Lista 10 lucrări relevante:

1. Ciocan T., Susan-Resiga R., **Muntean S.** (autor corespondent), (2016) Modelling and optimization of the velocity profiles at the draft tube inlet of a Francis turbine within an operating range, Journal of Hydraulic Research, ISSN 0022-1686, Vol. 54, Issue 1, pp. 74-89 (WOS:000370980100005)
2. Susan-Resiga R.F., **Muntean S.**, Avellan F., Anton I., (2011) Mathematical modeling of swirling flow in hydraulic turbines for the full operating range, Applied Mathematical Modelling, ISSN 0307-904X Vol. 35, Issue 10, pp. 4759-4773 (WOS:000292176200007)
3. Frunzaverde D., **Muntean S.**, Marginean G., Campian V., Marsavina L., Terzi R., Serban V., (2010) Failure analysis of a Francis turbine runner IOP Conference Series-Earth and Environmental Science, Vol. 12, Paper No 012115. ISSN: 1755-1307 DOI: 10.1088/1755-1315/12/1/012115 (WOS: 000325657000115)
4. Negru R., **Muntean S.**, Marsavina L., Susan-Resiga R.F., Pasca N., (2012) Computation of stress distribution in a Francis turbine runner induced by fluid flow, Computational Material Science, ISSN 0927-0256, Vol. 64, Issue 11, pp. 253-259, (WOS:000308396200053)
5. Bosioc A.I., Susan-Resiga R.F., **Muntean S.**, Tanasa C., (2012) Unsteady Pressure Analysis of a Swirling Flow with Vortex Rope and Axial Water Injection in a Discharge Cone, Journal of Fluids Engineering, ISSN 0098-2202, Vol. 134, Issue 8, Article number 081104. (WOS:000314759800004)
6. Javadi A., Bosioc A., Nilsson H., **Muntean S.**, Susan-Resiga R., (2016) Experimental and Numerical Investigation of the Precessing Helical Vortex in a Conical Diffuser, With Rotor-Stator Interaction, Journal of Fluids Engineering, ISSN 0098-2202, Vol. 138, Issue 8, Article number 081106. (WOS:000379589700006)
7. **Muntean S.**, Draghici I., Ginga G., Anton L.E., Baya A., (2015) Hydrodynamic Design of a Storage Pump Impeller using Inverse Method and Experimental Investigation of the Global Performances, WasserWirtschaft, ISSN 0043-0978, Vol. 105, Issue 1, pp. 28-32. (WOS:000354657300007)
8. Draghici I., **Muntean S.** (autor corespondent), Bosioc A.I., Anton L.E.,(2014) LDV measurements of the velocity field on the inlet section of a pumped storage equipped with a symmetrical suction elbow for variable discharge values, IOP Conference Series-Earth and Environmental Science, Vol. 22, Paper No 032017. ISSN: 1755-1307 DOI: 10.1088/1755-1315/22/3/032017 (WOS: 000347441900075)
9. Draghici I.A., **Muntean S.** (autor corespondent), Bosioc A.I., Ginga G., Anton LE., (2016) Unsteady Pressure Field Analysis at Pump Inlet Equipped with a Symmetrical Suction Elbow, Proceedings of the Romanian Academy Series A: Mathematics, Physics, Technical Sciences, Information Sciences, ISSN 1454-9069, Vol. 17, Issue 3, pp. 237-244 (WOS:000383527500007)
10. Moisa I.G., Susan-Resiga R., **Muntean S.**, (2013) Pump inducer optimization based on cavitation criterion, Proceedings of the Romanian Academy Series A: Mathematics, Physics, Technical Sciences, Information Sciences, ISSN 1454-9069, Vol. 14, Issue 4, pp. 317-325 (WOS:000328441500008)

IAHR AIRH Taylor & Francis Taylor & Francis Group

Journal of Hydraulic Research Vol. 54, No. 1 (2016), pp. 74–89
http://dx.doi.org/10.1080/00221686.2015.1119763

Research paper

Modelling and optimization of the velocity profiles at the draft tube inlet of a Francis turbine within an operating range

TIBERIU CIOCAN, Researcher, *Research Centre for Engineering of Systems with Complex Fluids, Politehnica University Timişoara, Timişoara, Romania*
Email: *tiberiu.ciocan@upt.ro*

ROMEO F. SUSAN-RESIGA, Professor, *Hydraulic Machinery Department, Politehnica University Timişoara, Timişoara, Romania*
Email: *romeo.resiga@upt.ro*

SEBASTIAN MUNTEAN, Senior Researcher, *Centre for Advanced Research in Engineering Sciences, Romanian Academy – Timişoara Branch, Timişoara, Romania*
Email: *seby@acad-tim.tm.edu.ro* *(author for correspondence)*

ABSTRACT
Operating hydraulic turbines over an extended range of discharge values requires an optimized runner that minimizes the weighted-average draft tube losses. It is shown that the draft tube losses can be minimized before actually designing the (new or refurbished) runner blades by optimizing the runner outlet flow. For this, a new runner surrogate model is introduced, where the relative flow angle is expressed via the so-called swirl-free velocity profile (represented with two parameters). Given a swirl-free velocity profile, as well as a turbine discharge value, the model provides swirling flow profiles consistent with the hydraulic turbine operation. With this swirling flow as inlet condition, the draft tube flow computation provides the relationship between the swirl-free velocity parameters and the hydraulic loss. It is shown that this approach can be used for optimizing the runner for a range of turbine operating regimes.

