FIFTH INTERNATIONAL SYMPOSIUM ON CAVITATION

November 1-4, 2003, Osaka, Japan

Workshop on physical models and CFD tools for computation of cavitating flows

proposed by Jean-Pierre FRANC (Jean-Pierre.Franc@hmg.inpg.fr)
and Guenter H. SCHNERR (schnerr@flm.mw.tu-muenchen.de)

Scope of the session

After the Fourth International Symposium on Cavitation (Pasadena, CA, June 2001) where several physical models and CFD tools were presented, it appeared of interest for our scientific community to better know the features of the various physical models available for computation of cavitating flows.

The proposed session is a first attempt to present these models in a unified way and to start a comparison of the computational results on a common test case. In a first step, we have deliberately chosen a very simple two-dimensional configuration for which experimental data are not yet available. Therefore, this session does not aim at validating tools against experiment but it simply offers a forum of discussion on existing physical and numerical cavitation models.

Modeling techniques

All techniques of calculation of cavitating flows including basic and simple methods and all assumptions are welcome, provided they are complete and clearly presented. The fluid may be considered as viscous or not, the flow as irrotational or not, steady or unsteady, laminar or turbulent…

Geometry

The proposed test case consists in a two-dimensional symmetrical foil set in the middle of a horizontal solid wall channel as shown below. For simplicity, gravity effects are to be neglected.

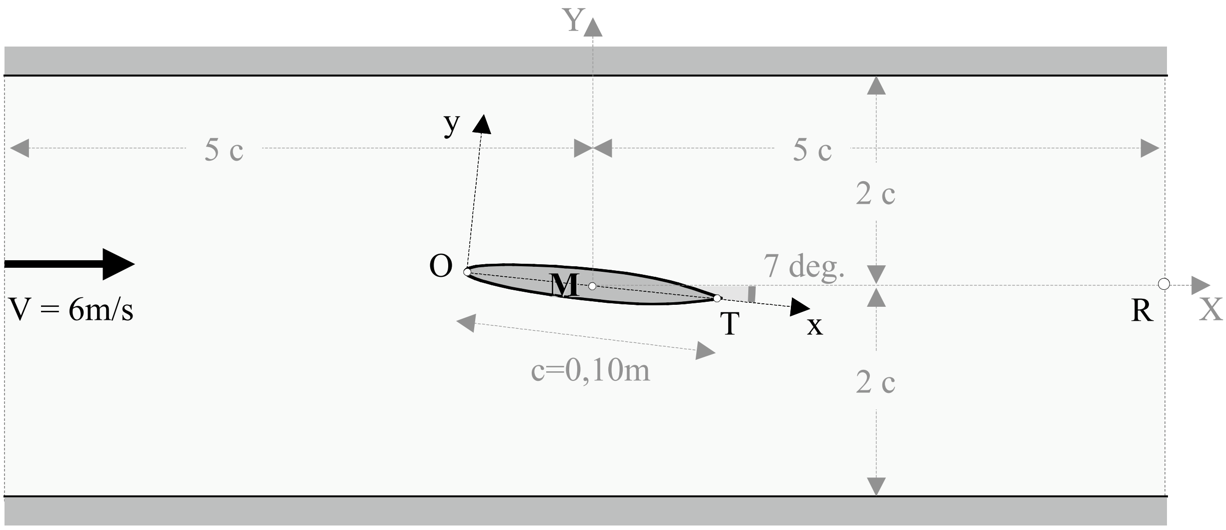

Figure 1

In the frame of reference Oxy of the foil, the upper side is defined analytically by the equation:

$$\frac{y}{c} = a_0\sqrt{\frac{x}{c}} + a_1\frac{x}{c} + a_2\left(\frac{x}{c}\right)^2 + a_3\left(\frac{x}{c}\right)^3 + a_4\left(\frac{x}{c}\right)^4$$

with the following values of the constants a_0 to a_4:

$$\begin{cases} a_0 = \ \ 0.11858 \\ a_1 = -0.02972 \\ a_2 = \ \ 0.00593 \\ a_3 = -0.07272 \\ a_4 = -0.02207 \end{cases}$$

The chord length c = OT of the foil is 0.10 m. The foil is set in the middle M of a channel of length 10 c and height 4 c. The angle of attack is 7 degrees. The foil is symmetrical with respect to the centerline, i.e. to the x-axis.

The point of maximum thickness is located at midchord M. The maximum thickness is 12% of the chord length. The thickness at the trailing edge is zero. The radius of curvature at the leading edge O is 0.703% of the chord length. The slope at the trailing edge T is $\left|\frac{dy}{dx}\right| = 0.265$. All this information is already included in the foil data.

