NUMERICAL SIMULATION OF UNSTEADY FLOW DUE TO DAM BREAK

[1]SEYYED HASSAN SEYYEDPOUR, [2]YOUSEF HASSANZADEH, [3]DAVOOD FARSADIZADEH, [4]SEYYED MOHAMMAD YASIN ALAVI

[1]Student of Water Engineering Department of Water Engineering Faculty of Agricultural University of Tabriz
[2]Professor of Water Engineering Department Faculty of Civil Engineering University of Tabriz
[3]Associate Professor of Department of Water Engineering Faculty of Agricultural University of Tabriz
[4]Ms. Student of Environment Faculty of Civil Engineering Shahid Beheshti Univesity of Tehran
E-mail: shassanspour@yahoo.com, yhassanzadeh02@yahoo.com, Farsadi_d@yahoo.com, smyasinalavi@yahoo.com

Abstract- To analyze and fully understand the natural phenomena such as unsteady flow phenomena resulting from dam Break, developing of numerical models that can simulate the boundary conditions on the free surface flows with large displacement that can and also deepen our understanding of the physical details of the interaction process of dam defeat waves is of great importance. The purpose of this study is to analyze the hydrodynamic flow of the dam Break in vertical two-dimensional way using fluent software and solving Navier-Stokes equations. This is achieved by collecting and processing all the necessary information, the correct definition of the physical model is presented. Then using test and comparison, the optimized parameters of the network size, suitable turbulence model and the turbulence term (plan) was determined for different separation. Then, the values of the dimensionless wave front position with the passage of time and the values of dimensionless water depth and pressure in certain sections have been calculated. To evaluate the model and the equations used, the simulation results were validated with experimental results and other numerical methods in the literature. Comparison results show the high accuracy and reliability of the method used. Based on the findings of this study it was proved that Fluent Model has great capabilities in simulation of analysis and evaluation the phenomenon of dam Break. In addition, it has the possibility of adaptation to different conditions and limitations of different problems. And after creating a model in the software environment, the effects of following scenarios on it can easily be observed:

Effect of downstream slope of the dam in the reservoir on negative wave
The effect of input varying with time (hydrograph) on the negative wave created in the tank
The effect of the initial depth on the progress of flow created by flood at the time of Break
Two important parameters' influence the grid size and time step option on results
The effect of velocity gradients in the tank drain
Software inflow effect on results at 6 and 8 seconds
Effect of substrate roughness and depth of the coastal on the wave progress due to dam Break

Keywords- Dam Break Flows, Fluent, Simulation, Turbulence Model..., Barrier-free Flow

I. INTRODUCTION

Constant increase in the number of large dams is essential for human societies from social, economic and political interests' point of view (1). Despite applying high reliability in the design and construction of dams, due to the potential for extreme hydrological events caused by climate changes (2) and unpredictable nature of nature, the likelihood of floods with a high return period and also extreme earthquakes around the reservoir, the risk of Break like any other man-made installations is not ruled out (1). Besides, the problem of sedimentation in reservoirs in the reservoir is seen as the main problems in the operation and maintenance of dams in the world.

According to statistics from the annual assessment, sediment in the reservoir happens at a rate of 3/0 per cent in the world. However, this rate is higher in the Asian region and is approximately between 5/0 to 1 percent (3). Sedimentation in dam reservoirs in Iran, has created a challenge for engineers. To address this problem it is enough to remind that every year approximately 175 to 250 million cubic meters of capacity of reservoirs being utilized in country sediments up (4).

With this volume of sediment behind the dam, in case the dam broke, unsteady waves roaring due to dam Break would cause loss losses. (5) Resulting in fast changes in the geometry of the valley of the river bed downstream. These changes lead to a severe impact on the behavior of the wave- the wave forehead when reaching down will be more than maximum water level and flow properties of the surface.

