COMPUTATION OF FLOW PAST THE IRREGULARITIES IN A CONDUIT

YAKUN GUO[1, 2], PINGYI WANG[1] and HUAJUN ZHOU[1]

[1]School of River and Ocean Engineering, Chongqing Jiaotong University, Chongqing, 400074, China
[2]Department of Engineering, University of Aberdeen, Aberdeen, AB24 3UE, UK.
(e-mail: y.guo@abdn.ac.uk)

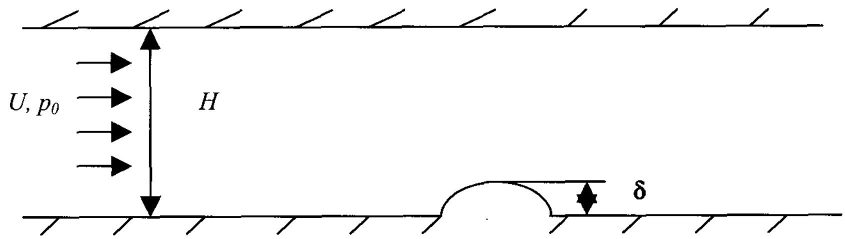
Cavitation damage often occurs in hydraulic structures, such as tunnels and spillways, as the high-speed fluid flows through. In engineering practice we can take some appropriate measures to reduce and avoid cavitation damage. One of the measures is to control the surface irregularities in structures. Previous experimental studies show that the cavitation number is related to the surface irregularity of structures. Therefore, some investigators proposed the demand of surface irregularity or the limit to the discharge per unit width of flow. In construction of hydraulic structures, however, it is impossible and uneconomic to build the concrete surface as smooth as a mirror surface. On the other hand, if we can compute the incipient cavitation number C_p of surface irregularity and control C_p during constructing so that it is smaller than cavitation number of flow, then the cavitation damage can be reduced and avoided during the operation.

The incipient cavitation number of surface irregularities has been studied numerically (Hsu and Ouyang, 1963, Guo and Wen (1993) and experimentally (Zhou et al, 1984). Over the last two or three decades, numerical models have been increasingly used to simulate the fluid flow due to the advantage of cheap, non-intruded the flow domain, transportability and scaling (Falconer and Lin, 2005). In this study we will use a CFD software package – FLUENT (2000), to carry out some modelling calculations for flow around surface irregularities in a pressure conduit. FLUENT is a powerful CFD package and has been widely used to simulate the complex fluid flow, heat transfer and chemical reaction (Karim and Ali, 2000). The problem in question is sketched in Figure 1. The incipient cavitation number C_p is defined as (for small irregularities $\delta/H << 1$):

$$C_p = \frac{p - p_0}{\rho U^2 / 2} = -\frac{v^2 - U^2}{U^2} \tag{1}$$

where p_0 and U are the undisturbed pressure and velocity far upstream of the irregularity, respectively, p and v are the pressure and velocity on the irregularity. For other symbols refer Fig 1.

Different shapes and sizes of irregularities are tested against the various flow conditions in order to verify the effectiveness and suitability of FLUENT in simulating such fluid flow. The modelling results are (i) velocity distribution around the irregularities, (ii) pressure distribution, and (iii) incipient cavitation number C_p of irregularities. Figure 2 is a typical example of the comparison between the numerical modelling results and experimental data of incipient cavitation number. It is seen that the simulated results are in good agreement with the available experimental data.



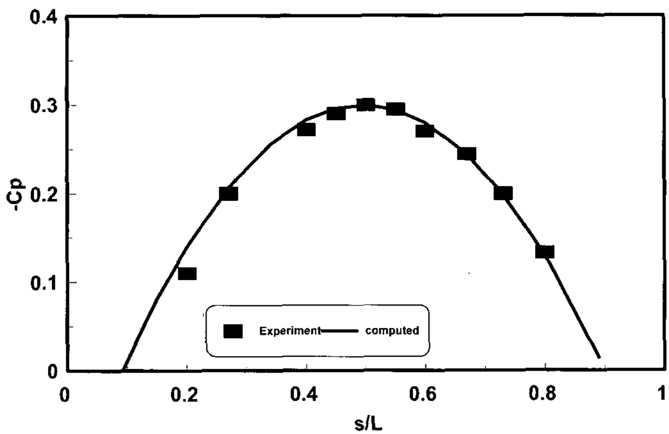


Fig. 1 sketch of the flow around the surface irregularity

Fig. 2 Comparison of computed and experimental incipient cavitation number on an arc irregularity

REFERENCES

Falconer, R.A. & Lin, B., 2005. Water quality management of river and estuarine waters, In J. H. W. Lee & K.M., Lam (eds), Environmental Hydraulics and Sustainable Water Management:45-55. Taylor & Francis Group, London.

FLUENT Incorporated, 2000. Fluent User's Manual, vol:1-4.

Guo, Y.K. & Wen, X., 1993. A boundary integral equation and its numerical solution of potential flow around surface-irregularities. Journal of Hydrodynamics, Ser. B, 3(1): 70-81.

Hsu, H.C. & Ouyang, L.X., 1963. The pressure distribution along boundary with circular arcs, Chinese Journal of Hydraulic Engineering, No: 4. (in Chinese)

Karim, O.A. & Ali, K.H.M., 2000. Prediction of flow patterns in local scour holes caused by turbulent water jets. Journal of Hydraulic Research, 38(4):279-287.

Zhou, S., *et al.*, 1984. Pressure distribution and incipient cavitation number of surface irregularities with rounded corners in a pressure conduit, Chinese Journal of Hydraulic Engineering, No:6. (in Chinese).