Keywords: CFD; draft tube; hydraulic turbine; optimization algorithm; swirling flow

1 Introduction

Changing demands on the energy market, as well as limited energy storage capabilities, require improved flexibility in the operation of hydraulic turbines for regulating the electrical grid. As a result, turbines tend to be operated over an extended range of regimes and maximizing the weighted-average efficiency increasingly becomes more important than simply improving the peak efficiency. In particular, for low or medium head Francis hydraulic turbines the shape of the efficiency hill chart is practically given by the steep increase in the draft tube losses at off-design operating points (Vu & Retieb, 2002). The main reason why the efficiency of a turbine significantly drops when operating far from the best operating regime is that the inherent residual swirl at runner outlet leads to large draft tube losses. Although this phenomenon cannot be avoided, one can adjust the runner outlet geometry such that a weighted-average draft tube loss becomes as low as possible over a certain range of operating points.

When refurbishing a hydraulic turbine the draft tube remains unmodified for economical reasons. As a result, the new runner should be the best match for the existing draft tube within a wide operating range. In order to achieve this goal, there are two main approaches: (i) several runner geometries are developed using classical design methodologies, then the overall turbine efficiency is determined and compared; and (ii) the swirling flow is first optimized at the draft tube inlet corresponding to a set of turbine operating points, then the runner blades are designed accordingly.

The second approach can be straightforwardly used for optimizing the swirl flow ingested by the draft tube at the design operating point (Galván, Rubio, Pacheco, Mendoza, & Toledo, 2013; Galván, Rubio, Pacheco, Gildardo, & Georgina, 2013). However, there is a distinct need for optimizing the swirling

Received 10 September 2014; accepted 6 November 2015/Open for discussion until 30 August 2016.

ISSN 0022-1686 print/ISSN 1814-2079 online
http://www.tandfonline.com

Applied Mathematical Modelling 35 (2011) 4759–4773

Contents lists available at ScienceDirect

Applied Mathematical Modelling

journal homepage: www.elsevier.com/locate/apm

Mathematical modelling of swirling flow in hydraulic turbines for the full operating range

R.F. Susan-Resiga [a,*], S. Muntean [b], F. Avellan [c], I. Anton [a]

[a] "Politehnica" University of Timişoara, Hydraulic Machinery Dpt., Bvd. Mihai Viteazu 1, RO-300222 Timişoara, Romania
[b] Romanian Academy - Timişoara Branch, Bvd. Mihai Viteazu 24, RO-300223, Timişoara, Romania
[c] École Polytechnique Fédérale de Lausanne, Laboratory for Hydraulic Machines, Av. de Cour 33Bis, CH-1007, Lausanne, Switzerland

ARTICLE INFO

Article history:
Received 29 September 2010
Received in revised form 16 March 2011
Accepted 31 March 2011
Available online 5 April 2011

Keywords:
Swirling flow
Hydraulic turbine
Constrained variational problem
Fourier–Bessel series

ABSTRACT

We introduce and validate a novel mathematical model for computing the radial profiles of both axial and circumferential velocity components, respectively, of the swirling flow exiting the runner of hydraulic turbines within the full operating range. We assume an incompressible, inviscid, axisymmetrical, and steady swirling flow, with vanishing radial velocity at runner outlet. First we find the correlation between the flux of moment of momentum downstream the turbine runner and the operating regime given by turbine's discharge and head. Second, we express the relationship between the axial and circumferential velocity components, corresponding to the fixed pitch runner blades, using the swirl-free velocity instead of the traditional relative flow angle at runner outlet. It is shown that the swirl-free velocity profile practically does not change with the operating regime. Third, we introduce a constrained variational problem corresponding to the minimization of the flow force while maintaining the prescribed discharge and flux of moment of momentum. This formulation also accounts for a possible central stagnant region to develop when operating the turbine far from the best efficiency point. Fourth, we show that by representing the unknown axial velocity profile with a suitable Fourier–Bessel series, the discharge constraint can be automatically satisfied. The resulting numerical algorithm is robust and produces results in good agreement with available data for both axial and circumferential velocity profiles measured on a model Francis turbine at several operating regimes. Our mathematical model is suitable for the early optimization stages of the runner design, as it provides the swirling flow configuration at runner outlet without actually computing the runner. By optimizing the parameterized swirl-free velocity profile one can achieve through the inverse design approaches the most suitable runner blades configuration at the trailing edge.

1. Introduction

Modern hydraulic turbines meet new challenges associated with the variable demand on the energy market as well as limited energy storage capabilities, resulting in great flexibility required in operation. Quite often turbines tend to be operated over an extended range of regimes far from the best efficiency point. In particular, Francis turbines, which have a fixed pitch runner, experience an abrupt decrease in efficiency and severe pressure fluctuations at off-design operating regimes. In

* Corresponding author. Tel.: +40 256403689; fax: +40 256403892.
E-mail addresses: resiga@mh.mec.upt.ro (R.F. Susan-Resiga), seby@acad-tim.tm.edu.ro (S. Muntean), francois.avellan@epfl.ch (F. Avellan), anton@acad-tim.tm.edu.ro (I. Anton).

