Technical Papers

6  DAM BURST IN THE SAPUCAÍ RIVER (MG): A HEC-RAS SOFTWARE SIMULATION
   Barbosa, Alexandre Augusto; Rissatto, Lucas Stéfano; Franco, Patrícia Mara; Silva, Laércio Rafael Colucci Marques da

11 SIMPLE MODELLING FOR MAXIMUM FLOW RATES DETERMINATION TO BE APPLIED IN ECONOMICALLY FEASIBLE SMALL HYDROPOWER PLANTS
   Santos, Ivan Felipe Silva dos; Vieira, Nathalia Duarte Brazi; Tiazgo Filho, Geraldo Lúcio; Barros, Regina Mambelli; Souza, André Luiz

14 ALE FINITE ELEMENT METHOD FOR FLUID FLOW SIMULATIONS OVER MOVING GUIDE VANES
   Noleto, Luciano Gonçalves; Barcelos Junior, Manuel Nascimento Dias; Brasil Junior, Antonio C. P.

19 ON THE HYDRODYNAMICS OF A ROW ARRANGEMENT OF HYDROKINETIC PROPELLER TURBINES
   Brasil Junior, Antonio C.P.; Mendes, Rafael C. F.; Oliveira, Taygoara F.; Andriampanary, Tamby; Koundri, Smaine; Mesquita, André L.A.

25 NUMERICAL SIMULATIONS OF COMPLEX FLOWS IN HYDRAULIC TURBOMACHINERY. ACTUAL R&D LINES IN THE ‘CFD/APPLIED MATHS LABORATORY’ GROUP
   Coussirat M., Fontanals A., Panella L., Henderson G., Aguirre R

35 AN ANALYSIS OF THE MECHANICAL STRESS IN HYDROELECTRIC TURBINES STAY-VANES UNDER DIFFERENT OPERATIONAL STATES OF THE ELECTRIC GENERATOR
   Gonzalez, Facundo E.; Kelm, Diego A.; Kolodziej, Javier E.; Tarnowski, Gabriel A.; Astelli, Raúl; Bordón, Hugo

39 FLOW PROFILE INVESTIGATION AT TUCURUI HYDRO POWER PLANT TAILRACE
   Bortoni, Edison C.; Bertrand, Olivier; Sauvaget, Patrick; Santos, Luciano T.; Vasconcellos, Ricardo C.

Technical Notes

44 ANALYSIS OF STATOR WINDINGS INSULATION OF COARACY NUNES HYDROELECTRIC PLANT GENERATING UNITSE
   P. R. M. Vilhena; F. S. Brasil; V. Dmitriev; C. J. S. Santos
American Journal of Hydropower, Water and Environment Systems

Editorial

A publication of Latin American Working Group of the International Association on Hydraulic Research

The search for renewable energy sources is presented as one of the pillars for actions aiming the reduction of greenhouse gases (GHG) net emissions. Among the renewable energy sources, hydropower is presented as fundamental because it provides greater security in the amount of energy when compared to other forms of energy, such as wind and solar. However, it is necessary to do mention that all forms of above mentioned energy are subject to the inherent uncertainties to predictability of power generation based on historical data and by the fact that their forecasts are based on statistical models. On the other hand, there is growing demand for energy in order to support industrial growth, society development, and both must be carried out in a sustainable manner, with social and environmental constraints.

In this search for a common point that makes feasible the growth based on sustainable development, and taking into account the limitations imposed by nature itself, the American Journal of Hydropower, Water and Environmental Systems (AJHWE) aims to present papers that can effectively contribute to the development of new technologies. Among the emerging technologies of scientific studies, the hydrokinetic turbines that take advantage from the velocity of a discharge for power generation, or the low- or very low-head turbines, which generate energy much more from the watercourse discharge contribution than by the net available head can be mentioned.

All these researches have the potential to support new policies to make the new technologies development feasible under technical, social and economic point of view and to subsidize to renewable energy sources aiming a search of a better future.

We hope to contribute in some way in this direction. We wish you to enjoy reading of the papers selected for this edition of AJHWES.

Yours sincerely,

Geraldo Lúcio Tiago Filho
Editor in Chief

Regina Mambeli Barros
Technical Editor
INSTRUCTIONS FOR AUTHORS

AMERICAN JOURNAL OF HYDROPOWER, WATER AND ENVIRONMENT SYSTEMS

A publication of Latin American Working Group of the International Association for Hydro-Environment Engineering and Research-IAHRS

All papers must be submitted in English. In case the author wants to translate the article through the journal all costs for the translation will be charged on the account of the author.

1. Formatting articles

1.1. Article structure

1.1.1 Subdivision - numbered sections

Divide your article into clearly defined and numbered sections. Subsections should be numbered 1.1 (then 1.1.1, 1.1.2, ...), 1.2, etc. (the abstract is not included in section numbering). Use this numbering also for internal cross-referencing: do not just refer to 'the text'. Any subsection may be given a brief heading. Each heading should appear on its own separate line.

1.1.2 Format

All text of the manuscript must be located within a 170 mm by 252 mm rectangle of a white A4 page or within 170 mm by 240 mm for the letter format. The margins are given in Table 1. An example of the page format is given in Fig. 1

[Table 1]: Page margin for manuscripts.

<table>
<thead>
<tr>
<th>Margin Position</th>
<th>Top</th>
<th>Bottom</th>
<th>Left</th>
<th>Right</th>
</tr>
</thead>
<tbody>
<tr>
<td>Margin size (cm)</td>
<td>2.0</td>
<td>2.5</td>
<td>2.0</td>
<td>2.0</td>
</tr>
</tbody>
</table>

All text should be single spaced, black and in 12-point type. “Times News Roman” or a similar proportional font should be used. Total length 15 pages in Word.

The terminology given in the IEC Technical Report for the Nomenclature of Hydraulic Machinery is recommended.

Introduction

State the objectives of the work and provide an adequate background, avoiding a detailed literature survey or a summary of the results.

Material and methods

Provide sufficient details to allow the work to be reproduced. Methods already published should be indicated by a reference: only relevant modifications should be described.

Theory/calculation

A Theory section should extend, not repeat, the background to the article already dealt with in the Introduction and lay the foundation for further work. In contrast, a Calculation section represents a practical development from a theoretical basis.

Results

Results should be clear and concise.

Discussion

This should explore the significance of the results of the work, not repeat them. A combined Results and Discussion section is often appropriate. Avoid extensive citations and discussion of published literature.

Conclusions

The main conclusions of the study may be presented in a short Conclusions section, which may stand alone or form a subsection of a Discussion or Results and Discussion section.

References

Within the text, references should be cited in numerical order according to their order of appearance. The numbered reference citation within text should be enclosed in brackets.

After the second edition all papers must have at least one reference of the American Journal of Hydropower, Water and Environment Systems.

Example: It was shown by Prusa [1] that the width of the plume decreases under these conditions.

In the case of two citations, the numbers should be separated by a comma [1,2]. In the case of more than two references, the numbers should be separated by a dash [5-7].

List of References. References to original sources for cited material should be listed together at the end of the paper; footnotes should not be used for this purpose. References should be arranged in numerical order according to the sequence of citations within the text. Each reference should include the last name of each author followed by his initials.

(1) Reference to journal articles and papers in serial publications should include:
- last name of each author followed by their initials
- year of publication
- abbreviated title of publication in which it appears
- full title of the cited article in quotes, title capitalization
- volume number (if any) (Do not include the abbreviation, "Vol.")
- issue number (if any) in parentheses (Do not include the abbreviation, "No.")
- inclusive page numbers of the cited article (include "pp.")

(2) Reference to textbooks and monographs should include:
- last name of each author followed by their initials
- year of publication
- titles in examples may be in italic
- publisher
- city of publication
- inclusive page numbers of the work being cited (include "pp.")
- chapter number (if any) at the end of the citation following the abbreviation, "Chap."

(3) Reference to individual conference papers, papers in compiled conference proceedings, or any other collection of works by numerous authors should include:
- last name of each author followed by their initials
- year of publication
- full title of the cited paper in quotes, title capitalization
- individual paper number (if any)
- full title of the publication
- initials followed by last name of editors (if any), followed by he abbreviation, "eds."
- publisher
- city of publication
- volume number (if any) in boldface if a single number, include, "Vol." if part of larger identifier (e.g., “PVP-Vol. 254”)
- inclusive page numbers of the work being cited (include "pp.")

(4) Reference to theses and technical reports should include:
- last name of each author followed by their initials
- year of publication
- full title in quotes, title capitalization
- report number (if any)
- publisher or institution name, city
Sample References


1.1.2 Essential title page information

• Title. Concise and informative. Titles are often used in information-retrieval systems. Avoid abbreviations and formulae where possible.

• Author names and affiliations. Where the family name may be ambiguous (e.g., a double name), please indicate this clearly. Indicate all affiliations with a number immediately after the author’s name and in front of the appropriate address. Provide the full postal address of each affiliation, including the country name and, if available, the e-mail address of each author.

• Author résumé. The author must inform the graduation degree, post graduation, affiliation and email address. The résumé must not exceed 150 characters.

• Corresponding author. Clearly indicate who will handle correspondence at all stages of refereeing and publication, also post-publication. Ensure that e-mail address and the complete postal address are provided. Contact details must be kept up to date by the corresponding author.

• Present/permanent address. If an author has moved since the work described in the article was done, or was visiting at the time, a ‘Present address’ (or ‘Permanent address’) may be indicated as a footnote to that author’s name. The address at which the author actually did the work must be retained as the main, affiliation address. Superscript Arabic numerals are used for such footnotes.

Abstract

A concise and factual abstract is required. The abstract should state briefly the purpose of the research, the principal results and major conclusions. An abstract is often presented separately from the article, so it must be able to stand alone. For this reason, References should be avoided, but if essential, then cite the author(s) and year(s). Also, non-standard or uncommon abbreviations should be avoided, but if essential they must be defined at their first mention in the abstract itself.

Keywords

Immediately after the abstract, provide a maximum of 6 keywords, using American spelling and avoiding general and plural terms and multiple concepts (avoid, for example, ‘and’, ‘of’). Be sparing with abbreviations: only abbreviations firmly established in the field may be eligible. These keywords will be used for indexing purposes.

Abbreviations

Define abbreviations that are not standard in this field in a footnote to be placed on the first page of the article. Such abbreviations that are unavoidable in the abstract must be defined at their first mention there, as well as in the footnote. Ensure consistency of abbreviations throughout the article.

Acknowledgements

Collate acknowledgements in a separate section at the end of the article before the references and do not, therefore, include them on the title page, as a footnote to the title or otherwise. List here those individuals who provided help during the research (e.g., providing language help, writing assistance or proof reading the article, etc.).

Nomenclature and units

Follow internationally accepted rules and conventions: use the international system of units (SI). If other quantities are mentioned, give their equivalent in SI.

Math formulae

Present simple formulae in the line of normal text where possible and use the solids (/) instead of a horizontal line for small fractional terms, e.g., X/Y. In principle, variables are to be presented in italics. Powers of e are often more conveniently denoted by exp. Number consecutively any equations that have to be displayed separately from the text (if referred to explicitly in the text).

Footnotes

Footnotes should be used sparingly. Number them consecutively throughout the article, using superscript Arabic numbers. Many wordprocessors build footnotes into the text, and this feature may be used. Should this not be the case, indicate the position of footnotes in the text and present the footnotes themselves separately at the end of the article. Do not include footnotes in the Reference list.

Table footnotes

Indicate each footnote in a table with a superscript lowercase letter.

Artwork

Electronic artwork

General points

• Make sure you use uniform lettering and sizing of your original artwork.

• Save text in illustrations as ‘graphics’ or enclose the font.

• Only use the following fonts in your illustrations: Arial, Courier, Times, Symbol.

• Number the illustrations according to their sequence in the text.

• Use a logical naming convention for your artwork files.

• Provide captions to illustrations separately.

• Produce images near to the desired size of the printed version.

• Submit each figure as a separate file.

• Pictures, graphics and images must be submitted in a JPG or GIF format with 300 dpi.

2 Conducting the Review

2.1 Originality

You might wish to do a quick literature search using tools such as Scopus to see if there are any reviews of the area. If the research has been covered previously, pass on references of those works to the editor.
INSTRUCTIONS FOR AUTHORS

2.2 Structure
Consider each element in turn: Title; Abstract; Introduction (It should describe the experiment, the hypothesis(es) and the general experimental design or method); Method; Results; Conclusion/Discussion; Language: you do not need to correct the English. You should bring this to the attention of the editor, however.

2.3 Previous Research
If the article builds upon previous research does it reference that work appropriately? Are there any important works that have been omitted? Are the references accurate?

2.4 Ethical Issues
Plagiarism: If you suspect that an article is a substantial copy of another work, please let the editor know, citing the previous work in as much detail as possible
Fraud: It is very difficult to detect the determined fraudster, but if you suspect the results in an article to be untrue, discuss it with the editor

AUTHORIZATION FOR PUBLICATION OF PAPERS
LICENSE FOR USE OF INTELLECTUAL WORK (Author)

For this private instrument the AUTHOR, below signed authorizes the IAHR Latin American Working Group, to publish its work authorship, without any obligation and in exclusiveness character for the period of six months starting from the publication in the AMERICAN JOURNAL OF HYDROPOWER, WATER AND ENVIRONMENT SYSTEMS, or in another official publication of IAHR.

In case of joint authorship, the first author signs as AUTHOR, assuming before IAHR the commitment of informing the other authors of the granted license.

AUTHOR (full name in form letter):

Title of the Paper:

JOINT AUTHORS [full name in form letter]:

ADDRESS:

________________________________________.

Email:
American Journal of Hydropower, Water and Environment Systems

Number 3    JULY 2016

Technical Papers

6 DAM BURST IN THE SAPUCAÍ RIVER (MG): A HEC-RAS SOFTWARE SIMULATION
Barbosa, Alexandre Augusto; Risotto, Lucas Stéfano; Franco, Patrícia Mara; Silva, Laércio Rafael
Colucci Marques da

11 SIMPLE MODELLING FOR MAXIMUM FLOW RATES DETERMINATION TO
BE APPLIED IN ECONOMICALLY FEASIBLE SMALL HYDROPOWER PLANTS
Santos, Ivan Felipe Silva dos; Vieira, Nathalia Duarte Brac; Tiago Filho, Geraldo Lucio; Barros,
Regina Mambeli; Souza, André Luiz

14 ALE FINITE ELEMENT METHOD FOR FLUID FLOW SIMULATIONS OVER
MOVING GUIDE VANES
Noletto, Luciano Gonçalves; Barcelos Junior, Manuel Nascimento Dias; Brasil Junior, Antonio C. P.

19 ON THE HYDRODYNAMICS OF A ROW ARRANGEMENT OF HYDROKINETIC
PROPELLER TURBINES
Brasil Junior, Antonio C.P.; Mendes, Rafael C. F.; Oliveira, Taygoara F.; Andriampanaray, Tamby;
Koudri, Smaine; Mesquita, André L.A.

25 NUMERICAL SIMULATIONS OF COMPLEX FLOWS IN HYDRAULIC TURBOMACHINERY.
ACTUAL R&D LINES IN THE ‘CFD/APPLIED MATHS LABORATORY’ GROUP
Coussirat M.; Fontanals A.; Panela L.; Henderson G.; Aguirre R.

35 AN ANALYSIS OF THE MECHANICAL STRESS IN HYDROELECTRIC TURBINES STAY-
VANES UNDER DIFFERENT OPERATIONAL STATES OF THE ELECTRIC GENERATOR
Gonzalez, Facundo E.; Kelm, Diego A.; Kolodziej, Javier E.; Tarnowski, Gabriel A.; Astelli,
Raúl; Bordón, Hugo

39 FLOW PROFILE INVESTIGATION AT TUCURUI HYDRO POWER PLANT TAILRACE
Bortoni, Edson C.; Bertrand, Olivier; Sauvaget, Patrick; Santos, Luciano T.; Vasconcellos, Ricardo C.

Technical Notes

44 ANALYSIS OF STATOR WINDINGS INSULATION OF COARACY NUNES HYDROELEC-
TRIC PLANT GENERATING UNITSE
P. R. M. Vilhena; F. S. Brasil; V. Dmitriev; C. J. S. Santos
ABSTRACT

This work aims to present a simulation of a dam break in the Sapucai River. This barrier will be located on the
neighborhood of 'Bairro dos Freires' in the city of Itajubá, Brazil. Although, the dam has not been constructed yet it was
totally designed and specified. For this study, we used the HEC-RAS software developed by the US Army Corps of Engineers.
This software matches the needs for the conduction of this investigation involving non-permanent flows. The results are
satisfactory and they show that there may be a risk of dam break whether the project is not executed with extensive
technical knowledge. Consequently, this means a potential threat to the valleys downstream, which can negatively affect the
local economy, the environment and the society, including human lives.

KEYWORDS: sustainability, micro hydro power, turbomachinery, pumps, PAT

1. INTRODUCTION

Dams are structures built crosswise to the river flow direction. The purpose of building a dam can be for hydroelectric energy
generation, for population water supply, for flood control or navigation. In order to reach these ends, dams raise the water
level upstream of its axis and, in some cases, they store a significant amount of water used to assure the regulation of the
affected waterbody (UEMURA; MARTINS; FADIGA JR, 2007). The construction of a barrier in a river, even when it is designed with
huge technical knowledge, it always involves the risk of incident, such as a structural break, that may hazard the valleys
downstream. The occurrence of breaking generates an abrupt waviness movement that propagates at high speeds and
considerable amplitude downstream the dams and, as consequence, a heavy flood of the secondary gutters, mainly the
ones closer to the barrier.

According to Castro (1999), a dam break weakens the ecosystem and also causes human, material and
environmental damage that imply in economic and social losses. In addition, this critical scenario may worse due to the
negative impact caused in essential services, such as energy supply, sanitation and health service.

As stated by Singh (1996), the records show that only on the XX century, 200 severe accidents have occurred in dams
higher than 15 meters around the world. These accidents have killed more than 8,000 people and left thousands homeless.

A dam break can have different causes. The evaluation of its potential impacts is a preventive process and is part of the
management tasks of Civil Defense. (TUCCI, 1997).

Some cases of dam burst stand out in Brazil. For instance, the break of a barrier in the Verde River in the state of Minas
Gerais in 2001. Another case happened in 2004 in Camará dam. According to Valencio (2006), in the latter, the
consequent waviness have caused 6 deaths and have left 3,000 homeless as well destroyed hundreds of houses in Mulungu and Alagoa Grande.

The past events and facts, such as those cited, evidence the concernment of studies on dam bursts and the formulation of
emergency action plans. Both are essential to minimize human and material damage when a dam accident happens.

The simulation of dam disruption and its consequent floods is decisive in the characterization and reduction of the threat caused
by failures in barriers (XIONG, 2011). The aims of this analysis are the evaluation of the impacts to the environment and dam safety,
and the implementation of operating emergency procedures such as an alert system and evacuation plans downstream the dam (DA
SILVA, 2011). According to Lnec (2003), a numerical model is the most appropriated tool to fast and systematic analysis of the
effects from a downstream valley’s flood. For this investigation, the used tool is the HEC-RAS software developed by the US Army
Corps of Engineers.

This report presents the study about a dam disruption in the Sapucai River in the city of Itajubá/MG. Moreover, an analysis
about the consequent flood displacement was simulated on HEC-RAS software and it is detailed on this article.

