THE SIMULATION OF THE SAINT PETERSBURG FLOOD DEFENSE SYSTEM GATE VIBRATION UNDER THE LOADS FROM THE MOVING WATER

Sergey Lupuleac
Saint Petersburg Polytechnical University, Russia

Alexander Bol'shev
Saint Petersburg Polytechnical University, Russia

Julia Shinder
Saint Petersburg Polytechnical University, Russia

Eugeny Petukhov
Saint Petersburg Polytechnical University, Russia

Vladimir Chernetsov
Central Design Bureau for Marine Engineering "Rubin", St Petersburg , Russia

ABSTRACT

The presented work is focused on detailed mathematical simulation of the dynamic behavior of the gates of Saint Petersburg flood defense system of ship passing channel under the loads from moving water. The mathematical model is based on accurate time dependent CFD (Computational Fluid Dynamics) simulations to determine the structure of the flow around the gate section. The Ansys CFX code was implemented for the simulation of the multiphase (air-water) flow. The dynamic behavior of the gate section was also modeled by solving the gate motion equation taking into account the loads from the flowing water on the basis of obtained CFD results. In doing so the coupled fluid structure interaction was modeled.

1. MOTIVATION AND TIMELINESS

Several times a year Saint Petersburg experiences floods. The project of dam construction was designed to prevent the Saint Petersburg city from periodical inundations. Now this project is partly realized and the dam is almost finished (see Figure 1). Only a small part of the dam needs to be constructed. Exactly in this place the ship passing channel is planned. The width of the channel is about 200 m and the depth is 16 m.

At the menace of flood the channel is closed by two gates having the form of pontoons (see Figure 2). The gates are moved to the center of the channel and then submerged.

The model tests disclosed that in some regimes the huge oscillations of gates are observed. This technical problem significantly decreases the reliability of whole Saint Petersburg flood defense system. The failure in gate system operation can result in huge material and life losses in the case of disastrous flood. Also the blocked ship passing channel will make it impossible for ships to get into the city port.

The first mathematical simulations of the flow over gate section (Boldyrev et al.) that were performed in the St. Petersburg Polytechnic University permitted to discover the reason of these oscillations. The oscillations were caused by the turbulent vortex shading from the lower edge of the gate. The mathematical model was developed for simulation of the gate section movement under loads from the flowing water.

The dynamics of the gate section under the loads from the moving water was modeled in (Denisikhin et al.). This work confirmed basic conclusions of (Boldyrev et al.) and gave some quantitative estimations of the oscillation process.

The approach described in present paper incorporates the most elaborate mathematical model for simulation of two phase fluid flow as well as gate oscillations caused by this flow. This model is free from geometrical and physical simplifications that were implemented in two above cited works.

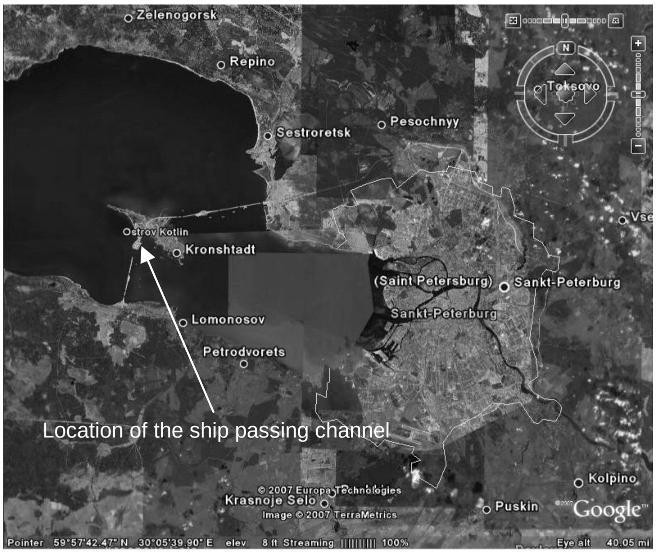

Figure 1 *The satellite photo of the eastern part of the Finnish gulf and Saint Petersburg city (taken from the Google Earth cite)*

