

Telemac-3D for ship lock hydraulics

Dr Miodrag JOVANOVIĆ

University of Belgrade, Serbia
Faculty of Civil Engineering
www.grf.bg.ac.rs/~mjovanovic
mjovanov@grf.bg.ac.rs
mjovanov@eunet.rs

10.11.2024

Abstract

One of the primary considerations in ship lock design is the selection of the lock chamber filling and emptying time. In order to make the right decision, numerical simulation of lock filling and emptying processes is necessary. Because of the interaction, superposition, and mixing of inflow jets, the flow in the lock chamber is predominantly three-dimensional. The TELEMAC-3D modeling framework is appropriate for investigation of the unsteady hydraulic regimes in ship locks. A case study is presented to demonstrate the applicability of this model. The computational results comprise several hydraulic scalar and vector fields, including unit discharge, velocity, streamlines, turbulent kinetic energy, and the rate of turbulent energy dissipation. The temporal evolution of the filling process is analyzed in terms of unit discharge and velocity fields, while streamlines and turbulence characteristics are presented for the period of maximum inflow, when the hydraulic disturbances are most pronounced. The results are supplemented by a brief discussion of the advantages and limitations of three-dimensional numerical modeling for design of ship lock hydraulic systems.

Key words: Hydraulics of ship locks, Telemac-3D

1 Introduction

The design of ship locks requires a detailed analysis of the flow regime in the lock chamber during filling and emptying. Such analyses are essential not only for optimizing the dimensions of the lock chamber and the characteristics of the hydraulic filling and emptying systems, but also for determining an appropriate duration of the locking cycle. This optimization directly affects the capacity of the lock, expressed in terms of the number of ships or the volume of cargo transported per day or year, and must therefore be consistent with the planned intensity of waterway traffic.

One of the primary design considerations is the selection of the filling and emptying time. This period must be sufficiently long to keep hawser forces below specified limits and ensure ship safety, while being sufficiently short to maximize the number of ship passages. Such analyses were traditionally performed using physical scale models, whereas numerical modeling has become the standard approach [4].

The complexity of numerical simulations of the unsteady flow depends on the geometry of the lock chamber and the configuration of the associated filling and emptying conduits. The problem can be simplified by considering an empty lock chamber, i.e. without ships, based on the assumption that this configuration produces more pronounced hydraulic disturbances and therefore provides a conservative assessment of the flow conditions [4].

Because of the interaction, superposition, and mixing of inflow jets, the flow in the lock chamber is predominantly three-dimensional. Consequently, the TELEMAC-3D modeling framework was selected to investigate the unsteady hydraulic conditions during the filling and emptying processes. A case study is presented to demonstrate the applicability of the model to ship-lock hydraulics and to assess its potential for evaluating flow conditions relevant to the selection of appropriate filling and emptying times.

2 Governing Equations

The three-dimensional flow is described by the Navier-Stokes equations, representing the mass and momentum conservation laws [3]:

$$\begin{aligned}
 \operatorname{div}(\mathbf{U}) &= 0 \\
 \frac{\partial u}{\partial t} + \mathbf{U} \cdot \mathbf{grad}(u) &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + \operatorname{div}(\nu_t \mathbf{grad}(u)) + f_x \\
 \frac{\partial v}{\partial t} + \mathbf{U} \cdot \mathbf{grad}(v) &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + \operatorname{div}(\nu_t \mathbf{grad}(v)) + f_y \\
 \frac{\partial w}{\partial t} + \mathbf{U} \cdot \mathbf{grad}(w) &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + g + \operatorname{div}(\nu_t \mathbf{grad}(w)) + f_z,
 \end{aligned} \tag{1}$$

where: t – time, (x, y, z) – space coordinates, (u, v, w) – components of velocity $\mathbf{U}$ in directions (x, y, z) , p – pressure, (f_x, f_y, f_z) – source terms, ν_t – coefficient of viscosity of turbulence, ρ – density i g – gravitational acceleration.

Impacts of friction and other factors (wind upon the surface, volume centrifugal forces, etc.) are taken into account by the source terms. System of equations (1) are coupled with "transport equations", used to model turbulence, and (when necessary) heat transfer, sediment transport, pollutant spreading, etc. [3, 8].

By numerically solving the system (1), the three-dimensional (3D) numerical models are used to define the flow field with three velocity components. In the case of ship locks, these models are useful for numerical simulation of all small and large-scale vortex movements in the lock chamber. The price is necessity of high numerical resolution of the computational domain, and significant engagement of computer resources.