To prevent or minimize the damage, exact knowledge of hydraulics and various methods that have been proposed based on the science of water is essential for engineering professionals. (6) Thus, in the past few decades, several mathematical models have been proposed to calculate the wave propagation speed, output hydrograph and hydraulic parameters of flood flows downstream at the dam breaking point (7). Because of analytical methods complexity, being time consuming and sometimes impossibility of being solved, difficulty of measuring the actual field

and the high cost of modeling complex phenomena in laboratory, numerical methods have found wide acceptance and application in numerical methods in the analysis of the current transmission from dam Breaks. Nowadays predictions of the wave resulting from dam Break often rely on numerical methods with complex data validation (6 and 5). And also due to the irregular geometry of channel of the natural river, only numerical methods can be justified. Calculations should be phased out of range for a longer period (6). On the other hand, when getting important information such as the speed and pressure distribution is difficult in the laboratory and requires a moment of great accurate equipment, numerical modeling is a matter of choice.

Numerical modeling of flow analysis regarding the methods used to solve the equations, ability to simulate one-dimensional or multi-dimensional flow, bed type and the boundary conditions have different kinds. And media and the boundary conditions are of various types. Flows resulting from dam Break is described using Navier-Stokes equations based on the continuity and momentum equations. Today, extensive efforts are made to verify the results of the numerical methods which has developed three numerical finite difference methods, finite volume, and implementation and has been limited. Among the models Fluent, DAMBRK and Flow3 can be named. With respect to the theoretical models applied to the model, simplifying assumptions were made to simulate varying degrees of accuracy. It should be noted that the theoretical basis of the model should be consistent with the location of the study. A brief look at the statistics of life and property damage caused by dam Break, shows the attention needed to be given to the issue by researchers.

Eaket and colleagues in 2005, in the laboratory, by using stereoscopy video system for unsteady flow measurements acquired the three-dimensional water structures and water flow rate depending on the occurrence of dam Breaks. Junqiang and colleagues in 2009 developed the two-dimensional Morpho Dynamic Model to predict the Break of the dam on the mobile platform. In this model two-dimensional normal equation models were modified, so observed the impact of sediment concentration and bed deformation in the flood wave propagation. Hong-ming Kao and colleagues in 2010, proposed a non-mesh numerical model for shallow water flows check in dam Break in open channels. This numerical model for shallow water solves equations (SWE) based on smooth particle hydrodynamics (SPH). Yang and colleagues in the 2011, in laboratory, studied the dams with reservoirs consecutive defeat in a glass flume with steep bottom slope. Laboratory results showed that the ratio of water depth of bottom of the reservoir to the depth of the upstream has a high impact on the flood at the downstream of the reservoir. Besides, a comparison between single reservoir and consecutive reservoirs showed differences in flow patterns of a dam Break.

There are various software products in the field of wave such as Flow 3D, Mike II and OPENFOAM software. Due to its functionality and capabilities of hydraulic calculation, we have used fluent software. Fluent Software is provided to the analysis of incompressible and compressible fluids on the basis of two and three-dimensional Stokes-Navier. The software doesn't allow the direct introduction of the waves of the parameters to the model, and to create a wave foreign functions of Fluent or UDFs must be used. UDFs are functions that can add to the software several features to the Fluent that are not included in the standard Fluent. These functions are implemented dynamically in iteration of the process of solving in the Fluent and apply special cases of the user to Fluent. For example, you can use these functions to determine the specific boundary conditions, the unique properties of a material, determine the specific source of the data used to initialize non-uniform field, and many other applications.

Repalle in 2007 by UDF Programming in Fluent modeled Stokes wave. Dong Zhi in 2009 using the same technique modeled Novydal wave in the channel.

In recent years the study of two-dimensional finite volume method in the presence and absence of the structure was carried downstream. Modeling of analysis of the Break of dam in 90 with the help of limited volume by Soares, Zech are at the stream of this field. In the meantime, in 2004, Abdolmaleki studied on the impact of shock wave resulting from dam Break with fluent software based on the finite volume method. In this article we have tried with the help fluent software and the use of numerical methods and solving the Navier-Stokes equations to simulate the positive wave caused by the Breaks of the dam. Fluent software is very powerful software with performance in a wide variety of fluid flow and heat transfer.

