ISSN 1453 – 7303 "HIDRAULICA" (No. 1/2014)
Magazine *of Hydraulics, Pneumatics, Tribology, Ecology, Sensorics, Mechatronics*

CFD MODEL OF FLOW IN THE OUTLET CHANNEL OF FLOATING CHAMBER

Ing. Karol STRAČÁR*, doc. Ing. Jozef KRCHNÁR*, CSc., doc. Ing. Karol PRIKKEL*, CSc.

*Institute of Chemical and Hydraulic Machines and Equipment, Faculty of Mechanical Engineering,
Slovak University of Technology, *Nám. slobody 17, 812 31 Bratislava 1*
karol.stracar@stuba.sk, jozef.krchnar@stuba.sk, karol.prikkel@stuba.sk

Abstract: *The article deals with the creation of a mathematical model of a floating chamber outlet. The created model should simulate the loading of the segment of a regulatory outlet closure caused by flowing of the fluid around this object. The result should help in designing changes that could help decrease the vibration on segments of the regulatory closure of floating chambers Waterworks Gabčíkovo – Nagymaros.*

Keywords: *simulation model, CFD, regulatory closures*

1. INTRODUCTION

The dynamical component from the flowing fluid will be determined by numerical simulations by means of CFD methods that are a necessary supplement of experiments on a real outlet object. They help us describe the effect of the flow on the regulatory closures segment of the outlet channel of a floating chamber. From this we could determine the loading of the segment without the necessity of time and material consuming and financially demanding measurement. Equally, we get the data about the distribution of examined parameters such as the distribution of pressure, phases in outlet channel or information about the course of the flow velocity in the outlet channel.

CFD methods are used in fluid mechanics where, by means of the numerical methods and solution algorithms, we can analyze problems dealing with the fluid flow. It is possible to obtain information about the interaction of fluids and gases with surfaces, while defining the boundary conditions [6], [7]. The fluid that flows through the outlet channel drains around the regulatory closure segment and influences it by force. The fluid force that acts on the segment is transmitted to a linear hydro-engine. To discover the force effecting on the regulatory closure segment, it is necessary to create a simulation model with a dynamic mesh. Similarly processed also the authors in article [1], where they used the CFD simulations to better understand the complicated flows inside the channel used for the free transfer of fish through the waterworks dam. The possibilities of a CFD solution with dynamic mesh are described in the article [2], where the author describes CFD simulation of a 2D computational analysis of fluid dynamics for a 3-paddle H-Darrieus rotor using nonstructural mesh with a moving model. The description of a flowing field using the CFD models are presented also in the article [4], where the author solved the flow in the area of wave-breaks within the free water surface in a full 3D model.

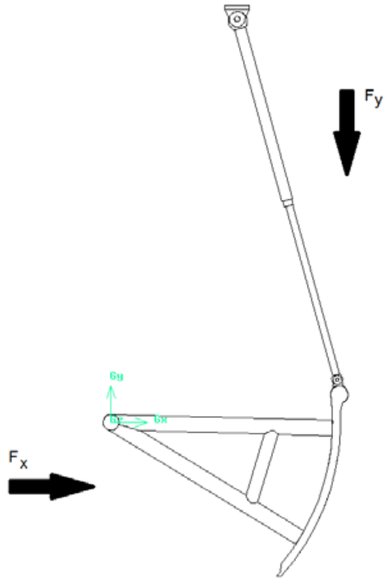

FIGURE 1: *The forces acting on the regulatory closure segment*

The outlet closures allow the entry of water from the floating chamber until an equilibrium of water levels within the lower level is reached in the concerned chamber. The chamber has square dimensions, which means that the setting of the segment is made with a barrier surface of squared shape and 4 x 4m dimensions. The segment is gimbaled, while the movement of the segment can be stopped in any position. All closures have the same structure. Each closure is directed by one linear hydro-engine filled with a pressurized fluid inserted by a hydraulic aggregate [5].

