Technical Papers

06 DEVELOPMENT OF PUMPS OPERATING AS TURBINES FOR USE IN DEVELOPING COMMUNITIES
Nascimento Filho, Jair; Losada y González, Manuel; Barreira Martinez, Carlos; Martins Stopa, Marcelo; Lopes, Rafael Emilio; Moncada Balmaceda, María Virginia

12 A STABILIZED FINITE ELEMENT METHOD FOR UNSTEADY POLLUTION DISPERSION IN RIVERS
Melo, Rafael P.; Brasil Junior, Antonio C.P.; Cavalcanti da Cunha, Alan

16 A KALMAN FILTER BASED APPROACH FOR MEASURING SOUND SPEED, AXIAL AND TRANSVERSAL FLOW COMPONENT IN ACOUSTIC TRANSIT TIME (ATT) MEASUREMENTS
Gruber, Peter

22 LEAKAGE ASSESSMENT IN WATER PIPE SYSTEMS: FROM STEADY-STATE TO TRANSIENT FLOW ANALYSIS
Kepler Soares, Alexandre; Covas, Dídia I.C.; Reis, Luísa Fernanda R

27 BEARING TEMPERATURE EFFECT ANALYSIS OF CENTRIFUGE PUMPS OPERATING WITH MODERATE CAVITATION
Schroder, Francisco Carlos; de Faria Lemos de Lucca, Yvone; Dalfré Filho, José Gilberto

34 VON KARMAN VORTEX STUDY IN HYDRAULIC MACHINES STAY VANES THROUGH CFD
Rojido, Martín; Rodríguez, Daniel; Cacciavillani, Juan Carlos

38 NUMERICAL APPROACH TO ASSESS THE HYDROELASTIC COUPLING OF MECHANICAL COMPONENTS FOR HYDRAULIC TURBOMACHINES
Tobón Espinosa, Daniel Felipe; Botero Herrera, Francisco Javier

42 EROSION PREDICTION BASED ON ILES METHOD
Hidalgo, Victor; Luo, Xianwu; Valencia, Esteban; Aguinaga, Alvaro; Cando, Edgar

Technical Notes

45 DOWNSTREAM FISH PASSAGE: THE NEW CHALLENGE OF THE HYDROPOWER SECTOR FOR THE CONSERVATION OF THE BRAZILIAN FISH FAUNA
Pompeu, Paulo S.; Suzuki, Fábio M.; Prado, Ivo G.; Souza, Rafael C.R.
During “25th IAHR Symposium on Hydraulic Machinery and Systems”, worldwide event of IAHR that took place on September 2010 in ROMENIA, a proposal was born for creating a Latin American group with the purpose of increasing discussions on hydraulic machines and systems area under the coordination of International Association for Hydro-Environment, Engineering and Research. In that instance, Prof. Geraldo Lúcio Tiago Filho was appointed as responsible for coordination of structuring the group that now counts with 22 participating companies and institutions, ALSTOM, ANDRITZ, BARDELLA, KSB, VOITH, Brazilian universities: IME, UFMG, UFMT, UFRJ, Unb, UNICAMP, UNIFEI, USP following universities UNAM, UNCOMA, UNCU, UNLP and UTN-Regional Mendoza (Argentina), EPN(Ecuador), IMFIA (Uruguay), UMSNH (Mexico) and USM (Chile).

The LAWG-IAHR group was created on September 2011 to fit demands of researchers and specialists in Latin America, in disclosure of knowledge on hydraulic machines, associated components and systems area.

IAHR, founded on 1935, is a worldwide and independent organization of engineers and specialists acting on areas related to hydraulics, environment and its applications. Activities range from basic hydraulics applied to rivers and tides to water resources development through the use of computer hydraulics tools and regular training. IAHR stimulates and promotes research and its application, and thus contributes for sustainable development, optimization of water resources management and industrial processes of worldwide effluents. IAHR reaches its purposes through a range of activities including groups of work, research performing, events, short term trainings, journals publication, monographs and papers. These activities count with engagement of international programs as UNESCO, WMO, IDNDR, GWP, ICSU and cooperation organizations related to water subject worldwide.

IAHR publishes two international scientific magazines from its headquarters in Madrid, Spain, in partnership with Taylor and Francis - Journal of Hydraulic Research and Journal of River Basin Management. The International Journal of Hydro-Environment Research (JHER) is published by Asia Division of IAHR in partnership with Water Resources Korean Association and Elsevier. IAHR publishes semimonthly “Hydrolink” Magazine and monthly “NewFlash”, newsletter targeted to international hydraulic community.

For disclosure of Latin American group scientific production, we have launched with this edition the American Journal of Hydropower, Water and Environment Systems, with trimestral periodicity and our goal being to fulfill existing gap for publications in Latin America. Thus, we hope to count on a much more expressive participation of researchers with the purpose to strengthen scientific initiation of American countries. Special thanks for our supporting members and Prof. Augusto N.C. Viana for their contribution for feasibility of this publication.

Geraldo Lúcio Tiago Filho
Editor in Chief
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-IAHR

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
- initial references followed by last name of editors (if any), followed by the 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 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 solidus (/) 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:
Technical Papers

06 DEVELOPMENT OF PUMPS OPERATING AS TURBINES FOR USE IN DEVELOPING COMMUNITIES
Nascimento Filho, Jair; Losada y González, Manuel; Barreira Martinez, Carlos; Martins Stopa, Marcelo; Lopes, Rafael Emilio; Moncada Balmaceda, María Virginia

12 A STABILIZED FINITE ELEMENT METHOD FOR UNSTEADY POLLUTION DISPERSION IN RIVERS
Melo, Rafael P.; Brasil Junior, Antonio C.P.; Cavalcanti da Cunha, Alan

16 A KALMAN FILTER BASED APPROACH FOR MEASURING SOUND SPEED, AXIAL AND TRANSVERSAL FLOW COMPONENT IN ACOUSTIC TRANSIT TIME (ATT) MEASUREMENTS
Gruber, Peter

22 LEAKAGE ASSESSMENT IN WATER PIPE SYSTEMS: FROM STEADY-STATE TO TRANSIENT FLOW ANALYSIS
Kepler Soares, Alexandre; Covas, Didia I.C.; Reis, Luisa Fernanda R

27 BEARING TEMPERATURE EFFECT ANALYSIS OF CENTRIFUGE PUMPS OPERATING WITH MODERATE CAVITATION
Schröder, Francisco Carlos; de Faria Lemos de Lucca, Yvone; Dalfré Filho, José Gilberto

34 VON KARMAN VORTEX STUDY IN HYDRAULIC MACHINES STAY VANES THROUGH CFD
Rojido, Martín; Rodríguez, Daniel; Cacciavillani, Juan Carlos

38 NUMERICAL APPROACH TO ASSESS THE HYDROELASTIC COUPLING OF MECHANICAL COMPONENTS FOR HYDRAULIC TURBOMACHINES
Tobon Espinosa, Daniel Felipe; Botero Herrera, Francisco Javier

42 EROSION PREDICTION BASED ON ILES METHOD
Hidalgo, Victor; Luo, Xianwu; Valencia, Esteban; Aguinaga, Alvaro; Candia, Edgar

Technical Notes

45 DOWNSTREAM FISH PASSAGE: THE NEW CHALLENGE OF THE HYDROPOWER SECTOR FOR THE CONSERVATION OF THE BRAZILIAN FISH FAUNA
Pompeu, Paulo S.; Suzuki, Fábio M.; Prado, Ivo G.; Souza, Rafael C.R.
DEVELOPMENT OF PUMPS OPERATING AS TURBINES FOR USE IN DEVELOPING COMMUNITIES

1Nascimento Filho, Jair; 2Losada y González, Manuel; 3Barreira Martinez, Carlos; 4Martins Stopa, Marcelo; 5Lopes, Rafael Emilio; 6Moncada Balmaceda, María Virginia

ABSTRACT

In this paper, the results of a research project that aims to provide a hydro – electro – mechanical equipment for the application of small loads, typical of developing communities and sustainable manner, are presented. In this case we refer to as a centrifugal pump running as turbine (PAT), which is coupled to an electric generator. The paper shows the manufacturing processes of the pump and the electromechanical assembly system. It also presents the conversion equations pump to operate as turbine and system performance results obtained from the tests conducted in the laboratory.

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

1. INTRODUCTION

The current dependency of fossil fuels for power generation is especially adverse in third world developing communities. The origin of fossil fuel sources is mostly exogenous for local productive resources at the expense of economy. In addition, it is necessary to purchase associated equipment for generation chain and consequently this represents an unknown technology for population. In spite of the internal combustion engines as alternative solution is interesting, it does not contribute to raise these communities to a compatible technology level with the benefit (electricity) generated. This can be observed in riverside communities at Brazilian Amazon Rivers where population completely distant from current productive resources, counts on typical equipment of an advanced society such as plasma television, mobile telephones, etc. Those equipment in spite of easing life of people, represent more and more a strong dependency on exogenous technologies, turning these communities more dependent on other people and culture. In other hand, these have as commerce elements the products produced by land, such as forest and mining products. This process constitutes a classic case of neo-colonialism that in spite of whole world’s efforts still persists in the third world.

Considering this adverse reality as reference context, a project focused in diffusion of a technology for power generation appropriate for communities’ development started fifteen years ago. The purposes of this project were: a) to provide an alternative for power generation, necessary for basic activities as lighting and low charges for isolated communities (or not) of developing countries; b) to provide a technology able to be assimilated by local population aiming to start a technology development process compatible with current global challenges; c) to allow this technology to enter in local social fabric underlying basis of a process that may lead to establish small technology centers (small workshops); d) to raise local technology level in order to reduce the difference between these communities and the rest of developed world. We made the choice to start the process from the primary source of hydropower. This choice is based in the fact that this source is available in the most part of Latin America. It also pursues a feasible technology to be used by communities with low level of development and that may be easily assimilated allowing that small workshops could manufacture such equipment. For this reason, the research is focused in centrifugal pumps running as turbines (PAT).

Centrifugal pumps have been established as a feasible technologic alternative since nineteenth century, though his origin is centuries before, Lima [1]. Former studies showing pumps running as turbines are from the 60’s with Kittredge [2], Stepanoff [3]. However, its use as reversible machine has unknown origin. Works performed by Viana [4], Chapallaz [5] and Williams [6] present different ways of selecting a pump for running as a turbine. The PAT selection method developed by Viana [4] consists in defining the head and the commercial pumps flow that will run as turbines (PAT) by the use of coefficients obtained by the author through tests on the basis of the work performed by Kittredge [2] and Buse [7] related to P'T rotation. The methodology proposed by Chapallaz [5] is similar to Viana’s [4] as it is also based on determination of pump rotation. Works performed by Williams [6] were already based in determination of pump using flow and available head. But in all cases, the study of pumps running as turbines (PAT) follows an action line based in its potential for optimization of small potentials. Lopes [8] presented a study case with efficient applications and commercially competitive in the range of applications from 5 kW to 150 kW. In all these situations, pumps used were commercial and available in the market.

This study presents calculation procedures of a centrifugal pump, steps for manufacturing, test running as pump, calculation of conversion for running as turbine and theoretical and experimental results.

2. RAPID CENTRIFUGAL PUMP WITH BLADES OF SIMPLE CURVATURE

The design of a centrifugal pump can be developed quickly with help of set of traditional and known equations in literature,
Macintyre [9]. The flow and head must be considered. In addition to this data, constructive data entering in the process as conditioners of project must be considered as well. The radial component of velocity is estimated by Macintyre method [9], with help of charts built according specific velocity. In this case under consideration, the radial component of velocity is determined according to constructive data such as rotor diameter and width of blades [10]. Determination of speed triangles in entry and exit of channel between blades requires definition of rotor geometric data. Therefore, the procedure for rotor calculation is made by definition of speed triangles. It is a test and error method based in determination of speeds in order to avoid excessive speeds [10].

For manufacturing a pump housing, a dual start solution was chosen to ease the access into it. Thus it was possible to extract eventual imperfections on surface and to finish the entrance rings. In spite of these facilities, the solution in two has as disadvantage the difficulty in building the connection. The bearings are sphere bearings and are stored in an aluminium housing screwed to the housing diametrically opposed to the pump suction tubing. The use of screws makes easy the set assembling and allows maintenance operations more quickly. The engine is coupled to the pump through a flexible gasket mould of volute casing in wood and epoxy which later was melted in aluminium.

Figure 1a shows geometry and dimensions of volute casing. Figure 1b shows a view of melted casing.

The blades of the pump were manufactured by forging of metal and later welded to the rotor disk. Superior and inferior rotor disks were built in carbon steel.

Figure 2a shows a plant view of superior and inferior disks and of steel plates forged to form the blades. Figure 2b shows a view of inferior plate.

After being forged, blades were welded to superior and inferior disks. It was decided to make a complete welding (all extension) in inferior disk as it is wider. The superior disk (bearing) was fixed only in the plate end. This made the welding process easier since inevitably one of the faces had to be welded in a quite compromised space.

The driving shaft was built in carbon steel and has in one end a flexible gasket for coupling to the engine. Figure 3a shows a detail of blades welding to the disk. Figure 3b shows a detail of casing set (two parts) and two suction and discharge connections ready to be fixed to housing by bolts. Five bolts for coupling bearings to housing can also be observed.

3. PUMP RUNNING AS TURBINE

Besides the conventional use of PAP (pump operating as a pump), one appliance of a centrifugal pump is its use as PAT. This kind of alternative has been studied by many researchers that identify several advantages and disadvantages under this kind of operation. Thus, we can mention as advantages the fact that pumps were standard equipment with manufacturing costs and low maintenance [8]. In other hand, simplicity and ease of maintenance of this equipment allow people with low technical background to work with PAT [12]. As disadvantage, we have difficulty in load control and a low performance compared to conventional turbines [8].

According to Sharma [13], PAT should be chosen considering "H" available head and available flow of "Q" source. Available
head and flow of site shall be named as HPAT and QPAT respectively. Available head and flow in function PAP shall be named respectively as HPAP and QPAP. Sharma [13] suggested to express the correlation of available head and flow through equations 1 and 2, as pair H x Q of pump.

\[ Q_{PAT} = \frac{Q_{PAP}}{\eta_{max}^{0.8}} \]  

\[ H_{PAT} = \frac{H_{PAP}}{\eta_{max}^{2}} \]  

To verify operating of pump as turbine, it is necessary to have the curves of pump running as pump in chosen rotation. The rotation of 1200 rpm was selected for running of PAT coupled to a synchronic generator and to 1250 rpm for an appliance with induction generators. To perform tests, a structure was installed as showed in Figures 4 and 5, where a pumping station furnished necessary pressure and flow.

![Figure 4: Representative Scheme of test bench.](image)

![Figure 5: View of test bench.](image)

### 4. SIMULATION OF PUMP RUNNING AS PUMP TO THE ROTATION OF PUMP OPERATING AS TURBINE

In used method, it is necessary to obtain experimental data on characteristic curves of pump operating as pump PAP (Figure 6, Tables 1 and 2, Source: [11]).

In Table 1, pe is the pressure in pump entry, ps is the exit pressure, Q is the flow and Pel is the electric power, obtained under test at nominal speed of 1.750 rpm. With the help of motor efficiency chart, the power in shaft is obtained.

**[Table 1]: Test at 1750 rpm – PAP - Data collection (Source: [11]).**

<table>
<thead>
<tr>
<th>Operation Point</th>
<th>pe (kgf/cm²)</th>
<th>ps (kgf/cm²)</th>
<th>Q (m³/h)</th>
<th>Pel (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0,12</td>
<td>0,95</td>
<td>0,00</td>
<td>1392</td>
</tr>
<tr>
<td>2</td>
<td>0,10</td>
<td>0,93</td>
<td>5,00</td>
<td>1563</td>
</tr>
<tr>
<td>3</td>
<td>0,08</td>
<td>0,92</td>
<td>9,96</td>
<td>1730</td>
</tr>
<tr>
<td>4</td>
<td>0,06</td>
<td>0,90</td>
<td>20,78</td>
<td>2081</td>
</tr>
<tr>
<td>5</td>
<td>0,04</td>
<td>0,86</td>
<td>30,53</td>
<td>2288</td>
</tr>
<tr>
<td>6</td>
<td>0,01</td>
<td>0,81</td>
<td>39,93</td>
<td>2528</td>
</tr>
<tr>
<td>7</td>
<td>-0,03</td>
<td>0,74</td>
<td>50,62</td>
<td>2835</td>
</tr>
<tr>
<td>8</td>
<td>-0,08</td>
<td>0,65</td>
<td>60,43</td>
<td>3052</td>
</tr>
<tr>
<td>9</td>
<td>-0,10</td>
<td>0,61</td>
<td>63,64</td>
<td>3127</td>
</tr>
</tbody>
</table>

In Table 2 are presented results of characteristic curves of pump to a rotation of 1.750 rpm, where H is head, N is power in motor shaft, \( \eta_{el} \) is performance of engine and \( \eta \) is performance of pump.

**[Table 2]: Results of tests at 1750 rpm – PAP (Source: [11]).**

<table>
<thead>
<tr>
<th>Point</th>
<th>Q (m³/h)</th>
<th>H (m)</th>
<th>P_e (kgf/cm²)</th>
<th>P_s (kgf/cm²)</th>
<th>Q (m³/h)</th>
<th>Pel (W)</th>
<th>( \eta_{el} ) (%)</th>
<th>( \eta ) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0,0</td>
<td>8,63</td>
<td>1392,0</td>
<td>1141,4</td>
<td>82</td>
<td>0,0</td>
<td>9,2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>5,0</td>
<td>8,64</td>
<td>1563,5</td>
<td>1282,1</td>
<td>82</td>
<td>9,2</td>
<td>16,7</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10,0</td>
<td>8,73</td>
<td>1730,7</td>
<td>1419,1</td>
<td>82</td>
<td>16,7</td>
<td>28,8</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>20,8</td>
<td>8,71</td>
<td>2081,9</td>
<td>1707,2</td>
<td>82</td>
<td>28,8</td>
<td>37,3</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>30,5</td>
<td>8,53</td>
<td>2288,3</td>
<td>1899,3</td>
<td>83</td>
<td>37,3</td>
<td>47,2</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>39,9</td>
<td>8,37</td>
<td>2528,2</td>
<td>2098,4</td>
<td>83</td>
<td>43,3</td>
<td>49,2</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>50,6</td>
<td>8,08</td>
<td>2835,9</td>
<td>2353,7</td>
<td>83</td>
<td>47,2</td>
<td>49,8</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>60,5</td>
<td>7,59</td>
<td>3052,4</td>
<td>2533,5</td>
<td>83</td>
<td>49,2</td>
<td>49,8</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>63,6</td>
<td>7,47</td>
<td>3127,8</td>
<td>2596,1</td>
<td>83</td>
<td>49,8</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

With help of Rateau equations and from experimental results of PAP running in normal conditions of 1.750 rpm, characteristics curves of PAP can be obtained, relative to rotations of 1.250 rpm and 1.200 rpm (Tables 3 and 4, figures 7, 8, 9 and 10).

**[Table 3]: Prediction of performance at 1.250 rpm – PAP.**

<table>
<thead>
<tr>
<th>Point</th>
<th>Q (m³/h)</th>
<th>H (m)</th>
<th>P_e (kgf/cm²)</th>
<th>P_s (kgf/cm²)</th>
<th>N (W)</th>
<th>( \eta_{el} ) (%)</th>
<th>( \eta ) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0,0</td>
<td>4,40</td>
<td>415</td>
<td>0,0</td>
<td>466</td>
<td>9,2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3,6</td>
<td>4,40</td>
<td>466</td>
<td>9,2</td>
<td>516</td>
<td>16,7</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7,1</td>
<td>4,45</td>
<td>621</td>
<td>16,7</td>
<td>691</td>
<td>28,8</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>14,8</td>
<td>4,44</td>
<td>691</td>
<td>37,3</td>
<td>764</td>
<td>43,3</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>21,8</td>
<td>4,35</td>
<td>857</td>
<td>47,2</td>
<td>922</td>
<td>49,2</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>28,5</td>
<td>4,26</td>
<td>945</td>
<td>49,8</td>
<td>387</td>
<td>49,8</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>36,1</td>
<td>4,12</td>
<td>3,81</td>
<td>49,8</td>
<td>3,81</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>43,2</td>
<td>3,87</td>
<td>922</td>
<td>49,2</td>
<td>3,81</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>45,4</td>
<td>3,81</td>
<td>945</td>
<td>49,8</td>
<td>3,81</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Table 4**: Prediction of performance at 1.200 rpm – PAP.