2.1 Illustrative case study

Let us consider filling of a lock chamber with a frontal water-supply system, consisting of two symmetrically positioned short galleries in the side walls [4]. The chamber is 100 m long, 12 m wide, with initial depth of 4 m. During the filling process, the two inflow jets are mutually opposed, resulting in a highly turbulent, three-dimensional flow and intensive mixing in the upstream part of the chamber. This part acts as a basin for dissipating the excess kinetic energy. Let it be called the "stilling basin". The length of the basin is 10 m, and its bottom is 1 m below that of the rest of the chamber.

The task is to analyse the hydraulic regime in the lock chamber, particularly the turbulent flow in the stilling basin. The inflow hydrograph $Q(t)$, previously determined for the system of two frontal water supply galleries, is given in Table 1.

Table 1: *The inflow hidrograph* [4].

t (s)	0	63	126	189	252	315	378	441	504	567	630	693	756	819	1000
Q (m ³ /s)	0	4,5	8,8	12,6	16,0	18,2	19,3	19,3	18,1	16,4	14,3	12,1	9,9	7,7	0

All computations were performed using the calibrated Telemac-3D model, based on the finite-element method and the fractional-step numerical procedure for solving the given equations. The basic solution consists of time-variable distribution of water depth $h(t, x, y)$ and the three velocity components $u(t, x, y, z)$, $v(t, x, y, z)$, and $w(t, x, y, z)$. Additional results include characteristic turbulent flow indicators.

The computational grid. The basic, nonstructured 2D grid (in the plane) consists of triangular elements¹. The grid is well adjusted to the given geometry of the lock chamber, with density of the grid increased in the regions of interest – around the gallery outlets (Fig. 1).

The 3D grid of prismatic elements with triangular base is also shown in the same Fig. In general, any irregular, variable 3D physical domain is at each computational step transformed to a regular 3D computational domain (such as the one in Fig. 1) by a special technique known is the "sigma tranformation" [3, 8].

¹In this particular case, the geometry is of the computational domain is quite simple, so regular, structured grids of rectangles or squares could have been used, as well.

