Abstract

Cavitating Flow refers to a complex hydrodynamic phenomenon occurring when the local static pressure in a fluid drops below its vapor pressure, causing vapor bubbles to form and collapse rapidly. This process generates intense shockwaves, leading to significant mechanical stresses on nearby surfaces. In turbomachinery, such as pumps, propellers, and turbines, cavitation can lead to erosion, vibration, noise generation, and reduced efficiency. As such, understanding and mitigating cavitation effects are crucial for maintaining the reliability and efficiency of turbomachinery devices.

In this doctoral thesis, the primary aim is to comprehensively understand the often-overlooked impact of dissolved air on cavitating flow. Acknowledging the inherent complexity in both numerical simulations and experimental setups, our research focuses on evaluating and elucidating the influence of dissolved air. We strive to develop methodologies that effectively integrate dissolved air considerations into both simulation and experimental analysis, aiming for a deep understanding of its effects.

The research unfolds in two distinct yet interrelated parts. Firstly, employing Computational Fluid Dynamics (CFD) modeling techniques, we delve into the intricate dynamics of three-phase cavitating flow. The CFD modeling phase utilizes Finite Volume Methodology (FVM) to discretize the transient three-dimensional Navier-Stokes equations. The emphasis here is on modification and development of turbulence models, cavitation models, and incorporation of dissolved air through a mixture model to account for the presence of dissolved air. Various models and approaches are explored, seeking to accurately simulate the behavior of cavitating flows in the presence of dissolved air. Secondly, experimental investigations are conducted to validate and augment the insights gained from numerical simulations. The experimental tests are conducted in a hydraulic setup, carefully designed to measure and observe the cavitation behaviors in water with controlled dissolved air levels. The test chamber, hydrofoil, and monitoring systems are configured to facilitate detailed observations and data collection.

The culmination of experiments and numerical simulations revealed a profound correlation between dissolved air levels and cavitating flow dynamics. Variations in dissolved oxygen levels distinctly influenced cavitation frequencies and cloud structures, notably amplifying shedding vortex frequencies with increasing cavitation numbers. Both experimental validations and numerical simulations underscored the pivotal role of dissolved air, showcasing significant reductions in pressure pulsation amplitudes at higher cavitation numbers, indicating a stabilizing effect on cavitating flow dynamics. Augmented dissolved air content not only expanded cavity closure volumes, leading to larger cloud cavities and stabilized cavitating flow, but also introduced smaller-scale instabilities in cavity closures. The introduction of dissolved air showcased enhancements in lift and drag coefficients, prominently altering the behavior of the re-entrant jet, and influencing pressure coefficient distributions in sheet and cloud cavity regions. Additionally, the injection of air demonstrated vital impacts on shedding frequencies and vibration frequencies in the test chamber, highlighting its effectiveness in altering cavitation characteristics, contingent upon injection specifics and flow conditions. These findings collectively emphasize the intricate relationship between dissolved air and cavitating flow, elucidating its multifaceted impacts on cavitation phenomena.