2. METODOLOGY

This work was conducted in the city of Itajubá, in the
south of Minas Gerais state. The barrage will be built on the
neighborhood of 'Bairro dos Freires' located between the cities of
Itajubá and Wenceslau Brás. The figure 1 illustrates the
geographic location of Itajubá.

[Figure 1: Location of the city of Itajubá.]
Source: Itajubá city hall (2010)

The system of dams will be built to reduce the peak of flood
hydrographs and, therefore, it will restrain part of the superficial
outflow arising from the Sapucaí River’s springs, upstream to Itajubá. The figure 2 shows the Sapucaí River basin and it also illustrates the dam location and the topobatimetric section of study.

This system will belong to a set of contention basins which is part of a flood prevention that often affects some towns as Pouso Alegre, Santa Rita do Sapucaí and Itajubá.

The dam will be a gravity dam made of a massif (solid) concrete on the Sapucaí riverbed, specifically, in the coordinates: 458440.082; 7508471.711. Table 1 presents the main features of this barrier.

**[Table 1]: Dam features.**

<table>
<thead>
<tr>
<th>Dam features</th>
<th>Cities of Itajubá and Piranguçu,</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Location</strong></td>
<td>in the south of the Minas Gerais</td>
</tr>
<tr>
<td><strong>River</strong></td>
<td>state, about 4 km before the</td>
</tr>
<tr>
<td><strong>Type of Dam</strong></td>
<td>bridge on BR 459 highway that</td>
</tr>
<tr>
<td><strong>Maximum Height</strong></td>
<td>links Itajubá to Wenceslau Brás.</td>
</tr>
<tr>
<td><strong>Crest length</strong></td>
<td>260 meters</td>
</tr>
<tr>
<td><strong>Drainage Area</strong></td>
<td>378 km²</td>
</tr>
<tr>
<td><strong>Accumulated volume at maximum level</strong></td>
<td>33 hm³</td>
</tr>
<tr>
<td><strong>Flooded area at maximum level</strong></td>
<td>192 ha</td>
</tr>
</tbody>
</table>


In order to make the hydrograph calculations and a proper representation of the channel’s geometric features, some measurements were taken in each monitoring section. For this, the measuring techniques used are edges topography and bathymetry, as they allow a fully covering of the riverbed and enough shifting to represent the notable points of a trace.

On Table 2 there are the main features of the transversal sections downstream the dam in relation to ebb levels.

**[Table 2]: Downstream sections hydrological details.**

<table>
<thead>
<tr>
<th>Section</th>
<th>Wet area [m²]</th>
<th>Wet perimeter [m]</th>
<th>Hydraulic radius [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Água Limpa</td>
<td>6,8</td>
<td>7,4</td>
<td>0,9</td>
</tr>
<tr>
<td>Canta Galo</td>
<td>25,8</td>
<td>20,6</td>
<td>1,3</td>
</tr>
<tr>
<td>Captação Copasa</td>
<td>23,0</td>
<td>19,0</td>
<td>1,2</td>
</tr>
<tr>
<td>Santana</td>
<td>4,9</td>
<td>7,4</td>
<td>0,7</td>
</tr>
</tbody>
</table>

A DGPS equipment obtained the topographic data, in this case, the DGPS is the model 900 CS developed by LEICA Geosystems. This instrument uses L1 and L2 satellite signals and it is installed at the Federal University of Itajubá.

The bathymetry of each section was performed with an ultrasonic meter, the OTT Oliner, which measures speed and flow rate of each topobathymetric section. This instrument gives accurate and detailed flow rate information, which implies in reliable and fast results. The meter moves crosswise to the section and gets, simultaneously, the profile of instantaneous speed and the section depth. The volumetric flow rate is calculated by the integration of the speeds and the bathymetry of the section. Finally, the results are presented in real time just after the end of these measurements.

In addition to the cited instruments, a simulation tool that is a software developed by the US Army Corps of Engineers was used to complete this investigation. The HEC-RAS software is available for free and it allows unidimensional calculations of outflow in steady and transient states. Furthermore, this tool allows quantitative and qualitative analysis of different water sources as watercourses or reservoirs. The HEC-RAS algorithm for the calculations is based on Saint-Venant equations for unidimensional outflow, which according to Porto (2006) are:

Continuity equation:

\[ A \frac{\partial V}{\partial x} + V \frac{\partial A}{\partial x} + B \frac{\partial y}{\partial t} = 0 \]

where:
- A = Cross sectional area of the outflow
- V = Cross sectional average speed
- B = Largura de topo da seção
- y = Outflow depth
- t = Time

Equation of motion:

\[ V \frac{\partial V}{\partial x} + \frac{\partial V}{\partial t} + g \frac{\partial y}{\partial x} = g(I_0 - I_f) \]

where:
- V = Cross sectional average speed
- y = Outflow depth
- g = Gravity acceleration
- t = Time
- I_0 = Channel bottom steepness
- I_f = Energy line steepness

A simulation on HEC-RAS software requires, beforehand, a project that gathers all necessary information for the numeric simulation as well as the obtained results. In aid to the study, on the AutoCAD CIVIL 3D 2011 it was set topography information as drainage network system and the location of bathymetric sections (stations) where the outflow calculation will be done.
The rivers, paths and stations of study are listed on Table 3 and illustrates on Figure 4.

[Figure 4: Fluvial path for calculations.]  
Source: HEC-RAS

[Table 3]: Description of studied area.

<table>
<thead>
<tr>
<th>Rivers</th>
<th>Path name</th>
<th>length [km]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sapucaí River</td>
<td>T1</td>
<td>5,6</td>
</tr>
<tr>
<td></td>
<td>T2</td>
<td>4,7</td>
</tr>
<tr>
<td></td>
<td>T3</td>
<td>10,2</td>
</tr>
<tr>
<td>Bicas River</td>
<td>T1</td>
<td>1,7</td>
</tr>
<tr>
<td>Ribeirão Santo Antônio</td>
<td>T1</td>
<td>3,4</td>
</tr>
</tbody>
</table>

The table 4 shows the topobathymetric sections used for the outflow analysis. The RS sections refer to those from the Sapucaí River; RB to the Bicas River sections and RSA refers to Ribeirão Santo Antônio parts.

[Table 4]: Topobathymetric sections explored.

<table>
<thead>
<tr>
<th>Section</th>
<th>Name</th>
<th>Channel Depth [m]</th>
<th>Distance to the dam [km]</th>
</tr>
</thead>
<tbody>
<tr>
<td>RS01</td>
<td>Barramento</td>
<td>857,3</td>
<td>0</td>
</tr>
<tr>
<td>RS02</td>
<td>Confluência Bicas</td>
<td>845,4</td>
<td>3,5</td>
</tr>
<tr>
<td>RS03</td>
<td>Confluência Bicas II</td>
<td>845,4</td>
<td>3,6</td>
</tr>
<tr>
<td>RS04</td>
<td>Confluência Santo Antônio</td>
<td>840,3</td>
<td>7,5</td>
</tr>
<tr>
<td>RS05</td>
<td>Cantagalo</td>
<td>840,3</td>
<td>16,0</td>
</tr>
<tr>
<td>RS06</td>
<td>Confluência José Pereira</td>
<td>829,6</td>
<td>22,0</td>
</tr>
<tr>
<td>RB01</td>
<td>Confluência Bicas</td>
<td>845,4</td>
<td>4,0</td>
</tr>
<tr>
<td>RB02</td>
<td>Santana</td>
<td>851,6</td>
<td>5,5</td>
</tr>
<tr>
<td>RSA01</td>
<td>Confluência Santo Antônio</td>
<td>840,3</td>
<td>8,5</td>
</tr>
<tr>
<td>RSA02</td>
<td>Água Limpa</td>
<td>847,1</td>
<td>12,5</td>
</tr>
</tbody>
</table>

The Manning coefficients (n) were estimated through tests made in fieldwork. It was avoided to use an unique value for the coefficient (n) once that the simulations aims to cover the variety of flow resistance from any gutters to any channel. According to Chow (1959), this coefficient ranges from 0.045 to 0.009 and, thus, is possible to represent the possible flow resistance in most of natural known channels.

The coefficients, considered constant to any water level, were varied for the path 1 (T1) and, thereby, 5 numerical simulations were performed.

The first path (T1) was simulated with a Manning value of 0.20 for both secondary gutters: right and left. The main gutter had a variation of Manning coefficient value, and they were: 0.045, 0.060, 0.070, 0.085 and 0.090.

For the paths 2 and 3, constant values were set: 0.12 for the secondary gutters while 0.045 for main gutter.

Based on hydrograph studies from the Laboratory of Hydro Information (LIH) of the Federal University of Itajubá, a hydrograph was drawn to represent the dam burst as can be seen on Figure 5. By producing a hydrograph of the most upstream point, the result will fed the HEC-RAS with information about the flood waves that will propagate. A steepness value of 0.0007 [m/m] was set for the path most downstream to the outflow.

[Figure 5: Hydrograph of a hypothetic dam break.]

This hydrograph production required the following data:
- Generation time: around 30 minutes;
- Initial quota: 891 m;
- Final quota: 858 m;
- Depth: 260 m.

The HEC-RAS software also needs information about the relation quota x volume of the reservoir. This is illustrates in Figure 6.

[Figure 6: Quota x reservoir’s volume chart.]  
Source: Magna Engenharia’s report (2001)

3. RESULTS

From figure 7 to figure 11 the results of the HEC-RAS (version 4.1) simulations are presented. The simulations where executed under 5 sets of geometric data in which each set has a different Manning coefficient (n). The changes of the coefficient is because the proper value for each topobathymetric section and paths was unknown. Based on observation made at the fieldwork, it was decided that the coefficient would range from 0.045 to 0.09 and, therefore, the simulated values were 0.045, 0.06, 0.07, 0.085 and 0.09. Following it is possible to see the results represented by charts.
The section RS01, located on the dam break point, equally behaves in all the 5 simulations as can be seen (blue line) on the hydrographs above. This means this section is not affected by Manning coefficient changes and hence the hydrograph is input at HEC-RAS as boundary conditions.

For the section RS02, 3.5 km from the barrier, the hydrographs show (in red lines) a damped flood wave, which has a maximum outflow rate approximately, equals to half of the maximum outflow at section RS01. In RS02, it was noticed some inconstancy for the simulated Manning values of 0.06, 0.07 and 0.085. The latter was the most unstable value and this can be explained by the closeness to the disruption point and the high steepness level of the path.

The section RS03, that is 7.5 km away of the dam is represented by a green line on the hydrographs. It is possible to observe a maximum flow rate value lower than in the other sections: RS01 and RS02. The softer behaviour is due to the existence of two rivers between the sections where the outcome water of the disruption has propagated. This fact has created negative flow rate in those waterways (in the upstream to downstream direction).

The steadiest values are seen (violet lines) on section RS06, located 22km from the dam burst area. This stability of these values is influence of the distance to the broken barrier and the damping of the previous sections.

In despite of the steadiest values are observed on section RS06, it is possible to state that for this section (22km away of the dam), the maximum flow rate will be greater than 1,000 m³/s and this is enough to overcome the secondary gutters and reach levels up to 3.5 meters (PINHEIRO, 2005). If this flow rate value is reached after a dam burst, the city of Itajubá may have a large part of its urban area flooded. By comparing the hydrographs, it was noticed that a smooth difference occurs as consequence of the variation of the Manning coefficient (n) in the simulations. The following figures (12, 13 and 14), respectively, evidence the simulation of RS02, RS05 and RS06 sections.

To enhance this analysis, a study about sensitivity was conducted in order to find the relation between Manning coefficients variation and changing of top flow rate and the respective river level.

For this analysis, the sections RS02, RS05 and RS06 have the calculus of sensitivity related to flow rate done. The following chart explores these relations.

---

By looking for the results, it is possible to affirm that the changes of the Manning coefficients implies in changes of the calculated flow rate however they are tiny changes. For instance, when increasing the coefficient in 40%, it results in a decrease of 4% in the maximum flow rate.

The OTT Oliner used for bathymetric tests gives accurate and detailed flow rate information, which implies in reliable and fast results. The meter moves crosswise to the section and gets, simultaneously, an instantaneous speed profile and the section depth. The volumetric flow rate is calculated by the integration of the speed profile and the bathymetry of the section. Finally, the results are presented in real time just after the end of these measurements.

The relation between Manning coefficient and the river level could be explored only for sections RS01 that is the area of the dam break. This limitation is because that only on RS01 the flow rate has not changed in different simulations. In other words, the hydrograph is the same for all the tests. Furthermore, the results show a tiny increase of the flow rate due to the changes in Manning coefficient for the maximum quota. Likely in the previous sensibility analysis, the variation of the values were around 0.2%.

4. CONCLUSION

Based on the results of this studies and the references it is possible to state that a dam burst may cause many damage as economic and social losses. A barrier break cares attention once that it can negatively affect essential services to the community such as energy supply, sanitation and health service.

Thereby, it is essential a technical control of the structures and the creation of a preventive plan to minimize the impacts of a dam break. This plan may include a compulsory study of dam bursts and the creation of a list of emergency acts that includes alert systems and mapping of affected areas by the barriers failure. Finally, the plan also includes detailed information about the establishment of protection actions and about people evacuation downstream the barrier.

This work reached relevant results that allowed the perception about the risk of a dam break and it also contributes to the effectiveness of mitigating measures and response to possible crisis situations. Moreover the HEC-RAS software, proved to be a reliable tool to simulate dam break cases. With the aid of this software, it was possible to notice that the use of a great number of topobathymetric sections increases the tests reliability and decreases instability problems. As consequence, this also helps in flood mapping production once that the better are the collected topobathymetric data, the more reliable the hydrographs are.

In summary, the most stable results were obtained for the 0.045 and 0.09 Manning coefficient values. By testing greater values for the coefficient while simulating sections close to the dam, the results are much stable and reliable. From the sensitivity analysis, it was proved the effects on the model due to the variation of Manning coefficients, even there were tiny changes in flow rate and water level.

Finally, to obtain more precise results, it would be necessary to collect more number of topobathymetric sections in order to decrease the interpolation of sections. Furthermore, a deep evaluation about Manning coefficients would be necessary to minimize the errors during the simulations.

5. REFERENCES

Simple modelling for maximum flow rates determination to be applied in economically feasible small hydropower plants

Santos, Ivan Felipe Silva dos; Vieira, Nathalia Duarte Braz; Filho, Geraldo Lúcio; Barros, Regina Mambeli; Souza, André Luiz

ABSTRACT

The implantation of hydroelectric plants, especially the small ones (SHP) supports the reduction of greenhouse gases and the expansion of renewable generation and can also be used for the development of distributed generation in a country. However, the development of such plants is subject to economic feasibility. In this context, this paper develops equations that relate the fall of a hydroelectric potential site with the maximum flow rate for a case in which implementation of a SHP is financially feasible. Through these equations, the flow to be adopted in one plant can be estimated basing on fall data, which helps to carry out prospecting and preliminary hydro potential research of economic feasibility. Correlation equations were determined by analytical deductions using the cost-benefit equation of a SHP. Variations of these in relation to different sales rate values of energy and capacity factor, were also discussed.

KEYWORDS: SHP, economic viability, peak flows.

1. INTRODUCTION

The growth in electricity generation from renewable sources is the key to the urgent need in reducing the greenhouse gases emissions released in to the atmosphere. In this context, the hydraulic power produced from small hydro powerplants (SHPs), a renewable and clean energy source, becomes a very important option [1].

The framing of a Hydroelectric Power Plant as SPH varies from country to country. For example, in Germany, SHP are those plants with P <1 [MW]. In Canada, the SHP must have a P <50 [MW] [2]. In Brazil, due to Law 13.097/2015, the SHP are framed in 3 <P <50 [MW] [3].

It is remarkable the significance of hydropower to Brazilian energy scenario. Due to the drought of recent months, the percentage hydropower in Brazilian energy matrix has been reduced in relation to prior years, but still quite high, being close to 71% in 2014 [4]. Also in 2014, the SHP accounted for 3.58% of all installed capacity in the country [5]. The Figure 1 below also shows the evolution of installed capacity of SHP in Brazil between 2001 and 2014 [5, 6 and 7].

![Figure 1: Evolution of the number of SHP and installed capacity of this [5] source.]

The SHPs stand out with respect to large hydro, because they implies in lower social environmental impacts due to lower flooded area and no need for regularization (low land expropriation and impacts on wildlife). In addition, the SHPs may be developed as the basis for a plan that prizes for distributed energy generation held close to the centers of consumption and lower losses on the transmission line.

In order to foster the development of SHP and assist the economic feasibility analysis of this type of use, this article aims to develop the equations that relates the head of a hydroelectric site potential with the implanted flow and the resulting economic viability the SHP.

2. MODELLING

According to [8] cited in [9] the amount of the annual net financial benefit generated by the SHP can be calculated by Equation 1.

\[
ANB = E \cdot T - I \cdot FRC - Com
\] (1)

Where: ANB = annual net benefit (R$); T = energy sales rate [R$/MWh]; E = Energy produced annually; I = Investment [R$]; FRC = Capital Recovery Factor and Com = operation and maintenance costs.

The cost of operation and maintenance costs of SHP can be adopted as 5% of the investment [10]. Then, rewriting the equation 1, replacing and writing with the energy E as a function of power and the capacity factor, we obtain the equation 2.

\[
ANB = P \cdot fc \cdot 8760 \cdot T - I(FRC + 0.05)
\] (2)

Where: P = power [MW]; fc = Capacity factor; and 8760 = number of annual hours.

The estimation of investment was performed using the method based on the aspect factor (AF), proposed by Tiago Filho et al. (2011) [11].
This model correlates unit costs ($C_{un}$) with the aspect factor (AF) of SHP. Factor derived from the equation used to determine the specific rotation of the selected turbine for a given hydropower plant (Equation 3):

$$n_{qa} = 1000. n \frac{Q^{0.5}}{(g . H)^{0.75}}$$  \hspace{1cm} (3)

Where: $n$ = hydraulic turbine rotation [rpm]; $Q$ = water discharge [m³/s]; $g$ = gravity acceleration = 9.81 [m²/s]; and $H$ = gross head in which the turbine is subjected [m] and $n_{qa}$ = specific rotation defined as the spin that the machine would operate with a head of one meter with a 1[m³/s] flow.

Dividing $n_{qa}$ by $n$ in equation 3, and substituting the relation between power and flow (Equation 4), it is possible to obtain a function of power and head which is denominated as aspect factor (AF) (Equation 5):

$$Q = \left( \frac{P \cdot 1000}{g H} \right)$$  \hspace{1cm} (4)

$$\frac{n_{qa}}{n} = 1821.43 \frac{P^{0.5}}{H^{1.25}} = AF$$  \hspace{1cm} (5)

Where: $P$ = Power [MW]; $\gamma$ = specific weight of water = 9800 [N/m³] and AF = aspect factor [rpm-1] and:

$$C_{un} \left[ \frac{\text{US$\$/kW}}{\text{kW}} \right] = k \cdot (AF)^n$$  \hspace{1cm} (6)

Where $k$ and $n$ are constants that vary according to the market characteristics of the study area.

It is observed from equation 5 that, besides the aspect factor (AF) relating to the rotation and specific rotation of the machines, which are chosen according to the SHP physical aspect, it can also be written as a function which varies with $H$ - $P$ values chosen for hydropower plant to the deployment in a potential site, factors that are also closely related to the physical format of the SHP, and are dominant in determining the costs of civil works and turbines to choose from.

