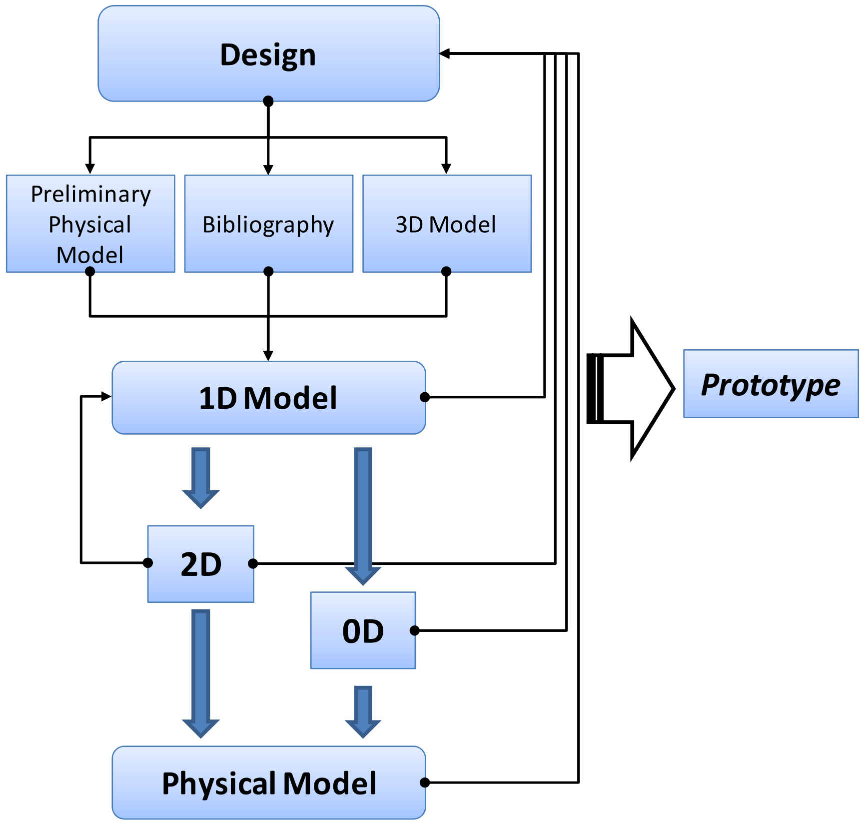
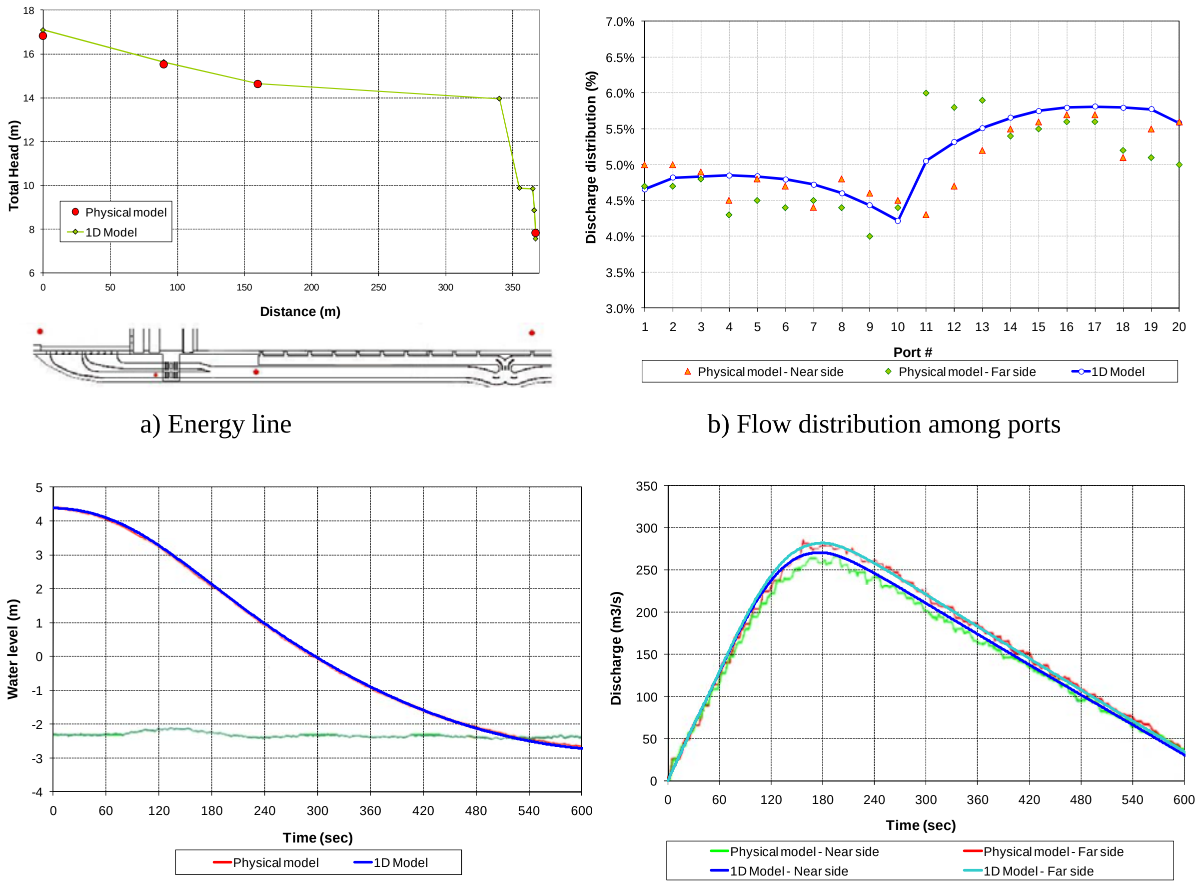


