Analysis of unsteady pulsating flows with a modified wall function

I S Grecu[1,ON1], **G Dunca**[1*,ON2], **D M Bucur**[1,ON3] **and M J Cervantes**[2,ON4]

[1]Department of Hydraulics, Hydraulic Machinery and Environmental Engineering, University POLITEHNICA of Bucharest, Bucharest, Romania
[2]Division of Fluid and Experimental Mechanics, Luleå University of Technology, Luleå, Sweden
ORCID numbers: ON1: 0000-0003-2047-7335, ON2: 0000-0003-2517-2610, ON3: 0000-0003-4771-5292, ON4: 0000-0001-7599-0895.

[*]Corresponding author: georgianadunca@yahoo.uk.com

Abstract. Nowadays, due to the increase in intermittent energy production (wind and solar), hydraulic turbines are more often operated under off-design conditions. In these operating conditions, dynamic phenomena occur in hydraulic turbines as flow instabilities, secondary flows, vortex rope developed in the draft tube etc. These phenomena can lead to pressure pulsations in the water conduit and structural vibrations of the hydraulic turbine. All these losses affect the hydraulic turbine performance and its lifetime. In the present paper a wall model, developed by Manhart et al. (2008), that considers the adverse pressure gradient of the flow is used with the *k-ω* SST turbulence model to study unsteady pulsating flows which can occur in a hydraulic turbine during part load operation. The Manhart wall model has the advantage of being used on a coarser mesh, $y^+ \leq 5$, leading to smaller simulation time and computational requirements of the numerical simulations compared to the general literature. The pulsating flows are analysed inside a geometry similar to the draft tube of a hydraulic turbine using the Code_Saturne CFD software.

1. Introduction

In the last decade an increase of the energy production from renewable energy sources (hydro, wind and solar photovoltaics (PV) was observed [1]. The wind and solar PV are forcing the hydraulic turbines to work under off-design conditions due to their intermittency. This can lead to part load or high load operation of hydraulic turbines which can result in the development of dynamic phenomena inside the hydraulic turbines. The dynamic phenomena can be represented by transient flows, secondary flows, flow instability, pulsating flows etc., which showed a great interest for the academia and industry community over the last decades [2, 3].

In the last decade, high-performance computers (HPC) have become more accessible to perform Computational Fluid Dynamics (CFD) numerical analysis of flow inside hydraulic turbines. This led to studies of the dynamic phenomena using less expensive methods compared to experimental measurements [4, 5]. One economic approach to perform a CFD analysis through a hydraulic turbine, from a computational power point of view, is the Reynolds Averaged Navier - Stokes (RANS) approach. Still, when complex flows are analyzed using the RANS approach, several challenges arise, such as flow separation and influence of the Adverse Pressure Gradient (APG) in the draft tube. The RANS approach