AMERICAN JOURNAL OF HYDROPOWER, WATER AND ENVIRONMENT SYSTEMS

Technical Papers

- 06** PUMPED HYDROELECTRIC ENERGY STORAGE SYSTEMS (PHES): CHALLENGES AND PERSPECTIVES IN THE BRAZILIAN ENERGY SCENARIO
Raimundo, Danielle Rodrigues; Filho, Geraldo Lúcio Tiago
- 11** HYDRODYNAMIC BEHAVIOR ON KAPLAN TURBINES: EXPERIMENTAL METHODOLOGY FOR THE EVALUATION OF CAVITATION DEVELOPMENT BASED ON DYNAMIC MEASUREMENTS
Rivetti, Arturo L.; ANGULO Mauricio A.; LUCINO Cecilia V.; BOTERO Francisco.; LISCIA, Sergio O.
- 16** TECHNICAL AND ECONOMIC EVALUATIONS FOR DECISION-MAKING ON THE REPOWERING OF SMALL HYDROPOWER PLANTS AND THE OPTIMUM EXPLOIT OF EXISTING PLANTS IN BRAZIL
Oliveira, Marcos André
- 25** EVALUATION OF HYDROELECTRIC, WIND AND SOLAR POTENTIAL: A CASE STUDY OF EXISTING PLANTS
Filho, Wilson Pereira Barbosa; Silva, Livia Maria Leite da; Silva, Nathan Vinícius Martins da; Oliveira, Karina Aleixo Benetti de; Abreu, Anna Luisa de Oliveira; Swiatovy, Gabriel Hepp
- 33** PREDICTION OF PRESSURE PULSATION IN FRANCIS TURBINES USING RANS SOLUTION: AN APPLIED STUDY OF EFFECTS OF THE TURBULENCE MODELS, MESH, DOMAIN EXTENSION AND MULTIPHASE FLOW
Marra, João M.; Gramani, Liliana M.; Zubeldia, Luiz F.; Kaviski, Eloy

IAHR DIVISION I: HYDRAULICS
TECHNICAL COMMITTEE: HYDRAULIC
MACHINERY AND SYSTEMS

Editors in Chief

Prof. Geraldo Lucio Tiago Fº - UNIFEI
Prof. Eduard Egusquiza - UPC

Executive Editors

Prof. Carlos Martinez - UFMG
Eng. Humberto Gissoni - VOITH

Technical Editors

Prof. Regina Mambeli Barros - UNIFEI
Prof. Cecilia Lucino - UNLP

Journalist in charge

Adriana Barbosa MTb - MG 05984JP

Graphic Project/ Diagramming

Lidiane Silva

Circulation

1,000 copies

Federal University of Itajubá - UNIFEI
Av. BPS, 1303 - Bairro Pinheirinho
Itajubá - MG - Brasil - CEP: 37500-903

Catalographic card prepared by Mauá Library -
Librarian Margareth Ribeiro - CRB_6/1700

R454

American Journal of Hydropower, Water and Environment
Systems, LAWG-IAHR, v.1, 2014 - Itajubá: LAWG-
IAHR, 2015 - v.6, jan. 2018.

Trimestral,
Editors Chief: Geraldo Lúcio Tiago Filho/Eduard Egusquiza
Journalist in Charge: Adriana Barbosa - MTb_MG 05984
ISSN 2317-126X

1. Hydropower. 2. Water. 3. Environment Systems.
II. Latin American Working Group

LAWG-IAHR Executive Secretariat
Lucia Garrido Rios
latinahrsecex@gmail.com

AMERICAN JOURNAL OF HYDROPOWER, WATER AND ENVIRONMENT SYSTEMS

Editorial

Demand for electricity is increasing rapidly in Latin America. Expanding and enhancing the current power generation infrastructure is thus a major concern for the region. In that context, renewable energy is bound to play a relevant role in the coming years and hydroelectric energy especially so in view of its relatively unexploited potential, which is estimated in around 690,000 MW in the region.

Our mission is to contribute to the regional integration by means of the production of knowledge in the field of hydroelectric power generation and hydromechanical systems, with focus on technological advance and the rise of quality standards, but with an eye on the priorities of a region where, well into the XXI century, millions of people find themselves deprived of electricity. That includes the topics such as the evaluation of the development of projects of low environmental impact, the role of hydroelectric powerplants in the face of the diversification of the energy matrix as well as mini- and small-hydro projects to meet the demand of areas with no access to the electrical network.

Three symposia have been held and six editions of this Journal have been published since the creation of this, the Latin Group, in 2012. Added to that are postgraduate courses and exchange programs that are being organized at present. Mexico, Brazil, Ecuador, Colombia, Uruguay and Argentina are the most active members of the Group and it is our goal to get more involved in the future.

We firmly believe in the ability of our universities to conduct the research in the field of hydraulic turbomachinery and that this knowledge will have a beneficial impact on the policy making of our countries, whether on matters related to existing infrastructure as well as the development of long-term projects and strategies.

We understand that keeping and strengthening the link between the nations currently participating is a major challenge in the framework of a constantly evolving reality, seen reflected in the availability of economic resources, the technological tools and the swings of institutional management. May this new edition of the Journal serve as evidence that, however difficult, such challenge is achievable.

Eng. Sergio O. Liscia
Presidente Latin Group 2017-2021

ISSN 2317-126X

INSTRUCTIONS FOR AUTHORS

American Journal of Hydropower, Water and Environment Systems

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

Margin Position	Top	Bottom	Left	Right
Margin size (cm)	2.0	2.5	2.0	2.0

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

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

Introduction

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

Material and methods

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

Theory/calculation

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

Results

Results should be clear and concise.

Discussion

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

Conclusions

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

References

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

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

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

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

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

(1) Reference to journal articles and papers in serial publications should include:

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

(2) Reference to textbooks and monographs should include:

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

(3) Reference to individual conference papers, papers in compiled conference proceedings, or any other collection of works by numerous authors should include:

- last name of each author followed by their initials
- year of publication
- full title of the cited paper in quotes, title capitalization
- individual paper number (if any)
- full title of the publication
- initials followed by last name of editors (if any), followed by 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] Ning, X., and Lovell, M. R., 2002, "On the Sliding Friction Characteristics of Unidirectional Continuous FRP Composites," *ASME J. Tribol.*, 124(1), pp. 5-13.
- [2] Barnes, M., 2001, "Stresses in Solenoids," *J. Appl. Phys.*, 48(5), pp. 2000-2008.
- [3] Jones, J., 2000, *Contact Mechanics*, Cambridge University Press, Cambridge, UK, Chap. 6.
- [4] Lee, Y., Korpela, S. A., and Horne, R. N., 1982, "Structure of Multi-Cellular Natural Convection in a Tall Vertical Annulus," *Proc. 7th International Heat Transfer Conference*, U. Grigul *et al.*, eds., Hemisphere, Washington, DC, 2, pp. 221-226.
- [5] Hashish, M., 2000, "600 MPa Waterjet Technology Development," *High Pressure Technology*, PVP-Vol. 406, pp. 135-140.
- [6] Watson, D. W., 1997, "Thermodynamic Analysis," *ASME Paper No. 97-GT-288*.
- [7] Tung, C. Y., 1982, "Evaporative Heat Transfer in the Contact Line of a Mixture," Ph.D. thesis, Rensselaer Polytechnic Institute, Troy, NY.
- [8] Kwon, O. K., and Pletcher, R. H., 1981, "Prediction of the Incompressible Flow Over A Rearward-Facing Step," *Technical Report No. HTL-26, CFD-4*, Iowa State Univ., Ames, IA.
- [9] Smith, R., 2002, "Conformal Lubricated Contact of Cylindrical Surfaces Involved in a Non-Steady Motion," Ph.D. thesis, <http://www.cas.phys.unm.edu/rsmith/homepage.html>

