POLYPUBLIE
Polytechnique Montréal

POLYTECHNIQUE MONTRÉAL
UNIVERSITÉ D'INGÉNIERIE

Titre: Optimization of the inlet velocity profile of the turbine 99 draft tube
Title:

Auteur: Sergio Ricardo Galvan
Author:

Date: 2007

Type: Mémoire ou thèse / Dissertation or Thesis

Référence: Galvan, S. R. (2007). Optimization of the inlet velocity profile of the turbine 99 draft tube [Thèse de doctorat, École Polytechnique de Montréal]. PolyPublie. https://publications.polymtl.ca/8057/
Citation:

Document en libre accès dans PolyPublie
Open Access document in PolyPublie

URL de PolyPublie: https://publications.polymtl.ca/8057/
PolyPublie URL:

Directeurs de recherche: Marcelo Reggio, & François Guibault
Advisors:

Programme: Non spécifié
Program:

Ce fichier a été téléchargé à partir de PolyPublie, le dépôt institutionnel de Polytechnique Montréal
This file has been downloaded from PolyPublie, the institutional repository of Polytechnique Montréal

https://publications.polymtl.ca

UNIVERSITÉ DE MONTRÉAL

OPTIMIZATION OF THE INLET VELOCITY PROFILE OF THE TURBINE 99 DRAFT TUBE

SERGIO RICARDO GALVAN
DÉPARTEMENT DE DÉPARTEMENT DE GÉNIE MÉCANIQUE
ÉCOLE POLYTECHNIQUE DE MONTRÉAL

THÈSE PRÉSENTÉE EN VUE DE L'OBTENTION
DU DIPLÔME DE PHILOSOPHIÆ DOCTOR (Ph.D.)
(GÉNIE MÉCANIQUE)
JUIN 2007

© Sergio Ricardo Galvan, 2007.

Library and Archives Canada

Published Heritage Branch

395 Wellington Street
Ottawa ON K1A 0N4
Canada

Bibliothèque et Archives Canada

Direction du Patrimoine de l'édition

395, rue Wellington
Ottawa ON K1A 0N4
Canada

Your file Votre référence
ISBN: 978-0-494-35512-1
Our file Notre référence
ISBN: 978-0-494-35512-1

NOTICE:
The author has granted a non-exclusive license allowing Library and Archives Canada to reproduce, publish, archive, preserve, conserve, communicate to the public by telecommunication or on the Internet, loan, distribute and sell theses worldwide, for commercial or non-commercial purposes, in microform, paper, electronic and/or any other formats.

The author retains copyright ownership and moral rights in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

AVIS:
L'auteur a accordé une licence non exclusive permettant à la Bibliothèque et Archives Canada de reproduire, publier, archiver, sauvegarder, conserver, transmettre au public par télécommunication ou par l'Internet, prêter, distribuer et vendre des thèses partout dans le monde, à des fins commerciales ou autres, sur support microforme, papier, électronique et/ou autres formats.

L'auteur conserve la propriété du droit d'auteur et des droits moraux qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

In compliance with the Canadian Privacy Act some supporting forms may have been removed from this thesis.

While these forms may be included in the document page count, their removal does not represent any loss of content from the thesis.

Conformément à la loi canadienne sur la protection de la vie privée, quelques formulaires secondaires ont été enlevés de cette thèse.

Bien que ces formulaires aient inclus dans la pagination, il n'y aura aucun contenu manquant.

Canada

UNIVERSITÉ DE MONTRÉAL

ÉCOLE POLYTECHNIQUE DE MONTRÉAL

Cette thèse intitulée :

OPTIMIZATION OF THE INLET VELOCITY PROFILE OF THE TURBINE 99 DRAFT TUBE

présentée par : GALVAN Sergio Ricardo

en vue de l'obtention du diplôme de : Philosophiæ Doctor

a été présentée au jury d'examen constitué de :

M. PELLETIER Dominique, Ph.D., président

M. REGGIO Marcelo, Ph.D., membre et directeur de recherche

M. GUIBAULT François, Ph.D., membre et codirecteur de recherche

M. TRÉPANIER Jean-Yves, Ph.D., membre

M. GHALY Wahid, Ph.D., membre

To my wife, Maria-Elena Villarreal for the emotional support, freedom, and encouragement to develop this research.