<table>
<thead>
<tr>
<th>Point</th>
<th>Q (m³/h)</th>
<th>H (m)</th>
<th>N (W)</th>
<th>η (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0,0</td>
<td>4,05</td>
<td>366</td>
<td>0,0</td>
</tr>
<tr>
<td>2</td>
<td>3,4</td>
<td>4,06</td>
<td>411</td>
<td>5,46</td>
</tr>
<tr>
<td>3</td>
<td>6,8</td>
<td>4,10</td>
<td>455</td>
<td>17,535</td>
</tr>
<tr>
<td>4</td>
<td>14,2</td>
<td>4,10</td>
<td>548</td>
<td>30,135</td>
</tr>
<tr>
<td>5</td>
<td>27,3</td>
<td>3,94</td>
<td>673</td>
<td>45,36</td>
</tr>
<tr>
<td>6</td>
<td>34,7</td>
<td>3,80</td>
<td>755</td>
<td>49,56</td>
</tr>
<tr>
<td>7</td>
<td>41,4</td>
<td>3,57</td>
<td>813</td>
<td>51,45</td>
</tr>
<tr>
<td>8</td>
<td>43,5</td>
<td>3,51</td>
<td>833</td>
<td>52,185</td>
</tr>
<tr>
<td>9</td>
<td>45,4</td>
<td>3,81</td>
<td>945</td>
<td>49,8</td>
</tr>
</tbody>
</table>

An important aspect is to determine characteristics of pump running as pump at the point with higher performance. Figures 9 and 10 show behaviour of efficiency according to Q flow, where point of higher efficiency occurs for 52% with flow of 43,5 m³/h for a rotation of 1.200 rpm.

### 5. RESULTS FOR PUMP RUNNING AS TURBINE

Tests in PAT were performed in rotations of 1.200 rpm and 1.250 rpm (Tables 5 and 6).

**Table 5**: Test at 1200 rpm – PAT coupled to a synchrongenerator - Data collection.

<table>
<thead>
<tr>
<th>pe (kgf/cm²)</th>
<th>ps (kgf/cm²)</th>
<th>Q (m³/h)</th>
<th>V (V)</th>
<th>I (A)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0,62</td>
<td>0,02</td>
<td>103,68</td>
<td>90,6</td>
<td>9,11</td>
</tr>
<tr>
<td>0,56</td>
<td>0,02</td>
<td>99,72</td>
<td>84,3</td>
<td>8,48</td>
</tr>
<tr>
<td>0,53</td>
<td>0,01</td>
<td>95,4</td>
<td>78,3</td>
<td>7,87</td>
</tr>
<tr>
<td>0,48</td>
<td>0,01</td>
<td>91,8</td>
<td>71,1</td>
<td>7,19</td>
</tr>
<tr>
<td>0,41</td>
<td>-0,02</td>
<td>83,16</td>
<td>57</td>
<td>5,75</td>
</tr>
<tr>
<td>0,37</td>
<td>0,01</td>
<td>78,48</td>
<td>49,9</td>
<td>5,04</td>
</tr>
<tr>
<td>0,37</td>
<td>0,02</td>
<td>74,88</td>
<td>41,5</td>
<td>4,19</td>
</tr>
<tr>
<td>0,36</td>
<td>0,04</td>
<td>70,92</td>
<td>32,9</td>
<td>3,32</td>
</tr>
<tr>
<td>0,36</td>
<td>0,04</td>
<td>67,32</td>
<td>23,4</td>
<td>2,37</td>
</tr>
<tr>
<td>0,36</td>
<td>0,06</td>
<td>65,16</td>
<td>13,9</td>
<td>1,4</td>
</tr>
<tr>
<td>0,36</td>
<td>0,05</td>
<td>64,08</td>
<td>0,0</td>
<td>0,0</td>
</tr>
</tbody>
</table>

**Table 6**: Test at 1250 rpm – PAT coupled to a synchrongenerator - Data collection.

<table>
<thead>
<tr>
<th>pe (kgf/cm²)</th>
<th>ps (kgf/cm²)</th>
<th>Q (m³/h)</th>
<th>V (V)</th>
<th>I (A)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0,66</td>
<td>0,02</td>
<td>106,2</td>
<td>93</td>
<td>9,33</td>
</tr>
<tr>
<td>0,64</td>
<td>0,02</td>
<td>103,68</td>
<td>90</td>
<td>8,99</td>
</tr>
<tr>
<td>0,6</td>
<td>0,02</td>
<td>101,16</td>
<td>85</td>
<td>8,59</td>
</tr>
<tr>
<td>0,58</td>
<td>0,02</td>
<td>99,72</td>
<td>83</td>
<td>8,36</td>
</tr>
<tr>
<td>0,57</td>
<td>0,02</td>
<td>97,56</td>
<td>80</td>
<td>8,08</td>
</tr>
<tr>
<td>0,52</td>
<td>0,01</td>
<td>92,88</td>
<td>73</td>
<td>7,37</td>
</tr>
<tr>
<td>0,5</td>
<td>0,02</td>
<td>89,64</td>
<td>67,6</td>
<td>6,77</td>
</tr>
<tr>
<td>0,45</td>
<td>0,01</td>
<td>85,32</td>
<td>59,5</td>
<td>6,02</td>
</tr>
<tr>
<td>0,42</td>
<td>0,03</td>
<td>80,64</td>
<td>51,1</td>
<td>5,15</td>
</tr>
<tr>
<td>0,4</td>
<td>0,04</td>
<td>77,04</td>
<td>42,7</td>
<td>4,33</td>
</tr>
</tbody>
</table>
[Table 6]

<table>
<thead>
<tr>
<th>$p_e$ (kgf/cm²)</th>
<th>$p_s$ (kgf/cm²)</th>
<th>$Q$ (m³/h)</th>
<th>$V$ (V)</th>
<th>$I$ (A)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0,39</td>
<td>0,05</td>
<td>72,72</td>
<td>33,2</td>
<td>3,35</td>
</tr>
<tr>
<td>0,4</td>
<td>0,05</td>
<td>69,12</td>
<td>23,9</td>
<td>2,4</td>
</tr>
<tr>
<td>0,4</td>
<td>0,06</td>
<td>67,68</td>
<td>14,4</td>
<td>1,45</td>
</tr>
<tr>
<td>0,4</td>
<td>0,06</td>
<td>65,52</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Numerical results are presented in tables 7 and 8, respectively. Graphical results are shown in Figures 11, 12, 13 and 14.

[Table 7]: Test at 1200 rpm – PAT coupled to a synchronic generator - Data collection.

<table>
<thead>
<tr>
<th>$H$ (m)</th>
<th>$Q$ (m³/h)</th>
<th>$P$ (W)</th>
<th>$N$ (W)</th>
<th>$\eta$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6,2</td>
<td>103,68</td>
<td>1785,6</td>
<td>1100,49</td>
<td>61,63</td>
</tr>
<tr>
<td>5,6</td>
<td>99,72</td>
<td>1551,2</td>
<td>953,15</td>
<td>61,45</td>
</tr>
<tr>
<td>5,3</td>
<td>95,4</td>
<td>1404,5</td>
<td>821,63</td>
<td>58,50</td>
</tr>
<tr>
<td>4,8</td>
<td>91,8</td>
<td>1224,0</td>
<td>681,61</td>
<td>55,69</td>
</tr>
<tr>
<td>4,1</td>
<td>83,16</td>
<td>947,1</td>
<td>437,00</td>
<td>46,14</td>
</tr>
<tr>
<td>3,7</td>
<td>78,48</td>
<td>806,6</td>
<td>335,33</td>
<td>41,57</td>
</tr>
<tr>
<td>3,7</td>
<td>74,88</td>
<td>769,6</td>
<td>231,85</td>
<td>30,13</td>
</tr>
<tr>
<td>3,6</td>
<td>70,92</td>
<td>709,2</td>
<td>145,64</td>
<td>20,54</td>
</tr>
<tr>
<td>3,6</td>
<td>67,32</td>
<td>673,2</td>
<td>73,94</td>
<td>10,98</td>
</tr>
<tr>
<td>3,6</td>
<td>65,16</td>
<td>651,6</td>
<td>25,95</td>
<td>3,98</td>
</tr>
<tr>
<td>3,6</td>
<td>64,08</td>
<td>640,8</td>
<td>0,00</td>
<td>0,00</td>
</tr>
</tbody>
</table>

[Table 8]: Results of test at 1250 rpm PAT.

<table>
<thead>
<tr>
<th>$H$ (m)</th>
<th>$Q$ (m³/h)</th>
<th>$P$ (W)</th>
<th>$N$ (W)</th>
<th>$\eta$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6,6</td>
<td>106,2</td>
<td>1947,0</td>
<td>1156,92</td>
<td>59,42</td>
</tr>
<tr>
<td>6,4</td>
<td>103,68</td>
<td>1843,2</td>
<td>1078,80</td>
<td>58,53</td>
</tr>
<tr>
<td>6,0</td>
<td>101,16</td>
<td>1686,0</td>
<td>973,53</td>
<td>57,74</td>
</tr>
<tr>
<td>5,8</td>
<td>99,72</td>
<td>1606,6</td>
<td>925,178</td>
<td>57,59</td>
</tr>
<tr>
<td>5,7</td>
<td>97,56</td>
<td>1544,7</td>
<td>861,87</td>
<td>55,80</td>
</tr>
<tr>
<td>5,2</td>
<td>92,88</td>
<td>1341,6</td>
<td>717,35</td>
<td>53,47</td>
</tr>
<tr>
<td>5,0</td>
<td>89,64</td>
<td>1245,0</td>
<td>610,20</td>
<td>49,01</td>
</tr>
<tr>
<td>4,5</td>
<td>85,32</td>
<td>1066,5</td>
<td>477,59</td>
<td>44,78</td>
</tr>
<tr>
<td>4,2</td>
<td>80,64</td>
<td>940,8</td>
<td>350,89</td>
<td>37,30</td>
</tr>
<tr>
<td>4,0</td>
<td>77,04</td>
<td>856,0</td>
<td>246,52</td>
<td>28,80</td>
</tr>
<tr>
<td>3,9</td>
<td>72,72</td>
<td>787,8</td>
<td>148,29</td>
<td>18,82</td>
</tr>
<tr>
<td>4,0</td>
<td>69,12</td>
<td>768,0</td>
<td>76,48</td>
<td>9,96</td>
</tr>
<tr>
<td>4,0</td>
<td>67,68</td>
<td>752,0</td>
<td>27,84</td>
<td>3,70</td>
</tr>
<tr>
<td>4,0</td>
<td>65,52</td>
<td>728,0</td>
<td>0,00</td>
<td>0,00</td>
</tr>
</tbody>
</table>

As expected, the pump running as turbine obtained a superior performance than when running as pump, reaching a value of 62% much superior to performance as PAP, for the same rotation of 52%.
Besides, head as well as flow had maximum values to operate as turbine and regarding operation as pump in point of major performance as shows Table 9. The pump showed a satisfactory operation being able to provide more than 1.000 watts of net power in its shaft.

[Table 9]: Comparative table between PAT and PAP in point of higher performance.

<table>
<thead>
<tr>
<th>Operation Type</th>
<th>H (m)</th>
<th>Q (m³/s)</th>
<th>η (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PAP</td>
<td>3,51</td>
<td>0,0124</td>
<td>52,18</td>
</tr>
<tr>
<td>PAT</td>
<td>6,20</td>
<td>0,0288</td>
<td>61,63</td>
</tr>
<tr>
<td>Results at 1.200 rpm</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PAP</td>
<td>3,81</td>
<td>0,0130</td>
<td>49,84</td>
</tr>
<tr>
<td>PAT</td>
<td>6,60</td>
<td>0,0295</td>
<td>59,42</td>
</tr>
<tr>
<td>Results at 1.250 rpm</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

6. CONCLUSION

Results reached enable to confirm that it is perfectly possible to raise a centrifugal pump with acceptable performances (~52%). It is also possible to check that using existing equations in described literature, an operation point of this pump running as pump can be identified and that in this mode reasonable performance values are reached (~62%). Construction technology of centrifugal pump is simple and can be incorporated in remote communities of development centers, with relative ease.

7. ACKNOWLEDGEMENTS

Authors express their special thanks to FAPEMIG and CNPq (306722/2011-9) for financing support for achievement of this paper.

8. REFERENCES

A STABILIZED FINITE ELEMENT METHOD FOR UNSTEADY POLLUTION DISPERSION IN RIVERS

1Melo, Rafael P., 2Brasil Junior, Antonio C.P., 3Cavalcanti da Cunha, Alan

ABSTRACT

In this work a stabilized finite element method is proposed for the simulation of unsteady pollution in rivers. The transport of the concentration of pollution constituents is modeled using a set of one dimensional and transient advection-diffusion-reaction equations. The flow rate and boundary conditions can vary in time, and all constituents can interact among each other through dynamical kinetics. The proposed numerical method was validated using the analytical test cases associated to the advection of a pollution front considering transient boundary conditions. The DO-BOD model was implemented and the numerical results for a real river pollution problem were analyzed comparing it to the experimental data.

KEYWORDS: unsteady river pollution; Finite Element Method; DO-BOD model.

1. INTRODUCTION

The dynamic characteristic of the river pollution is a fundamental aspect in many environmental problems related to the dispersion of chemical or organic compounds. In many situations, the flow rate of a river varies in time as consequence of natural tidal or runoff effects or due to operation of dam water releases. The transient behavior of the river flow influences the concentrations of chemical or biological compounds. The unsteady flow changes the rate of concentration decay after its releasing from the source positions. It can compromise the environmental health of the river, taking into account the upper limit of an admissible maximum compound concentration, for any time or any position in the stream. Simple steady state analysis based on time-averaged values is inappropriate to describe this process.

The unsteady modeling of pollution in rivers is integrated in many modern simulation tools (EDP-RIV1 from USA-EPA or MIKE from DHI for instance). For this kind of problem, fast and robust numerical solution schemes have to be implemented in order to provide stable results, without introduction of excessive numerical diffusivity.

The dynamic modeling of contaminants dispersion has been explored in many works concerning the implementation of numerical schemes (e.g. Borah and Bera [1]). Concerning simulation methods, Fisher et al. [5] had highlighted a group of criteria for the numerical simulation of unsteady contaminant transport in rivers. The absence of oscillations in the solution (stable schemes), small truncation errors and low distortion of short-wavelength Fourier components can be considered as the main properties required for the numerical schemes. For very complex problems of water pollution, the set of inter-dependent transport equations allows a very strong non-linear behavior. Proper numerical methods for time and space discretization schemes are the base to reach stable and realistic results. Taken it into account the development of stable and accurate algorithms to obtain concentrations fields in time and space have been proposed in the literature of applied numerical methods for water resources (e.g. Iranpour et al. [9], Romanowicz et al. [12]).

The aim of the present work is to present an alternative finite element methodology for unsteady problem related to the formulation of a set of advection-diffusion-reaction equations. The proposed method is based on a stabilized formulation of finite element method that achieves adequate numerical properties for implemented algorithms through high-order time discretization.

2. MATERIALS AND METHODS

2.1 Mathematical Modeling

In the present paper, unsteady flow in an open channel (or river) is considered. A reference axis x along the stream is assumed and the flow carries a set of species \( i = 1, N \), diluted in water, at a concentration \( C_i(x, t) \). The concentrations of the species are distributed through the river \( \Omega = \{ x \mid x \in [0, L] \} \), where the transport problem is modeled by a set of unsteady advection-diffusion-reaction equations given by:

\[
\frac{\partial(A C_i)}{\partial t} + \frac{\partial(Q C_i)}{\partial x} = \frac{\partial}{\partial x}
\left(D_i \frac{\partial C_i}{\partial x} \right) + A \sigma_i C_i + A S_i \quad ; \quad i = 1, N
\]

where \( A(x) \) denotes the area of the river cross-section and \( Q(t) \) is the water flow rate. The distribution of those variables for a river is obtained from the database of its bathymetry and hydrology. The variables \( D_i, \sigma_i \) and \( S_i \) denote respectively the longitudinal dispersion coefficient, the reaction and source terms, related to the specie \( i \). This general formulation can solve a great number of pollution problems in rivers. Detailed aspects of chemical or biological pollution dynamics can be taken into account for each constituent equation by means of the modeling of diffusion and source terms (e.g. Chapra [2], Deng et al. [3]).

Considering here a focus on organic pollution problems, a slightly modified DO-BOD model can be considered. This simple model for organic pollution in rivers takes into account only two species: dissolved oxygen (DO) and biochemical oxygen demand (BOD). The index 1 will be used for the oxygen concentration and 2 for the concentration of BOD. In the model employed here the source and reaction terms for the transport equations are given by:

For \( C_1 \geq 0.1 \text{ mg/l} \) (Aerobic condition)

\[
\begin{align*}
\sigma_1 &= -k_1 \quad ; \quad S_1 &= -k_1 C_2 + k_2 C_3 \\
\sigma_2 &= 0 \quad ; \quad S_2 &= 0
\end{align*}
\]

For \( C_1 < 0.1 \text{ mg/l} \) (Anoxic condition)

\[
\begin{align*}
\sigma_1 &= -k_1 \quad ; \quad S_1 &= -k_1 C_2 + k_2 C_3 \\
\sigma_2 &= 0 \quad ; \quad S_2 &= -k_2 C_3
\end{align*}
\]

\( k_1, k_2 \) and \( k_3 \) are the kinetic parameters.
where the coefficient \( k_i \) is the rate of oxidation of the BOD and \( k_r \) is the re-aeration rate coefficient. \( C_i \) is the value of saturation of dissolved oxygen in water.

To complete the mathematical formulation, a set of initial and boundary conditions for the concentration fields have to be prescribed. The following equations consider an initial distribution of the concentrations in the entire domain, a prescribed transient condition for the species at \( x = 0 \) and an homogeneous Neumann condition for \( x = L \):

\[
C_i = C_{i,0}(x); \quad \forall x \in \Omega
\]  
(4)

\[
C_i(0,t) = \overline{C}_i \left( 1 - \chi_i \sin \left( \frac{2\pi}{\tau} t \right) \right); \quad \frac{\partial C_i}{\partial x} \bigg|_{x=L} = 0
\]  
(5)

In those equation \( \overline{C}_i \) and \( \chi_i \) are the average and the amplitude of the species concentrations. \( \tau \) is the period of the oscillation of the pollution charge at the river.

The presented model can reproduce a real pollution problem related to a transient organic charge at a source point. This transient behavior, induced by the boundary condition at \( x = 0 \) and in some occasions by the hydrodynamics of the river, will require the use of proper discretization techniques in space and time.

### 2.2 Numerical Time Integration

The transport equation (1) is firstly discretized in time using a second order scheme for time derivative. Considering that the concentration fields at the time \( t-\Delta t \), \( t \) and \( t+\Delta t \) are denoted by the superscripts \( n-1, n \) and \( n+1 \) respectively, the conservation equation (1) can be re-written for a time \( t+\Delta t \) using an implicitly scheme as:

\[
\tau_0 C^{n+1}_i + \tau_1 C^n_i + \tau_2 C^{n+1}_i = \frac{\Delta t}{\Delta x} \left( D_i A \frac{\partial C^n_i}{\partial x} + A_i \delta C^n_i + A_S \right) \bigg|^{n+1} \quad ; \quad i = 1, N
\]  
(6)

This scheme has a truncation error of \( O(\Delta t^2) \) if:

\[
\tau_0 = \frac{3\Delta t}{2\Delta x} ; \quad \tau_1 = -\frac{\Delta t}{2\Delta x} ; \quad \tau_2 = \frac{\Delta t}{2\Delta x}
\]  
(7)

Using this discretization scheme, the governing equation for a time step \( n+1 \) can be expressed by the following general advective-diffusive-reactive boundary value problem:

For this scheme the stability parameter is used as a proposed by Franca and Farhat [6] (Unusual Stabilized Finite Element Method).

\[
\tau_h = \frac{h}{2 |Q^{n+1}|} \xi(Pe_h)
\]  
(19)

### 2.3 Finite Element Discretization

In this section a space discretization of the problem (8) has been outlined. Firstly, the weak formulation corresponding to (8) can be obtained classically by the use of weight-residual method. An integral problem can be expressed by:

\[
(a_i C^{n+1}_i,w) + \left( Q^{n+1} \frac{\partial C^{n+1}_i}{\partial x}, w \right) + \left( \nu_i \frac{\partial C^{n+1}_i}{\partial x}, \frac{\partial w}{\partial x} \right) = (f_i,w) ; \quad i = 1, N
\]  
(12)

In the equation (12) \( w \in H^1_{0h}(\Omega) \) denotes the test function (defined in the Sobolev functional space) and \( (u,v) \int_\Omega u v \, dx \) an integral.

