Second International Symposium on Marine Propulsors
smp'11, Hamburg, Germany, June 2011
Workshop: Cavitation on foil

Preliminary Cavitation Simulations on the Twisted Delft Hydrofoil

Dimitrios Papoulias[1], Manolis Gavaises[1]

[1]Energy and Transport Research Centre, School of Engineering and Mathematical Sciences, City University London, UK

1 INTRODUCTION

CUL has been involved in the field of cavitation, related to the operation of fuel injection systems, by combining innovative experimentation techniques [1-3] and state-of-the-art modeling methodologies [4-6]. As part of these investigations, a Eulerian-Lagrangian CFD model has been developed and extensively validated by a variety of relevant collected experimental measurements. Conceptually, the Lagrangian cavitation model is based on the tracking of sample cavitation bubble parcels that simulating the formation and development of the cavitation cloud. The incipience and latter evolution of the bubbly phase is affected by a number of process (growth/collapse, break-up, coalescence and collisions) which are predicted by suitable sub-grid scale phenomenological sub-models. The existence of the cavitation bubbles is realized by the surrounding liquid through implemented source terms that describe the momentum exchange between the two phases and include the influence of vapour volume fraction.

As part of the present work, a preliminary CFD study has been conducted over the last two months in an attempt to simulate cavitation around the Delft twisted hydrofoil. For this purpose both commercial and the in-door multiphase CFD solvers have been utilized. In particular, the predictability of the Eulerian-Eulerian mixture methodologies available in FLUENT-12 are evaluated and assessed against the capabilities of the Monte-Carlo multiphase Eulerian-Lagrangian GFS model. The Eulerian cavitation methodologies regard the two-phase flow as a mixture of liquid and vapor; a transport equation is coupled with the Navier-Stokes system of equations for tracing the gas phase. In contrast, the Lagrangian GFS model relies on numerous statistical, stochastic and mechanistic processes for predicting the nucleation, growth/collapse and break-up events of individual cavitation bubbles. Instead of treating the vapour phase as a continuum medium, the Lagrangian methodology predicts the trajectory of each discrete spherical bubble across the fluid field in accordance with Newton's second law and the acting flow forces (drag F_D , lift F_L , pressure force F_P , etc).

In this synopsis, a summary of the simulated flow cases is included together with a short discussion on the tested solver inputs. The majority of the simulated flow cases are summarized in Table 1, which is partitioned into two parts; firstly the non-cavitating steady-state flow simulations (CN=3.64) are categorized according to the grid-size and turbulence model while a second matrix includes the cavitating computations (CN=1.04) which likewise are classified in respect to the tested cavitation modeling parameters.

In particular, for the Eulerian cavitation model three different mixture methodologies are examined (CASE 6-8) under steady-state flow conditions but also the effect of a transient time-step (CASE 9) is investigated. On the other hand, the sensitivity of the Lagrangian cavitation model (CASE 10-15) is evaluated for different nucleation threshold pressures (CASE 10-12) - that is the pressure below which cavitation occurs - and also the influence of the temporal resolution (CASE 11,15) is inspected. For this purpose the timescale of the fluid flow (dt_{fluid}) is increased (CASE 15) but the integration step of the dispersed phase (dt_{bubble}) is maintained at low levels for preserving the accuracy of the bubble dynamics. Besides the aforementioned parametric modeling factors, also the available break-up models are investigated (CASE 11, 13, 14) in order to reveals the effects of this process on the main flow development.

Table 1: Parametric single-phase and cavitating flow study - sensitive CFD solver inputs and physical criteria

Non-Cavitating Steady-State Flow ***CN=3.64***				
	Case	Grid size	Discretization scheme	Turbulence model
Fluid Solver	**1,2,3**	800 k	2^{nd} Order Upwind	k-ε: ***STD*** - ***RNG*** - ***RLZ***
	4	1.5 M		k-ε ***STD***
	5	3.0 M		k-ε ***STD***

Cavitating Transient & Steady-State Flow ***CN=1.04***						
	Case	Cavitation model		Threshold pressure (bar)	Time-step (ms)	Break-up model
Vapor Phase	**6**	***Eulerian***	***Singhal***	0.03	***Steady-state***	-
	7		***Zwart***			
	8		***Sauer***			
	9		***Zwart***		dt_{fluid}=1.0 dt_{bubble}=0.01	
	10	***Lagrangian***		0.0	dt_{fluid}=1.0 dt_{bubble}=0.01	turbulent
	11			0.03		
	12			0.05		
	13			0.03		hydrodynamic
	14					inactive
	15			0.03	dt_{fluid}=2.5 dt_{bubble}=0.01	turbulent

2 SAMPLE FLOW RESULTS & SHORT DISCUSSION

A preliminary inspection of the developing cavitating flow (Figure 1 and 2) reveals the shedding of a vapour cloud which travels across the hydrofoil surface and the recirculation of the flow at the trailing edge. Interestingly, the downstream cavitation bubbles block the incoming flow streams and effectively reduce the liquid's acceleration over the leading edge of the hydrofoil. This in turns restricts the activity of the upstream cavitation site and thus allows the fluid to regain its high momentum over the twisted hydrofoil surface. The fluctuating nature of the nucleation site is depicted in Figure 3. This plot shows the iso-surface of the threshold vapour pressure (cyan boundary; p_{vapour} = 3000.0 Pa), below which cavitation bubbles are formed.

