Science, Engineering & Education, 1, (1), 2016, 3-10

Validation procedure for numerical model of hydraulic inflatable dam under pulse load

Veselin Iliev*, Nikolay Nikolov

University of Chemical Technology and Metallurgy, 8 Kl. Ohridski, 1756 Sofia, Bulgaria

Received 15 March 2016, Accepted 05 September 2016

ABSTRACT

Hydraulic inflatable dams are cylindrical dams, formed by load bearing rubberized fabrics, placed across streams to raise the upstream water level when inflated. During operation, they are exposed to the water flow and its static and dynamic characteristics. While the behavior of these structures under static load is well studied, the dynamic load is still under investigation. This is particularly emphasized when the water flow is in the form of wave causing pulse load. Such behavior is difficult to being studied analytically and usual method here is a numerical simulation. This article presents a validation procedure for a model for numerical simulation of a hydraulic inflatable dam under pulse load, caused by water wave. The used software is ANSYS CFX. The experimental equipment and experimental procedure are described for short, two-side fixed dam in a rectangular bed. Visual and numerical criterions are proposed in order to estimate the adequacy of the model.

Keywords: inflatable dam, CFD analysis, validation procedure.

INTRODUCTION

There is well-tried experimental technique for testing of inflatable dams in ordinary operating conditions [1 - 3]. To obtain the dam's behavior under complicate load, such as dynamic load of a parallel flow and flow fluctuation, the numerical methods are necessary to be used. The majority of researchers, working in this field, explore the vibrations caused by micro-pulsations of the water flow. A two-dimensional analysis is applied in [4] to the vibrations of air-inflated dam under overflow conditions. The cylindrical membrane is taken as inextensible and anchored along two generators. The flow is incompressible and the total head is specified. The eigenvalues and eigenvectors obtained are employed to describe the small vibrations of the dam. The effects of the dam's density and damping coefficient on the stability and response of the dam are investigated. In the same conditions, the finite element method is used to determine the dynamic response of

** Correspondence to: Veselin Iliev, University of Chemical Technology and Metallurgy, 8 Kl. Ohridski, 1756 Sofia, Bulgaria, E-mail: veso@uctm.edu*

the inflatable dam subjected to a flood using ABAQUS with a shell element [5]. The authors solve the problem in the time domain which allows them to predict a number of transient phenomena such as generation of upstream advancing waves, dynamic structural response and structural failure. The suppression of vibrations in air-inflated membrane dams is investigated in [6]. The vibrations are assumed to result from disturbances that include unsteady water flow and wind pressure. There are several attempts to solve the vibration problem in 3D conditions. In [7] the three-dimensional vibration behavior of two single-anchor inflatable dams with fins is analyzed, both in the absence of water and in the presence of stationary or parallel flowing water.

The dam is modeled as an elastic shell inflated with air and resting on a rigid foundation. The effects of the internal pressure, external water head and parallel flow velocity on the vibrations of the dam are investigated and the results are compared to those for the dam in the absence of external water. The vibration frequencies and the corresponding mode shapes of a rubber dam are computed in [8] by using finite element software produced by ANSYS Inc. During their operation the inflated dams are subjected not only on unsteady water flow and wind pressure, but on the pulse load of the incoming waves, caused by a sharp rise in water level in the river or canal. The numerical model for investigation of the mechanical behavior of inflatable dam under pulse load is proposed in [9]. The model is based on the well-known equations of Navier-Stokes and k-epsilon model for turbulence. The computer simulation technique and verification of the model is presented in [10]. The simulation is performed by finite volume method, implemented with ANSYS CFX software. For simulation of the free-surface flow, the VOF (volume of fluid) method is used with buoyancy function switched on. This article presents validation of results from this model for free-surface flow, by comparing with results from laboratory experiment. Two criteria (visual and numerical) are proposed for estimation of the model adequacy.

EXPERIMENTAL

Laboratory experiment

The specialized laboratory system is designed and constructed to meet the specific requirements of the experiment. The system should create conditions for modeling of free surface flow, allow the deployment of the inflatable dam model and measure the following parameters: the depth of the stream before the dam, overflow height and the depth of the stream after the dam. Additionally, the velocity of the flow has to be measured in several particular places at the dam position. The measured process parameters will be compared with values, obtained by computer simulation under similar conditions.