II. MATERIALS AND METHODS

A. MATHEMATICAL MODELING OF THE PROBLEM

a. DIFFERENTIAL EQUATIONS OF INCOMPRESSIBLE FLOW OF WATER AND NAVIER-STOKES EQUATIONS Non-constant fluid flow equations like the phenomenon of dam Break, originally are the Navier-Stokes equations which based on the principle of conservation of matter (continuity equation) and momentum (Newton's second law) with known variables which are the balance of water level on the basis of supposed water level and velocity components are established. These

equations for three-dimensional case and for an incompressible fluid can be
defined as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (1)$$

Equation (1) is the continuity equation form. It is noteworthy that u is the velocity component in the direction of X and V is the component of the velocity in the direction of Y and W is the velocity component in the direction of Z. below momentum equation of Navier-Stokes are mentioned.

$$\frac{\partial \bar{u}}{\partial t}+\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}+\bar{w}\frac{\partial \bar{u}}{\partial z}=\bar{X}-\frac{1}{\rho}\frac{\partial P}{\partial x}+\frac{1}{\rho}\frac{\partial}{\partial x}\left[\mu\frac{\partial \bar{u}}{\partial x}-\rho\bar{u}\bar{u}\right]+\frac{1}{\rho}\frac{\partial}{\partial y}\left[\mu\frac{\partial \bar{u}}{\partial y}-\rho\bar{u}\bar{v}\right]+\frac{1}{\rho}\frac{\partial}{\partial z}\left[\mu\frac{\partial \bar{u}}{\partial z}-\rho\bar{u}\bar{w}\right] \quad (2)$$

At the right side of equality, similar sentences $-\frac{1}{\rho}\frac{\partial}{\partial x}(\rho u'w')$ and $-\frac{1}{\rho}\frac{\partial}{\partial x}(\rho u'u')$ and $-\frac{1}{\rho}\frac{\partial}{\partial x}(\rho u'v')$ in the above equation $-(\rho u'u')$ and $-(\rho u'v')$ and $-(\rho u'w')$ is known as the Reynolds stresses that are similar to stresses which are slimy shears. These additional terms of average time obtained for a laminar for zero. In addition u are included in average speed and u' are part of the swing speed and the same thing is true for $w' and\ w\ and\ v' and\ v$. Equation (2) is the momentum equation for the flow in x direction and the momentum equation for the flow in the direction of Y and Z are as follows:

$$\frac{\partial \bar{v}}{\partial t}+\bar{u}\frac{\partial \bar{v}}{\partial x}+\bar{v}\frac{\partial \bar{v}}{\partial y}+\bar{w}\frac{\partial \bar{v}}{\partial z}=\bar{X}-\frac{1}{\rho}\frac{\partial P}{\partial y}+\frac{1}{\rho}\frac{\partial}{\partial x}\left[\mu\frac{\partial \bar{v}}{\partial x}-\rho\bar{v}\bar{u}\right]+\frac{1}{\rho}\frac{\partial}{\partial y}\left[\mu\frac{\partial \bar{v}}{\partial y}-\rho\bar{v}\bar{v}\right]+\frac{1}{\rho}\frac{\partial}{\partial z}\left[\mu\frac{\partial v}{\partial z}-\rho\bar{v}\bar{w}\right] \quad (3)$$

$$\frac{\partial \bar{w}}{\partial t}+\bar{u}\frac{\partial \bar{w}}{\partial x}+\bar{v}\frac{\partial \bar{w}}{\partial y}+\bar{w}\frac{\partial \bar{w}}{\partial z}=\bar{X}-\frac{1}{\rho}\frac{\partial P}{\partial z}+\frac{1}{\rho}\frac{\partial}{\partial x}\left[\mu\frac{\partial \bar{w}}{\partial x}-\rho\bar{w}\bar{u}\right]+\frac{1}{\rho}\frac{\partial}{\partial y}\left[\mu\frac{\partial \bar{w}}{\partial y}-\rho\bar{w}\bar{v}\right]+\frac{1}{\rho}\frac{\partial}{\partial z}\left[\mu\frac{\partial w}{\partial z}-\rho\bar{w}\bar{w}\right] \quad (4)$$