1.1.2 Essential title page information

- **Title.** Concise and informative. Titles are often used in information-retrieval systems. Avoid abbreviations and formulae where possible.
- **Author names and affiliations.** Where the family name may be ambiguous (e.g., a double name), please indicate this clearly. Indicate all affiliations with a number immediately after the author's name and in front of the appropriate address. Provide the full postal address of each affiliation, including the country name and, if available, the e-mail address of each author.
- **Author résumé.** The author must inform the graduation degree, post graduation, affiliation and email address. The résumé must not exceed 150 characters.
- **Corresponding author.** Clearly indicate who will handle correspondence at all stages of refereeing and publication, also post-publication. Ensure that e-mail address and the complete postal address are provided. Contact details must be kept up to date by the corresponding author.
- **Present/permanent address.** If an author has moved since the work described in the article was done, or was visiting at the time, a 'Present address' (or 'Permanent address') may be indicated as a footnote to that author's name. The address at which the author actually did the work must be retained as the main, affiliation address. Superscript Arabic numerals are used for such footnotes.

Abstract

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

Keywords

Immediately after the abstract, provide a maximum of 6 keywords, using American spelling and avoiding general and plural terms and multiple concepts (avoid, for example, 'and',

'of'). Be sparing with abbreviations: only abbreviations firmly established in the field may be eligible. These keywords will be used for indexing purposes.

Abbreviations

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

Acknowledgements

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

Nomenclature and units

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

Math formulae

Present simple formulae in the line of normal text where possible and use the 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:_____

IAHR DIVISION I: HYDRAULICS
TECHNICAL COMMITTEE: HYDRAULIC
MACHINERY AND SYSTEMS

Scientific Committee

Alexandre Kepler
André Mesquita
Antonio Brasil Júnior
Arthur Leotta
Augusto Nelson Viana
Benedito Márcio de Oliveira
Carlos Barreira Martinez
Cecilia Lucino
Daniel Fernández
Daniel Rodriguez
Edmundo Koelle
Facundo González
Fernando Zárate
Gabriel Tarnowski
Geraldo Lucio Tiago Filho
Humberto de Camargo Gissoni
Jaime Espinoza
José Carlos Amorim
José Cataldo
José Geraldo P. de Andrade
Juan Carlos Cacciavillani
Lubienska Cristina Lucas Jaquie Ribeiro
Luciano dos Santos
Miguel Tornell
Orlando Anibal Audisio
Rafael Acedo Lopes
Regina Mambeli Barros
Ricardo Vasconcellos
Roque Zanata
Segen Farid Estefen
Sergio Galván
Sergio Liscia
Ticao Siguemoto
Victor Hidalgo
Zulcy de Souza

AMERICAN JOURNAL OF HYDROPOWER, WATER AND ENVIRONMENT SYSTEMS

Number 6

Jan 2018

Technical Papers

- 06** PUMPED HYDROELECTRIC ENERGY STORAGE SYSTEMS (PHES): CHALLENGES AND PERSPECTIVES IN THE BRAZILIAN ENERGY SCENARIO
Raimundo, Danielle Rodrigues; Filho, Geraldo Lúcio Tiago
- 11** HYDRODYNAMIC BEHAVIOR ON KAPLAN TURBINES: EXPERIMENTAL METHODOLOGY FOR THE EVALUATION OF CAVITATION DEVELOPMENT BASED ON DYNAMIC MEASUREMENTS
Rivetti, Arturo; Angulo Mauricio A.; Lucino Cecilia V.; Botero Francisco; Liscia, Sergio O.
- 16** TECHNICAL AND ECONOMIC EVALUATIONS FOR DECISION-MAKING ON THE REPOWERING OF SMALL HYDROPOWER PLANTS AND THE OPTIMUM EXPLOIT OF EXISTING PLANTS IN BRAZIL
Oliveira, Marcos André
- 25** EVALUATION OF HYDROELECTRIC, WIND AND SOLAR POTENTIAL: A CASE STUDY OF EXISTING PLANTS
Filho, Wilson Pereira Barbosa; Silva, Livia Maria Leite da; Silva, Nathan Vinicius Martins da; Oliveira, Karina Aleixo Benetti de; Abreu, Anna Luisa de Oliveira; Swiatovy, Gabriel Hepp
- 33** PREDICTION OF PRESSURE PULSATION IN FRANCIS TURBINES USING RANS SOLUTION: AN APPLIED STUDY OF EFFECTS OF THE TURBULENCE MODELS, MESH, DOMAIN EXTENSION AND MULTIPHASE FLOW
Marra, João M.; Gramani, Liliana M.; Zubeldia, Luiz F.; Kaviski, Eloy

PREDICTION OF PRESSURE PULSATION IN FRANCIS TURBINES USING RANS SOLUTION: AN APPLIED STUDY OF EFFECTS OF THE TURBULENCE MODELS, MESH, DOMAIN EXTENSION AND MULTIPHASE FLOW

¹Marra, João M.; ²Gramani, Liliana M.; ³Zubeldia, Luiz F.; ⁴Kaviski, Eloy

ABSTRACT

The present work shows a comparison between turbulence models, computational meshes, and domains extension, as regards its ability to numerically modeling the flow in a Francis turbine operating in part load and full load conditions. The main objective of this study is to evaluate the influence of these parameters at prediction of the amplitude and frequency of pressure pulsations in 3D flow in the turbine hydraulic system in any operative condition, when using RANS solution for single-phase incompressible model for the ruler equations of the flow or two-phase compressible model to consider the effects of the occurrence of cavitation on the flow. In this study were used the SST, k- ϵ and k- ω turbulence models with three different mesh densities and two extensions of the domain. The configuration of mesh and turbulence model whose results best adhered to available data was used to simulate two-phase flow and entire domain. The study was applied to the ITAIPU turbines and results obtained in each simulation were compared with data available for the operating conditions considered.

KEYWORDS: Francis turbine, Turbulence models, CFD, numerical simulation.

1. INTRODUCTION

As further a hydraulic turbine operates out of its designing conditions, worst is the flow at the turbine and greater is the possibility of hydraulic instability phenomena, mainly in turbines with fixed blades runners, as Francis.

Such changes in its operating condition cause pressure fluctuations with damage potential to the physical integrity of the turbine, generating unit and even civil facilities from the powerhouse to the dam.

One cause of this damage is due to the development of vortices at the runner outlet which induces pressure fluctuations which propagate throughout the hydraulic circuit [1], which may cause dynamic interactions which cause hydraulic instabilities and resonance in the turbine's hydraulic circuit.