Thus it appears that the AF factor can be related to the total costs of hydroelectric projects. Tiago Filho et al. (2011) [11] also used Brazilian design data to determine the AF relation with SHP costs in this region, presented in equation 7.

$$\frac{\text{US$/kW}}{\text{kW}} = 1654 \cdot AF^{0.085}$$  \hspace{1cm} (7)

Substituting (5) in (7) and considering a conversion rate between US Dollar equal to 2.5 [R$], equation 8 could be obtained.

$$I[10^6 \text{R$}] = 7.82 \cdot \frac{P[\text{MW}]^{0.425}}{H^{1.0625}}$$  \hspace{1cm} (8)

The economic viability of SHP is obtained from net annual benefits greater than 0 (profitable value). Thus the match- $ANB = 0$ in equation (2), to substitute I for the result of equation (8), $P$ by $Q$ using the equation (4), one can relate the maximum viability flow ($Q_{MV}$ - Maximum flow to $ANB = 0$) with the head $H$ to obtain the equation 9, which can be summarized in accordance with equation 10.

$$Q_{MV} = \frac{f c. 8760 . T}{7.63 \cdot 10^6 . (F R C + 0.05)} . H^{3/2}$$  \hspace{1cm} (9)

$$Q_{MV} = \alpha . H^\beta$$  \hspace{1cm} (10)

Through equation 10 is observed that the maximum flow viability is related to the head through an exponential curve defined by two factors: $\alpha$ and $\beta$. For the above equation it can be seen that these constants are influenced by the market conditions, specific energy terms for each region (like the average $fc$), local conditions of the chosen market and the estimated cost. Therefore, the proposed equations can be replicated to other countries, or even to Brazil in different market conditions.

3. APPLICATION

For the purposes of this study it was considered an interest rate equal to 10%, a capacity factor of 60% and a lifespan of 50 years. The $QMV \times H$ ratio curves were constructed using three values of representative rates of the Brazilian scenario, denominated 125 (pessimistic value), 150 (Moderate) and 175 (Pessimistic) [R$/ \text{MWh}]. These are shown in Figure 2 (on a logarithmic scale). On the other hand, figure 3 shows the relation of the parameter $\alpha$ with rate variations. In this figure, there is a nonlinear relation between $T$ and $\alpha$ (which can also be seen in equations 9 and 10).

Analyzing figure 2 it can be seen that higher energy sales tariffs allow an ensured viability of SHP for higher flow rates, keeping the same head. This fact enables the profitability of a configuration that provides more energy per head unit. In contrast, Figure 3 shows that the correlation factor $\alpha$ also increases exponentially with the tariff increases. This fact shows the importance of setting a fair price $T$ for the development of SHP.

Still observing Figure 2 it is possible to analyze how the estimate proposed in this work enables a simplified analysis of the viability of a potential site with a known head. Assuming a head of 10 [m], flow rates up to 10 [m³/s] can be used, if $T = 175$ [R$/\text{MWh}]$, so that the viability of the project is analyzed.
For any flow higher than 10 [m³/s], the enterprise will not be attractive. Such fact shows that it is not always interesting, economically, the implementation of higher available flows.

4. CONCLUSIONS

In order to assist energy potential studies and economic feasibility of SHP, this paper develops an equation for a simplified analysis of the economic feasibility of a potential site with a known head. The developed equation allows the correlation between the head of a hydroelectric potential site with the maximum flow rate to be deployed that would allow the viability of the SHP.

The developed equations are a function of two parameters: α and β. These parameters in turn are a function of parameters linked to the macroeconomic scenario in each region (discount rate values i and energy sale tariff), the typical Capacity Factor of such plants in the area (fc) and the estimate adopted cost (which also vary regionally).

Therefore, it can determine the parameters α and β and compare them in order to define the best regions for the development of SHPs. The best regions will be those that provide greater QMV for a given head. This comparison remains as a suggestion for future work.

5. ACKNOWLEDGEMENTS

The authors thank the Higher Education Personnel Improvement Coordination (CAPES) for granting master’s scholarships.

6. REFERENCES


ABSTRACT

The present paper presents numerical results of the turbulent flow over a moving NACA 0018 hydrofoil. This hydrofoil is used as a moving vane of the wicket gate of a hydraulic turbine. In hydraulic turbines, this movement is related to the flow loading changes, where the gate is opening or closing. Recurring phenomena linked to the flow over moving vanes inside hydraulic turbines are transition to turbulence, boundary layer separation and dynamic stall. The hydrofoil is submitted into upstroke and downstroke movement to observe the dynamic stall. An explicit projection finite element algorithm is employed with an Arbitrary Eulerian Lagrangean (ALE) scheme to conduct the numerical simulation. The ALE method has the ability to conduct mesh movement and fluid flow independently. Turbulence modeling based in URANS (Unsteady Reynolds Averaged Navier Stokes), given by the Shear Stress Transport (SST) and the hybrid Detached Eddy Simulation Shear Stress Transport (DES SST) modeling are implemented. The obtained results are consistent with the measured reality displayed at the literature.

KEYWORDS: arbitrary lagrangean eulerian, finite element method, guide Vanes.

1. INTRODUCTION

The flow over moving boundaries holds great engineering interest. These flows are encountered in several areas, such as civil, mechanical, aerospace and biomechanical engineering. One can found at the literature several examples of practical applications which use the combination of finite element techniques and Arbitrary Eulerian Lagrangean (ALE) methodology [1, 2]. The flow inside a spiral casing and within the wicket gate is one of these cases. It holds great complexity due to the inner nature of the turbulent flow and the appearance of dynamical stresses over its mechanical parts [5, 6]. Those stresses are, in a great part, originated by a complex 3D turbulent flow topology that interacts with the mechanical parts of the turbine, in particular with the wicket gate, and with the runner blades [8].

Some of its structural stresses are taken into account for the project and design of the gate. This is an area known as fluid-structure interaction, where knowledge on fluid mechanics, solid mechanics and vibrations are needed. A great number of problems in turbomachinery have hydrodynamical origins, such as pressure oscillations and cavitation [7]. Eects of fluid-structure interaction and the stresses originated by this interaction induces cumulative damage and mechanical failures at those parts due to cyclical stresses.

Those pathologies can force an approximately 10-day halt at the turbine operation for maintenance. To calculate the flow over moving vanes, one can employ mesh motion techniques. The rationale involved at the mesh moving strategies has an early approach based on a pseudo-structure dynamics. A more complete mesh motion approach uses the superposition of linear and torsional springs [15]. From the beginning, the linear-torsional spring method was developed for two-dimensional triangular elements. The technique adaptation to three-dimensional elements such as tetrahedrons required a position modification of the torsional spring action plane.

The present work show numerical results of the turbulent flow over a moving hydrofoil using finite element and ALE methods. For the moving mesh, a pseudo-structural approach is employed, based on linear-torsional springs. The obtained results will be compared with the work of [14]. Therefore, the following article is divided as follows: The next section will state the flow problem with the moving boundary, starting with the flow equations to its final discretized weak form and the moving mesh problem. The following sections will describe the results, analysis and conclusions.

2. THEORY

2.1. Problem Statement

In the Arbitrary Lagrangean Eulerian (ALE) methodology, the domain movement is carried by a mapping process, defined into the continuum mechanics theory (Figure 1).

![Figure 1: ALE rationale.]

Considering now the ALE mapping, for any function \( f: \Omega_0 \mapsto \mathbb{R} \) the material derivative can be written as showed by Equation 1:

\[
\frac{Df}{Dt} = \frac{\partial f}{\partial t} + \mathbf{u} \cdot \nabla f
\]

(1)

The relative velocity \( \mathbf{u} \) is denoted by the dierence between flow and mesh velocities. The continuity, momentum and SST transport equations are reformulated in a ALE description, given as follows (Equations 2 to 5):

Find the velocity, pressure and turbulent fields \( \mathbf{u}(x,t), p(x,t), k(x,t), \omega(x,t) \) defined in \( \Omega_t \{0; T\} \), such that:

\[
\nabla \cdot \mathbf{u} = 0
\]

(2)
\[
\frac{\partial \mathbf{u}}{\partial t} + \hat{\mathbf{u}} \cdot \nabla \mathbf{u} = \frac{1}{\rho} \nabla p + (\nu + \nu_t) \nabla^2 \mathbf{u} + \mathbf{f}
\]  
(3)

\[
\frac{\partial k}{\partial t} + \hat{\mathbf{u}} \cdot \nabla k = P_k - \beta \kappa \omega + \left( \frac{\nu_t}{\sigma_k} \right) \nabla^2 k
\]  
(4)

\[
\frac{\partial \omega}{\partial t} + \hat{\mathbf{u}} \cdot \nabla \omega = \alpha S^2 + \beta \omega + \left( \frac{\nu_t}{\sigma_\omega} \right) \nabla^2 \omega + 2(1 - F_1) \alpha \omega + \frac{1}{\sigma_\omega} (\nabla k) \cdot (\nabla \omega)
\]  
(5)

Where:
- \( k \) and \( \omega \) are the turbulent kinetic energy and turbulent frequency respectively.
- \( P_k \) is a production limiter, used to avoid the excessive generation of turbulence in stagnation points.
- \( \alpha \) and \( \beta \) are constants.

The eddy viscosity is defined by Equation 6:
\[
\nu_t = \frac{\alpha_t k}{k} \left( \frac{\max(\kappa, S \beta)}{F_{DES}} \right)
\]  
(6)

The DES SST formulation uses the SST turbulence model formulation with a modification of the kequation destruction term, showed by Equation 7 [24]:
\[
k \beta \omega = k \beta_0 F_{DES}
\]  
(7)

Where the DES blending function is given by Equation 8:
\[
F_{DES} = \max \left( \frac{\kappa}{C_{DES} \Delta \beta_0}, 1 \right)
\]  
(8)

For moving boundary problems of small and medium scale, the solution strategy involves direct methods for linear system resolution, due to its simplicity and robustness. Mesh position \( x_n+1 \) is updated by the sum of the mesh displacement vector to the previous configuration \( x_n \), as given by Equation 9 [42]:
\[
\Delta x = \mathbf{u} + \mathbf{x} \quad \text{at} \quad \Gamma_d
\]  
(9)

\[
\Delta p = \mathbf{p} \quad \text{at} \quad \Gamma_o
\]  
(10)

\[
\mathbf{u}(x, t) = \mathbf{u}_d \quad \text{on} \quad \Gamma_d
\]  
(11)

\[
\mathbf{u}(x, t) = \mathbf{w}_i \quad \text{on} \quad \Gamma_{in}
\]  
(12)

\[
\mathbf{p}(x, t) = P_{ref} \quad \text{on} \quad \Gamma_o
\]  
(13)

\[
k(x, t) = k_d \quad \text{on} \quad \Gamma_d
\]  
(14)

\[
\omega(x, t) = \omega_d \quad \text{on} \quad \Gamma_d
\]  
(15)

Where the displacement boundary condition is prescribed as given by Equation 17:
\[
x = x_n \quad \text{on} \quad \Gamma_{in}
\]  
(17)

Here \( \mathcal{L}(\cdot) \) is the mathematical operator responsible for domain mapping. One can state that the mesh nodes move independently of the flow in an ALE framework. Hence, it is a suitable method for analyzing oscillating rigid body eects into any given flow.

The stiffness matrix and the reaction force vector can be divided in parts related to the internal (\( \Omega \)) and external (\( \Gamma \)) mesh degrees of freedom, as given by Equation 18:
\[
\begin{pmatrix}
K_{\Omega \Omega} & K_{\Omega \Gamma} \\
K_{\Gamma \Omega} & K_{\Gamma \Gamma}
\end{pmatrix}
\begin{pmatrix}
\mathbf{x}_\Omega \\
\mathbf{x}_\Gamma
\end{pmatrix} = \begin{pmatrix}
\mathbf{r}_\Omega \\
\mathbf{r}_\Gamma
\end{pmatrix} = \mathcal{L}(\mathbf{x})
\]  
(18)

Equation 18 is solved with a null force condition at the internal mesh nodes (\( \mathbf{r}_\Omega = 0 \)). Besides, a quasi-static model is used for this problem. The stiffness matrix is built using the improved spring analogy method from [42]. Therefore, a linear-torsional method is employed. Its stiness is calculated by geometric parameters [42]. For moving boundary problems of small and medium scale, the solution strategy involves direct methods for linear system resolution, due to its simplicity and robustness. Mesh position \( x_n+1 \) is updated by the sum of the mesh displacement vector to the previous configuration \( x_n \), as given by Equation 19 [42]:
\[
\Delta x = \mathbf{u} + \mathbf{x} \quad \text{at} \quad \Gamma_d
\]  
(19)

3. MATERIAL AND METHODS

Considering the dimension of the spaces given by and base functions denoted \( \text{dim}(\mathbf{V}) = \text{dim}(\mathbf{U}) = \text{dim}(\Omega) = N \) by \( \{ \mathbf{N}_i, j = 1 \}, \{ \mathbf{N}_j, i = 1 \} \) and \( \{ \mathbf{N}_j, j = 1 \} \), the discrete matricial problem form is given as follows (Equations 20 to 24):

Step 1: Velocity Calculation:
\[
A \Delta \mathbf{p} = \mathbf{F}_p
\]  
(20)

Step 2: Pressure Calculation - Poisson problem:
\[
A \Delta \mathbf{p} = \mathbf{F}_p
\]  
(21)

Step 3: Velocity projection in a divergent-free space:
\[
\mathbf{u}(x, t) = \mathbf{u}_d
\]  
(22)

Step 4: Kinetic turbulent energy calculation:
\[
M_{\omega \omega} \Delta \omega^2 = \mathbf{F}_{\omega \omega}
\]  
(23)

Step 5: Turbulent frequency calculation:
\[
M_{ij} \Delta \omega^2 = \mathbf{F}_{ij}
\]  
(24)

Where the matrices \( M_{\omega \omega}, M_{ij} \) and \( M_{ij} \) are the mass matrices for the velocity and turbulent quantities respectively and \( A \) is the Laplacian matrix for the pressure. Those matrices are given by Equation 33:
\[
M_{ij} = \frac{1}{A} \mathbf{N}_i^T \mathbf{N}_j ; \quad A_{ij} = (\nabla \mathbf{N}_i, \nabla \mathbf{N}_j)
\]  
(25)

The vectors \( \mathbf{F}_d^*, \mathbf{F}_p^*, \mathbf{F}_{\omega d}^* \) and \( \mathbf{F}_{\omega d}^* \) are related to the right hand side discretization of the matrix equations on steps 1 to 5. The boundary integral terms, related to the boundary conditions, are added into these vectors.
Remarks:

• The linear system solution of steps 1 and 3 involve the mass matrix. In order to enhance the convergence rate, this matrix is lumped in a diagonal form. The lumping is performed once in the beginning of the iterative process.

• The linear system for the pressure correction problem, step 2, is solved by the Conjugated Gradient Method, preconditioned by the partial Cholesky factorization. This matrix is stored by a Morse strategy, and the preconditioning is also performed once when this matrix is firstly computed.

• The time step is controlled by a weighted average between convective (Δt_{conv}) and diffusive (Δt_{diff}) time steps, given by Equation 26 [35]:

\[ \Delta t \leq \frac{\Delta t_{conv} \Delta t_{diff}}{\Delta t_{conv} + \Delta t_{diff}} \]  

(26)

3.1. Computational Details

The case in analysis is a NACA 0018 airfoil in upward and downward movement, going above the normal stall angle of attack (Figure 2) [14]. The airfoil movement is governed by the following equation:

\[ \alpha(t) = 10 + 15 \sin(\omega t) \]  

(27)

At the inlet, the velocity is set to match a Reynolds number of 105. The outlet boundary condition has a zero reference pressure. It is imposed at the airfoil mesh velocity, in order to recover a zero fluid velocity at the ALE equations. The mesh has 8842 nodes and 16396 P1 elements. The airfoil is moved up to 25 degrees and then, moved back to 0 degrees (Figure 2).

The lift and drag coefficients are given as follows [43]:

\[ C_l = \frac{F_l}{0.5 \rho u_\infty^2 A} \]  

(28)

\[ C_d = \frac{F_d}{0.5 \rho u_\infty^2 A} \]  

(29)

Where \( u_\infty \) is the inlet velocity and \( A \) is the airfoil surface area.

4. RESULTS

Figures 3 shows the flow topology during the airfoil movement. One can observe a flow reversal characterized by a layer close to the upper camber at the upstroke. There is a slight indication of flow reversal at \( \alpha = 13^\circ \) (Figures 3(b) and 3(c)). The reversal becomes more evident with the increase of the angle of attack. The boundary layer is attached to the airfoil, preceded by a shortened separation bubble. The flow reversal has an intermittent nature, caused by turbulent fluctuations at this region. When the angle of attack surpasses 20 degrees, flow reversal and turbulent boundary layer breakdown occurs. The leading edge vortex is originated right after \( \alpha = 13^\circ \), in conjunction with flow reversal onset. Its size at that moment is very small. The flow reversal begins to slowly decrease the pressure at the leading edge, and at \( \alpha = 19^\circ \) (Figures 3(c) and 3(d)) the leading edge vortex growth is accelerated and it begins to move downstream. At \( \alpha = 25^\circ \) the leading-edge vortex covers 80% of the airfoil chord length. On the downstroke a front-to-rear flow reattachment on the upper camber is noted, preceded by a secondary vortex that happens at the trailing edge, noted about \( \alpha = 16^\circ \) (Figures 4(a) and 4(b)). This vortex moves the separation point upward, leading to flow reattachment and therefore, reversing upstroke effects.

For oscillation above normal stall, the boundary layer and stall effects are complex, due to the presence of reversal flow, turbulent breakdown and leading edge vortex dynamics [14]. Other effects are seen at the aerodynamical coefficients: Its curves shows hysteresis, as shown in figure 5. This hysteresis is
originated from the angles of attack non-symmetry for separation and reattachment. On the airfoil upstroke, the leading-edge transition has a major effect on the flow by making the leading edge vortex grow and move downstream. The effect of leading-edge transition has a laminar nature. Therefore, it is identified as a difficulty for numerical simulation to predict adequately its effects [10]. On the airfoil downstroke, it is noted a brief stabilization on the drag coefficient by the experimental data. This stabilization marks the beginning of reattachment flow, where the leading edge vortex diminishes and the boundary layer reattaches.

The reattachment effects are defined by a pressure gradient, which reattaches the flow and halts the vortex shedding process. One can note that the drag and lift are well predicted below 14 degrees. But when it goes above this value, the numerical results begin to depart from experimental data. One can note also that the downstroke numerical results for reattachment (below $\alpha = 15^\circ$) are underestimated when compared with experimental data. This effect can be explained by leading edge transition, which becomes crucial when the airfoil achieves certain values of angle of attack. The DES results tend to be closer to experimental data at the lift curve. For the drag results, both models have similar results. Other factor to be considered is $y^+$ parameter. If its value is not adjusted to the turbulence model, the simulated results will fail in predict drag effects [23]. The mesh motion altered that parameter due to element deformation, which also explains the obtained results.

5. CONCLUSIONS

Numerical results of dynamic stall effects over a moving NACA 0018 airfoil are presented. The results are showed in velocity vectors, streamlines, pressure and vorticity contours. Lift and drag results were compared with experimental data. The simulation was performed in a finite element solver with the Arbitrary Lagrangean Eulerian methodology. A Pseudo-structural methodology was employed to mesh motion.

