17th International Conference
ENGINEERING MECHANICS 2011
Svratka, Czech Republic, 9 – 12 May 2011

TURBULENCE MODELS FOR SIMULATION OF FLOW OVER WEIRS

M. Spano*, V. Stara*

Abstract: *Weirs belong to one of most common water structures. Various shapes of weirs are used at various conditions (e.g. emergency spillway, river weir, volumetric weir, etc.). For estimation of the weir capacity and determination of discharge coefficient the physical modelling is usually performed. Recently, numerical simulations take place here too. The main advantages of numerical simulation compared to hydraulic research are the volume of information gained, relatively lower price, and no scale effects. However, results from simulations are strongly affected by the computation setup. The choice of turbulence model belongs to most important options. Within this study the flow over weir was simulated with seven flow models. Those were inviscid, laminar and seven turbulence models. The aim was to find the most suitable one. All simulations were compared with results from hydraulic research.*

Keywords: ***Flow over weir, turbulence models, numerical simulations.***

1. Introduction

Simulations of flow were performed for a given 0.4 m height and 0.1 m thick weir. The problem was solved with commercial CFD software ANSYS-Fluent which uses a finite volume method (FVM). All achieved results were further compared with results from hydraulic research. Simulations were made for a flow rate of 0.029 $m^3.s^{-1}$ because the water head (above the weir crest) of 0.1 m was measured at this flow rate. Thus, the weir thickness and water head ratio is equal to 1.0. Compared were water and total heads above the weir crest measured 0.5 m upstream of the weir and pressures along the weir crest. Shape and dimensions of the weir are shown in Fig. 1.

Hydraulic research was performed in a 0.4 m wide channel. A model of the weir was made from plexiglass. Water levels upstream of the weir were measured with a point gauge. Pressures along the weir surface were measured using piezometric holes. Various sets of measurements were performed within the hydraulic research. Detailed description of the hydraulic research is summarized in Koutková & Stara (2003).

2. Models of flow, boundary conditions and basic domain

Nine 2D models of incompressible open channel flow were used for simulation of flow over the weir. Short overview of used modes is summarized in Tab. 1. Detailed description of each model can be found in user guide Fluent Inc. (2006). Water levels were reconstructed with the volume of fluid (VOF) method. Water was considered viscous (except of inviscid flow) with a constant density of $\rho_w = 998.2$ $kg.m^{-3}$ and viscosity $\mu_w = 1.003.10^{-3}$ Pa.s. The density and viscosity of air were also considered as constant.

Boundary conditions (BC) were set as a wall at the bottom, pressure outlet with zero gauge pressure at the outflow and at the top boundary. Velocity inlet BC was set up at the inflow as known velocity profile. In this case a uniform velocity of 0.184 $m.s^{-1}$ was used at lower 0.4 m of the boundary and 0.0 $m.s^{-1}$ at the rest part of the boundary. A detailed description of the boundary types and its application can be found in user guide Fluent Inc. (2006). The initial condition was set up such that a volume of fluid with a head $h = 0.100$ m was located at the crest of the weir. Fig. 1 shows a general layout of the problem.

* Ing. Miroslav Spano, Ph.D. and assoc. prof. Ing. Vlastimil Stara, CSc.: Water structures Institute, Faculty of Civil Engineering, Brno University of Technology, Veveri 95; 602 00, Brno; CZ, e-mails: spano.m@fce.vutbr.cz, stara.v@fce.vutbr.cz

Tab. 1: Overview of turbulence models used for simulation.

