

# Development of numerical model in the RANS solver for 3D cavitation simulation on marine propeller

Keun Woo Shin (kws@mek.dtu.dk) Dept. of Mechanical Engineering, Technical University of Denmark

## 1 Introduction

Although cavitation is mostly an undesirable occurrence in high-speed liquid flows of various engineering devices like ship propellers, pumps, hydraulic systems etc, due to such negative effects as noise, vibration, erosion and performance degradation, the demand of industry for heavier loads makes it unavoidable. As computational fluid dynamics becomes common, the capability for numerical simulation of cavitating flows is of critical importance for efficient design of high-performance engineering devices. To develop a reliable numerical model, several cavitation models are implemented in EllipSys, an incompressible Reynolds-averaged Navier-Stokes (RANS) solver, which is developed in cooperation of the Department of Mechanical Engineering at DTU and the Department of Wind Energy at RISØ [5,8].

## 2 Numerical model

We adopt the homogeneous equilibrium modeling (HEM) approach, which handles the two-phase mixture as a single fluid with variable fluid properties, averaged on a volume fraction basis as follows

$$\rho_m = \alpha_v \rho_v + (1 - \alpha_v) \rho_l, \quad \mu_m = \alpha_v \mu_v + (1 - \alpha_v) \mu_l \quad (1)$$

where  $\rho$  is the density,  $\mu$  is the dynamic viscosity, the subscripts  $m, l, v$  indicate the mixture, liquid and vapor, respectively,  $\alpha_v$  is the vapor volume fraction,  $\alpha_v \in [0, 1]$ .

A single set of the RANS equations for the mixture fluid is solved as follows

$$\frac{\partial}{\partial t}(\rho_m u_i) + \frac{\partial}{\partial x_j}(\rho_m u_i u_j) - \frac{\partial}{\partial x_j} \left[ (\mu_m + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial p}{\partial x_i} = 0, \quad \frac{\partial u_j}{\partial x_j} = -\frac{\dot{m}}{\rho_v} \quad (2)$$

where  $\dot{m}$  is the mass transfer per unit volume between two phases.

The phase change is based on either a vapor transport equation or a barotropic state law. We implement three cavitation models (Model 1 [9], 2 [7], 3 [3]) with a transport equation and a model (Model 4 [1]) with a barotropic state law. For example, the vapor transport equation in [9] is

$$\frac{\partial}{\partial t}(\rho_v \alpha_v) + \frac{\partial}{\partial x_j}(\rho_v u_j \alpha_v) = -\dot{m}, \quad \dot{m} = \begin{cases} -C_e \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_l}} (1 - \alpha_v) & \text{for } p < p_v \\ C_c \sqrt{\frac{2}{3} \frac{p - p_v}{\rho_l}} \alpha_v & \text{for } p > p_v. \end{cases} \quad (3)$$

## 3 Numerical result

At first, the computation for a 2D NACA66(mod) hydrofoil is made with four cavitation models. In the distribution of the pressure coefficient  $C_p$  on the suction side for the steady leading-edge cavitation (Fig.1(a,c)) and transient mid-chord cavitation (Fig.1(b,d)), the result shows a reasonable agreement with the measurement [6], but for the leading-edge cavitation, the cavity length from the computation is shorter than that from the experiment and the  $C_p$  distribution behind the cavity differs. The solution from Model 1 is more stable and its result is closer to the measurement, compared to the other models, and hence we proceed to 3D computations with Model 1.

The computation is made for the 3D NACA16-206 hydrofoils of non-swept (Fig.2(a)) and swept (Fig.2(b)) forms. The results show reasonable agreements in the vapor distribution, but the cavity extends less than the measurements [10] and cloud cavitation is not generated.

The computation is made for a conventional propeller in a uniform inflow with a rotating polar coordinates. For the weak cavitation (Fig.2(c)), the result show good agreements in the vapor distribution, but the tip vortex cavitation does not extend like in the experiment [4] due to the coarse grid some distance from the blade surface. The result from the computation in a uniform inflow is compared with that from the experiment in a behind-hull condition (Fig.2(d)). The advance ratio  $J$  corresponding to the wake peak is applied to the computation. The cavity from the computation extends more radially, but less chordwise compared to the experimental result.

## 4 Conclusion

The cavitation model in a RANS solver is proved to be accurate and robust, to some extent, for the steady and unsteady sheet cavitation, but it is still to be improved for the cloud cavitation and to be tested for the cavitating flows on a 3D propeller in non-uniform inflow and on a highly skewed propeller.