To better understand the complex flow in a Francis turbine and evaluate disturbances which occur throughout the hydraulic circuit due to pressure fluctuations can be used the Computational Fluid Dynamics (CFD). For this, knowledge of inherent peculiarities to the application of this technique is extremely important for obtaining satisfactory results. In this work were conducted studies with variation of system's range (domain), spatial discretization (mesh) and turbulence models (SST, k- ϵ e k- ω) in solving the governing equations of three-dimensional turbulent flow, aiming to evaluate the adopted numerical solution ability of reproducing the hydraulic system behavior as pressure pulsation and associated frequencies to draft tube vortex cores.

Nevertheless, as states [2], 'many dynamics problems occurring in hydraulic machines can only be properly understood if their interaction with the fluid column in the waterways of the plant is taken into account,' which is related to the properties of inertia, compressibility, and dissipation of the system.

Three-dimensional geometric modeling of the entire hydraulic system and numerical simulations were performed using applications of CAD and CFD available at Center for Advanced Studies on Safety of Dams – CEASB considering as an object of study the turbines of Itaipu Hydroelectric Powerplant, which one has a rated power of 14,000 MW generated by 20 Francis turbines of 700 MW.

2. GOVERNING EQUATIONS AND NUMERICAL SOLUTION

Navier-Stokes equations have full capacity to describe the behavior of turbulent flows without needing of additional terms. However, as quoted in [3], its analytical solution for general cases is still open and, to meet the need of knowing the behavior of fluids in its countless applications, several numeric methods have been developed.

According to [4], one of the most successful methods is the Finite Volumes Method, which discretizes differential equations by a conservation balance of each property, e.g., the mass flow for each volume element.

However, turbulent flows of realistic high Reynolds numbers presents a wide range of spatial and temporal turbulence scales which generally involves spatial scales much smaller than the smallest finite volume of a mesh still viable to use. To allow that effects of turbulence be analyzed, a large amount of research in CFD has focused on developing of turbulence models conforming to [5].

Regarding the numerical solution of the Navier-Stokes equations, is common sense in the scientific world that its direct solution (DNS) is only computationally viable for more simplified systems. Given the inherent difficulty of a solution of these equations for real systems, as the flow in the hydraulic system

¹Department of Maintenance Engineering, Itaipu Binacional, Iguassu Falls, 85856-970, Brazil, marra@itaipu.gov.br; Tel: (+55 45)3520-2690; Fax: (+55 45)3520-3835

² Department of Mathematics, Federal University of Parana - UFPR, Curitiba, 81531-990, Brazil, gramani@ufpr.br

³ Center for Advanced Studies in Dam Safety - FPTI, Iguassu Falls, 85.867-900, Brazil, luiz.zubeldia@pti.org.br

⁴ Department of Hydraulic, Federal University of Parana - UFPR, Curitiba, 81531-990, Brazil, eloy.dhs@ufpr.br

of a turbine Francis, the most widely used alternative is using the Reynolds Averaged Navier-Stokes (RANS) equations. However, due to the required modeling of original equation terms, such procedure represents a filter in the system dynamic response, as demonstrated by [6], limiting its application in assessing the dynamic flow behavior. RANS solution modality and other types of solutions of the Navier-Stokes equations will be briefly discussed in the next section, aiming only characterize them.

2.1. Turbulence models

The numerical methods that model the turbulence are grouped into the following three categories:

- Reynolds Averaged Navier-Stokes (RANS) is a method which solves the mean flow and models the effects that all turbulence scales have over the mean flow. ([5]).
- Large Eddy Simulation (LES) is an intermediary methodology that solves the large turbulence scales, which carry most of the energy and models the smaller scales. ([7]).
- Direct Numerical Simulation (DNS) is able to solve the mean flow and all speed fluctuations due to turbulence. This method is extremely costly in terms of computational resources, so it is not used by the industry to solve flows. ([8]).

Among these methods, in line with [9] the RANS modeling is preferentially adopted for engineering cases, as it significantly reduces the required computational effort. In this method, the variables in a turbulent flow, e.g., flow velocity, are decomposed in an average term and fluctuating term, as shown in Equation (1).

$$U_i = \bar{U}_i + u_i \quad (1)$$

The substitution of the average values in the original transport equations for single-phase fluid results in Reynolds Averaged Equations given below in index notation, in which the bar indicates the average values, ρ the specific mass, p the pressure, τ_{ij} the stress tensor, S_M the sum of the body forces per unit of volume. x and t are respectively the spatial and temporal variables.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{U}_j) = 0 \quad (2)$$

$$\frac{\partial \rho \bar{U}_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \bar{u}_i \bar{u}_j) + S_M \quad (3)$$

Equation (2) is the continuity equation and the Equation (3) is the conservation of momentum equation, in which appears an additional term, the Reynolds stresses, $\rho \bar{u}_i \bar{u}_j$. This term represents correlations between the floating speeds and it is one more unknown in the system, according to [9]. To close the equations system it's necessary a model for the Reynolds tensor.

The eddy viscosity hypothesis, proposed by Boussinesq in 1877, establishes that the Reynolds tensor is proportional to the mean velocity gradient, mathematically analogous to the stress and strain tensor in a Newtonian fluid. Thus it is given by:

$$-\rho \bar{u}_i \bar{u}_j = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\rho k + \frac{\mu_t \partial U_k}{\partial x_k} \right) \quad (4)$$

Where μ_t is the eddy viscosity and k is the kinetic energy per unit of mass, like in [10].

When applying the hypothesis to Reynolds Averaged Equation for the momentum is obtained the following:

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial p}{\partial x_i} \left(p + \frac{2}{3} \rho k \right) + \frac{\partial}{\partial x_j} \left[\mu_{ef} \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \right] \quad (5)$$

Where μ_{ef} is the effective viscosity composed by molecular (μ) and eddy (μ_t) viscosity, such that:

$$\mu_{ef} = \mu + \mu_t \quad (6)$$

Eddy viscosity models are distinguished by the manner in which they prescribe the eddy viscosity and eddy diffusivity. ([10]).

2.1.1. The k-ε model

The model k-ε is based on the eddy viscosity hypothesis, as reported in [10]. In this model, the following expression relates the eddy viscosity μ_t to the specific turbulent kinetic energy (k) and specific turbulence eddy dissipation rate (ϵ), where $C_\mu = 0,09$ is a constant, and ρ the specific mass:

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon} \quad (7)$$

The transport equations for the turbulent kinetic energy (Equation 8) and eddy dissipation rate (Equation 9) give the values of k and ϵ :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \epsilon \quad (8)$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \epsilon) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \frac{\epsilon}{k} (C_{\epsilon 1} P_k - C_{\epsilon 2} \rho \epsilon) \quad (9)$$

where $C_{\epsilon 1} = 1,44$, $C_{\epsilon 2} = 1,92$, $\sigma_k = 1,0$ and $\sigma_\epsilon = 1,3$ are constants. The term P_k is the production of turbulence due to viscous forces modeled by the following expression:

$$P_k = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \left(3 \mu_t \frac{\partial U_k}{\partial x_k} + \rho k \right) \quad (10)$$

2.1.2. The k-ω model

According to the same [10], the k-ω model offers as an advantage treatment for calculating low Reynolds numbers near to the wall. Its formulation does not involve nonlinear damping functions as in the k-ε model and is generally more accurate and robust.