A discrete approximation of the weak formulation (12) can be constructed using the standard Galerkin method considering the subspace \( V_{n,h} \subset H^1_{0h}(\Omega) \). By this way, the problem is reduced to:

\[
B_i(C^{n+1}_i,w_h) = F_i(w_h) ; \quad \forall w_h \in V_{n,h} ; \quad i = 1, N
\]  
(13)

where

\[
B_i(C^{n+1}_i,w_h) \equiv (a_i C^{n+1}_i,w_h) + \left( Q^{n+1} \frac{\partial C^{n+1}_i}{\partial x}, w_h \right) + \left( \nu_i \frac{\partial C^{n+1}_i}{\partial x}, \frac{\partial w_h}{\partial x} \right)
\]  
(14)

and

\[
F_i(w_h) = (f_i,w_h)
\]  
(15)

The analysis of the standard Galerkin discretization applied to the advection-diffusion-reaction problem, equations (13)-(15), allows a well-known result concerning the stability limits for the local Peclet number \( (Pe) = |Q_h|/2\nu \) (e.g Donea and Huerta [4]). For convection dominated flows (high \( Pe \)) spurious numerical oscillations may occur degenerating the numerical results. A stabilization method has to be employed to obtain non-contaminating results for any time step or mesh size.

In order to stabilize the convection dominated problem, many techniques have been proposed including an extra term in function of the residual of equation (8). The stabilized method using here is based in the proposal of Franca and Farhat [6], and it was recently revisited to Henao et al. [8] to the advection-diffusive-reactive equation. This formulation is proposed as:

\[
B_i(C^{n+1}_i,w_h) = F_i(w_h) ; \quad i = 1, N
\]  
(16)

where

\[
B_i(C^{n+1}_i,w_h) \equiv (a_i C^{n+1}_i,w_h) + \left( Q^{n+1} \frac{\partial C^{n+1}_i}{\partial x}, w_h \right) + \left( \nu_i \frac{\partial C^{n+1}_i}{\partial x}, \frac{\partial w_h}{\partial x} \right) - \sum e \left( a_i C^{n+1}_i + Q^{n+1} \frac{\partial C^{n+1}_i}{\partial x} - \tau_h a_i w_h - Q^{n+1} \frac{\partial w_h}{\partial x} \right)
\]  
(17)

and

\[
F_i(w_h) = (f_i,w_h) - \sum e \left( f_i \tau_h a_i w_h - Q^{n+1} \frac{\partial w_h}{\partial x} \right)
\]  
(18)

For this scheme the stability parameter is used as a proposed by Franca and Farhat [6] (Unusual Stabilized Finite Element Method).
with a Peclet number redefined as

$$P_{e_h} \equiv \frac{2a|\alpha|}{\sigma h^2 + v_1}$$  \hspace{1cm} (20)$$

where h is the length of 1-D element and $\xi(P_{e_h})$ where $P_{e_h} \in [0, 1]$ and $\xi(P_{e_h}) = \begin{cases} P_{e_h} & 0 \leq P_{e_h} < 1 \\ 1 & P_{e_h} \geq 1 \end{cases}$ \hspace{1cm} (21)

The proposed formulation of the discrete problem (16)-(21) has sufficient regularity and stability as reported in Franca and Valentin [7]. It was tested for many 2D numerical experiments and the authors had deeply evaluated an error and stability by analysis.

### 2.4 Numerical implementation

An unsteady 1-D finite element code was implemented in MATLAB computational environment. River characteristics related to the flow rate, cross-flow area, initial and boundary conditions parameters (equations 4 and 5) are taken from GIS, bathymetry and hydrological database. A simple mesh generation was implemented for 1D stream coordinate, which can generate uniform or stretched grids. The numerical scheme is unconditionally stable and time-step controls the solution truncation error. Typical values for elements lengths around 100 meters and time-steps of 0.1-1s are used. The finite element matrix issued for discrete problem (16)-(18) is constructed using simple linear elements and it is storage as diagonal vectors. The linear system for each time step is solved by Thomas algorithm, considering the tri-diagonal structure of the matrix. Three numerical experiments were considered to validate the code and the numerical methodologies used to stabilize the discrete problem.

### 3. RESULTS AND DISCUSSION

#### 3.1 Test case 1: Single reaction specie

This section presents a comparison between the results obtained with the numerical method developed in last section and an analytical solution for the case of single reacting specie, based on the traveling of a concentration peak expressed by a spatial Gaussian distribution. Twelve different conditions were simulated combining three diffusivity coefficients and two decay coefficients and two initial conditions. Manson et al. [10] studied the same problem using another numerical method based in finite difference approach. Analytical solution for (1) with $N=1$, $\sigma=\kappa$ and $S=0$ is possible for the case of a single reacting specie with a sudden release of mass M in a steady flow and uniform channel area. Equation (22) presents the transient concentrations solution, $C(x,t)$ obtained if the boundary conditions are given by the same equation for $x=0$. A constant stream velocity $U=Q/A$ is considered and the analytical solution is given by:

$$C(x,t) = \frac{M}{A\sqrt{4\pi Dt}} \exp\left(-\frac{x-Ut}{4Dt}\right) \exp(-kt)$$  \hspace{1cm} (22)

The solution above is Gaussian in space, traveling with the convection velocity U. It has a variance given by $\sigma^2 = 2Dt$. The channel velocity is constant and equal 1 m/s in a cross-flow area with $10^5 m^2$. Three diffusion coefficients were used in this test case: 1, 5 and $100^2 m^2/s$. These coefficients are in accord with those present in real rivers. The decay coefficients used were equal to 0 and 0.693/day. The numerical method is started with the distribution given by equation (22) with two initial variances $37636 m^2$ and $180000 m^2$. The numerical domain used had $120000 m$ in length and was divided in 1200 elements with $100m$ each one, the same condition used in Manson et al. (2000). The time step is 0.1 s and the simulation time is equal 14400 s.

Results in Figure 1 compare the numerical and analytical results. These figures show the remarkable agreement obtained for this case and an estimative for the error dimension is only possible when the error in peak percentage is compared (Table 1). It can be observed that no spurious oscillations were verified or any negative undershooting variations. This is important because the reactive terms in more complex models can produce a stiff behavior of the set of equations that can be unstable during the time integration of the problem. Small and spurious negative values of concentration can be amplified, considering the nonlinearity of some water quality models, contaminating all solution. The stable behavior of the proposed methodology in this tested case shows a good condition for more complex situations.

![Figure 1: Gaussian plume traveling.](image-url)
3.2 Test case 2: Multiple interacting species (DO-BOD model)

In 1979, an accident in a factory in New-Zeland led to a large amount of milk discharged in the Waipa River. This accident was properly monitored in order to control the environmental risk over the local ecosystems and to observe the water quality indicators. This situation became a good test case for validation of modeling and numerical methods. In this numerical experiment, the milk was used as a water tracer transported in the river where water quality parameters where measured for three fixed times along the river for DO and BOD. The data collected in this accident have been used in some numerical papers as a benchmark problem (e.g. Manson et al. [10], McBride and Rutherford [11]). The experimental data provided by McBride and Rutherford [11] include spatial profile for BOD and DO along the river for three times 11 hours apart. In order to validate the numerical algorithm the data obtained for the second time was used as initial condition and results obtained numerically for the last time was compared with the measured data. The same velocity and depth profiles reported in the experiments were used in the present work (equations (23) and (24)), with x given in kilometers.

\[
U(x) = \begin{cases} 
0.08 + 0.002(49.4 - x); & \text{for } x \leq 19.4 \text{ km} \\
0.4 + 0.018(49.4 - x); & \text{for } x > 19.4 \text{ km} 
\end{cases} 
\]  
\[h = 6.0 - 0.08(49.4 - x)\]  
\[k_1 = 1.0 \, \text{day}^{-1}; \quad k_2 = 3.74\sqrt{u/h^3}\]  

The re-aeration rate \(k^2\) was formulated according to the surface renewal model of O’Connor-Dobbins. The diffusion coefficient \(D\) was estimated as 10 \(\text{m}^2/\text{s}\) based on the same bibliographic references.

Figure 2 presents the experimental data for BOD and DO used as initial condition for the numerical problem. Notice that in this real case the release of the tracer had occurred periodically on time, with three different peaks (as shown in the figure). The first and second milk release reduced the oxygen concentration in the river to an anoxic condition. After those peaks the re-oxygenation of the BOD and aeration processes in the stream tended towards a recovery of oxygen level, but this was inhibited by the third release of tracer (lower than the other peaks related to the previous releases). The oxygen concentration shot down another time and in some kilometers it reached again its normal levels. This specific dynamical behavior is useful for validation of numerical codes of unsteady pollution problems. The model and its numerical implementation have to recover the reported dynamical characteristics of the BOD and DO concentration along the river.

Figure 3 presents the comparison between the numerical and experimental data, with particular interest in the changes in concentration between the second and the third peaks. Considering all the errors associated to the field measurement and all the assumptions made for estimates of \(k_2\), \(k_{\gamma}\), and the approximations for \(U\) and \(A\), the results can be considered satisfactory. No spurious behavior of numerical solutions was observed and the results for DO and BOD transport evolution presented realistic levels. The concentrations were properly amplified and attenuated after 11 hours of stream transport and the levels of DO were correctly predicted. The levels of BOD had a satisfactory agreement between the field and numerical results.

4. CONCLUSIONS

The prosed algorithms allow good estimates of concentration field for river pollution problem varying in time and space. The second order time discretization promotes lower levels of numerical errors observed after the comparison of the numerical results with the analytical propagation of a Gaussian pulse. The stabilization scheme was able to reduce the numerical stabilities and to obtain good results for real situations. The proposed methodology can be considered as a stable, accurate and fast to simulate unsteady dispersion problems in rivers and water channels.

5. REFERENCES


American Journal of Hydropower, Water and Environment Systems, August 2015
A Kalman filter based approach for measuring sound speed, axial and transversal flow component in acoustic transit time (ATT) measurements

1Gruber, Peter

ABSTRACT

The ATT method for measuring the flow in hydropower applications is an accurate and well established method. The method uses normally a high number of transit time measurements in up- and downstream direction, from which the axial and transversal flow components are determined. In order to obtain a sufficient accuracy, the obtained transit times and transit time differences have to be filtered and averaged. Typical ping rates of a pair of up- and downstream measurement for one path is 50 to 100Hz. In the case that a transverse component of the flow is present, crossed paths are used in one layer, reducing therefore the rate of determining the axial and transverse flow by a factor of two. The conventional determination of the axial velocity of one path relies on the assumption that the speed of sound does not change during the measurement of the up- and downstream measurement process. If this assumption is violated, the conventional method treats this change as a measurement error for the transit times. In the following a Kalman filter based approach is chosen, which allows to distinguish between system noise and measurement noise. The system noise corresponds to the variations of the speed of sound, the axial and the transversal flow component of the flow, while the measurement noise reflects the inaccuracies introduced by determining the transit and transit time differences. The chosen system model is the so called random walk model for the three states speed of sound, the axial and the transversal flow component of the flow and four output quantities for the up- and downstream transit times for two crossed paths. The Kalman filter approach allows to weigh the variation of each state separately and to relate the magnitude of each of the noise sources to the measurement noise. Different situations of estimating the states in a noisy environment are examined with the help of simulation: stepwise change in speed of sound, axial and transverse flow. The influence of the choice of the weighting of the noise sources are investigated, showing the trade-off of noise rejection and signal tracking capabilities of the different filter parameterizations.

KEYWORDS: filter, paramerization, velocity

1. INTRODUCTION

The widely used acoustic transit time (ATT) method for discharge measurement determines the flow \( Q \) by a weighted sum over a number \( N \) of averaged axial path velocities \( \overline{v_{ax}} \) (equation (1)), which can be obtained from the measured transit times of the acoustic pulses along the paths.

\[
Q = k \frac{D}{2} \sum_{i=1}^{N} W_i \overline{v_{ax}}(z_i) b(z_i)
\]  

The positions (heights) \( z_i \) (or \( d_i \)) and weights \( W_i \) of the paths are determined by the integration method used, \( b(z) \) is the width of the conduit at position \( z \), \( D \) the diameter of the conduit and \( k \) a possible geometrical correction factor. Fig. 1 illustrates the velocity components which play a role. The three dimensional velocity \( v(s) \) at a position \( s \) along the path \( A \) can be split in a vertical component which does not contribute to the path velocity and in two horizontal velocities \( v_{ax}(s) \) and \( v_{tr}(s) \) which build the layer velocity \( v_{layer}(s) \). If the layer velocity is projected on the path, the transverse component leads to an erroneous velocity contribution \( v_{c}(s) \). In order to eliminate this contribution a second crossed path \( B \) is added at the same height. Assuming the same averaged transversal component on each of the two crossed paths, the influence of the transversal component to the averaged path velocities can be eliminated.

Equation (2) and (3) give the relation between projected path velocity, speed of sound \( c \) and transit time \( t \):

\[
L = \frac{L}{c + \sqrt{v_{proj}}}
\]

\[
\sqrt{v_{proj}} = \frac{1}{L} \int_{0}^{L} v_{proj}(s) ds
\]

The actual speed of sound is also varying along the path to a minor degree, so \( c \) corresponds to an average speed along a path. If a crossed two path arrangement is chosen, four different transit times (two in up- and downstream direction) can be determined (equations (4)).
\[ t_{d1} = \frac{L}{c + \frac{\nu_a}{\cos \phi + \frac{\nu_v}{\sin \phi}}} \]

\[ t_{u1} = \frac{L}{c - \frac{\nu_a}{\cos \phi - \frac{\nu_v}{\sin \phi}}} \]

\[ t_{d2} = \frac{L}{c + \frac{\nu_a}{\cos \phi + \frac{\nu_v}{\sin \phi}}} \]

\[ t_{u2} = \frac{L}{c - \frac{\nu_a}{\cos \phi + \frac{\nu_v}{\sin \phi}}} \]  

This quadruple of transit times is repeated up to 50 to 100 times per second, depending on the ping rate of the system. Assuming constant speed of sound and averaged velocities during the measuring process of the four transit times, the averaged axial velocity and the speed of sound are found as:

\[ \bar{v}_{ax,1} = \frac{L}{2 \cos \phi} \left( \frac{1}{t_{d1}} - \frac{1}{t_{u1}} \right) - \frac{\nu_v}{\tan \phi} \]  

(5a, b)

\[ \bar{v}_{ax,2} = \frac{L}{2 \cos \phi} \left( \frac{1}{t_{d2}} - \frac{1}{t_{u2}} \right) + \frac{\nu_v}{\tan \phi} \]

\[ \bar{v}_{ax} = \frac{\bar{v}_{ax,1} + \bar{v}_{ax,2}}{2} \]

\[ c = \frac{L}{4} \left[ \left( \frac{1}{t_{d1}} + \frac{1}{t_{u1}} \right) + \left( \frac{1}{t_{d2}} + \frac{1}{t_{u2}} \right) \right] \]  

(6a, b)

Commercial measurement techniques apply the equations (5) and (6) for the evaluation of the flow and the speed of sound. If consecutive measurements are logged without being filtered, a noisy behaviour of the instantaneous velocities can be observed, which can amount to a magnitude of up to +/- 20% of the long term average. Therefore averaging of the transit time and differences of the transit time must be applied in order to stabilize the flow measurement. Typically time constants of 1 sec or more are used. It is an open question from where these variations in time measurement do arise. Is it due to underlying physical processes of turbulence or is it due to the acquisition of the signal and the determination of the times from the recorded signals?

2. TIME-VARYING MODEL FOR VELOCITIES

The determination of the velocities as given by equations (4) assumes a noise free environment and constant velocity components during the measurement process. In many ultrasonic devices the measurement process is done sequentially and not at the same time instant \( k \). A possible time varying measurement process could be formulated as follows:

\[ t_{d1}(k) = \frac{L}{c + \frac{\nu_a}{\cos \phi + \frac{\nu_v}{\sin \phi}}} + n_1(k) \]

\[ t_{u1}(k) = \frac{L}{c_1(k + 1) - \frac{\nu_a}{\cos \phi - \frac{\nu_v}{\sin \phi}}} + n_2(k) \]

\[ t_{d2}(k) = \frac{L}{c_2(k) + \frac{\nu_a}{\cos \phi - \frac{\nu_v}{\sin \phi}}} + n_3(k) \]

\[ t_{u2}(k) = \frac{L}{c_2(k + 1) - \frac{\nu_a}{\cos \phi + \frac{\nu_v}{\sin \phi}}} + n_4(k) \]

The model of equations (7) takes into account that the two acoustic paths are averaging varying local path velocities. There are six time dependent averaged velocities and four independent noise sources involved (equations (8)):

- Velocities: \( c_1(k), \bar{v}_{ax,1}(k), \bar{v}_{ax,2}(k), c_2(k), \bar{v}_{ax,1}(k), \bar{v}_{ax,2}(k) \)
  
- Noise sources: \( n_1(k), n_2(k), n_3(k), n_4(k) \)

With the abbreviations (equations (9))

\[ x(k) = \begin{bmatrix} c_1(k) \\ \bar{v}_{ax,1}(k) \\ \bar{v}_{ax,2}(k) \end{bmatrix} \]  

\[ y(k) = \begin{bmatrix} n_1(k) \\ n_2(k) \\ n_3(k) \\ n_4(k) \end{bmatrix} \]

\[ z(k) \]  

the measurement process can be described in a compact form:

\[ x(k + 1) = f(x(k)) + \Sigma(k) \]

\[ y(k) = g(x(k)) + n(k) \]

(10a-10b)

The nonlinear vector function \( f \) (equation (10a)) is given by the expressions (7a-7d), while the unknown vector function \( g \) (equation (10b)) describes the time evolution of the averaged velocity components.

The problem with this model is the definition of \( f \). This definition must contain the effect of local turbulences on the vector components and is of a random nature. At the moment no such model was yet applied but if suitable random models are available, it certainly would be interesting to use these in this context.

3. RECURSIVE ESTIMATION OF VELOCITIES VIA KALMAN FILTER

Lanzensdorfer [1] proposed a simplified model (equation (11)) for which a least square solution can be found easily. It is assumed that

\[ x = \begin{bmatrix} c_1(k) \\ \bar{v}_{ax,1}(k) \\ \bar{v}_{ax,2}(k) \end{bmatrix} = \begin{bmatrix} c \\ \bar{v}_v \\ \bar{v}_v \end{bmatrix} \]

(11)

and additionally that the inverse of the transit times are defined as outputs. If the inverse of the transit times are defined
as outputs, the measurement equations (12) are linear with respect to the three unknown but constant states $c$, $\bar{v}_a$, and $\bar{v}_r$.

\[
\begin{align*}
z_1(k) &= \frac{1}{t_{d1}(k)} = \frac{1}{L} \left[ c + \bar{v}_a \cos \varphi + \bar{v}_r \sin \varphi \right] + \varepsilon_1(k) \\
z_2(k) &= \frac{1}{t_{u1}(k)} = \frac{1}{L} \left[ c - \bar{v}_a \cos \varphi - \bar{v}_r \sin \varphi \right] + \varepsilon_2(k) \\
z_3(k) &= \frac{1}{t_{d2}(k)} = \frac{1}{L} \left[ c + \bar{v}_a \cos \varphi - \bar{v}_r \sin \varphi \right] + \varepsilon_3(k) \\
z_4(k) &= \frac{1}{t_{u2}(k)} = \frac{1}{L} \left[ c - \bar{v}_a \cos \varphi + \bar{v}_r \sin \varphi \right] + \varepsilon_4(k)
\end{align*}
\] (12)

The new measurement noise sources are rescaled quantities from the original noise sources under the assumption that the noise contribution is small. Equations (12) can be put into matrix form (equations (13a):

\[
\tilde{z}(k) = H \tilde{x} + \tilde{e}(k)
\]

\[
H = \begin{bmatrix}
h_1^T \\ h_2^T \\ h_3^T \\ h_4^T
\end{bmatrix} = \begin{bmatrix}
1/L \cos \varphi / L & \sin \varphi / L \\
1/L - \cos \varphi / L & -\sin \varphi / L \\
1/L \cos \varphi / L & -\sin \varphi / L \\
1/L - \cos \varphi / L & \sin \varphi / L
\end{bmatrix}
\] (13a-13b)

and a linear problem least square problem for the unknown $\tilde{X}$ in equation (13a) can be formulated (Lanzensdorfer [1]). For $N$ consecutive measurements the minimization of the performance index

\[
J = \sum_{k=1}^{N} \sum_{i=1}^{M} (\varepsilon_i(k))^2 = \sum_{k=1}^{N} \sum_{i=1}^{M} (\tilde{z}_i(k) - h_i^T \tilde{x}(k))^2 = \min
\]

leads to the normal equations. The solution for $\tilde{X}$ of equation (14) can be found by direct inversion of a block of measurements or it can also be solved by an iterative procedure via a recursive least square method. By introducing a forgetting factor $\lambda$, it is possible to weigh new measurements more than older ones. This weighting prevents the recursive methods from getting insensitive to changes in the $\tilde{X}$ (velocities) values. In flow measurement applications changes in these values are a fact. So either the block length or the forgetting factor must be chosen adequately in order to obtain satisfying results.