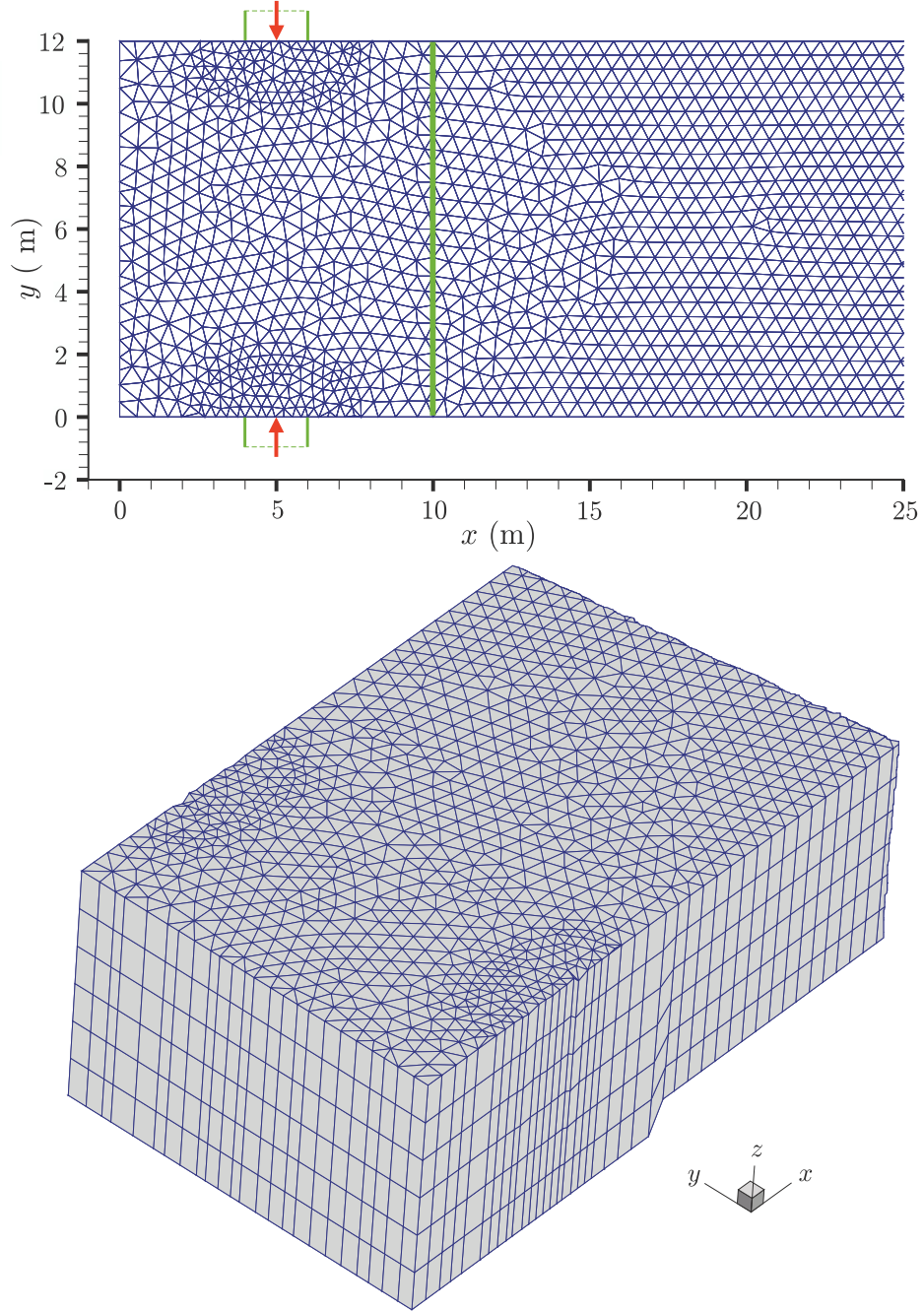


Figure 1: *The nonstructured computational grid; only the upstream part of the lock chamber is shown; above: the basic 2D grid consists of 9544 triangular elements and 4986 nodes; below: the 3D grid is formed by "copying" the basic grid in additionl 6 planes over the depth [4].*

Results displayed in a plane. As during the computation the results are memorized for each layer (plane) of the 3D domain, the *depth-averaged* values of velocities (and other variables of interest) can be easily obtained and displayed in the form of ordinary diagrams². The purpose of presenting the results of 3D numerical modeling in a plane is the necessity to reduce the amount of information, simplify relationships, and thus gain a clearer view of flow dynamics.

Evolution of the flow in the entire lock chamber is presented at several time steps in Figs. 2–3. Flow in the stilling basin is shown in Fig.4. The chosen time period is from the beginning of the filling process, until the inflow peak is reached ($Q = 19,3 \text{ m}^3/\text{s}$).

Results displayed in space. This kind of presentation is required for complex flow fields with pronounced disturbances in all three coordinate directions. In this case, this is true for the upper part of the chamber called "the stilling basin". Interaction of the jets from the galleries results in a very complex flow structure, as shown in Fig. ??.

Efficiency of the stilling basin. The main advantage of multidimensional numerical models is the possibility to quantify the turbulence characteristics, which in this case can be used to measure the efficiency of the stilling basin, and determine its optimal size.

Some basic notions from the theory of turbulence [5, 6, 7] are recalled at this point, as necessary for interpretation of the computational results.

As is well known, turbulent flows consist of a broad spectrum of vortex sizes and corresponding fluctuation frequencies. The scale of the largest vortices, which are associated with low-frequency fluctuations, is determined by the size of the flow domain, whereas the scale of the smallest vortices, associated with high-frequency fluctuations, is determined by the fluid viscosity. The large vortices extract kinetic energy from the mean flow and transfer part of this energy to progressively smaller vortices. This process, known as the turbulent energy cascade, continues until the kinetic energy reaches the smallest scales, where viscous effects ultimately dissipate it as heat.

Turbulent flows are random in character. Conceptually, the local instantaneous velocity can be divided into time-averaged components $\bar{u}$, $\bar{v}$, $\bar{w}$ and fluctuating components u' , v' , w' :

$$u = \bar{u} + u'; \quad v = \bar{v} + v'; \quad w = \bar{w} + w' \quad (2)$$

Scalar quantities are transported by convection due to the mean flow, characterized by the time-averaged velocity, and by turbulent diffusion associated with the fluctuating velocity components.

²Results produced in this way are similar to those obtained by the Telemac-2D model, but are not identical.