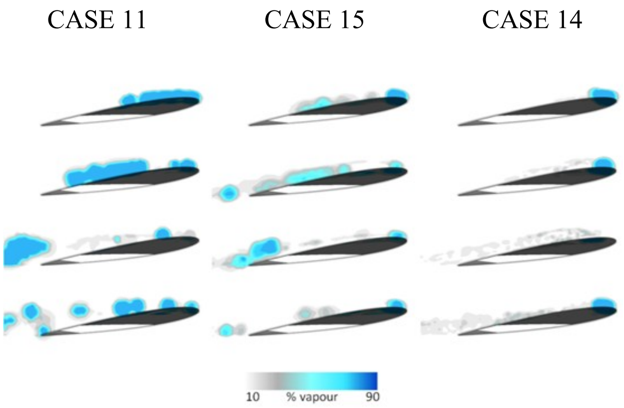

Figure 1: Instantaneous Lagrangian predictions of the developing vapour fraction (%)

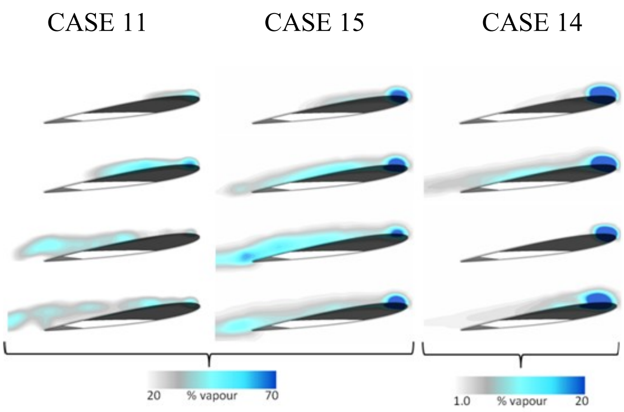

Figure 2: Time-averaged (over the Eulerian time step) Lagrangian predictions of the developing vapour fraction (%)

Intriguingly the developing hydrodynamic flow is somehow linked with the behaviour of the cavitation bubbles but also with the input parameters of the fluid solver and in particular the selected time-step. For instance the flow simulation with the relatively larger time-step, that is case 15, seems to smooth out the appearing vortical structures which are indicated when the temporal resolution is increased (CASE 11). Likewise if the bubble break-up process is neglected (CASE 14) apart from the reduced cavitation levels (Figure 1, 2) also the drifting flow seems attached to the hydrofoil's surface most of the time (Figure 3). In both cases the interactions between the bulk liquid and the bubbly phase are minimized either by reducing the available interfering time (CASE 15) or by omitting a dominant cavitation process (CASE 14).

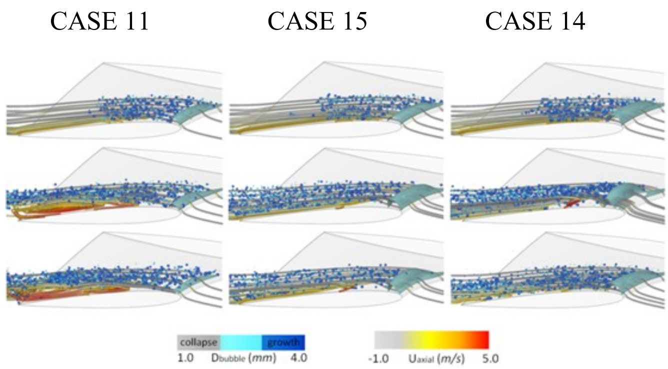

Figure 3: Instantaneous flow streamlines colored with the stream-wise velocity component, discrete cavitation bubbles highlighted according to the size and volume under tension at the leading edge (cyan iso-surface: p=3000 Pa)

In addition to the main flow features, FIGURE 3 displays also the travelling cavitation bubbles which are coloured according to their size.

REFERENCES

Roth, H., Gavaises, M. & Arcoumanis, C. 2002 Cavitation initiation, its development and link with flow turbulence in Diesel injector nozzles. SAE paper 2002-01-0214.

Gavaises, M. & Andriotis, A. 2006 Cavitation inside multi-hole injectors for large-scale Diesel engines and its effects on the near-nozzle spray structure. SAE paper 2006-01-1114.

Gavaises, M., Andriotis, A., Papoulias, D., Mitroglou, N. & Theodorakakos, A. 2009 Characterazation of string cavitation in large-scale diesel nozzles with tapered holes. Phys. Fluids, **21**: p. 1-9.

Giannadakis, E. & Gavaises, M. 2008 Modelling of cavitation in Diesel injector nozzles. J. Fluid Mech., **616**: p. 153-193.

Giannadakis, E., Papoulias, D., Gavaises, M., Arcoumanis, C., et al. 2007 Evaluation of the predictive capability of Diesel nozzle cavitation models. SAE paper 2007-01-0245.

Gavaises, M., Papoulias, D., Andriotis, A., Giannadakis, E. & Theodorakakos, A. 2007 Link between cavitation and erosion damage in diesel injector nozzles. SAE paper 2007-01-0246.