Here another approach is followed. A dynamic model of 3rd order and constant coefficients is assumed for the time evolvement of the velocities, see Fig. 2.

The states of this model are driven by the system noise $w(k)$ (equation (15a)).

\[
\tilde{x}(k+1) = A \tilde{x}(k) + B w(k)
\]

\[
\tilde{z}(k) = H \tilde{x}(k) + \tilde{e}(k)
\] (15a-15b)

The measurement equation for $\tilde{z}(k)$ (equation 15b) is the system output $\tilde{z}(k) = H \tilde{x}(k)$ corrupted by the measurement noise $\tilde{e}(k)$. The dynamic model used is a random walk model (Gruber, Tödtli [2]) for the unknown but time varying velocities:

\[
c(k+1) = c(k) + w_1(k)
\]

\[
\bar{v}_a(k+1) = \bar{v}_a(k) + w_2(k)
\]

\[
\bar{v}_r(k+1) = \bar{v}_r(k) + w_3(k)
\]

\[
(16a-16c)
\]

Equations (16) can be written in matrix form (equations (15)) with

\[
A = \begin{bmatrix}
1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1
\end{bmatrix} \quad B = \begin{bmatrix}
1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1
\end{bmatrix}
\]

With the statistical assumptions of equations (18), the random walk model for a state has the statistical properties that the mean is constant and the variance is growing linear with time (equations (19)).

\[
E\{\tilde{z}(k)\} = E\{\tilde{e}(k)\} = 0 \quad E\{z(0)\} = \bar{z}_0
\]

\[
\text{cov}[\tilde{z}(j), \tilde{z}(k)] = \sum_{i=1}^{r} \begin{bmatrix}
q_j & 0 \\ 0 & q_j
\end{bmatrix} \text{cov}[\tilde{z}(j), \tilde{z}(k)] = \sum_{i=1}^{r} \begin{bmatrix}
0 & r_j \\ 0 & 0
\end{bmatrix}
\]

\[
\text{cov}[\tilde{z}(j), \tilde{z}(k)] = \text{cov}[\tilde{z}(0), \tilde{z}(k)] = \text{cov}[\tilde{z}(0), \tilde{z}(0)] = 0 \quad \text{var}[\tilde{z}(0)] = P(0)
\]

\[
E\{z(k)\} = \bar{z}_0 \quad \text{var}[\tilde{z}(k)] = P(0) + kQ + R
\] (19a-19b)

With the formulation from equations (13), (15), (17), and (18) a standard Kalman filter can be applied to the dynamic system (Sage, Melsa [3]). The Kalman filter is a filter that tries to estimate the unknown state of a system by minimizing the error covariance between the true state and an estimate of the state:

\[
P(k) = \text{var}[\tilde{z}(k) - \tilde{z}(k)]
\]

(20)

The filter consists of two parts: a first part with a one step ahead prediction $\tilde{z}(k)$ of the state estimate $\tilde{z}(k)$ and a one step ahead prediction $P(k)$ of the error covariance matrix $P(k)$ of equation (20). In a second part the predicted value is updated and corrected with the weighted (with the Kalman
gain) difference between the new measurement \(z(k+1)\) and the output prediction without measurement noise \(H\hat{x}(k)=HA\hat{x}(k)\).

If the model is time invariant, the steady state Kalman gain matrix can be computed off-line:

\[
P(0) = P_0 \quad k = 0,1,2,\ldots,\infty
\]

\[
P'(k+1) = AP(k)A^T + Q
\]

\[
K(k+1) = P'(k+1)HA^{-1}
\]

\[
P(k+1) = P'(k+1) - K(k+1)A^T + Q
\]

(21a-21d)

If the fixed point of the difference equations of equations (21) is reached \((k \to \infty)\), can be obtained. \(K\) is in this case only a function of \(A, H, Q\) and \(R\). \(K\) can then be used for the update equation of the state estimate corresponding to the velocities:

\[
\hat{x}(0) = x_0 \quad k = 0,1,2,\ldots,\infty
\]

\[
\hat{x}(k+1) = (I-KH)A\hat{x}(k) + Kz(k+1)
\]

(22)

Figure 3 shows the block diagram of the filter with the real measurements \(z(k)\). Equation (22) allows estimate the speed of sound of water \(c(k)\), the axial and the transversal component \(v_{ax}(k)\) and \(v_{tr}(k)\) of the flow velocity.

4. SET-UP FOR FILTER PARAMETRIZATION STUDY

The benefit of the Kalman filter compared to other recursive estimation methods is that it is possible to influence the temporal behavior of the filter in two ways:

1) The ratio of the covariance matrices of the system noise \(Q\) over the measurement noise \(R\). If \(Q/R\) is much larger than 1, the measurement noise is much less weighted than the system noise, that means the filter tries to follow the measured output in a fast way. Each variation in the measurement is considered to stem from a change in \(z(k)\). If \(Q/R\) is much lower than 1, the system noise is much less weighted than the measurement noise, that means all uncertainties in \(z(k)\) arises from measuring the transit times. In this case the filter acts as a low pass filter trying to eliminate the measurement noise. With the ratio \(Q/R\) it is for instance possible to tune the filter such that a predefined settling time for step change in the velocities can be achieved. The higher the ratio \(Q/R\) is, the more confidence in the measured value is assumed, that means the changes in the measured quantities \(z(k)\) are assumed to be cause by changes in the system states (velocities).

2) As the system noise consist of 3 noise sources, one for each velocity, it is possible to weigh them individually. That makes sense in this application because changes in speed of sound, axial and transversal components are not the same. This possibility enhances the flexibility of the filter.

We consider a situation with typical values as given in Fig. 4a-4d. In order to make the temporal behavior of the filter visible, step changes for the speed of sound at \(k=1000\), the axial velocity at \(k=2000\) and the transversal velocity at \(k=3000\) are applied sequentially and the whole run is simulated during 4000 time steps.

Fig. 5a -5d displays the ideal and noisy velocity components and the noisy transit time measurements due to velocity noise and measurement noise. The scaling of the horizontal axes is such that a time period without step change in the velocities is shown. The scaling of the vertical axes is different for each graph.

![Figure 3: Steady state Kalman filter implementation.]

![Figure 4: Simulated process, a: axial flow velocity in [m/s], b: transversal velocity in [m/s], c: speed of sound in [m/s], d: measured transit time (td1) in [s].]

![Figure 5: Examples of noisy velocity components (all in [m/s]) and transit time measurement in [s].]
5. EXAMPLES OF PARAMETRIZATION

The Kalman filter solution is now applied to the set-up of section 4. A simple moving average solution for filtering the transit times is applied along the Kalman filter in order to be able to judge the Kalman filter. Two filter parametrizations have been chosen: first a parametrization with no distinction between the noise sources in terms of individual weights (example 1) and then a parametrization which is tuned more to the relative importance of the noise sources in the physical process and its measurement.

Example 1: A first example of a Kalman filter treats all noise sources as equal that means all noise covariance matrices are chosen to be the unity matrix:

$$Q = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad R = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The steady state Kalman filter gain matrix is then given by:

$$K = \begin{bmatrix} 0.2573 & 0.2573 & 0.2573 & 0.2573 \\ 0.3090 & 0.3090 & 0.3090 & 0.3090 \\ 0.3062 & 0.3062 & 0.3062 & 0.3062 \end{bmatrix}$$

$$J_{Kalman} = \sum_{k=1}^{N} |v_{axial\_ideal}(k) - v_{axial\_estimated}(k)|$$

$$J_{measured} = \sum_{k=1}^{N} |v_{axial\_ideal}(k) - v_{axial\_measured}(k)|$$

$$J_{filtered} = \sum_{k=1}^{N} |v_{axial\_ideal}(k) - v_{axial\_filtered}(k)|$$ (23a-23c)

The values of these indices are tabulated in Table 1. Fig. 6d shows the performance of the Kalman filter for the estimation of the speed of sound.

Example 2: In this example the covariance matrices are adapted as follows: The diagonal elements of the covariance matrix are weighted individually with diminishing magnitude of the velocity components. The values 1000, 10 and 1 are chosen arbitrarily (the same holds for R), the ratio between the elements however is chosen according to the magnitudes. Also the diagonal elements of $R$ are all equal and at least ten times larger than the values for $Q$. Therefore a low pass filter effect can be expected.

$$Q = \begin{bmatrix} 1000 & 0 & 0 \\ 0 & 10 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad R = \begin{bmatrix} 10000 & 0 & 0 & 0 \\ 0 & 10000 & 0 & 0 \\ 0 & 0 & 1000 & 0 \end{bmatrix}$$

The steady state Kalman filter gain matrix is now changed to

$$K = \begin{bmatrix} 0.1264 & 0.1264 & 0.1264 & 0.1264 \\ 0.0156 & -0.0156 & 0.0156 & -0.0156 \\ 0.0050 & -0.0156 & -0.0156 & 0.0156 \end{bmatrix}$$

Figure 7 displays the same quantities as in example 1. It is clearly visible that the Kalman filter performs much better than before and even outperforms the moving average filtered estimate (see Table 1). That means that with an appropriate Kalman filter parametrization good results can be obtained. Keep in mind that the above choice of the filter was not optimized explicitly to the noise sources but were chosen by inspection.

Figure 6a shows the determination of the axial velocity if equations (5a, b) are used. No additional low pass filtering was applied. Fig. 6b shows the tracking capability and noise rejection of the chosen Kalman filter for the estimated axial path velocity $v_{axial\_estimated}$ and a low pass filtered (moving average of length 25) signal $v_{axial\_filtered}$ of the measured axial velocity together with the ideal axial velocity. Fig. 6c shows the same for the transversal component. Both figures indicate that the Kalman filter is not well tuned to the simulation conditions, while the low pass filter behaves much better. In order to compare the filters the following performance criteria have been used:

- $J_{Kalman}$
- $J_{measured}$
- $J_{filtered}$

$$J_{Kalman} = \sum_{k=1}^{N} |v_{axial\_ideal}(k) - v_{axial\_estimated}(k)|$$

$$J_{measured} = \sum_{k=1}^{N} |v_{axial\_ideal}(k) - v_{axial\_measured}(k)|$$

$$J_{filtered} = \sum_{k=1}^{N} |v_{axial\_ideal}(k) - v_{axial\_filtered}(k)|$$ (23a-23c)
Table 1: Performance indices according to equations (23) for the different filtering techniques for the two examples (units of J is [m/s]).

<table>
<thead>
<tr>
<th>Performance index</th>
<th>Example 1</th>
<th>Example 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>unfiltered data: ( J_{\text{measured}} )</td>
<td>1386</td>
<td>1420</td>
</tr>
<tr>
<td>Filtered by a moving average filter of length 25: ( J_{\text{filtered}} )</td>
<td>304.6</td>
<td>336.0</td>
</tr>
<tr>
<td>Kalman filter: ( J_{\text{Kalman}} )</td>
<td>933.5</td>
<td>251.4</td>
</tr>
</tbody>
</table>

6. OUTLOOK AND CONCLUSION

By tuning a Kalman filter for a random walk model of the physical measurement process properly it has been shown that recursive estimations of the time varying velocity components \( v_\text{ax}(k), v_\text{trans}(k), c(k) \) are possible without the use of the delta time information of the transit times found via the correlation of up- and downstream acoustic pulses. Due to this fact, the following problems and steps to be taken in further works could be addressed:

- Behavior of the filter for slowly varying situation (sinusoidal, ramp-like)
- Optimization of the filter for measured noise levels
- Comparison of the filter performance for different methods to determine the transit times to a sufficiently high accuracy
- Evaluate the filter with real measurements
- Can the Kalman filter solution be extended to a situation where also the delta time between up- and downstream pulses can be included?
- Modelling the measurement noise dependent on the method of transit time determination
- Modelling of the system noise by using some turbulence model.

7. REFERENCES

ABSTRACT

The current research work focuses on the application and testing of two optimization-based approaches for quantifying and locating leakage losses by means of hydraulic model calibration: (i) a steady-state analysis and (ii) a transient-based technique. The former is applied to an existing water distribution network in order to evaluate leakage losses, which is diffused on a larger area of the network; a hydraulic solver that takes into account pressure-dependent leakage is presented. The latter is tested using transient pressure data collected from an experimental multi-pipe system composed of PVC pipes. A hydraulic transient solver that describes fluid transients in plastic pipes has been developed. A key factor for the success of the technique is the accurate calibration of the transient solver, namely adequate boundary conditions and an accurate creep function of the pipe wall. The application of the inverse transient method to leak location and sizing is carried out by a genetic algorithm search method starting from different random seeds. Results obtained by using both methodologies are discussed.

KEYWORDS: Hydraulic transients; Pipe systems; Simulation models; Water losses.

1. INTRODUCTION

The increasing population of the cities requires an efficient water resources management, particularly when natural resources are growing scarce. On the other hand, water distribution networks usually present a high percentage of water losses – the volume left after subtracting all authorized billed and unbilled water consumption from the system input volume can reach up to 50-60%. According to Alegre et al. [1] and AWWA Water Loss Control Committee [2], there are two types of water losses: apparent and real losses. Apparent losses include customer meter inaccuracy, data handling errors and unauthorized use. Apparent losses depend on the degree of effort in the compilation of accurate water usage data. Real losses are the physical water losses from the distribution system and include leakage on transmission and distribution mains, on service connections and storage overflows, and pipe bursts. The percentage of real losses depends not only on the age and physical deterioration of the network, but also on normal and extreme operating pressures of the system.

Leakage losses can occur either by the total failure (breaks) of one or more pipes or by small openings and leaks located in the pipe walls and around the pipe junctions. In the first case, breaks are usually located but often cause considerable damage and they may also affect the distribution system water quality as they are a potential source of contamination during low-pressure or back-flow conditions [2, 3]. In the second case, the water loss is diffused on a larger area of the network pipes and usually accounts for a significant proportion of the water that is supplied into the distribution system.

Quantifying the total amount and location of leakage is of great importance once leakage reduction and control is a crucial step for the financial sustainability of water utilities. To this end, hydraulic simulation models are world-wide considered as effective and efficient tools that have been successfully applied to water losses management. This information is available as the bottom-up contribution to the water audit. However, such procedure is feasible only using a robust and reliable calibrated model that considers the dependency between demands, leaks and pressures. Calibrating water distribution network models is a decisive stage for the model being able to reproduce their hydraulic behaviour for different operational conditions.

In this paper, two optimization-based approaches are investigated for simultaneously quantifying and locating leakage losses by means of hydraulic model calibration: (i) a steady-state flow analysis using an extension of the widely used hydraulic solver EPANET 2 [4] and applied to evaluate leakage losses in a real-life water distribution network; and (ii) a transient-based technique to detect leaks in a PVC experimental facility.

2. LEAKAGE EVALUATION IN WATER NETWORKS:

STEADY-STATE APPROACH

Leakage levels can be reasonably well assessed by the analysis of minimum night-flows in discrete zones of the water distribution system - district metered areas (DMAs) – where all in and out flows are well controlled and measured [5-7]. However, hydraulic simulation models considering pressure-dependent relationships have effectively provided a quantitative measure of leakage under different scenarios and distinct pressure levels [8-12].

Tucciarelli et al. [10] proposed a hydraulic simulation model used in the calibration procedure according to which water losses are computed assuming that, in the pipes of each zone, there is a constant leakage per unit area of the pipe's surface, based on the idea that the aging effect is uniform on all pipe walls (assuming pipes have the same age and material). The authors assumed that the spatial distribution of the potential demand could be estimated with some degree of confidence based on the knowledge of the consumers supplied by each node of the network. The same assumption was adopted by Soares et al. [11] for the development of a hydraulic simulation model based on leakage and on pressure-dependent demand

1Kepler Soares, Alexandre; 2Covas, Dídia I.C.; 3Reis, Luísa Fernanda R.
using an extension of the software EPANET 2. In this study, it has been assumed that the total amount of supplied water (TS) can be divided in the total demand actually supplied to the consumers (TD) and leakage (V), i.e., \( TS = TD + V \). The total demand TD can be calculated in terms of average monthly demand (baseline demand), demand pattern multiplier factor and restrictions on pressure levels in the network (pressure-driven model). Leakage is computed by the following equation:

\[
V = \sum_{i=1}^{N} v_i \rightarrow v_i = (H_i - z_i) \frac{M_i}{2} \sum_{j=1}^{N} D_{ij} \theta_j L_{ij}
\]

where \( N \) = number of nodes; \( H_i \) = piezometric head at node \( i \); \( z_i \) = topographic elevation; \( \theta \) = loss exponent; \( M_i \) = total number of pipes linked to node \( i \); \( D_{ij} \) = pipe diameter; \( L_{ij} \) = pipe length; and \( \theta_j \) = leakage coefficient per unit pipe surface of the pipe linking nodes \( i \) and \( j \).

In order to determine the unknown network parameters (i.e., pipe roughness, leakage coefficient \( \theta \) and exponent \( \beta \) in the current case), the proposed hydraulic solver has been linked to a hybrid optimisation model. The idea is to solve the inverse problem by linking an effective global search method (Genetic Algorithms – GA) to an efficient local search method (Simplex method – Nelder and Mead [13]). The indirect approach to solve the inverse problem of parameters identification is set as the minimisation of least square errors between observed and calculated pressures and flow rates, as follows:

\[
\min_{Z} OF = \sum_{n=1}^{n^o} \left[ \sum_{i=1}^{n^p} (P_i - P^*_i)^2 \right]^{\frac{1}{2}} + \sum_{i=1}^{n^p} \left( Q_i - Q^*_i \right)^2 \sum_{i=1}^{n^o} (\theta_i - \theta^*_i)^2 \right]
\]

where \( n^o = \) number of scenarios considered in the simulations; \( n^p = \) number of nodes where pressure has been measured; \( n^o = \) number of pipes where flow rate has been measured; \( P = \) computed pressure vector; \( Q = \) computed flow rate vector; \( P^* = \) measured pressure vector; \( Q^* = \) measured flow rate vector; and \( Z = \) decision variables vector corresponding to:

\[
Z = (\varepsilon_1, \theta_1, \theta_2, ..., \theta_n, \beta_1, ..., \beta_n)
\]

where \( \varepsilon = \) pipe roughness; \( n = \) number of zones with constant pipe roughness (i.e., similar pipe material, age); \( n_1 = \) number of zones with constant leakage coefficients \( \theta \); and \( n_1 = \) number of zones with constant loss exponent \( \beta \). In order to decrease the level of parameterization (complexity) of the problem it is assumed that leakage parameters are homogeneous within specified areas of the network (i.e., in DMAs).

### 2.1. Analysis of leakage losses in a real-life water distribution network

In this study, leakage losses have been evaluated in an existing water distribution area – Parque Santa Rita sector - in the city of Goiânia, Brazil (Figure 1). This system is composed of 35 nodes, 41 pipes made of PVC and 287 service connections.

In order to collect data for model calibration, pressure was monitored at three nodes of the distribution sector (Points 1, 2 and 3) and flow rate was measured at the inlet pipe (Figure 1). The obtained curves of inflows are shown in Figure 2, which have indicated a well-defined residential pattern of consumption with lower flows during the night and maximum flows along the daytime hours (higher values at around 10:00 am). The average daily inflow resulted in 3.4 L/s, and the minimum night flow measured was 0.9 L/s, which can be explained by the supply of residential tanks as well as by the occurrence of leakage in the network. To verify the water consumption during the night (supply of residential tanks) it would be necessary to carry out the monitoring of consumer demand.