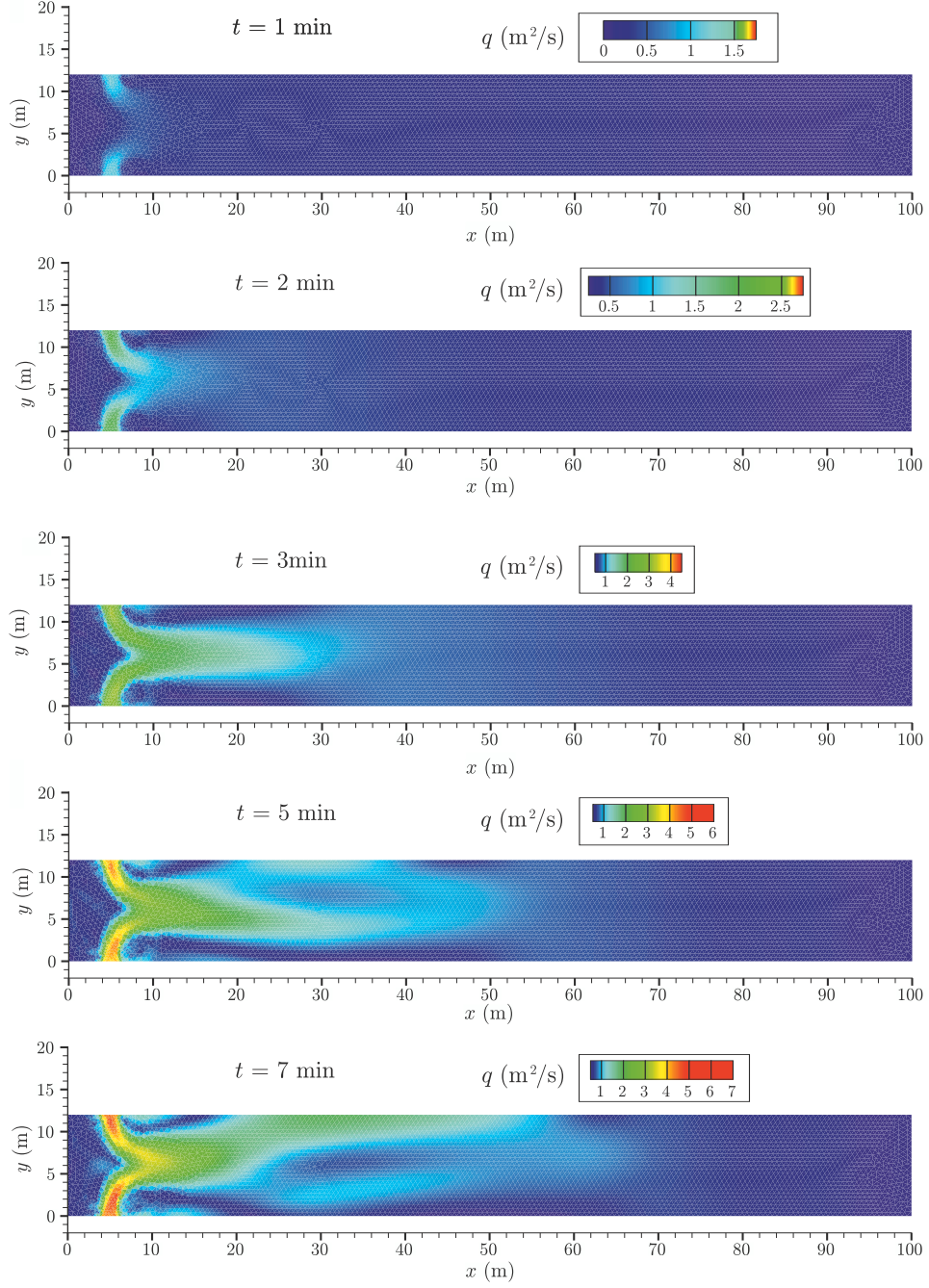


Figure 2: *Distribution of the unit discharge in the lock chamber [4].*

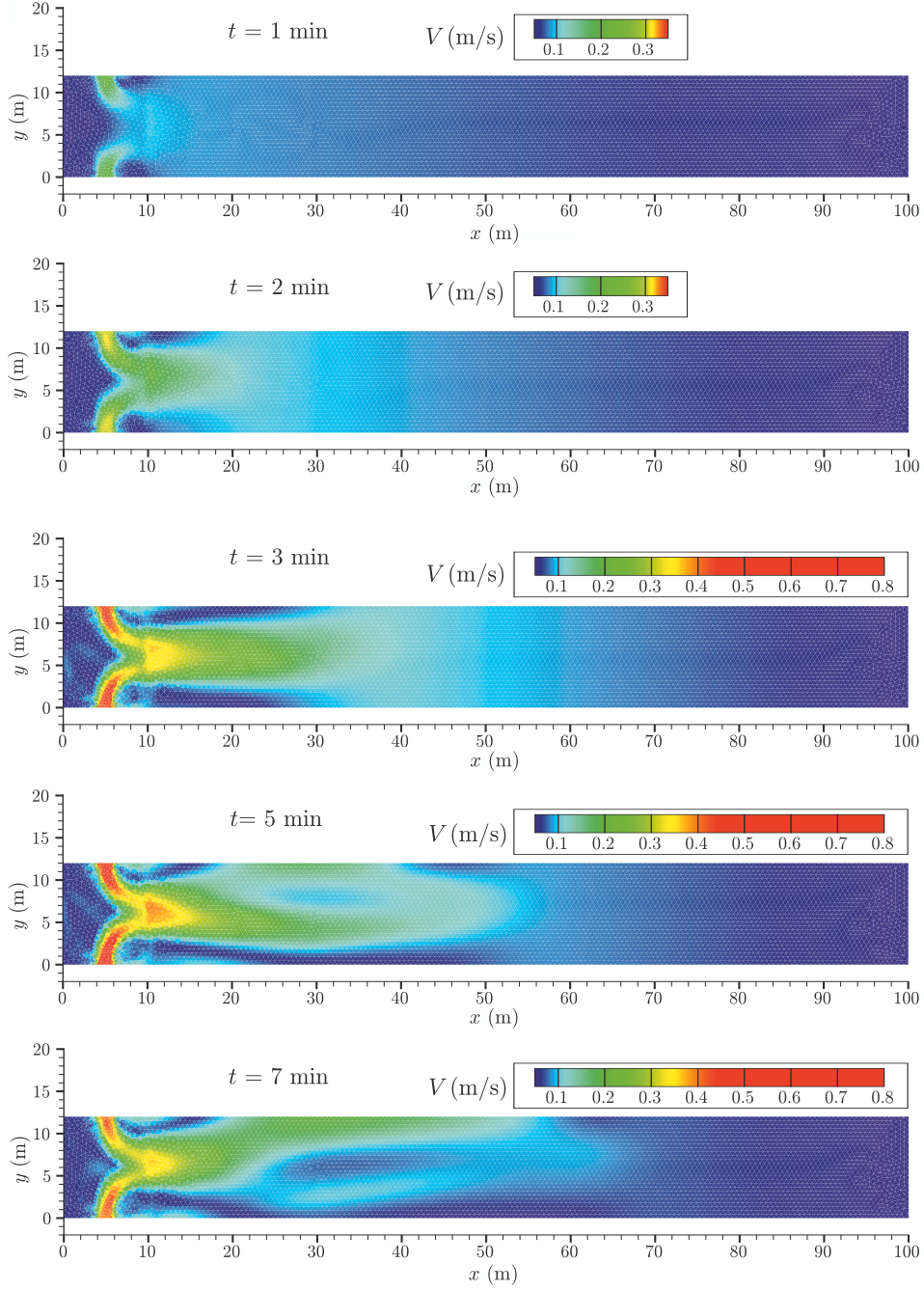


Figure 3: *Distribution of velocity in the lock chamber [4].*