Figure 2: Modeling system

The 1D and 2D were validated by comparing its predictions with experimental data obtained at a specially built preliminary physical model (by CNR – Compagnie Nationale du Rhône – Lyon, France). The 1D model correctly represented the energy line along the different stretches of the F/E system, the flow distribution among ports, the water level evolution at the chambers, and the discharge evolution at the culverts, as illustrated in Figure 3. The 2D model was successful in reproducing longitudinal water slopes, as illustrated in Figure 4. No experimental validation was possible for the 0D model, but its performance was fully consistent with previous models developed for the original project design.



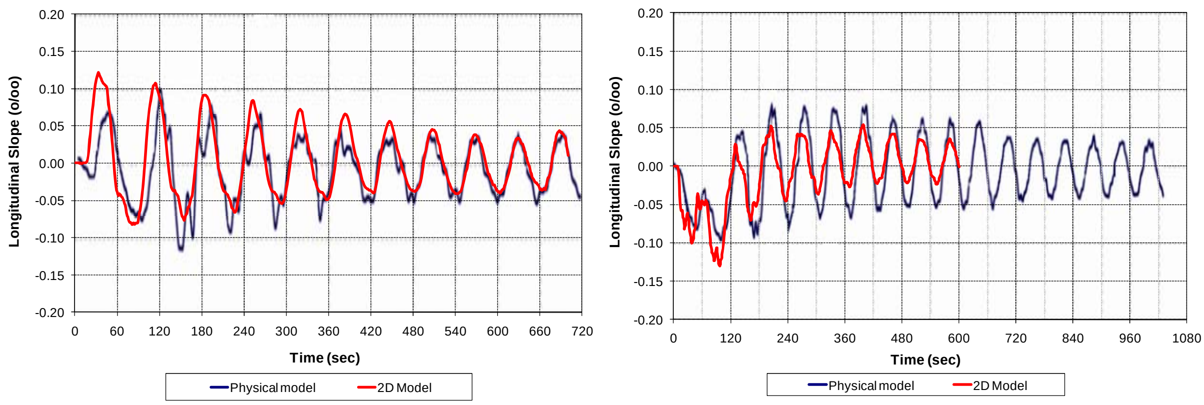
a) Energy line

b) Flow distribution among ports

c) Water level at chamber

d) Total discharge through the culvert

Figure 3: Validation of 1D model



a) Standard operation

b) Non-standard operation

Figure 4: Validation of 2D model: longitudinal water slope at Chamber, for Lock to Lock operation

3 CFD FOR HYDRAULIC DESIGN

Historically, under the limitations of computing capacity, numerical modeling in Hydraulics concentrated in relatively large spatial scales, based on 1D or, later, 2D formulations, i.e., mathematical models in which one or two physical dimensions are not explicitly solved, but accounted for only through their integrated effects. 3D models at geophysical scales, which include as a key simplification the hydrostatic approximation, were later incorporated. Truly 3D short-scale phenomena were studied on ‘physical models’, i.e., experimentally on appropriately scaled physical representations. Presently, CFD is starting to be used as a tool to study this type of 3D problems.

3.1 Governing equations

The problems to be solved are essentially of the ‘internal flow’ type, i.e., flows constrained within ducts, with no free surface, except for a few cases. Additionally, they are considered to be in fully turbulent conditions (very high Reynolds number).

The governing equations correspond to a RANS (Reynolds Averaged Navier-Stokes) model. LES (Large Eddy Simulation) approaches are still considered to be too costly in computer time to be used for hydraulic engineering studies. The Realizable k-ε model (Shih et al., 1995) was chosen for turbulence representation.

Wall functions were used as boundary conditions near solid walls, avoiding the necessity of solving the problem down to the wall with a very fine near-wall mesh and a low-Reynolds number model, thus considerably reducing the computer time.

3.2 Numerical code

OpenFOAM® (Open Field Operation and Manipulation) CFD Toolbox, Version 1.7 was chosen as the numerical code for modeling (http://www.opencfd.co.uk/index.html).

OpenFOAM solves systems of partial differential equations using the Finite Volume Method, on 3D unstructured meshes of polyhedral cells. The fluid flow solvers are developed within a solution framework of the implicit, pressure-velocity, iterative type. Domain decomposition parallelism is integrated at a low level.

The chosen discretizations were linear for gradient, upwind for divergence, and linear corrected for Laplacian. The SIMPLE algorithm was used for the resolution of the numerical system.

When using wall functions, the first near-wall grid node must lie within the fully turbulent inner region ($30 < y^+ < 300$), where the logarithmic law for the velocity profile holds, in order to provide a correct account of frictional energy losses.

