Master's thesis project:

Modal analysis of the cavitating flow around a model-scale ship propeller:
a combined experimental and numerical study.

Background

Cavitation is the change of phase from liquid to vapor when the fluid pressure falls locally below the saturation pressure. When occurring on ship propellers, it can affect the energy efficiency of the propulsive system and lead to large levels of underwater noise.

The unsteady nature of propeller cavitation, together with the range of length and time scales involved, makes the understanding of this phenomenon a challenging task. In order to investigate the mechanisms behind cavitation, modal decomposition techniques have been employed to analyze both experimental data and numerical results. In the former case, snapshots from high speed videos are processed; on the other hand, numerical modeling allows for a direct modal decomposition of the flow fields.

One method for modal analysis is the Proper Orthogonal Decomposition (POD) technique, employed for example in Reference [1]. POD enables to find the coherent structures in a complex flow pattern, with their time scale and energy content.

This project has two objectives. First of all, provide validation by comparison of modal analysis of experimental data on one side, and numerical results on the other side. Secondly, the combined experimental and numerical work has the potential for a detailed description of the spatio-temporal characteristics of propeller cavitation.

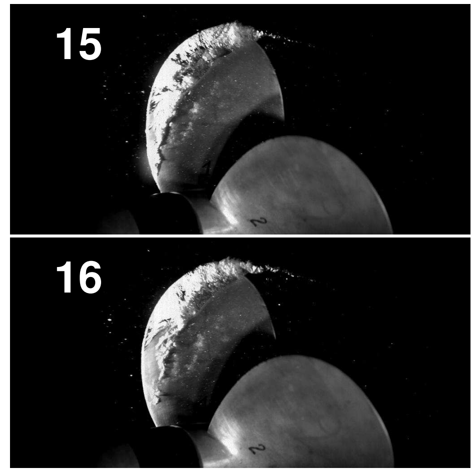

Figure 1: Two frames from a high-speed video of unsteady cavitation on a model-scale propeller in open water. Pereira et al.[2]

Tasks

1. Make a literature review of the methods for modal decomposition used in hydrodynamics and draft a workplan. The literature review should provide an indication for a suitable test case. One possible geometry is the E779A propeller, for which model tests were made by Pereira et al. [2].
2. Review the available experimental data and analyze the flow field through, for examples, POD on a collection of snapshots from high speed videos.
3. Carry out the numerical simulations and apply the POD analysis as part of the post-processing phase. Validate the results against the experimental data.
4. Write a (concise) report with conclusion from the modal analysis, regarding both the physics of cavitation and the comparison between simulations and experiments. Moreover, provide best-practice guidelines for the analysis of propeller cavitation using the POD technique.

Additional information

For the CFD simulations, the multi-phase viscous flow solver ReFRESCO will be used [3], developed at the Maritime Research Institute of the Netherlands (MARIN). A final presentation at MARIN, in the Netherlands, is possible upon the conclusion of the thesis. The work should be conducted preferably by two students.

Send your application to

Carlo Negrato, PhD student
negrato@chalmers.se

Prof. Rickard Bensow, Full Professor
rickard.bensow@chalmers.se

References

[1] C. Negrato, T. van Terwisga, R.E. Bensow. Analysis of hydrofoil cavitation using Proper Orthogonal Decomposition. Proceedings of NuTTS 2017.

[2] F. Pereira, F. Salvatore, F. Di Felice, M. Soave. Experimental investigation of a cavitating propeller in non-Uniform inflow. Proceedings of ONR 2004.

[3] `http://www.refresco.org/` Last access 22-October-2018.

CHALMERS

MARIN