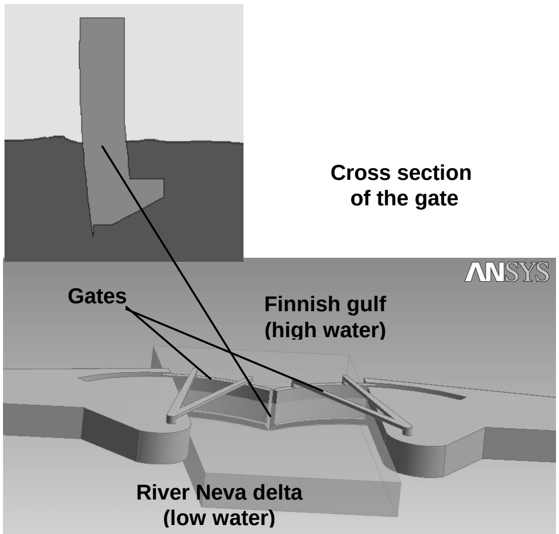

Figure 2 *The gate system (CAD model)*

2. MATHEMATICAL MODEL

We used two-dimensional CFD model and Ansys CFX code to simulate the flow over gate section. The computational area is shown in Figure 3. It involves the areas filled with water (dark grey) and air (light grey) as well as gate cross section.

The Reynolds averaged Navier-Stokes equations were numerically solved using Menter's k-ω SST turbulence model and VOF (volume of fluid) model as realized in Ansys CFX. The boundary conditions are presented in Figure 3.

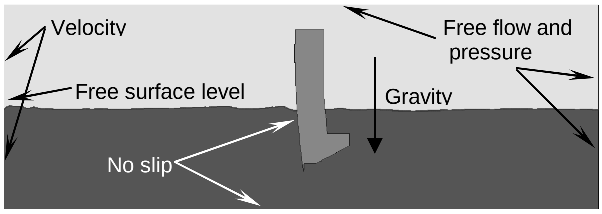

Figure 3 *Computation area and boundary conditions.*

Performing the CFD simulations we obtain the load F_{hd} acting on the gate section from the moving water and air. In presented model the gate section is able to move only in vertical direction. The gate section motion is described by the following differential equation:

$$m\ddot{X} + C\dot{X} = mg + F_{hd} \tag{1}$$

wherein m is linear mass of the gate section; C is damping coefficient that is introduced into the model to take into account the effect of different damping devices neglected in CFD analysis (in presented work we set C=0), mg is gravity, $\dot{X}$ and $\ddot{X}$ are respectively velocity and acceleration of the gate section in vertical direction.

We take note that the coupled problem of simulation of fluid flow as well as gate oscillations caused by this flow. On every time step we numerically solve the Navier-Stocks equations describing the viscous two component (air-water) flow. Next by integration we obtain the hydrodynamic force F_{hd} on the right hand side of the gate section motion equation (1). Then using the explicit Euler scheme we solve the motion equation (1) and find the gate displacement at this time step. Next taking into account the obtained gate displacement we rebuild the CFD mesh and repeat the algorithm. In that way we take into account the interference of the fluid flow and gate displacement.

The gate section mass in equation (1) is adjusted before the above described simulation to obtain desirable deepening of the gate in the stagnant water. Gate deepening here is the distance between the gate lower edge and the channel bottom in stall position. The desirable high and low water level drop is adjusted by matching the input water velocity.

The numerical solution of this problem needs the deformable CFD mesh. Figure 4 shows this mesh.

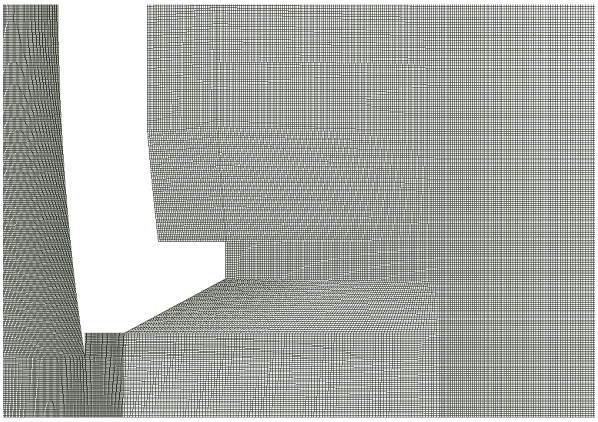

Figure 4 *CFD mesh (fragment).*

3. OBTAINED NUMERICAL RESULTS

We performed simulation of the gate dynamic behavior for different scenarios that is different high and low water level drop and different gate deepening. We site here only one scenario when the water level drop is 0.4 m and the gate deepening is equal to 3 m.

Figure 5 gives dependence of the gate displacement (in meters) from time. We can see that after the 70 sec start-up stage the gate section passes into oscillation regime with 12 sec period and 0.8 m swing.