The convergence criterion (for iterations within a time step, and for stabilization towards steady state conditions) is based on relative residuals of the primary variables (velocity components, pressure, k, and ε), and defined as the RMS value of the difference between successive solutions on the grid, normalized by the average value of the variable throughout the domain. Within each step of the SIMPLE algorithm, the tolerance for convergence was fixed at 10^{-4}, except for k and ε, for which it was reduced to 10^{-8}. This condition was usually met after several thousands of SIMPLE iterations. For problems involving a relatively wide range of flow velocities within the domain, the resulting flow structure was checked through visualization in order to guarantee that stabilization was also achieved at the slower flow zones.

OpenFOAM runs under LINUX operative system. Parallel computing was used via domain decomposition, using the MPI protocol for exchanging information between parallel threads. Up to 8 threads were run in single i7 Quad Core Processors, depending on the complexity of the model. For especially big models, calculations were made in parallel using several i7 PCs, connected over a Local Area Network. Due to the efficiency of the domain decomposition approach in this kind of problems, the overhead of parallelizing calculations at this scale was found out to be practically negligible.

3.3 Mesh generation

Mesh generation was undertaken with Gmsh (Generic Mesh). This is an automatic 3D Finite Element grid generator with a built-in CAD engine and post-processor (http://geuz.org/gmsh/). Unstructured tetrahedral meshes were used.

In order to correctly represent the boundary layer adjacent to solid surfaces, and so fulfill the restriction for the first near-wall grid nodes to lie within the fully turbulent inner region, code enGrid was selected (http://www.ohloh.net/p/engrid) for generating thin elements adjacent to solid walls. This open source mesh-generation software, especially developed with CFD applications in mind, regenerates the original tetrahedral mesh adding layers of prismatic element near solid boundaries.

Typical model were discretized using between 1 and 2 million elements. Up to 10 million elements were used for special cases. The computer time required for the stabilization of steady state runs of typical problems, with a cold start, was between 12 and 36 hours using 8 parallel threads.

Based on a priori analyses, and sometimes through successive trials, adapted meshes were built, i.e., higher resolution was ascribed to flow zones with higher gradients, in order to attain maximum precision for the adopted number of elements.

4 VALIDATION OF CFD CODE

Prior to applying the numerical code, tests were undertaken in order to analyze its performance for problems similar to those solved, as a validation procedure. The validation criterion consisted in comparing its results with experimental data.

4.1 Pipe elbow

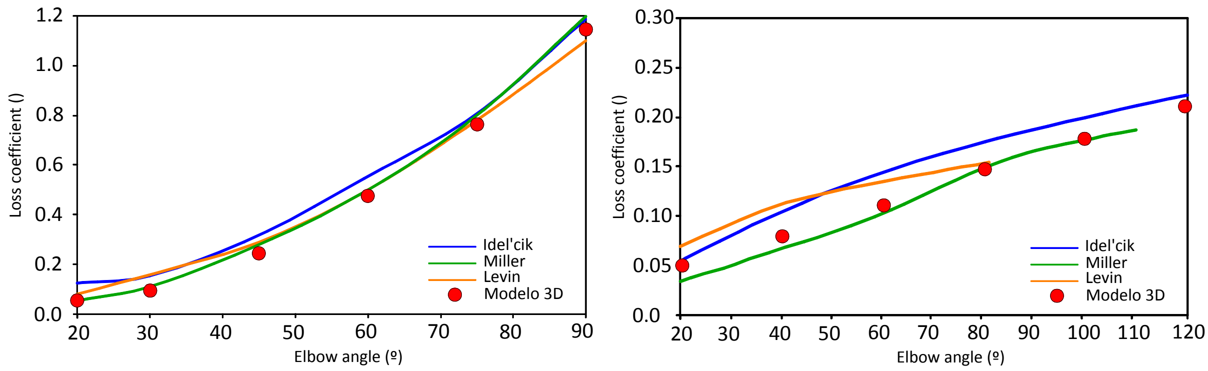
Two cases were considered: a sharp elbow, i.e., one for which there is a sudden change in the flow direction, and a smooth elbow, i.e., one for which the direction changes smoothly. They were modeled for a wide range of elbow angles. The last test is more demanding than the former one, as the location of the flow separation point is not determined a priori, and changes with the elbow angle.

A square cross section (2 m x 2 m) pipe was taken. The pipe material was considered to be concrete, with an effective roughness height of 0.25 mm, which behaves as hydraulically smooth. The discharge was 16 m^3/s. The radius of curvature of the pipe axis for the smooth elbow was 3 m.

The head loss coefficient, K, was obtained from the difference between section-averaged total mechanical energy at the ends of streamtubes. The inlet and outlet sections of the streamtubes are chosen far enough from the model boundaries, in order to minimize the influence of the imposed (simplified) boundary conditions. These section averages were obtained using ParaView (http://www.paraview.org/), an open-source, multi-platform data analysis and visualization application.

Experimental results were taken from Idel'cik (1979), Levine (1968), and Miller (1971). The results correspond to diverse experimental conditions that are not reported in detail. Moreover, they are presented as continuous curves as a function of the elbow angle, which arise from some fitting procedure, with no information about an uncertainty range. Hence, the differences among the three different curves were considered to give an indication of that uncertainty range. The validation criterion for the model consisted in reproducing the experimental trends, with a deviation similar to the so-defined uncertainty range.