The flow velocity V at inlet is uniform and equal to 6 m/s. The cavitation parameter at the reference point R is defined by:

$$\sigma_R = \frac{p_R - p_v}{\frac{1}{2}\rho_\ell V^2}$$

where p_R is a reference pressure at the downstream point of reference R (cf. Figure 1) and p_v the vapor pressure. The fluid is water at 20°C, ρ_ℓ is the liquid density, ρ_ℓ = 1000 kg/m^3 and p_v = 2329 Pa. The turbulence level at the inlet is Tu = 0.1%. For clarification, the reference pressure is not constant in the entire outlet cross sectional area, but only at the reference point R.

Flow conditions

Three different operating conditions are proposed:

- condition #0: no cavitation
- cavitation condition #1: $\sigma_{R1} = 0.8$
- cavitation condition #2: $\sigma_{R2} = 0.4$

Quantities of interest

When possible, the following quantities will be determined (cf. Figure 2):

- lift and drag coefficients per unit meter span $C_L = \dfrac{\text{Lift Force}}{\frac{1}{2}\rho_\ell V^2 c}$ and $C_D = \dfrac{\text{Drag Force}}{\frac{1}{2}\rho_\ell V^2 c}$ (for non-cavitating and cavitating conditions)
- distribution of pressure coefficient $C_p = \dfrac{p - p_R}{\frac{1}{2}\rho_\ell V^2}$ as a function of reduced abscissa x/c on the upper side and the lower side (for non-cavitating and cavitating conditions)
- reduced abscissa of cavity detachment $\bar{\ell}_d = \ell_d / c$ (ℓ_d measured from the leading edge O along x-axis)
- (maximum) reduced cavity length $\bar{\ell}_{max} = \ell_{max} / c$ (measured from cavity detachment along x-axis)
- maximum reduced cavity thickness $\bar{t}_{max} = t_{max} / c$ (t_{max} measured along y-axis)
- reduced position of maximum cavity thickness $\bar{\ell}_{t\,max} = \ell_{t\,max} / \ell_{max}$ ($\ell_{t\,max}$ measured from the detachment point along x-axis)
- cavity interface and/or field of vapor volume fraction: for identification of the cavity shape within the field of vapor volume fraction, a threshold minimum volume fraction of $\alpha = 10\%$

should be assumed (the vapor volume fraction is defined as the ratio of the volume of vapor to the total volume of liquid and vapor)

- the integral total volume of vapor $\mathcal{V}$ per unit meter span in 10^{-4} [m^3], in the computational domain.

- Strouhal number $S = \frac{f\ell_{max}}{V}$ (if a periodic behavior of the cavity appears at a frequency f)

Time averaged values will be considered in priority. However, in case of unsteady cavitation the time dependent variation of all quantities are of interest, especially of C_L, C_D, C_p and of the vapor volume variation, and should be presented together with time averaged values.

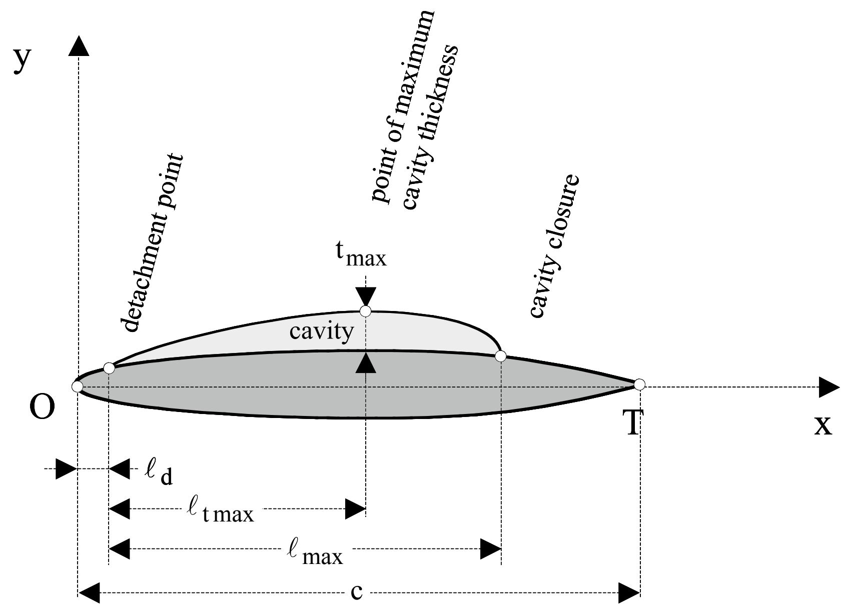

Figure 2

Paper content

The paper should contain the following information (if applicable):

- detailed presentation of the physical model with all model parameters, governing equations (including boundary and initial conditions) and unknowns
- numerical procedure
- time step, grid size, computational parameters, computational time, CPU time (grid studies with systematic refinements would be appreciated)
- computational results for conditions #0, #1 and #2 and discussion.

Thank you for your contribution !