In this model, it is assumed that the turbulent kinetic energy and the turbulent frequency are related to eddy viscosity by the expression

$$\mu_t = \rho \frac{k}{\omega} \quad (11)$$

Two transport equations are solved, one for the turbulent kinetic energy (Equation 12) and another for the turbulent frequency (Equation 13):

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega \quad (12)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \frac{\omega}{k} P_k - \beta \rho k \omega^2 \quad (13)$$

where the specific mass, ρ , and the mean velocity vector, U , are treated as known quantities and P_k is the turbulence production rate given by Equation (10).

The model constants are given by $\beta' = 0.09$, $\alpha = 5/9$, $\beta = 0.075$, $\sigma_k = 2$ e $\sigma_\omega = 2$. The Reynolds Stress, (eq), is an unknown given by Equation (4).

2.1.3. Model Shear Stress Transport (SST)

In the free flow region far from the wall, the results obtained with the k-ε model are few influenced by the boundary conditions, however, near to the wall performance is not satisfactory for boundary layers with adverse pressure gradients as in agreement with F. R. Menter (1992a) apud [8]. To solve this deficiency, a hybrid model was suggested applying a transformation of the k-ε model for the k-ω model in regions near the wall and the k-ε model in completely turbulent regions far from the wall as reported by Menter (1992a, b, 1994, 1997) apud [8].

A complete formulation of the SST model, given below, is extracted from [11].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho U_i k) = \tilde{P}_k - \beta' \rho k \omega + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] \quad (14)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho U_i \omega)}{\partial x_i} = \alpha \rho S^2 - \beta \rho \omega^2 + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_{\omega} \mu_t) \frac{\partial \omega}{\partial x_i} \right] + 2(1-F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (15)$$

where the blending function F_1 is defined by:

$$F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta' \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho \sigma_{\omega 2} k}{CD_{kw} y^2} \right] \right\}^4 \right\} \quad (16)$$

Where M is the distance to the closest wall and

$$CD_{kw} = \max \left(2\rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \quad (17)$$

The value of A_2 is equal to zero when far from the wall, thus defining the use of k-ε model and becomes unitary when inside the boundary layer indicating the use of the k-ω model.

The eddy viscosity is defined by:

$$\mu_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)} \quad (18)$$

where S is the invariant measure of the deformation rate F_2 , and is a second blending function defined by:

$$F_2 = \tanh \left\{ \left[\max \left(\frac{2\sqrt{k}}{\beta' \omega y}, \frac{500\nu}{y^2 \omega} \right) \right]^2 \right\} \quad (19)$$

A turbulent energy production limiter is used for SST model to prevent the buildup of turbulence in stagnant regions:

$$P_k = \mu_t \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \rightarrow \tilde{P}_k = \min(P_k, 10 \cdot \beta' \rho k \omega) \quad (20)$$

All constants are calculated from a combination of constants corresponding to the k-ε model ($F=0$) and the k-ω model ($F=1$) by the general expression

$$\alpha = \alpha_1 F + \alpha_2 (1 - F) \quad (21)$$

Where α corresponds to one of the constants to be calculated. The constants for this model are:

$$\beta' = 0.09, \alpha_1 = 5/9, \beta_1 = 3/40, \sigma_{k1} = 0.85, \sigma_{\omega 1} = 0.5, \alpha_2 = 0.44, \sigma_{k2} = 1, \beta_2 = 0.0828, \sigma_{\omega 2} = 0.0856.$$

2.2. Two-phase flow

The occurrence of cavitation in the turbine flow requires two-phase flow treatment for better represent the flow phenomena. Considering a homogeneous mixing model, in which both phases share the same pressure and velocity

fields, it is sufficient to solve one equation for each the field, instead of solving it for each phase of field. However, apart the continuity equation and of the moment, it is necessary an additional transport equation coupled those, forming thus a set of nine governing flow equations in the considered control volume element for the case of three-dimensional flow, more the restriction that the pressure in all phases be equal to the average pressure of the mixture.

The additional transport equation can be obtained by conservation volume law which expresses the sum of partial fractions of the phases volumes should be equal to the unity. According to this law, a compressible viscous fluid mixture formed of liquid and a β vapor volume fraction, the density ρ_m of the mixture is given by Equation (22), where subscripts c and l refer respectively to the cavitation and liquid phase, as quoted by [1].

$$\rho_m = \beta \rho_c + (1 - \beta) \rho_l \quad (22)$$

Replacing this conservation condition at the continuity equation the additional transport equation (25) can be obtained, forming a set of coupled equations to the mass and momentum conservation of the compressible two-phase mixture, where the subscripts m were omitted for variables $\rho, \bar{U}, p$ at Reynolds Averaged Equations (23), (24) and (25), and the cavitation volume β is an additional variable of the system:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{U}_j) = 0 \quad (23)$$

$$\frac{\partial \rho \bar{U}_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \bar{u}_i \bar{u}_j) + S_M \quad (24)$$

$$\frac{\partial \beta}{\partial t} + \frac{\partial}{\partial x_j}(\beta \bar{U}_j) = \frac{1}{\rho_c} S_{cl} \quad (25)$$

The term S_{cl} is a source term relative to the mass flow rate of the vapor phase per unit volume, considering condensation and vaporization processes, which were modeled by Rayleigh-Plesset equation.

3. LOW FREQUENCY PHENOMENA IN SWIRLING FLOW

This section aims to present the origin of low-frequency phenomena that happens when a Francis turbine operates out of its optimum flow conditions. This section discusses also the limitation of the satisfactory representation of hydraulic system behavior at these operatives' conditions through RANS numerical solution of governing equations of three-dimensional flows using the turbulence models indicated in the previous section.

In its simplest form, the flow in a Francis turbine can be represented by a one-dimensional model. Euler equation for hydraulic machines relates the power absorbed by the turbine only with the fluid inlet and outlet conditions in its runner. According to Equation (22) in order to have an optimum efficiency, the fluid must enter in the turbine runner without shock with its blades, being tangential to them, and exiting axially ($U_{v2} = 0$), as happen when the turbine flow rate $Q = Q_{ot}$. However, as the Francis turbines have a fixed blades runner and rotate at a constant synchronous speed, whenever the flow in the turbine get away from their optimum conditions the speeds at inlet and outlet may no longer meet the ideal

condition, which normally occurs to meet the required power generation adjustment to its demands, as charge dispatches usually practiced.

$$N = \dot{m}(U_{V1}V_1 - U_{V2}V_2) \quad (22)$$

Due to operation outside of optimum conditions, the speed at the outlet is no longer fully axial, giving rise to tangencies components which swirl the fluid, generating the well-known vortices cores at the draft tube. These components cause fluctuations in pressure and can produce significant oscillation in the electrical power, as mentioned by [2]. This author also mentions that vortices core depends on the distribution of the speed field at runner outlet, the geometry of draft tube and the dynamic response of the hydraulic circuit as a whole.

Such behavior can be seen in Figure 1 by the speed triangle at the outlet of the turbine blades for different flow rate, Q , in the turbine. In part load ($Q < Q_{ot}$) and overload ($Q > Q_{ot}$) the tangential component $\vec{U}_{v2}$ is responsible for the rotational effect on the fluid. In ideal condition ($Q = Q_{ot}$) the flow exits axially and this effect does not occur.

When the boundary conditions allow the occurrence of the water vapor phase, the vortex core is configured as an underpressure in the core region of the fluid flow. Under these conditions, they become visually clear in a transparent draft tube, like the used in scale models of turbine testing laboratories, as illustrated in Figure 1.