The general description of the laboratory equipment is shown in Fig. 1. The system consists of a metal bed with a shape of rectangular prism (1), 1.2 m long and 0.6 m wide. At both ends of the bed are formed additional volumes (2) and (3). Above the bed is arranged a transverse rack bar (6) which at both ends has a slides (7) and screws (8). On the rack bar there is a slide (9) on which is disposed a vertical graduated strip (10) and screws (11) (12). On the board (4) are positioned a pump unit (13) connected to volumes (2) and (3) by tubes (14) and (15), and a pump unit (16) connected to volumes (2) and (3) by pipes (17) and (18). On the walls of the bed are fixed four rods (19) and graduated tube (20).

The experimental system works as follows: In the starting position the volume (3) is filled with water up to the experimental level and is keeping in that position by flat water gate, anchored on the bottom by cylindrical joint. The inflated dam model (balloon, attached on the vertical walls), is located on the bottom of the bed, approximately in the middle of the longitude. By operating the pump unit (13) the balloon is filled to the correct pressure, which is monitored by piezometric tube (20). The experiment starts with opening the water gate and then stream enters the bed, flows over the balloon and reenters in the extension (2). The depth of the stream is monitored through the

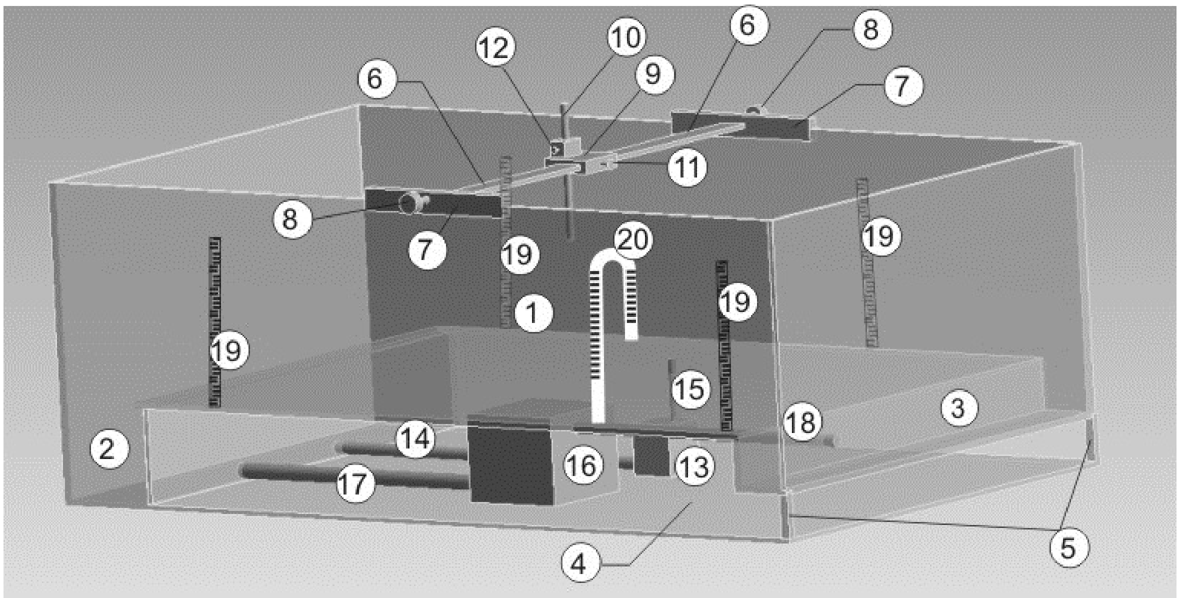

Fig. 1. Laboratory equipment.

graduated strips (19). The overflow height of the flow over the balloon is measured by a graduated strip (10) which is moved vertically by a screw (12), transversely together with a slide (9) by a screw (11) and longitudinally together with the rack bar (6) by screws (8). The all process of the fluid flow over the dam is recorded by a hi-speed camera thru the transparent side wall.