Figure 5 presents the comparison between the numerical head loss coefficient (once frictional losses were subtracted) for the different elbow angles, and the experimental curves. Note that the uncertainty range is of the order of 0.05 throughout the elbow angle range for the two cases. It can be seen that the model results lie within this uncertainty range.



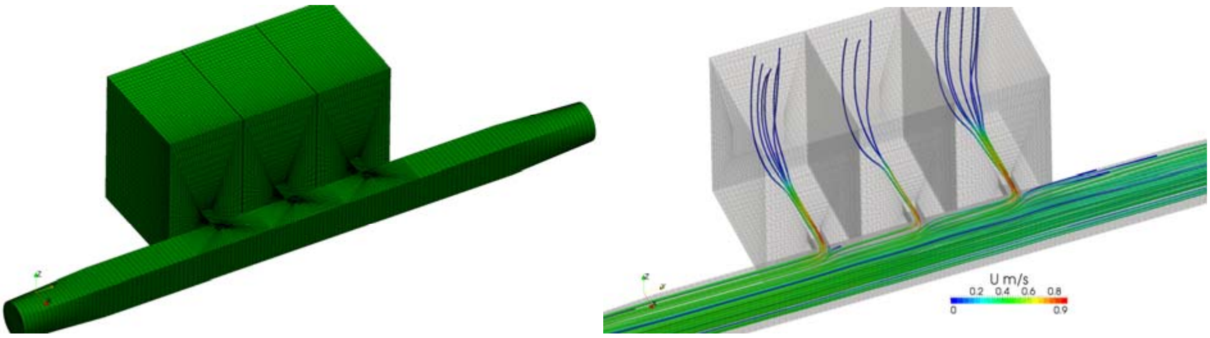
a) Sharp elbow b) Smooth elbow

Figure 5: Validation of 3D model with elbow problem

4.2 Ports system

Experimental results from a physical model for a port system were available (CPP, 2007). The system consisted in a conduit, three ports located at the ceiling of the conduit, and three chambers receiving the discharge. Figure 6a) shows the mesh built.

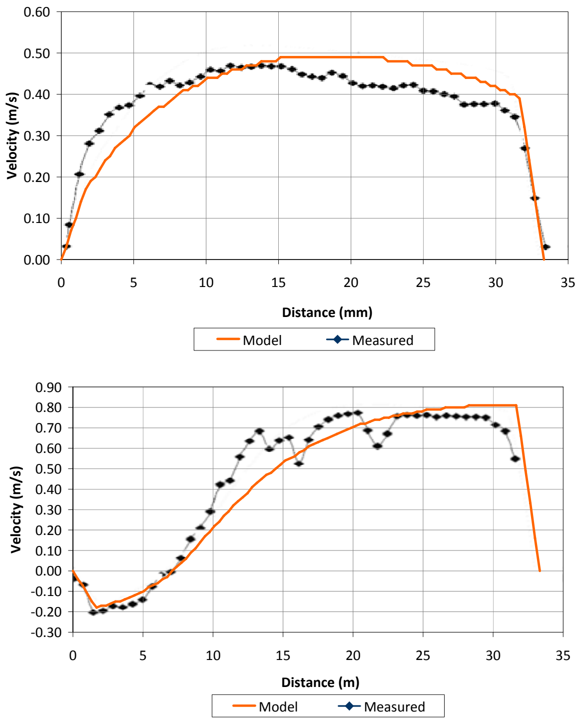
As boundary conditions, the inlet and outlet flow rates at the upstream and downstream sections of the conduit, respectively, were imposed. Uniform pressure (corresponding to the respective measured water height) was set at the top of each chamber, together with a restriction to avoid inflow, in order to simulate what in practice is a side discharge through a spillway. Figure 6b) presents the streamlines pattern.



a) Mesh b) 3D view of streamlines

Figure 6: Validation of 3D model with port system problem

Velocity measurements at two ports cross sections, obtained using PIV (Particle Image Velocimetry), were available. Figure 7 shows the comparison between calculated and measured velocity profiles for one of the two reported cases. The agreement is considered quite satisfactory.



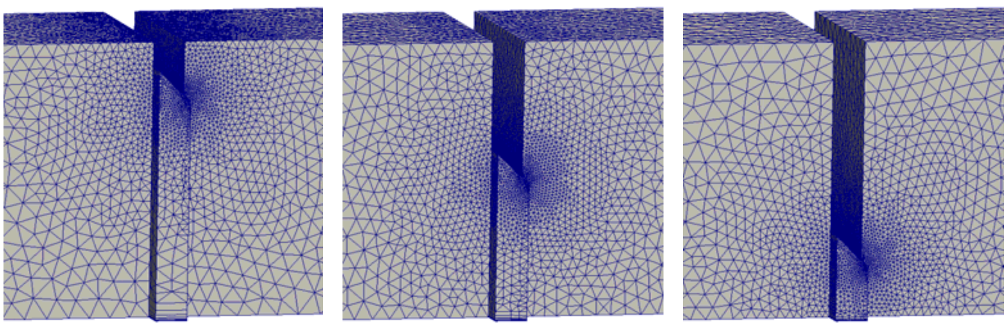
a) Inlet cross section

b) Cross section 20 mm downstream from inlet

Figure 7: Velocity profiles at mid-plane of ports.

