Volume 3 • July, 2016
ISSN 2317-126X • US$: 15.00

American Journal of Hydropower, Water and Environment Systems

Technical Papers

Technical Notes

IAHR AIRH
Published with the support of
Hydraulic Machinery and Systems
International Association

latinamerican
WORKING GROUP

IAHR AIRH

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

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.3, july. 2016.

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. Enviroment Systems.
II. Latin American Working Group

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

ISSN 2317-126X

9 772317 126001 00002

American Journal of Hydropower, Water and Environment Systems

Editorial

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

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

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

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

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

Yours sincerely,

Geraldo Lúcio Tiago Filho
Editor in Chief

Regina Mambeli Barros
Technical Editor

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 he abbreviation, "eds."
- publisher
- city of publication
- volume number (if any) in boldface if a single number, include, "Vol." if part of larger identifier (e.g., "PVP-Vol. 254")
- inclusive page numbers of the work being cited (include "pp.")

(4) Reference to theses and technical reports should include:

- last name of each author followed by their initials
- year of publication
- full title in quotes, title capitalization
- report number (if any)
- publisher or institution name, city

Sample References

[1] 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.

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

American Journal of Hydropower, Water and Environment Systems

Number 3 **JULY 2016**

Technical Papers

Technical Notes

American Journal of Hydropower, Water and Environment Systems
Publisher: Acta Editora/LAWG-IAHR
DOI:10.14268/ajhwes.2016.00033
ISSN: 2317-126X
Subject Collection: Engineering,
Subject: engineering, measurement, environment Systems

Numerical Simulations of Complex Flows in Hydraulic Turbomachinery. Actual R&D Lines in the 'CFD/Applied Maths Laboratory' Group

[1]Coussirat M., [2]Fontanals A., [1]Panella L., [1]Henderson G., [1]Aguirre R

ABSTRACT

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

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

1. INTRODUCTION

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

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

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

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

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

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

$$\frac{\partial \rho_m \phi_m}{\partial t} + \nabla \cdot \left(\rho_m \vec{c}_m \phi_m\right) = \nabla \cdot \left(\Gamma_m \nabla \phi_m\right) + S_{\phi,m}, \quad \phi = \left\{1, \vec{c}, T, Y_j, k, \varepsilon, \ldots\right\} \qquad (1)$$

Where **ρ** is the density, $\vec{c}$ is the velocity vector, $\mathbf{S_\Phi}$ is a general source term, $\mathbf{\Gamma_m}$ is the diffusion coefficient; and Φ represents the dependent variables (scalars as the fluid temperature $\boldsymbol{T}$, the

[1]CFD/LAMA, Depto Ing. Electromecánica, Universidad Tecnológica Nacional.C/Rodriguez 273. 5500 Mendoza, Argentina, corresponding author: miguel.coussirat@frm.utn.edu.ar [2]Escola Universitaria d'Enginyeria Tècnica Industrial de Barcelona.C\ Compte d'Urgell, 187. 08036. Barcelona, Spain. Centre de Diagnòstic Industrial i Fluidodinàmica (CDIF), Universitat Politècnica de Catalunya Av. Diagonal, 647. ETSEIB. – 08028 Barcelona, Spain

mass fraction for j_{th} species Y_j, the turbulence kinetic energy k, or its dissipation rate ε ; or vectorials as the velocity $\vec{c}$) in the system. It is highlighted again that the subscript m designates 'mixture', i.e., remarking that the fluid is a two- or multi- phase fluid. With the introduction of a cavitation model, the complete set of governing equations for turbulent cavitating flows can be derived from the foregoing generalized scalar transport equation.

1.1 Common strategies for modeling the turbulence

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

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

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

1.2 Common strategies for modeling the cavitation

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

$$\frac{\partial \alpha \rho_v}{\partial t} + \nabla \cdot (\alpha \vec{c} \rho_v) = R_e - R_c \quad (2)$$

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

$$\frac{DR_B}{Dt} = \sqrt{\frac{2}{3}\frac{P_B - P}{\rho_l}} \quad (3)$$

$$\frac{\partial \alpha \rho_v}{\partial t} + \nabla \cdot (\alpha \vec{c} \rho_v) = \frac{\rho_v \rho_l}{\rho_m}\frac{D\alpha}{Dt} \quad (4)$$

Here, R_B is bubble radius; ρ_l, ρ_v, ρ_m are liquid, vapor and mixture densities; P and P_B are fluid and bubble surface pressures respectively.