Numerical simulation

The mathematical model, the computer simulation technique and verification of the model is presented in [10]. Here only equations directly connected whit experimental technique will be presented. The emphasis is on the mathematical modeling of the free surface fluid flow. For this, the volume of fluid (VOF) method, based on the concept of a fractional volume of fluid, is used. A volume fraction (VFRC) parameter is introduced in calculation scheme for determine the degree of occupancy of the element with a fluid. The boundary between empty (VFRC=0) elements and full elements (VFRC=1) defines a polygon, crossing non-empty elements with VFRC value between unity and zero (Fig. 2).

The first step of the calculation procedure is to solve the set of Navier-Stokes equations in 2D environment xO_y and to obtain velocity field V_x and V_y. The next step is to calculate the Lagrangian displacements ζ (along x) and η (along y) using false time step δt:

$$\zeta = \int_t^{t+\delta t} Vx\, dt$$

$$\eta = \int_t^{t+\delta t} Vy\, dt \qquad (1)$$

By obtained displacements, the new locations of the polygon vertices can be calculated (Fig. 3). As a result, a portion of the new polygon can cross into the neighboring elements (n, p, q in Fig.3a). After that, the new VFRC is calculated, the new front of the filled portions of elements is defined

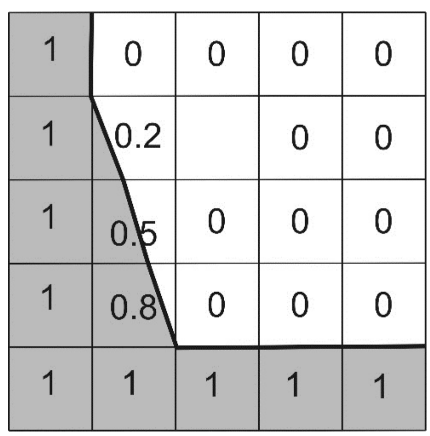

Fig. 2. Volume fraction distribution.

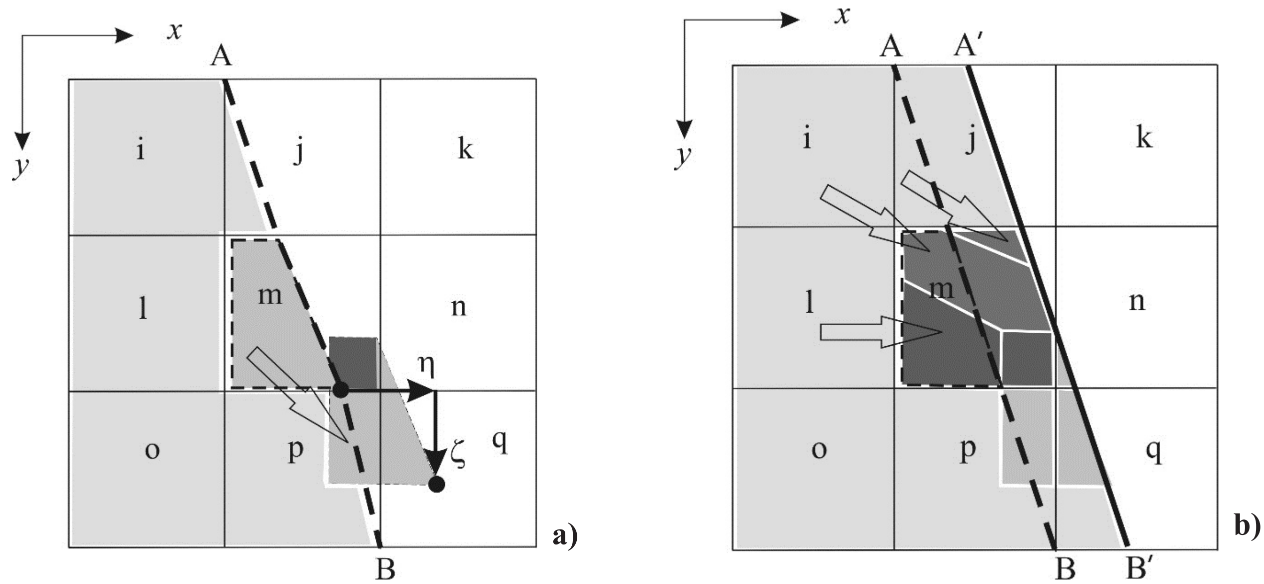

Fig. 3. Volume fraction polygon motion.