Due to the fact that the EPANET model is only applicable to turbulent flows, it was necessary to check the Reynolds number for each pipe link of the network under distinct periods of the day. The only scenario in which turbulent flows occur in all pipe links is that of maximum consumption. Therefore, the scenario used in the calibration tests was the highest average flow rate (5.11 L/s) observed at 10 am. It should also be noted that the friction factor calculation for smooth turbulent flow, as occurs in PVC pipes without incrustation, is with the Blasius formula. This model is not codified in EPANET and it depends on the Reynolds number only.

The calibration process was carried out in order to determine leakage parameters (leakage coefficient \( \theta \) and loss exponent \( \beta \)) as well as pipe roughness by running five distinct initial populations of solutions (GA random seeds). The loss exponent has been considered homogeneous for all pipes of the sector. Two areas were established for both leakage coefficient and pipe roughness, being the first area composed of pipes with 100 mm diameter, and the second area consisting of pipes with 50 mm diameter.

Parameter values of the leakage model and pipe roughness obtained by using the calibration process are as follows: \( \varepsilon_1 = 0.46 \text{ mm}; \varepsilon_2 = 0.01 \text{ mm}; \theta_1 = 2.9 \times 10^{-8}; \theta_2 = 2.5 \times 10^{-8}; \beta = 0.55.\) Table 1 shows the values of calculated pressure heads obtained, which are compared to the pressure heads observed in the monitoring points. The maximum absolute error is less than 1.5 m, which can be considered acceptable for a good model calibration. The calculated inflow and leakage losses are presented in Table 2 and Figure 3. The relative error between the observed inflow (266.510 m³/day) and the calculated inflow (267.144 m³/day) – sum of calculated consumption and
leakage – resulted in 0.24%. The total daily volume of leakage losses is less than 1% of the total inflow. The small percentage is because of relatively low age of the network as well as the oversizing of the pipes. In this way, leakage may be due to small leaks distributed on the network or on the service connections. It should be noted that the analyses reported in this study do not take into account apparent losses, which can raise the final amount of water losses in the water distribution network sector.

Table 1: Calculated and observed pressure heads.

<table>
<thead>
<tr>
<th>Random seed</th>
<th>Observed pressure head (m)</th>
<th>Calculated pressure head (m)</th>
<th>Absolute error (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>Point 2</td>
<td>Point 3</td>
<td>Point 1</td>
</tr>
<tr>
<td>1</td>
<td>20.13</td>
<td>22.70</td>
<td>22.13</td>
</tr>
<tr>
<td>2</td>
<td>20.06</td>
<td>21.22</td>
<td>22.28</td>
</tr>
<tr>
<td>3</td>
<td>20.12</td>
<td>21.21</td>
<td>22.31</td>
</tr>
<tr>
<td>4</td>
<td>20.10</td>
<td>21.21</td>
<td>22.27</td>
</tr>
<tr>
<td>5</td>
<td>20.09</td>
<td>21.21</td>
<td>22.30</td>
</tr>
</tbody>
</table>

Table 2: Calculated consumption and leakage and observed inflows.

<table>
<thead>
<tr>
<th>Hour of Day</th>
<th>Observed inflow (L/s)</th>
<th>Calculated consumption (L/s)</th>
<th>Calculated leakage (L/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>00:00</td>
<td>1,537</td>
<td>1,515</td>
<td>0.029</td>
</tr>
<tr>
<td>01:00</td>
<td>0.953</td>
<td>0.931</td>
<td>0.029</td>
</tr>
<tr>
<td>02:00</td>
<td>0.909</td>
<td>0.887</td>
<td>0.029</td>
</tr>
<tr>
<td>03:00</td>
<td>0.816</td>
<td>0.794</td>
<td>0.030</td>
</tr>
<tr>
<td>04:00</td>
<td>0.837</td>
<td>0.815</td>
<td>0.030</td>
</tr>
<tr>
<td>05:00</td>
<td>1.146</td>
<td>1.124</td>
<td>0.029</td>
</tr>
<tr>
<td>06:00</td>
<td>1.749</td>
<td>1.727</td>
<td>0.029</td>
</tr>
<tr>
<td>07:00</td>
<td>2.677</td>
<td>2.655</td>
<td>0.029</td>
</tr>
<tr>
<td>08:00</td>
<td>3.902</td>
<td>3.880</td>
<td>0.029</td>
</tr>
<tr>
<td>09:00</td>
<td>4.307</td>
<td>4.285</td>
<td>0.029</td>
</tr>
<tr>
<td>10:00</td>
<td>5.107</td>
<td>5.085</td>
<td>0.028</td>
</tr>
<tr>
<td>11:00</td>
<td>5.063</td>
<td>5.041</td>
<td>0.028</td>
</tr>
<tr>
<td>12:00</td>
<td>4.685</td>
<td>4.663</td>
<td>0.029</td>
</tr>
<tr>
<td>13:00</td>
<td>4.487</td>
<td>4.465</td>
<td>0.029</td>
</tr>
<tr>
<td>14:00</td>
<td>4.020</td>
<td>3.998</td>
<td>0.029</td>
</tr>
<tr>
<td>15:00</td>
<td>3.957</td>
<td>3.935</td>
<td>0.029</td>
</tr>
<tr>
<td>16:00</td>
<td>3.660</td>
<td>3.638</td>
<td>0.029</td>
</tr>
<tr>
<td>17:00</td>
<td>4.293</td>
<td>4.271</td>
<td>0.029</td>
</tr>
<tr>
<td>18:00</td>
<td>4.764</td>
<td>4.742</td>
<td>0.029</td>
</tr>
<tr>
<td>19:00</td>
<td>3.882</td>
<td>3.860</td>
<td>0.029</td>
</tr>
<tr>
<td>20:00</td>
<td>3.282</td>
<td>3.260</td>
<td>0.029</td>
</tr>
<tr>
<td>21:00</td>
<td>3.216</td>
<td>3.194</td>
<td>0.029</td>
</tr>
<tr>
<td>22:00</td>
<td>2.696</td>
<td>2.674</td>
<td>0.029</td>
</tr>
<tr>
<td>23:00</td>
<td>2.085</td>
<td>2.063</td>
<td>0.029</td>
</tr>
</tbody>
</table>

Total (m³): 266,510 264,598 2,546

Figure 3: Numerical results of the hydraulic model versus data collected: inflow, consumption, leakage.

[Figure 4: Experimental Hydraulic Circuit (EHC): simplified system configuration (thick continuous line) and neglected pipes (thin dashed line).]
3.1. Inverse transient analysis (ITA)

The indirect transient approach to solve the inverse problem of leak detection is set as the minimisation of least square errors (LSE) between collected and calculated pressures, as follows:

$$\min_{Z} OF = \sum_{i=1}^{DT} \left\{ \sum_{j=1}^{n_p} \left( P_{i,j} - P^*_{i,j} \right)^2 / \left( \sum_{j=1}^{n_p} P^*_j / n_t \right)^2 \right\}$$

(4)

where $OF$ = objective function; $DT$ = number of time steps of observed hydraulic transient event; $P$ = vector of pressures calculated by the transient solver; $P^*$ = transient vector of observed pressures (collected pressure data); $n^P$ = number of measurement locations in the system; and $Z$ = decision variables vector corresponding to unknown calibration parameters:

$$Z = (\tau_1, \ldots, \tau_{N_{KV}}, J_1, \ldots, J_{N_{KV}}, C_d A_0, \ldots, C_d A_{0n_1})$$

where $\tau_i$ = retardation time of the dashpot of the Kelvin-Voigt element $k$; $J_i$ = creep-compliance of the spring of the Kelvin-Voigt element $k$; $N_{KV}$ = number of Kelvin-Voigt elements; $C_d A_0$ = effective leak area (product of the discharge coefficient $C_d$ and the actual leak area $A_0$) of node $i$; and $n_t$ = number of potential leak locations.

Retardation times, $\tau_i$, and creep coefficients, $J_i$, are parameters of the viscoelastic model that describe the creep function of PVC pipes. In this study, the creep function is described by a mathematical expression, which is incorporated in the hydraulic transient equations by the generalized Kelvin-Voigt mechanical model of a viscoelastic solid. This model is a combination of elements (springs and dashpots) that numerically describe the rheological response of viscoelastic solids [23]:

$$J(t) = J_0 + \sum_{k=1}^{N_{KV}} J_k \left( 1 - e^{-t/\tau_k} \right)$$

(6)

where $J_0$ = creep-compliance of the first spring defined by $J_0 = 1/E_0$; $E_0$ = Young’s modulus of elasticity of pipe.

The developed hydraulic model is solved by the Method of Characteristics. Using a rectangular computational grid and neglecting convective terms (the fluid velocity is negligible compared to elastic wave speed), piezometric heads ($H$) and flow rates ($Q$) can be solved numerically by the following scheme:

$$C^t (H_i - H_{i+1,j-1}) + a \frac{\partial Q_j}{\partial t} = a \frac{\partial \psi_j}{\partial t}$$

(7)

where $i$ = coordinate along the pipe axis; $t$ = time; $H$ = piezometric head; $Q$ = flow rate; $a$ = celerity or elastic wave speed (dependent on the fluid compressibility, and on the physical properties and external constraints of the pipe); $g$ = gravity acceleration; $A$ = pipe cross-sectional area; $\psi$ = retarded strain component (in viscoelastic pipes the total strain can be decomposed into an instantaneous-elastic strain and a retarded strain); and $h_j$ = head loss per unit length ($h_j = f Q_j |Q_j|/2DA^4$ in turbulent conditions, in which $f$ = Darcy-Weisbach friction factor and $D$ = pipe inner diameter). In these equations, the retarded strain time-derivative (fourth term) cannot be directly calculated and require further numerical discretization. Covas [24] presents the basis for calculating these terms.

The inverse model described has been used to determine the creep compliance function $J(t)$. Wave speed was estimated in 460 m/s and parameters $J_i$ and $\tau_i$ were estimated by using GA search method considering only transient pressure data at Location P07. Finished GA optimisation, parameters $\tau_i$ were fixed and coefficients $J_i$ were estimated by using Levenberg-Marquardt local search method. Several initial numerical simulations were run to find the best number of Kelvin-Voigt elements. Combinations of 1, 2 and 3 elements were tested to better describe the creep compliance function in PVC pipes. The optimal number of Kelvin-Voigt elements was obtained by using one single element ($\tau_i = 0.05$ s and $J_i = 0.02250$ GPa$^{-1}$).

3.2. Leak detection simulations

During the experiments, the leak was located at Node 72 (Location P02). ITA was run using GA to locate and size the leak for two steady state turbulent flows: Case A - inflow = 1.84 L/s, leak = 0.84 L/s; and Case B - inflow = 2.47 L/s, leak = 0.73 L/s. Obtained optimal leak locations by using ITA are presented in Figure 5 in terms of both frequency of GA detection (for 10 GA random seeds) and minimum value of objective function for each step. Calculated pressures for the optimal leak location at each step are presented in Figure 6 for the Location P07. In this research work, leaks can be located with an accuracy corresponding to 4% of the total length of discharge line, despite measurement errors (noise in the pressure transient signal), errors in calibrated viscoelastic parameters of the hydraulic transient solver, and optimisation method capability of determining the best solutions.

4. SUMMARY AND CONCLUSIONS

The current paper presented the application and testing of two optimization-based approaches for simultaneously quantifying and locating leakage losses by means of hydraulic model calibration. In order to evaluate leakage losses in a
complex real-life water distribution network, in which the leakage is diffused in a larger area of the network, a steady-state flow analysis using an extension of the hydraulic solver EPANET 2 has been applied. Parameters of the leakage model were determined via inverse analysis considering data collected during the extended period. The total real losses (leakage) can be assessed and results have shown lower values for leakage (less than 1%). The steady-state analysis has proved to be promising for evaluating leakage losses in complex pipe networks, in which the leakage is diffused on a larger area of the network. On the other hand, transient-based techniques for leak detection can be an arduous task in multi-pipe systems (such as pipe networks), due to their complexity and uncertainties on both interior and boundary conditions. In this way, an inverse transient method has been applied to an experimental pipeline system with simulated pipe breaks. The application of inverse transient analysis to leak location was based on a multi-step procedure carried out by a GA search method. Physical data were collected from the experimental facility composed of PVC pipes, characterized by viscoelastic mechanical behaviour of the pipe-walls. Results have shown that leaks can be accurately located with a 4% uncertainty in a 67 m long pipe system. As a further study, the inverse transient method presented here has to be applied to a real-life pipe system.

5. ACKNOWLEDGMENTS

The authors gratefully acknowledge the financial support of both “Fundaçao de Amparo à Pesquisa do Estado de Sao Paulo” (FAPESP, Brazil) and “Conselho Nacional de Desenvolvimento Científico e Tecnológico” (CNPq, Brazil). The authors would also like to thank Saneago water utility and Cryslara Lemes, Agne Cunha and Willian Pinto for support in the development of field tests.

6. REFERENCES

**ABSTRACT**

The bearing performance of a centrifuge pump is evaluated by the number of revolutions occurring before its respective degradation. Degradation can be a result of insufficient lubrication caused by reduced viscosity of the employed oil, due to contamination or temperature increase. Increase of the bearing temperature is mainly caused by the friction of continuous or occasional loads, misalignment, dimensional diversion or use of inadequate lubricants. An ideal pumping system must be designed to operate and perform without cavitation. However, in practice, this condition is not always met and can induce supplemental hydrodynamic exertion. Such exertion is a combination of axial and radial-thrust which act upon the bearing, with their measurement requiring sensors such as hot-wire anemometers and variable resistance extensometers that are not easily installed in the field. A simple and economical diagnostic for cavitation is desirable to classify its intensity and thus establish the necessary corrective actions. If the contrary exists, damages and accidents with substantial losses can occur. An experimental unit was built to test a pump operating under normal conditions as well as moderate cavitation or incipient damages. Comparing the two conditions, it is concluded that temperature increase in the bearing can be used as a means of detecting moderate cavitation and trigger the necessary corrective actions, despite the complexity of the phenomenon in question.

**KEYWORDS:** cavitation, centrifuge pumps, bearing temperature, experimental analysis, diagnosis.

1. **INTRODUCTION**

1.1 Cavitation Characteristics and Classification

During the flow of a given liquid, at a given temperature, the reduction of pressure leading up to the vapor pressure generates a liquid and gas mixture. The origination of bubbles or cavities gives name to the term cavitation. Cavitation can emerge, for example, from the suction of pumps at room temperature, when it limits the aspiration capacity of the pump causing undesirable effects in its performance, such as rotor erosion, hydraulic yield decrease, noise level elevations, vibrations, etc. The maintenance of pumps under this condition typically requires employing high cost instruments in the analysis, such as vibration analyzers. For this reason, this research aims at introducing simple methods and instrumentation involving the analysis of practical values and results obtained during the experiment. In order to achieve this goal, temperature variation of the pump bearing was recorded with a pump operating under normal conditions (without cavitation) as well as one with moderate cavitation.

The knowledge of this procedure can be used both to serve as a diagnostic regarding the intensity of the phenomenon as well as provide consequent information needed for decision-making requirements.

In cavitation characterization specific measurements are employed, such as the NPSH (Net Positive Suction Head), Equation (1), and the cavitation index (σ), Equation (2).

To achieve a more precise characterization, these measurements are employed along with measurements of noise levels, vibrations and rotor erosion. Whenever possible, the amount of bubbles formed during the phenomenon can also be observed. The measured NPSH is presented in relation to the installation, when it is named available NPSH, or in relation to the pump, named required NPSH.

The NPSH corresponds to the energy available to transport the liquid through the suction tubes until the housing pump flange. Equation (1) is utilized to obtain

\[ NPSH_d = \frac{P_a - P_v}{\gamma} + H_s \]  

(1)

The NPSH, on the other hand, represents the energy of the liquid above the vapor pressure within the suction of the pump, which is necessary for aspiration, and depends on the pump design, the rotor diameter and the specific velocity, all supplied by the manufacturer through testing.

This standardized test (ANSI/HI 1.6-2000, American National Standard for Centrifugal Pump Tests and Hydraulic Institute) consists of a circuit installed and subjected to reducing aspiration or NPSH, all the while maintaining a constant flow.

The gradual reduction of the NPSH until the emergence of cavitation corresponds to a 3% decrease in pump pressure, with the appearance of bubbles and noise typical of the phenomenon serving as references that establish incipient cavitation.

With regards to the cavitation index, σ, a non-dimensional measure which establishes the relationship between the intensity of forces that oppose the emergence of cavitation and those that favor it, the possibility that the phenomenon will occur is measured.

Despite the ease of calculating its value, according to Equation (2), such solution does not occur with the reproduction of various hydrodynamic configurations for different intensities.

\[ \sigma = \frac{P_{abs} - P_v}{\frac{\gamma v^2}{2g}} \]  

(2)

This study employs the NPSH, together with other parameters, as an index to determine cavitation.

Intensity levels of cavitation in general can only be determined by experiments with specific regimens, which create difficulty in regards to standardization of cavitation classification. Proposals such as TOMAS (1) reflect the degree of cavitation classified according to five areas:
• Incipient cavitation determined by aforementioned standardized test conditions;
• Reduced cavitation expressed as the first manifestations of the phenomenon;
• Moderate cavitation corresponding to stable flow conditions;
• Fully developed cavitation representing maximum levels of noise and vibration being observed in high frequency;
• Advanced cavitation represented as the phase after the maximum intensity, although with a contradictory reduction of noise and vibration levels.

In this study, the advanced cavitation level was not reproduced within testing as to avoid risks related to the equipment and personnel during the pump operation.

Other names commonly employed by researchers such as KNAPP (2) and BOUZIAD (3) correspond to the classification proposed by TOMAS (1):

Critical cavitation = Reduced cavitation
Cavitation with incipient damages = Moderate cavitation

The majority of tests reproduce the effects of a pump operating with incipient cavitation, while only a few investigate the behavior of the pump operating with cavitation levels above incipient due to the hydrodynamic conditions of such an unstable phenomenon. Nevertheless, the set up of a robust experiment allowed for stability to be achieved in the system, yielding results that may serve as a diagnostic tool for conditions under a more severe cavitation.

Mechanical failures caused by variations of the pump’s cavitation levels include erosion of the rotor, ring and volute, decrease in yield, excessive mechanical exertion in roller bearings, vibration increase and consequently, breakage of fasteners and shafts, among others.

1.2 Pump Bearing Temperature

The performance of bearings is evaluated by the number of revolutions which have occurred before its proper degradation.

Degradation is a consequence of inadequate lubrication generated by a reduction in the viscosity of the oil, which is associated with contamination or temperature increase caused mainly by friction related to:
• Continuous or occasional increase of mechanical exertion;
• Misalignment or dimensional deviations;
• Inappropriate quantity or quality of lubricant.

In this study the pump performs both in normal conditions and with moderate cavitation, without the mentioned causes, except for the exertion imposed by the phenomenon.

Such exertion is a combination of the axial and radial thrusts that act upon the bearing. Axial thrust is generated by pressure differences that act on the rotor surface, similar to suction and dynamic effects, which exist due to a change in the direction of flow when liquid goes through the rotor. The sum of the unbalanced pressure forces at the axial direction is expressed by Equation (3):

\[ E_r = F_{p_a} + F_{p_r} - F_{p_u} - F_{p_s} - F_r \]  

\( F_{p_a} \): Force of the variable pressure at the back wall of the rotor;
\( F_{p_r} \): Force of the atmospheric pressure at the extremity of the axis (rotor’s rear);
\( F_{p_u} \): Force of the suction pressure at the front of the rotor;
\( F_{p_s} \): Force of the variable pressure at the front wall of the rotor;
\( F_r \): Force of the impulsion caused by the change in direction of the trajectory of the rotor.

The radial thrust is generated by the differences in pressures distributed in a circumferential manner at the exit of the rotor’s flow. Such pressure differences are caused by axial deflections, reaction strengths and by non-uniform discharge flows.

The values of the radial thrust are smaller at the point of best efficiency in opposition to the condition of null or maximum flow, where unbalances are evident.