1.3 Applying CFD for turbomachinery design

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

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

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

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

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

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

2. VALIDATION/CALIBRATIONS TASKS, RESULTS AND DISCUSSION

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

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

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

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

[Table 1]: Turbulence models used, (see details of the references cited in this Table in [2,13]).

	Turbulence models	Notation	Reference
1	Spalart Allmaras	SA	Spalart and Allmaras [Spalart, 1994]
2	Standard k-ε	Std k-ε	Launder and Spalding [Launder, 1974]
3	Realizable k-ε	Rlz k-ε	Shi et al.[Shih, 1995]
4	Re Normalization Groups k-ε	RNG k-ε	Yakhot and Orzag [Yakhot, 1986]
5	Standard k-ω	Std k-ω	Wilcox [Wilcox, 1998]
6	Shear Stress Transport k-ω	SST k-ω	Menter, 1994 [Menter, 1994]
7	Reynolds Stress Model	RSM	Launder at al.[Launder, 1975]

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

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

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

c) When a flow around a stage of fixed and moving cascade of blades was modeled, the EVM used shows underestimations of the turbulence intensity $(I\% = \sqrt{u'^2}/U_0)$ compared against experiments, as well as for isolated foils having thin trailing edges and low flow incidence angles.

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

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

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

2.1 Database selected for Rotor Stator Interaction (RSI) Phenomena

In turbomachinery design, the RSI is an important phenomenon, which has a strong influence in the machine behavior. These interactions can have a significant impact in the vibrational and acoustical characteristics of the machine, because unsteadiness and turbulence play a fundamental role in complex flow structures. The RSI can be divided into two different mechanisms: potential flow interaction and wake interaction. The nature of the flow due to the wake interaction is unsteady and turbulent, and there it also appear three-dimensional (3D) boundary layers, curvature and system rotation effects (see more details in [15]).

The choice of an appropriate turbulence model and the boundary layer treatment is far from trivial, and a suitable turbulence modeling plays an important role for successful CFD results [10]. The experimental data used here are for a single-stage pump with a specific number, n_q~53 [rpm, $(m_3/s)_{0,5}$, $m_{0,75}$); being a vaned diffuser pump with five impeller blades Z_i, eight diffuser vanes, Z_g and a volute casing as shown in Fig.1, (see complete details in [11]). A detailed description of the pump and its specifications of the essential components are summarized in Tsukamoto et al. 1995 [11]. The unsteady pressure measurements was made at several places (stations) in the vaned diffuser passage (see Fig.1), and a comprehensive survey of instantaneous pressure within the diffuser passages was obtained, (see Figs. 8 and 9).

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

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

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

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

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

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

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

For checking the performance of the turbulent models from Table 1, the results obtained for pressure fluctuations in three monitoring points r_1c_1, r_1c_3 and r_2c_3 of the vaned diffuser passage were recorded for every numerical simulation developed. These points correspond to the reported measurement points for the experimental data set from Tsukamoto, and their location can be seen in Fig.1 and Fig.6.

In order to capture the RSI effects, the relationship between the pressure fluctuations and the movement of the rotor vanes in front of the diffuser vanes was extracted from the computed unsteady pressure field. Using a Fourier transform, the characteristic frequencies of the pressure fluctuations were obtained at points r_1c_1, r_1c_3 and r_2c_3. Results obtained show that the pressure fluctuates with the impeller blade passing frequency ZiN and its higher harmonics, (e.g. see Fig.8, more details in Refs. [15,16]).

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

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

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

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

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

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

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

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

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

In addition to the standard input for the length scale in form of a velocity gradient tensor $\partial U_i / \partial x_j$, SAS model computes a second length scale, called the von Kàrmàn length-scale, L_{vK} from the second derivative of the velocity field (see details in [24]). The information provided by L_{vK} allows the model to react more dynamically to capture scales in the flow field which cannot be handled by standard URANS models, because URANS recovers only the large-scale unsteadiness, whereas the **SAS-SST *k-ω*** model adjusts to the already resolved scales in a dynamic way and allows the development of a turbulent spectrum in the detached regions.

