Here are some Master Thesis propositions with OpenFOAM:

ERCOFTAC Centrifugal Pump

Numerical studies of the flow in the ERCOFTAC centrifugal pump. In this work you should do thorough literature studies of previous results, and then you should study this case in a number of different ways. The problem involves both rotating and steady parts, which means that there are differens numerical methods of coupling these parts. You should study these different methods thoroughly. The main CFD code used for the studies will be OpenFOAM. You will evaluate already existing features of the code, but there is also a need to develop new features. This case will be used as a turbomachinery case-study for the fourth OpenFOAM workshop in Montreal June 1-4 2009, and you should then also take part in setting up the case study - meshing, case-set-up, post-processing routines and best-practice-guidelines (note that I do unfortunately not have funding to finance any trip for you to the workshop, but if we can find the funding you may/should of course join the workshop). You can see http://www.openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_conical_diffuser for an example of the case-study that was set up for the previous OpenFOAM workshop. The ERCOFTAC centrifugal pump case-study should be placed next to this one, in the OpenFOAM Wiki. Some studies of this case have already been made at Hydro Quebec in Canada, using CFX, and my PhD student Olivier Petit will also study this case between February and June, while he is visiting Hydro Quebec. We will have a close contact with them. If you are joining the OpenFOAM course, you can of course start working on the master thesis as part of your OpenFOAM course project.

Hydraulic design of intakes of water turbines

Hydraulic design of intakes of water turbines. This project involves free-surface flow, and the focus is on the flow and the losses in the intake of water turbines. This is a pilot project, aiming for future financing. The problem is not as well-defined as the one above, so you will have to find out relevant geometries and flow properties as part of the project. It should be possible to use generic geometries, but we should try to find a case with experimental or numerical results to compare with. OpenFOAM should be used also in this project, and there is a need to validate existing features in OpenFOAM for these kinds of applications. There might also be some need for development of OpenFOAM. I have been the examiner of a somewhat similar project before, where we did numerical simulations of the flow in a spillway, using Fluent (see http://www.tfd.chalmers.se/~hani/pdf_files/Computational_Modeling_of_Flow_over_a_Spillway_final_version.pdf and http://www.tfd.chalmers.se/~hani/pdf_files/MS_Thesis_presentation_BjornMargeirsson.ppt). As an introductory part of the present project it might be of interest to reproduce these results (The mesh is available, so it would 'just' be to set the boundary conditions and make the solution converge). However, this should be more of an exercise, rather than a large part of the work. Also in this project you should set up some generic case-study that could be used for future OpenFOAM workshops. If you are joining the OpenFOAM course, you can of course start working on the master thesis as part of your OpenFOAM course project.

Other propositions of less importance than the above:

- Setting up a case-study of the Dellenback combustor using OpenFOAM, for all the different cases that were measured by Dellenback. You should of course also do some thorough studies of the flow for these cases using different methods. For this we first need an agreement that we can distribute the measurements, but I don't think that will be a problem.
- Finding and studying validation cases for numerical cavitation modeling. Also in this case we are aiming at setting up OpenFOAM case-studies. One of the purposes of this project is to support an

existing PhD project, where the student is on parental leave.

Contact Håkan Nilsson, hani@chalmers.se for more information