4.3 Sluice gate

The energy loss through a sluice gate, which acts as a valve for a square conduit, was modeled for three different apertures: 80%, 50%, and 20%. Figure 8 shows details of the mesh around the gate. Figure 9 presents the streamlines and velocity distribution in the vertical middle plane for the three cases.

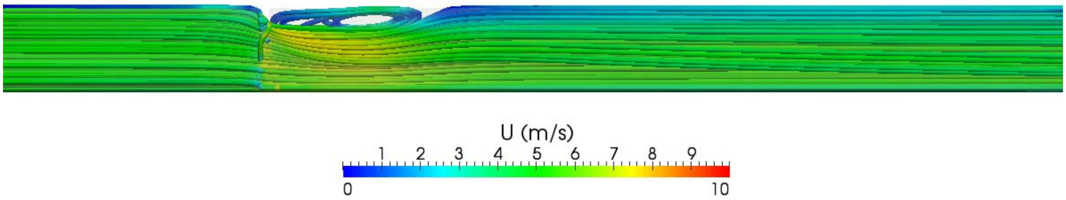


a) 80%

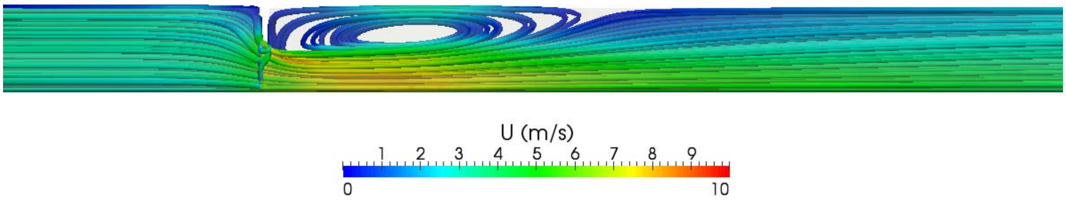
b) 50%

c) 20%

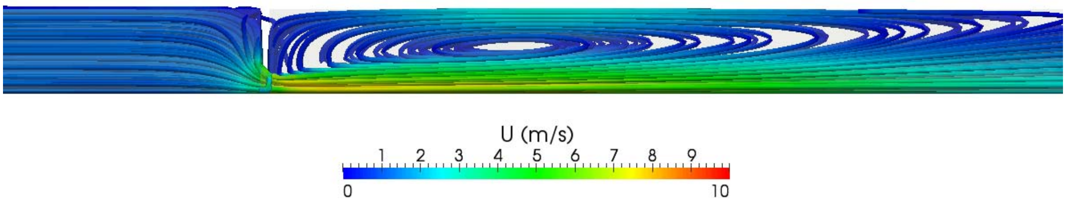
Figure 8: Detail of mesh for the different valve apertures



a) 80%



b) 50%



c) 20%

Figure 9: Streamlines and velocity for the different valve apertures

The head loss coefficients, obtained from the 3D models, are compared with those provided by Idel'cik (1979), Levine (1968), and in Flowmaster in Figure 10. It is observed that the agreement is very satisfactory.

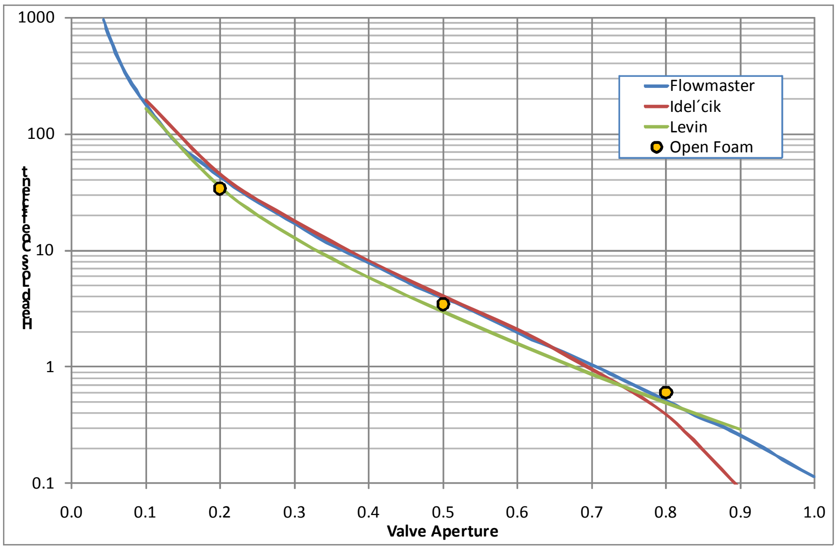


Figure 10: Streamlines and velocity for the different valve apertures

5 APPLICATION OF CFD CODE

5.1 Conceptual framework

Hydraulic studies for lock systems are historically performed based on physical models. One of the main contributions of CFD to the present hydraulic studies was to undertake, through numerical simulation, the design of project alternatives for some of the system hydraulic components, to be tested at the physical model.