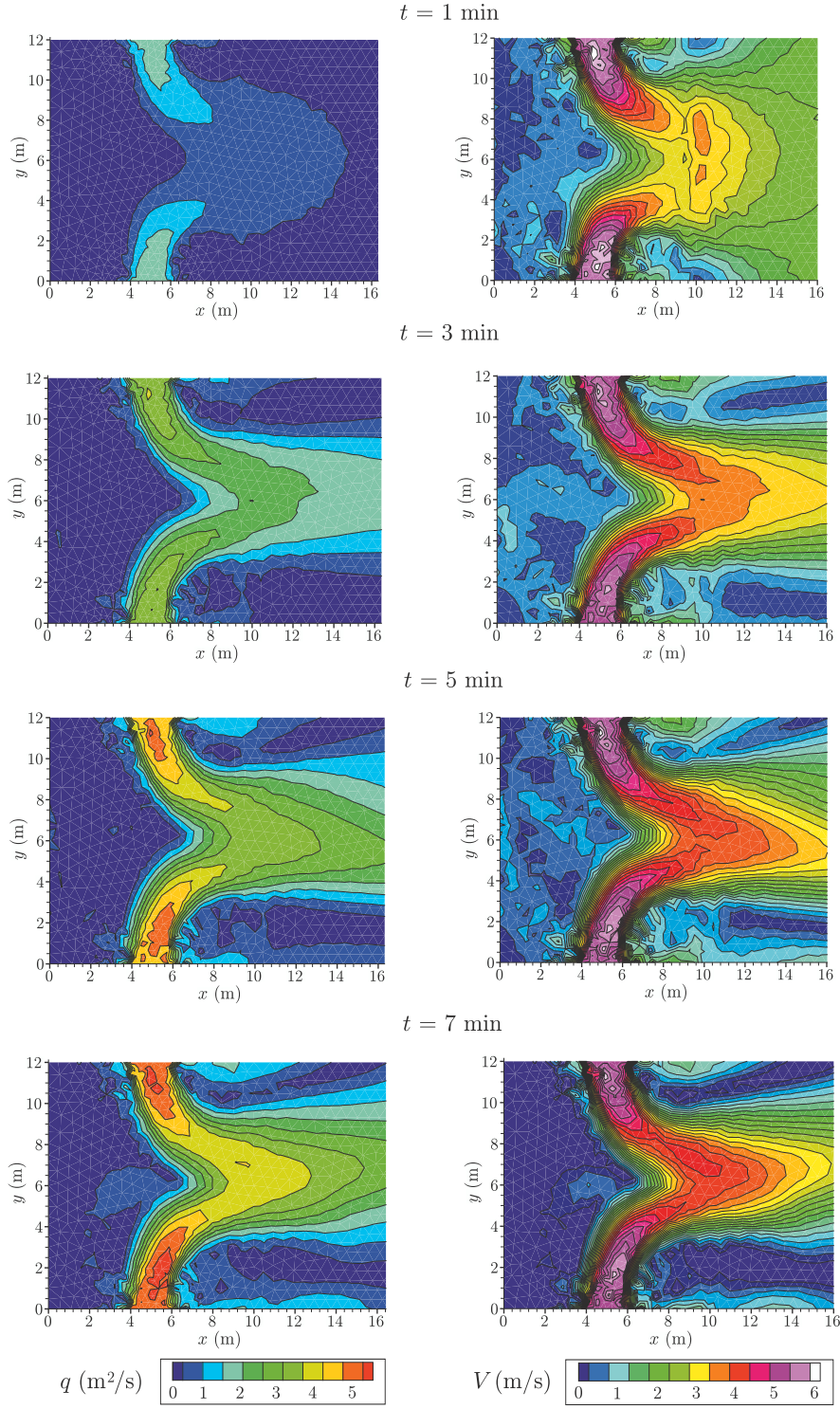


Figure 4: *Flow in the stilling basin* [4].

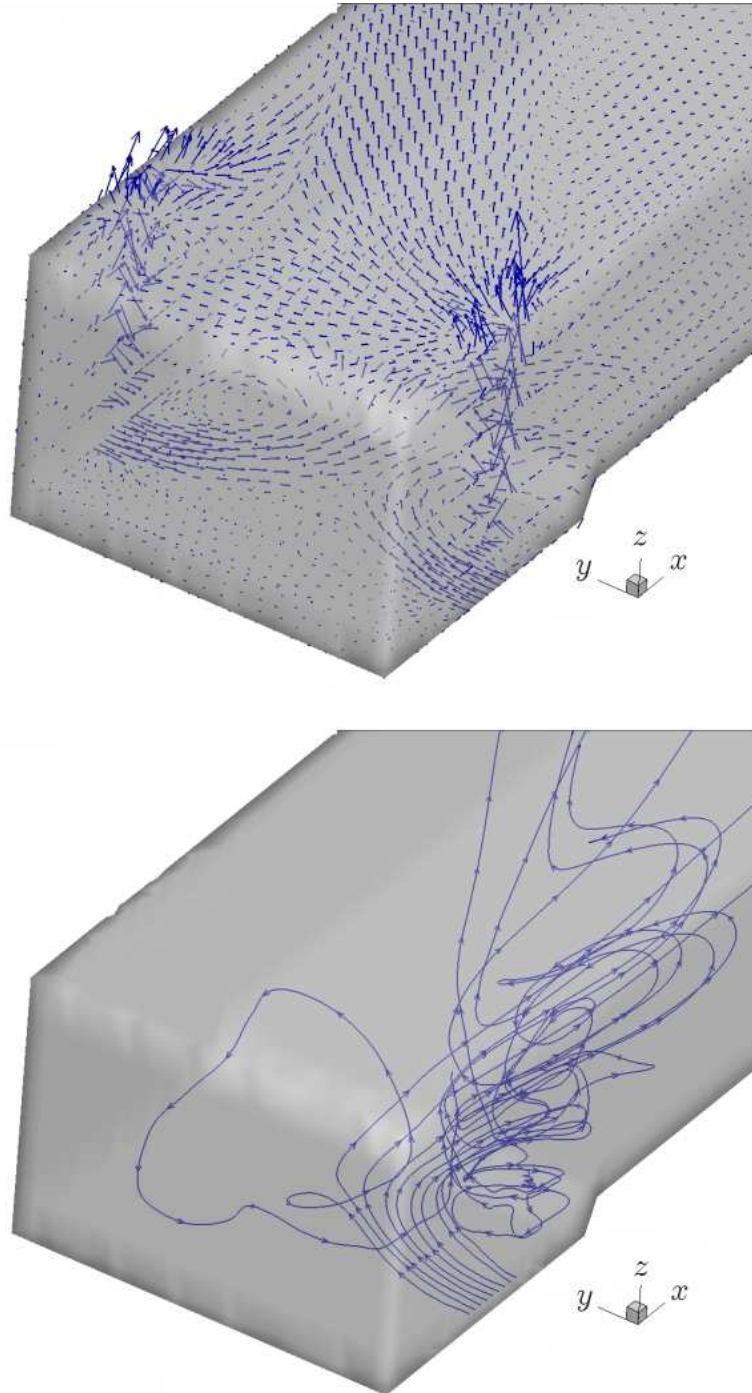


Figure 5: *Three-dimensional flow in the stilling basin; above: vector velocity field at the moment of the peak inflow ($t \approx 7$ min, $Q = 19,3 \text{ m}^3/\text{s}$); vectors reveal jet emerging on the surface; below: streamlines in the vicinity of one of the two gallery outlets supplying the lock chamber with water [4].*