(line AB from Fig. 3a moves in new position A′ B′ in Fig. 3b) and procedure continued with next time step. As in our case the flow is 3D, it is necessary to extend results from 2D approach to higher dimensions by "reconstruction" of the interface in each cell, given the volume fraction in neighboring cells.

Another problem, typical for this investigation, consist of defining boundary conditions reflecting the real working state, and set initial conditions, leading to convergence of the solution. The water gate is joined on the bottom of the bed, so in the start of the experiment the opening direction is from top to bottom. In addition, the influence of the gravity on the velocity distribution along the height has to be taken into account in advance. In this case the 3D geometrical model of the bed is designed in the ANSYS Design Modeler environment, the right side of the bed is defined as inlet and the next expression is used for description of the velocity distribution (V_y) along the inlet height:

$$V_y = 4V_{av}/H_{in}\,[Y\,\mathrm{step}(H_{in}/2 - Y) + (H_{in} - Y)\,\mathrm{step}(Y - H_{in}/2)], \qquad (2)$$

where $V_{av} = 0.5(2gH_{in})^{\wedge 0.5}$ is an average value of the velocity in the inlet, H_{in} is an inlet water height, Y is a vertical coordinate (zero at the bed bottom, Fig. 4) and step is a function (step(x) = 1 when x > 0, step(x) = 0 when x < 0 and step(x) = 0.5 when x = 0).

Another mathematical model parameter, which should be clarified, is the initial VFRC distribution. As it is expected, clear separation between the volume part occupied by water, and the one occupied by air is not possible, then a homogenous fluid flow is accepted. In starting position the entire examination volume is accepted as filled with air, (air VFRC = 1), but at inlet the water height has to be defined. In this case this is achieved by setting the water VFRC ($VFRC_W$) and air VFRC ($VFRC_A$) along the height at the inlet portion (Fig. 4).

$$(VFRC_W) = \mathrm{step}(H_{in} - Y),$$
$$(VFRC_A) = 1 - (VFRC_W) \qquad (3)$$

The top surface of the investigated volume is defined as "opening", which means that the fluid can cross the boundary surface in either direction. An opening boundary condition is specified with

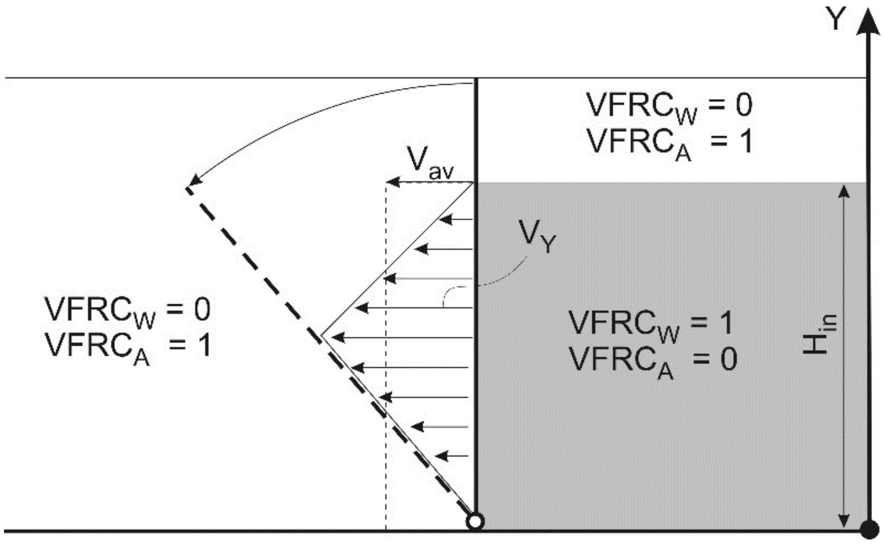

Fig. 4. Boundary conditions at the inlet.

a relative pressure value, interpreted as relative total pressure for inflow and relative static pressure for outflow. All other boundary surfaces are defined as walls. The roughness of the bottom, left and back side is assumed to meet that of the steel. The roughness of the front side is aligned with glass. The dam surface is assumed as “free slip wall”, which corresponds to the very low value of the coefficient of friction between fluid and dam material. The initial shape of dam corresponds to hydrostatic load (primary load of the solution procedure). The experimental equipment and geometry of the model with boundary conditions are given in Fig. 5.