Figure 6 presents the streamlines and Figure 7 pressure distributions (without hydrostatic pressure) in three time instances at the period that is marked in the Figure 5. The time interval between instances is 4 sec.

As it is seen from Figure 7, the turbulence vortex shading is occurred from the lower edge of gate that leads to periodical change of pressure under the gate. This process initiates the gate oscillations. Moreover after start-up stage the vortex shading is synchronized with gate oscillation and the self-oscillation regime is observed. We take note that similar effect was observed for all simulated scenarios.

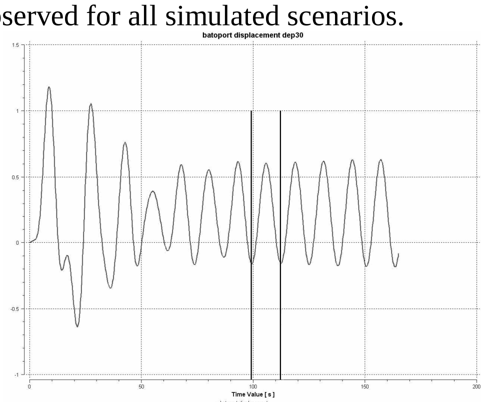

Figure 5 *Dependence of the gate displacement (in meters) from time.*

4. COMPARISON WITH PHYSICAL EXPERIMENTS

The special simulations of the scale (1:30) gate section model were made to meet the conditions of physical experiments described in (Klimovich, Chernetsov et al.). The general appearances of the flow obtained both in the numerical analysis and in physical experiments were very similar. The flow over gate section observed in the experiment is shown in the Figure 8. The water was slightly colored to make the vortex structure visible. The Figure 9 presents the streamlines of the flow obtained in the numerical simulations. The proposed computational approach permitted to perform the simulation of the gate dynamic behavior and obtain good coincidence with experimental results.

The quantitative parameters of the gate section oscillations obtained in the numerical analysis for scale model are also in good agreement with experimental results. The period of oscillation for scale model obtained in numerical analysis as well as in physical experiment is about 2 seconds. The discrepancy between computed and observed oscillation amplitude for different scenarios is within 15-20%.

5. SUMMARY AND FURTHER RESEARCH AVENUES

The numerical analysis alongside with physical experiments was very widely used for thorough examination of the effectiveness and reliability of the gate system.

The project is still under way. Recently it has been decided to modify the shape of the gate section. Computations are implemented for search of the most suitable gate section shape.