The general equation to determine radial thrust is as follows by Equation (4):

\[ E_r = k_r \cdot \rho \cdot H \cdot D_2 \cdot b_2 \]  

\( k_r \): Radial thrust coefficient
\( \rho \): density
\( H \): Total head in meters
\( D_2 \): Rotor diameter
\( b_2 \): Width of the rotor’s vane

Where \( k_r \), radial thrust coefficient, is given by Equation (5):

\[ k_r = k_{r0} \cdot \left(1 - \frac{Q}{Q_{BEP}}\right)^2 \]  

\( k_{r0} \): Radial thrust coefficient in shut-off mode
\( Q \): Flow at the operating point
\( Q_{BEP} \): Flow at the point of best efficiency

Measurements for both thrusts require difficult sensor installations in the field, such as a hot wire anemometer distributed through the volute’s circumference, or even transmitters that support the load at the pump bearing. In this study the goal is to indirectly measure those exertions through the temperature levels, particularly in the moderate cavitation condition, and then compare these findings with the temperature levels of the pump during normal conditions. FRANZ et al. (4) measured radial thrusts in centrifuge pumps with cavitation operation and established relationships between their magnitude and the vibration.

Wilkinson et al. (5) tested centrifuge pumps in order to correlate bearing failures caused by overload during the operation with cavitation and concluded that under this condition, bearing temperatures tend to elevate.

2. EXPERIMENTAL SET UP

The experiment was set up at the Laboratory of Fluid Hydraulics and Mechanics at the College of Civil Engineering at UNICAMP (LHMF). As shown in Figure (1), the experiment was arranged of the following:

• Water circuit powered from a lower tank; booster pump motor set; upper tank with turbulence tranquilizer; water level reader; direct suction valve with fence for the main pump; suction and discharge piping with control valves; pressure, suction and discharge transducers; electromagnetic flow meter downstream of the pump motor set; and lower tank return discharge siphon;
• Electrical pump actuation/release through a three phase network: control and protection panel, frequency inverter and 22.0 kW electrical motor;
• Radial horizontal centrifuge pump; single stage, base sleeve type semi-open rotor with a 264mm diameter and angular contact with oil lubricated roll bearings.

[Figure 1: Motor pump with suction and discharge piping.]
Table (1) displays the instrument selection made to allow for the field reproduction of the following measurements:

**[Table 1]: Displays the instrument selection made to allow for the field reproduction of the following measurements.**

<table>
<thead>
<tr>
<th>Instrument</th>
<th>Brand</th>
<th>Operating Range</th>
<th>Precision</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure suction transducer</td>
<td>Smart</td>
<td>-250 to + 250 kPa</td>
<td>0.20%</td>
</tr>
<tr>
<td>Discharge pressure transducer</td>
<td>Jumo</td>
<td>-250 to + 2500 kPa</td>
<td>0.10%</td>
</tr>
<tr>
<td>Electromagnetic flow gauge</td>
<td>Emerson</td>
<td>0 to 18 m/s</td>
<td>0.25%</td>
</tr>
<tr>
<td>Oximeter</td>
<td>Alfakit</td>
<td>0 to 19.9 mg/l</td>
<td>2.00%</td>
</tr>
<tr>
<td>Infrared thermometer</td>
<td>Raytek</td>
<td>-20 to + 275 °C</td>
<td>1.0°C</td>
</tr>
<tr>
<td>Digital thermometer</td>
<td>Soma</td>
<td>-25 to + 60 °C</td>
<td>0.1 °C</td>
</tr>
<tr>
<td>Mercury thermometer</td>
<td>BD</td>
<td>0 to 40 °C</td>
<td>0.5 °C</td>
</tr>
<tr>
<td>Aneroid barometer</td>
<td>Fischer</td>
<td>76 to 105 kPa</td>
<td>1.00%</td>
</tr>
</tbody>
</table>

Measurements with the infrared thermometer were done at the same points for both pump operating conditions, with and without cavitation, at specific time intervals, as shown in Figure (2):

3. METHODOLOGY

Five stages were observed for the operating conditions of the pump with flows of 0.0586, 0.0619, 0.0653, 0.0686, 0.0719 and 0.0753 m³/s. Measured in each stage were Q, Pfs, Pfr, the bearing temperature at LA and LOA, as well as calculations of H, NPSHd, ΔH and ΔNPSH (difference between available and required).

- Stage 1 – Operation under normal conditions with measurements obtained every 30 minutes.
- Stage 2 – Calculation of the NPSHr according to the standardized test from the Hydraulic Institute.
- Stage 3 – Characterization of the most severe levels of cavitation.
- Stage 4 – Calculation of the time necessary for the stabilization of the bearing temperature at the flow of 0.0586 m³/s under normal conditions and at the flow of 0.0719 m³/s with moderate cavitation.
- Stage 5 – Operation under the moderate cavitation condition with measurements executed every 30 minutes.

The simulations for the cavitation conditions were obtained through pressure reduction by closing the control valve installed upstream of the pump.

3.1 Stage One – Normal Operating Conditions

In this stage the following measurements were registered: hydraulic measurements of Q, Pm, Pfr, the bearing temperature at LA and LOA and calculations of H, NPSHr, ΔH and ΔNPSH (difference between available and required).

To calculate the parameters of the study, Equations (6) to (8) were employed:

**Total head (in meters):**

\[ H = Hr - Hs \]  

(6)

**Discharge head (in meters):**

\[ Hr = \frac{Pfr}{\gamma} + \frac{v^2}{2g} \]  

(7)

**Suction head (in meters):**

\[ Hs = \frac{Pfs}{\gamma} + \frac{v^2}{2g} \]  

(8)

The summary of steps for this stage is composed of:

1- Checking the water level in the upper tank with the gauge;
2- Complete opening of the suction control valve;
3- Priming of the oil pump;
4- Actuation of the pump by the frequency invertor with ramp, until the speed of 1740 rpm;
5- Opening of the discharge control valve until the specified flow Q;
6- Measurement of Pfs, Pfr, Q, bearing temperature at LA and LOA, water temperature and atmospheric pressure.

3.2 Stage Two – Determining the NPSH

In this stage the NPSH was determined with the pump working at a fixed flow level and with the pressure reduced by a choke near the suction control valve until a drop of 3% in the total head H was achieved, in accordance with the HI. The emergence of incipient cavitation was the basis to determine more severe levels in the next test.

The summary of steps is similar to the prior stage, however with items 7 to 9 being added due to pressure reduction, as follows:

7- Pressure reduction achieved by closing the suction control valve, approximately 5 kPa each adjustment;
8- Adjustment of the flow at the electromagnetic measurer after stabilizing the flow;
9- Repetition of procedures until the total head suffered a drop corresponding to a variation of 3%.

3.3 Stage Three – Characterization of the Most Severe Levels of Cavitation

This stage classifies the operating conditions with cavitation at the incipient, reduced, moderate and fully developed levels through a greater choke of the suction control valve and pressure reduction. The bearing temperature of both sides is not measure in this stage.

Different from the already standardized NPSH for incipient cavitation, the classification for higher levels requires observation and recording of variables that can determine the phenomenon’s change of intensity level. In this experiment the main tool for such determination was the correlation of magnitudes measured and calculated for different flows. Those magnitudes allowed for a classification based on significant variations during the phase of gradual reduction regarding the NPSHr installation, as shown in Table (2).

**[Table 2]: Magnitudes for the classification of cavitation levels.**

<table>
<thead>
<tr>
<th>Measured Magnitudes</th>
<th>Calculated Magnitudes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pfs</td>
<td>x</td>
</tr>
<tr>
<td>Pfr</td>
<td>x</td>
</tr>
<tr>
<td>Δ NPSH</td>
<td>x</td>
</tr>
<tr>
<td>ΔH</td>
<td>x</td>
</tr>
</tbody>
</table>
3.4 Stage Four – Determining the Time Needed to Stabilize Bearing Temperature

The bearing temperature reflects the operating conditions supported by the bearings and depends on several factors, such as lubrication, mechanical set up interference, eventual contaminations or external environmental refrigeration influence.

Notwithstanding, these factors are common in all tests, thus it is assumed that only the load differences resulting from the operations both with normal and with cavitation reflect increase over time.

3.5 Stage Five – Moderate Cavitation Condition

In this stage the first six operating points are repeated for the pump to function with moderate cavitation, and adjustments obtained within the characterization stage are employed.

4. RESULTS

4.1 Normal Operating Conditions

After four hours, during which measurements were taken every 30 minutes, temperatures of the pump performing under normal conditions were recorded and are shown in the Table (3):

**[Table 3]: Measurements and calculation during normal condition testing, where Q = 0.0653 m³/s.**

<table>
<thead>
<tr>
<th>Q</th>
<th>Pns</th>
<th>Pfr</th>
<th>H</th>
<th>TLA °C</th>
<th>TLOA °C</th>
<th>Time s</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0653</td>
<td>-8.6</td>
<td>205.2</td>
<td>234.7</td>
<td>24</td>
<td>24</td>
<td>0</td>
</tr>
<tr>
<td>0.0653</td>
<td>-9.1</td>
<td>205.2</td>
<td>235.2</td>
<td>35</td>
<td>29</td>
<td>1,08E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-9.2</td>
<td>205.6</td>
<td>235.7</td>
<td>41</td>
<td>33</td>
<td>2,16E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-8.7</td>
<td>205.4</td>
<td>235.0</td>
<td>44</td>
<td>36</td>
<td>3,24E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-8.8</td>
<td>205.3</td>
<td>235.0</td>
<td>45</td>
<td>39</td>
<td>4,32E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-8.5</td>
<td>205.1</td>
<td>234.5</td>
<td>46</td>
<td>40</td>
<td>5,40E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-8.8</td>
<td>205.3</td>
<td>235.0</td>
<td>47</td>
<td>41</td>
<td>6,48E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-8.7</td>
<td>205.1</td>
<td>234.7</td>
<td>48</td>
<td>42</td>
<td>7,56E+08</td>
</tr>
<tr>
<td>0.0653</td>
<td>-8.6</td>
<td>205.3</td>
<td>234.8</td>
<td>48</td>
<td>43</td>
<td>8,64E+08</td>
</tr>
</tbody>
</table>

The significant margin of the NPSHd allowed for enhanced aspiration conditions during all tests.

4.2 Determining the NPSH

The test conducted to obtain the NPSH for all flows used in the study followed HI recommendations, with results shown in Table (4):

**[Table 4]: Measurements and calculations to obtain the NPSHr, where Q = 0.0653 m³/s.**

<table>
<thead>
<tr>
<th>Q</th>
<th>Pns</th>
<th>NPSH</th>
<th>NPSH</th>
<th>ΔNPSH</th>
<th>H</th>
<th>Δ H %</th>
<th>Characterization</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0653</td>
<td>-8.7</td>
<td>89.0</td>
<td>35.2</td>
<td>53.8</td>
<td>234.9</td>
<td>0.00</td>
<td>Normal</td>
</tr>
<tr>
<td>0.0653</td>
<td>-10.4</td>
<td>87.3</td>
<td>35.2</td>
<td>52.1</td>
<td>234.4</td>
<td>-0.21</td>
<td>Normal</td>
</tr>
<tr>
<td>0.0653</td>
<td>-15.2</td>
<td>82.5</td>
<td>35.2</td>
<td>47.3</td>
<td>234.1</td>
<td>-0.34</td>
<td>Normal</td>
</tr>
<tr>
<td>0.0653</td>
<td>-15.3</td>
<td>82.6</td>
<td>35.2</td>
<td>6.4</td>
<td>229.0</td>
<td>-2.30</td>
<td>Normal</td>
</tr>
<tr>
<td>0.0653</td>
<td>-9.8</td>
<td>38.1</td>
<td>35.2</td>
<td>2.9</td>
<td>228.7</td>
<td>-2.64</td>
<td>Normal</td>
</tr>
<tr>
<td>0.0653</td>
<td>-62.7</td>
<td>35.2</td>
<td>35.2</td>
<td>0.0</td>
<td>227.8</td>
<td>-3.02</td>
<td>Incipient cavitation</td>
</tr>
</tbody>
</table>

Suction pressure values were calibrated to characterize the first stage of the phenomenon. While starting at the best operating condition, the conditions continuously decreased at a rate of approximately 5 kPa through the closing of the suction control valve, which was repeated until the symptoms of incipient cavitation appeared for each specific flow.

After the consolidation of tests for the remaining flows, Figure (3) displays the following tendency curves:

**[Figure 3: NPSHr graphic for all experimental flows.]**

External conditions such as atmospheric pressure and temperature of the room and of the re-circulated water, varied insignificantly in the controlled circuit, and the curves for all tested flows showed similar tendencies with suction pressure between approximately 34 and 39 kPa at the start of the incipient cavitation.

4.3 Characterization of More Severe Cavitation Levels

To characterize the remaining cavitation levels of reduced, moderate and fully developed, the observed measurements and calculations of the magnitudes are presented in Table (1). Significant increases between measurements depicted a change in the cavitation level. Table (5) summarizes the measurements and calculations for a flow of 0.0653 m³/s, considering the methodology.

**[Table 5]: Magnitudes for cavitation level characterization.**

<table>
<thead>
<tr>
<th>Q</th>
<th>Pns</th>
<th>NPSH</th>
<th>NPSH</th>
<th>ΔNPSH</th>
<th>H</th>
<th>Δ H %</th>
<th>Characterization</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0653</td>
<td>-62.3</td>
<td>65.5</td>
<td>35.2</td>
<td>32.0</td>
<td>0.0</td>
<td>227.9</td>
<td>-3.0</td>
</tr>
<tr>
<td>0.0653</td>
<td>-69.7</td>
<td>50.9</td>
<td>35.2</td>
<td>-4.2</td>
<td>214.5</td>
<td>-8.7</td>
<td>Reduced cavitation</td>
</tr>
<tr>
<td>0.0653</td>
<td>-70.3</td>
<td>49.7</td>
<td>35.2</td>
<td>-9.3</td>
<td>203.2</td>
<td>-13.5</td>
<td>Moderate cavitation</td>
</tr>
<tr>
<td>0.0653</td>
<td>-75.5</td>
<td>45.3</td>
<td>35.2</td>
<td>-17.5</td>
<td>193.5</td>
<td>-17.6</td>
<td>Moderate cavitation</td>
</tr>
<tr>
<td>0.0653</td>
<td>-84.9</td>
<td>-79.1</td>
<td>12.5</td>
<td>35.2</td>
<td>-22.6</td>
<td>178.3</td>
<td>-24.1</td>
</tr>
</tbody>
</table>

During the gradual pressure decrease resulting from the narrowing of the aspiration control valve, variations between the magnitudes were mostly of little significance within the same level. However, a more detailed and organized observation of an expressive number of repetitions with similar or different flows showed differences that propose a new classification for the phenomenon's intensity.

It is worth noting that, in another hydrodynamic system, such variations alone may not completely characterize the intensity and the classification recorded. This study’s findings, however, can certainly serve as a reference to such classification.

4.4 Temperature Stabilization

With the pump performing for eight uninterrupted hours under normal conditions at a flow of 0.0586 m³/s and with moderate cavitation at a flow of 0.0719 m³/s, it was possible to verify the time needed to reach a stable point in the bearing temperature, as shown in Figures (4) and (5).
After the fourth hour, there was no significant change in temperature. This period was adopted as enough time to test both the absence and presence of cavitation.

4.5 Moderate Cavitation Condition

With the pump working under suction pressure similar to the one obtained in the characterization test, the test was performed for four hours and measurements recorded every 30 minutes. Measurement records were compared to those of the pump operating under normal conditions as shown in Table (6).

**Table 6**: Measurements and calculations of the operation with moderate cavitation, where Q = 0.0653 m³/s.

<table>
<thead>
<tr>
<th>Q m³/s</th>
<th>P_2 kPa</th>
<th>NPSH_L kPa</th>
<th>NPSH_L kPa</th>
<th>A NPSH kPa</th>
<th>H kPa</th>
<th>Δ H %</th>
<th>Characterization</th>
<th>TLA °C</th>
<th>TLOA °C</th>
<th>Time s</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0653</td>
<td>-76.9</td>
<td>20.7</td>
<td>35.2</td>
<td>-14.5</td>
<td>193.6</td>
<td>-17.6</td>
<td>Moderate cavitation</td>
<td>24</td>
<td>24</td>
<td>0</td>
</tr>
<tr>
<td>0.0653</td>
<td>-77.1</td>
<td>20.5</td>
<td>35.2</td>
<td>14.7</td>
<td>193.8</td>
<td>-17.5</td>
<td>Moderate cavitation</td>
<td>36</td>
<td>30</td>
<td>1.080E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-76.8</td>
<td>19.7</td>
<td>35.2</td>
<td>-15.5</td>
<td>193.8</td>
<td>-17.5</td>
<td>Moderate cavitation</td>
<td>42</td>
<td>34</td>
<td>2.160E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-76.5</td>
<td>20.0</td>
<td>35.2</td>
<td>-15.2</td>
<td>193.7</td>
<td>-17.5</td>
<td>Moderate cavitation</td>
<td>44</td>
<td>37</td>
<td>3.240E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-75.8</td>
<td>20.6</td>
<td>35.2</td>
<td>-14.6</td>
<td>192.7</td>
<td>-18.0</td>
<td>Moderate cavitation</td>
<td>46</td>
<td>40</td>
<td>4.320E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-76.9</td>
<td>19.5</td>
<td>35.2</td>
<td>-15.7</td>
<td>193.3</td>
<td>-17.7</td>
<td>Moderate cavitation</td>
<td>48</td>
<td>41</td>
<td>5.400E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-76.9</td>
<td>19.5</td>
<td>35.2</td>
<td>-15.7</td>
<td>193.1</td>
<td>-17.8</td>
<td>Moderate cavitation</td>
<td>49</td>
<td>43</td>
<td>6.480E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-76.7</td>
<td>19.7</td>
<td>35.2</td>
<td>-15.5</td>
<td>194.0</td>
<td>-17.4</td>
<td>Moderate cavitation</td>
<td>50</td>
<td>45</td>
<td>7.560E+05</td>
</tr>
<tr>
<td>0.0653</td>
<td>-76.8</td>
<td>19.6</td>
<td>35.2</td>
<td>-15.6</td>
<td>194.5</td>
<td>-17.2</td>
<td>Moderate cavitation</td>
<td>51</td>
<td>46</td>
<td>8.640E+05</td>
</tr>
</tbody>
</table>

Data recording for all flows allowed the direct comparison of bearing temperature between operation with and without cavitation, as seen in Table 6.

4.6 Diagnosis from the Bearing Temperature

The relations obtained from the temperatures of the bearing on the attachment side and on the opposite side of the motor, operating with moderate cavitation and without cavitation, Figures (6) to (9), demonstrate the feasibility of employing this indicator as a diagnostic tool as it presents relevant and linear variations in absolute values.
When comparing the tests with and without cavitation, for all flows, and on both sides of the bearing, the temperature variation between 3 and 4 in degrees centigrade signifies that the hydrodynamic phenomenon influences the load generation on the bearings. Consequently, a temperature profile during tests of regular performance, as supplied by the manufacturer, can be applied to the study flows and serve as an additional diagnostic tool along with rotor erosion, noise and vibration already used in preventive maintenance, as shown in Figures (10) and (11).

The tests show a temperature difference between the pump with normal conditions and the pump with cavitation, on both sides of the bearing (attached to the motor and to the pump adapter), recorded in Table (7):

**Table 7**: Classification criteria for moderate cavitation.

<table>
<thead>
<tr>
<th>$P_{fs}$ kPa</th>
<th>$\Delta$ NPSH kPa</th>
<th>$\Delta$ H kPa</th>
<th>$\Delta T$ BEARING LA °C</th>
<th>$\Delta T$ BEARING LOA °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>-70 to -80</td>
<td>-6 to -18</td>
<td>-10 to -18</td>
<td>3 to 4</td>
<td>3 to 4</td>
</tr>
</tbody>
</table>

Although there are other evaluation methods available for cavitation classification, such evaluation based on the bearing temperature has the advantages of utilizing a low cost instrument and the ease of obtaining the measurements directly from the pump bearing.

**5. CONCLUSIONS**

In order to employ the results as a diagnostic tool within centrifuge pumps, it is important to clarify how the classification of the phenomenon’s intensity motivates corrective or preventive actions from users, be it in industrial or utility installations.

Based on numerous case studies, the practice recommends that the levels of approval and reaction time depend on the risks involved in the pump operation, such as maintenance costs, production losses or even accidents.