0307-904X/$ - see front matter
doi:10.1016/j.apm.2011.03.052

Failure analysis of a Francis turbine runner

D Frunzăverde[1], S Muntean[2], G Mărginean[3], V Câmpian[1], L Marşavina[4], R Terzi[5] and V Şerban[5]

[1]Research Center in Hydraulics, Automation and Heat Transfer, "Eftimie Murgu" University of Reşiţa
P-ţa Traian Vuia 1-4, RO-320085, Reşiţa, Romania
[2]Centre of Advanced Research in Engineering Sciences, Romanian Academy – Timişoara Branch
Bv. Mihai Viteazu 24, RO-300223, Timişoara, Romania
[3] University of Applied Sciences Gelsenkirchen, Neidenburger Str. 10, 45877 Gelsenkirchen, Germany, Email: gabriela.marginean@fh-gelsenkirchen.de
[4]Department of Strength, "Politehnica" University of Timişoara, Bv. Mihai Viteazu 1, RO-300222, Timişoara, Romania
[5]Râmnicu Vâlcea Subsidiary, S.C. Hidroelectrica S.A., Str. Decebal 11, RO-240255, Râmnicu Vâlcea, Romania

E-mail: d.frunzaverde@uem.ro

Abstract. The variable demand on the energy market requires great flexibility in operating hydraulic turbines. Therefore, turbines are frequently operated over an extended range of regimes. Francis turbines operating at partial load present pressure fluctuations due to the vortex rope in the draft tube cone. This phenomenon generates strong vibrations and noise that may produce failures on the mechanical elements of the machine. This paper presents the failure analysis of a broken Francis turbine runner blade. The failure appeared some months after the welding repair work realized in situ on fatigue cracks initiated near to the trailing edge at the junction with the crown, where stress concentration occurs. In order to determine the causes that led to the fracture of the runner blade, the metallographic investigations on a sample obtained from the blade is carried out. The metallographic investigations included macroscopic and microscopic examinations, both performed with light and scanning electron microscopy, as well as EDX - analyses. These investigations led to the conclusion, that the cracking of the blade was caused by fatigue, initiated by the surface unevenness of the welding seam. The failure was accelerated by the hydrogen embrittlement of the filling material, which appeared as a consequence of improper welding conditions. In addition to the metallographic investigations, numerical computations with finite element analysis are performed in order to evaluate the deformation and stress distribution on blade.

1. Introduction

The variable demand on the energy market, as well as the limited energy storage capabilities, requires a great flexibility in operating hydraulic turbines. Therefore, turbines are frequently operated over an extended range of regimes. Francis turbines operating at partial load present pressure fluctuations due to the vortex rope in the draft tube cone. Moreover, the draft tube surges – so-called "Rheingans oscillations" [1] – due to interaction between hydraulic excitation sources (i. e., vortex rope precession) and eigenfrequencies are expected to induce high frequency loading [2]. This phenomenon generates strong vibrations and noise that may produce fatigue failures on the mechanical elements of the machine [3].

Consequently, there are different stages of fatigue damage where defects may nucleate in an initially undamaged section and propagate in a stable manner until catastrophic fracture occurs. For this most general situation, the progression of fatigue damage can be broadly classified into the following stages: i) structural and microstructural changes which cause nucleation of permanent damage; ii) the creation of microscopic cracks; iii) the growth and coalescence of microscopic flaws to form 'dominant' cracks; iv) stable propagation of the dominant macrocrack; v)

Computational Materials Science 64 (2012) 253–259

Contents lists available at SciVerse ScienceDirect

Computational Materials Science

journal homepage: www.elsevier.com/locate/commatsci

Computation of stress distribution in a Francis turbine runner induced by fluid flow

R. Negru [a,*], S. Muntean [b], L. Marsavina [a], R. Susan-Resiga [a], N. Pasca [a]

[a] Politehnica University of Timisoara, Bv. Mihai Viteazu 1, 300222 Timisoara, Romania
[b] Centre of Advanced Research in Engineering Sciences, Romania Academy – Timisoara Branch, Bv. Mihai Viteazu 24, 300223 Timisoara, Romania

ARTICLE INFO

Article history:
Received 2 November 2011
Received in revised form 15 May 2012
Accepted 31 May 2012
Available online 27 June 2012

Keywords:
Hydraulic turbine blade
Fluid flow analysis
Structural stress analysis

ABSTRACT

In this paper the hydraulic stresses induced in a Francis turbine runner blade by steady fluid flow were investigated. Based on the one-way coupled simulation, the approach consists of a fluid flow analysis which provides the distribution of the fluid pressure on the blade, followed by the structural finite element analysis. The three dimensional turbulent flow in both distributor and runner of Francis turbines were computed. The computational domains correspond to interblade channels for the Francis turbine distributor and runner, respectively. In order to couple the steady absolute distributor flow field with the runner steady relative flow, a mixing interface technique is used on the conical distributor–runner interface. The hydrodynamic field is computed in seven operating points at constant head from part load to full load conditions. The pressure coefficient distribution on the blade is plotted in order to evaluate the blade loading and region with cavitational risk. Further, the stress distributions were obtained by a structural finite element analysis performed for the steady loading in order to determine the areas with highest stress values.

1. Introduction

Hydropower is the largest source of renewable energy and it is the most efficient way to generate electricity. Besides being an environmentally clean and renewable source of energy, the variable demand on the energy market, as well as the limited energy storage capabilities, requires a high flexibility in operating hydraulic turbines. As a result, hydraulic turbines are frequently operated with a large number of start–stop cycles and over an extended range of regimes quite far from the best efficiency point (BEP), often at part load conditions (PL) with high pressure fluctuations generated by vortex rope in the draft tube. Therefore, strong vibrations are induced which can produce fatigue failure on the mechanical components [1], especially on the turbine runner blades [2].

The loads acting on the Francis turbine runner can be steady or unsteady. The first type includes the fluid pressure and the centrifugal force. The second type is composed of the high frequency pressure fluctuations due to stator–rotor interaction as well as vortex rope phenomenon. Under these loading conditions, the development of fatigue cracks is the major concern with regards to the structural integrity of the runners and its represent a threat to safe operation of the hydraulic turbines.