The local intensity of turbulence is characterized by the magnitude of the velocity fluctuations, which can be expressed in terms of their standard deviations:

$$\sigma_u = \sqrt{\overline{u'u'}}; \quad \sigma_v = \sqrt{\overline{v'v'}}; \quad \sigma_w = \sqrt{\overline{w'w'}}. \quad (3)$$

Based on these quantities, a characteristic measure of turbulence intensity, known as the turbulent kinetic energy, can be defined as:

$$k = \frac{1}{2}(\sigma_u^2 + \sigma_v^2 + \sigma_w^2) = \frac{1}{2}(\overline{u'u'} + \overline{v'v'} + \overline{w'w'}). \quad (4)$$

In addition to (k) , the well-known $(k - \varepsilon)$ turbulence model uses another variable, the rate of kinetic energy dissipation, denoted by ε . The two quantities provide characteristic velocity and length scales of the turbulent eddies. By solving the transport equations for k and ε , the spatial and temporal distribution of these characteristic turbulence scales can be determined [3, 6, 8].

The $k - \varepsilon$ model, as well as other turbulence models, contains a number of empirical parameters. It should be noted that turbulence models do not explicitly describe the onset of turbulent structures or their actual evolution. Rather, they describe the averaged effect of turbulent fluctuations on the mean flow. What this looks like, is shown in Fig. 6.

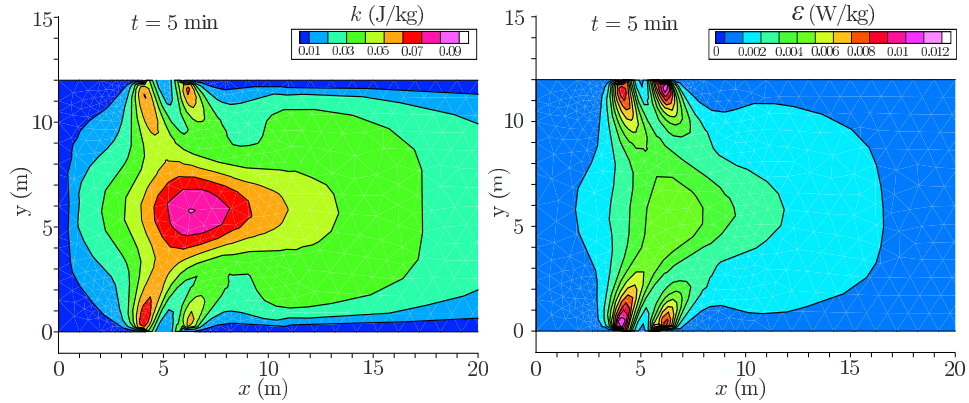


Figure 6: *On the left: the maximum values of the turbulent kinetic energy (k) occur in the center of the stilling basin, where jets from the galleries collide; on the right: the rate of turbulent energy dissipation (ε) is highest near the outlets, where energy is dissipated through eddies formed around the emerging jets [4].*

By comparing such diagrams, associated with different sizes of the stilling basin, its size can be optimized for the most efficient solution.

3 Limitations of numerical modeling

Numerical models undoubtedly have a number of advantages over physical (hydraulic) models, but they are not without limitations and problems in application.

Parametrization. Numerical models are usable only if calibrated. Without calibration, the values of model parameters cannot be considered reliable. In the case of ship locks, calibration is usually performed using physical scale models, and less often by means of field measurements on actual ship locks.

Numerical computations concerning an existing ship lock are aimed at upgrading the performance of the particular lock, which can subsequently serve as a "raw model" for new ship locks. As the amount of input data required for a 3D model is large, and the time needed to process and analyze the results is considerable, it may be concluded that the use of such a model is reasonable only at the highest design level.

Turbulence modeling. One of the main uncertainties in the application of multidimensional models is due to the choice of the turbulence model. There are two approaches to turbulence modeling. The more common RANS ("Reynolds-Averaged Navier-Stokes") approach approximates the turbulent flow structure by introducing "turbulent viscosity." The $k - \epsilon$ model is representative of this approach. The main limitation of this model lies in the fact that it involves several empirical parameters that, in practice, cannot be calibrated. The "standard" values are therefore used, and a kind of verification can be performed by comparing the results of the $k - \epsilon$ model with those obtained using other turbulence models.