No	Mark	Description
1	inv	Inviscid flow
2	lam	Laminar flow
3	sa	Spalart-Allmaras model
4	ske	Standard k-ε model
5	rngke	RNG k-ε model
6	rke	Realizable k-ε model
7	skw	Standard k-ω model
8	lowrersm	Reynolds stress model with low Re stress-ω model
9	quadrsm	Reynolds stress model with quadratic pressure-strain model

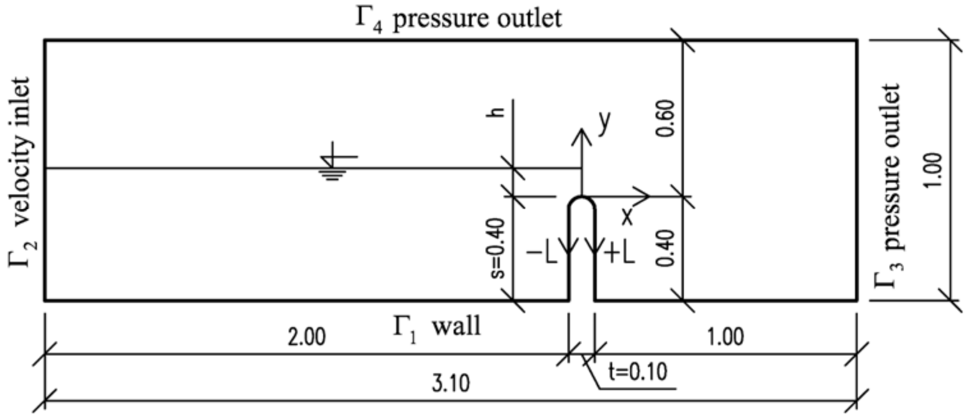

Fig. 1: A general layout of the problem and boundary conditions.

Triangle cells were used for discretisation of the domain. The grid size varied from 2 mm near the weir (at rounded section and downstream face) to 10 mm near the top boundary and in the basin upstream of the weir. The grid consist from 79 459 cells with 40 165 nodes.

A proper time step which provides converged results is a function of grid size. Time step value of $\Delta t = 1.10^{-4}$ s was chosen. According to the results in (Abdolmaleki et al., 2004 or Bhajantri et al., 2006), the value of the Courant number $C = 0.9$ or 0.7, respectively, should ensure a stable solution. The maximum Courant number across the domain here reached up to $C = 0.4$. A pressure based solver with the parameter setup recommended for this purpose was used, see Fluent Inc. (2006).

3. Results and discussion

Results of simulations describing the flow field around the weir are shown in Fig. 2.

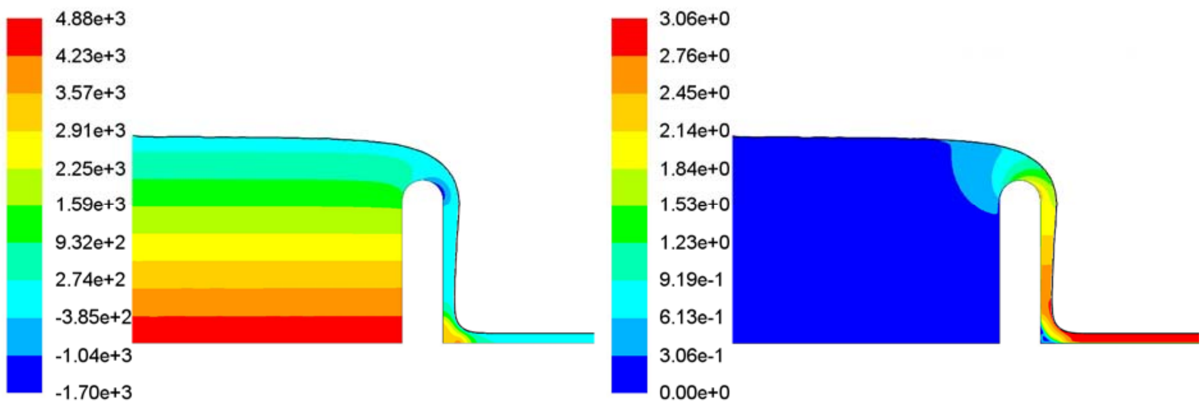

Fig. 2: Flow field around the weir - from left pressure [Pa], velocity [m.s^{-1}].