Several studies were reported in the last years for the computation and experimental measurement of the static and dynamic stresses induced in Francis turbine blades by the steady and the unsteady loading, respectively. The static stresses in the blades of a Francis turbine runner at different operating points were investigated by Saeed [3] by using the finite element method with inlet boundary conditions imposed from analytical computation. Moreover, in this case the three-dimensional flow generated by guide vanes is not taken into account. A good agreement between the finite element analysis and the strain gauge experimental measurements was found by Bjørndal et al. [4] for the flow induced static stresses in a medium head Francis runner. Based on a sequential coupled fluid–structure interaction, a similar approach was used by Sadowski and Golewski [5] in the multidisciplinary analysis of the thermal heat transfer of the turbine blades in combustion engines.

The purpose of this paper is the analysis of stress field induced in the runner blades of a medium specific speed Francis turbine by the steady loading at seven operating regimes at constant head. The following analysis is based on the one-way coupled simulation approach. Section 2 consists of a computational fluid dynamics (CFD) simulation which provides the fluid pressure distribution on the blade. It is followed by the structural finite element analysis (FEA) in Section 3. The conclusions are drawn in last section.

2. Fluid flow analysis

The case corresponds to a medium specific speed Francis turbine with parameters presented in Table 1, where: Q [m^3/s] – discharge,

* Corresponding author. Address: Politehnica University of Timisoara, Department of Mechanics and Strength of Materials, Bv. Mihai Viteazu 1, 300222 Timisoara, Romania. Tel.: +40 256 40 35 75, fax: +40 256 40 35 23.

E-mail addresses: blackradu@yahoo.com, radun@mec.upt.ro (R. Negru), seby@acad-tim.tm.edu.ro (S. Muntean), msvina@mec.upt.ro (L. Marsavina), resiga@mh.mec.upt.ro (R. Susan-Resiga), nicu_pasca21@yahoo.com (N. Pasca).

0927-0256/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.commatsci.2012.05.073

Alin Ilie Bosioc
Scientific Researcher
Center for Advanced Research
in Engineering Science,
Romanian Academy – Timisoara Branch,
Bvd. Mihai, Viteazu 24,
RO-300223, Timisoara, Romania
e-mail: alin@mh.mec.upt.ro

Romeo Susan-Resiga[1]
Professor
Hydraulic Machinery Department,
"Politehnica" University of Timisoara,
Romania, Bvd. Mihai Viteazu 1,
RO-300222, Timisoara, Romania
e-mail: resiga@mh.mec.upt.ro

Sebastian Muntean
Senior Researcher
Center for Advanced Research
in Engineering Science,
Romanian Academy – Timisoara Branch,
Bvd. Mihai Viteazu 24,
RO-300223, Timisoara, Romania
e-mail: seby@acad-tim.tm.edu.ro

Constantin Tanasa
Research Center for Engineering
of Systems with Complex Fluids,
"Politehnica" University of Timisoara,
Bvd. Mihai Viteazu 1,
RO-300222, Timisoara, Romania
e-mail: costel@mh.mec.upt.ro

Unsteady Pressure Analysis of a Swirling Flow With Vortex Rope and Axial Water Injection in a Discharge Cone

The variable demand of the energy market requires that hydraulic turbines operate at variable conditions, which includes regimes far from the best efficiency point. The vortex rope developed at partial discharges in the conical diffuser is responsible for large pressure pulsations, runner blades breakdowns and may lead to power swing phenomena. A novel method introduced by Resiga et al. (2006, "Jet Control of the Draft Tube in Francis Turbines at Partial Discharge," Proceedings of the 23rd IAHR Symposium on Hydraulic Machinery and Systems, Yokohama, Japan, Paper No. F192) injects an axial water jet from the runner crown downstream in the draft tube cone to mitigate the vortex rope and its consequences. A special test rig was developed at "Politehnica" University of Timisoara in order to investigate different flow control techniques. Consequently, a vortex rope similar to the one developed in a Francis turbine cone at 70% partial discharge is generated in the rig's test section. In order to investigate the new jet control method an auxiliary hydraulic circuit was designed in order to supply the jet. The experimental investigations presented in this paper are concerned with pressure measurements at the wall of the conical diffuser. The pressure fluctuations' Fourier spectra are analyzed in order to assess how the amplitude and dominating frequency are modified by the water injection. It is shown that the water jet injection significantly reduces both the amplitude and the frequency of pressure fluctuations, while improving the pressure recovery in the conical diffuser. [DOI: 10.1115/1.4007074]

Keywords: decelerated swirling flow, vortex rope, water injection method, unsteady pressure, experimental investigation

1 Introduction

The swirling flow emerging from a Francis turbine runner has a major influence in a draft tube cone downstream. It produces self-induced flow instabilities leading to pressure fluctuations and ultimately to the draft tube surge [1]. At part load operation it develops a precessing helical vortex (also known as vortex rope) in the Francis turbine draft tube cone. Consequently, the vortex rope generates pressure fluctuations, additional hydraulic losses, and power swing phenomena at the electrical generator [2]. Unsteady pressure measurements for hydraulic Francis turbines operating at part load have been performed on site by Wang et al. [3] and Baya et al. [4]. They reveal a low frequency oscillation (from 1/5 to 1/3 of the runner rotation frequency) associated with the vortex rope. Extensive unsteady wall pressure measurements in the elbow draft tube of the hydraulic Francis turbine model at partial discharge are performed by Arpe et al. [5]. The pressure waves' source was located near the inner part of the elbow draft tube based on experimental data. Moreover, these waves are propagated in all hydraulic systems. The synchronous nature of the pressure fluctuations and the pressure distribution along the draft tube suggests hydro acoustic resonance of the entire hydraulic system.