The other approach is LES ("Large Eddy Simulation") [1]. The large eddies, larger than the resolution of the numerical grid, are computed directly from the governing equations, while the small SGS ("Subgrid-Scale") eddies are modeled separately [2]. In most cases, the results obtained using the two approaches are in agreement for the flow field at the scale of the lock chamber width, while the LES method enables a more detailed reproduction of small eddies.

Geometric scales. Computer modeling of a ship lock is faced with the problem of different geometric scales. There is a considerable disparity between the hydraulic quantities computed at the scale of the lock chamber and those associated with the scale of the jet outlets, either in the walls of the lock chamber or on its bottom. It is difficult to account for both scales in the same model, especially when the hydraulic filling/emptying system of the lock is very complex. One solution may be to define the flow regime of the jets using a physical model and then incorporate the results into the numerical model of the entire lock.

Model usage in practice. It is indisputable that the application of numerical 3D models has already entered engineering practice, owing to the availability of such models through the internet and other sources. Such models are important tools for the design of ship locks, complementing physical models. However, the tempting availability of high-quality commercial and open-source software is at the same time a peculiar curse, owing to the potential misuse of such software by engineers with insufficient hydraulic and numerical knowledge. The risk is even greater because modern software may produce results even under unsuitable conditions (for instance, with an inadequate computational grid). Blind faith in the results obtained using renowned software is catastrophic for engineering practice, especially because the results of 2D and 3D computations are often presented in the form of attractive multicolored graphs, which, by themselves, do not warrant any credibility. A mocking slogan was introduced for this long ago: "Colored Fluid Dynamics," alluding to the term "Computational Fluid Dynamics," CFD being the abbreviation of both terms.

4 Conclusions

1. The Telemac-3D modelling framework is a powerful tool for solving practical ship lock design problems.
2. The principal advantages of using the Telemac-3D software are its numerical stability and speed of execution, ensured by the very efficient fractional-step method for solving partial differential equations. Flexible (un)structured gridding allows an increase in numerical resolution in the lock chamber regions of special interest, i.e. inflow outlets, the basin provided for dissipating excess kinetic energy, locations of jet interaction, etc.
3. Computational results, obtained in the presented case study of frontal filling of a lock chamber through two water supply galleries in the walls, provide a very reliable insight into the unsteady flow regime during the filling process. The dynamics of this process is clearly described by the evolution of the discharge and velocity fields along the chamber. Numerically simulated three-dimensional velocity vector fields and the corresponding streamlines depict a very complex flow pattern at the jet outlets.
4. Analysis of turbulence is of particular interest, as it is the basis for design considerations associated with ship stability during the locking process and the determination of its optimal duration. Turbulence models in general, and in this case the $k - \epsilon$ model in particular, can provide evidence of the efficiency of the basin for excess-energy

dissipation, which is essential in the design of locks with frontal filling systems.

5. In practice, multidimensional numerical models have a number of advantages over physical scale models, but are not without limitations. The principal ones are parametrisation (calibration), the choice of the turbulence model, and the disparity in the geometric scales of different parts of the locking system. All of these factors complicate the numerical modelling of ship locks. Finally, there is a risk of potential misuse by practising engineers with insufficient knowledge of, or training in, the use of 3D models.

References

- [1] Bourgoïn, A., Guillo, S., Thiébot, J., Benhamadouche, S., Ata, R. (2017) *Development of a Large-Eddy-Simulation* for Free Surface Complex Flows, 37th IAHR World Congress.
- [2] Germano, M., Piomelli, U., Moin, P., Cabot, W.H. (1991) *Dynamic Subgrid-Scale Eddy Viscosity Model*, Physics of Fluids A, 3(7), 17601765.
- [3] Hervouet, J-M. (2007) *Hydrodynamics of Free Surface Flows – modelling with the finite element method*, John Wiley & Sons.
- [4] Jovanovic, M. (2024) *Ship Locks - Hydraulic computations and efficiency assessment*, University of Belgrade (in Serbian).
- [5] Launder, B.E., Spalding, D.B. (1972) *Lectures in Mathematical Models of Turbulence*, Academic Press, London.
- [6] Rodi, W. (1980) *Turbulence Models and their Application in Hydraulics*, IAHR.
- [7] Tennekes, H., Lumley, J.L. (1972) *A First Course in Turbulence*, MIT-Press.
- [8] Telemac modelling system (2010), *Operating and Reference Manuals*.