In this paper, the method of VOF fluid volume is used, this method is a technique of calculation at the applied level, to mesh fixed Olerin where the species of transport equations for determining the volumetric ratio of the two phases in each computational cell is used. Complete set of equations for the flow is as follows:

$$\nabla.u=0 \quad (5)$$

$$\frac{\partial \rho u}{\partial t}+\nabla.(\rho uu)-\nabla.\left((\mu+\mu_t)s\right)=-\nabla p+\rho g+\delta k\frac{\nabla\gamma}{|\nabla\gamma|} \quad (6)$$

$$\frac{\partial \gamma}{\partial t}+\nabla.(u\gamma)=0 \quad (7)$$

P stands for fluid pressure, μ_t for viscosity of turbulent flow. δ for surface tension. K for surface curvature and S is the amount of deformation of tensor which is expressed by$(\nabla u+\nabla u^T), S=\frac{1}{2}$.

b. BOUNDARY CONDITIONS

To solve the mathematical equations governing the flow in rivers and reservoirs baskets, boundary conditions must be specified. The flow is the data at the top or bottom border of the state of topography or geometry model. The more the input data is accurate the more the result taken from the model is accurate. Model boundary conditions are considered as follows: The lateral sides are of the Wall kind and the lower side also are of the wall kind, but also to consider and control the number Yplus the model in the calculation possess of these two defined separately to software. Upper side also considered as symmetric and the air flow dedicated to it.

c. MODELLING WATER FREE SURFACES

Other types of initial and boundary conditions used in this study include:
The clear velocity is related to the case that the Debbie is known (including zero Debbie or clear hydrograph). In the case that velocity or flow rate is zero, i.e. the walls are perpendicular to the direction of flow, and you can call the boundary condition, the anti-symmetry boundary. The Break of the dam at this project is considered as the inner boundary condition where the dam introduced as overtopping that its profile changes during the time and the Debbie through the dam is calculated.

Developed flow is assumed to be zero gradients in the direction perpendicular to the boundary or the boundary condition is assumed that the flow is uniform. It can be called a symmetry boundary condition. Fluid reservoir is defined by the weather. All rates in the tank at the initial moment are zero.

d. MODEL OPTIONS

The method of separation solving for incompressible flows has been used. Therefore, this method has been used in the simulations. Different turbulence models of k-RSM k- RNG k-standard have been used and finally the model k-RNG selected as the best model. Various leading the first order and second order power Law and Quick leading to suspension of sentences displacement equations and selected the best option for the other stages. PISO Algorithms for coupling coefficients of velocity terms and pressure is applied and the Coefficient of skewness has been considered two. To simulate the free surface of a two-phase model of dilated VOF, and unsteady and implicit mode is used. Since the goal is to simulate free surface flow channels open the item to adapt the model to activate and the acceleration of gravity is equal to y -9.806 m/s2 be considered.

e. ZHOU PHYSICAL MODEL BY DRAWING A DIAGRAM

Reservoir dimensions and elevation of water surface are assumed according to Zhou [8] physical modeling that shown in fig 1.

B. NUMERICAL METHOD

Following the Break of the dam and the flow is supercritical simultaneously, there is a Discontinuity.

The numerical method used to simulate the flow of the Break of the dam should be able ,which flows into the exact manner and with minimum loss peak phase error and amplitude of the numerical simulation.

C. FEATURES, BENEFITS AND IMPLICATIONS OF FLUENT SOFTWARE

As noted, in the fluent software finite volume method is used to solve the equations of fluid flow. The software includes complex user interfaces for data entry and analysis. The software consists of three main components which are: preprocessor, solver and post-processor. In pre-processor, in addition to modeling, determining the geometry, the network or mesh required for the tracking solution can be made. In the solver modeling process are crafted. And operating out takes place in the post-processor to help users