Different methods were proposed in order to mitigate the instabilities produced by the vortex rope. Examples include aerators mounted at the inlet of the cone, stabilizer fins or runner cone extensions [6]. Numerical Francis turbine simulation of the flow was performed by Qian et al. [7] in order to investigate the air admission from the spindle hole. Analysis of the draft tube cone air admission showed that the amplitude and the pressure difference in the cross section of the draft tube decreases while the blade frequency pressure pulsation increases in front of the runner. Therefore, proper air discharge to mitigate the pressure pulsations in the draft tube cone of the hydraulic turbine should be chosen according to specific operating conditions. These methods lead to some improvements in reducing the pressure pulsations for a narrow regime but they are not effective or even increase the unwanted effects. Given by the energy injected in the draft tube cone these methods can be divided into active, passive or semipassive control. If an external energy source is used to mitigate or eliminate the vortex rope, the control is called active. Examples of active control include air injection either downstream (through runner cone) or upstream (through wicket gates trailing edge) of the runner [6,8], or tangential water jets at the discharge cone wall [9]. The control involving no additional energy to destroy the vortex rope is called passive. Passive control methods include fins mounted on the cone [10,11], extending cones mounted on the runner's crown [12] or using J-grooves [13].

Resiga et al. [14] proposed a new method in order to mitigate the vortex rope, axial water injected through the runner's crown along to the discharge cone. An experimental test rig was designed and developed in the Hydraulic Machinery Laboratory at "Politehnica" University of Timisoara in order to investigate this new method. The rig is used to determine the parameters of the swirling flow with vortex rope and the optimum water jet in order to mitigate the pressure fluctuations.

[1]Corresponding author.

Contributed by the Fluids Engineering Division of ASME for publication in the JOURNAL OF FLUIDS ENGINEERING. Manuscript received August 10, 2011; final manuscript received June 14, 2012; published online July 30, 2012. Assoc. Editor: Hassan Peerhossaini.

A. Javadi[1]
Division of Fluid Dynamics,
Department of Applied Mechanics,
Chalmers University of Technology,
Göteborg SE-412 96, Sweden
e-mail: ardalan.javadi@chalmers.se

A. Bosioc
Assistant Professor
Department of Hydraulic Machinery,
University Politehnica Timişoara,
Bv. Mihai Viteazu, No. 1,
Timişoara Ro-300222, Romania
e-mail: alin.bosioc@upt.ro

H. Nilsson
Professor
Division of Fluid Dynamics,
Department of Applied Mechanics,
Chalmers University of Technology,
Göteborg SE-412 96, Sweden
e-mail: hani@chalmers.se

S. Muntean
Center for Advanced Research in
Engineering Sciences,
Romanian Academy—Timişoara Branch,
Bv. Mihai Viteazu, No. 24,
Timişoara Ro-300223, Romania
e-mail: seby@acad-tim.tm.edu.ro

R. Susan-Resiga
Professor
Department of Hydraulic Machinery,
University Politehnica Timişoara,
Bv. Mihai Viteazu, No. 1,
Timişoara Ro-300222, Romania
e-mail: romeo.resiga@upt.ro

Experimental and Numerical Investigation of the Precessing Helical Vortex in a Conical Diffuser, With Rotor–Stator Interaction

The flow unsteadiness generated in a swirl apparatus is investigated experimentally and numerically. The swirl apparatus has two parts: a swirl generator and a test section. The swirl generator which includes two blade rows, one stationary and one rotating, is designed such that the emanating flow at free runner rotational speed resembles that of a Francis hydroturbine operated at partial discharge. The test section consists of a conical diffuser similar to the draft tube cone of a Francis turbine. Several swirling flow regimes are produced, and the laser Doppler anemometry (LDA) measurements are performed along three survey axes in the test section for different runner rotational speeds (400–920 rpm), with a constant flow rate, 30 l/s. The measured mean velocity components and its fluctuating parts are used to validate the results of unsteady numerical simulations, conducted using the FOAM-EXTEND*-3.0 CFD code. Furthermore, phase-averaged pressure measured at two positions in the draft tube is compared with those of numerical simulations. A dynamic mesh is used together with the sliding general grid interfaces (GGIs) to mimic the effect of the rotating runner. The delayed detached-eddy simulation method, conjugated with the Spalart–Allmaras turbulence model (DDES–SA), is applied to achieve a deep insight about the ability of this advanced modeling technique and the physics of the flow. The RNG $k-\varepsilon$ model is also used to represent state-of-the-art of industrial turbulence modeling. Both models predict the mean velocity reasonably well while DDES–SA presents more realistic flow features at the highest and lowest rotational speeds. The highest level of turbulence occurs at the highest and lowest rotational speeds which DDES–SA is able to predict well in the conical diffuser. The special shape of the blade plays more prominent role at lower rotational speeds and creates coherent structures with opposite sign of vorticity. The vortex rope is captured by both turbulence models while DDES–SA presents more realistic one at higher rotational speeds.*
[DOI: 10.1115/1.4033416]