Some physical effects were observed by the simulation: The results also showed the downstream movement of this vortex. A front-to-rear reattachment on the airfoil downstroke is noted by diminishment of the leading edge vortex. The turbulent separation moves upstream until the flow returns to its pre-stall topology. The lift and drag curves show a hysteretic behavior. Physical effects were observed by the simulation at angles of attack close to its experimental counterparts. The lift curve is in good agreement with experimental data. The drag curve predicted well the upstroke effects, but the peaks and reattachment zones holds dierence with experimental data.

One can conclude that the results can be considered as good. For precision improvement, a transition model, as shown in [10], can enhance the reattachment results. Also, a better control of the $y^+$ during the mesh motion can predict better the drag effects.

6. REFERENCES

On the hydrodynamics of a row arrangement of hydrokinetic propeller turbines

Brasil Junior, Antonio C.P.; Mendes, Rafael C. F.; Oliveira, Taygoara F.; Andriamparany, Tamby; Kouidri, Smaine; Mesquita, André L.A.

ABSTRACT
A numerical study of the hydrodynamics of propeller hydrokinetic turbines is presented. A set of horizontal axis hydrokinetic turbines with diameter of 2 meters and 10 kW of rated power were installed in a row arrangement of machines operating together. The CFD simulations have found the better distance between the machines axis, where the influence in the performance of each machine is small and the spurious near wake influences can be neglected. An evaluation of the wake effects in the shadowing a second row of machines, placed in a staggered arrangement, is also presented.

KEYWORDS: Hydrokinetic turbines, Array arrangement of free flow turbines, CFD and BEM methods.

1. INTRODUCTION
The hydrokinetic turbines have been considered as an important emergent technology for renewable energy conversion. The energy resources associated to the marine or river streams can represent an alternative way to produce electricity issued from the energy of natural water streams.

This technology has been explored in different worldwide projects [1-4] with applications in tidal [5] and river streams [6]. The actual status of the technological development of hydrokinetic turbines has reached important innovation advances, consolidating a base of a robust and sustainable technology.

The use of hydrokinetic turbines in river streams is particularly interesting to promote local sustainability for decentralized electricity generation in riverside areas [6-8]. In those situations, small or medium size turbines, aligned to the stream, can be used to convert the kinetic energy available in the river cross sections. Those devices can be installed with low environmental impact using floating systems or submersed bottom-fixed machines. Compatibility to the local fluvial transportation and other competitive usage can be always attained.

The hydrodynamic problem related to the hydrokinetic machines working in rows arrangements has been explored in some recent works [9-11]. Many practical aspects related to the river conditions can constraint the operation of large diameter machines. A small depth of a given site requires the use of smaller machines. Considering a required power, one needs a group of turbines working together in a framework of an array of hydrokinetic turbines. The study of the array-effects in those arrangements, will define the strategy to explore a hydrokinetic energy site, determining the losses of the turbines performance taken into account the proximity of neighbor machines.

The present work will explore the hydrodynamic behavior of a set of propeller type horizontal axis hydrokinetic turbines operating in a row a first approach to use the available energy resource.

The simulation of the fluid flow, by means of computations fluid dynamics approach, allows the estimation of the energy conversion in each machine and the influence of the unsteady wakes in neighbor turbines and in the next row of turbines.

The research question in this problem is related to the determination of the reduction of machine performance, taken into account the unsteadiness of the turbine wakes. The time-dependent fluid flow induces a dynamic behavior in energy conversion rate, allowing additional vibration levels.

1.1 Horizontal axis propeller hydrokinetic turbine
The test case studied here employs a four-blades propeller-type floating hydrokinetic turbine, presented in the Figure 1. This machine has a rated power of 10 kW for a water stream velocity of 2.5 m/s. It is a machine with 2 m of diameter, which will work together in a row arrangement as shown in Figure 2. This machine will be installed with floaters. The Energy and Environmental Laboratory, at University of Brasilia, had held the hydrodynamic and mechanical design of the machine with collaboration with group of R&D partners. The main purpose of the machine developments is to obtain a sustainable technology to convert energy for small remote communities or for recovery the remaining energy potential at downstream of hydroelectric power plants. In this case, a group of machines has to work together, to maintain the energy demand of a village with few hundreds habitants.
2. MATERIAL AND METHODS

In order to study the fluid flow in the row arrangement of hydrokinetic machines, two different approaches will be employed: The BEM method (blade element momentum), which is used to obtain the turbine performance for a stand-alone machine in different operation conditions, and CFD computations, which are carried out to calculate a complete description of the unsteady 3D turbulent flow though the turbine runner.

Firstly, simulations of a stand-alone turbine are carried out using two different methodologies. In a second case, the simulations of a row of turbines have been carried out using CFD approach, with periodic boundary conditions in lateral planes], which is equivalent to have a row of machines.

2.1 Blade Element Momentum method

The BEM method is an important approach to evaluate the performance of a free flow turbines (wind or hydrokinetic machines). This method is presented in many textbooks ([12-13] for instance) and some additional ad hoc corrections had been formulated and reported in the literature, allowing better realistic results for the turbine performance simulations [14-16].

Considering the water flow velocity denoted by \( V_0 \) and the angular velocity of the machine rotor given by \( \omega \), classically the axial velocity of the flow in the rotor plane can be expressed by:

\[
V = (1 - a)V_0
\]  
(1)

and the tangential velocity in the wake induced by the rotor is given by:

\[
V_\theta = 2\omega r a'
\]  
(2)

The equations (1) and (2) introduce the axial and tangential induction factors denote by \( a \) and \( a' \) respectively. Those variables are dependent of the radial position \( r \) in the rotor plane.

The conservation of axial and angular momentum of fluid flow through the rotor allows the equations for thrust and moment in the radial position \( r \) in a cross-flow section \( dr \) (Figure 3). The lumped balance in \( dr \) can obtain the following expressions:

\[
dT = 4\pi \rho r V_\theta^2 a(1 - a)dr
\]  
(3)

\[
dM = 4\pi \rho r V_\theta(1 - a) a'dr
\]  
(4)

where \( \rho \) denotes the water density and \( dT \) and \( dM \) are the incremental in the thrust and torque at the section \( dr \).

The formulation is complemented by the computation of the hydrodynamic forces in a blade element (Figure 3), by means of the estimate of drag and lift in the hydrofoil section. Considering a rotor with \( Nb \) blades, the thrust and the moment at \( dr \) can be this be computed complementarily by:

\[
dT = \frac{1}{2} \rho N_b c \frac{V_\theta^2 (1 - a)^2}{\sin^2 \phi} C_m dr
\]  
(5)

\[
dM = \frac{1}{2} \rho N_b c \frac{V_\theta(1 - a) \omega r (1 + a')}{\sin \phi \cos \phi} C_t dr
\]  
(6)

In those equations \( c \) is the chord length in the radial position \( r \) and \( \phi \) denotes the inflow angle, determined from the velocity triangle (Figure 3) as:

\[
\tan \phi = \frac{(1 - a)V_\theta}{(1 + a') \omega r}
\]  
(7)

The variables \( C_n \) and \( C_t \) are calculated using the lift and drag airfoil coefficients (CL and CD) as:

\[
C_n = C_L \cos \phi + C_D \sin \phi
\]  
(8)

\[
C_t = C_L \sin \phi - C_D \cos \phi
\]  
(9)

Using the equations (3)-(6), the expressions for the induction factors can be obtained:

\[
a = \left( \frac{4\pi \sin^2 \phi}{\sigma c_n} + 1 \right)^{-1}
\]  
(10)

\[
a' = \left( \frac{4\pi \sin \phi \cos \phi}{\sigma c_t} - 1 \right)^{-1}
\]  
(11)

In those equations \( \sigma \) denotes the local solidity, which is defined as:

\[
\sigma = \frac{c N_b}{2\pi r}
\]  
(12)

A correction function \( F \) is introduced to take into account the hydrodynamical effects of proximity of blade section to the tip and to the hub. The present implementation of the BEM methodology has considered these corrections, with the additional Buhl–Glauert formula for \( a > 0.4 \) [16].

The equations (10)-(11) define a non-linear problem associated to obtain the pair \( (a,a') \) for each radial position in the runner. The solution of this problem, for each radial position \( r \), can be attained by an iterative procedure. For each iteration, the inflow angle is computed by the equation (7) and the coefficients \( C_n \) and \( C_t \) can be updated by means of the computation of the polar hydrofoil coefficients \( C_L(a) \) and \( C_D(a) \), with attack angle \( a \) obtained from:

\[
\alpha = \phi - \theta
\]  
(13)

In this equation, \( \theta \) is the blade-setting angle, which is a known data from the design geometry of the turbine blade.

Classically, iterative fixed-point relaxation methods are employed to find the results for the induction factors given by the equations (10)-(11). In some situations, this simple approach fails in the conditions of post-stall attack angle and/ or in high values of the axial induction factor. In those cases the convergence of the search algorithm is not achieved. To circumvent this problem two additional changings in the methodology have to be implemented: the first one is to extrapolate the polar curves (lift and drag coefficients) for high attack angles (e.g. [17]). The second, advanced non-linear
searching algorithms has to be used, allowing robustness by means of optimization methods [16] or to algebraic reformulation of the non-linear problem [18]. All those enhancements have been implemented and tested in the present paper.

Integrating the equation (4), an expression for the power coefficient can be obtained. Knowing the converged values of \( \{a, a'\} \) at each radial position in the blade, the \( C_p \) can be computed by:

\[
C_p = \frac{8}{\pi^2} \int_0^1 \alpha (1-a) \lambda_r^2 \, d\lambda_r \tag{14}
\]

\[
\lambda_r = \frac{\omega r}{V_0} \tag{15}
\]

The performance variables of free flow turbines are classically defined by:

\[
C_p = \frac{P}{\frac{1}{2} \rho A V_0^2} \tag{16}
\]

\[
\lambda = \frac{\omega R}{V_0} \tag{17}
\]

In those equations \( P, A \) and \( R \) denote respectively the power, area and radius of the machine.

A BEM code was written in Matlab environment, with interface of XFOIL [19] to compute the hydrofoil drag and lift coefficients. The iterative procedure proposed in [18], which guarantees the convergence in all condition of inflow angles, is employed. The Matlab code allows the robust results for power coefficient for all range of machine operation \( \lambda \), in stand alone condition.

### 2.2 CFD computations

The fluid flow through the axial turbine was simulated also using the computation fluid dynamics approach. An unsteady Reynolds averaged approach (URANS) was used to describe the turbine flows with time-dependent velocity, pressure and turbulence variables fields. This strategy allows the description of the transient behavior of the flow, in particular in the wake region.

The ANSYS-CFX commercial code was used in the present simulations. The turbulence model k-ω/SST was employed in two different sub-domains: The first one is a parallelepiped domain that defines the large fluid region influenced by the machine. The second one is an immersed cylindrical region that contains the runner. In this volume the flow is described in rotating reference frame with the same rotation speed of the turbine rotor. The domain dimensions for the stand-alone machine simulation are shown in the Figure 4 and the boundary conditions employed in the simulations are described in the Figure 5.

The first step to perform realistic simulations of the flow field through a hydrokinetic turbine rotor is to develop a systematic mesh study. The mesh generation in present study was performed using the ANSYS-CFX/MESH software. Tetrahedral elements where used in an unstructured mesh, with refinement in wake and near wall regions as illustrated in the Figure 6. The near wall refinement guarantees small values of \( y^+ \) variable that is an important condition to have better results with the turbulence model.

Table 1 presents three different mesh densities used for simulations in the mesh study, where the convergence for the values for \( C_p \) is achieved for meshes around 5.5 millions nodes. The present grid resolution is equivalent to the simulations reported in some recent papers [20-21]. The results reported in those references could be able to describe realistically all the flow patterns (mainly the wake hydrodynamics) and had allowed good previsions for the machine performance.

<table>
<thead>
<tr>
<th>Mesh Nodes</th>
<th>( y^+ )</th>
<th>( C_p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>426 216</td>
<td>323.6</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>1 285 254</td>
<td>117.8</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>5 572 162</td>
<td>1.6</td>
</tr>
</tbody>
</table>

3. RESULTS AND DISCUSSION

#### 3.1 Stand alone machine simulation

Simulations of a stand-alone machine were performed for inlet flow velocity of 2.5 m/s. The range of Tip Speed Ratio (TSR) from 0.5 to 3.0 could be reached varying the rotation...
speed of the runner (rotation of referenced frame sub-domain). The performance curves of the machine can be obtained using CFD and BEM methodologies in the entire range of the curve \( C_p \times \lambda \).

For each different operation condition, the performance of the turbine was computed by means of the calculation of the hydrodynamical torque on the blades, using the surface integral of the normal and tangential hydrodynamic forces (using pressure and viscous stresses) issued from CFD computations.

Flow visualizations of the main fields (pressure, velocity, kinetic energy of turbulence and vorticity) were obtained to describe the effects of the wake. Spurious recirculating zones can be also being identified using the blade surfaces streaklines. The boundary layer detachment for the different conditions can be observed for specific ranges of TSR.

### 3.1.1 Performance results

Figure 7 presents the performance results for the stand-alone machine, obtained from the CFD and BEM simulations. The results obtained by the two different approaches have good coherence in a large range of TSR. A typical behavior of a hydrokinetic turbine can be observed in this plot, with a maximum value of \( C_p \) around TSR equal to 2.0. The maximum value of \( C_p \) is attained close to the design conditions of the machine.

For high values of TSR, the flow close to the blades is completely attached on the suction and pressure sides. Those conditions are related to the small values of attack angles (defined by the velocity triangle in each blade section). The situation changes for small values of TSR (equivalent to high attack angles) where the flow is detached to the blade surface on the suction side for \( \lambda < 0.9 \). The performance in those conditions decreased fast.

The results obtained by CFD and BEM are very adherent in particular in the vicinity of the maximum power coefficient. It is the most important key point to the validation of the optimized design approach. The design target to this turbine was to achieve a rated \( C_p \) around to 0.4, as can observed in the Figure 7.

The visualizations presented in the Figures 8 illustrate the behavior of the flow close to the turbine blades. The condition of detaching flow for low values of TSR can be observed.

The turbine optimized design methodology has attained a performance curve with flatness variation around the maximum value of power coefficient. In a practical point of view, it means that the machine will work near its best performance condition in a large range of tip speed ratio (1.2 - 2.5). It is a good advantage for the real machine operating conditions taken into account the existence of small variations of flow velocity in water stream.

### 3.1.2 Flow patterns

The flow visualizations of the Figure 9 present the typical conditions of the turbine wake and its characteristics vortex. The streamlines and the levels of vorticity components are used to characterize the wake. The tip and hub vortices can be observed. Those structures have been obtained from the transient solution of the flow and these figures represent an instantaneous snapshot of the vortex structure.

It can observe that the tip vortex establishes a wake influence zone in the rear region close to the runner. The vortex has a small radial dissipation, enlarging the radial influence zone up to 1.2D.

On the other hand, the viscous dissipation also acts in order to reduce the level of rotation in the axial direction. In a practical point of view, the wake shadowing effect of one machine is always presented in the near region (< 8D), and all wake rotation effects are not dissipated for a large distance from the rotor.

The influence of the wake can also be observed in the flow visualizations in the Figure 10. In this Figure levels of axial velocity and kinetic energy of turbulence are plotted in the turbine middle-plane, for different condition of TSR. The diffusion of wake effects can be observed, radially limited to a 1.2D. For small values of TSR, the wake shadowing is more important and restraints the zone of installation of the machines in the rear region.

---

[Figure 7: Performance of stand alone machine (points CFD - line BEM).]

[Figure 8: Streaklines on blade surfaces and pressures levels on the middle blade cross-section plane.]

[Figure 9: Wake flow visualization – Iso-vorticity surface and streamlines.]

[Figure 10: Axial velocity and kinetic energy of turbulence on turbine middle-plane.]
3.2 Turbine in row arrangement

In order to obtain the description of the flow through the row of hydrokinetic turbines, CFD computations were performed in a box with periodic lateral planes (see Figure 5). For different distances between the machines, the width of the box is dimensioned for each situation of the machines arrangements. Using this approach, the same mesh density of the stand-alone simulations can be employed, with a properly periodically boundary conditions prescribed in the lateral plane and the same refinement in the near wall and the wake regions. Unsteady simulations were carried out for different distances of the turbines in the row. The condition of TSR close to the best point in the stand-alone situation is considered ($A=2.0$).

The flow visualizations in the Figure 12 and 13 present the axial velocity and kinetic energy of turbulence levels for different machine distances. No visual important interference between the two wakes is observed. In these figures a free flow corridor region with a width of one turbine diameter is plotted between the neighbors turbines. The aim of this visualization is to place a second line of turbines, without a spurious influenced of the first row. The distance of 2.27 D between the machines takes a properly free influenced region for the installation of a second row. In fact, the axial velocity on the second row is slightly augmented due to the hydrodynamical mirroring effect (see Figure 13). Unfortunately, this effect cannot be explored cause that the unsteadiness of the fluid flow that is also observed as consequence of the dynamical wakes vortices effects.

In order to verify the influence in the performance of the neighboring lateral machines the Table 2 presents the $C_p$ for different distances between the machines – the results present slightly changings, which confirms the small interference between the wakes of the both machines. No time influence of the $C_p$ values was observed in any simulation cases.

![Figure 14: Axial velocity in the turbine wake – Best position for a second row machine.](image)

**Table 2:** Power coefficient for different turbines distances.

<table>
<thead>
<tr>
<th>Machine distance</th>
<th>1.36 D</th>
<th>1.81 D</th>
<th>2.27 D</th>
<th>2.72 D</th>
<th>Stand-alone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power coefficient</td>
<td>0.418</td>
<td>0.401</td>
<td>0.395</td>
<td>0.387</td>
<td>0.383</td>
</tr>
</tbody>
</table>
4. CONCLUSIONS

Numerical simulations of hydrokinetic flow in propeller hydrokinetic turbines were performed. For stand-alone machines, the simulations were performed by BEM and CFD methodologies. The results had shown a good coherence for the both approaches, for the large range of Tip Speed Ratio, validating the methodologies in terms of numerical parameters (mesh density, turbulence modeling, domain dimensions, etc.) and the qualitative behavior of the fluid flow.

The simulations of turbines operating in a row arrangement have shown a small influence in terms of performance of neighboring machines (steady and unsteady flow).

The flow visualization of the wake had permitted to define the best position of the machines located in the second line, in order to reduce the effects of the shadowing of the first row. The best position of a third machine is illustrated in the Figure 14. Using this arrangement, with the proposed values for the machine distances, the installation of two rows of machines can operate properly, with slightly hydrodynamical spurious effects between the neighbor’s turbines.

5. ACKNOWLEDGEMENT

This work is partially financed by the HYDRO-K project in a context of ANEEL P&D Grant with sponsoring of AES-Tietê company.

6. REFERENCES

ABSTRACT

A modern design of hydrodynamic equipment requires a sophisticated analysis. In the specific case of present turbomachinery exploitation requirements involve a good performance when the machine is operated under several flow conditions. The consequence is that an ‘elastic hydrodynamic behavior’ is required, but an undesired outcome of these requirements is the apparition of both fluid-structure interaction and cavitation phenomena in many cases. Several issues concerning to turbulence and cavitating flows must be considered to obtain suitable results in these designs and to obtain a better knowledge of the machine behavior. Numerical simulations become a useful tool in design hydraulic equipments, but in order to obtain accurate simulations it is necessary to know the capabilities of the available numerical models for turbulence and cavitation processes which involve several calibration/optimization tasks based on the physics of these kinds of flow. It is possible to study the complex flow in hydraulic devices once this calibration/optimization work is achieved, because physical experimentation is very expensive and, in many cases, it is not possible to observe the precise details of the fluid flow that allow to improve the final design of the device. This work demonstrates that it is possible to capture these fine details of the unsteady fluid flow in turbomachinery working under design and off-design conditions, and aims to provide several quantitative criteria for suitable simulations. It was also demonstrated that the numerical results obtained with non-calibrated models could be improved by means of a careful selection of the turbulence models and their suitable calibration.