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

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

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

In order to explore this option, in the present work, the results by using SAS simulations were obtained and compared with the previous ones obtained using EVM, for the vaned diffuser pump under design conditions. Fig.12 shows a comparison among the results obtained using several EVM models, and SAS modeling with some calibration of the C_s parameter. This parameter has a control over the Lvk scale allowing the model to react more or less dynamically, and in this way it is possible to recover more or less unsteady effects. It is showed that a suitable calibration of SAS permits to adjust the pressure peaks in a better way at the r_1c_3 position, i.e., in the middle of the stator channel, just when the flow from the rotor goes into it (see Fig.6). Near the wall, i.e., at the r_1c_1 position, the viscous effects are more important than inertial ones and then, the unsteady effect is not so relevant and the SAS term is more insensitive to calibrations.

3. CFD FOR THE TURBOMACHINE UNDER OFF-DESIGN CONDITIONS

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

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

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

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

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

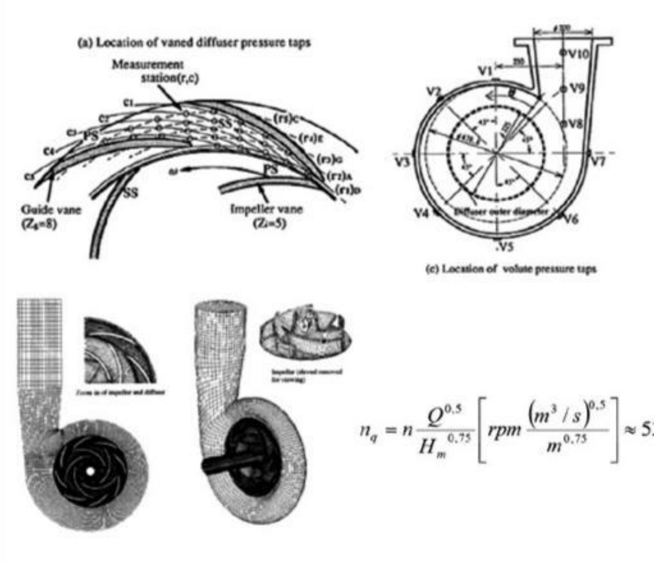
A complete analysis of these stalling cycles can be performed by means of CFD. In Fig.15 the pressure fluctuations in the diffuser are presented, showing an unsteady behavior related with the 'stalling cycle' in the channel diffuser. The cycle showed is related to an equivalent time of nine cycles of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

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

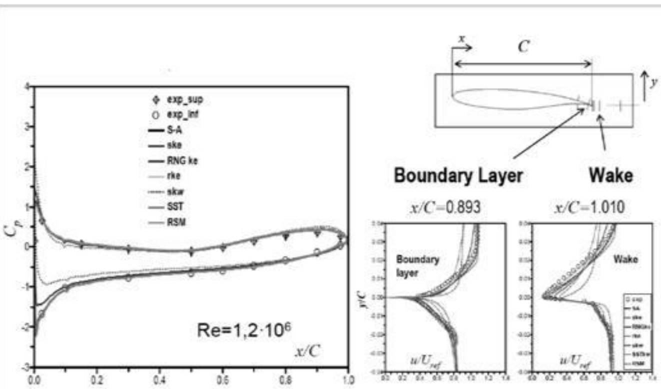
Finally it is necessary to remark that all the CFD results presented here, give good adjustments of the experimental results despite they are obtained in 2D geometry. The obtained results were also compared against CFD results for other 2D geometry from the literature [22] giving similar trends for the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

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

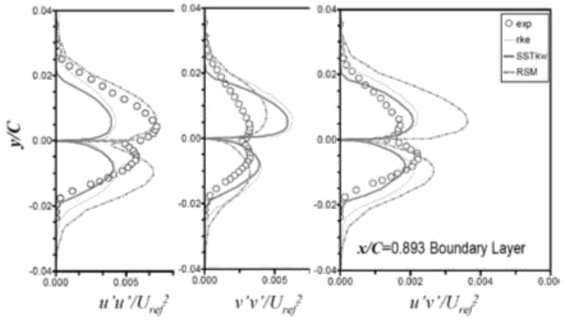
To reach this goal, it is necessary to reconstruct the 3D geometry using the given characteristics of the machine [11], by means of a 'classical design' of turbomachinery, (a skill not very common in the industrial engineers nowadays). A classical preliminary design was carried out to obtain the 3D geometry for this machine (see Fig.16), and the obtained 3D geometry of the rotor can be seen in Fig.17. It can be compared to the geometry used by Shi et al., 2001 [26], in his CFD work. After completing the 3D geometry design, a line of the future work comprises the 3D simulation of the complete machine using the quantitative criteria for the assessment of internal flow state obtained in this work.