3.1. Total head

The total head was computed from equation (1):

$$h_0 = h + \frac{\alpha}{2g}\left[\frac{Q}{B\cdot(h+s)}\right]^2 \quad [\text{m}] \tag{1}$$

where Q is discharge $Q = 0.029$ m^3.s^{-1}, g – gravity acceleration $g = 9.81$ m.s^{-2}, h and h_0 –water and total head, respectively, above the weir crest [m], α – energy coefficient [-], B – channel width upstream of the weir $B = 0.40$ m, s – height of the weir $s = 0.40$ m.

The energy coefficient depends on the velocity profile and therefore slightly differs for each model. An analysis has shown that the value of energy coefficient is up to $\alpha = 1.08$ (1.05 for most models). A comparison of measured and computed total heads is summarized in Tab. 2.

Tab. 2: Summary of measured and computed total heads.

Variable		Measured	inv	lam	sa	ske	rngke	rke	skw	lowrersm	quadrsm
h	m	0.100	0.099	0.098	0.098	0.095	0.096	0.098	0.097	0.096	0.109
h_0	m	0.101	0.100	0.100	0.099	0.096	0.097	0.099	0.098	0.097	0.110
v_{mean}	m.s^{-1}	0.147	0.148	0.146	0.146	0.149	0.148	0.146	0.147	0.146	0.139
Re	-	$84\cdot10^3$	$84\cdot10^3$	$83\cdot10^3$	$83\cdot10^3$	$84\cdot10^3$	$84\cdot10^3$	$83\cdot10^3$	$83\cdot10^3$	$83\cdot10^3$	$79\cdot10^3$
α	-	1.05*	1.01	1.05	1.05	1.06	1.05	1.05	1.08	1.05	1.04

* *estimated value*

The results have shown a quite good agreement between the measurements and simulations. Most of models compute lower total head in order of millimetres in comparison with measurements. This is probably due to the assumption of a perfectly smooth wall (see Ho et al., 2003). In reality (on a physical model), some additional energy losses can occur. Also, the effect of the turbulent boundary layer and the wall function should be further investigated.

3.2. Pressure along weir surface

A comparison of pressures along the spillway surface is shown in Fig. 3. The pressures computed fit well with the data measured. However, some models (skw, ske, quadrsm) tend to underestimate the peak value of negative gauge pressure. Also, some deviation can be seen in the pressure values on the downstream face of the weir regardless the model used.

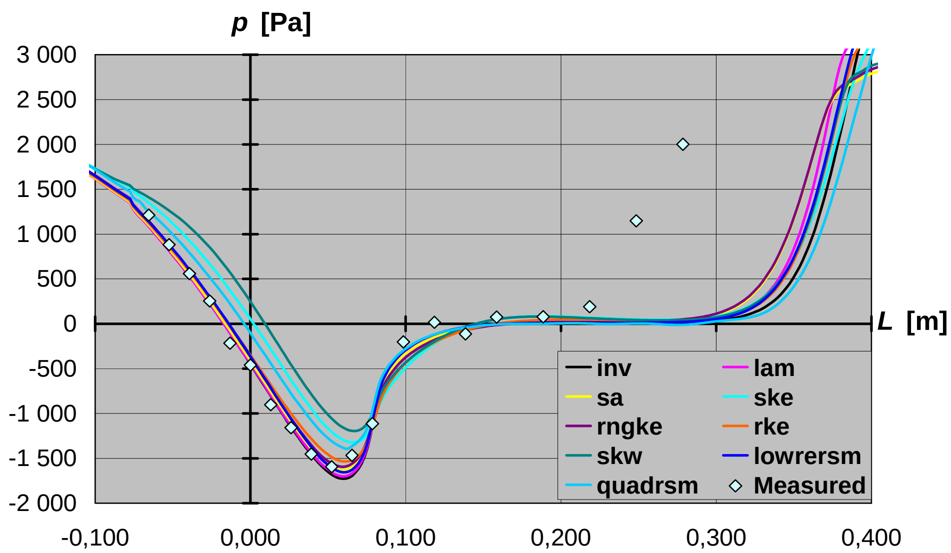

Fig. 3: A comparison of pressure distribution along the spillway surface; L – distance along weir surface [m], p – pressure (relative to atmospheric pressure) [Pa]; L = 0 at the weir crest.