It is well known, though, that physical models are subject to scale effects, i.e., their response deviates from the one in the prototype due to the fact that some effects – essentially those associated to energy dissipation – do not scale properly (the Reynolds numbers are different). In particular, for lock systems, common practice indicates that "…A prototype lock filling-and-emptying system is normally more efficient than predicted by its [physical] model. The difference in efficiency…can be accommodated empirically…(filling time and overtravel, specifically)" (USACE, 2006).

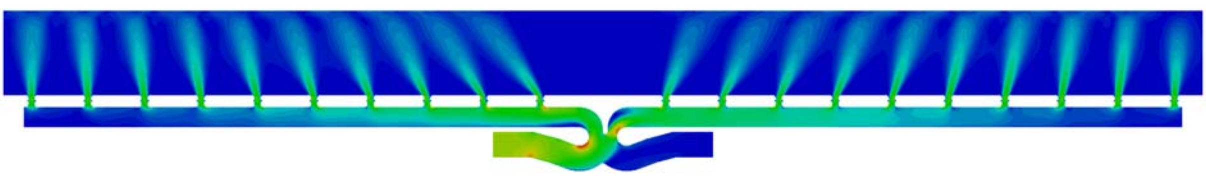
The numerical models do not have the same type of limitations. The deviations of their results, relative to prototype, can be attributed to two main causes: (i) insufficient resolution; and (ii) limitations of the turbulence models. Theoretically, the first limitation could be overcome by reducing the step of the numerical grid, while the second one by recurring to LES (Large Eddy Simulation) or, eventually, DNS (Direct Numerical Simulation). In practice, the present available computer processing capacity puts a limit to these possible solutions.

Based on these observations, the strategy developed in the present study was to numerically simulate also the flow in the physical model, i.e., to use the dimensions of the physical model in order to show that the so obtained Numerical Scaled Model can, in fact, explain most of the scale effects. Being that the case, the adequate way of extrapolating the results to the prototype should be through the numerical model of the prototype. This second contribution of CFD to hydraulic studies constitutes a new paradigm, still to be accepted by the hydraulics community.

5.2 Flow distribution in manifold

As an illustration of the above mentioned approach, the problem of the 'Central Connection' – i.e., a specially designed device that splits the flow coming through the main conduit of the F/E system into the two secondary conduits, where the ports are located – is described.

The Central Connection design studied at the physical model was obtained through CFD. The model domain included, in addition to the Central Connection itself, a stretch of primary conduit, the two secondary conduits, the ports, and half the chamber (till the plane of symmetry). Figure 11 shows the velocity distribution (in prototype units) on a horizontal plane (located at half the ports height) for the whole model domain, while Figure 12 presents a 3D view of the streamlines in the Central Connection. The numerical model indicated that this design provides equal partition of the flow between the two branches.