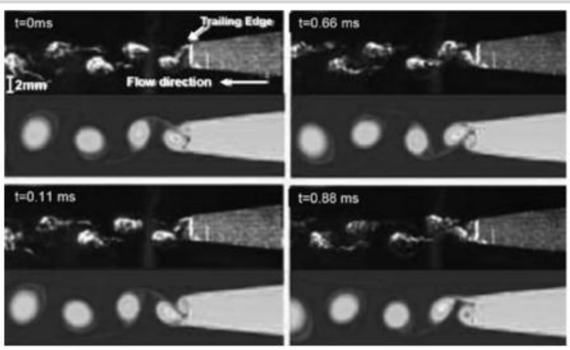
*[**Figure 1:** Experiments from Tsukamoto et al., 1995 [11]. It can be seen a schematics of the tested diffuser pump showing positions of the pressure measurement stations for the vaned diffuser and the locations of the volute pressure taps. Meshes showed correspond to 2D and 3D CFD cases from Shi et al, 2001[26].]*



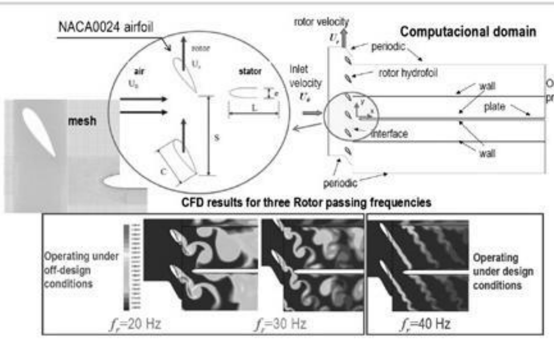
*[**Figure 2:** CFD Results for Boundary Layer and Wake flows over a supercritical profile airfoil: **Left, C_p** coef.; **Right**, mean velocities. Exp. from Nakayama et al., 1985 (see details in Refs. [16,17]). Nomenclature: **x/C** distance from leading edge of the hydrofoill/chord, **u/Uref**: local velocity value/mean velocity value.]*



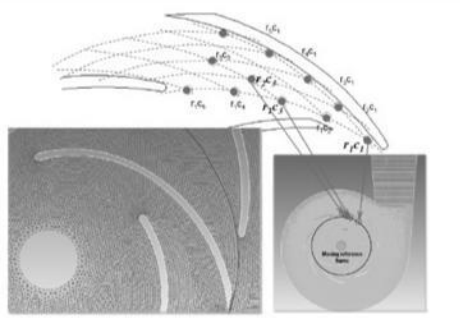
[Figure 3: CFD Results for Boundary Layer flow around a supercritical airfoil: fluctuating velocities along the boundary layer. Exp. from Nakayama et al., 1985 (see details in Refs. [16,17]). Nomenclature: $\boldsymbol{x'x'/U_{ref}^2}$*: local fluctuation / mean velocity value.]*



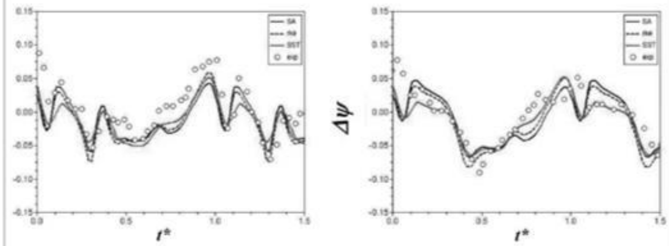
[Figure 4: CFD Results, Unsteady flow over a NACA 0009 truncated hydrofoil. The frequency of the von Kàrmàn vortex street is compared against experiments from Ausoni et al. 2005 (B/W pictures, see details in Ref. [17]).]



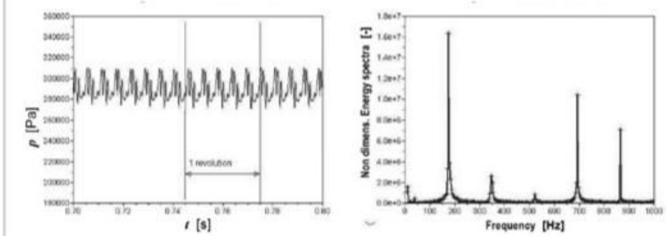
[Figure 5: CFD Results for a RSI phenomena. Up, mesh and geometry of the experimental setup for CFD. Down, Results obtained for fr=20Hz, 30Hz and 40Hz for turbulence intensity, $\left(I\%=\sqrt{\overline{u'^2}}/U_0\right)$ *, (see details in Refs. [16,17]. CFD results show that depending on fr, a different structure of the flow along the plate was obtained.]*