The time of elevation of the regulatory closure segment in the outlet channel of a floating chamber is 120s. The emptying of the floating chamber to a water level suitable for a boat to sail out is approximately 15-20 minutes. However, this is not the time of total water level equilibrium in the floating chamber.

2. THE CREATION OF A SIMULATION MODEL USING THE DYNAMIC MESH

The creation of a dynamic mesh allows us in one simulation to describe the loading rates as well as the pressure and velocity courses during the whole 120 second cycle of the opening of the regulatory closure segment. The example of such a model is in (Fig. 2). It is advantageous to use structural mesh on parts that are sufficiently far away from the regulatory closure segment. However, it is also necessary to use non-structured mesh on areas around the regulatory closure segment that are intended to move. During the designing of the dynamic mesh we made use of the MAP, SUBMAP and PAVE meshing schemes. The final model was composed of 700 000 elements. The input into the simulation model of the outlet channel of the floating chamber is on the left side, the output on the right side (Fig. 2).

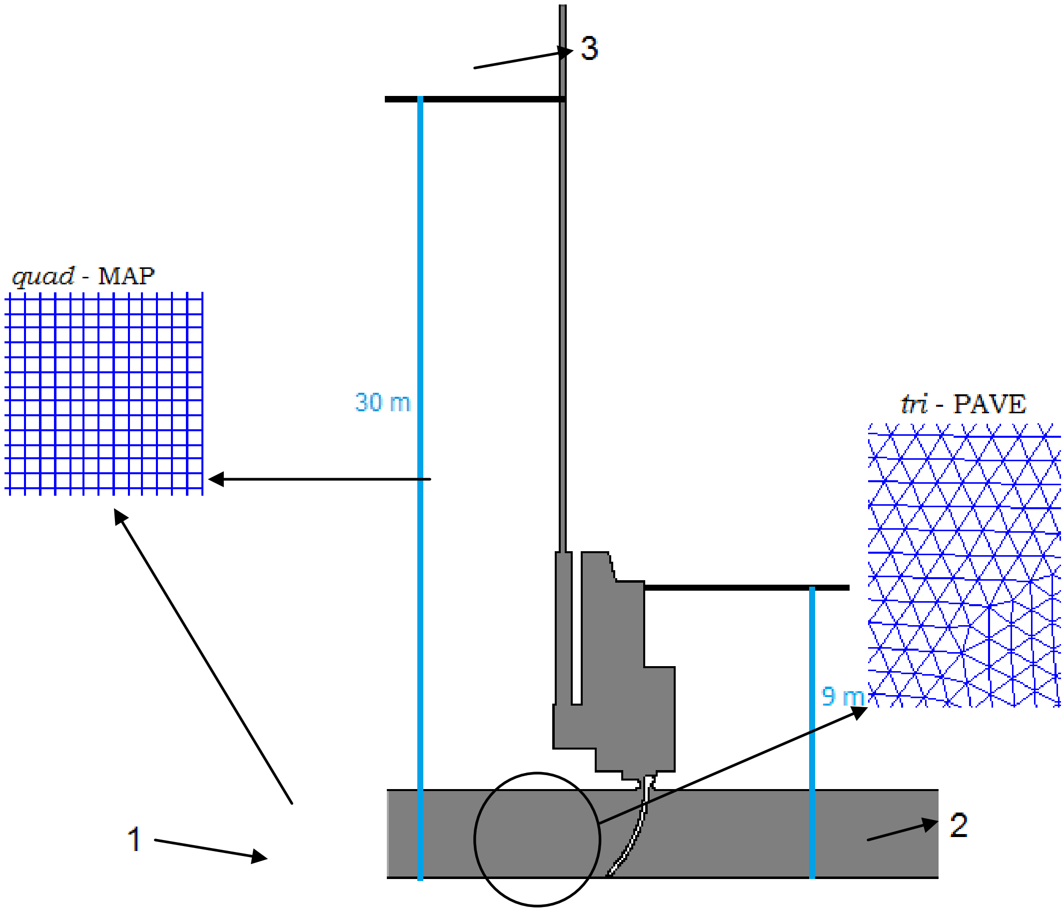

Figure 2: *The defined boundary conditions of the model in software Ansys/FLUENT*