These vortices act as an exciter source that dynamically interacts with the hydraulic system through a forced oscillation process or a hydraulic instability mechanism. At part load condition the vortex moves like a spiral with precessional movement and acts like a forced vibration with a characteristic frequency. If this frequency coincides with a natural frequency of the system there is a hydraulic resonance situation. At full load condition, the vortex can pulse radially at eigenvalue frequency of the system, configuring a self-excited phenomenon that produces oscillation even without a forcing source, as explained by [2]. Due to the influence of the compliance of the vortex in this phenomenon is essential to adopt a model that considers the occurrence of cavitation in the flow.

[Figure 1: Vortex core (Adapted from [1]).]

Although the intensity of the rotational component a , in the fluid has singular importance in pressure fluctuation, other factors such as cavitation, depending on its intensity, also influence this process. The explanation for the effect of cavitation in the pressure pulsation in the draft tube is due to the dynamic gain provided by the resonance with the draft tube fluid system natural frequency, as indicated in [12].

To finalize this section, it is observed that single-phase solution of the governing equations (2) and (3) does not allow

the appearance of the cavitation, excluding the evaluation of its influence on the hydraulic system behavior. Other limiting factors in the proposed mathematical model are not considering the effects of elasticity and viscoelasticity throughout the whole hydraulic boundary surface and mitigation system of pressure fluctuations by atmospheric aeration or compressed air injection in the turbine.

4. GEOMETRICAL MODELING OF THE HYDRAULIC SYSTEM

The hydraulic system considered as object of study was the existing at Francis turbines of Itaipu Hydroelectric Power Plant, equipped with 20 vertical generating units with the following nominal values for its turbines:

- Shaft power: 715 MW
- Net head: 112.9 mca
- Rotation: 91.6 RPM

The hydraulic system of Itaipu turbine is illustrated in the following figure:

[Figure 2: Hydraulic system of Itaipu turbines - Vertical section - [13].]

The geometrical models used for developing this study were developed based on design's bidimensional technical drawings of the turbine and of the hydraulic system and on some measurements made on site. The models of the set Spiral Case, Pre-distributor first two geometrical models ones were dev developed by [14] and the last one by [15]. Besides these components were also molded geometries of part of the Reservoir, Water Intake, and Penstock.

4.1. Modeling of stationary domains

4.1.1. Set Spiral Case, Pre-distributor and Distributor

[Figure 3: Geometric model of the set Spiral Case, Pre-distributor and Distributor - [14].]

The geometrical models of the set Spiral Case, Pre-distributor, and Distributor, for the operating conditions contemplated in this work were developed by [14]. The opening of the distributor is different for each condition and its value is given by the smallest distance between two blades of the Distributor.

4.1.2. Draft Tube

The three-dimensional model of the Draft Tube used in this study was developed by [15]. In order to avoid recirculating problems in the outlet region of the draft tube, was introduced an extension to ward off the exit boundary condition, thus making the numerical solution process more stable.

4.1.3. Reservoir, Water Intake and Penstock

In order to allow evaluation of the influence that presence of components upstream of the spiral case has on simulation results, the geometric modeling of the components was made, as illustrated in Figure 5.

4.2. Modeling of the rotating domain: the Turbine Runner

The geometric model of the Turbine Runner was developed by [14]. Due to the lack of blade's profile's drawings, was necessary a field survey in order to get points on the blade. The nominal value of the runner outlet diameter is 8.1 m.

5. BOUNDARY CONDITIONS AND MEASUREMENTS

Aiming more robust results, the boundary conditions chosen were the mass flow at the inlet and the static pressure at the outlet of the domain. For the domain without the penstock, the inlet boundary condition corresponds to the flow (or speed) at spiral case inlet and the outlet condition corresponds to the static pressure at draft tube outlet. For these conditions, the total pressure is an implicit result of the solution.

In order to have an appreciation of the numerical solution of measurements that was developed for two turbine operating points tested in scale model and prototype, being one for operation with discharge below design point and another with discharge above design point, of which there are images of the vortex core at draft tube of scale model, as in [16]. In unitary values, the simulated operating points are:

- Part load: $n_{11} = 65.9$ RPM; $Q_{11} = 0.580$ m³/s
- Overload: $n_{11} = 68.9$ RPM; $Q_{11} = 0.995$ m³/s

The location of these points on the unitary hill chart of a scale model of the turbine and the image of the respective vortex core are shown in Figure 7.

Based on the scale model data [16] and measurements made on hydraulic stability tests of turbines of the units U04 [17] and U11 [18] the values considered to evaluate the adherence of numerical simulations are presented in Table 1.

[Table 1]: Operating parameters and measurements in prototype.

Parameters	Partload	Fullload
Distributor's opening [%]	49.3	88.8
Net Head [mca]	126.8	116.0
Flow Rate [m3/s]	434.2	712,4
Efficiency [%]	90.7	92.7
Power [MW]	488.8	748,6
Pressure oscillation at draft tube [kPa _{p/p}] – Model Transposition	65	19
Pressure oscillation at spiral case [kPa _{p/p}] – Model Transposition	22	16
Main frequency of the pressure oscillation [Hz] - Model Transposition	0.38	1.60
Pressure oscillation at draft tube [kPa _{p/p}] – Prototype	56 (U04)	43 (U11)
Pressure oscillation at spiral case [kPa _{p/p}] – Prototype	35(U04)	47 (U11)
Main frequency of the pressure oscillation [Hz] - Prototype	0.42/ (U04)	1.10 (U11)

6. NUMERICAL ANALYSIS

6.1. Computational mesh

In order to evaluate the influence, the spatial discretization has on the results were tested three meshes with a different number of elements for each one of the two evaluated operating conditions. Table 2 shows the total number of elements of the respective meshes utilized for each simulated condition and also shows the number of elements of the meshes per component of the turbine's domain.

[Table 2]: Number of elements of the mesh for each simulated condition.

PartLoad				Fullload		
Component	Mesh1	Mesh2	Mesh3	Mesh1	Mesh2	Mesh3
Spiral Case	1593308	3123673	5356802	1547968	3174044	5226729
Runner	1904397	2593739	3298709	1991912	2510822	3338452
Draft Tube	867022	1652722	4399713	867030	1652722	4398676
Total	4364727	7370134	13055224	4406910	7337588	12963857

A viewing surface of the Mesh 3 is shown in Figure 8, on what the virtual prolongation of the draft tube shown in Figure 4 is partially represented.

6.2. Temporal discretization, computational resources and simulation times

In order to enhance the convergence of simulations, it was decided to divide them into two stages. The first stage of simulation has the function of providing a good initial condition for the following stage. For this stage was utilized varied time steps, as shown in Table 3, and the first order advection scheme (upwind), which convergence is easier.

[Table 3]: Time steps used in simulations.

Operation Condition	Time Steps [s]	
	Initialstage	Final stage
Part load	20 x 0.01; 20 x 0.1; 20 x 0.01; 40 x 0,00181	1440 x 0.00181
Full load	30 x 0.01; 30 x 0.1; 30 x 0.01; 30 x 0,00181	1080 x 0.00181

For the second simulation, stage was utilized the second-order advection scheme and constant time step equivalent to the period that runner takes to rotate one degree. The simulated time for Part Load condition is equivalent to approximately 7,8 runner revolutions and the time for the Full load is equivalent to 8,6 revolutions.

