Postdoctoral fellowship

18 months

High Performance Computing for Turbulent Flows: application to hydraulic turbines

Research center: The Laboratory of Geophysical and Industrial Flows (LEGI)
Address: Domaine Universitaire, BP 53, 38041 Grenoble Cedex 9
Research team: Modelling and Simulation of Turbulence
(MOST, http://www.legi.grenoble-inp.fr/web/spip.php?rubrique43)
Contact: Guillaume Balarac, guillaume.balarac@grenoble-inp.fr

Remuneration: 3000 € brut / month.

Context

New usages of hydro-electric turbines bring new issues. To overcome these issues, a better understanding is needed of the flow dynamic in various components of the hydraulic power station. For example, unsteady hydrodynamic instabilities can appear when the turbine is used at partial load during a long time: vortex rope, inter blade vortices, …

Numerical simulation appears as an effective tool to investigate these phenomena. However, classic statistic approaches can fail for these kind of strongly unstable regimes. So, to better analyze this kind of flow, unsteady simulations should be used to improve the reliability of the numerical prediction. In high performance computing (HPC) context, the growth of computing power means such simulations for industrial geometries can be carried out. However, some challenges persist.

The goal of this project is to address these challenges. The candidate will evolve in the MOST team, internationally known for its work in the field of numerical simulation of turbulent flows. He/she will have access to the local, French (GENCI) and European (PRACE) computational centers. Moreover, this project will be conducted in strong collaboration with the technological centre of Alstom Hydro, which is based in Grenoble. This work will form a part of the industrial chair between Grenoble Institute of Technology and Alstom Hydro.

Main objectives

The goal of this project is therefore to develop needed methods to allow for the performing of accurate unsteady simulations of flow in hydraulic turbine components. To carry out these simulations, the MOST team has taken part in a CNRS joint initiative called SUCCESS (http://success.coria-cfd.fr) to promote super-computing in fluid mechanics. The team participates in the development of the YALES2 solver, a massively parallel solver allowing simulation in realistic geometries (http://www.coria-cfd.fr/index.php/YALES2). One of the main issues to predict accurately the flow in industrial applications is to have a precise description of the turbulent effects. To overcome this issue, the large-eddy simulation (LES) approach consists in explicitly solving the large scales flow and modelling only small scales. The direct description of the large scales allows improvement in the simulation results. However, the theoretical framework of LES necessitates taking into account specific constraints on the mesh generation.

Moreover, to be able to apply LES approach to simulate flow in hydraulic turbines, new numerical methods have to be developed. In particular, numerical methods have to be developed to take into account moving solids. For example, this is needed to simulate conjointly runner and draft tubes. Simulations of such configurations in HPC context are still challenging. The goal of the project is therefore to develop an efficient and accurate numerical tool. As a second step, the new developed numerical tool will be used to make an in-depth analysis of the hydrodynamic instabilities and the consequences of these instabilities on the turbine performance. For this step, collaboration with the laboratory for hydraulic machines (EPFL, Lausanne, Switzerland) is planned.