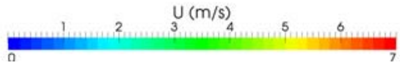


Figure 11: Plan view of velocity distribution for Central Connection

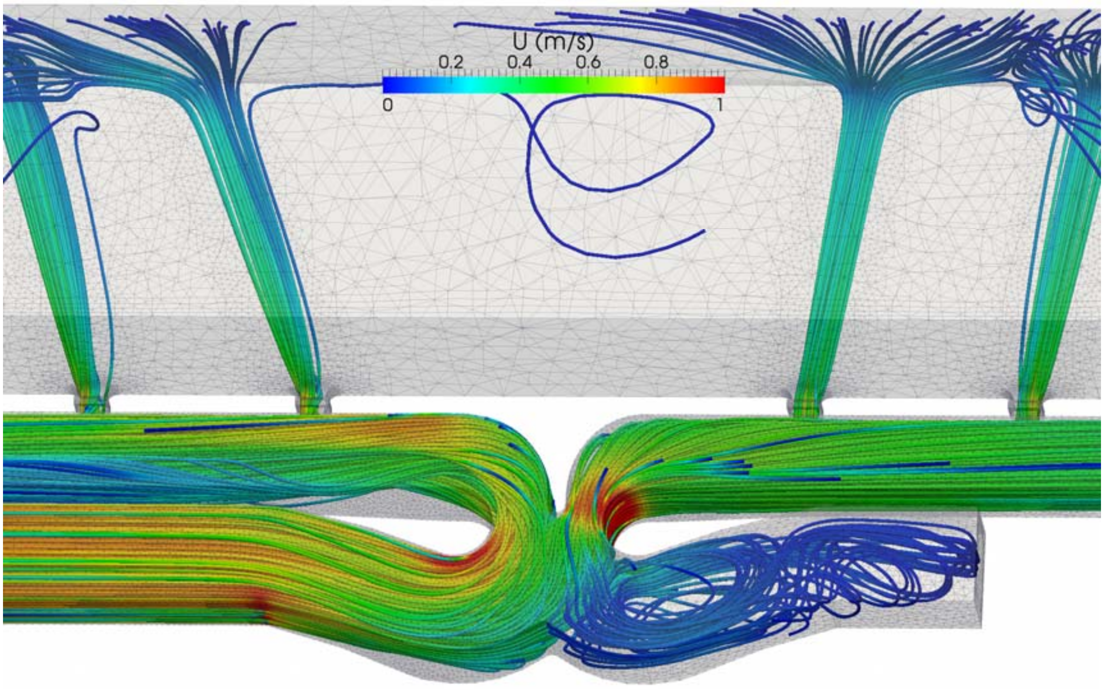


Figure 12: Streamlines for Central Connection

During the first set of physical model tests, besides some accuracy limitations for discharge measurements (made with a propeller), a clear asymmetry was detected in the flow distribution, with about 55% of the discharge proceeding through the ‘S’ branch (rightwards in the figure). This bias can lead to increase hawser forces for the ships standing in the chamber during lockage. The Numerical Scaled Model demonstrated that the physical dimensions were partly responsible for that trend (scale effect), indicating that about 52% of the discharge should proceed through the S branch. The remaining bias was tracked down to wall roughness: though the physical model was basically built in Plexiglass, the curved forms of the Central Connection were made out of Styrofoam (providing an efficient way for eventually introducing changes in order to test design alternatives), behaving as hydraulically rough instead of hydraulically smooth as the prototype, a second scale effect. By smoothing out the Styrofoam surface, through a coating, two series of experiments (PM-1 and PM-2) indicated 52.4% and 51.6% of the flow proceeding through the S branch, in close agreement with the Numerical Scaled Model (NSM). This is shown in Figure 13, where the result according to the Numerical Prototype Model (NPM) is also presented. Figure 14 shows the comparison of flow distribution according to the smoothed PM and the Numerical Scaled Model. The measurement at Port #11 was shown to be grossly underestimated due to the high flow concentration close to the outer port boundary, as illustrated in Figure 15.