1 Introduction

Hydraulic turbomachines have played an important role in the procurement of renewable energy for more than a century. Accompanied with the context of extensive technological development, their design for efficiency and reliability has reached an excellent point of quality. The fixed-speed turbines in the power plants work most efficiently at the design point that delineates the best combination of the speed, head, and discharge. At the design point, water turbines generally operate with little swirl entering the draft tube and without flow separations from the draft tube walls, whereas at off-design, at both full and part load, the flow leaving the turbine has a large tangential component. The hydraulic efficiency drops sharply due to this tangential component. It is known that a mitigated level of residual swirl downstream the runner delays boundary layer separation at the draft tube wall in the conical section, and thus aids the pressure recovery [1]. The large swirl intensities, however, can decrease the performance by forming an on-axis recirculation region [2]. Clausen et al. [1] showed that there is a narrow range of swirl intensities that prevents both recirculation and separation. When swirl increases, a vortex breakdown occurs. The vortex breakdown is an abrupt change in the core of a slender vortex and typically develops downstream into a recirculatory "bubble" or a helical pattern [3–6]. The breakdown in a pipe with constant cross section is an inviscid process, where a slight viscosity of the fluid, divergence or convergence of the pipe, or increment of tangential vorticity at the inlet are perturbations from the inviscid case. The characteristics of various breakdown states, or modes, in swirling flows depend on the swirl number, the Reynolds number (Re), and geometry-induced axial pressure gradients in the flow. The swirl number is defined as

$$\mathrm{Sr} = \frac{1}{R_2}\frac{\int_{R_1}^{R_2} r^2 UW dr}{\int_{R_1}^{R_2} rW^2 dr} \quad (1)$$

where R_2 is the outer radius of the cross section where Sr is calculated, R_1 is the inner radius of the cross section, and W and U are the mean axial and tangential velocity. The velocity components are averaged circumferentially and temporarily. The inclusion of a slight swirl ($\mathrm{Sr}=0.1$) at the inlet can reduce the helical precession speed and may cause the rotation of the precessing helical vortex to be against the mean swirl [7]. Several states of precession are presented as the swirl intensity increases, in which the helical precession, as well as the spiral structure, reverses the direction of rotation [8]. When the swirl increases to $\mathrm{Sr}=0.5$, a central recirculation zone occurs, which is a typical manifestation of vortex

[1]Corresponding author.

Contributed by the Fluids Engineering Division of ASME for publication in the JOURNAL OF FLUIDS ENGINEERING. Manuscript received May 12, 2015; final manuscript received April 5, 2016; published online June 8, 2016. Assoc. Editor: Bart van Esch.

Sebastian Muntean, Ionel Drăghici, Gheorghiţă Gînga, Liviu Eugen Anton and Alexandru Baya

Hydrodynamic Design of a Storage Pump Impeller using Inverse Method and Experimental Investigation of the Global Performances

The paper presents the hydrodynamic design of a new impeller using inverse method for one single stage double-flux pumped storage. The new impeller is designed for an existing hydraulic passage using inverse method. The old and new impellers are experimentally investigated without and with an inducer in order to assess the global performances. An improved cavitational behaviour is obtained for both solutions with inducer.

1 Introduction

The pumped storage hydropower is valid and competitive solution for large energy storage in order to balance fluctuating renewable sources (such as wind and solar). Therefore, refurbishment and modernization of the units is needed [1]. In our case, one single stage double-flux pumped storage unit is considered (**Figure 1**). The unit was designed in order to deliver large discharge values at high efficiency operating under tolerant cavitation conditions. In situ measurements were performed after 27 years of operation in order to assess the unit performances [2].

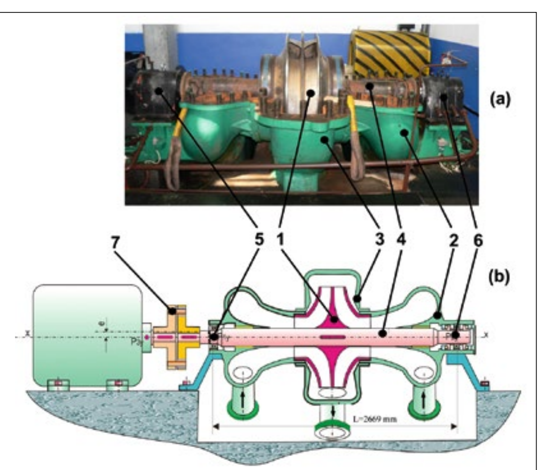

Figure 1: Single stage double-flux pumped storage view (a) and cross section (b); the pumped storage components are (1) double flux impeller, (2) suction elbow, (3) volute, (4) shaft, (5) radial bearing, (6) radial-axial bearing and (7) elastic coupling (Source: Muntean et al.)

The paper presents the hydrodynamic design of a new impeller using inverse method for one single stage double-flux pumped storage. In our case, a new impeller is designed for an existing hydraulic passage. There are two main approaches in order to achieve this goal:

i) a new geometry of the impeller based on a classical design method followed by a-posteriori analysis to assess the performances [3], [4];
ii) a new geometry of the impeller for a known flow configuration using the inverse design method developed by Zangeneh [5].

The design objective is to improve the cavitational performances of the pumped storage impeller. The paper presents the hydrodynamic criteria used in order to design the impeller. The conclusions are drawn in the last section.

2 Hydrodynamic design of the impeller

2.1 Inverse design method

The preliminary analysis and design of turbomachinery flow can be performed within the inviscid fluid assumption, since losses can be considered negligible at the design operating point. Of course, maximizing the machine efficiency requires the evaluation of viscous losses, but the first step is to get a preliminary design within the loss-free framework.

