Unsteady Rotor-Stator Simulation of the U9 Kaplan Turbine Model

Olivier Petit‡ and Håkan Nilsson

Chalmers University of Technology
Hörsalsvägen 7A
412 96 Göteborg, Sweden

March 08, 2011

Keywords: Water Turbine, OpenFOAM, Unsteady, Rotor-Stator

The present work compares simulations made using the OpenFOAM CFD code with experimental measurements of the flow in the U9 Kaplan turbine model. The U9 Kaplan turbine prototype, located in Porjus, Sweden, has an inlet pipe that is curved close to the inlet of the spiral casing. Nowadays, this curved pipe and its effect on the flow in the turbine is not taken into account when numerical simulations are performed at design stage. To study the impact of the inlet pipe curvature on the flow, a 1:3.1 scaled model of the U9 Kaplan Turbine was experimentally investigated at the Vattenfall Research and Development (VRD) test rig in Älvkarleby, Sweden. Pressure and velocity measurements were made at the spiral casing inlet, in the spiral casing just before the guide vanes and in the draft tube [1,2,3,4].

The computational domain used in the present work starts with the high-pressure tank and includes the totality of the U9 Kaplan turbine model, the curved inlet pipe, the spiral casing, the wicket gate, the runner and the draft tube. The domain was realized in ICEM Hexa, and is divided in 6 different parts: the inlet tank, the curved pipe, the wicket gate, the runner and the draft tube. The different parts are then coupled together using the General Grid Interface (GGI) [5]. The mesh is fully hexahedral and consists of 12 million cells.

Unsteady simulation using the transientSimpleDyMFoam solver was realized and the present work compares the numerical results of the flow with the measured data. The computed results are fairly similar to the measured ones.

‡ Corresponding Author: Olivier Petit (olivierp@chalmers.se)
REFERENCES