KEYWORDS: turbomachinery design, turbulence, CFD, validation, calibration, RSI.

1. INTRODUCTION

Nowadays, the trend in the imposed working requirements for hydraulic equipments is an ‘elastic hydrodynamic behavior’ what means to cover several operation conditions during its exploitation.

In many cases (e.g. hydraulic turbomachinery, valves, injectors, others) and under certain working conditions, the flow behavior inside the equipment leads to the apparition of complex phenomena such as a fluid-structure interaction, FSI, and a cavitating flow. Both phenomena are potentially dangerous because they provoke a low performance of the machine in its operation and the life of the equipment is shortened too.

The design of the hydraulic equipment that fulfills this elastic behavior requires a broader and deeper knowledge of the Mechanics science. It is broadly known that the branch of the Mechanics science called Computation Fluid Dynamics, CFD, is a useful tool in engineering design; but, to obtain accurate simulations of industrial flows it is necessary to assess the capabilities of the available numerical models for turbulence and cavitation. This assessment involves several calibration/optimization tasks based on the physics of these kinds of flow. Industrial engineers from companies involved in CFD tasks rarely spend enough time to perform these analyses, because it is broadly known that ‘time is money’ for companies. At present, collaboration between companies and universities are trying to ‘fill this gap’.

In order to cover the growing requirements of human resources possessing these specific skills (that imply tasks meant to form/train people competent enough to deal with these requirements), a Research&Development+innovation Group, R&D+i CFD/LAMA Group, was created in the Departamento de Electromecánica of the Universidad Tecnológica Nacional, Facultad Regional Mendoza, Argentina. This University has recently joined the LAWG/IiHR Group in order to enhance this collaborative environment among several Latin American universities and companies.

The main subject of the CFD/LAMA is to apply numerical methods to the engineering design, both in the branches of the thermo- fluid- mechanics as well as in the solid mechanics. One of the goals of CFD/LAMA is to apply the CFD science for modeling complex flows in hydraulic equipment trying to improve the equipment designs, by means of the study of the flow behavior inside this kind of equipment with more detail. Some of the specific subjects are the study of the FSI in turbomachinery, pointed to the pressure fluctuation simulations, and the multiphase flows oriented to the simulation of cavitating flows.

It was mentioned that a good knowledge of the ‘CFD environment’ in complex flows involving turbulence and cavitation phenomena implies a deeper understanding of several transport phenomena depicted by transport equations (mass, momentum, energy, turbulence and multi-phase mixture). It is possible to write them starting from each particular form (i.e., mass, momentum, energy, turbulence, etc. equation) in a generalized form, by introducing a general scalar \( \varphi \) that represents the ‘transported’ property, and a subscript \( m \) that represents the ‘mixture’, i.e., the multiphase flow [1]:

\[
\frac{\partial (\rho \varphi)}{\partial t} + \nabla \cdot (\rho \mathbf{c} \varphi) = \nabla \cdot (\mathbf{G}_m \varphi) + S_{\varphi_m}, \quad \varphi = \left[ \rho, \mathbf{c}, T, Y, k, c, \ldots \right]
\]

(1)

Where \( \rho \) is the density, \( \mathbf{c} \) is the velocity vector, \( S_{\varphi_m} \) is a general source term, \( \mathbf{G}_m \) is the diffusion coefficient; and \( \varphi \) represents the dependent variables (scalars as the fluid temperature \( T \), the...
mass fraction for \( j_m \) species \( Y_j \), the turbulence kinetic energy \( k \), or its dissipation rate \( \varepsilon \); or vectorials as the velocity \( c \) in the system. It is highlighted again that the subscript \( m \) designates ‘mixture’, i.e., remarking that the fluid is a two- or multi-phase fluid. With the introduction of a cavitation model, the complete set of governing equations for turbulent cavitating flows can be derived from the foregoing generalized scalar transport equation.

1.1 Common strategies for modeling the turbulence

In order to define a turbulence model, the momentum equation from Eq. 1 is transformed by means of a special averaging, obtaining the so-called Reynolds Averaged Navier Stokes equations, RANS, where the turbulence fluctuations are set in an explicit form, giving place to a new term in this equation. Then, this term is modeled by means of a turbulence model to transfer the effects provoked by these fluctuations to the computed mean flow (see details of some Refs. in Table 1, and Ref. [2]). This modeling gives place to the apparition of, e.g. \( k \) and \( \varepsilon \) variables in the generalized Eq. 1, when the variable \( \Phi \) is replaced by \( k \) or by \( \varepsilon \), typical variables of the broadly known Std \( k-\varepsilon \), model, see Refs. in Table 1, (see more details in [2]).

Several modeling strategies for the turbulence are available nowadays, starting from model of zero equations to models of one, two or four equations in the family of the Eddy Viscosity Models, EVM, so-called scalar models. More sophisticated ones are the Reynolds Stress Models, RSM, (so-called tensorial models), and arriving finally to the more recent ones, the so-called Large Eddy Simulation, LES models [2]. The latter models use quite a different strategy for averaging the Navier Stokes equations (spatial filtering), but this one also leads to a turbulence (or eddy) viscosity definition. Here, it is important to note that once a turbulence model is introduced into the momentum equations, these equations no longer carry any information concerning to their derivation (i.e., some kind of averaging technique). Both the RANS as well as the LES models are EVM that are used to substitute either the Reynolds- or the sub-grid stress tensor respectively.

After the introduction of a turbulent (or eddy) viscosity, both the RANS and LES equations are formally identical. The difference lies exclusively in the turbulence scale represented by the eddy viscosity used/computed by the underlying turbulence model.

1.2 Common strategies for modeling the cavitation

In order to define a cavitation model in a general CFD code a vapor transport equation is introduced, since there is a process of liquid-vapor mass transfer for cavitating flows. It is considered that this liquid-vapor mass transfer process (evaporation and condensation) is governed by the following vapor transport equation, (see details in [1]):

\[
\frac{\partial \alpha \rho_v}{\partial t} + \nabla \cdot (\alpha \rho_v c) = R_c - R_v
\]  

Where \( \alpha \) is the phase volume fraction, the subscript ‘v’ indicates the vapor phase; Re and Rc are respectively the mass transfer source terms connected to the growth and collapse of the vapor bubbles. These terms account for the mass exchange between the vapor and the liquid phases during the cavitation process. Then, these terms can be modeled using the Rayleigh-Plesset equation [3-7] that describes the growth of a single vapor bubble in a liquid. The final set of Eqs. (3-4) form the basis of several two-phase cavitation transport models included in several CFD codes after some manipulations, (e.g., see details in [1]).

\[
\frac{dR_B}{dt} = \frac{2P_B - P}{\sqrt{3} \rho_l}
\]

\[
\frac{\partial \alpha \rho_v}{\partial t} + \nabla \cdot (\alpha \rho_v c) = \frac{\rho_v \rho_l}{\rho_m} \frac{d\alpha}{dt}
\]

Here, \( R_B \) is bubble radius; \( \rho_v, \rho_l, \rho_m \) are liquid, vapor and mixture densities; \( P \) and \( P_B \) are fluid and bubble surface pressures respectively.

1.3 Applying CFD for turbomachinery design

A general code to apply to the modeling of a real turbomachinery comprises the set of equations aforementioned. To solve this set of equations comprising turbulence and cavitation is a not trivial task, and it is necessary to take into account a lot of calibration parameters, together with the necessity of a careful validation of submodels (turbulence and cavitation ones). In order to perform the validation/ calibration task, experimental databases are necessary and they are of paramount importance. Unfortunately, experimental measurements for turbulent cavitating flows in turbomachinery are very scarce because the major problem of physical experimentation is its high cost. In general, the experimental studies related to the vapor cavities structures present in the cavitating flow and their behavior are challenging due to the fact that cavitation is a very complex phenomenon and typically occurs in locations where the access to measuring instruments is limited; and it is also due to the presence of high velocities, high void fraction, and a considerable splitting of the dispersed phase [8,9]. It is also necessary either to measure or to control a lot of parameters (i.e., local pressure and temperatures in the flow, the dissolved gases content, others).

More complexity is added when, in some cases, cavitating flows start showing a periodic behavior in its development, strongly dependent on the fluid-flow state.

Similar observations concerning to the measurement difficulties can be made at the moment of take measurements of the unsteady turbulent flow pattern when different FSI phenomena are present in turbomachinery, e.g., to characterize the complete flow pattern, the boundary layer growing in the rotor or to measure fluctuating pressures when the Rotor Stator Interaction, RSI, occurs between the rows of fixed and moving blades.

It is interesting to highlight that turbulence and cavitation are closely related. Turbulence affects cavitation inception since a nucleus may be found in the core of a vortex, where the local pressure level is lower than the mean value of the pressure in the flow. Hence, the nucleus could cavitate when it might not do so under the influence of the mean pressure level. This fact points out that cavitation may alter the global pressure field by altering the location of flow separation and the induced variations of the local turbulence level; thus, turbulence may promote cavitation and vice versa.

Despite the fact that there are some general characteristics in several turbulent cavitating flows, the observed flow structures depend not only on the hydraulic device geometry but also on the fluid/flow parameters. Unfortunately, experiments
provide detailed information ‘case-by-case’ only. For all the above mentioned reasons, CFD applied to this kind of flows is an active field of research and has significant importance for design engineers.

Both turbulence and cavitation phenomena offer several challenges for a suitable modeling by means of the available CFD codes; and the moving reference frame, a tool commonly used for unsteady modeling of flow in turbomachinery, adds more difficulties to obtain credible CFD results.

2. VALIDATION/CALIBRATIONS TASKS, RESULTS AND DISCUSSION

To obtain a deeper knowledge of the available CFD codes capabilities for modeling complex flow, a brief summary of the work recently developed by the CFD/LAMA Group, where several modeling test cases studied using experimental databases, is presented here. The main goal of these works was to provide several quantitative criteria for suitable simulations in the branch of turbomachinery design, due to the fact that databases having detailed experimental information for turbulent cavitating flow in turbomachinery are very scarce. Using this previous experience, this work is focused on the study of the RSI phenomena and the set of cases selected here are related to specific RSI phenomena that appear in turbomachinery under normal or off-design operation conditions (a similar work focused to study cavitating flows being also performed but the obtained results are not presented here).

It was already mentioned that a RSI phenomenon is a complex flow involving potential and turbulent effects. In order to have a systematic approach to this complex phenomenon, simpler test cases than a real turbomachine, but closely linked to the RSI in the machine were selected for CFD modeling. These experimental databases are related to steady and unsteady flow around isolated airfoils/hydrofoils (see Figs.2, 3 and 4) and RSI in a cascade of moving blades, (see Fig.5). Several validation/calibration tasks were carried out to obtain information about the capabilities for modeling this kind of flows by means of available turbulence models (see more details in Refs. [10,15-18]).

The main goal of these previous studies carried out and summarized here, was to observe the behavior of several EVM models (see Table 1) related to: a) reproduce the pressure coefficient Cp, the mean velocity field (see Fig.2) and its turbulent fluctuations (see Fig.3) for a steady fluid flow over a supercritical profile airfoil; b) reproduce the fluid flow field and the typical oscillation frequency in the wake (von Kármán vortex street) for an unsteady flow over a NACA profile hydrofoil (see Fig.4); c) reproduce the boundary layer fluctuations along a plane plate (stator) and their characteristic oscillation frequencies due to the RSI phenomena in a cascade of moving blades (see Fig.5); d) reproduce the flow pattern in caviting flows in injectors, Venturis and other hydrodynamic devices with simple geometry (results obtained in this last item are not shown in the present work).

This previous work allows to define a suitable CFD setup for subsequent RSI simulations in turbomachinery (geometry meshing, boundary layer definitions, discretization schemes, etc.); because, several sensitivity tests related to: a) the mesh size and cells distribution in the boundary layer, and b) the discretization schemes influence on the results, were performed (not shown, see details in [15]). By means of these previous simulations, some ideas about the capabilities of the different turbulence models used for recovering the detailed structure of these flow patterns were obtained too.

Several useful conclusions were obtained from these previous CFD works, and some of them are summarized here:

a) When a flow around a thin trailing edge isolated foil, or around a stage of fixed and moving cascade of blades was modeled, the SST k-ω turbulence model shows a good performance to compute the profile of velocities in the boundary layer and the general flow pattern of the wake; results obtained with other EVM (see Table 1) do not show big differences in both cases. The fluctuating velocities u’ in general were underest imated; but, ν’ and u ν’ fluctuations were overestimated for all the EVM tested.

b) When a flow around a truncated (i.e., non thin trailing edge) isolated foil was modeled, the SST k-ω turbulence model shows a good performance to compute the velocity profiles along the boundary layer, and it also shows a slight overestimation of the shedding frequency.

c) When a flow around a stage of fixed and moving cascade of blades was modeled, the EVM used shows underestimations of the turbulence intensity \( \frac{\sqrt{\overline{U'v'}}}{U_0} \) compared against experiments, as well as for isolated foils having thin trailing edges and low flow incidence angles.

d) In all the cases tested not big differences were observed in the results obtained by means of EVM, DES or LES models.

If big differences were not observed in the results obtained, it is possible to say that the main advantage of the EVM models is its low computational cost when compared to more accurate turbulence models (but more resource-consuming) such as the RSM or the LES (see details of these comparison in Refs. [10,19]); being this fact of paramount importance when a flow in a complete turbomachine is modeled.

On the other hand, and despite that the DES is an hybrid formulation between EVM and LES that need less computational resources than LES, some results obtained (not shown) allow to say that there is not a clear improvement compared to the results obtained by using EVM [10,19].

2.1 Database selected for Rotor Stator Interaction (RSI) Phenomena

In turbomachinery design, the RSI is an important phenomenon, which has a strong influence in the machine behavior. These interactions can have a significant impact in the vibrational and acoustical characteristics of the machine, because unsteadiness and turbulence play a fundamental role in complex flow structures. The RSI can be divided into two different mechanisms: potential flow interaction and wake interaction. The nature of the flow due to the wake interaction

<table>
<thead>
<tr>
<th>Turbulence models</th>
<th>Notation</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart Allmaras</td>
<td>SA</td>
<td>Spalart and Allmaras [Spalart, 1994]</td>
</tr>
<tr>
<td>Standard k-ε</td>
<td>Std k-ε</td>
<td>Launder and Spalding [Launder, 1974]</td>
</tr>
<tr>
<td>Realizable k-ε</td>
<td>Rlz k-ε</td>
<td>Shi et al.[Shih, 1995]</td>
</tr>
<tr>
<td>Re Normalization</td>
<td>Groups k-ε</td>
<td>RNG k-ε</td>
</tr>
<tr>
<td>Standard k-ω</td>
<td>Std k-ω</td>
<td>Wilcox [Wilcox, 1998]</td>
</tr>
<tr>
<td>Shear Stress Transport k-ω</td>
<td>SST k-ω</td>
<td>Menter, 1994 [Menter, 1994]</td>
</tr>
<tr>
<td>Reynolds Stress Model</td>
<td>RSM</td>
<td>Launder at al.[Launder, 1975]</td>
</tr>
</tbody>
</table>

[Table 1]: Turbulence models used, (see details of the references cited in this Table in [2,13]).
is unsteady and turbulent, and there it also appear three-dimensional (3D) boundary layers, curvature and system rotation effects (see more details in [15]).

The choice of an appropriate turbulence model and the boundary layer treatment is far from trivial, and a suitable turbulence modeling plays an important role for successful CFD results [10]. The experimental data used here are for a single-stage pump with a specific number, \( n_p = 53 \) [rpm, (\( m/s \)), \( m/s \)], being a vaned diffuser pump with five impeller blades \( Z_p \), eight diffuser vanes, \( Z_v \) and a volute casing as shown in Fig.1, (see complete details in [11]). A detailed description of the pump and its specifications of the essential components are summarized in Tsukamoto et al. 1995 [11]. The unsteady pressure measurements was made at several places (stations) in the vaned diffuser passage (see Fig.1), and a comprehensive survey of instantaneous pressure within the diffuser passages was obtained, (see Figs. 8 and 9).

The unsteady pressures were measured by semi-conductor-type pressure transducers, which were installed directly on the pressure taps to prevent the decrease of natural frequency in the pressure measurement systems. Figs.8 and 9 illustrates the unsteady pressure measurement stations in the shroud casing side of the diffuser in the test pump. Due to the limited space in the measuring sections, the pressure taps for tangential traverse were located at one radial location in each passage of the diffuser only. The blade-to-blade distributions of unsteady pressure were identified by a phase shift of the measured data. The coordinates of the static pressure taps were formed by the cross of five radial grid lines and five stream-wise grid lines in a blade-to-blade passage as shown in Fig.1.

### 2.2 CFD results obtained for the pump working under the design condition

Two dimensional, unsteady incompressible Reynolds-averaged Navier-Stokes equations were solved by means of the commercial CFD code Ansys Fluent 12.1 [12]. An entire 2D stage of a diffuser pump was modeled to study the pressure fluctuations due to the interaction between the impeller and the diffuser of the pump. Unsteady fluctuations of pressure in the steady vane and frequencies of the pressure fluctuations in the diffuser passage were computed and compared against experimental results from Tsukamoto et al., 1995 [11]. Full RANS equations coupled with several EVM (see Table 1) were solved for the 2D stage to establish the most accurate modeling strategy for a diffuser pump.

A constant pressure value was imposed at the fluid inlet and a constant pressure value was imposed at the pump outlet. These values were obtained from the characteristic curve of the pump (see mode details in [11]), checking that the maximum efficiency is obtained at this point, i.e., working under the design condition. A non slip boundary condition was imposed in the runner blades, diffuser vanes and volute casing wall. A rotational speed, \( N_z \), of 2066 rpm was imposed to the blade impeller implying that this validation/calibration study is made for the turbomachine working under design conditions (optimal performance).

Applying this previous experience, an entire 2D stage of a diffuser pump was modeled to accomplish the proposed turbulence model performance study. The setup defined for this case consists in an unsteady simulation including a second-order implicit velocity formulation and a pressure-based solver. The SIMPLE pressure-velocity coupling algorithm was used, and a second order scheme discretization was selected for the numerical experiments.

The maximum number of iterations for each time step was set to 40 in order to reduce all computed normalized numerical residuals to an order of O(10^{-5}). The interface between the rotor blade and the diffuser vane was set to a sliding mesh, in which the relative position between the rotor and the stator was updated every time step. The adopted computational time step was 1/360 of the rotor revolution time. Due to the unsteady nature of the flow, it is required that the whole flow domain be affected by the unsteady fluctuations. In order to check the aforementioned situation, a flow rate monitoring was made at the domain outlet.

After starting the simulation, the machine must spin some time while the flow pattern is computed to obtain the uniform unsteady flow and then, check the uniformity in the shape of pressure fluctuations; uniform unsteady flow behavior was obtained after 10 revolutions.

For checking the performance of the turbulent models from Table 1, the results obtained for pressure fluctuations in three monitoring points \( r_1 c_1, r_1 c_3 \) and \( r_2 c_3 \) of the vaned diffuser passage were recorded for every numerical simulation developed. These points correspond to the reported measurement points for the experimental data set from Tsukamoto, and their location can be seen in Fig.1 and Fig.6.

