Numerical Simulation of Flow in a Francis Turbine at Nominal and Off-Design Operating Conditions

P. T. K. Loan[1] **and B. V. Ga**[1]

Abstract: This paper presents the use of a commercial Navier-Stokes turbulent flow code as a mean to evaluate the behavior of a runner of a Francis turbine for the design and off-design conditions. The flow in the Francis turbine runner is analysed numerically. Different operating points are calculated using the software package based on a finite volume method (FLUENT). The numerical results permit to observe physical phenomena in the runner that are important in the process of hydraulic turbomachinery design. Values of different velocity components, blade pressure distribution and recirculation in the flow are compared experimental data at nominal and off-design flow conditions that show a very accurate prediction of the turbine performance.

Computer resource involves in the flow analysis should be compatible with the need of design process of a runner. Therefore 12 hours of CPU time can be considered as acceptable for each operating point on a computer workstation of medium size power.

In this study, the FLUENT software, using a k-e standard turbulent model, is used to analyse the behavior of the runner designed by Neyrpic and used for experimental research work in the laboratory LEGI, INPG (Institut National Polytechnique Grenoble). This method consists in modelling and computing the flow in a single channel of the 13 blade runner with peridiocity assumed between the channels of the turbine. The channel grid contains 75113 elements including the tetrahedrons, the hexaedrons and the prisms.

This method permit to use the boundary conditions as outlet with radial equilibrium pressure distribution; lateral faces with periodic conditions in order to reduce calculation domain; shroud, hub and the blade with wall condition and inlet with the boundary condition for the runner guide vane interaction.

After control of numerical convergence, comparison between experimental data and numerical results of the specific energy and momentum is proposed. The physical coherence is shown with many operating points. The analysis of the numerical results is in agreement with the experimental ones. The pressure distribution on both side of the blade is obtained. The blade loading is well calculated.

For the outlet of the grid the numerical results coincide well with experimentals flow survey, the evolution of the velocity components on the outlet is well predicted for all operating points.

The unsteady calculation gives the results more accurate than those of the steady simulation, however it requires more CPU time for the numerical convergence.

[1] University of Danang
01, Cao Thang, Danang, Vietnam
Tel. (84) 511 835 706; Fax (84) 511 894 884
buivanga@dng.vnn.vn