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.

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.