The convergence criterion adopted for all simulations was 1×10^{-5} to the RMS value of residues.

The computational resources used were a computer with Intel® Xeon® E5-1650 CPU of 3.2GHz processor of which was

utilized ten cores, 16GB of RAM and 64-bit operating system. The simulation times are shown in Table 4:

[Table 4]: Simulation time [h].

Turbulence Model	PartLoad			Fullload		
	Mesh1	Mesh2	Mesh3	Mesh1	Mesh2	Mesh3
SST	18.9	29.2	48.3	11.0	14.3	29
k-ε	17.7	27.8	47.6	9.8	11.7	28.4
k-ω	17.4	27.0	43.1	8.7	13.7	25.9

7. RESULTS

Flow in a Francis turbine operating at Part Load or in Full load produces its characteristics vortex hydraulic phenomena in the flow in the draft tube at the turbine runner outlet, as previously mentioned. All these phenomena produce pressure pulsation with a determined amplitude and frequency, being the amplitude strongly dependent on dynamic interaction with the entire hydraulic system involved in the flow. Monitor points in the numerical simulation were inserted at spiral case inlet and at draft tube inlet for acquiring these values at each simulated time step.

The results related to incompressible single-phase simulations are presented at items 7.1 to 7.3. The results related to compressible two-phase simulations are presented at item 7.4.

7.1. Core vortices prediction

The main flow characteristic in part load and full load is vortices formation at the draft tube with a characteristic geometry for each condition, with volume and length dependents of the turbine suction head, that is, of Thoma coefficient, σ .

At the part load condition, all considered turbulence models and meshes were able of reproducing the vortex characteristic shape, but without reaching the expected water vapor pressure for the considered suction head corresponding to $\sigma=0.17$. However, the isopressure surfaces indicated in Figure 9, corresponding to the pressure of the suction head considered (300kPa) for this condition, shows a vortex core at turbine output with similar characteristics to the one observed on scale model test, as available in [16]. Below this value, lower is the pressure, the isopressure surfaces obtained at this simulation becomes ever more deformed and divergent of the results of the scale model test.

For full load condition, the considered turbulence models and meshes were successful in reproducing the characteristic geometry of the vortex of this operative condition and also in reaching the vapor pressure expected for the considered suction head corresponding to $\sigma=0.132$, but with very small volume for coarser mesh M1. The effects of turbulence models and mesh refinement can be compared by images shown Figure 10 for isopressure surfaces of 130kPa, wherein the configuration with SST model and mesh M3 presented aspects more adherent to the data available in [16] to this operative condition.

[Figure 10: Full load vortices for each simulated configuration ($P = 130$ kPa).]

The vortex volume variation with the reference pressure for isopressure surfaces of 130kPa and vapor pressure (3.2kPa) can

be seen in Figure 11 for configuration of mesh and turbulence model M3-SST, where the blue core refers to vapor pressure.

[Figure 11: Full load vortices – Volume comparison ($P = 130$ kPa e $P=3.2$ kPa).]

7.2. Pressure pulsation

The frequency and amplitude of pressure fluctuation due to vortex core presence in numerical simulations were obtained by applying Fast Fourier Transform to temporal values obtained with virtual monitor probe at the spiral case and draft tube.

For part load condition, the virtual monitor probe of pressure at draft tube recorded the frequency of 0.33Hz for all simulated configurations of turbulence models and meshes considered, against 0.38 Hz for the transposition from scale model and 0.42 Hz for the prototype measurement. As pressure fluctuations caused by vortex propagate to upstream and downstream, disregarding dynamic gains or nonlinearities, as expected, pressure fluctuation at monitor points of spiral casing should register the same frequency of draft tube, but with smaller amplitude. This happened for all configurations except for the frequency at the configurations M2 k-ε e M2 k-ω, whose obtained value was 0.16 Hz, as indicated in Table 5.

Depending on configuration, the value peak to peak of the pressure pulsation amplitude varied from 34 kPa to 44 kPa in draft tube and 12,9kPa to 17,5kPa on spiral casing. Considering the average value of the pulsations found in the simulations, there was a deviation for less of about 36% relative to the reduced model and for the prototype 30% at draft tube and 58% at the spiral casing. Additional simulation of the part load with mesh M3-SST for longer simulation of 10,73s resulted at 50,6 kPa p/p in the draft tube and 24,7kPa p/p in the spiral casing, reducing the respective deviations for about -22% and 12% compared to values from scale model and 10% and 29% for the measurements at prototype. The simulation with two-phase model could also contribute to reducing these deviations since the cavitation effect on the response of the suction tube to the excitation of the vortex would be better represented, but this simulation was not performed.

[Table 5]: Power, pressure pulsation and frequency for part load condition.

Parameter	M1 SST	M1 k-ε	M1 k-ω	M2 SST	M2 k-ε	M2 k-ω	M3 SST	M3 k-ε	M3 k-ω
N [MW]	485.7	480.8	484.8	490.4	483.2	491.4	487.3	480.1	487.5
ΔH [kPa _{p/p}] (Draf Tube)	40.0	43.9	42.4	35.8	34.7	35.2	36.5	41.7	41.5
f [Hz] (Draft Tube)	0.33	0.33	0.33	0.33	0.33	0.33	0.33	0.33	0.33
ΔH [kPa _{p/p}] (Spiral Case)	13.8	14.7	14.7	12.3	12.9	13.5	15.6	16.9	17.5
f [Hz] (Spiral Case)	0.33	0.33	0.33	0.33	0.33	0.33	0.33	0.33	0.33

Another parameter that also indicated adherence of the simulation results on part load to those provided by the scale model are the value of the generated power, whose expected value for part load conditions is 488.8MW at the simulated point. Regarding the ability to represent this parameter, k- ϵ model had the worst result at all meshes and SST model the best. However, SST and k- ω results present adherent results for the more refined mesh M3.

For full load condition, none of the configurations of simulation was able to reproduce the vortex dynamics, resulting in an almost static pressure field, therefore, without pressure pulsation, in disagreement to what is observed in the prototype at the operating point considered. Additionally, also it was performed a single-phase simulation with the complete domain at full load. The mesh used was M3 and the turbulence model was SST. This configuration also resulted in a static field of pressure.

The obtaining of null pressure fluctuation in full load may be related to the fact that on ITAIPU turbines such fluctuation is due to a hydraulic instability phenomenon. At this situation, the single-phase model used for governing equations does not allow the existence of water vapor at the central vortex core, as well as does not consider the elastic and viscoelastic effects of the fluid and the entire hydraulic contour, respectively considered as incompressible and ideally rigid.

Notwithstanding, the values obtained for the turbine power were reasonable, presenting a deviation from 2.6% to 3.5% compared to the value of 748,6MW predicted by the scale model transposed to prototype. The better results for this parameter was found with mesh M1, while the M2 and M3 meshes presented worst results for all turbulence models, except for configuration M2/ k- ϵ . Table 6 shows the values of power obtained and respective deviation for the full load simulations.

[Table 6]: Power and deviation for full load condition at the simulated configurations.