The input was defined by boundary condition pressure-inlet (1), where the maximum height of the water level was determined and also the height at which the water level decreases during opening. The output was defined by boundary condition pressure-outlet (2), where the height of the water level was set at 9m, which is also the river level. In the upper shaft, similarly to output we set boundary condition pressure-outlet (3), which was defined by the maximum and minimum heights of the water level. The configuration of the height of the water level as input parameter was set by the condition that defined the free level surface in this shaft (the definition of free surface means determination of the borders between fluid and atmosphere). In the numerical simulation, as a flowing media was used water as the primary medium with real values defined for 4°C and as a secondary phase, air. The simulation must be resolved in steps because this method improves the stability as well as the precision of the computation

Moreover, in this case it is necessary to set up in the computational software the parameters of the dynamic mesh, in other words to set up the values for smoothing and re-meshing of the result that is appropriate for the numerical simulation. Apart from the creation of a dynamic mesh, dynamic zones must also be created. In this case a dynamic zone is composed only from the moving regulatory closure segment that is defined as the rigid body. This zone has to have the direction and velocity of the motion also determined, since in our case the segment is moving continuously alongside the circle with velocity 0.035 m/s. In cases of simulation models built up with dynamic mesh, one small disadvantage is the need for a UDF, User Defined Function, subprogram. It is represented by the function determined by the software or environment user. The disadvantage is that this function slows down the computational process in the simulation model, as it influences the internal computation system of the own simulation program. Using dynamic mesh through UDF, we define the weight of the regulatory closure segment and the moment point of inertia.

In simulation models where the dynamic mesh is used, we have to first of all find the simulation solution in a stable time mode. In the beginning was created a simple solver scheme, whereas the computation was done in stable time mode with setting up 100 time steps, as previous models were worked out. Later the simulation was set in second and third orders of precision, as is noted in Table 1.

Table 1: *Setting up of simulation in the stationary mode in the computational software Ansys/FLUENT*

Number of time steps	Pressure - velocity coupling	Pressure	Volume fraction	Momentum
100	SIMPLE	PRESTO!	1st order upwind	1st order upwind
100	Coupled	PRESTO!	1st order upwind	1st order upwind
100	Coupled	PRESTO!	2nd order upwind	2nd order upwind
100	Coupled	PRESTO!	2nd order upwind	3rd order muscl
100	Coupled	Body Force Weighted	2nd order upwind	3rd order muscl

As the regulatory closure segment is elevating during the simulation, it is not sufficient to find the solution only by establishing a stable time step. It is necessary therefore to switch the model to an unsteady random? time mode. This mode allows finding a time-dependent solution, as it permits simulation of the entire range of time-dependent phenomena, such as whirl or for example the filling and emptying of containers. Simulation is set up in time in this mode of processing when the simulation model is working in the third order of precision. During the numerical simulations in unsteady time mode, it is necessary to correctly set certain parameters. From our experience, it was useful to set the value of the time step to 0.1s. The time of real elevation of the segments is120s and therefore it is necessary to set the value of the time steps to 12000 for the entire length of the cycle.In the previous figures (Fig. 3 a, b) was presented the creation of a new mesh in a simulation model under the segment during its motion in selected time modes. This generation and evanescence of the mesh is resolved by means of a dynamic mesh, where the computation software during the simulation automatically calculates the elements over which has the segment raised.