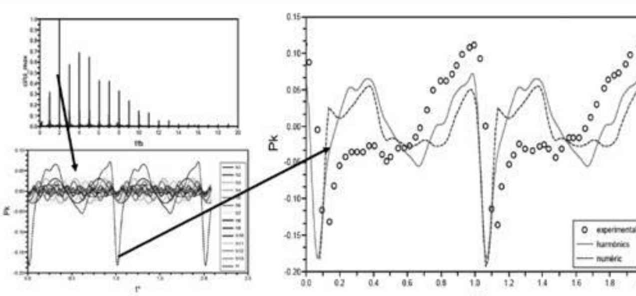
[Figure 6: Details of the meshed geometry for a diffuser pump. Notation: rici places where the unsteady pressures were measured [11].]



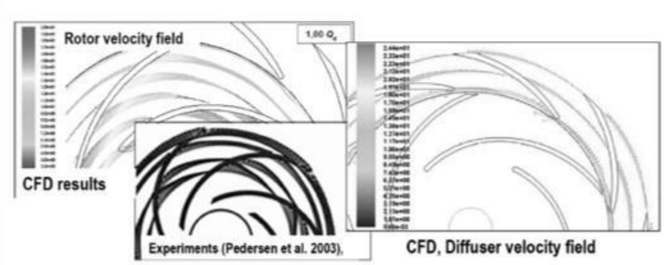
[Figure 7: CFD results using a mesh with $y_+\sim25$*. Left, point* r_1c_3*; Right, point* r_2c_3*.* $\boldsymbol{\Delta\Psi}=(p\text{-}ps)/(0.5\rho U_2U_2)$ *vs t*, where:* $\boldsymbol{\Delta\Psi}$*, non-dimensional unsteady pressure; (p-p*$_s$*) unsteady component of the relative pressure;* p_s*, total pressure at pump suction;* U_2*, peripheral speed of the impeller; t*= t/*T_i *, non-dimensional time;* T_i *, time required to traverse one pitch of impeller blade.]*



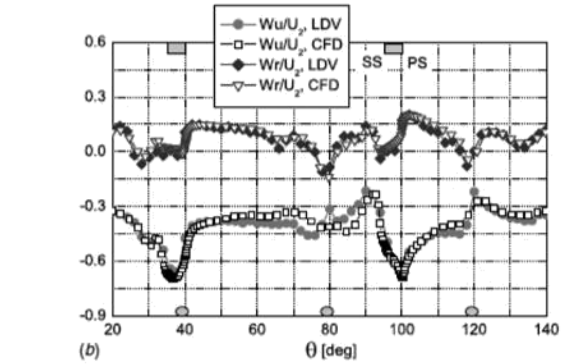
[Figure 8: CFD results for design conditions, (Q=1,0Q$_d$*) for the Spalart-Allmaras model.* ***Left****, CFD results for the pressure fluctuations at point* r_1c_3 *(see Fig.1);* ***Right****, frequency domain showing the harmonics of the unsteady pressure fluctuations obtained from the CFD computations.]*



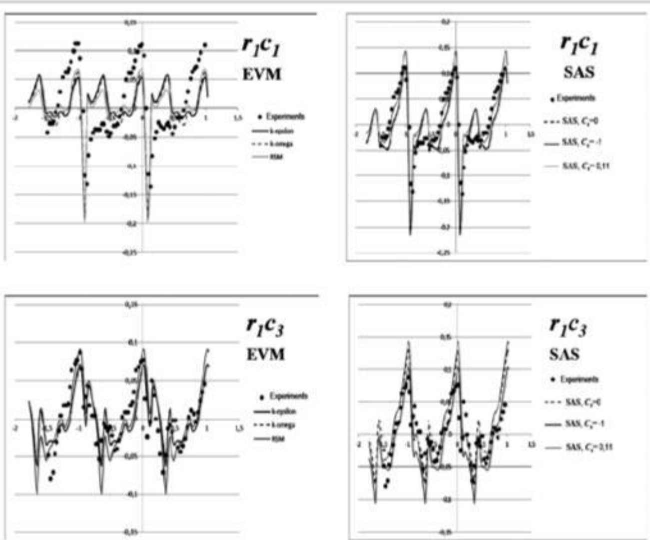
[Figure 9: CFD results using for design conditions, (Q=1,0Q$_d$*). From the frequency spectra obtained by CFD it is possible to recover the complete (potential+turbulence) unsteady signal. The arrows point out the composition process for obtaining the complete signal starting with individual frequencies by means of their addition.]*