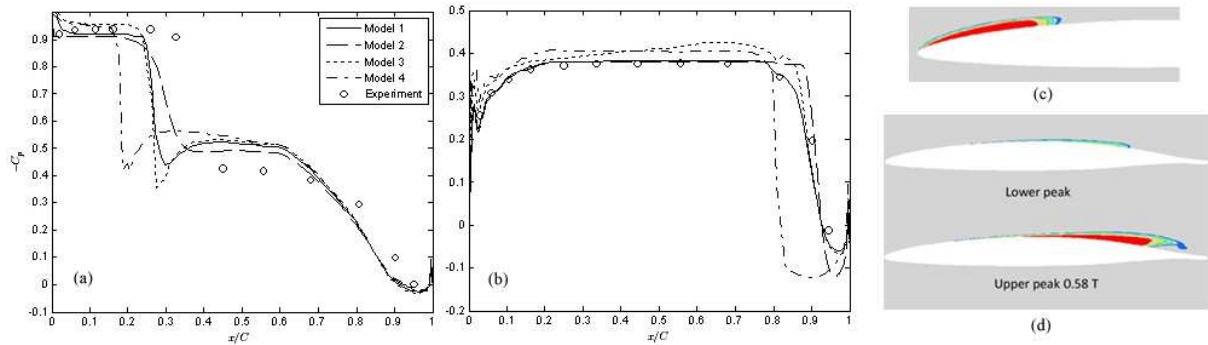


Figure 1:  $-C_p$  on the suction side as a function of  $x/C$  for  $\alpha = 4^\circ, \sigma = 0.91$  (a) and for  $\alpha = 1^\circ, \sigma = 0.38$  (b),  $\alpha_v$  (outermost layer indicating  $\alpha_v = 0.1$  and interval of 0.1) for  $\alpha = 4^\circ, \sigma = 0.91$  (c) and for  $\alpha = 1^\circ, \sigma = 0.38$  (d)

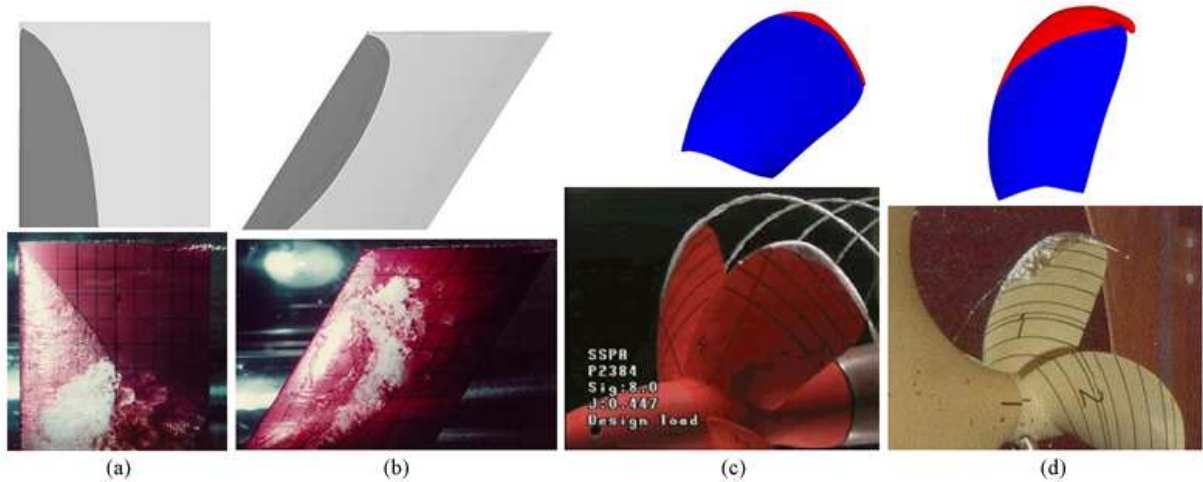


Figure 2: (a) Non-swept hydrofoil for  $\alpha = 6^\circ, \sigma = 0.628$ , (b) swept hydrofoil for  $\alpha = 6^\circ, \sigma = 0.585$ , (c) conventional propeller for  $J = 0.447, \sigma_n = 1.6$  and (d) for  $J = 0.2$ (computation),  $J = 0.4$ (experiment),  $\sigma_n = 2.2$

## References

- [1] Hoeijmakers, H. W. M., Janssens, M. E., Kwan, W., "Numerical simulation of sheet cavitation", 3rd Int. Symp. on Cav., 1998
- [2] Kuiper, G., *Leading edge: Data of selected propellers*, Report No.16206-2-RD, Marin, 2004
- [3]Kunz, R. F., Boger, D. A., Stinebring, D. R., Chyczewski, T. S., Lindau, J. W., Gibeling, H. J., Venkateswaran, S., Govindan, T. R., "A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction", *Computers & Fluids*, 2000
- [4] Li, D. Q., Lundström, P., *Leading edge: open water characteristics and cavitation inception tests of a conventional propeller and a highly skewed propeller*, SSPA report, 2002
- [5] Michelsen, J. A., *Basis3D- A platform for development of multiblock PDE solvers*, Technical Report AFM 92-05, DTU, Denmark, 1992
- [6] Shen, Y. J. and Dimotakis, J. S., "The influence of surface cavitation on hydrodynamics forces", 22nd ATTC, 1989
- [7] Singhal, A. K., Athavale, M. M., Li, H. Y., Jiang, Y., "Mathematical basis and validation of the full cavitation model", *ASME J. Fluids Eng.*, 2002
- [8] Sørensen N. N., *General purpose flow solver applied to flow over hills*, Risø, 2003
- [9] Zwart, P. J., Gerber, A. G., Belamri T., "A two-phase flow model for predicting cavitation dynamics", *ICMF*, 2004
- [10] Ukon, Y., "Cavitation characteristics of a finite swept wing and cavitation noise reduction due to air injection", *Int. Symp. Propeller and Cavitation*, Wuxi, pp.383-390, 1986