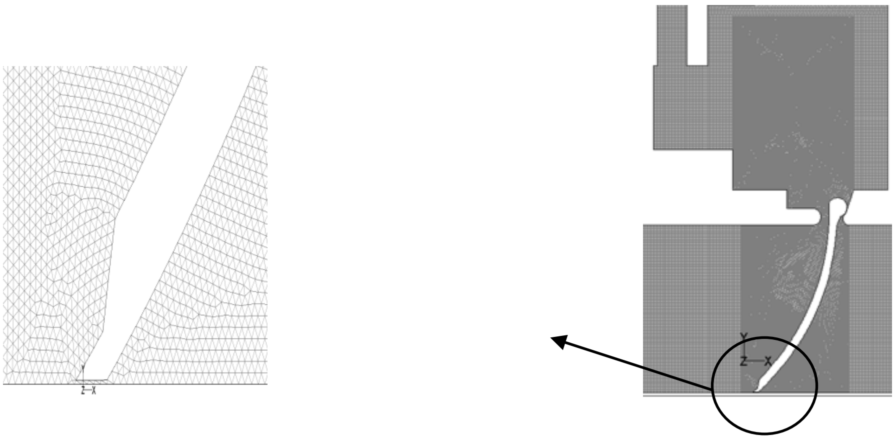

Figure 3 a: *The shape of dynamic mesh for time t = 1s*

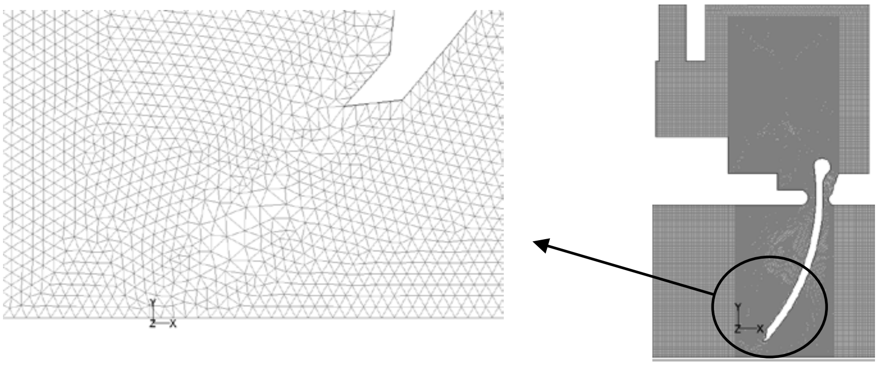

Figure 3 b: *The shape of the dynamic mesh for time t = 15s*

During the simulation, in each time step we recorded the forces along the axes „*x*“and „*y*“. The values of the forces are represented on the graphs (Fig. 4, Fig. 5). The barrier is connected to a linear hydro-engine, which secures its lifting. One part of the force acting on the barrier is transmitted to the hydro-engine. As the segment is also fixed onto the wall of the outlet channel, one part of the force from the axes „*x*“ is consumed just at this point and is represented by friction.