In order to capture the RSI effects, the relationship between the pressure fluctuations and the movement of the rotor vanes in front of the diffuser vanes was extracted from the computed unsteady pressure field. Using a Fourier transform, the characteristic frequencies of the pressure fluctuations were obtained at points \( r_1 c_1, r_1 c_3 \) and \( r_2 c_3 \). Results obtained show that the pressure fluctuates with the impeller blade passing frequency \( Z_{IN} \) and its higher harmonics, (e.g. see Fig.8, more details in Refs. [15,16]).

After that, a detailed analysis of the obtained results allows to notice that using a mesh with \( y^+ \sim 25 \) (i.e., first cell center at a distance of \( y^+ \sim 25 \) from the wall) the Spalart-Allmaras, the Riz \( k-\varepsilon \) and the SST \( k-\omega \) turbulence models show the best adjustment for the experimental pressure fluctuations (see Fig.7). This mesh is also suitable to use Wall Functions, WF, for turbulence models that need this kind of near-wall treatment. Similar results were obtained for refined meshes, (i.e. \( y^+ \sim 5 \)), requiring a Two-Layer Modeling, TLM, for the near-wall treatment for some turbulence models (not shown, see details in [15-17,19]).

It can be noticed that both the Std \( k-\omega \) as well as SST \( k-\omega \) turbulence models have some difficulties to capture some of the representative frequencies of the fluctuations, while all the other models accurately tested capture the characteristic frequencies of the phenomena (see details in Table 3, Ref. [15]).

An interesting result is that it could be possible to recover the unsteady signal from the frequency spectra obtained, allowing to separate the potential and viscous effects. Since the potential effect can be recovered applying a typical theoretical frequency analysis [20], a subsequent subtraction of the potential effect from the total signal obtained by CFD computation (potential+ turbulence) would permit the recovering of the turbulence effect, (see Fig.9). This proposal could be useful to analyze turbulence spectra or which frequency affects more the pressure fluctuations.

On the other hand, the analysis of the associate velocity field to the pressure fluctuations is of great interest in order to understand RSI phenomenon more completely. Unfortunately, the Tsukamoto database has only pressure fluctuations measurements and there is no information about the velocity.
field. Despite this fact, the velocity field was computed and compared qualitatively against similar experiments from Pedersen et al., 2003 [21] and CFD results from Feng et al., 2011 [22] to obtain a more complete idea about the capabilities of CFD to reproduce the flow pattern (see Figs.10 and11). Fig.10 shows the velocity field obtained for the Tsukamoto centrifugal pump both for the rotor as well as for the stator. The CFD results obtained for the rotor were compared to qualitatively against the experimental ones from Pedersen et al., 2003 [21] showing similar velocity distributions in the rotor. The potential effect in the rotor vane is clearly observed, and the structure of the boundary layers along the pressure side and the suction side of the blades was observed too.

Comparisons of the relative velocity components against a numerical database for a centrifugal pump with a vaned diffuser from Feng et al., 2011 [22] was also performed, giving similar flow field configuration, (see Fig.11, and more details in Refs. [16,19]).

After a careful analysis of the obtained CFD results, some conclusions were obtained. All the models tested showed good results for the pressure fluctuations in the vaned diffuser when compared against experimental results from Tsukamoto, except the Std k-ω model. It is possible to notice that the Spalart-Allmaras, Riz k-ε and SST k-ω turbulence models show better adjustments than other EVM for the experimental pressure fluctuations (not shown, see Refs. [15,16,19] for details). When the influence of the boundary layer treatment applied on the results is analysed, it can be observed that meshes with γ+>25 are able to reproduce the RSI pressure fluctuations accurately, despite the fact that they do not allow to recover the boundary layer flow in detail.

In summary, a general conclusion from the CFD results obtained for a vaned pump working under design conditions is that the relationship between the pressure fluctuations and the movement of the rotor vanes in front of the diffuser vanes was well determined. The characteristic frequencies of the pressure fluctuations were obtained, resulting that the pressure fluctuates with the impeller blade passing frequency ZN and its higher harmonics as the experiments show. All the turbulence models tested showed a periodic pattern in the pressure fluctuation in which each cycle is produced by the movement of a rotor blade in front of a diffuser vane, except for the Std k-ω and the SST k-ω turbulence models. For these two models, the pressure fluctuations values were different from each pass of an impeller blade in front of a diffuser blade, but the overall behavior for one revolution of the impeller followed a regular pattern.

Despite some difficulties found to capture all the frequencies of the pressure fluctuations (only the 2ZN was not well captured), the idea that the promising behavior of the SST k-ω model could lead to future unsteady simulations by using the Scale Adaptive Simulation (SAS) option, available for this turbulence model, was explored.

SAS is a recent development related to unsteady turbulence modeling, [23–25], and it is an improved Unsteady Reynolds Averaged Navier Stokes (URANS) formulation, which allows the resolution of the turbulence spectrum in unstable flow conditions. Contrary to the standard URANS, SAS provides two independent scales to the source terms of the underlaying two equations model (e.g., SST k-ω model).

In addition to the standard input for the length scale in form of a velocity gradient tensor ∇U / ∂xj, SAS model computes a second length scale, called the von Kármán length-scale, L_k.

from the second derivative of the velocity field (see details in [24]). The information provided by L_k allows the model to react more dynamically to capture scales in the flow field which cannot be handled by standard URANS models, because URANS recovers only the large-scale unsteadiness, whereas the SAS-SST k-ω model adjusts to the already resolved scales in a dynamic way and allows the development of a turbulent spectrum in the detached regions.

As a result, SAS offers a single framework, which covers steady state regions (computed normally by RANS) as well as unsteady detached flow regions (which must be computed by LES to solve their details), without an explicit switch in the model formulation [24,25]. SAS would allow studies of unsteady flow behavior by means of an Unsteady RANS simulation including a technique for adapting the length scales automatically (SAS-URANS) instead of the more expensive, in terms of CPU requirements, LES option. The SAS-URANS option can be a very interesting tool, since LES modeling for turbulent flows in complex geometries (industrial flows) are not affordable nowadays [14,25].

In summary, the functionality of SAS is similar to the Detached Eddy Simulations (DES), being DES a hybrid formulation that uses both EVM and LES. The LES activity in DES is enforced by the grid limiter, whereas SAS allows a breakdown of the large unsteady structures by adapting the turbulence model to the locally resolved length scale.

This functionality could be explored more extensively to open the possibility to perform unsteady CFD simulations with affordable CPU costs not using so big computational meshes. At the moment the SAS option is only coupled to the SST k-ω model in Ansys Fluent v12.1 [12], and for this reason the behavior of the SST k-ω model was observed carefully.

In order to explore this option, in the present work, the results by using SAS simulations were obtained and compared with the previous ones obtained using EVM, for the vaned diffuser pump under design conditions. Fig.12 shows a comparison among the results obtained using several EVM models, and SAS modeling with some calibration of the Cs parameter. This parameter has a control over the Lvk scale allowing the model to react more or less dynamically, and in this way it is possible to recover more or less unsteady effects. It is showed that a suitable calibration of SAS permits to adjust the pressure peaks in a better way at the r1C2 position, i.e., in the middle of the stator channel, just when the flow from the rotor goes into it (see Fig.6). Near the wall, i.e., at the r1C1 position, the viscous effects are more important than inertial ones and then, the unsteady effect is not so relevant and the SAS term is more insensitive to calibrations.

3. CFD FOR THE TURBOMACHINE UNDER OFF-DESIGN CONDITIONS

Bearing in mind the presented validation/calibration work, now it is possible to select better options for a subsequent CFD study for this turbomachine working under off-design work conditions, being this subject the main goal of this work.

Despite that an experimental database for off-design conditions is not available for this case, it is interesting to highlight that after an extensive validation/calibration of the models it is possible to ‘extrapolate the models performance’, but carefully, to obtain new results when the machine is working under off-design conditions. It is necessary to remark that these conditions provoke an enhancement of the boundary...
layer thickness and the possibility of flow detachment becomes possible, being a challenge for the EVM models without any calibration.

The knowledge of the behavior of the turbomachine working under off-design conditions is of paramount importance for design engineers, and to obtain credible trends in its behavior by means of CFD becomes an interesting option to explore. By varying the flow rate, and defining the suitable head, some results were obtained for off-design conditions, (see Fig.13).

Fig.13 shows the results obtained for several cases, (marked with squares) by means of changing the boundary conditions to simulate the turbomachine working under design or off-design conditions. Notice that if the machine was working under the design point, the mean value for the pressure (non dimensional value \( \psi \)) is well captured by the CFD simulation. The enhancement of secondary flows under off-design operating conditions provokes strong secondary flows that affect the boundary layers thickness and their structure leading to an increase of the turbulence level. To manipulate this flow condition is very hard for EVM, and SAS option could become an interesting one, because LES option is very consuming in terms of CPU resources. Some details of the fluid flow under design and off-design work conditions can be seen in Fig.14. Notice that the CFD adjustments of the curve for off-design conditions go down, but the trend is well reproduced.

Therefore, by means of CFD it is possible to observe details of the complex flow in the turbomachine that sometimes is very hard to obtain by means of experiments. Fig.14 shows that under off-design operation conditions secondary flows and instabilities appear in the rotor, affecting the general flow pattern. These perturbations are convected to the rotor outlet and then, start affecting the stator flow into the channels, leading to a ‘stalling cycle’ in the channel. This ‘stalling cycle’ has a completely unsteady behavior that affects the general performance of the machine. In some cases, and depending on the local flow behavior, a cavitating flow can appear too.

A complete analysis of these stalling cycles can be performed by means of CFD. In Fig.15 the pressure fluctuations in the diffuser are presented, showing an unsteady behavior related with the ‘stalling cycle’ in the channel diffuser. The cycle showed is related to an equivalent time of nine cycles of the rotor blades passing at this point (i.e., \( t=9t^* \)). It can also be seen that the sequence of stall/unstall appears every two diffuser channels.

For cases of low flow rates, i.e., \( Q<Q_d \), the pressure pulsations are stronger than for high flow ratios; this fact is confirmed by experimental results for hydraulic machinery working under this condition (see details in [16]).

Finally it is necessary to remark that all the CFD results presented here, give good adjustments of the experimental results despite they are obtained in 2D geometry. The obtained results were also compared against CFD results for other 2D geometry from the literature [22] giving similar trends for the pressure fluctuations. The experimental measurements from Tsukamoto were obtained in a mixed flow pump (\( n_r \sim 53 \)). It is broadly known that mixed machines have double curvature blades; therefore, the experimental machine has a full 3D geometry.

It is quite easy to obtain a 2D geometry of a turbomachine from the mean characteristics of the pump, pictures and drawings are given in the references found in the literature (e.g. see Tsukamoto, 1995 [11], Feng, 2011 [22]). Instead, obtaining a 3D geometry from the available data is not a trivial task, because the available experimental databases do not provide complete information of the full 3D geometry normally. Despite the fact that to obtain a 3D geometry is not easy, future work will consist in 3D simulations in order to check the RSI phenomena and other fluid-structure interaction (e.g. stator-volute interaction, casing tongue-stator interaction, others).

To reach this goal, it is necessary to reconstruct the 3D geometry using the given characteristics of the machine [11], by means of a ‘classical design’ of turbomachinery, (a skill not very common in the industrial engineers nowadays). A classical preliminary design was carried out to obtain the 3D geometry for this machine (see Fig.16), and the obtained 3D geometry of the rotor can be seen in Fig.17. It can be compared to the geometry used by Shi et al., 2001 [26], in his CFD work. After completing the 3D geometry design, a line of the future work comprises the 3D simulation of the complete machine using the quantitative criteria for the assessment of internal flow state obtained in this work.
[Figure 3: CFD Results for Boundary Layer flow around a supercritical airfoil: fluctuating velocities along the boundary layer. Exp. from Nakayama et al., 1985 (see details in Refs. [16,17]). Nomenclature: \( x'x'/U_{ref}^2 \): local fluctuation / mean velocity value.]

[Figure 4: CFD Results, Unsteady flow over a NACA 0009 truncated hydrofoil. The frequency of the von Kármán vortex street is compared against experiments from Ausoni et al. 2005 (B/W pictures, see details in Ref. [17]).]

[Figure 5: CFD Results for a RSI phenomena. Up, mesh and geometry of the experimental setup for CFD. Down, Results obtained for fr=20Hz, 30Hz and 40Hz for turbulence intensity, \( (16\% = \sqrt{\text{\(\bar{u}'\bar{u}'\)}} U_0) \), (see details in Refs. [16,17]. CFD results show that depending on fr, a different structure of the flow along the plate was obtained.)

[Figure 6: Details of the meshed geometry for a diffuser pump. Notation: rici places where the unsteady pressures were measured [11].]

[Figure 7: CFD results using a mesh with \( y+ \sim 25 \). Left, point r1c3; Right, point r2c3. \( \Delta \Psi = (p-p_s)/(0.5 \rho U_2^2) \) vs \( t^* \), where: \( \Delta \Psi \), non-dimensional unsteady pressure; \( (p-p_s) \) unsteady component of the relative pressure; \( p_s \) total pressure at pump suction; \( U_2 \) peripheral speed of the impeller; \( t^* = t/T_i \), non-dimensional time; \( T_i \), time required to traverse one pitch of impeller blade.]

[Figure 8: CFD results for design conditions, \( Q=1.0Q_d \) for the Spalart-Allmaras model. Left, CFD results for the pressure fluctuations at point r1c3 (see Fig.1); Right, frequency domain showing the harmonics of the unsteady pressure fluctuations obtained from the CFD computations.]

[Figure 9: CFD results using for design conditions, \( Q=1.0Q_d \). From the frequency spectra obtained by CFD it is possible to recover the complete (potential+turbulence) unsteady signal. The arrows point out the composition process for obtaining the complete signal starting with individual frequencies by means of their addition.]

[Figure 10: CFD results using a mesh with \( y+ \sim 25 \) for design conditions \( Q=1.0Q_d \). Left, rotor velocity field; Right, diffuser (stator) velocity field. Notice that quantitatively, the velocity field is similar as one from Pedersen, 2003 [21]. This database corresponds to a centrifugal pump without diffuser.]
Figure 11: Comparison of the CFD results obtained against a similar radial machine (Feng et al., 2011 [22]): velocity field obtained, for design conditions ($Q=1.0 Q_d$). Notation: LDV Laser Doppler Velocimeter measurements [22]; $W_u$, tangential component of the relative velocity; $W_r$, radial component of the relative velocity.]

[Figure 12: CFD results for the non-dimensional head-flow rate $\psi = (2gH_m U_r^2 - 2)$ vs $t^*$. Up: $r_1c_1$ position; Down: $r_1c_3$ position; Left: EVM; Right: SAS. Nomenclature: • Experimental data (Tsukamoto, 1995).]

[Figure 13: CFD results, ($\psi$, $\phi$) Left: curve for the vaned pump; Right: Associated fluid flow pattern for some working conditions. Notation: Non-dimensional head-flow rate curve ($\psi$, $\phi$); pressure coeff., $\psi = (2gH_m U_r^2)$; flow rate coeff., $\phi = (Q) \times \left( \frac{2 \phi_d 2U_r^2}{\rho U_r^2} \right)^{1/2}$; $\psi$, Exp. from Tsukamoto, 1995 [11].]

[Figure 14: CFD results for the flow field under three operation conditions, vaned pump: Left: Flow pattern in the rotor, Right: Flow pattern in the diffuser (stator).]

[Figure 15: CFD results for velocity field and pressure fluctuations, point $r_1c_3$, $Q/Q_d=0.61$. In the sequence of pictures the stalling cycle in the stator (curve in blue) joined with the related velocity field is shown.]

[Figure 16: Summary of steps followed for obtaining a 3D rotor configuration by means of a semi-empirical ‘classical design’ of a mixed pump (Tables and curves are extracted from the common literature found, developed for specific undergraduate Fluid Mechanics courses).]
4. CONCLUSIONS

It is known that EVM models have advantages related to CPU costs compared against other more sophisticated options (e.g. RSM or LES), but it is also true that they have some known deficiencies when used for turbulent cavitating flows simulations. Then, in order to check the capabilities of the available EVM models applied to the turbulent flows simulations, an extensive validation/calibration work was performed. This work was focused on the study of the capabilities of EVM to capture the characteristics of the RSI phenomenon in a vaned diffuser pump.

This work proves by means of a very detailed comparison of the results obtained against experimental databases and other CFD works found in the literature, that it is possible to observe the flow instabilities due to RSI phenomenon that affect the general flow pattern when the machine works under design and off-design conditions. This information is very useful for the design engineer in order to analyse and to propose a more efficient design without carrying out several and very expensive experiments.

It was observed that the SAS modeling option improves the results obtained in some cases when it is compared with the ones obtained by means of EVM models for meshes of similar size. This allows performing unsteady simulations with lower CPU costs compared with DES/LES simulations, because DES/LES simulations need bigger meshes than SAS modeling.

Despite the promising results obtained with SAS, more intensive validation/calibration work is needed for modeling both design as well as off-design conditions using experimental databases having more complete information. At moment these databases are very scarce in the literature.

5. REFERENCES


American Journal of Hydropower, Water and Environment Systems, July 2016 33


An analysis of the mechanical stress in hydroelectric turbines stay-vanes under different operational states of the electric generator

Gonzalez, Facundo E.; Kelm, Diego A.; Kolodziej, Javier E.; Tarnowski, Gabriel A.; Astelli, Raúl; Bordón, Hugo

ABSTRACT
In a large Kaplan turbine of 160MW the vanes of the stay vane direct the water from the spiral chamber to the wicket gate that regulates the turbine functioning condition. These vanes also support the civil structure that holds the combination of the turbine-generator. Like these vanes are working under different load, the knowledge about their dynamic behavior has special importance to decide the operational strategies of the electric generator. An analysis by means of finite elements allowed the determination of the natural frequencies and results are used for choose the localization of the strain gages for stress analysis.

KEYWORDS: Stress Measurements; Hydroelectric turbines; Vibrations; Strain Gages.

1. INTRODUCTION
The vanes of the stay vane have the purpose of directing the flow from the spiral chamber to the mobile vanes of the wicket gate. These vanes are exposed to constant loads, known as the weight of the civil structure, as well as to the force applied by the water flow in a change of direction. At the same time, the flow excites the structure through the pressure waves. These pressure waves have different origins, as the excitation frequency of the mobile parts, variation in the flow, the external vibrations of surrounding machines and the dam conditions.

This article complements the analysis done in Milán and Mirasso [1], with the main difference that in the present analysis the maximum operational conditions of the design are accomplished, and two more functioning conditions are dealt with, the start of the turbine-generator set and the excitation of the electrical generator. These limits are reached based on the net head increase between the time when the measurements in Milán and Mirasso [1] were done and the date of the measurements presented in this article.

2. EXPERIMENTAL PROCEDURE / METHODOLOGY
The vanes analyzed in this article are shown in Figure 1. The analysis is done in vane P4, because it is the one located opposite to the water entrance and the strength of the water, due to the change of direction, is shown in this vane with greater intensity. Vanes P12 and P13 were analyzed with the objective of contrasting the results with studies previously done in Milán and Mirasso [1].