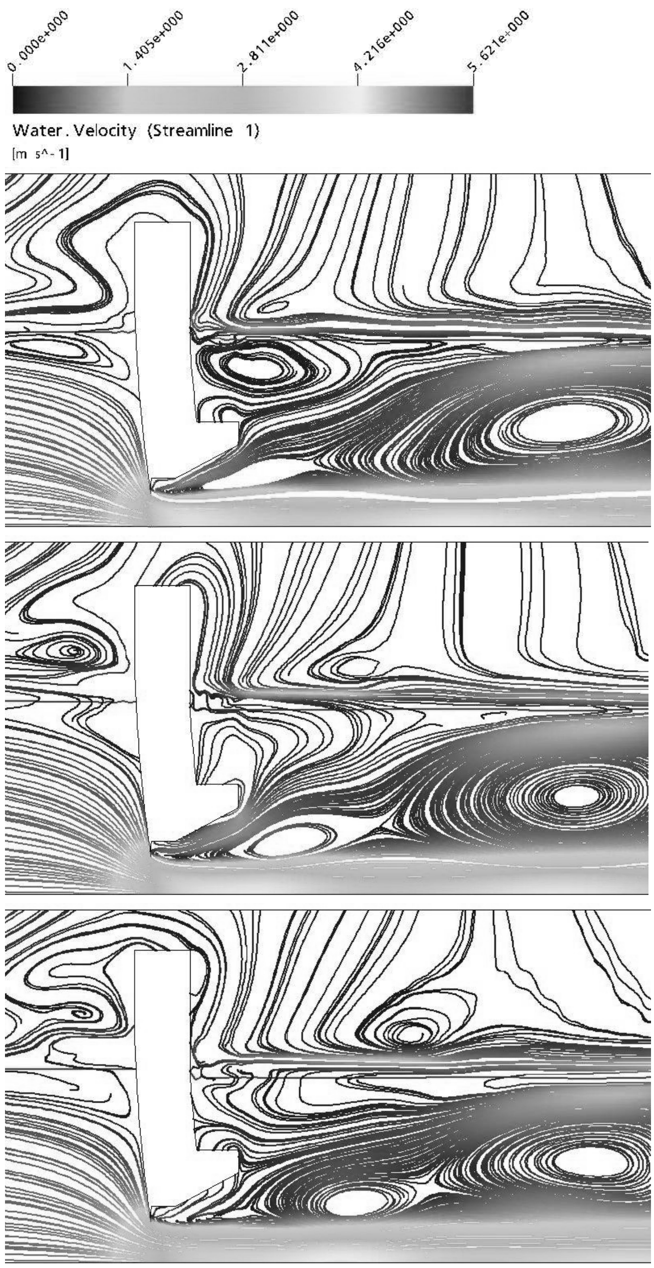

Figure 6 *The streamlines in three instances.*

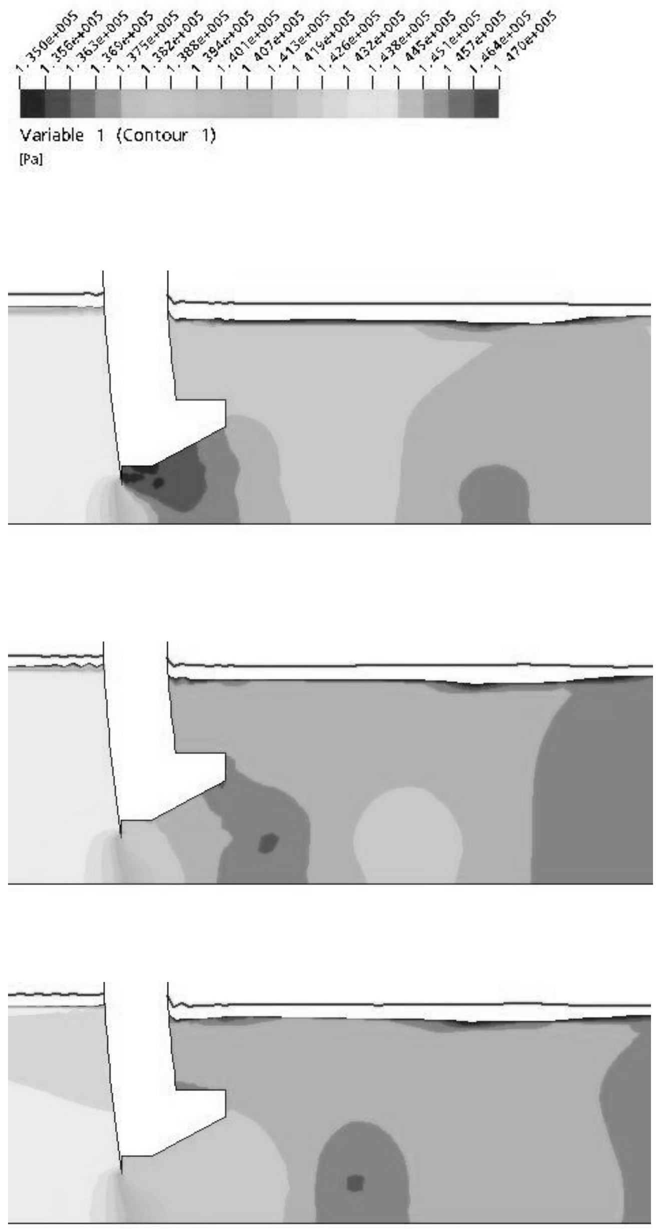

Figure 7 *The pressure distributions in three instances.*

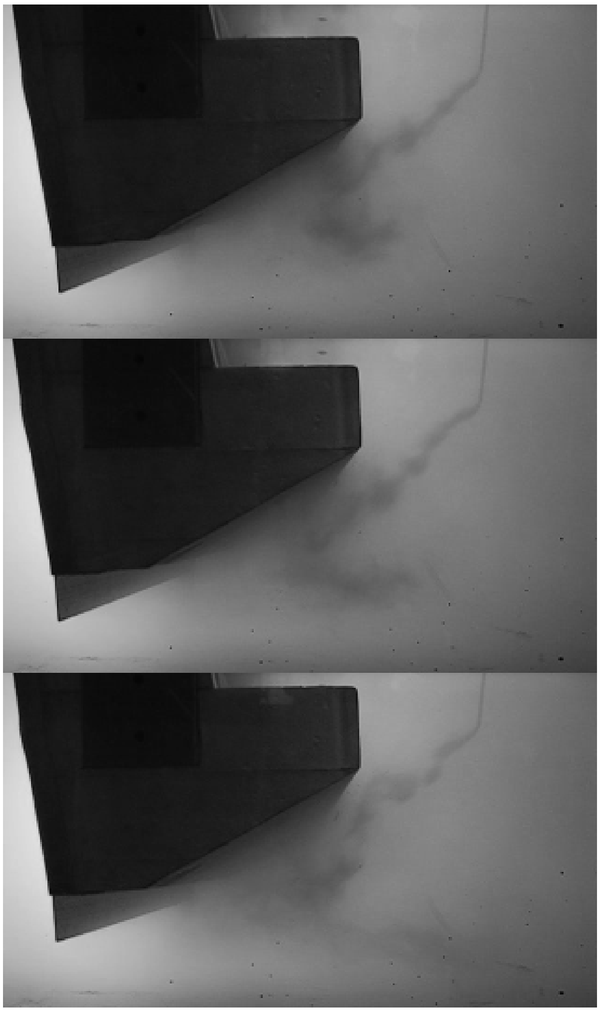

Figure 8 *Experimental observation of the turbulence vortex under the lower edge of gate section.*

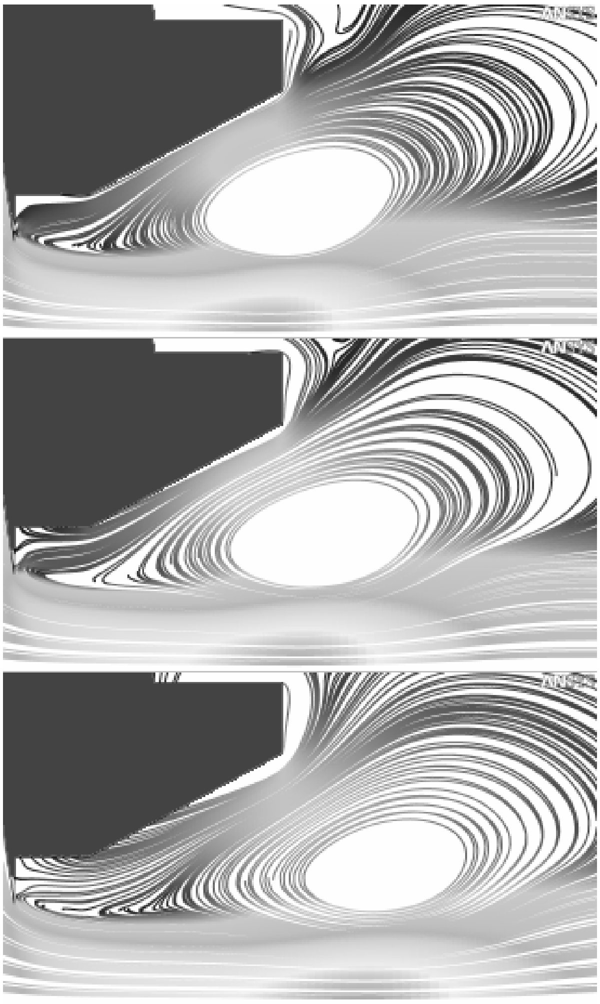

Figure 9 *Numerical simulation of the turbulence vortex under the lower edge of gate section.*

6. ACKNOWLEDGEMENTS

This work was supported by Microsoft Corporation in the framework of Technical Computing Initiative (TCI) program.

Contributions by Ms Nadezhda Fedosenko in CFD model development and by Mr Kirill Zamotin in paper design are gratefully acknowledged.

7. REFERENCES

Boldyrev, Yu., Lupuleac, S., Zhelezhyakova, O., 2004, Numerical simulation of the turbulent flow in the dam ship passing channel. *In proceedings of the conference Innovations and technical policy ,St. Petersburg, Russia.*

Denisikhin, S., Kupreev, V., Sukhorukov, A., Chernetsov, V., 2007, Mathematical simulation of floating gate dynamics during submerging process. *ISPC STAR-2007 Computer technologies in applications to heat and mass transfer problems, Nizhnii Novgorod, Russia.*

Klimovich, V., Chernetsov, V., Kupreev, V., 2006, improvement of floating gate design for C-1 navigation pass of St. Petersburg flood protection barrier. *32nd Congress of IAHR the International Association of Hydraulic Engineering and Research, Venice, Italy.*