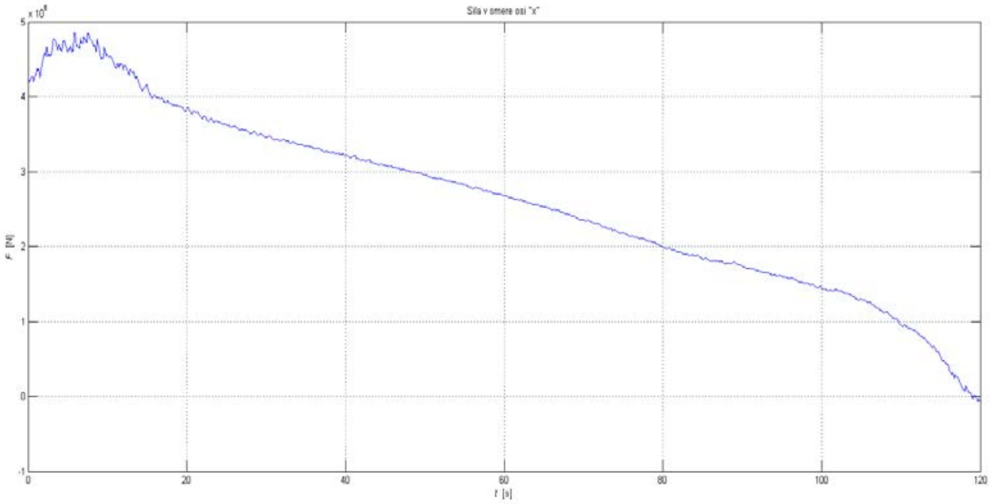

Figure 4: *The force acting on the segment in direction of x-axis*

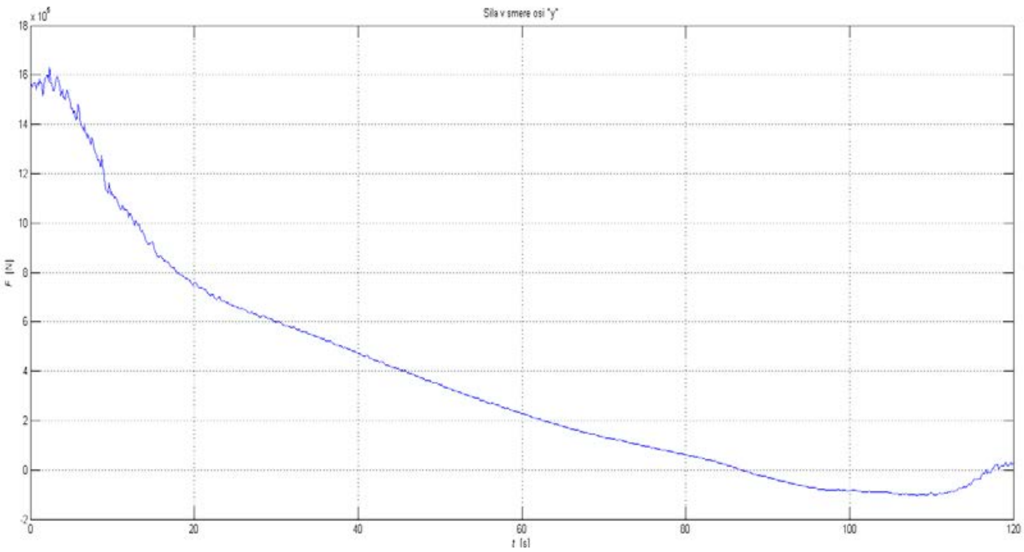

Figure 5: *The force acting on the segment in direction of y-axis*

The simulation model allows us to get information about forces acting from the flowing medium. From the numerical simulation using CFD methods, it is possible moreover to determine the pressures, velocities and other parameters at the individual points of the simulation model, as this model was created in correspondence with real parameters. This gives the possibility to compare the simulated parameters at selected points of the outlet channel with data that were measured on the upper cover of the outlet channel in front of and behind the regulatory closure segment. The comparison of simulated and measured data is shown on graph (Fig. 6). The selection of the evaluated area of the simulation model was determined according to the layout drawing of the outlet channel of the floating chamber at the places where the pressure sensors were placed. The presented simulated pressures in the graph (Fig. 6) at selected points of the simulation model were evaluated in apost-processing program which is useful for evaluation of the results of CFD simulation.