ACKNOWLEDGMENTS

I would like to thank to the National Council of Science and Technology of Mexico (CONACYT) for their financial support during my Ph.D. studies. I also wish to thank Prof. Marcelo Reggio and Prof. Francois Guibault for direction, scientific supervision and support through this work. I also would like to thank to Dr. Maryse Page and Robert Magnan, from the Institut de Recherche d'Hydro-Québec (IREQ) and Mr. Thi Vu from General Electric Hydro, experts in draft tube performance, for their valuable discussions and comments. I also would like to thank Dr. Christophe Tribes for his priceless help on the optimization process setup. Specials tanks also Dr. Juan Abanto for the FLUENT support. Many thanks also to all the people who help me in any way to develop this work.

Montréal Québec, June 12, 2007.

Sergio Galvan

RÉSUMÉ

Au cours des dernières années, les recherches numériques et expérimentales sur la performance des aspirateurs dans les turbines hydrauliques ont confirmé l'importance du tourbillon causé par les aubes de la roue sur le rendement global de la turbine. Les résultats indiquent que la spécification d'une distribution optimale de l'écoulement d'entrée qui donne la meilleure performance de la machine représente encore un défi. Il existe par conséquent un besoin d'adapter le couplage roue-aspirateur pour minimiser les pertes d'énergie en fonction de l'écoulement à l'entrée de l'aspirateur. Ainsi, cette étude vise à développer une méthode d'optimisation pour améliorer les caractéristiques de l'écoulement à l'entrée des aspirateurs. L'intégration des logiciels iSIGHT, FLUENT et MATLAB au sein d'une méthodologie globale d'optimisation a permis de déterminer la meilleure forme du profil de vitesse qui permette d'augmenter l'efficacité totale de la machine. On a étudié une représentation analytique de l'équation de trois de vortex développée pour une turbine Francis, laquelle a été modifiée pour tenir compte de la vitesse près des parois. Puis, on a effectué une étude de sensibilité pour identifier la meilleure fonction objectif par rapport aux paramètres représentatifs du profil d'entrée de l'aspirateur. L'algorithme d'optimisation et le modèle numérique de l'aspirateur ont été validés également pour assurer l'obtention de résultats fiables. Finalement, afin d'accroître notre compréhension de la physique de l'écoulement, une analyse des champs de pression et de vitesse causés par les profils optimisés a été effectuée. Suite à l'application de cette méthodologie, il a été possible de confirmer que la symetrie du vortex vers la sortie de l'aspirateur est un facteur très important pour minimiser les pertes d'énergie. On s'attend à ce qu'un profil correctement modélisé de vitesse d'entrée puisse supprimer ou atténuer les caractéristiques indésirables dans l'aspirateur telles que les écoulements secondaires, l'évolution irrégulière de l'écoulement (flux inverse) et la vitesse excessive.

ABSTRACT

In recent years, numerical and experimental investigations on the draft tube performance have confirmed the importance of the inlet swirling flow created by the runner vanes. The results indicate that it is still a challenge to get the optimal flow distribution at the draft tube inlet which gives the best machine performance over a range of operation points. Consequently, there is a need to adjust the runner-draft tube coupling to minimize the losses arising from the inlet flow distribution.

In consequence, this study focuses on establishing an optimization method to improve the swirling flow characteristics at the draft tube inlet. The coupling of iSIGHT, FLUENT and MATLAB commercial softwares has been performed to determine the best inlet flow shape which leads to increase overall machine performance. An analytical representation for the outlet swirling flow of the Kaplan turbine runner is studied based on a modification of the three-vortex system equations applied for a Francis turbine to account for the near wall axial velocity increase at the draft tube entrance. Then, to identify the best objective function, a study has been carried out of the most representative swirl parameters on the draft tube performance quantities. The optimization algorithm and the draft tube numerical model were validated to obtain reliable results. Finally, in order to increase the understanding of the flow physics, an analysis of the velocity and pressure fields caused by the optimized profiles is presented.

