11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

# COMPARISON OF PERFORMANCE OF TURBULENCE CLOSURES IN FREE-SURFACE FLOWS PAST HYDRAULIC STRUCTURES

**Fabián A. Bombardelli[1], JoongcheolPaik[2] and Kenneth J. Loh[3]**

[1]Department of Civil and Environmental Engineering, University of California, Davis, 2001 One Shields Ave., Davis, CA 95616,fabianbombardelli2@gmail.com, http://cee.engr.ucdavis.edu/faculty/bombardelli/default.htm
[2]Department of Civil Engineering, Gangneung-Wonju National University, Gangwon, 201702, Korea; currently, Visiting Scholar at Department of Civil and Environmental Engineering, University of California, Davis, 2001 One Shields Ave., Davis, CA 95616, paik@gwnu.ac.kr
[3]Department of Civil and Environmental Engineering, University of California, Davis, 2001 One Shields Ave., Davis, CA 95616,kjloh@ucdavis.edu, HTTP://NESST.ENGR.UCDAVIS.EDU/

**Key Words:** *Turbulence, Free surface, Open channel, Numerical Modeling, Hydraulic Structures.*

The flow past cylinders piercing the water free surface, or past sluice gates, exhibits interesting features in the vicinity of such structures. The former flow presents a run-up in the front and a depression around the sides and at the back and its vortical structures strongly interact with the free surface as well as bottom wall at high Froude numbers.The free surface variation and the corresponding pressure gradient result in a remarkable downward flow. The latter flow shows complex three-dimensional flow features including eddies of vertical axes at both sides of the channel in the upstream and the downstream of the gate.The interactions of turbulent vortical structures with the free surface are numerically investigated for a turbulent flow past a circular cylinder in a rectangular channel, which was experimentally investigated by Graf and Yulistiyanto [1], at a Reynolds number of $1.47\times10^5$ and a Froude number of 0.5.In the case of the flow passing a gate, numerical simulations are conducted to reproduce the experiments investigated by Albayrak et al. [2] at the Reynolds number of 33,500.The governing equations are discretized by the second-order-accurate finite volume methods both in space and in time. The turbulent flowsare modeled using the $k$-$\omega$ shear stress transport (SST) model and the scale-adapted simulation (SAS)approach based on the $k$-$\omega$ SST model. A two-phase volume of fluid technique is employed to simulatethe free surface variation. The numerical results show that both numerical approaches are able to capture reasonably well the mean flows around the hydraulic obstacles observed in the experiments, while some details of vortical structures with intense unsteadiness and their corresponding turbulence quantities are reproduced only by the SAS approach in good agreement with the measurements.

## REFERENCES

[1] W.H. Graf and B. Yulistiyanto,Experiments on flows around a cylinder: the velocity and vorticity fields.J. Hydraul. Res., Vol. **36**(4), p 673 (1998).
[2] I. Albayrak, E.J. HopfingerandU. Lemmin,Near field flow structure of a confined wall jet on flat and concave rough walls. J. Fluid Mech., Vol.**606**, 27 (2008).