3D CFD study of a centrifugal pump with OpenFOAM

Purpose

The purpose of this project is to analyse the 3D flow in the ERCOFTAC Centrifugal Pump, using the OpenSource CFD tool OpenFOAM (www.openfoam.org). The results should be presented both as a master thesis report, and as a Wiki tutorial on how to perform such simulations. The case was studied in 2D at the fourth OpenFOAM workshop in Montreal ([http://www.openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_centrifugal_pump_with_a_vaned_diffuser)](http://www.openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_centrifugal_pump_with_a_vaned_diffuser).

This project should extend the 2D simulations to 3D, elaborate on the choices of boundary conditions, turbulence models and solvers, and validate the numerical results with all the available experimental results.

Description of the thesis work

The master thesis will cover the following:

- Literature study: A number of previous publications have been gathered, and a thorough study of them is necessary to understand the case.
- Grid generation of the 3D model, using ICEM CFD. A 2D grid of the model is already available that will help in generating the 3D grid.
- CFD of the flow using the grid generated above. The problem involves both rotating and steady parts. There are different numerical methods of coupling these parts, among else Multiple Reference Frames (steady simulation), and sliding grid (unsteady simulation). You will evaluate those different methods thoroughly.
- Analysis of the results and post processing using OpenSource Gnuplot and Paraview, Other alternatives can also be considered.
- Comparison with the earlier results, from the literature as well as from the 2D study.
- Creating an automatisation of the process, from pre-processing in OpenFOAM to post-processing, so that it will be easier to run a tutorial for newcomers to OpenFOAM.
- Writing a master Thesis report and give an oral presentation. Ultimately, an abstract can be sent to the 5th OpenFOAM workshop in Göteborg, June 21-24 2010. If it is accepted (which is likely if the project is successful), you will have the opportunity to present your work at the workshop.

This Master thesis will provide you with knowledge of an OpenSource CFD software that becomes more and more used in both industry and academia. It will give you a basic knowledge of C++, as well as a good understanding of the CFD process, from grid generation to post-processing and analysis of the results.

Suitable background

A good knowledge in fluid mechanics and CFD is required. A basic knowledge of Linux and C++ is a plus, but not a requirement.

Starting date

Fall of 2009 or early 2010.

Contacts

Håkan Nilsson   031-772 1414   hani@chalmers.se
Olivier Petit   031-772 1405   olivierp@chalmers.se