With the experimental results it is possible to develop an analytical decision-making tool where the bearing temperature confers additional reliability to the classification of moderate cavitation and provides alerts related to the immediate need for a corrective action, given the phenomenon’s intensity and destructive abilities as described above.

Based on Table (7), it is possible to determine the temperature increase range that corresponds to an alteration in the pump’s operation, as long as the observed magnitudes also present variations compatible with the operation mode with moderate cavitation.

The difference of 3 to 4 degrees centigrade above the bearing temperature when the pump is operating normally (without cavitation) reflects extra hydrodynamic exertion caused by continuous cavitation above incipient. Evidently, this increase corresponds to the experimental system, be it the pump, hydraulic installation or respective operational dynamics. Nonetheless, it still serves as reference to other systems provided there are parameters regarding the test conditions of the original equipment, supplied by the respective manufacturers upon request.

Temperature tests paralleled to performance tests are currently not common among manufacturers; however they can be requested and as such provided. With regards to the magnitude data, this is usually obtained from monitoring instruments already installed along with basic manometric calculations.

The use of temperature as an additional magnitude to characterize moderate cavitation is recommended to improve analysis reliability even with the difficulties inherent to this phenomenon.

**LIST OF SYMBOLS**

- $b_2$ width of the rotor’s vane
- °C centigrade
- $E_a$ axial thrust
- $E_r$ radial thrust
- $D_2$ rotor diameter
- $g$ gravity acceleration
- $H$ total head
- $H_d$ discharge head
- $H_s$ suction head
- $h_{fr}$ load loss at discharge
- $P_{fs}$ pressure at suction flange
- $P_{fr}$ pressure at discharge flange
- $P_a$ atmospheric pressure
- $P_{abs}$ absolute pressure
- $P_v$ vapor pressure
- $Q$ flow
- $Q_{bep}$ flow at the best efficiency point
- $s$ second
- $v$ speed
- $\rho$ density
- $\eta$ yield
- $\sigma$ cavitation index
- $\gamma$ specific weight
- LA attachment side
- LOA side opposite to attachment
The authors thank the Higher Education Personnel Training Coordination (CAPES) for financial support and scholarship for this study and experiment.

6. REFERENCES

ABSTRACT

The vortex emission known as von Karman street, is a characteristic phenomenon in flows around immersed objects for Reynolds number greater than 50. The stay vanes of low head Francis and Kaplan turbines are especially prone to this, as the stay vanes are slender. In these cases the von Karman shedding and the natural mechanical frequencies are near. Resonance might occur in this situation and, due to the high oscillation frequencies, fatigue problems may arise. Therefore, a careful study must be performed at early design stages to ensure that the vortex shedding does not excite the stay vanes natural frequencies. The present paper first validates the CFD calculation procedure comparing the results with experimental data and then describes the von Karman frequencies calculation and stay vane profile design through CFD for 2 cases: a new Francis project with \( n_q = 78 \) and a new Kaplan project with \( n_q = 160 \).

KEYWORDS: cavitation, centrifuge pumps, bearing temperature, experimental analysis, diagnosis.

1. INTRODUCTION

In order to minimize the energy losses of the hydraulic machines, the stay vanes dimensions, especially its width, tend to be minimized. In machines with high specific number (like low head Francis and Kaplan machines), this makes the stay vanes very slender bodies, as its height increases with the specific number.

In this scenario, the correct prediction and study of the dynamics loads generated by the von Karman vortex street is very important to avoid resonance and coupling effects as the shedding frequency may excite some natural mechanical frequencies whose values decreases the more slender are the stay vanes.

The stay vane profile design is made so as to separate the hydraulics (von Karman) and mechanical frequencies (both bending and torsional modes) and minimize or suppress the dynamics loads magnitudes caused by the vortex shedding. The process combines theoretical analysis, experimental data and CFD.

2. VON KARMAN VORTEX THEORY AND EXPERIMENTAL DATA

2.1. Classical theory

The stay vane vortex emission frequency may be studied by means of the classical Strouhal formula:

\[
f = \frac{S^* V_{ps}}{\delta} \quad (1)
\]

Where, \( S^* \) is the Strouhal number, \( V_{ps} \) is the free stream flow velocity at the separation point and \( \delta \) is the wake width at the separation points. The frequency depends on two factors:

- The Strouhal number, that depends on \( Re \) and the stay vane profile (especially the trailing edge shape); and is often determined by experiments.
- Wake width at the separation points including both the geometrical and displacement boundary layer thickness.

Theoretically, the exiting oscillating load amplitude induced by the vortex shedding can be determined following the Kutta-Joukowski theorem:

\[
F = \rho \Gamma V \quad (2)
\]

Considering that the circulation variation in one period is:

\[
\Delta \Gamma = \frac{V^2}{2f} \quad (3)
\]

The load amplitude per unit length is:

\[
F = \rho \frac{V^3}{2f} \epsilon \quad (4)
\]

The experimental coefficient \( \epsilon \) is introduced to consider the effect of the trailing edge shape on the amplitude.

Taking into account that the frequency has a linear variation with the velocity (1), formula (4) shows that the load amplitude varies with the square of the velocity.

This theoretical analysis helps understands the physics behind the phenomena but has serious shortcoming in the fact that the effect of the trailing edge shape on the frequency (Strouhal) and load amplitude and application point is not considered directly. That is one of the main reasons why CFD is used in the design process.

2.2. Trailing edge studies

Experimental studies over flat plates in [1, 2, 3] with several trailing edge shapes, showed that there is a heavy dependence between the vortex shedding and the trailing edge shape. The boundary layer over the trailing edge zone varies significantly with the different geometries and modifies the vortex shedding behavior.

3. CFD VALIDATION

In order to calibrate and validate the method, CFD simulations were compared against the test results presented in [1]. The results for the flat plate with blunt trailing edge are presented.
The numerical simulations were performed using the ANSYS CFX 13.0 SP2 software which, using a RANS scheme, solves for the pressure and velocity modeling turbulence thought an extra set of equations, as described in [4].

Given the heavy dependence of the phenomena on the near wall flow, the boundary layer was fully resolved and the simulations were performed using the SST and SAS turbulence models [7].

The CFD domain includes the measurement section extended in the trailing edge direction. A full Hexa mesh was constructed with Ansys ICEM to fulfill the requirements of the turbulence models (Figure 2). The trailing edge zone is specially refined, Figure 3.

The velocity of the points outside the boundary layer (with no variation) was considered the reference velocity.

### 3.1. Simulation and Results

Four velocities were simulated. The vortex shedding frequencies were obtained from the temporal variation of the load normal to the profile. As a check, the frequencies from the wake velocity were also obtained.

In all cases the von Karman street developed. Table 1 shows the simulated velocities and the frequencies obtained. The case with maximum velocity was simulated with both SST and SAS models. The difference between the frequencies obtained with both models is less than 1%.

[Table 1]: Flat plate CFD results.

<table>
<thead>
<tr>
<th>$U_\text{ref}$ [m/s]</th>
<th>$f_{\text{SST}}$ [Hz]</th>
<th>$f_{\text{SAS}}$ [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.88</td>
<td>119.30</td>
<td>-</td>
</tr>
<tr>
<td>6.05</td>
<td>188.28</td>
<td>-</td>
</tr>
<tr>
<td>12.68</td>
<td>404.49</td>
<td>-</td>
</tr>
<tr>
<td>16.73</td>
<td>538.02</td>
<td>543.38</td>
</tr>
</tbody>
</table>

The following figure shows the von Karman vortices over the flat plate wake for the maximum velocity case.

![Figure 5: Vorticity contour for $U_{\text{ref}}=16.7$ m/s.]

Between 300Hz and 500Hz approximately, a “lock-in” [5,6] between the exiting von Karman and the mechanical frequencies was observed in the test results. As a mechanical phenomenon is involved, the CFD frequencies cannot compared with the experimental in this range.

![Figure 6: Tests and CFD frequencies comparison.]

Outside this range, the frequencies obtained by CFD are in very good agreement with the experimental ones. The same CFD methods and techniques were used in the simulation of the stay vanes.

### 4. VON KARMAN CFD STUDY IN HYDRAULIC MACHINES STAY VANES

The following section describes the stay vanes von Karman vortex study with CFD for a Francis machine with $n_q=78$ and a Kaplan machine with $n_q=160$. The dimensions and characteristics of both projects made the von Karman vortex shedding study very important to avoid potential frequency coupling problems.
4.1. Stay vane geometry

The stay vanes profiles were designed and optimised by CFD to have a uniform flow at the cascade exit edge and to minimize the energy losses.

For this reason, the hydraulic profile of the stay vanes in the cascade is not unique.

There are 4 different profiles families in the Francis machine and 9 in the Kaplan, as shown in Figure 7. Each group has a particular maximum width, mean chamber line and, in some cases, chord.

Given this scenario, only the trailing edge region could be altered to have the desired von Karman behaviour (shedding frequency and load amplitude).

4.2. Trailing edge design strategy

Given the mechanical natural frequencies expected in both cases, the aim of the design was to achieve a safety margin between them and the shedding frequencies and also to have the lowest possible alternating load.

From [1, 3], a trailing edges with about 30° reports “no vibration” and “no vortex visible”, showing that the amplitude is very low or that no von Karman street develops. A similar configuration was therefore used in the stay vanes trailing edge designs.

4.3. CFD simulation

To consider the guide vanes cascade effect over the stay vanes cascade, the simulations included both components with the corresponding guide vanes opening to cover the whole machine operating range.

The inlet flow angle to the stay vanes cascade is very important to determine the correct von Karman frequency and the fluctuating load amplitude. Its value was taken from the complete spiral casing CFD simulations. As it is not the same for all stay vanes in the cascades, each stay vane group was simulated with its minimum and maximum flow angle.

Since the effects on the stay vanes height direction are not considered dominant, the simulations to determine the frequency were made in a Quasi-3D mesh, with two elements in the height direction. Also, due to the periodicity of the geometry group, the domain included only one stay vane and one guide vane (Figure 10).

These simplifications allow generating good quality meshes keeping a reasonable node number. Meshes were generated with Ansys ICEM and were Hexa-Tri (Francis case) or Hexa-Quad (Kaplan) with about 270,000 nodes per domain, fulfilling all requirements of the SST and SAS turbulence models, especially having y+ values smaller than 2 (Figure 9).

The simulation process was the same in all cases. First, a steady state simulation was run and then used as initial condition for the transient simulation. On the transient simulation a small perturbation (wall velocity) was introduced over the trailing edge during the first 500 time iterations. Without this perturbation, no vortex shedding develops in the simulation.

4.4. Frequency

The von Karman vortex shedding frequency was obtained from the temporal variation of the normal force and wake velocity, once the phenomenon was stabilized (Figure 11).

On the transient simulation a small perturbation (wall velocity) was introduced over the trailing edge during the first 500 time iterations. Without this perturbation, no vortex shedding develops in the simulation.

During the simulation, the behavior of several parameters was monitored. The most important were the force over the stay vane (normal and tangential to the flow direction) and the velocity and pressure in the wake.

4.4. Frequency

The von Karman vortex shedding frequency was obtained from the temporal variation of the normal force and wake velocity, once the phenomenon was stabilized (Figure 11).
The von Karman frequencies were compared against the mechanical natural frequencies for the first bending and torsional modes of the stay vanes. This comparison was made for each stay vane group. A safe enough separation to avoid dynamic coupling between the shedding and natural frequencies was achieved for the whole continuous operating range of the turbine.

4.5. Fluctuating load

In all simulations, once the perturbation was removed, the von Karman vortex developed. However, the shedding suffers damped in greater or lesser number of cycles depending on the shape of the profile and the flow conditions (mass flow and flow angle), disappearing completely in some cases. Figure 12 shows an example of the latter case.

![Figure 12: Normal force evolution in time. Once the perturbation is removed, the fluctuation damps after few cycles.]

The damping and low amplitude observed in the CFD results are consistent with [1-3], where no vortices were visualized on the flat plates with trailing edges similar to those used in the stay vane design, indicating that these were not formed or were of very low intensity to be captured during the tests.

Another factor that contributes to the damping of the vortices, as shown in [1,2,7], is the asymmetry of the flow in the trailing edge region and a displacement thickness of the order of trailing edge geometrical width.

The next figures show the velocity distribution on the trailing edge region for the stay vanes of the Kaplan case.

![Figure 13: Velocity contours over the trailing edge region of the Kaplan case stay vanes, on the left, “cambered” profiles, on the right, "straight" profiles.]

The load amplitude due to the von Karman vortex shedding determined by CFD is very dependent on the simulation conditions. To obtain reliable amplitude values is necessary to ensure a very low numerical damping level in the simulation. This was archived by carefully selecting the element type and density of the meshes as well as the simulation conditions, especially the turbulence model and discretization scheme for the equations to be solved by the code. The SAS [4] turbulence model was used in this cases along the lowest possible numerical damping of the CFX code.

5. CONCLUSIONS

The von Karman vortex shedding phenomena in the stay vanes of a Francis (nq=78) and a Kaplan (nq=160) hydraulic machines was studied by means of CFD simulations.

Both, the frequencies and load amplitude for the whole operation range were determined and the stay vanes trailing edge designed to ensure a proper separation between the von Karman and mechanical natural frequencies, avoiding potential coupling problems. A careful analysis of the results was made comparing them with the available experimental data and theoretical concepts.

Studies are underway to further improve and validate the CFD results, especially the fluctuating load prediction.

6. REFERENCES


Nomenclature

- $f$: Frequency [Hz]
- $S^*$: Strouhal number [-]
- $V$: Flow Velocity [m/s]
- $n_{11}$: Unit speed [-]
- $n_q$: Specific speed = $n_{11} \cdot Q_{11}^{0.5}$
- $Q_{11}$: Unit flow [-]
- $R_e$: Reynold number [-]
- $\delta$: Wake width [m]
- $\Gamma$: Circulation [m²/s]
- $\varepsilon$: Amplitude coefficient [-]
- $\rho$: Density [kg/m³]
1. INTRODUCTION

Turbomachines are those that absorb energy from a fluid and generally restitute mechanical energy at the shaft. Alternatively, absorb mechanical energy at the shaft and restore the energy to a fluid. The fluid may be a liquid or a gas. The mechanical energy and fluid exchanger is endowed with rotatory motion; hence the word “turbo” or in Latin Turbinis, means “the rotating machine”. The fundamental Euler equation of turbo machines, based on the theorem of angular momentum, is basic for the study of these machines [1]. Hydraulic turbines play a major role in nowadays stage since the development of new technologies of sustainable energy such as solar, tidal or wind, but those technologies will be useful, perhaps, in the medium and long-term application.

Meanwhile, hydropower is a great source of a clean worldwide energy in a short and a medium term application [2]. For this reason, it is important to continue developing maintenance procedures and related diagnostics techniques in order to keep it clean, safe and under-controlled.

These machines have a certain type of operating problems such as cavitation [3], stagnation, pressure fluctuations, among many more; which generate losses and declines in both mechanical and hydraulic performance. Therefore is significant for industries to keep all monitoring the least invasive and as remote as possible [4], [5].

Pressure fluctuations, as those derived from Von Karman streets, could become a very dangerous phenomenon. If the frequency of the pressure pulsations developed during the vortex formation coincide with one of the natural frequencies of the machine, a hydroelastic coupling takes place increasing the fatigue failure risk. Severe damage in generating units have been reported by Egusquiza [6] and Finnegan [7]. Von Karman theories and their implications for the vibrations have been studied previously, particularly in numerical analysis [8]–[10]. New theories have been developed for monitoring [11]–[14]; but it remains complicated to ensure convergence and numerical stability. Thus, a fundamental study case, which was already considered in the literature [15] is solved using numeric methods. It is a NACA profile, normally used as guide vanes of Francis type machines. In order to validate the proposed methodology, the hydroelastic behavior of the guide vane under different working regimes was considered.

2. MATERIALS AND METHODS

In order to perform the analysis drawn on a typical guide vane of turbomachinery, it was needed a flow cross-section area 150 mm wide, by 150 mm high, with a total length of 750 mm, as shown in Figure 1. Same configuration was previously studied by Ausoni [15] and details of the experimental setup can be found in the work of Zobeiri [10]. The guide vane was in all the cases oriented horizontally. In the experimental tests conducted by Zobeiri flow rates ranging from 14.0 to 17.0 m/s were used [10]. Same speeds are used for this numerical analysis. In addition, pressure was regulated in order to avoid cavitation in the cores of the vortex.

Numerical simulation of such situation is a challenging. Multi-physics setups may or may not develop the target phenomenon and, even, if it develops, convergence are not fully guaranted. To accelerate the numerical solution, an initial deformation was imposed on trailing edge. This deformation, equivalent to a Dirac, is described mathematically in Equation 2. The Dirac disturbance pretends to excite the natural frequencies of the guide vane and therefore induce the hydroelastic coupling if the flow conditions are conducive for it.
2.1. Guide Vane Geometry

The geometry of the guide vane can be observed in Figure 2. It is based on the 0009 standard NACA - 7.8 45 / 1.93 Structural Steel [16], originality conceived to reduce drag in airfoils. However, in this case study (see Figure 2), the trailing edge was truncated, in order to be in-line with the experimental works reported by Aussoni [15] and Zobeiri [10]. It is well known that such modified profile generate a hydro-elastic coupling when tested in a high-speed cavitation tunnel.

The reported range of speeds, where the phenomenon appears, is between 13 and 17 m/s [10]. For such reason, same ones are imposed as simulation speeds in this work.

2.2. Numerical Analysis

For the numerical analysis it is used Finite Element Method (FEM), and the solver for this study is the Software Elmer [17], which allows a multi-physics problems; allowing great control of the solution methods.

Other Software used was Gmsh [18], which aids generating all meshes used in the analyses.

Due to the complexity of the problem it is necessary to use a significant computing power, which was provided by APOLO Scientific Computing Center of EAFIT University [19]. It has an installed capacity of 420 nodes.

3. Calculation

For numerical analysis water and the steel were used. Steel was used for the guide vane and water is the fluid study. Their properties are found in Table 1.

<table>
<thead>
<tr>
<th>Table 1: Material Properties [17].</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
</tr>
<tr>
<td>------------------------------------</td>
</tr>
<tr>
<td>Density [kg/m³]</td>
</tr>
<tr>
<td>Viscosity [Pa*s]</td>
</tr>
<tr>
<td>Speed of Sound [m/s]</td>
</tr>
<tr>
<td>Meshing Elastic Module[Pa]</td>
</tr>
<tr>
<td>Meshing Poisson</td>
</tr>
<tr>
<td>Density [kg/m³]</td>
</tr>
<tr>
<td>Speed of Sound [m/s]</td>
</tr>
<tr>
<td>Poisson</td>
</tr>
</tbody>
</table>

As proved in previous studies [4],[10], vortices can be simplified to a 2D model if a hydro-elastic coupling appears. If this does not appear, the vortices are generated in a 3D distribution [15]. Since our interest is to analyze the hydro-elastic coupling, a 2D model was used as shown in Figure 2. This approach can be extended to a 3D analysis for further and more complex studies.

The following boundary conditions, referenced in Figure 3, are keep for all simulations:
- On the upper and lower edges, "Wall" edge condition was applied.
- At the right edge speed was imposed on the y-axis with a value of $V_y = 0.0$, because this is the outlet edge.
- On the left edge, which serves as inlet, the speed was imposed on the x-axis with a parabolic distribution, shown in Equation 1.

$$\delta_y = 1 - \frac{x}{50}$$

Equation 1, Speed distribution.

Where the variable $y$ is the vertical coordinate value of each point, and $V_y$ is the desired velocity value in the range between 13 and 17 m/s.

- Over all the edges of the guide vane, the so-called Fluid Structure Interaction (FSI) condition is imposed. It allows for the coupled fluid-structural analysis. These edges have an "update mesh" setup as well, so that nodes of both of the bodies (structural and fluid) suffer the same displacements.

An important aspect is the step time. Because of the Nyquist's Law [21] the simulation step time should be maximum the half of the phenomenon period. For that reason, the simulation step time used was 0.001 s, which in the experimental arrangement would correspond to a sampling frequency of 1.0 kHz. Moreover, a total of 60 seconds were simulated. Initial conditions are crucial getting faster solution convergence rates. Two of them were imposed:
- The first one was that the initial speed of all the fluid has a parabolic distribution as shown in Equation 1.
- The second one refers to the guide van. It imposes an initial displacement along the axis as shown in Equation 2.

$$V_x = V_i \times \left(1 - \frac{y^2}{5626}\right)$$

Equation 2, Guide Vane initial displacement.