This study confirms that the vortex symmetry at the outlet is an important issue to minimize the energy losses through the draft tube which has been achieved by the optimization methodology. It is expected that a properly modeled inlet velocity profile can suppress or mitigate the undesirable draft tube flow characteristics such as secondary flow, irregular evolution and excessive velocity through this important component of the turbine.

CONDENSÉ EN FRANÇAIS

Introduction

Une turbine hydraulique est un dispositif mécanique qui convertit l'énergie potentielle de l'eau, liée à une différence d'altitude, en travail. Elle se compose principalement du réservoir de stockage suivi du conduit forcé, de la bâche spirale et de la roue, où l'énergie de l'écoulement se transforme en énergie mécanique. L'écoulement qui sort de la roue perd de la vitesse dans l'aspirateur, ce qui transforme l'énergie cinétique résiduelle de l'eau en énergie de pression. Cette récupération d'énergie affecte significativement l'efficacité et la puissance de sortie des machines, surtout celles qui présentent une basse chute (haute vitesse spécifique) et qui travaillent régulièrement hors du point de meilleure efficacité. Dans des machines de basse chute et grand débit, les pertes de l'aspirateur peuvent atteindre jusqu'à 50% des pertes totales de la machine.

C'est ainsi que l'analyse de l'écoulement dans l'aspirateur est d'un grand intérêt pratique pour les concepteurs de nouvelles machines ou ceux qui effectuent la réadaptation des machines déjà existantes. Par conséquent, ceci a généré de nombreuses études du calcul de l'écoulement dans cette composante ces dernières décennies, (Avellan, 2000, Masse, 1999, Ruprecht et al., 2000b, Mauri et al., 2002b, Iliescu et al., 2002). Les résultats de ces recherches ont montré que la plupart des pertes d'efficacité dans l'unité sont provoquées par les caractéristiques de l'écoulement à l'entrée du diffuseur. Dans ce cas, le tourbillon induit par les aubes de la roue en est la cause principale. Ceci explique l'importance d'analyser et de prédire de façon conjointe, le comportement hydrodynamique de la roue et de l'aspirateur.

Une revue de l'état de l'art démontre que le couplage hydrodynamique idéal reste encore un défi. On peut alors poser l'hypothèse qu'il est possible de modifier les caractéristiques de l'écoulement à l'entrée de l'aspirateur de telle façon que la performance soit augmentée. Il devient alors essentiel de déterminer quelles sont les caractéristiques d'un

écoulement qui provoquent des pertes d'énergie dans l'aspirateur.

Notre proposition pour essayer de répondre à ces questionnements est d'optimiser le profil de vitesse à l'entrée de l'aspirateur. Lors de cette optimisation, on devra explorer l'espace de design des profils pour trouver une famille spécifique de profils qui minimise les pertes d'énergie, ce qui pourrait améliorer le couplage roue et aspirateur.

La motivation qui nous amène à développer ce projet est d'obtenir une avance scientifique dans la compréhension de l'influence des conditions aux frontières sur la performance de l'aspirateur. Et une avance industrielle, puisque le gain de quelques points seulement sur le rendement du diffuseur peut augmenter la génération d'électricité de plusieurs kWh.

État de l'art.

On sait que l'écoulement dans l'aspirateur est très complexe, non stationnaire et turbulent. Tout ceci à cause de la nature tournante de l'écoulement à la sortie de la roue et de la géométrie de l'aspirateur. Ceci entraîne la non-uniformité des champs de vitesse et de pression et des instabilités des valeurs en chaque point du champ. Ainsi, les caractéristiques de l'écoulement à la sortie de la roue ont une structure difficile à représenter.