For the runner bladed region it is convenient to consider the relative flow equations, where the relative velocity is $\vec{W} = \vec{V} - \vec{\Omega} \cdot r$ and the relative specific energy $E_R = E - \Omega(rV_\theta)$. The corresponding steady relative flow equations are:

$$\nabla \cdot \vec{W} = 0$$

$$(\nabla \cdot \vec{V}) \cdot \vec{W} = - \nabla E_R \qquad (1)$$

$$E_R \equiv \frac{p}{\rho} + \frac{W^2}{2} - \frac{(\Omega r)^2}{2}$$

From the momentum equation for relative flow we have $\vec{W} \cdot \nabla E_R = 0$, i.e. the relative specific energy E_R remains constant along relative flow streamlines. For the three-dimensional flow in the bladed regions, the blade produces a pressure difference between suction and pressure sides, which in turn produces a circumferential pressure gradient that deflects the flow. The flow inside the interblade channel is essentially three-dimensional since the streamlines originating on a circle (normal to the machine axis) do not remain on an axi-symmetric surface [6]. Instead, the streamlines close to the blade pressure side are pushed radial towards the hub, while the streamlines in the neighbourhood of blade suction side are deflected toward the shroud. It is obvious that at the blade trailing edge there is a significant departure in radial direction between streamlines originating at the same radius upstream the blade. However, a simplified axi-symmetric model for the hub-to-shroud turbomachinery flow considers that the stream surfaces retain axial symmetry within the blade regions as well.

The simplified axi-symmetric computation retains only the average inter-blade pressure since no circumferential gradient is allowed. As a result, it is required an ar-

LDV measurements of the velocity field on the inlet section of a pumped storage equipped with a symmetrical suction elbow for variable discharge values

I Drăghici[1], S Muntean[2], A I Bosioc[1] and L E Anton[1]
[1] Department of Hydraulic Machinery, University Politehnica Timisoara, Bv. Mihai Viteazu, No. 1, Ro-300222, Timisoara, Romania
[2] Center for Advanced Research in Engineering Sciences, Romanian Academy – Timisoara Branch, Bv. Mihai Viteazu, No. 24, Ro-300223, Timisoara, Romania

E-mail: seby@acad-tim.tm.edu.ro

Abstract. The storage pumps are equipped with various types of inlet casings. The flow non-uniformity is generated by the suction elbows being ingested by the impeller leading to unsteady phenomena and worse cavitational behaviour. A symmetrical suction elbow model corresponding to the double flux storage pump was manufactured and installed on the test rig in order to assess the flow field at the pump inlet. The experimental investigations are performed for 9 discharge values from 0.5 to 1.3 of nominal discharge. LDV measurements are performed on the annular section of the pump inlet in order to quantify the flow non-uniformity generated by the symmetrical suction elbow. Both axial and circumferential velocity components are simultaneously measured on the half plane (180°) of the annular inlet section along to 19 survey axis with 62 points on each. The flow field on the next half plane is determined tacking into account the symmetry. As a result, the flow map on the pump inlet annular section is reconstructed revealing a significant variation of the circumferential velocity component. The absolute flow angle is computed showing a significant variation of ±38°.

1. Introduction

The large pumping units are widely used in industry to store energy, to provide cooling, to ensure propulsion and to transfer fluids. These pumps can reach high values of flow rate at high efficiency with a tolerate level of cavitation behaviour if its selection criteria with respect to suction behaviour and optimum efficiency meet various system requirements [1]. The large pumping units consume a significant amount of the electrical energy. Therefore, even smaller improvements of energetical and cavitational performances of the large pumps are significant to minimize the overall costs.

Three pumped storage power plants (PSPP) are integrated in one of the most complex hydropower system from Romania located on the Lotru river, Cojocar [2]. PSPP considered like test case is equipped with two units. Each unit includes synchronous electrical motor (blue), elastic coupling (yellow) and double flux pumped storage (green) like in Fig. 1. The unit was designed in order to deliver large flow rate at high efficiency operating under tolerant cavitation conditions. In situ measurements were performed after 25 years of operation in order to evaluate the unit performances, Anton [3]. A worse cavitational behaviour was revealed in service. The same problems were reported by Škerlavaj et al. [4] in operation of the units from Fuhren HPP.

THE PUBLISHING HOUSE
OF THE ROMANIAN ACADEMY

PROCEEDINGS OF THE ROMANIAN ACADEMY, Series A,
Volume 17, Number 3/2016, pp. 237–244

UNSTEADY PRESSURE FIELD ANALYSIS AT PUMP INLET EQUIPPED WITH A SYMMETRICAL SUCTION ELBOW

Ionel A. DRĂGHICI[1], Sebastian MUNTEAN[2], Alin I. BOSIOC[1], Gheorghiţă GÎNGA[3], Liviu E. ANTON[1]

[1] "Politehnica" University of Timisoara, Department of Hydraulic Machinery, RO-300222, Timisoara, Romania
[2] Center for Advanced Research in Engineering Sciences, Romanian Academy-Timisoara Branch, RO-300223, Timisoara, Romania
[3] Research Center for Engineering Systems with Complex Fluids, "Politehnica" University of Timisoara, RO-300222, Timisoara, Romania
Corresponding author: Sebastian MUNTEAN, E-mail: seby@acad-tim.tm.edu.ro

Abstract. The flexible operation of the large hydraulic pumps requires to be run at variable discharges and speeds. The large pumps are equipped with a suction elbow which generates non-uniform flow at the impeller inlet leading to unsteady hydrodynamic phenomena. The experimental investigation presented in this paper is focused on unsteady pressure measurements at pump inlet. Two vortices generated by suction elbow are visualized being ingested by impeller. The equivalent amplitude and frequency associated to hydrodynamic phenomenon at pump inlet are determined at variable discharges and several speeds in order to be quantified the unsteady flow field. A discrimination procedure is applied on unsteady signals in order to be evaluated the plunging and rotating components. The experimental investigations are performed for nine discharge values from 16.75 l/s to 43.55 l/s and four impeller speed values from 2 700 rpm to 3 000 rpm.