RESULTS AND DISCUSSION

The solution provides a complete set of results concerning pressure, velocity and other basic characteristics of the flow. Here results only for water volume fraction, which is the basic validation parameter, has been discussed. The validation is performed by comparing the pictures of free flow surfaces from laboratory experiment and numerical simulation, created in the same moment. The video from laboratory experiment is created by 1080 p, 60 fps. The numerical solution is performed by 0.2 s time step and every time step is recorded.

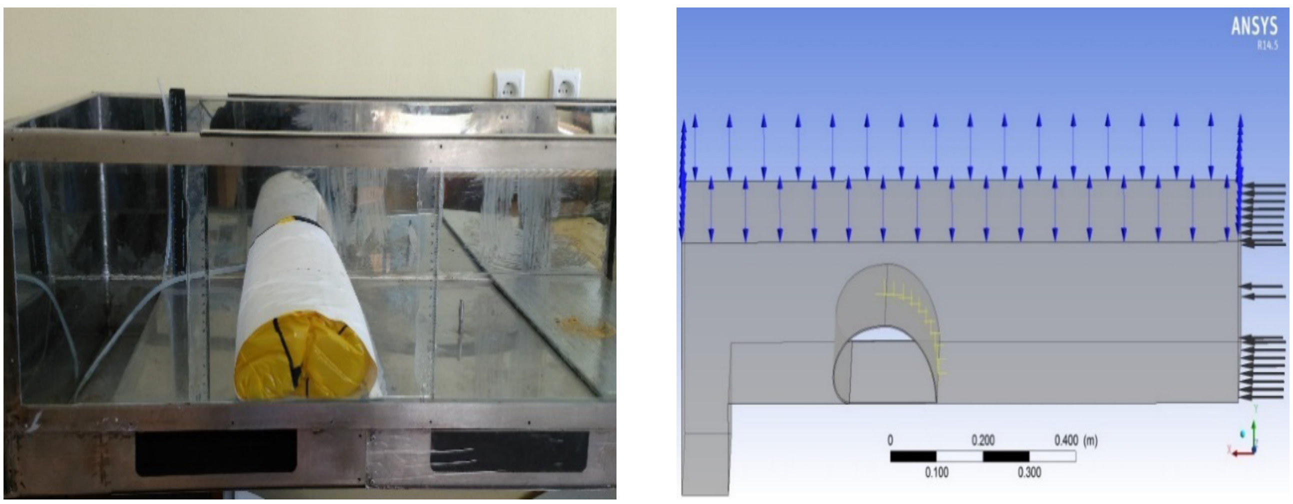

Fig. 5. Boundary conditions for the fluid area.

In Fig. 6 are given three basic pictures, which allow concluding, that results, obtained by numerical simulation are close enough to these, obtained by laboratory experiment.

The main purpose of the numerical solution is to obtain results for the mechanical behavior of the dam during pulse load of the stream and water overflow. The solution can be implemented in two steps – transient fluid flow analysis to obtain the influence of the water flow on the dam surface (pressure as a function of time) and transient mechanical analysis with load, obtained from fluid flow analysis. For validation the results for influence of the water flow on the dam surface, the overflow height of the flow over the balloon is accepted as numerical criteria for matching results. In Fig. 7 two curves are given in order to illustrate the results from experiment and numerical solution.

The curves coincide as values and in the first derivative, which allow concluding, that results are close enough.

