Postdoctoral fellowship

High Performance Computing for Turbulent Flows in Complex Geometries: application to hydraulic turbines

Research center: The Laboratory of Geophysical and Industrial Flows (LEGI)
Address: Domaine Universitaire, BP 53, 38058 Grenoble Cedex 9

Research group: Modelling and Simulation of Turbulence
(MOST, http://www.legi.grenoble-inp.fr/web/spip.php?article322)

Contact: Guillaume Balarac, guillaume.balarac@grenoble-inp.fr

Remuneration: between 2400 and 3000 € brut / month, depending on previous experience

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 kinds 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.

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. The candidate will evolve in the MOST team, internationally known for its work in the field of numerical simulation of turbulent flows. 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 participate to the development of an efficient and accurate numerical tool. The developments could consider turbulence modeling, dynamic remeshing, numerical methods, boundary layer treatment, cavitation modeling, fluid-structure interaction…

Application

The candidate should have a strong background in numerical simulation of turbulent flows. The application should include a Curriculum vitae and a brief research proposal addressing a question linked with the objectives described above.