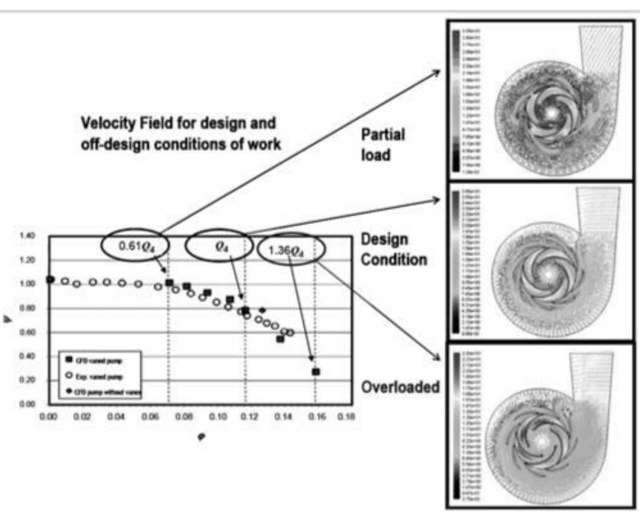
[Figure 10: CFD results using a mesh with $y_+\sim25$ *for design conditions (Q=1,0Q*$_d$*).* ***Left****, rotor velocity field;* ***Right****, diffuser (stator) velocity field. Notice that quantitatively, the velocity field is similar as one from Pedersen, 2003 [21]. This database corresponds to a centrifugal pump without diffuser.]*



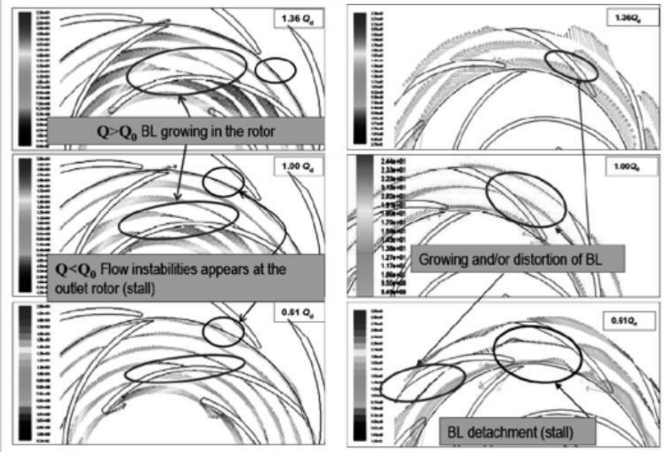
*[Figure 11: Comparison of the CFD results obtained against a similar radial machine (Feng et al., 2011 [22]): velocity field obtained, for design conditions ($Q=1{,}0Q_d$). Notation: LDV Laser Doppler Velocimeter measurements [22]; **Wu**, tangential component of the relative velocity; **Wr**, radial component of the relative velocity.]*



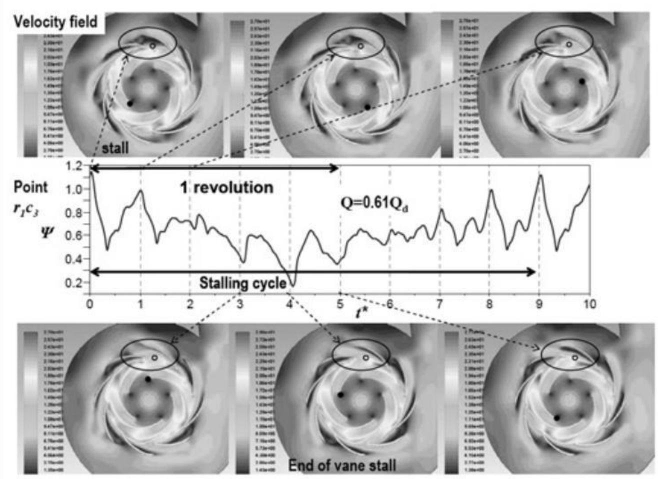
[Figure 12: CFD results for the non-dimensional head-flow rate $\psi = (2gH_mU_2^{-2})$ vs t. **Up**: r_1c_1 position; Down: r_1c_3 position; **Left: EVM**; **Right: SAS**. Nomenclature: • Experimental data (Tsukamoto, 1995).]*



