Predicting Marine Propeller Cavitation

Case Study of SC/Tetra

Using SC/Tetra to predict propeller cavitation including tip vortex region

Cavitation Flow Analysis

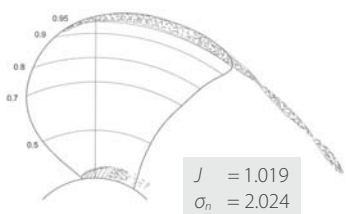
Cavitation in fluid machinery causes device degradation, vibration, and erosion. CFD can be used to predict the extent of cavitation during the propeller design and development phases, which reduces design cycle time and cost.

In this case study, CFD was used to simulate cavitation in a marine propeller, focusing especially on tip vortex cavitation. Analysis results and experimental measurements were compared and evaluated.

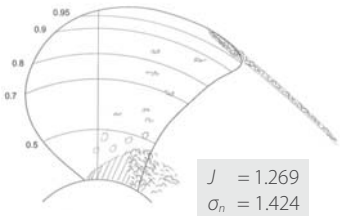
[1] Fujiyama, K. et al, smp'11 Workshop on Cavitation and Propeller Performance, 2011

Predicting the extent of cavitation

Measurement

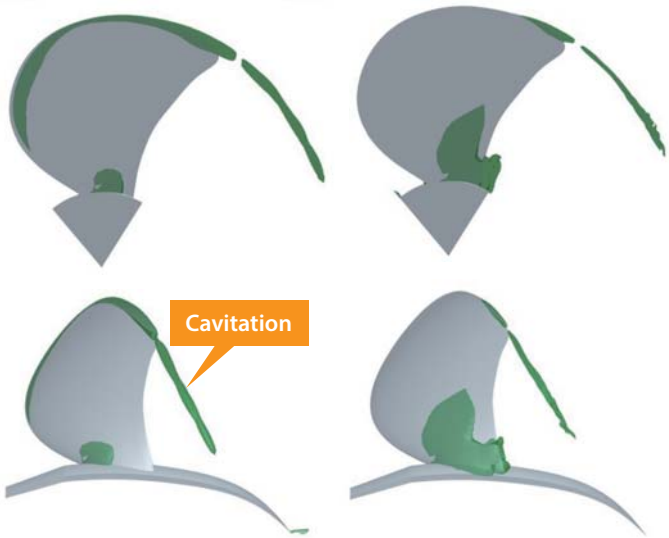


J = 1.019
σ_n = 2.024



J = 1.269
σ_n = 1.424

Analysis results



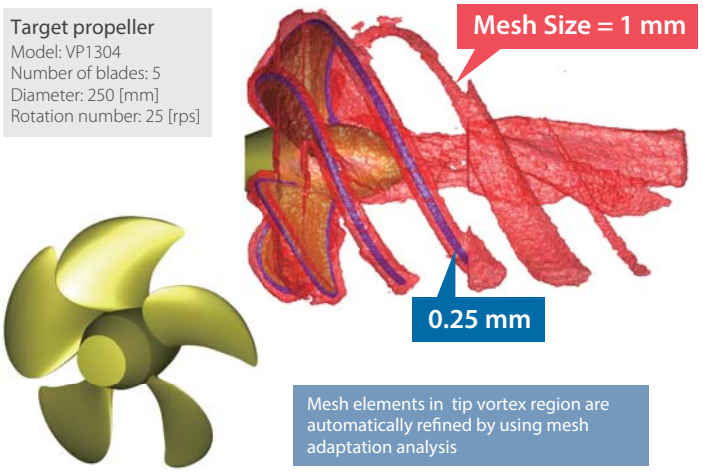
Case 1	K_T	Case 2
0.3750	(Thrust values)	0.2064
+0.67%	Versus test values	-3.59%

The extent of cavitation and thrust values are accurately estimated

Mesh generation using mesh adaptation analysis

Potsdam Propeller Test Case (PPTC)

Target propeller
Model: VP1304
Number of blades: 5
Diameter: 250 [mm]
Rotation number: 25 [rps]

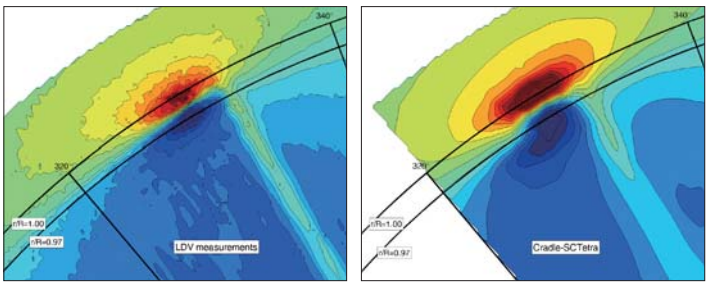


Mesh elements in tip vortex region are automatically refined by using mesh adaptation analysis

Tip vortex resolution

Vortices around propeller

Propeller velocity distribution at 0.1D downstream



Measurement using LDV | Analysis results

Velocity distribution of blade tip vortex are accurately predicted

Notes

SC/Tetra was used to accurately predict both the extent of cavitation around a marine propeller and the changes in thrust associated with the cavitation. Using mesh adaptation analysis to generate fine mesh elements, SC/Tetra accurately simulated local phenomena, such as tip vortex cavitation.