Depuis des années, cette structure de l'écoulement a intéressé plusieurs chercheurs et a été étudiée de façon expérimentale, en utilisant des simulations numériques et à travers des modèles mathématiques. Cependant, la modélisation et l'étude du phénomène du tourbillon ne sont pas précisément le domaine d'application de cette recherche.

Par contre, nous nous intéressons à l'influence du tourbillon d'entrée sur le rendement de l'aspirateur. Sur ce sujet, des études expérimentales ont été effectuées en utilisant différentes représentations de l'écoulement d'entrée. Cette relation tourbillon-performance, a été étudié autant de façon expérimentale que numérique.

Récemment, des outils basés sur la dynamique des fluides numérique et les algorithmes d'optimisation ont été utilisés pour améliorer la performance des turbomachines. Plusieurs recherches ont été effectuées pour optimiser les profils d'aubes de roues de type

Francis et Kaplan et dans la conception de la géométrie des aspirateurs. Jusqu'à présent, l'optimisation des profils de vitesse a été dirigée essentiellement vers les processus de forgeage du métal et de processus chimiques. Ces recherches montrent que la manipulation du profil de vitesse par des algorithmes d'optimisation permet d'améliorer la performance de ces processus.

Ainsi, cette revue bibliographique nous ammène à conclure que l'optimisation du profil de vitesse à l'entrée d'un aspirateur, en réponse à une simulation numérique, aidera à augmenter la performance des installations hydroélectriques. Cette recherche n'a pas été développée jusqu'à présent.

Méthodologie

La revue bibliographique a montré que l'optimisation numérique du profil de vitesse à l'entrée des aspirateurs est un nouveau défi parce que le flux est tridimensionnel donc très cher à évaluer numériquement et très sensible aux conditions aux frontières. Entre autres, il faut prendre en compte la non-linéarité et la multimodalité des résultats de performance de l'aspirateur provoqué par la modification des conditions d'entrée. Ainsi, on a proposé une méthodologie pour l'optimisation numérique de l'écoulement dans l'aspirateur comprenant 3 étapes, soit :

- La paramétrisation du profil de vitesse.
- La validation du modèle numérique.
- L'évaluation de l'algorithme d'optimisation, y compris la fonction objectif.

Cependant, il existe le risque de produire un profil dont la forme ne soit pas facilement obtenue par le concepteur. Mais, dans cette recherche, l'objectif est de savoir s'il est possible d'obtenir le rendement maximal de l'aspirateur plutôt que d'évaluer la faisabilité technique et structurale de l'aube.

Paramètrisation du profil de vitesse.

La paramétrisation du profil de vitesse est cruciale pour bien représenter l'écoulement d'entrée et bien effectuer le processus d'optimisation. En conséquence, les principales caractéristiques d'une bonne paramétrisation doivent être :

1. D'utiliser le nombre minimal de variables tout en maintenant les propriétés de continuité et en permettant d'approximer les données expérimentales.
2. De bien correspondre et expliquer la structure physique de l'écoulement.

Afin de satisfaire ces conditions, nous avons étudié trois méthodes de représentation de l'écoulement à l'entrée de l'aspirateur.

La première méthode analysée s'appuyait sur la dépendance entre la forme du profil de vitesse sur l'aube de la roue et les régions d'attachement du fluide. Nechleba (1957) and Kazan (1962) ont utilisé un réseau formé par une vitesse méridienne constante le long de l'arête de sorti de la roue et des lignes de courant sur chaque canal de la roue, en se basant sur l'équation de continuité le long de cette courbe. De cette façon, en utilisant les triangles de vitesse d'entrée, de vitesse de sortie et des équations fondamentales de la théorie des turbomachines, il a été possible d'obtenir les composantes des vitesses pour n'importe quel point le long du bord de fuite de l'aube de la roue. Malheureusement, comme les conditions à l'entrée de l'aspirateur ne sont pas identiques pour chaque ligne de courant, il y aura une grande interdépendance entre les variables de conception pour représenter le profil de vitesse. Ainsi, on devra avoir un nombre important de points pour bien construire le profil de vitesse. Puisque plusieurs paramètres définissent l'aube de la roue, le processus se transforme en une optimisation multivariable , ce qui rend le processus complexe et numériquement très cher.