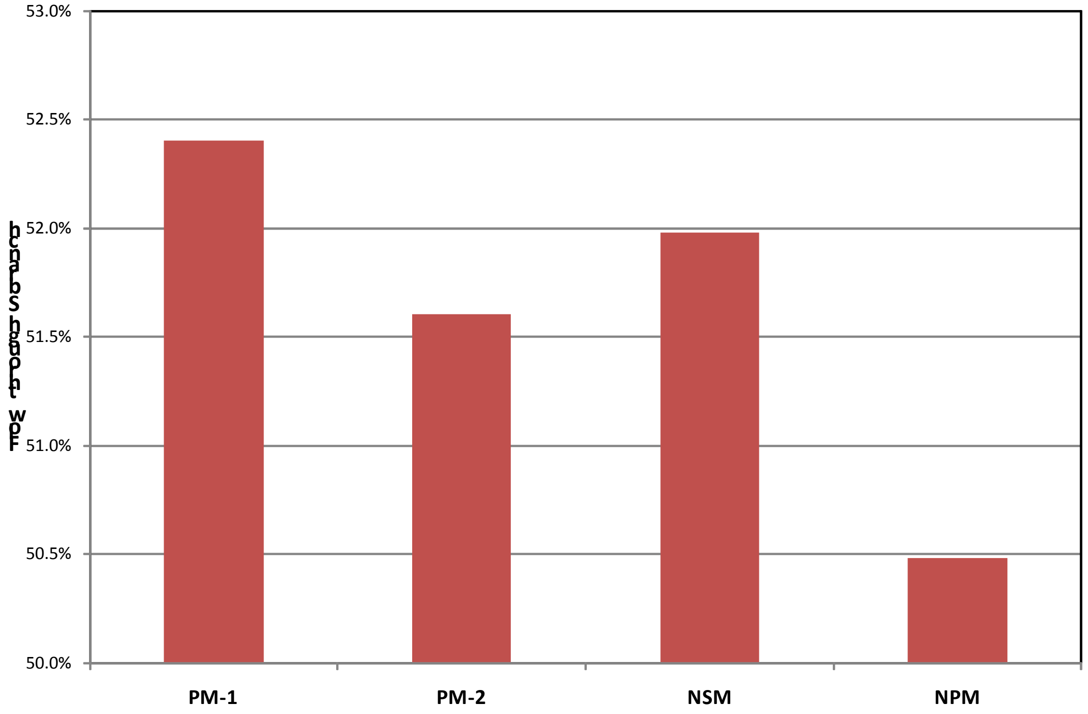


Figure 13: Flow through the S branch of the Central Connection

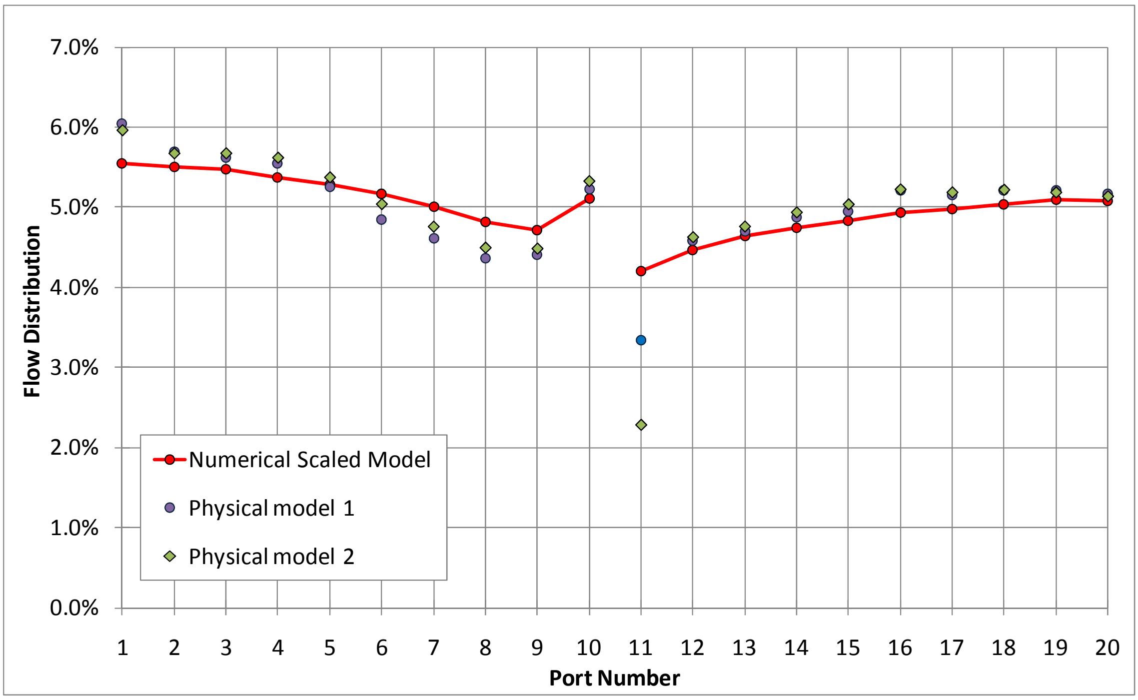


Figure 14: Flow distribution among ports according to physical model and Numerical Scaled Model

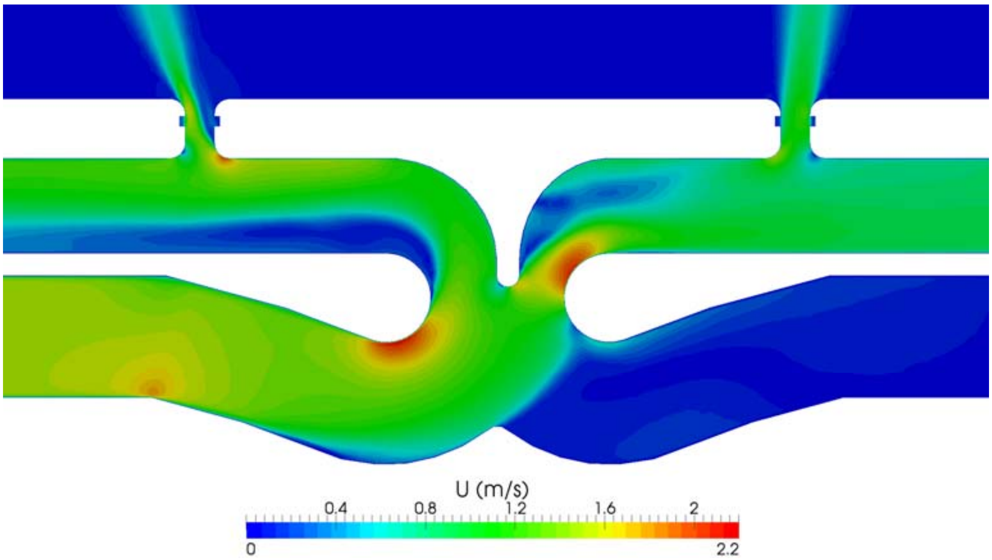


Figure 15: Velocity distribution according to Numerical Scaled Model

6 CONCLUSIONS

From the modeling experience with CFD for the Panamá Canal hydraulic project, the following main conclusions arise:

- Hydraulic design of special components which operation leads to flow separation and/or secondary currents, historically performed through physical modeling, is starting to be tackled with CFD, which provides the preliminary design to be tested.
- Scale effects – mainly related to improper representation of energy dissipation –, a major limitation for physical models, can be explained by numerical modeling at the physical scale (Numerical Scaled Model). A new paradigm is then emerging: the extrapolation to prototype should be made from the prototype numerical model.
- These two functions of CFD is triggering a new era of complementation between physical and numerical model in hydraulic design.
- OpenFOAM constitutes an efficient and accurate tool for hydraulic design with CFD.

REFERENCES

CPP (Consorcio Post Panamax), Task 1.2.5. *Numerical model of variations for the selected F/E System*. Final Report. September, 2007.

Idel'cik, I.E., *Memento des Pertes de Charge*. Eyrolles, Paris, 1979.

Levine, L., Formulaire des conduites forces oleoducts et conduits d'aeration. Dunon, Paris, 1968.

Menéndez, A. N., Sistema HIDROBID II para simular corrientes en cuencos. *Revista internacional de métodos numéricos para cálculo y diseño en ingeniería*, 6(1):25-36, 1990.

Miller, D., *Internal Flow*. England, 1971.

Shih, T.-H., Liou, W. W., Shabbir, A. and Zhu, J., A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. *Computers Fluids*, 24(3):227-238, 1995.

USACE, Engineering and Design - Hydraulic Design of Navigation Locks. EM 1110-2-1604, May, 2006.