2.1. Finite element model
For the finite elements analysis, the Abaqus® program was used. The analysis consisted on reproducing two of the solid vanes geometries, with their respective boundary conditions. For this purpose, a finite elements model for vanes P4 and P12 was created (Idem P13). In Figure 2a the concrete that defines the model under the boundary condition that limits the movement in the three directions in relation to the global reference system is illustrated. In Figure 2b the interactions between the vanes and the concrete and water through TIE limitations are indicated. In Figure 2c the water model to which an acoustic boundary condition is applied is presented.

The mesh of the different parts of the model consists of volumetric tetrahedral elements of quadratic and linear interpolation. The concrete and the vane are 3D solid elements and the water is an acoustic element. In total there are 1.2 million elements; 465.815 are quadratic and 728.322 are linear. The parameters used for the different materials that constitute the model were extracted from project plans and operation data from the power plant and are presented in Table 1.
**Table 1:** Physical properties used.

<table>
<thead>
<tr>
<th>Material</th>
<th>Elastic Module (MPa)</th>
<th>Poisson Module</th>
<th>Density (kg/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete H39</td>
<td>29421</td>
<td>0.2</td>
<td>2400</td>
</tr>
<tr>
<td>Steel ASTM A516-60</td>
<td>205000</td>
<td>0.3</td>
<td>7800</td>
</tr>
<tr>
<td>Normalized Water at 20°C</td>
<td>2200</td>
<td></td>
<td>1000</td>
</tr>
</tbody>
</table>

Then, the movement analysis of the nodes was done. From that analysis, the natural frequencies for the first 10 vibration modes are possible to determine considering the contribution of the concrete plus the metallic mesh and the effect of the water where the vane is. The results obtained by means of the finite elements model make it possible to visualize and identify the different vibration modes. In Figure 3, the first three vibration modes are shown.

The finite elements analysis allows visualizing the nodes of the different vibration modes so that in the regions where the nodes of the first dominant modes are, Strain Gages are not used. Because of this, more sensitivity in the measurements done is obtained. The natural frequencies for each vibration mode are presented in Table 2.

**Table 2:** Natural Frequencies for the first 10 vibration modes.

<table>
<thead>
<tr>
<th>Vibration Mode</th>
<th>Natural Frequency (Hz) P4</th>
<th>Natural Frequency (Hz) P12-P13</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30.14</td>
<td>27.17</td>
</tr>
<tr>
<td>2</td>
<td>64.28</td>
<td>57.14</td>
</tr>
<tr>
<td>3</td>
<td>83.14</td>
<td>76.18</td>
</tr>
<tr>
<td>4</td>
<td>141.29</td>
<td>126.01</td>
</tr>
<tr>
<td>5</td>
<td>162.07</td>
<td>150.72</td>
</tr>
<tr>
<td>6</td>
<td>237.85</td>
<td>212.70</td>
</tr>
<tr>
<td>7</td>
<td>257.61</td>
<td>243.92</td>
</tr>
<tr>
<td>8</td>
<td>264.99</td>
<td>248.82</td>
</tr>
<tr>
<td>9</td>
<td>298.83</td>
<td>270.75</td>
</tr>
<tr>
<td>10</td>
<td>354.40</td>
<td>308.03</td>
</tr>
</tbody>
</table>

2.2. Vanes Instrumentation

The positions of the Strain Gages on the vanes are shown in Figure 4. Positions #1 y #2 are maintained in the three vanes.

The strains are related to the stress in the longitudinal axis of the vane due to the solid mechanical laws so that [3]:

\[
\sigma_o = E \epsilon_o
\]  \hspace{1cm} (2.1)

So as to analyze the dynamic responses of the vanes for the different operational modes, an acquisition with a sampling frequency of 2 kHz to follow the Nyquist criteria for the highest natural frequencies was done [2].

The strains and stresses in the longitudinal axis of the vane due to the solid mechanical laws so that [3]:

\[
\sigma_o = E \epsilon_o
\]  \hspace{1cm} (2.1)

Where \( \sigma_o \) is the stress calculated; \( \epsilon_o \) is the strain measured and \( E \) is the Young modulus. In this case we adopted the 205 GPa [4].

2.3. Operational sequence of the generator

The operational conditions in which the vanes strain data was obtained are the following:

a) Start-up to no-load operation;

b) Generator excitation;

c) Power steps (See Table 4);

**Table 3:** Instrumentation used.

<table>
<thead>
<tr>
<th>Equipment</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strain Gage</td>
<td>Vishay LWK-06-W250B-350</td>
</tr>
<tr>
<td>Acquisition board</td>
<td>HBM Somat EBRG 350</td>
</tr>
<tr>
<td>PC</td>
<td>Compaq 610</td>
</tr>
</tbody>
</table>

2.2. Vanes Instrumentation

The positions of the Strain Gages on the vanes are shown in Figure 4. Positions #1 y #2 are maintained in the three vanes.

For the measurements carried out on the vanes of the stay vane, it has been assembled a measurement chain with a strain gages connection in quarter bridge which were connected to an acquisition data board in communication via Ethernet to a storage PC, settled in the turbine head cover. The wiring of the strain gages was settled in a region close to the edge of the vanes and was covered with epoxy so as to soften their geometry. In Table 3, the elements used in the measurement chain are detailed.

**Table 4:** Power steps parameters.

<table>
<thead>
<tr>
<th>Step</th>
<th>Power (MW)</th>
<th>Net Head (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>No-load</td>
<td>23.11</td>
</tr>
<tr>
<td>1</td>
<td>71.5</td>
<td>22.97</td>
</tr>
<tr>
<td>2</td>
<td>95.2</td>
<td>22.94</td>
</tr>
<tr>
<td>3</td>
<td>126.4</td>
<td>22.92</td>
</tr>
<tr>
<td>4</td>
<td>134.2</td>
<td>22.91</td>
</tr>
<tr>
<td>5</td>
<td>138.9</td>
<td>22.90</td>
</tr>
<tr>
<td>6</td>
<td>142.0</td>
<td>22.89</td>
</tr>
<tr>
<td>7</td>
<td>150.1</td>
<td>22.88</td>
</tr>
<tr>
<td>8</td>
<td>153.0</td>
<td>22.87</td>
</tr>
<tr>
<td>9</td>
<td>157.8</td>
<td>22.88</td>
</tr>
</tbody>
</table>
3. RESULTS AND DISCUSSION

In this section, the data obtained from the measurements for the operational conditions specified in Section 2.3 is presented and analyzed. For point a, the transitional condition is analyzed until start-up to synchronous speed is reached. For points b and c, the dynamic responses for the mode changes are analyzed so as to verify the location of the excitation frequencies to compare them to the natural frequencies of the modeled vanes.

3.1. Start-up to no-load operation

In Figure 5, it is possible to identify the change in the measured stress on the vanes of the stay vane from the start of the machine until the synchronous speed. It has been verified that the mechanical stresses for the vanes in the leading edge of the stay vanes reach values that are four times higher than in the trailing edge where the Von Karman turbulences [1] also apply.

The tension peaks, which the vanes are submitted to, have a higher value in the trailing edges.

3.2. Excitation of the generator

First, an analysis is done in the temporal field in which no considerable change is observed in the mean values of the stress present in the field. Then a frequency analysis is done to study the evolution of the components and to compare those frequencies to the natural frequencies obtained in the finite elements analysis.

The analysis on vane P4 didn’t show a coupling between the strain frequencies measured with Strain Gages and the natural frequencies. Vanes P12 and P13 have excitation frequencies which are close to the natural frequencies as can be appreciated in Figure 6.

It is possible to observe a response for the first natural frequency (27.17 Hz) and for the eighth natural frequency (248.82 Hz) as indicated in Figure 6. To analyze the value of the stress obtained, Marin’s equation was applied according Shigley et al. [4] and the value of the ultimate resistance $S_{ut}$ is obtained from the (3.1) equation:

$$S_e = k_a k_b 0.504 S_{ut}$$

In this equation $k_a$ equals 0.7628 and corresponds to the surface factor for a hot-rolled element; $k_b$ equals 1 which represents the load factor for a bending load. The value of the ultimate resistance $S_{ut}$ comes from the steel in question and is 413.68 MPa. Then, the limit of the ultimate resistance is 159 MPa. From this analysis it can be observed that the stress amplitude close to the resonance region for frequencies 2 and 8 reach values under the admissible useful life limits.

3.3. Power steps

The last analysis is done from the progression of mean values and the oscillation amplitude departing from the mean values and standard deviation of each set of data with unchangeable intervals for each power step. Apart from this, a spectral waterfall analysis is done on the stress between the electric power steps where the generator operates.

In Figure 7 the stress measured in points #1 and #2 in vanes P4, P12 y P13 are observed. The stress that P4 stands is clearly marked because of previously mentioned reasons, in position #1 in P4 stress values up to 36MPa are reached, these values are still below the fatigue limit of the steel used in P4 vane. In P12 and P13 the mean stress values diminish in relation to the no-load operation; however, the amplitude increases. From 126 MW the vaues for #1 in P12 experiment an increase in its mean value, which tends to diminish for higher power. In #2 in P12 that corresponds to the trailing edge, it has a tendency to increase up to the maximum mean values of 8 MPa where peaks of 19 MPa are reached. For vane P13 the tendency in the leading edge is to diminish and in the trailing edge the measurements reach 126 MW, where the measurement of the gage was lost.

Finally, in Figure 8, results of the analysis done on vane P4 of the stay vane for different power measures are shown. This vane has been chosen so as to obtain the higher values in the measured stress. The behavior for low frequencies up to 200 Hz is of marked frequencies up to 95.2 MW. These frequencies correspond to the generator turbine unit frequencies, such as the rotation frequency, the frequency considering the path of the turbine blades, the disturbance because of the number of vanes in the wicket gate, etc. In these curves no coupling region is identified with the vane natural frequencies.
4. CONCLUSIONS

In this article, different analysis on the vanes of the stay vane in a Kaplan turbine with 160 MW nominal power have been presented. In the first analysis the results for the starting condition up to the no-load operation are presented. This condition is especially important since the turbine’s wicket gate has a lot of activity here and it is where the higher concentration of acceleration and deceleration activity is found. The results found in this condition showed that the stress values reached do not attain significant values in relation to the structural static limits or fatigue limits found in Shigley et al. (2005).

The second condition analyzed corresponds to the excitation of the electrical generator. Although at this stage the transitional phenomena related to the electrical operation are considerable, these are not shown to be a structural perturbation to the vanes in the stay vane. In vanes P12 and P13 measured frequencies are observed to be close to the natural frequencies previously calculated through finite elements. However, the amplitude of these excitations reaches low values when compared to the fatigue limits in steel used in this kind of structure. Finally, the stress experimented by the measured vanes for the different operational power are shown, verifying that in some cases such as P12 and P13, some stress values are higher for the no-load operation than for operational power around 90 MW. The stress peaks appear eventually with higher operational power. These peaks, as in some the other cases, are below the limits of the mechanical design [4].

To finish, a scan in the frequencies for the different power steps in the U4 are presented. Here, marked frequency peaks associated to the frequencies in the generator turbine unit can be identified and some others where frequency bands are identified which need deeper analysis but can be associated to hydrodynamic phenomena.

5. ACKNOWLEDGMENTS

An special acknowledgement to regulation department from the hydroelectric power plant of Yacyretá, also to National University of Misiones through the Center of Studies of Energies for Development, for the technical support in the development of this work.

6. REFERENCES

ABSTRACT
This paper presents the application of acoustic Doppler current profiler as a tool for adjustment and validation of computational fluid dynamic models in river environment. The two tailraces of the Tucurui hydro power plant present strong differences. The bed of the left tailrace has a high bank and a curvature oriented towards the right. Thus, an initial analysis of the overall dynamics points to the potential existence of local circulation, which can produce adverse pressure in front of some units. In order to validate this hypothesis, the Tucurui hydro power plant tailrace was modeled and field measurements were applied to verify the model behavior. This article will focus on results obtained and comparison between the physical and numerical approach.

KEYWORDS: Doppler flow profiler, hydro power plants, open channel flow, CFD modeling and validation.

1. INTRODUCTION
Tucurui is an 8,370 MW hydro power plant located at the Amazon basin (on the Tocantins River, more than 300 kilometers upstream of the city of Belém) and is connected to the Brazilian power grid. The whole power plant was constructed in two phases, twelve units of 350 MW rated power during the 1980’s and, more recently, eleven units of 375 MW rated power. Figure 1 shows the power plant general view along with its location at the 3°49’S 49°38’W coordinates.

The power substation of Phase I was constructed very close to the left margin of the river, leaving very little space for the construction of the Phase II tailrace. Therefore, such channel had to be constructed with a very particular shape. It is a curved channel with different input and output cross section areas, with intermediate transition.

In order to simulate the plant operation and its tailrace hydraulic performance under several loading conditions and different number of operating units, a Computational Fluid Dynamics (CFD) model was developed. On the other hand, the simulations results could only be accepted if validated from field measurements. A series of acoustic Doppler current profiler (aDcp) measurements was carried out to adjust and validate simulation.

ADcp are based on digital signal processing, Doppler effect and other acoustic propagation properties and are capable to measure the water flow profile over the cross section of a channel, river or in the oceanic currents. Such technology has becoming very popular, being widely applied in studies on deep sea, ocean modeling, estuaries, sediment transport [1-2], and river flow [3-9]. More recently, aDcp have also been applied to hydro power plants analysis [10].

This paper presents the application of aDcp as a tool for adjustment and validation of computational fluid dynamic models. The Tucurui hydro power plant tailrace was modeled and field measurements were applied to verify the model behavior, which allows the simulation of several operating conditions.

2. PROBLEM DESCRIPTION
Tucurui is one of the biggest Brazilian hydro power plants and was constructed in two phases. Phase I included the power plant dam, spillway, substation, and a first power house, with twelve 350 MW Francis generating groups, unit #1 to unit #12, completed in 1984. The main Tucuruí Dam is a concrete-gravity 78 m high and 6.9 km long, impounding a 45 km³ reservoir, with a live volume of 32 km³. The main dam’s Creager spillway has a maximum flow capacity of 110,000 m³/s, controlled by 20 floodgates. The Phase I power house is fitted with an intake and penstocks. Phase II began in June 1998 and was called for the construction of a new powerhouse with eleven 375 MW Francis turbines, unit #13 to unit #23. The new powerhouse is located to the left of the old one. Phase II was completed in 2007. Figure 2 shows the satellite view of the power plant arrangement.

The power substation of Phase I, nevertheless, showed in the upper left corner of Fig. 2, was constructed very close to the left margin of the river, leaving very little space for the construction of the Phase II tailrace. Therefore, such channel had to be constructed with a very particular shape. It is a curved channel with different input and output cross section areas, with intermediate transition. The input section is about 360 m wide and roughly 12.6 m depth average. Its output cross section area is about a third of the input area, 2,650 m², as showed in Figure 3.
The flow pattern in such injector type channel is unique for this kind of application and the study of the flow profile in several working situations is of utmost importance, as it can be the source of problems of different orders. To better understand the flow profile in different working situations, a CFD model was developed. The model and simulations were validated through flow profile measurements using aDcp measurements. The results of measurements and simulations are presented as follows.

3. FIELD MEASUREMENTS

The flow measurement campaign was carefully established in order to conform to the power plant operation. Measurements were taken from an aluminum boat along seventeen transversal paths at the tailrace. In each path, at least two crossings were performed to obtain enough information for statistical analysis of total flow. Water speed and channel bed profile were recorded. Table 1 presents some of the aDcp parameters used to configure the device for the field measurements. Figure 4 (a) and (b) show the field measurements, while Fig. 5 presents the measured transversal paths.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>System Frequency</td>
<td>1228.8 kHz</td>
<td>Transducer</td>
<td>4 Beam Janus’</td>
</tr>
<tr>
<td>1st Bin</td>
<td>-0.53 m</td>
<td>False Target</td>
<td>50 counts</td>
</tr>
<tr>
<td>Bin Size</td>
<td>0.20 m</td>
<td>Band Width</td>
<td>0</td>
</tr>
<tr>
<td>No. Bins</td>
<td>125</td>
<td>Cor. Thres.</td>
<td>64 counts</td>
</tr>
<tr>
<td>Pings/Ens</td>
<td>2</td>
<td>Err Thres.</td>
<td>1500 mm/s</td>
</tr>
<tr>
<td>Sensor Configuration</td>
<td>#1</td>
<td>Blank</td>
<td>0.25 m</td>
</tr>
<tr>
<td>Transducer Head Attached</td>
<td>TRUE</td>
<td>Min Pgood</td>
<td>0</td>
</tr>
<tr>
<td>Orientation</td>
<td>DOWN</td>
<td>Ref Layer</td>
<td>0, 5 first bin, last bin</td>
</tr>
<tr>
<td>Beam Angle</td>
<td>20 Degrees</td>
<td>Mode</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 2 presents average values of the distance of each path from powerhouse wall and its corresponding depth. This information is important to obtain the topography of the channel, to be used in the CFD modeling. Fig.6 shows the obtained depth profile. It should be noticed the existence of a salience in the channel path, which is a dam left in the channel after its construction. This small dam was erected to guarantee dryness during the channel construction, and was expected to be removed afterwards. Its existence would increase the water speed profile at that region for a given flow. Vertical walls should also be noticed close to units #13 and #23. Complete depth profile were employed during further simulations.

<table>
<thead>
<tr>
<th>Section</th>
<th>L (m)</th>
<th>h (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40</td>
<td>18.08</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
<td>17.38</td>
</tr>
<tr>
<td>3</td>
<td>65</td>
<td>16.88</td>
</tr>
<tr>
<td>4</td>
<td>75</td>
<td>16.76</td>
</tr>
<tr>
<td>5</td>
<td>85</td>
<td>16.70</td>
</tr>
<tr>
<td>6</td>
<td>105</td>
<td>16.58</td>
</tr>
</tbody>
</table>
During the measurement campaign, generating units #13, #15, #17, and #19 were operating at approximately full load, while units #14, #16, #18, and #20 to #23, were turned off. However, the depth profile was very close to the expected, the speed profile measurements has showed to have a region of null flux on the channel north direction. A negative flow was observed in the left margin of the channel. Figure 7 presents the flux profile at the first section (a) and at the tenth section (b).

The average speed vectors at different depths range, from 0 to 5 m (a), from 5 to 10 m, (b), from 15 to 20 (c), and from 20 to 25 m (d) are presented in Figure 8.

Figure 9 presents a 3D view of the flow speed vectors in the Tucurui hydro power plant tailrace. It should be noticed the non-conventional ramp in the turbines output. It can be seen that only four generating groups were in operation during the measurements. In this operating condition, the flow is concentrated in the right margin of the tailrace. A vortex can be noticed in the left margin, with a recirculating flow passing very in front of the left turbines output. This observation is very important to guarantee secure turbines operation far from their vibration forbidden limits. With these measurements, it is possible to validate CFD models and simulate different operating conditions.

4. CFD MODELING AND VALIDATION

We have used the Telemac 3D software (www.opentelemac.org) to compute the flow inside the tailrace. Telemac 3D considers mass and movement conservation equations, one state equation, hypothesis of Boussinesq and Newtonian fluid, free surface equation and bottom friction contour conditions. In addition, one strong hypothesis of the modeling is the consideration of the vertical speed by solving the non-hydrostatic Navier-Stokes equations. For doing that, the pressure is broken down into a hydrostatic term and a "dynamic" one as in the following formulation [11].

\[
p = p_h + p_d = \rho g (\eta - z) + \rho_0 g \int_{\eta}^{\eta + \Delta \eta} \frac{\rho_0}{\rho} dz' + p_d
\]  

Where \( \rho, \rho_h, \rho_d \) stands for the total, static and dynamic pressure (N/m²), respectively; \( \rho \) is the density (kg/m³), \( g \) is the gravitational constant (m/s²), \( \eta \) and \( z \) are reference heights, upper and bottom levels (m) respectively, and also the limits of integration, \( dz' \) is the differential of the height.

