SUPERCRITICAL TURBULENT FREE-SURFACE FLOW OVER A SEMICIRCULAR OBSTACLE

EVAN KOUTANDOS [1] and PANAYOTIS PRINOS [2]

[1] Post Doctoral Fellow, Hydraulics Lab., Dept. of Civil Engineering, Aristotle University of Thessaloniki, Thessaloniki 54124, Greece,
(Tel: +30-2310-995856, Fax: +30-2310-995672, e-mail: ekoutant@civil.auth.gr)
[2] Professor, Hydraulics Lab., Dept. of Civil Engineering, Aristotle University of Thessaloniki, Thessaloniki 54124, Greece,
(Tel: +30-2310-995856, Fax: +30-2310-995672, e-mail: prinosp@civil.auth.gr

In the present study supercritical free-surface turbulent flow over a semicircular obstacle is studied numerically and experimentally in a smooth open channel of rectangular cross section. Both upstream and downstream of the obstacle, the flow is supercritical for all cases examined. A comparison between numerical and experimental free surface variation is presented, for four different Froude numbers, varying between 1.92 and 2.44 and Reynolds numbers between 38000 and 45000. Numerical results are also presented, concerning the distributions of the computed mean velocity and turbulent kinetic energy around the obstacle are presented, revealing some interesting features of the mean and turbulence flow structure.

The two dimensional RANS equations together with the continuity are solved using the finite volume method with the FLUENT 5.5. The RSM of Launder et al (1972) is used for calculating the turbulent stresses by solving the transport equations for the respective stresses. In addition a modified volume of Fluid (VOF) method Hirt & Nichols (1981) is employed for computing the free surface.

Continuity equation and transport equations of velocities, turbulence quantities and volume fraction were solved in the computational domain with appropriate boundary conditions. At the inlet boundary the water depth was h_{inlet} while the whole domain was extended up to two times the h_{inlet}. The total length of the computational domain in all cases was 1 m. At the water part of the boundary velocity inlet conditions were imposed (fully developed logarithmic velocity profile and the Reynolds stresses profiles according to the semi-empirical relationships developed by Nezu and Nakagawa (1993) and at the air part symmetry conditions were imposed. At the outlet boundary the water depth was set equal to h_{inlet} and outflow boundary conditions were imposed while symmetry conditions were imposed at the air part. The height of the obstacle h_l was 3 cm while the width, B, was equal to 6 cm.

Four cases with different Fr numbers (based on the mean velocity and water depth at the inlet) were examined numerically and experimentally. At the water inlet the water depth was set equal to 4.6, 4.2, 4.2 and 3.9 cm, while the respective Fr numbers were found equal to 1.92, 2.18, 2.27 and 2.44 and the Re numbers (based on the mean velocity and water depth at the inlet) were varied between 38730 and 45300. The vertical distributions of the velocity and the Reynolds stresses at the specific boundary were set using UDF subroutines in C++6 according to the semi-empirical equations proposed by Nezu & Nakagawa (1993) for fully developed open channel flow. Several grids have been employed to test the grid dependency of the results. It is found that the same results are even if a coarser grid is used. This is not the case for the turbulent quantities for which the grid dependency is stronger.

The experimental study was conducted in the Hydraulics laboratory of the Department of

Civil Engineering, AUTh. The dimensions of the tilting open channel were 12 m length, 0.25 m width and 0.5 m height. The water depth in the downstream part of the flume was regulated using properly made iron wings while at the entrance of the flume a smoothing filter was used. The measurements of the free surface were conducted using point gauges with an accuracy of ± 0.01 mm. All measurements were conducted at the centerline of the channel along the main flow direction. The obstacle, made of stainless steel, was placed at seven meters distance from the channel inlet in a region of fully developed incoming flow. For the measurement of the discharge a triangular weir was used at the end of the channel.

Numerical and experimental free-surface variation in the region of the obstacle is presented in figure 1 for all cases examined. The curvature of the free surface is observed mainly in the regions upstream and over the obstacle depending on the Froude number. In all cases examined, the dimensionless diagrams present a water depth increase with a maximum value of 1.5 times the inlet water depth. As it was observed in the experiments the renormalization of the water depth downstream of the obstacle was achieved in a distance greater than 3 times the obstacles width. Numerical results agree fairly well with the corresponding experimental concerning the free-surface shape, in all cases examined, revealing the satisfactory performance of the VOF method, in open channel flows with free surface curvature.

As it was found in experiments and computations, the renormalization of the water depth downstream of the obstacle was achieved in a distance greater than 3 times the obstacles width. Numerical results indicate the existence of a recirculation bubble in the downstream part of the obstacle for all Fr numbers examined. It is concluded that the renormalization of the velocity starts earlier than the corresponding one of the turbulent kinetic energy.

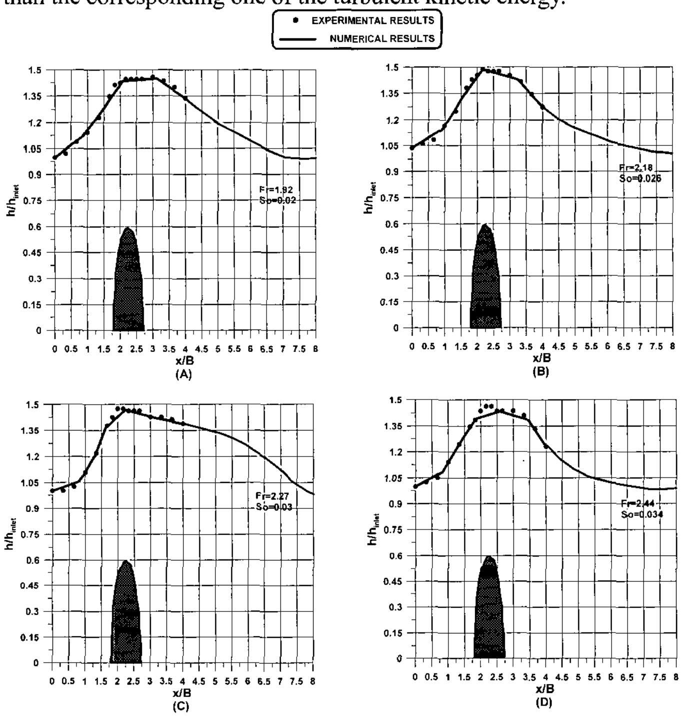

Fig. 1 Free surface variation in the region of the obstacle