# Numerical simulations of cavitating flows in the U9-400 Kaplan turbine model

**Mohammad Hossein Arabnejad, Håkan Nilsson and Rickard E. Bensow**

Department of Mechanics and Maritime Sciences, Chalmers University of Technology, 41296 Gothenburg, Sweden

E-mail: mohammad.h.arabnejad@chalmers.se

**Abstract.** In this paper, we use high-fidelity Computational Fluid Dynamics (CFD) to examine cavitating flows in the U9-400 Kaplan turbine model near cavitation breakdown. We study both Part-Load (PL) and Best Efficiency Point (BEP) conditions. The results are compared with experimental data obtained from Vattenfall, including observations of the cavitating regions in the blade passage and the efficiency drop due to cavitation breakdown. The comparison shows that the simulations can capture similar cavitation patterns as in the experiments and that the drop in efficiency due to cavitation breakdown is well predicted. We further investigate the reasons behind the efficiency drop near the cavitation breakdown and reveal that it is caused by the reduction in the pressure difference between the suction side and the pressure side of the runner blade due to cavitation. This reduction subsequently leads to a decrease in torque and efficiency.

## 1. Introduction

Hydro turbines are increasingly being used to regulate the electrical grid, which leads to more operation at off-design conditions. To enable this operational flexibility, it becomes necessary to expand the operational range of hydro-turbine systems. One challenge in this expansion is the occurrence of cavitation, which is the formation of vapor due to a drop in pressure. Cavitation can significantly impair performance, induce vibration, and lead to cavitation erosion [1]. Therefore, it is crucial to study the formation of cavitation and its effects on the performance and reliable operation of hydro turbines.

To study cavitation behavior, numerical simulations have been shown to be a useful tool as they provide access to the flow field, enabling investigation of various phenomena [2]. In this paper, we employ such numerical simulation techniques to investigate cavitating flows in the U9-400 Kaplan turbine model. Our focus is to provide a better understanding of the cavitation behavior near the onset of cavitation breakdown under both Part-Load (PL) and Best Efficiency Point (BEP) conditions. Initially, we compare the numerical results with the experimental results obtained by Vattenfall, which indicate that the simulations successfully capture the main flow characteristics in the studied conditions. Subsequently, we utilize the numerical results to investigate the underlying causes for the performance drop at the onset of cavitation breakdown.