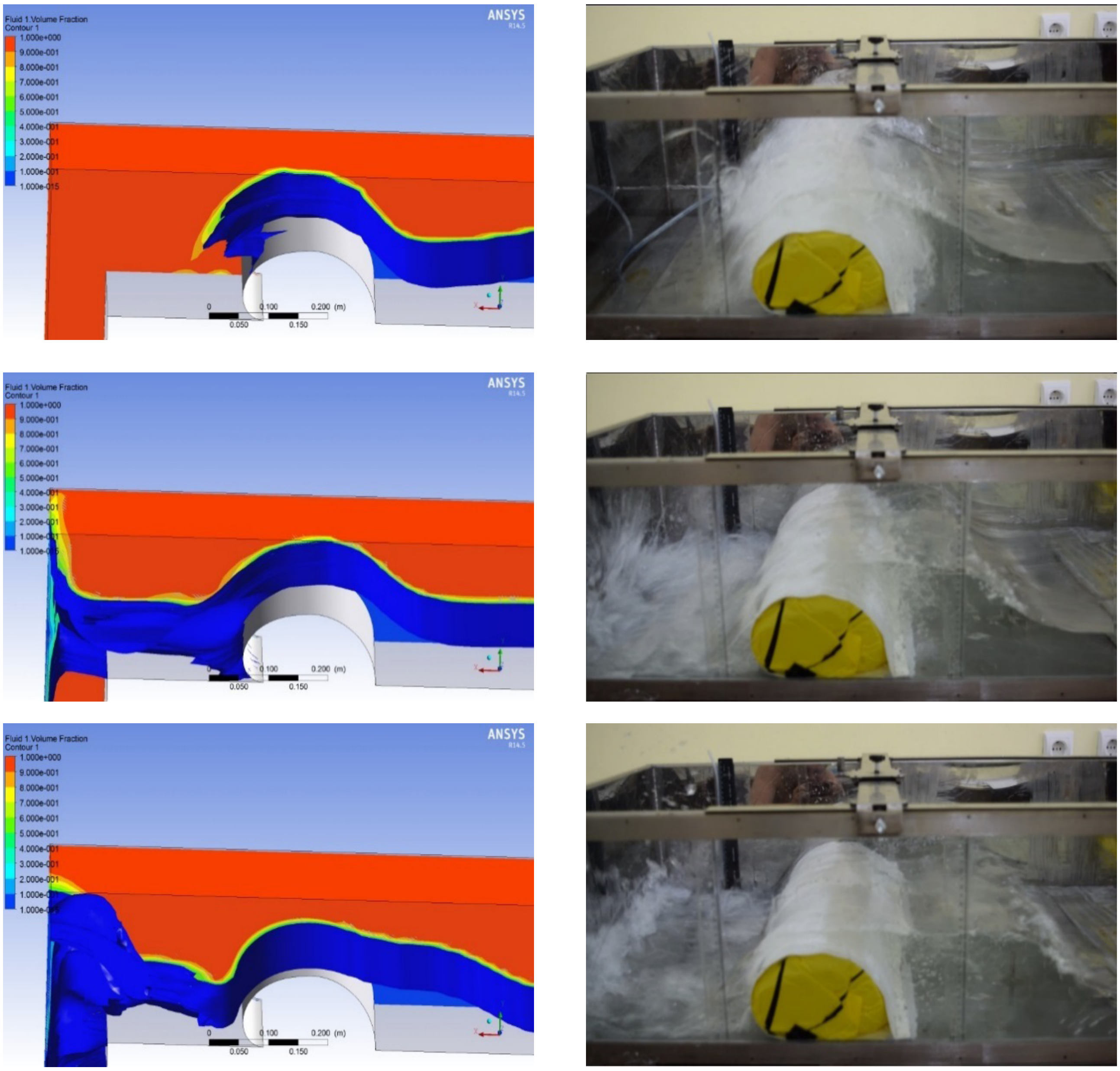

Fig. 6. Free surface and volume fraction distribution.