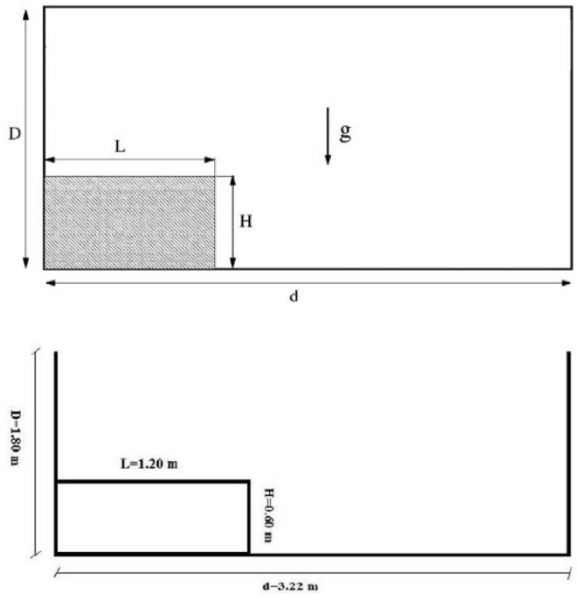


Fig1. Schematic sketch of the initial set-up for two-dimensional dam break

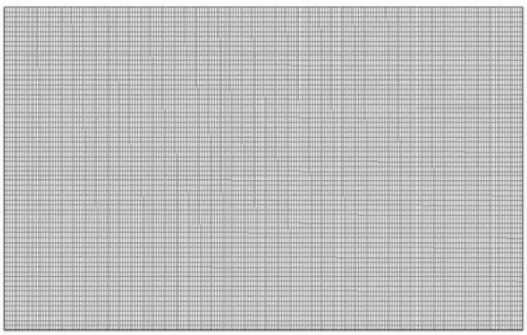


Fig 2. Schematic model of software GAMBIT

to solve the control condition. Finally, review the needed results. After modeling, the results of the software should be compared with the results of the pilot study. GAMBIT software as pre-processor is used for modeling. The model is built in this software and then the required mesh for model is acquired. Finally, the boundary conditions are enacted on the model. And the model is ready to be called to Fluent. Schematic model of software GAMBIT shown in Fig 2.

III. DISCUSSION AND ANALYSIS OF RESULTS

To evaluate the accuracy of the proposed model at all stages of development and completion, from actual physical model and (benchmark) study of research literature is used the most important of these three general categories are listed Figure 3 to 11.