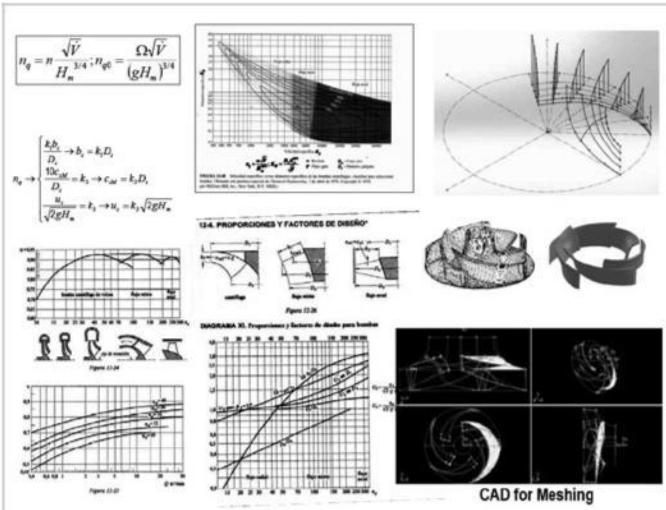
*[Figure 13: CFD results, (**ψ**, **φ**) **Left**: curve for the vaned pump; **Rigth**: Assocciated fluid flow pattern for some working conditions. Notation: Non-dimensional head-flow rate curve (**ψ**, **φ**); pressure coeff., $\psi = (2gH_mU_2^{-2})$; flow rate coeff., $\varphi = (Q) \times (2�b_2R_2U_2)^{-1}$; **O**, Exp. from Tsukamoto, 1995 [11].]*



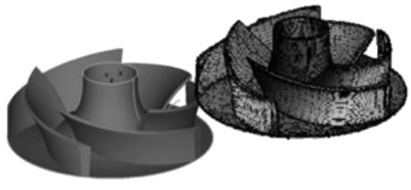
*[Figure 14: CFD results for the flow field under three operation conditions, vaned pump: **Left**: Flow pattern in the rotor, **Right**: Flow pattern in the diffuser (stator).]*



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



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



*[**Figure 17:** **Left**: Rotor configuration obtained by means of a 'classical design' of a mixed pump. **Right**: rotor configuration from Shi et al, 2001 [26].]*

4. CONCLUSIONS

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

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

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

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

5. REFERENCES

[1] Li H., Kelecy F., Egelja-Maruszewski A., Vasquez S., 2008, 'Advanced Computational Modeling of Steady And Unsteady Cavitating Flows', Proceedings of IMECE2008, 2008 ASME International Mechanical Engineering Congress and Exposition, Boston, Massachusetts, USA.

[2] Coussirat M., 2003, 'Theoretical/Numerical Study of Flows with Strong Streamlines Curvature', PhD Thesis, Universitat Politècnica de Catalunya, Barcelona, Spain.

[3] Kubota A., Kato H., and Yamaguchi H., 1992, 'A New Modeling of Cavitating Flows: A Numerical Study of Unsteady Cavitation on a hydrofoil section', J. Fluid Mech., 240, pp. 59–96.

[4] Singhal A., Athavale M., Li H., and Jiang Y., 2002,'Mathematical Basis and Validation of Full Cavitation Model', ASME J. Fluids Eng. 124, pp. 617–624.

[5] Zwart J., Gerber A., Belamri T., 2004, 'Two-Phase Flow Model for Predicting Cavitation Dynamics', Proc. 5th International Conference on Multiphase Flow, Yokohama, Japan, paper 152.

[6] Kozubková M., Rautová J., and Bojko M., 2012,'Mathematical Model of Cavitation and Modeling of Fluid Flow in Cone', Procedia Engineering 39, pp. 9–18.

[7] Franc J. and Michel J,, 2004, Fundamentals of Cavitation, Kluwer Academic Publ., New York USA.

[8] Peters A., Sagar H., Lantermann U., Moctar O., 2015, 'Numerical modeling and prediction of cavitation erosion', Wear 338 339, pp. 189 201.

[9] Duan L., Yuan S., Hub L., Yang W., Yu J., Xia X., 2016, 'Injection performance and cavitation analysis of an advanced 250 MPa common rail diesel injector', Int. J. of Heat and Mass Transfer 93, pp. 388-397.

