Proceedings of

HYDRO 2015 – October, 2015, Bordeaux, France

The role of unsteady flow simulation in Francis turbine development

Bernd Nennemann, Oliver Braun

Abstract

The performance of Francis turbines has been improved continuously since their invention in the 1800's. Units with power output upwards of 300MW are now commonplace. In the past 20 years steady state flow simulations by means of Computational Fluid Dynamics (CFD) have significantly contributed to the progress in turbine technology in the form of efficiency gains and cavitation margins [1]. In the past decade, unsteady computational fluid dynamics has been added to the suite of tools used in the analysis of Francis turbines. In many instances this has been driven by the occurrence of damage such as cracks or high levels of vibrations and pressure pulsations [2, 3]. In many cases unsteady CFD has proven a useful tool in the process of understanding the root causes of such vibrations or pressure pulsations [4, 5].

Subsequent to such root cause analyses, the methods used, and sometimes newly developed, have regularly entered use in the development process of the turbine components. Increasingly this has become necessary to ensure machine safety and reliability as machines have been pushed towards performance limits, thus potentially reducing mechanical robustness. Wider operating ranges have also been required by customers as machines are operated more frequently in strongly variable operation. Unsteady CFD analyses have permitted the prediction of dynamic pressure loads from a number of hydrodynamic phenomena. An overview of five typical unsteady phenomena is given in the sections below.