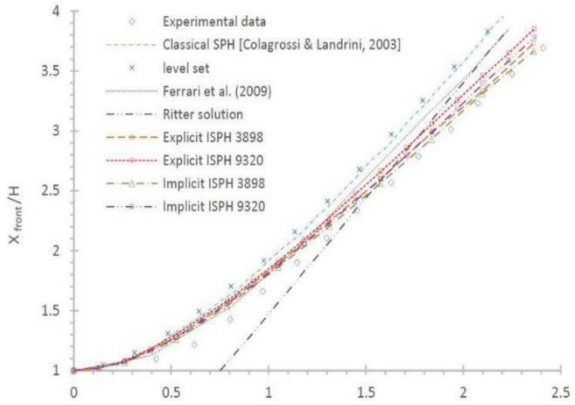


Fig 3. Time history of the surge front position

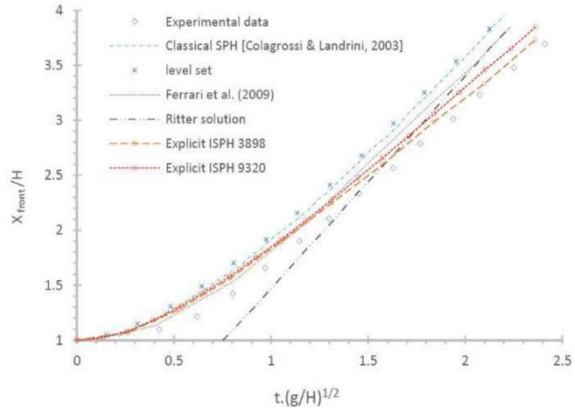


Fig 4. Explicit-Time history front position

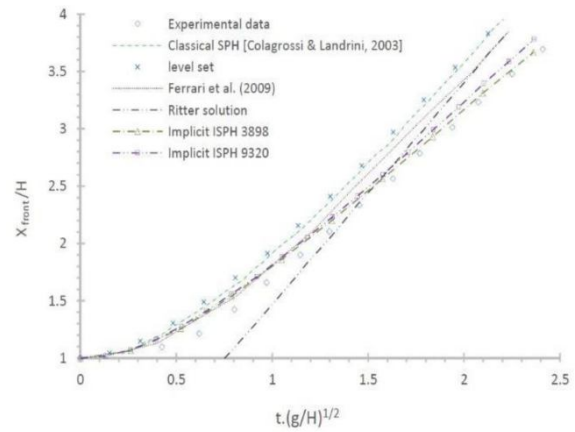


Fig 5. Implicit-Time history of the surge front position

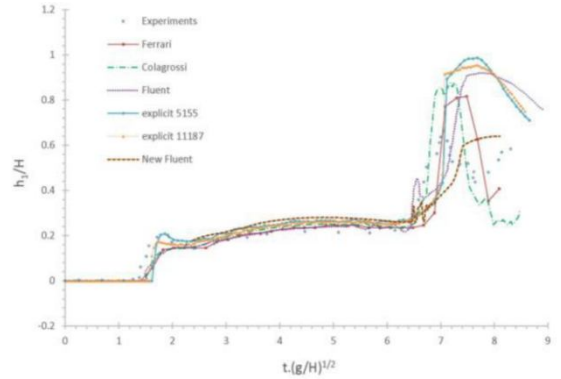


Fig 6. Explicit-Time evolution of the free surface height at location x=2.228m

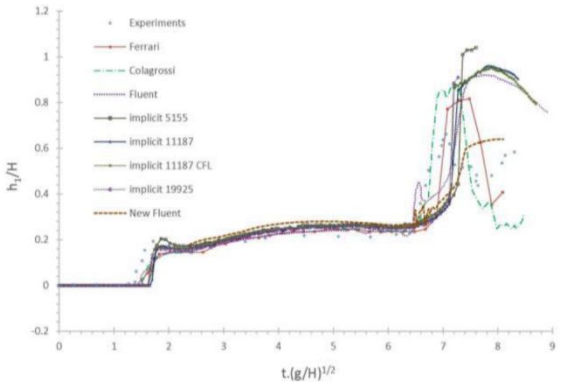


Fig 7. Implicit-Time evolution of the free surface height at location x=2.288m

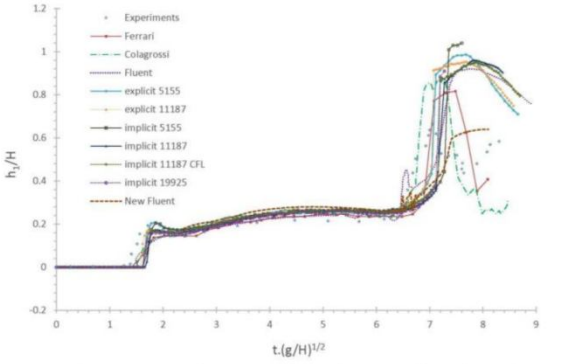


Fig 8. The time evolution of the free surface height at location x=2.228m

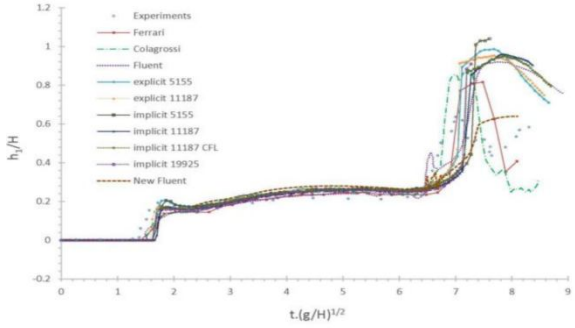


Fig 9. Explicit-Time evolution of the free surface height at location x=2.725m

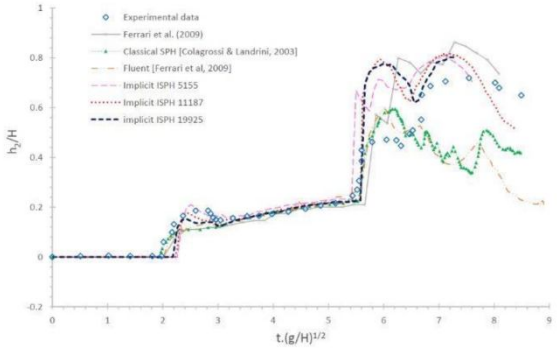


Fig 10. Implicit-Time evolution of the free surface height at location x=2.725m

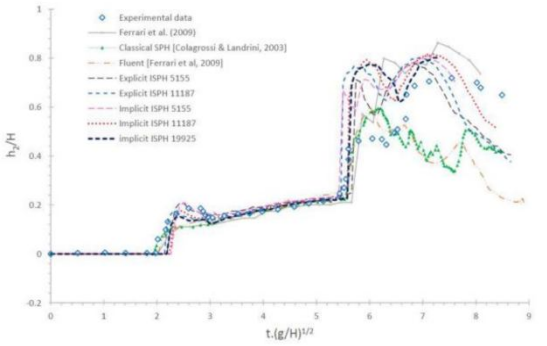


Fig 11. The time evolution of the free surface height at location x=2.725m

CONCLUSION

Potential consequences of dam Break and the subsequent massive surge of water is severe enough that sparing no effort to anticipate and reduction damages should not be. In order to minimize losses incurred, detailed knowledge of hydraulics different models have been provided based on this knowledge is essential. The early and the most important initiatives in this field, the output hydrograph reservoir flow simulation and forecasting by using mathematical models.

According to the nature of flow of the dam Break at high speed in the range of a very wide spread, using simulation models of completed and current transfers especially mathematical models is essential consider that in order to increase preventive measures, Consider the possibility of dam Break And the analysis of its features and factors affecting floodwaters Seek to reduction consequences and damages it managed.

REFERENCES