Parameter	M1	M1	M1	M2	M2	M2	M3	M3	M3
	SST	k- ϵ	k- ω	SST	k- ϵ	k- ω	SST	k- ϵ	k- ω
N [MW]	768.5	769.4	767.4	777.9	768.2	775.5	775.9	775.4	773.9
Deviation [%]	2.7	2.8	2.5	3.9	2.6	3.6	3.7	3.6	3.4

7.3 Pressure and velocity fields

Figure 12 displays the streamlines for part load and full load condition. In part load, streamlines allow to distinguish the swirling effect of flow at the same direction of runner's rotation and the flow disequilibrium in the right and left draft tube channels due to tangential component of absolute speed a , at the same direction of rotation. In full load, is observed a symmetrical concentration of flow at the periphery of the draft tube conical section and elbow, but without disequilibrium of flow in channels.

[Figure 12: Streamlines for M3 SST - Part load (left) and full load (right).]

Figure 13 shows vertical sections of pressure and velocity fields in turbine domain at part load condition.

The image corresponding to the pressure field allows visualization of different areas along the pressure transition from spiral case to draft tube, as well as the region corresponding to the spiraled vortex of the part load. The image corresponding to velocity field allows observing the velocity transition between the spiral case and the draft tube, denoting the lack of symmetry of velocity field at draft tube due to the presence of the spiraled vortex.

[Figure 13: Pressure (left) and velocity (right) fields for M1 SST - Part load.]

Figure 14 shows vertical sections of the pressure and velocity fields in turbine domain at full load condition. The image corresponding to the pressure field allows visualization of different areas along the pressure transition from spiral case to draft tube, as well as the region corresponding to the oblong shaped vortex core for the full load. The image corresponding to velocity field allows observing the velocity transition between the spiral case and the draft tube, denoting the almost-symmetry of velocity field at the conical initial stretch of draft tube due to the presence of symmetrical vortex core.

[Figure 14: Pressure (left) and velocity (right) for M3 SST - Full load Considering the topographical difference between the level of downstream and the center of the.]

Considering the topographical difference between the level of downstream and the center of the spiral case inlet section and disregarding the head loss in this stretch, the expected pressure in the spiral case is 1325 kPa.

Therefore, is observed a deviation of approximately +368 kPa on the expected pressure value for part load condition and of approximately +198 kPa for full load condition. This difference is probably due to an insufficient spatial discretization, which is directly influencing the integration process of viscous friction in solving governing equations.

7.4. Two-phase simulations

The implementation of the two-phase simulations were mainly motivated by the fact that RANS incompressible single-

phase simulations had not captured the vortex dynamics in full load condition, as seen at the prototype turbine used as an object of study at this work and whose origin is attributed to a known process of hydraulic instability due to the self-excited phenomenon mentioned in section 3. So, to evaluate the contribution of the hydraulic system and cavitation in the behavior of the pressure pulsations, compressible two-phase simulations were performed at two domains: Partial - from the inlet of the spiral case up to the outlet of the draft tube; Total - from the surface of upstream reservoir up to the outlet of the draft tube. A display of the surface mesh the entire domain and its elements numbers are shown in Figure 15.

Considering the very long time for processing the two-phase simulations using the computational resources used in the single-phase simulations, the two-phase simulations were performed at a cluster of 8 servers with Intel® Xeon® E5-2697 CPU of 2.7 GHz processor and 64 logics cores, 20GB of RAM/Server and 64-bit operating system Linux.

According to recommendations of [10], two-phase simulations were performed at two stages for both domains considered. At the first stage, the cavitation model is set unabled and this converged preliminary solution will provide the initial condition for the second stage of the simulation, to be performed with the cavitation model activated. Due to the greater complexity of the equations to be solved for this case, was adopted a milder convergence criterion of 1×10^{-4} . However, the RMS value of the residues was lower than 1×10^{-5} at the second stage of the solution. The mesh used was M3 presented at Table 2 and turbulence model was SST. All others configurations mentioned for single-phase were kept at the two-phase simulations. The total simulated time was 10.2s, corresponding to 15.5 revolutions of the turbine. The variation of time steps and time of simulation are shown in Table 7.

[Table 7]: Time steps and time simulation in full load two-phase simulations.

Domain	Time Steps [s]		Time Simulation [h]
	Initialstage	Final stage	Total
Parcial	30 x 0.01; 30 x 0.1; 30 x 0.01; 30 x 0,00181	3600 x 0.00181	119
Total	30 x 0.01; 30 x 0.1; 30 x 0.01; 30 x 0,00181	3600 x 0.00181	204

The values peak-to-peak of the pressure pulsation and respective frequency obtained at two-phase simulations for both domains considered are shown in Table 8. In contrast with single-phase simulation, the virtual monitoring probes at Spiral Case and Draft Tube registered pressure pulsations at two-phase simulations. The values were calculated by Fast Fourier Transform of the time variation of the pressure at Spiral Casing and Draft Tube from 4 to 10.2 seconds of time simulation,

which corresponds to an already range stabilized of the final stage of the simulation. The reference values of the model and prototype were taken from existing time pressure records, considering the maximum value peak-to-peak.

The pressure pulsation in the prototype was not adherent to the results obtained from the scale model, as shown in Table 8. Possibly this is due to an interaction with the hydraulic system in the prototype turbine, leading to a lack of dynamic similarity with the test rig and possible effect of hydraulic instability at considered Full Load operation point. For this reason, the turbines of ITAIPU were preventively provided with an axial atmospheric aeration system to smooth full load operation of its turbines, as explained at [19].

[Table 8]: Pressure pulsation and frequency for two-phase full load simulations.

Parameter	Model	Prototype Measurements	Domain Simulated	
	Test Rig	U11	Partial	Total
ΔH [kPap/p] (Draft Tube)	19	43	50.0	58.0
ΔH [kPap/p] (Spiral Case)	16	47	79.0	62.0
f [Hz]	1.6	1.10	0.32	1.14

With respect to the results of pressure pulsation obtained at the simulations, the maximum peak-to-peak values greatly exceeded the reference value of the prototype, as illustrated by the graphs of Figure 16 for the case of Total Domain. However, considering the value peak-to-peak of the FFT at the interval of 4.0 s 10.2 s they presented less deviation in the amplitude with relation to the measurements on the prototype, as shown in Table 8. Nevertheless, Partial Domain simulation not adequately represented the frequency of pressure pulsations on the prototype, since the value obtained was 0.32 Hz and measured value in the prototype was 1.10 Hz. Meantime, it should be noted some positive aspects related to results of two-phase simulations, mainly considering that prototype measurements present characteristics that it operates near a hydraulic instability point well controlled by an atmospheric aeration system of the turbine, not included in the numerical simulations of the flow. At prototype, a similar damped pressure pulsation to Figure 16 appears every 12s at the operating point in reference.

Pressure pulsations in the simulation with the entire domain in full load showed an average deviation of approximately 33,4% higher than the amplitude of reference in the prototype. Even so, in the author's opinion, a deviation of this magnitude could be considered an acceptable prediction to the pressure pulsations in hydraulic turbines in its design phase. In this simulation, the frequency of the pulsations was also well represented, as well as the occurrence of damping pulsations

observed in the prototype. Another positive factor for the CFD simulations was the detection of the gain in the amplitude of pressure pulsations in the spiral casing relative to the draft tube be of the same order as in the prototype and synchronously. This discrete gain is a symptom the prototype turbine operates with some interaction with the hydraulic system. However, at scale model test this phenomenon was not observed, signaling the test rig did not operate with hydraulic instability. Possibly due to that the deviation of the results regarding the scale model test was very large, up 200%, evidencing the high risk of validating the prototype turbine pressure pulsation prediction at design stage based only on a model test.