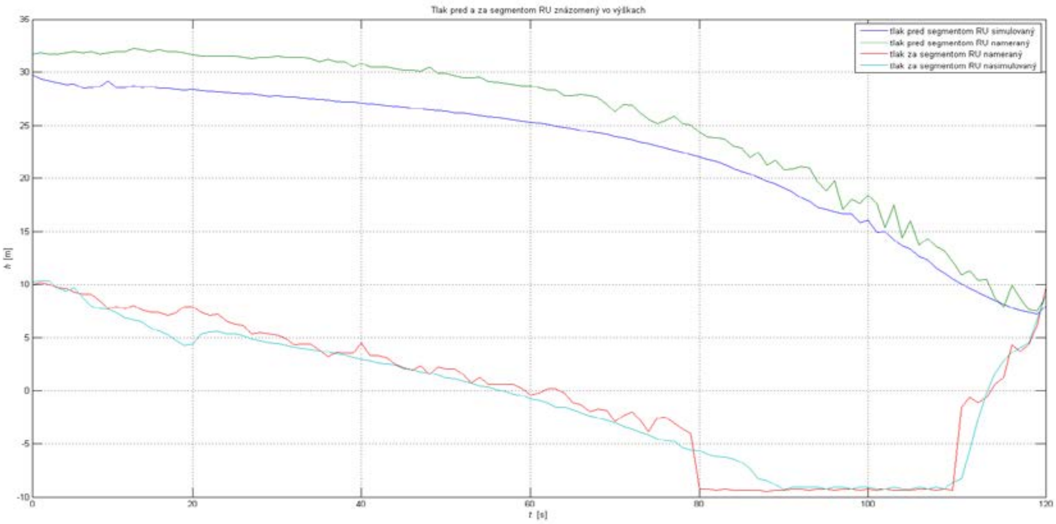

Figure 6: *The pressure in the outlet channel in front of and behind the segment RU (Plot at the height of the fluid column)*

In the graph (Fig. 6) the difference between the measured and simulated data is visible. It is caused by measurement, mostly by the difference of the water level height in the floating chamber. During the real measurement, we measured the height of the water level at almost 32m, which is an above-average water level for that period of the year. In the simulation, we are using a water level height of30m, which is the average height during the whole year.

3. CONCLUSION

In this case, the CFD simulations provided information about the forces created during the fluid flow around the segment and its influence on the regulatory closure. These forces were needed mostly in terms of increasing of power of the hydraulic circuit that secures the elevation of the regulatory closure. The advantage is saving of costs and time that would be needed for the preparation and proper performance of the experiment to determine the forces acting on the segment. Another advantage is in addition to the course of the forces acting on the regulatory closure segment that we get from the simulation also the courses of pressure differences in front of and behind the segment that could be compared with measured values during the whole cycle of closures opening. Finally, the simulation provided a colorful video of fluid flow velocity propagation alongside the floating chamber outlet channel that could be used in future modifications of this channel.

REFERENCES

[1] FERRARI, E. G., POLITANO, M., WEBER, L. *Numerical simulation of free surface flows on a fish bypass.* Hydroscience and Engineering, Iowa, Elsevier Ltd., 2008

[2] GUPTA, R., BISWAS, A. *Computational fluid dynamics analysis of a twisted three-bladed H-Darrieus rotor.* Journal of renewable and sustainable energy, American Institute of Physics, 2010, ISSN 1941-7012/2010/2(4)/043111/15

[3] PODLESNÝ, J. *Diagnostika lineárneho hydrostatického pohonu.* STU, Strojnícka fakulta, Bratislava, Dizertačná práca, 2012

[4] YAZDI, J., SARKARDEH, H., AZAMATHULLA, H. MD., GHANI, AB. A. *3D simulation of flow around a single spur dike with free-surface flow.* Intl. J. River Basin Management, Vol. 8, No. 1, International Association for Hydro-Environment Engineering and Research, 2010, ISSN 1814-2060 online

[5] *Haťová prevádzka a suchý dok plavebnej komory VD Gabčíkovo*, STU, Strojnícka fakulta, Bratislava, 2000

[6] *Ansys/FLUENT 12.0/12.1 Documentation.* Ansys, Inc. 2009

[7] *Computational fluid Dynamics*. Základná teória CFD, Internet: http://en.wikipedia.org/wiki/Computational_fluid_dynamics