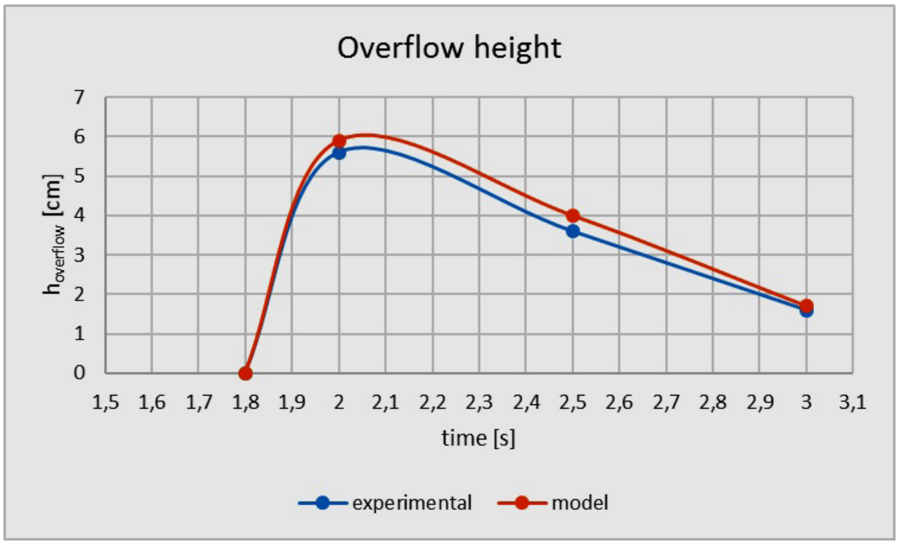

Fig. 7. Results for overflow.

CONCLUSIONS

The water flow over the inflatable dam and pulse load formation can be modeled by "volume of fluid" technique, using ANSYS CFX software. To do this, an adequate numerical model has to be created. The model creation process includes verification and validation procedures. The visual and numerical criteria can be used for the procedure for validating the results.

The visual criterion consists of comparing the pictures of free flow surfaces obtained from laboratory experiment and numerical simulation. It is shown that using appropriate mathematical model, boundary and initial conditions, the obtained results are close enough. For the numerical criterion can be accepted overflow height of the flow over the dam. The curves presenting the results from experiment and numerical solutions coincide as well as the first derivative, which allows concluding that both of the results are close enough. After validation the model can be used to determine the dynamic load on the dam for future strength calculations. For this purpose on the surface of the dam can be defined monitoring points (yellow crosses in Fig. 5) where the full pressure can be recorded as a function of time during the fluid solution, and after that used as a load during calculation of the mechanical behavior of the dam as a solid.

Acknowledgements

The financial support by the RSC of University of Chemical Technology and Metallurgy - Sofia, under contract 11394 "Validation a model of an inflatable dam" is gratefully acknowledged.

REFERENCES

1. A. Alwan, The Analysis and Design of Inflatable Dams. Ph.D. Thesis, University of Sheffield, Imagine Services North, 1979.
2. S. Diaz, J. Gonzalez, Hydraulic Effects of an Inflatable Rubber Weir Located Over a Wes Original Spillway Profile: Experimental and CFD Approaches. E-proceedings of the 36th IAHR World Congress 28 June - 3 July, 2015, The Hague, the Netherlands, http://89.31.100.18/~iahrpapers/81526.pdf
3. A. Alhamati at al., Behavior of inflatable dams under hydrostatic conditions, Suranaree J. Sci. Technol., 12, (1), 2005, 1-18.
4. P. Wu, R. Plaut, Analysis of the Vibration of Inflatable Dams Under Overflow Conditions, Thin-Walled Structures, Elsevier Science Ltd., 26, (4), 1996, 241-259.
5. K. Lowery, S. Liapis, Dynamic analysis of an inflatable dam subjected to a flood, Computational Mechanics, Springer-Verlag, 24, 1999, 52-64.

6. S. Choura, Suppression of structural vibrations of an air-inflated membrane dam by its internal pressure, Compurers & Structures, Elsevier Science Ltd., 65, (5), 1997, 669-677.
7. G. Mysore, S. Liapis, R. Plaut, Dynamic Analysis of single-anchor inflatable dams, J. of Sound and Vibration, 215, (2), 1998, 251-272.
8. M. Shafai-Bejestan at al., 3 Dimensional analysis of linear vibrations of the rubber dam, J. of Applied Sci., 9, (7), 2009, 1309-1316.
9. V. Iliev, Load forming of the load-bearing hydraulic membrane in initial period of fluid stream storage, In: Contemporary problems and ways for their solution in science, transport, production and education, SWorld, 32, 2010, 30-36, (in Russian).
10. N. Nikolov, V. Iliev, Load forming of the inflatable hydraulic membrane under pulse load of incoming water flow, E-Journal Mathematical Modeling and Computer Simulation, III, 7, 2015, 5-11.