3.3. Comparison

Water heads were compared upon relative error related to measured value according to equation (2). Pressure was compared by Pearson correlation coefficient computed from measured and simulated pressure values. Here the pressure values within interval -0.1 < L < 0.2 m were taken into account only. Results of comparison are summarized in Tab. 3.

$$r_h = 1 - \frac{|h_s - h_m|}{h_m} \text{ [-]} \tag{2}$$

where r_h is comparison coefficient for water head [-], h_s and h_m – simulated and measured water head, respectively [m].

Tab. 3: Comparison of turbulence models.

comparison coefficient for water head r_h	***measured***	***inv***	***lam***	***sa***	***rke***	***skw***	***rngke***	***lowrersm***	***ske***	***quadrsm***
	1.000	*0.982*	*0.982*	*0.979*	*0.973*	*0.964*	*0.958*	*0.957*	*0.946*	*0.917*
Pearson correlation coeff. for pressure	***measured***	***lam***	***inv***	***sa***	***lowrersm***	***rngke***	***rke***	***quadrsm***	***ske***	***skw***
	1.000	*0.995*	*0.994*	*0.994*	*0.992*	*0.991*	*0.987*	*0.973*	*0.940*	*0.910*

Comparison coefficients in Tab. 3 are sorted decreasingly. They shows that most accurate results of simulations related to measurements are given by Inviscid flow, Laminar flow, and Spallart-Almaras model.

4. Conclusion

Upon the above-mentioned results it can be concluded that most of the models used are suitable for prediction of the spillway capacity (and the discharge coefficient) of presented round-shaped weir.

Generally, the Inviscid and Laminar flow models provide very accurate information about the weir capacity, discharge coefficient values, and pressure along the weir surface. Probable reason is that the flow upstream is very slow (Reynolds number approx. 84.10^3) and the effect of turbulence is small here. O the other hand, the flow downstream of the weir is very fast (Reynolds number here is approx. 220.10^3) and the use of turbulence model for simulation is appropriate. Among turbulence models the most suitable for this case are Spalart-Allmaras model, Realizable k-ε model and RNG k-ε model.

Acknowledgement

This work has been carried out with a financial support of the Ministry of Education, Youth, and Sports of the Czech Republic, Project No. 1M0579, and the Czech Science Foundation, Project No. GA103/08/P538, and the internal foundation agency of Brno University of Technology, Project No. FAST-S-10-54.

References

Koutková, H. & Stara, V. (2003) Součinitel přepadu přelivu s kruhově zaoblenou korunou z fyzikálních experimentů [Coefficients of discharge for round-shaped weirs obtained from physical experiments], in *3. Vodohospodářská konference 2003 s mezinárodní účastí 2003*, Conference Proceedings, Brno University of Technology, Brno, 2003, pp. 379-387.

Fluent Inc. (2006) *FLUENT 6.3 User guide*, Fluent Inc., Centerra Resource Park, 10 Cavendish Court, Lebanon, NH 03766, 2006.

Abdolmaleki, K., Thiagarajan, K. P. & Morris-Thomas, M. T. (2004) Simulation of the dam break problem and impact flows using a Navier-Stokes solver, in *15th Australian Fluid Mechanics Conference-2004*, Conference Proceedings, The University of Sydney, Sydney, Australia, 2004.

Bhajantri, M. R., Eldho, T. I. & Deolalikar, P. B. (2006) Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model, *Sadhana-Academy Proceedings in Engineering Sciences,* Vol. 31, Part 6, pp. 743-754, 2006.

Ho, D., Boyes, K., Donohoo, S. & Cooper, B. (2003) Numerical flow analysis for spillways, in *43rd ANCOLD Conference 2003*, Conference Proceedings, ANCOLD, Hobart, Tasmania, 2003.