Key words: unsteady pressure, large pump, symmetrical suction elbow, variable discharge and speed.

1. INTRODUCTION

The large pumping units are widely used in industry to store energy [1], to ensure cooling or heating in different systems [2] and to transport water [3]. The flexible operation of the large hydraulic pumps requires to be run at variable discharges and speeds. Constructively, the solutions for large pumps are different than regular ones [4]. A suction elbow with complex three-dimensional geometry is installed upstream to the impeller of large pumps or to the first impeller of the multistage pumps. This suction elbow generates circumferential non-uniformity in velocity distribution at the impeller eye due to the geometry and the flow around the shaft [5–8]. Consequently, the flow with pre-rotation is generated over roughly one half of the impeller inlet section and counter-rotation in the second half [7–12]. This non-uniform flow is ingested by the impeller [9–12] leading to the following: (i) loss in efficiency [14]; (ii) noise and vibrations are excited [15–17]; (iii) radial forces are generated [18]; (iv) cavitation erosion on the impeller blades and even its damage [19–21]; (v) lifetime of mechanical components is reduced [22].

The paper investigates the unsteady pressure field at the inlet section of a pump equipped with a symmetrical suction elbow for variable discharges and speeds. The experimental setup is described in Section 2. The pressure data analysis is presented in Section 3 while the conclusions are drawn in last section.1.

2. EXPERIMENTAL SETUP

A test rig is available at "Politehnica" University Timisoara in order to investigate the pump hydrodynamics (Fig. 1). The main components consist in two reservoirs of 1 m^3, a hydraulic pump with characteristic speed $n_q = n Q^{0.5} / H^{0.75}$ ~30 and a 37 kW electromotor. The test rig is equipped with a variable speed system control. The DTC-inverter varies the speed of the induction motor from 500 rpm up to 3000 rpm [23]. An acquisition system was implemented to acquire sensors data for overall pressure, discharge and

THE PUBLISHING HOUSE
OF THE ROMANIAN ACADEMY

PROCEEDINGS OF THE ROMANIAN ACADEMY, Series A,
Volume 14, Number 4/2013, pp. 317–325

PUMP INDUCER OPTIMIZATION BASED ON CAVITATION CRITERION

Irina Georgeta MOISĂ[1], Romeo SUSAN-RESIGA[1], Sebastian MUNTEAN[2]

[1] "Politehnica" University of Timişoara, Romania
[2] Center of Advanced Research in Engineering Sciences, Romanian Academy - Timişoara Branch, Romania
E-mail: `resiga@mh.mec.upt.ro`

The paper presents the optimization procedure of a pump inducer taking into account the cavitation criterion. The pump inducer optimization procedure is based on inverse design method where the inflow and outflow conditions are known. Axial flow is considered as upstream condition while the downstream swirling flow is suited for the pump impeller inlet, respectively. The key ingredient in this procedure is the function of loading shape which provides the distribution of blade loading from leading edge to trailing edge. A new parametric analytical expression is introduced in this paper for the loading shape function. The objective function corresponds to the minimum pressure coefficient with reverse sign, which is minimized within the chosen parameter space. This procedure is applied for optimize a pump inducer with three blades, with improved cavitating performance.

Key words: pump inducer, inverse design method, cavitational optimization.

1. INTRODUCTION

Nowadays, pumping systems account for nearly 20% of the world's electrical energy demand and range from 25–50% of the energy usage in certain industrial plant operations [1]. The pumping systems are widely used in industry to store energy, to provide cooling and lubrication services, to transfer fluids for processing, and to provide the driving force in hydraulic systems. Clearly, pumping systems consume a significant amount of the total electrical energy. The key element of the pumping system is the pumping unit which includes the electrical motor and the hydraulic pump. In the manufacturing sector, pumps represent 27% of the electricity used by industrial systems [2]. As a result, the improvements of energetic and cavitational performances of the pumps become more and more important. A common technical solution is to install an inducer at the pump impeller inlet in order to improve the cavitational behavior of the pumps [3]. Generally, inducers are axial flow impellers with a small blade numbers and long blades. As a result, an inducer increases the inlet static pressure at the impeller inlet thus improving the overall cavitational performances. Using conventional design the inducers blades with helical surfaces and a straight line or a combination of straight lines and curves is chosen for the camber line [4]. The extensive experimental and numerical investigations have been performed in our group in order to improve the performances of these inducers during last two decades [4–6]. However, the inducer design guidelines are based on the empirical relationship between geometry parameters and performance characteristics [7]. Moreover, the designed inducers based on conventional methods may have leading edge backflow even at the design point. Therefore, the three-dimensional inverse design method developed by Zangeneh et al. [8] was applied to improve the inlet backflow characteristics of highly loaded turbopump inducers for a liquid hydrogen rocket engine [9]. Optimizing the blade loading distribution using the three-dimensional inverse design method eliminates this inlet backflow [10]. Consequently, the numerical and experimental investigations were performed in order to compare the performance of inducers with different blade loadings, [11].

In situ experimental investigations performed by Anton [12] on large double suction storage pumps have revealed serious cavitational problems. As a result, a model inducer (Fig. 1) was designed using inverse design method [13] based on the technical specifications shown in Table 1. The loading function $\partial(rv_u)/\partial x$ imposed along to three sections of the blade: S_H near the hub, S_M in the middle and S_T near the tip and