American Journal of Hydropower, Water and Environment Systems, august 2015 39
Where the variable $x$ is the value of the coordinate in the $x$ axis of each point, and $\delta_y$ is the value of the resulting initial displacement. The mesh used for all simulations was the same. A first order triangular mesh was generated in Gmsh [18], which has 640,808 nodes, surface elements 1,277,684 and 1,918,492 boundary elements, it can be observed in Figure 6.

4. RESULTS

After several simulations, it reached a similitude between the real and the simulated phenomenon, which yielded reliable results, as it is shown in Figure 4. Vortex was generated, and the pressure distribution associated to these vortices was as expected. It is qualitatively close to the experimental works. As can be observed in Figure 5, after the generation of vortices, pressure waves travel and hit the pipe walls. Pressures on the wall were monitored in all simulated cases and presented in the Figure 7; actually, higher pressure fluctuations, which are important for diagnostics purposes in turbomachinery issues, are seen at a distance of once the span provided that the vortex is completely detached. The pressure behavior can be gathered into well-defined groups. The first collecting speeds below 16 m/s, and the second one those over 16 m/s. The latter generates higher pressure differences due to the vortices appeared in the second group. This difference in behavior is attributed to the structural-fluid coupling, as proposed by Zobeiri [10].

5. CONCLUSIONS

• Using a numerical simulation it is possible to reproduce the generation of vortices on a turbomachine guide vane.
• The introduced approach leads to an accentuated convergence of the simulation and fast development of the structure-fluid interaction phenomenon.
• It was determined that a length equal to the length of the guide vane after it, is the highest-pressure fluctuations.
• It is recommended to perform simulations with smaller speed intervals to understand the behavior of the two groups of data found.
• The boundary layer is very important in the generation of vortices, therefore the mesh in this area must be as clean and small as possible.
• Numerical methods can support monitoring theories to find the optimal data collecting locations.
• Preliminary results revealed regions experimenting higher pressure fluctuations once the vortices appeared. It can be implemented in more complex simulations to help finding fatigue sensitive points on the turbomachine, during operation and even throughout the design process.

6. REFERENCES


Acknowledgements
The authors thank in a very special way, the center of scientific computing Apolo [19], who supported the project and EAFIT University [22] who finances our research.
EROSION PREDICTION
BASED ON ILES METHOD

1,2*Hidalgo, Victor, 1Luo, Xianwu, 2Valencia, Esteban, 2Aguinaga, Alvaro, 1,2 Cando, Edgar

ABSTRACT

The present paper deals with a method to find the more probable eroded areas produced by the collapse of the cloud of bubbles under unsteady cavitating flows conditions. In order to improve this erosion prediction, numerical simulations of partial cavitation have been carried out by using OpenFOAM, the implicit large eddy simulation method (ILES) and the cavitation model of Zwart. Results show that the affected area is related with the pressure gradient as a cavitation-erosion index, which has been validated by experimental results from previous studies from the erosion test at the Université Grenoble Alpes (LEGI).

KEYWORDS: Cavitation, Erosion, ILES, OpenFOAM, Numerical Simulation.

1. INTRODUCTION

According to Franc [1], pits are the result of the plastic deformation in the material for an interval of time due to the collapse of a cloud of bubbles, then the erosion is clear observed [2]. Hattori et al. [3] proved that the impact loads of the bubbles collapsing could be used to predict the pit incubation. In addition, experimental studies have given important information, which have been used to propose mathematical cavitation-erosion models [4]. However, it is very expensive to run experimental tests and measuring equipments have limitations, so that, CFD shows to be the answer to improve the erosion prediction.

Based on previous premises, Nohmi et al. [5] have proposed four equations to predict the aggressive erosion by using CFD, they considered that the vapor volume fraction, α, should be more than zero, its time derivative, ∂α/∂t, less than zero and the pressure, p, bigger than vapor pressure, pV. Then, Li [6] based on Nohmi ideas worked with different erosion aggressive indexes for 3D cavitation, results shows that the time derivative of pressure, ∂p/∂t, is an important parameter to predict areas that could be damaged by unsteady cavitating flows. Other works have been proposed based on Li’s index an the total time derivative of pressure rather than partial derivation [7] by using RANS methodology and Fluent software. Though these indexes have a good accuracy with experimental results, RANS turbulence method presents limitations to capture the cavitation phenomena, due to the turbulence is modeled, special changes should be doing to enhance the numerical simulation performance [8]. In context, large eddy simulation (LES) [9] captures the essential part of the attached cavity sheet and the horseshoe cloud breaking off. In this way, improving the aggressive erosion index calculation.

On the other hand, Bensow [10] has demonstrated that the numerical results obtained by OpenFOAM, and implicit large eddy simulation (ILES) rather than traditional explicit LES methods, present better accuracy with experimental results, because the excessive dissipation has been prevented by avoiding the explicit subgrid model [11].

In this framework, the challenge of the present research is to find an index based on ILES to predict the damaged areas produced by unsteady cavitating flows.

2. PHYSICAL DESCRIPTION

2.1 Multiphase flows consideration

Mixture homogeneous flows are the basis of the present numerical study and expressed in Equations from (1) to (3).

\[ \alpha = \frac{\forall V}{V}, \]  
\[ \rho = (1 - \alpha)\rho_L + \alpha\rho_V, \]  
\[ \mu = (1 - \alpha)\mu_L + \alpha\mu_V, \]

where \( \alpha \) is the vapor volume fraction, \( \forall \) is the volume, \( \rho \) is the fluid density, \( \mu \) is the dynamic viscosity, \( L \) and \( V \) are the subscripts for liquid and vapor respectively.

2.2 Mathematical Considerations

The filtered equations of continuity and momentum are used in the numerical simulation as indicated in Equations (4) and (5).

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0, \]
\[ \frac{\partial (\rho u_i u_j)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial \tau_{ij}}{\partial x_i} + \frac{\partial (\tau_{ij} - \tau'_{ij})}{\partial x_j}, \]

where \( \tau_{ij} = \rho u_i u_j \) is the product of the filtered velocities, \( \tau'_{ij} = 2\rho \nu S_{ij} \) is the filtered viscous stress tensor, \( S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \) is the filtered strain tensor rate, \( \tau'_{ij} = \rho u_i u_j - \rho u_i u_j \) is the Reynolds stress tensor and is the kinematic viscosity. In ILES the large eddies of \( \tau'_{ij} \) are calculated and the small eddies are modeled as dissipative actions equivalent to the subgrid scale actions [10, 11].

Finally it is consider the vapor volume fraction into the continuity equation to get the transport Equation (6), which is related to the interphase mass transfer rate per volume unit, \( \dot{m} \).

\[ \frac{\partial (\alpha \rho_V)}{\partial t} + \frac{\partial (\alpha \rho_V u_i)}{\partial x_i} = \dot{m}. \]
2.3 Zwart Cavitation Model

The cavitation model of Zwart based on equations of Rayleigh-Plesset were implemented in OpenFOAM for the numerical simulation, as indicated in Equation (7).

\[
m^+ = \begin{cases} 
  \frac{3}{2} \left( \frac{F_v}{R_B} \right) \left( \frac{p_v - p}{\rho_L} \right) \left( \frac{1}{3} \right) & \text{if } p < p_v \\
  -\frac{3}{2} \left( \frac{F_c}{R_B} \right) \left( \frac{p_v - p}{\rho_L} \right) \left( \frac{1}{3} \right) & \text{if } p > p_v
\end{cases},
\]

where \( F_v = 300 \) and \( F_c = 0.03 \) are the selected calibration constants for vaporization and condensation \([12] \), \( r_{nuc} = 5.0 \times 10^{-6} \) is the nucleation site volume fraction and \( R_B = 1.9 \times 10^{-6} \) [m] is the typical bubble size in water \([9] \).

2.4 Cavitation condition

The cavitation condition is based on the cavitation number \( \sigma \) equal to 0.9. It has been calculated by Equation (8) in the experiment and by Equation (9) in the present study.

\[
\sigma = \frac{p_d - p_v}{p_u - p_d}, \tag{8}
\]

\[
\sigma = \frac{p_d - p_v}{0.5 \rho U^2}, \tag{9}
\]

where \( p_d \) is the pressure downstream, \( p_u \) is the pressure upstream and \( U \) is the reference velocity.

3. COMPUTATION DOMAIN AND MESHING

The idea of the erosion test disk from the study of incubation and cavitation erosion rate of work-hardening materials \([1] \) at the Université Grenoble Alpes (LEGI) have been used in this research. The main issue of the test is that the water flows to the inlet at 47.13 [m/s] inside of a round pipe (16 [mm]), then it is abruptly conduced to a small space between to circular planes equal to 2.5 [mm], as indicated in figure 1 (a) with a downstream pressure equal to 10 [bar] at the outlet.

Inlet, outlet, wall and symmetry conditions are shown in figure 1 (b), which is the simulation domain, and a quart part of the disk used by Franc et al. \([1] \). This figure has a structured mesh with 473490 elements. The mean calculated yPlus around the wall of the disk is 5.0, which matches ILES requirements for meshes and based on Equation (10).

\[
y^+ = \frac{u_2 y}{v}, \tag{10}
\]

where \( y^+ \) is the yPlus, \( u_2 \) is the friction velocity at the nearest wall and \( y \) is the distance to the wall.

4. RESULTS AND DISCUSSION

The time comparison study between experimental result and simulation is not possible to carry out, due to the limited experimental data available and measuring equipment. In this way, dimensionless parameters, based on Equation (11) are used to understand the partial cavitation behavior and the collapse of the cloud of bubbles in a typical cycle, as shown in figure 2.

\[
\xi = \frac{t - t_o}{t_f - t_f}, \tag{11}
\]

where \( \xi \) is the dimensionless parameter, \( t \) is the time, \( o \) and \( f \) are the subscripts for initial and final time respectively.

Figure 2 shows that there is a ring of bubbles at \( \xi = 0 \) with \( \alpha \) similar to 0.9, which means that the vapor is more gas than liquid without phase changes. The growth of the cavity sheet and detachment is shown from \( \xi = 1 \) to \( \xi = 2/3 \), but the reentrant jet effects are observed from \( \xi = 1/2 \) to \( \xi = 2/3 \). The last part of the cycle is the cloud of bubbles breaks off at \( \xi = 5/6 \), and its collapse at \( \xi = 1 \), though the probable affected zone is shown in this part, the vapor fraction includes unnecessary areas, which are not affected for the collapse of bubbles.

According to the collapse time of figure 2, the pressure, \( p \), and its gradient, \( \nabla p \), in z direction, \( \partial p/\partial z \), have been plotted with logarithm scale in figure 3 at \( \xi = 6/7 \) to find the affected zone. The \( p \) in (a) shows a zone that does not match the experimental result in (c), but the \( \partial p/\partial z \) in (b) matches (c) and indicates that the mean difference is about 7.5% to 11.76% (based on D, which is the diameter of the round pipe). This affected zone is located on the disk base when the \( \partial p/\partial z = 1 \times 10^{10} \) [Pa/m], the zone near to the round pipe is ruled out when the main phenomena occurs \([5] \).
5. CONCLUSIONS

A numerical simulation with ILES method was performed in this research and validated by experimental results from LEGI. Therefore, the following milestones are concluded:

1. The ILES method and the Zwart cavitation model in OpenFOAM gave results for erosion prediction similar to experimental results between 7.5% to 11.76% based on the pressure gradient, which could be considerer as an aggressive erosion index.

2. The erosion is caused after the cavity sheet breaks off and when the cloud of bubbles starts to collapse.

6. REFERENCES


Acknowledgement

This work was supported by National Polytechnic University of Ecuador (EPN) with collaboration of Tsinghua University - China.
ABSTRACT

Fish passes have been built in Brazil in order to mitigate the adverse effect of hydropower dams on the movement of migratory fish. The effectiveness of these mechanisms depends on the successful upstream and downstream passage, the latter a fundamental aspect that has been neglected. The implementation of mechanisms into the dam structure, which facilitate or allow the passage downstream, represent a major challenge to the engineering applied to biodiversity conservation in Brazil. The aim of this study is to evaluate the international and Brazilian experiences on downstream passage of eggs, larvae and adults, as well as the main challenges to be faced. Alternatives to improve downstream fish passage would contribute to match hydropower generation with the conservation of the Brazilian fish fauna.

KEYWORDS: dams, fish pass, migration, ichthyoplankton.

1. INTRODUCTION

Practically all rivers in Brazil have been dammed or are under the influence of dams [1], as the construction of hydroelectric plants has increased considerably in recent decades [2]. Despite their importance for industrial and economic development, hydroelectric plants are responsible for severe and irreversible changes to the natural hydrologic regime of rivers, while also altering the quality and availability of habitats as well as the entire complex functioning of communities at those sites [1,3]. Given the high diversity and presence of migratory species in Brazil [4], impacts on the ichthyofauna are of great relevance and are regarded as one of the most delicate issues when licensing new hydroelectric power plants. [5-7]. Due to their size [7,8] and greater abundance [9], migratory fish are most appreciated by professional [7,10-12] and recreational fisheries [7].

Migratory movements in southeast Brazil, known as "piracema" includes several displacements [13]: during the flood period migrating adults move from feeding sites for breeding areas, located usually upstream; eggs and larvae from breeding are carried toward the floodplains; adults return to the feeding sites; young individuals from the floodplains in a following flood move toward the river or small tributaries. The formation of reservoirs directly impacts migratory fish as they block the communication between the habitats required in their life cycle [14].

In order to attenuate this adverse effect on the movement of migratory fishes, Fish Passes have been installed since the early 20th century [15]. However, despite the increased popularity of these mechanisms, most of them were never monitored for their efficiency in maintaining migratory species stocks. Some of the evaluated mechanisms were considered ineffective, while others may be resulting in the collapse of local populations. Part of that failure can be attributed to a lack of downstream fish passages, including passage of eggs and larvae through the reservoir [16].

In order to reach areas downstream the dam, fish must be able to cross the reservoir and pass through the turbines or spillway gates. In Brazil there are no specific mechanisms to facilitate the downstream passage, and this topic must be prioritized to minimize the impacts of dams on fish.

This paper deals with Brazilian and international experience on downstream passage of eggs, larvae and adult fish, as well as the main challenges to be faced.

2. MATERIAL AND METHODS

Literature review was conducted with the main bibliographic databases to identify international experiences on downstream fish passages, as well as the Brazilian experience with regard to fish passage, including adults, eggs and larvae. Based on these information, it was identified the main challenges and knowledge gaps to be faced in order to better enable compatibility between power generation and maintenance of the movements of migratory species in Brazil.

3. RESULTS AND DISCUSSION

3.1 INTERNATIONAL EXPERIENCES

Downstream passage has been the main concern of recent studies on fish passage in North America and Europe [17], since problems associated with damage caused by passage through turbines and spillways have been considered the main factors affecting the maintaining of migratory fish populations [18].

The fish mortality during downstream migration can be reduced by the construction of specific downstream passes or through modifications in the structures of the dam, in order to reduce possible injuries to fish [19]. The use of structures to minimize this problem is relatively new, and the main reports dating back just over 50 years [20]. However, only in recent decades their use started to become popular in North America.

Different apparatus to facilitate downstream fish passage have been proposed: physical barriers that prevent fish from entering the turbines; behavioral barriers that guide fish (attract or repel) through some stimulus to the spillways; specific systems for descent; and bypass systems, installed at the water intake of the turbines [19]. The latter has been used in numerous large projects in North America (the Columbia River and its tributaries), specially associated to large turbines.
(300 to 600 m3/s flow) [21]. Screens of up to 12 meters have been installed in the upper portion of the water intake of the turbines to divert the fish to a bypass system, which directs them to the tailrace, or to a transport mechanism. However, no downstream passage device has proved 100% effective. Even well-designed bypass systems protect only a small portion of the fish [22]. For this reason, numerous studies have been conducted to maximize the survival of fish passing through turbines [18,23]. Sudden variation in pressure, shock and friction on the blades, disorientation due to high turbulence in the tailrace and consequent increased susceptibility to predators are the main causes of death or injury of migratory fish as they pass through the turbines [18]. Studies conducted with salmonids indicate that mortality rates during the passage through turbines, vary from 0% to 100% in Francis turbines [20] and is rarely below 10% [24], and range from 0 to 90% for Kaplan turbines, usually between 5% and 20%, with a mean of 15% [20].

Mortality during passage through the spillways may also occur due to the shock when the dam height is considerable, or by supersaturation of gases. This shock may occur against the dam structures, or against the bed and banks of the river. Proper design of the spillway can significantly reduce the frequency of injuries, that depend also on the hydraulic conditions of the dissipation basin and tailrace [20]. The maximum dam height to allow maximum survival of Pacific salmon was estimated to be 21-40 m for young (15-18 cm long) and approximately 13 m for adults (greater than 60 cm) [25]. Above this height, mortality varies among species, location and height of the dam spillway gate. Nevertheless, studies indicate that anadromous species have 98% chance of survival when subjected to a free fall of 90 m [25]. Moreover, nitrogen supersaturation is related to the dam height and flow characteristics as well [26].

Another important issue related to the impact of hydroelectric is related to effect of the reservoir and the dam on the downstream passage of the ichthyoplankton populations. Significant decline of the ichthyoplankton population, associated with high rates of natural mortality may have significant long-term effects in adult populations [27]. However, the downstream migration in North America and Europe is primarily accomplished by juveniles and adults, little emphasis has been given to the possibility of passage of eggs and larvae.  

3.2 THE DOWNSTREAM PASSAGE IN SOUTH AMERICA

The downstream migration has been widely neglected in South American countries, ignoring the fact that the return of adults, as well as the passage of eggs and larvae by the reservoir toward the downstream areas, is a key factor to the long term maintenance of migratory populations [16]. Hence, the passages should ideally restore the free flow through the dam in both ways: downstream and upstream.

For adult fish, the few studies that evaluate the phenomenon indicate that the descending passage is far more limited than the ascending [28-30], or even absent [28,31,32]. These early indications show the urgent need to assess the descending passage of adults, since there is the possibility that the phenomenon was not happening in many power plants.

In the case of the downstream passage eggs and larvae through the reservoir, recent assessments indicate that large reservoirs can profoundly alter the spatial distribution of ichthyoplankton, with evidence that the eggs and larvae disappear in lentic portions of the dam, not reaching the dam [28,33,34]. Only in small reservoirs, where the residence time of the water is small, the passage, with high selectivity, seems to be possible [16]. Spawning during the rainy season when the rivers have turbid water is an adaptation of migratory species to protect eggs and larvae [35]. However, when these initial forms, along the downstream migration reach a reservoir with high transparency and a large number of small predators, its passage is unlikely to happen [36]. This is particularly critical when the spawning grounds are located just upstream of the reservoir and the floodplain (nursery areas) can be found only downstream of the dam. In these cases, the maintenance of the population depends directly on the success of this passage [16].

3.3 FUTURE PROSPECTS

Considering the relevance and the challenges related to this issue on the migratory fish fauna conservation, some questions should be priority answered:

- What is the maximum size of the reservoir (length, residence time, etc) able to allow the downstream passage of the ichthyoplankton?
- Are there ways (flow manipulation, spillway operation, etc) to promote an increase in the success of implementing downstream passage through the dam?
- Specific downstream passage devices can be applied in Brazil? At what cost?

The answer to the first question depends on studies assessing the issue for different reservoirs with different dimensions and flow conditions, including the presence or not of large tributaries upstream. Current estimates indicate that this threshold may be a reservoir between 10 and 30 km², and residence time of less than one week [16,34], values that need to be checked for different basins.

The simplest mechanical technique to increase the downstream fish passage with less injuries on the fish consists in promote spills, which is particularly appropriate when the downstream migration period is short and occurs during high flow rates [37]. In this case, increase knowledge on the spatial and temporal distribution of fish immediately upstream of the dam is essential in order to maximize the cost/benefit relationship of this operation.

4. CONCLUSION

Development of alternatives in order to facilitate or allow the downstream passage represents a major challenge to the engineering applied to the conservation of the aquatic biodiversity in Brazil. Solutions in this sense would certainly contribute to a better harmonizing between hydropower and the conservation of the Brazilian migratory fish fauna.

5. BIBLIOGRAPHY

America Latina: alguns comentários”, COPESCAL Documento Ocasional, pp.1-17.