The fact that the amplitude of recorded pressure pulsation in the spiral case have been higher and in sync with the pulsation in the draft tube indicates, as described by [20] and [1], the propagation of a permanent plane wave in the hydraulic circuit, confirming the need to take into account the cavitation modeling at simulations of self-excited oscillation phenomena, normally associated with the possibility of hydraulic instability.

The fact that the frequency of the pressure pulsations at simulations with Partial Domain presented no adherent results shows the dependence of this parameter with the hydroacoustic effects related to the compressibility of the cavitation vapor volume and to domain extension at acoustic wave propagation, better represented at Total Domain simulations.

The evolution of the volume and geometry of the vortex at the two-phase full load simulation is shown in Figure 17 in four pictures for each domain taken respectively in similar moments of the simulations. The vortexes presented are images rendered of the elements where the volumetric fraction of the vapor is higher than 75%. The vortex core is quite axisymmetric as expected, but its shape is not oblong comparatively to observations at scale model test. The volume and length of the vortex core at the simulations with entire domain was lower than that with Partial Domain.

[Figure 17: Full load vortices for two-phase simulation: a) Partial Domain (top); b) Total Domain (bottom).]

8. FINAL CONSIDERATIONS

All the used configurations of turbulence models and mesh satisfactorily represented the shape and frequency of part load vortex at single-phase simulations. However, it not allows reaching the expected vapor pressure at vortex core to the simulated operative point. However, on a broad way, the configuration M3-SST showed the best results. The pressure pulsation amplitude showed a maximum deviation of

22% to less on scale model and 29% relative to prototype. Furthermore, the CFD simulation showed a consistent pressure pulsation attenuation from draft tube to spiral case consistent with results of the model and prototype. The simulation with the two-phase model at part load wasn't performed, but probably could contribute to reducing these deviations, since the cavitation effect on the response of the suction tube to the excitation of the vortex would be better represented.

All the used configurations of turbulence models and mesh satisfactorily represented the vortex shape at full load condition at single-phase simulations and allowed to reach the expected vapor pressure at vortex core, but the configuration with better relative dimensional similarity with the vortex from scale model was M3-SST. The amplitude and frequency of the pressure pulsation were unsatisfactorily represented even considering the entire hydraulic domain since the simulation had not captured the vortex dynamics at this configuration. However, the simulation with two-phase model and the complete domain has captured the dynamics of the pressure pulsation observed at prototype as to the frequency and stability of the pressure pulsations, although operates close an instability point. The pressure pulsation amplitude showed a maximum deviation of 33% to greater than prototype records. Furthermore, the CFD simulation showed a discrete pressure pulsation gain from draft tube to spiral casing consistent with results of the model and prototype.

The comparison of results of CFD with scale model results transposed to prototype indicated an excessive deviation at full load condition, indicating a high risk of validating the prototype turbine pressure pulsation prediction at design stage based only on a model test when a resonance or instability cannot be discarded at the prototype.

A more realistic comparison at full load condition requires realizing a prototype test with the aeration system of the turbine inhibited or consider the air injection at the simulations, configuring a three-phase simulation, consuming greater computational effort to run the simulation. Other improvements would be consider the contribution of the fluctuation of boundary conditions of the inlet (flow) and outlet (pressure) and also the elasticity and viscoelasticity of the lateral walls of the boundary of the hydraulic system.

Nevertheless, the results obtained in these simulations signalize that RANS solution of the governing equations considering the presence of the cavitation at a two-phase compressible model and the entire extension of the hydraulic domain can present satisfactory results at the prediction of pressure pulsation of Francis turbines operating out of the optimum point.

Aiming to improve the results of the prediction of pressure pulsation, authors are developing a new study with a model of three-dimensional flow three-phase and/or with updating conditions through a one-dimensional model which considers the vortex dynamics and the flexibility and viscoelasticity of the walls of the hydraulic contour. To better comparison with prototype measurements, was developed a device that permits vary and/or inhibit the aeration atmospheric of the turbines during the test. The results of these complementary studies will be part of another article.

LIST OF SYMBOLS

U_i	Velocity	α	Constant model $k-\omega$
$\bar{U}_i$	Average velocity component	β	Constant model $k-\omega$
u_i	Time dependent velocity component	σ_ω	Constant model $k-\omega$

t	Time	$\tilde{\rho}_k$	Limiter SST turbulence model production
ρ	Specific mass	S_{cl}	Source term relative to mass flow rate of the vapor phase per unit volume
p	Pressure	F_1	First merge function of the SST model
x	Variable of space	F_2	Second merge function of the SST model
τ_{ij}	Stress tensor	F	
Δ	Variation	y	Distance to the nearest wall in the SST model
S_M	Sum of the body force per unit volume	σ_{k1}	Constant model SST
e_q	Reynolds tensor	$\sigma_{\omega1}$	Constant model SST
μ	Dynamic viscosity	α_1	Constant model SST
μ_t	Turbulent viscosity	α_2	Constant model SST
μ_{ef}	Effective viscosity	σ_{k2}	Constant model SST
δ_{ij}	Kronecker delta function	$\sigma_{\omega2}$	Constant model SST
k	Turbulent kinetic energy	β_1	Constant model SST
ε	specific turbulent dissipation rate	β_2	Constant model SST
C_μ	Constant model k- ε	σ	Constant model SST
$C_{\varepsilon1}$	Constant	H	Head of the turbine [mWc]
$C_{\varepsilon2}$	Constant	N	Power
σ_k	Constant	m	Mass flow
σ_ε	Constant		
P_k	Produced turbulence by viscous forces		
ω	Turbulent frequency		
β'	Constant model k- ω		

9. ACKNOWLEDGMENTS

The present work only was possible due to the support of the Center for Advanced Studies on Safety of Dams – CEASB belonging to the Foundation Itaipu Technological Park – FPTI which especially we thank the availability of computer equipment and application necessary for this work.

10. REFERENCES

- [1] Alligné, S. Forced and self-oscillations of hydraulic systems induced by cavitation vortex rope of francis turbines. PhD thesis, École Polytechnique Fédéral de Lausanne, Lausanne, Switzerland, 2011.
- [2] Dörfler, P.; Mirjam, S.; Coutou, A. Flow Induced Pulsation and Vibrations in Hydroelectric Machinery: Engineers Guidebook for Planning, Design and Troubleshooting. Springer, London, 2013.
- [3] Fox, R. W.; McDonald A. T.; Pritchard, P. J. Introduction to Fluid Mechanics. 7. ed. LTC, 2013.
- [4] Maliska, C. R. Heat Transfer and Computational Fluid Mechanics. 2. ed. LTC, Rio de Janeiro, Brazil, 2004.
- [5] ANSYS. Ansys CFX - Solver modeling guide- Release 15.0. Ansys Inc, 2013.
- [6] Smagorinsk, J. General circulation experiments with the primitive equations. Monthly Weather Review, n. 91, p. 66, 1963.