[10] Coussirat, M., Fontanals, A., Grau, J., Guardo, A. and Egusquiza. E., 2008, 'CFD study of the boundary layer influence on the wake for turbulent unsteady flow in rotor-stator interaction'. IAHR 24th Symposium on Hydraulic Machinery and Systems. Foz do Iguassu, Brazil (2008).

[11] Tsukamoto, H., Uno, M., Hamafuku, H., and Okamura, T., 1995, 'Pressure Fluctuation Downstream of a Diffuser Pump Impeller,' The 2nd Joint ASME/JSEM Fluids Engineering Conference, Forum of Unsteady Flow, FED-Vol. 216, pp. 133– 138.

[12] Ansys Inc. Ansys Fluent Software, http://www.ansys.com/ Industries/Academic/Tools/

[13] Coussirat M., Moll F., Cappa F., Fontanals A., 2016, 'Study of Available Turbulence and Cavitation Models to Reproduce Flow Patterns In Confined Flows',Journal of Fluids Engineering, (in press)

[14] Fontanals A., Coussirat M., Guardo A. and Egusquiza E., 2010, 'Detailed Study of the Rotor-Stator Interaction Phenomenon in a Moving Cascade of Airfoils', IAHR 25th Symposium on Hydraulic Machinery and Systems. Timisoara, Romania.

[15] Fontanals A., Guardo A., Coussirat M. and Egusquiza E., 2011, 'Numerical Study of the Fluid–Structure Interaction in the Diffuser Passage of a Centrifugal Pump', COUPLED PROBLEMS 2011, IV Int. Conf. on Comput. Methods for Coupled Problems in Scie. and Eng.

[16] Fontanals A., 2012,'Caracterització del flux a causa de la interacció rotor-estator en turbomàquines mitjançant CFD', PhD Thesis, Universitat Politètcnica Catalunya, Barcelona, Spain.

[17] Coussirat M., Fontanals A., Guardo A., 2012 'CFD Study of the Rotor-Stator Interaction in a Centrifugal Pump With Diffuser', Anales AFA Vol. 23 Fluidos (34-38) Bs.As., Argentina.

[18] Fontanals A., Guardo A. Zobeiri A., Egusquiza E., Farhat M. and Avellan F., 2014, 'Boundary Layer Effects on the Vortex Shedding in a Donaldson- type Hydrofoil', IAHR 27th Symposium on Hydraulic Machinery and Systems. Montreal, Canada.

[19] Guardo A., Fontanals A., Coussirat M., 2013, 'Estudio Numerico de la Interaccion Rotor Estator en el Difusor se Una Bomba,' Mecánica Computacional Vol XXXII, AMCA, págs. 1155-1168.

[20] Rodriguez C., Egusquiza E. and Santos I., 2007, 'Frequencies in the vibration induced by the rotor stator interaction in acentrifugal pump turbine', Journal of Fluids Engineering, vol.129, no. 11, pp. 1428–1435

[21] Pedersen, N., Larsen, P., Jacobsen, C., 2003 'Flow in centrifugal pump impeller at design and off design conditions- Part I: Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) measurements'. Journal of Fluids Engineering, Vol. 125.

[22] Feng J. Benra F, Dohmen H., 2011, 'Investigation of periodically unsteady flow in a radial pump by CFD simulations and LDV measurements'. Journal of Turbomachinery, Vol 133.

[23] Menter F., Kuntz M. and Langtry R., 2003, 'Ten Years of Industrial Experience with the SST Turbulence Model', Turbulence, Heat and Mass Transfer 4, K. Hanjalic et al., eds., House, Inc., pp. 625–632.

[24] Menter F. and Egorov Y., 2010, 'The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description', Flow Turbulence Combust. 85, pp. 113–138.

[25] Egorov Y., Menter F., Lechner R., and Cokljat D., 2010,'The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Complex Flows', Flow Turbulence Combust. 85, pp. 139–165.

[26] Shi, F., and Tsukamoto, H., 2001, 'Numerical Study of Pressure Fluctuations Caused by Impeller-Diffuser Interaction in a Diffuser Pump Stage, 'ASME J. Fluids Eng., 123, pp. 466–474. [Launder, 1974] Launder B. and Spalding D., 1974, 'The Numerical Computation of Turbulent Flows', Computer Methods in Applied Mechanics and Engineering 3, pp. 269– 289.