After solving the hydrostatic step, the gradient of dynamic pressure acts as a correction ensuring the divergence-free velocity. The dynamic pressure is solved by the "projection step".

\[
\text{div} \left[ \frac{1}{\rho} \text{grad}(p_d) \right] = \frac{1}{\Delta t} \text{div} (\vec{U})
\]

Figure 10 shows the tailrace 3D mesh adopted for the computational analysis. The unstructured triangular horizontal mesh is composed of more 4,800 nodes. Fifteen vertical layers results in more than 73,000 nodes, and more than 130,000 prismatic elements.
And (4)

The modeling included the bottom treatment in between rough surface and smooth concrete with a Manning-Strickler coefficient of 50 and time steps of 0.2 s.

The validation of the simulations is made by comparing the calculated speed profiles to the results obtained with the performed measurements. The CFD model was adjusted in order to achieve the highest agreement between measurements and simulations. Figure 11 presents the flux patterns for the four operating units for several depth slices.

A 3D plot of the flow pattern for a depth of 12.5 m is showed in Fig. 12, where a great recirculation can be observed.

There were observed a good agreement between measurements and CFD results. The main flow characteristics are well represented such as the great recirculation observed in the left margin and vertical distribution of the flow. From this comparison, the CFD model was considered validated and operational for study of typical operating conditions. Comparison between velocity values from the CFD and measurements results on three typical sections downstream of the dam and at four different depths, are presented in Tables 3, 4, and 5.

Table 3: Measured and Calculated speeds for Section 2.

<table>
<thead>
<tr>
<th>Depth (m)</th>
<th>0-5</th>
<th>5-10</th>
<th>10-15</th>
<th>15-20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vmeas (m/s)</td>
<td>0.72</td>
<td>0.82</td>
<td>0.76</td>
<td>0.68</td>
</tr>
<tr>
<td>Vcalc (m/s)</td>
<td>0.83</td>
<td>0.81</td>
<td>0.79</td>
<td>0.76</td>
</tr>
<tr>
<td>Error (%)</td>
<td>15.40</td>
<td>-0.29</td>
<td>4.10</td>
<td>11.95</td>
</tr>
</tbody>
</table>

Table 4: Measured and Calculated speeds for Section 5.

<table>
<thead>
<tr>
<th>Depth (m)</th>
<th>0-5</th>
<th>5-10</th>
<th>10-15</th>
<th>15-20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vmeas (m/s)</td>
<td>0.85</td>
<td>0.82</td>
<td>0.78</td>
<td>0.76</td>
</tr>
<tr>
<td>Vcalc (m/s)</td>
<td>0.88</td>
<td>0.82</td>
<td>0.70</td>
<td>0.62</td>
</tr>
<tr>
<td>Error (%)</td>
<td>3.16</td>
<td>0.39</td>
<td>-9.62</td>
<td>-18.97</td>
</tr>
</tbody>
</table>

Table 5: Measured and Calculated speeds for Section 9.

<table>
<thead>
<tr>
<th>Depth (m)</th>
<th>0-5</th>
<th>5-10</th>
<th>10-15</th>
<th>15-20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vmeas (m/s)</td>
<td>0.86</td>
<td>0.81</td>
<td>0.64</td>
<td>0.62</td>
</tr>
<tr>
<td>Vcalc (m/s)</td>
<td>0.83</td>
<td>0.75</td>
<td>0.65</td>
<td>0.57</td>
</tr>
<tr>
<td>Error (%)</td>
<td>-2.83</td>
<td>-7.24</td>
<td>1.84</td>
<td>-7.58</td>
</tr>
</tbody>
</table>

An application of the CFD model was conducted considering units #13 to #23 operating at 630 m³/s each, excepting for units #14 and #22 that were operating at 450 m³/s. Graphical results is presented in Fig. 13.
The analysis of the results shows that the CFD model was able to detect a speed increase close to the salience observed in the middle of the channel. It is also observed a speed increase from units #13 and #23, which are close to the channel walls.

The former case study considered all units operating at 630 m³/s and the following figures present the main results. Figure 14 presents the average flux lines of the output flow of each generating unit. It can be seen the adjustment of surface flow to the small output area, while the deepest flow accommodates itself.

![Figure 14: Average water speed profile along the channel.](image1)

Figure 15 presents a 3D view of the average flux lines for each generating unit. It can be notice that both end units have their flux ejected to the surface due to the channel wall that is close to them. Due to the small output area, one can also observe the flux speed increase.

![Figure 15: Average water speed profile along the channel.](image2)

5. CONCLUSIONS

This work presented an evaluation of the flow profile at the tailrace of the Tucuruí hydro power plant. Whilst an acoustic Doppler current profiler was used to perform measurements, a CFD model was adjusted in order to achieve the highest agreement between measurements and simulations.

The complex geometry of the tailrace with a bend followed by a contraction along the right margin induces a heterogeneous configuration flow at the plant downstream. The differences between input and output cross-section areas, with intermediate transition, results in a very atypical configuration and a particular shape, whose flow profile was not studied yet.

It was noticed that the presence of the vertical walls close to units #13 and #23 leads to a rise in the turbine output flux speed. This one is then faster than those located at the center of the power plant. Outside the zone of walls’ influence, the bottom stream outputs are relatively parallel to each other into the terrain downstream of convergence.

On the surface, the free surface gradient between the left margin and right margin leads surface currents to be diverted to the right. Shear velocities between the bottom and the surface, can be also observed in the left margin, generating rip currents, presenting itself as a source of energy dissipation.

The contraction of the channel and the tide against this at this point push the jet from the unit #23 towards the center of the flow and lead to merge with the flux emitted by the unit #22. In general, the flow is further disrupted by right and left margins. Turbulent hydrodynamic phenomena observed in the left margin are sources of instability, which should be avoided during the power plant operation.

6. ACKNOWLEDGMENTS

The first author would like to thank CNPq, FAPEMIG and INERGE for their financial support in conducting researches.

7. REFERENCES

The electrical insulation system of the generators must be capable to resist the high voltage stress. Therefore, it is necessary to check the operational condition during its operating time. This is possible by monitoring partial discharges (PD) which are electrical discharges of low intensity that occur in a dielectric imperfect area subject to high electric field. A general purpose of this article is to describe evaluation of the insulation system of stator winding in Coaracy Nunes Hydroelectric Plant generating units through an online monitoring system of partial discharges. It was noted during the continuous monitoring that there was not great evolution of PD in normal conditions, but when an increase of generated rated power above 1 p.u. happened the machines had an increase in the PD levels. In addition, it was noticed that increase of PD levels has a strong relation with the increase of vibrations.

**KEYWORDS:** Partial Discharges, Hydrogenerators, Online Monitoring System, Predictive Maintenance.

### 1. INTRODUCTION

INAUGURATED in the 70’s, Coaracy Nunes hydroelectric plant (UHCN) was built on the Araguaia river, city of Ferreira Gomes (Amapá, Brazil), and originally two generating units of 20 MW each were projected. Then this potential was increased with the insertion of one more unit of 30 MW and the repowering of the units of 20 MW to 24 MW each, as a whole 78 MW. The principal electric energy supply components, namely, generators, need to be kept in perfect operational conditions, because they represent the most expensive part of the plant. The insulation system of the generators must be capable to resist the high voltage stress. Therefore it is necessary to check the operational conditions during its useful life.

Examination of 69 incidents [1] showed that the main causes of hydrogenerator failures could be categorized in the next sequence: insulation system failures, mechanical defects, thermic problems and, at last, bearings failures [1]. Figure 1 illustrates this classification.

The failures in the electrical insulation systems were investigated in more details and the results are shown in Figure 2. Among the main causes of these failures only the protection over voltages and coils loosening do not generate PDs, that is, approximately 90% of causes of failures has PD appearing as a symptom.

Partial discharge (PD) is a phenomenon that manifests itself in the insulation system of rotating machines. It can be defined as an electrical discharge of small intensity that occurs in an area of imperfection of a dielectric environment subjected to an electric field. The path formed by the discharge does not connect the two ends of that area in a complete way, and, because there is no complete rupture, it is called “partial discharge” [2]. As part of PDs can be potential sources of defects in the electric insulations, their presence can be a strong indicator of a process that can bring the insulation to total failure.

In the electric power sector the maintenance practices have been changed in the last years. The maintenance methodologies based on time (preventive maintenance) and those that occur after a failure (corrective maintenance) have been replaced by maintenance methodologies based in the equipment conditions (predictive maintenance). This change happened because of impossibility to exclude the machines for a long time in the installations because of the possible financial losses. Thus, the disconnections must be realized in a programmed manner to minimize the losses.

PD monitoring enables the reliability improvement of generators because it can identify the degradation of its insulation system, and therefore, a possibility of repair or replace it before a catastrophic failure in use [3].

Since 2009, our engineers worked to install PDs monitoring system in the UHCN generating units. At the beginning, the first passes were started through sporadic measurements performed together with Research Center of Electric Energy – Cepel of Eletrobras until definitive installation of all PDs monitoring system in all machines in 2013 [4].

UHCN suffered two failures caused by a strange body left in the stator which could have been avoided if there were a PD monitoring system. There was a failure in CNUGH-02 unit (three months and six days of stop machine, source SAP R3 - Eletronorte). The cause
of stator burning was a screw found in the stator winding, see Figure 3. Also, there was a failure in CNUGH-03 (nine months of stop machine to repair, source SAP R3 – Electronorte) because of a screw which was situated in an air guide and damaged the stator insulation. The large time of stop machine is associated with some other factors related to insulation.

The objective of this article is to present the measurement results and evaluation of the dielectric state of the stator windings of Coaracy Nunes hydrogenerators by PDs monitoring.

2. PREDICTIVE MAINTENANCE

The choice of an appropriate maintenance technique of electric energy generation plant must take into account the following factors: the importance of equipment to the productive process, its acquisition cost and its failure rate [5].

The preventive maintenance is based on statistics over average time between failures. The problem is that the average time can not be always assertive resulting in a maintenance for unnecessary repair or a maintenance for catastrophic failure.

Thus, one needs corrective techniques that, according to [6], imply high costs because the unexpected breaks can result in production losses, loss of product quality and high indirect maintenance costs. That is why this type of maintenance is changed gradually for predictive ones, where from parameters analyses, it is possible to interfere before the catastrophic failure.

One can see defects evidences through change of vibration signals, temperature variation, change of PD signals, etc. In [7], predictive maintenance is defined as “any monitoring activity that is capable to provide sufficient data to a tendencies analysis, diagnostic emission and decisionmaking”.

Therefore one can consider PD analysis as a method of predictive maintenance.

3. PARTIAL DISCHARGES

Partial discharges are small electric sparks resulting from electric fatigue of a gas (for instance, air) contained in interior of a cavity. If the cavity is within an organic solid or liquid, the PD will degrade the material and can eventually cause the failure of the electric insulation [3].

Successions of incomplete electric discharges, quick and intermittent occur in a gaseous environment in series with solids insulators or liquids. The PDs occur in defined regions and not complete a close circuit (short circuit). They are considered to be quick because occur in very short periods in comparison with the period of test voltage (1/60 seconds). Figure 4 presents a typical pattern of PD in a stator bar of a generator.

The PDs can produce current pulses, luminescence, electromagnetic waves, acoustic waves, energy consumption, thermic variations, chemical variations, mechanical vibrations. In a broad sense, the detection and measuring techniques can be divided into two groups: electric and non-electric ones.

A. Electric Method of PDs Measurement

In the electric methods, one inserts a detecting and measuring circuit in the circuit where discharges do not occur. The non-electric methods usually serve to support the electric ones.

A particular challenge in PD measuring is found in online measuring of turbogenerator in operation. Once the machine is turned on to the energy system, electrical interferences (noises) are present during most of the time. Noise sources include corona from the power system, collector ring, commutator, ignition, sparks of poor electrical connections, etc. This noise overshadows the PD pulses and can cause erroneous technical conclusions that the stator winding has high levels of PD. The consequence is that a good winding is evaluated incorrectly as being defective, which means that a false alarm is given suggesting that winding has its insulation compromised. Such false alarms reduce the credibility of online PD tests [9]. This problem of possible errors of analysis along time has resulted, in particular, in the decrease of interest of plant maintenance team to continue the measurements [10]. However, there are methods of separation of PD pulses from noise pulses.

After elimination of the noise, the number, magnitude and the phase position of PDs related to the cycle of 50 or 60 Hz of the alternate current are registered in PD measuring system. Figure 5 shows a typical graphic of one phase of a stator winding of a motor [9]. The pulse magnitude is measured in millivolt units (mV). From each test, two integral indicators are extracted representing collected data of PD pulses. The positive and negative peaks of PD magnitudes (+Qm and −Qm) symbolizing the highest measured PD pulses with a minimal rate of PD repetition of 10 pulses per second. Qm is a reasonable predictor of the condition of the stator insulation. A high Qm measured in a winding in comparison with a minor Qm in other windings, commonly implies that the winding is more damaged [9], [10].
The destructive nature of PDs in the insulation of high voltage equipment is known already more than 80 years [11]. Therefore, manufacturers of high voltage cables, transformers, capacitors and commutators, in which the insulation is composed mainly of organic materials (paper, oil, polyethylene, rubber, epoxy and/or polyesters) have to take care when projecting an insulation system. The insulation should not contain empty spaces in regions of electric stress of high voltage [10]. The measuring methods to determine PD has been object of development since the 1930s.

B. PD Types in Rotating Machines [10,12]

The stator coil of a rotating machine presents the PD activities in different conditions. Each location of the PD activity has a specific condition with different materials involved. So the PD activity is specific to each different setting. Figure 6 shows a transverse cut of a stator bar with location of typical defects marked:

a) Intense electric field is in the conductor corner (corona in the coil heads). Depending on manufacturing process, a sharp border can cause PD beginning in an early stage.

b) Delamination of the conductor-insulator is the principal insulation wear. The delamination of the insulator layers of the dielectric system usually happens when a machine is overloaded.

c) Insulation detachment is an inside delamination of principal insulation which is part of the normal process of thermal ageing of mica-epoxy insulation.

d) A region without layers of mica.

e) Discharge in the groove caused by bar vibration inside the groove due to inadequate wedge or partial vibration of the core.

f) Cavities.

C. Patterns Recognition

Figure 7 shows in a schematic way some examples of phase resolved PD patterns. Such patterns can be found usually for defects in the insulation systems of stator winding [12]. The users should be aware that several additional effects, not discussed here, can happen in stator windings which can produce other characteristic patterns of PD.

The measurements of PD are sensitive to some of the external sources of deterioration that affected these stator windings. It was noted [13] that defects found reflect the defined pattern in the standard IEC TS 60034-27-2-2012. It is important emphasized that evaluated stators were from different manufacturers and were installed in different dates.

D. Partial Discharges Measuring System

In Figure 8, a measuring circuit for a coupler installed in one phase is shown. In the low voltage side of each capacitor, a measuring cable is connected that leads the current signal of the PDs up to a box of terminals located in the generator external part. In this box of terminals, there is a resistor of 2000 Ω in parallel to the coaxial cable. From this box of terminals, the cables are launched up to the entrance of the monitoring system mounted in a standard rack.

The instrumentation proposal of the system was developed in a PXI commercial platform obeying the following criteria’s: mounting of modulate hardware in a way that is easier update the system and to involve minor maintenance costs; facility of software development, separating the development of acquisition processes of the storage processes, analysis and diagnostic tools. In the case of Coaracy Nunes power plant it serves for three machines, without a limitation of couplers number that can be installed. Figure 9 shows a schematic diagram of the physical connections made in the generating units CNUGH-01, 02 and 03 of Coaracy Nunes.

The PD measuring is performed in time domain with a digital instrumentation with sampling capacity of up to 100 MS/s. As the environment of the power plant has different sources of electromagnetic noises with different bands of frequency, it was necessary an intermediate conditioning of signals in a way that just
the components of interest would reach the measuring instrument. Actually, this conditioning of signals can be fulfilled in different ways according to the characteristics of each stator winding.

The PDs are current pulses with high repetition rate, i.e. with a very large quantity of pulses during the measuring time, which can be one minute, for instance. Therefore, storing all this information would be memory consuming from the computational point of view. So, it was established the so-called statistic map representative of PDs. Actually, this map has three magnitudes that are jointly stored, namely, amplitudes of discharges (millivolts (mV) or picocoulombs (pC)) – y-axis, voltage angle related to the reference signal of 60 Hz – x-axis, and the number of discharges (repetition) – z-axis represented by the colored graphics.

The processing scheme performed by the system is presented in Figure 10. The analyses and the diagnostics are based on these maps to help in the making decision based on the PD occurrence in the stator windings of a determined machine.

5. RESULTS

The data analysis has been done in the three phases of each generating unit of UHCN during the year 2014. It was possible to verify that in the months of continuous monitoring there was not great PDs evolution, i.e. the machines operated properly. However, it has been noticed that when an increase in the generated power happened, the machines showed a significant increase in PD levels. It has been verified that PDs have relation with the machine operational condition: the PDs accompanied the power and temperature changes in the machine. Also, it was noticed that PDs increase had a strong correlation with increasing of mechanical vibrations. It was possible to observe through curves of tendencies obtained by the monitoring system.

A. Measured Discharges Patterns.

In Figures 12 and 13, one can compare the pattern of each discharge type presented in the standard IEC 60034-27-2 with the patterns found by IMA-DP.

B. Correlation between PD measurements and mechanical vibrations.

This topic demonstrates a case that happened in 2014/06/30 in a generating unit of the UHCN, where the PD phenomenon had correlation of with the Mechanical Vibrations. Figure 14 presents the curve of tendency of the PDs in machine 2, phase A, during 30/06/2014.

4. MONITORING SYSTEM

The monitoring system utilized by UHCN power plant was developed by Research Center of Electric Energy (CEPEL) and Eletrobras Eletronorte in partnership. It is called Instrumentation for Monitoring and Analysis of PDs (IMADP). Figure 11 presents the diagram of this monitoring system which possesses a developed instrumentation in a commercial platform of an industrial computer. The terminal server is a service of remote computation using disseminated terminals by the net. In this system, there is a server computer for centralized processing, where the desktop computers connected to the net function as access terminals. Thus, it is possible to do remote access to the IMADP system by the Eletrobras internal network.

[Figure 11: IMA-DP diagram.]
Finally it can be concluded with this study that PDs magnitudes as well as their number of occurrences and, consequently, their pattern presented a definite correlation with the increase of mechanical vibrations. This fact should be taken into consideration for a better diagnostic of electrical machines.

6. CONCLUSION

This work had as general objective an evaluation of the electric insulation condition of stator windings of the hydrogenerators using the online monitoring system of PDs developed by CEPEL and Eletrobras Eletronorte called Instrumentation for Monitoring and Analysis of PD. We presented the principal results of online monitoring of partial discharges along the year of 2014. It was shown that there is a strong correlation between partial discharges and mechanical vibrations for the given operational conditions of the machine (active power above 1 p.u.).

7. ACKNOLEDGEMENTS

We thank CEPEL by technical support in the installation of the partial discharges monitoring system at UHCN.

8. REFERENCES