[1] Y. Hassanzadeh, "Hydraulic Dam Break," Department of Energy – National Committee on Large Dams in Iran , Issue No.61, First edition, Tehran. 2002

[2] Z. Cao, G. Pender, S. Wallis, P. Carling, "Computional Dam-Break Hydraulic Over Erodible Bed." ASCE, Journal of Hydraulic Eng., Vol. 130, No.7, pp.689-703, 2004.

[3] K. W. Olesen, G. Basson, "A 2D model for reservoir sedimentation and flushing," Conference of Hydraulics of Dams, Iran. 2004.

[4] M. Abbaspour, "Forecasting Pattern of Sedimentation in Reservoirs by Using Software Hec-Ras and Gstars3.0," M.s Thesis, Department of Civil Engineering, University of Tabriz, 2010.

[5] H. Chanson, "Analytical solution of dam break wave with flow resistance", Application to tsunami surges. In: B.H. Jun, S.I. Lee, I.W. Seo & G.W. Choi (eds.) 31st IAHR Congress, 11–17, Seoul, Korea. pp. 3341–3353. September 2005.

[6] M. Shafaei Bajestan, "Hydraulic Sediment Transport", Martyr Shahid Chamran University of Ahwaz, Second Edition, 2008.

[7] Y. Zech, B. Spinewine, "Dam-break induced floods and sediment movement-state of the art and need for research." In: Pro ceedings of the First Workshop of EU Project IMPACT, HR Wallingford, 16－17, 2002.

[8] ZQ. Zhou, J. Kat, B. Bunchner. "A nonlinear 3D approach to simulate green water dynamics on deck." Seventh international conference on numerical ship hydrodynamics; p. 1–4, 1999.

[9] K. Abdolmaleki, KP. Thiagarajan, MT. Morris-Thomas, " Simulation of the dam break problem and impact flows using a Navier-Stokes solver." 15th Australasian fluid mechanics conference; p. 1–4, 2004.

[10] A. Ferrari, M. Dumbser, EF. Toro, A. Armanini, "A new stable version of the SPH method in Lagrangian coordinates." Commun Comput Phys; 4: 378–404. 2008.

[11] M. Dumbser, M. Kaser, "Arbitrary high order finite volume schemes for seismic wave propagation on unstructured meshes in 2D and 3D." Geophys J Int; 171:665–94. 2007.

★★★