Une autre approche a été étudiée, qui utilise des équations analytiques inspirée de la modélisation de l'évolution des propriétés des fluides dans un processus de fabrication de matériaux composites, encapsulées dans le logiciel Polyfit, (Ruiz, 2004). Ce logiciel a été

utilisé pour approcher des donnes expérimentales de la vitesse d'entrée de l'aspirateur des centrales hydrauliques La Grande 3 (Québec) et Huites (Mexique), ceux établis par Mauri et al. (2002a) à l'aide de différentes équations mathématiques. Ce logiciel a recourt à la méthode de régression non linéaire (Levenverg-Marquardt) en utilisant un algorithme itératif pour améliorer cette approximation. Malheureusement, les équations évaluées ne donnent pas une bonne approximation des profils expérimentaux.

Susan-Resiga et al. (2004) a exposé une façon analytique de représenter l'écoulement d'entrée d'un aspirateur stationnaire non visqueux et axisymétrique en utilisant une superposition de trois tourbillons. Pour la vitesse tangentielle et la vitesse axiale : deux vortex de Batchelor, l'un avec le même sens de rotation et une vitesse axiale positive et l'autre en sens oposée et une vitesse axiale negative par rapport à un vortex forcée ($w = \Omega_0\, r$) avec une vitesse uniforme ($u = U_0$) au coeur de la section d'entrée. Le coefficient de décharge de la machine est obtenu par l'intégration de l'équation de la vitesse axiale. Les résultats d'approximation aux données expérimentales (La Grande 3 et Huites) montrent un bon ajustement pour les trois composantes de la vitesse, cependant, un des aspects que l'on doit considérer est le défaut dans la modélisation près de la paroi. Donc, on a ajouté une équation pour mieux représenter le profil de vitesse près de la paroi et aussi près du cône sans augmenter le nombre de variables.

Toutefois, l'addition de la vitesse radiale et de la vitesse près de la paroi pourrait provoquer certains problèmes mathématiques. Des recherches sur la performance de l'aspirateur ont montré l'importance de ces vitesses pour bien effectuer une simulation numérique. Mais, avec l'adition de ces deux équations, on note au passage que ça ne satisfaisait pas la continuité si on prenait dans le volume, mais comme on le prend au bord de la section d'entrée, les composants dans le volume vont s'ajuster. Ainsi, le champ divergent de la vitesse n'est pas nécessairement inappliqué et cette anomalie sera immédiatement corrigée par le premier élément contigu à la section d'entrée.

Validation du modèle numérique de l'aspirateur.

Plusieurs facteurs essentiels dans une simulation numérique, tels que la génération du maillage, l'algorithme de discrétisation et la modélisation de la turbulence, sont loin d'être validés pour tous les types d'application, entre autres à cause de la complexité physique des phénomènes en jeu. Une validation systématique de chacun de ces aspects a donc été entreprise. Àu niveau des modèles de turbulences, le modèle $k-\epsilon$ (standard) est très populaire à cause de sa simplicité et de sa robustesse. Récemment, suite à la grande utilisation de ce modèle, certains désavantages du modèle ont été identifiés et des améliorations ont été apportées (Renormalization group theory, realizable) pour augmenter sa performance. La modélisation numérique de l'écoulement dans l'aspirateur est un exemple de l'utilisation de la Dynamique de Fluides Numérique pour étudier des écoulements industriels. Ainsi, le logiciel FLUENT a été utilisé pour étudier l'interaction des variations du modèle de turbulence $k-\epsilon$ avec les schémas de discretisation et la modélisation près de la paroi. Les résultats ont montré que qualitativement le modèle de turbulence RNG donne la meilleure approximation des résultats expérimentaux et que cette approximation dépend du paramètre d'intensité du tourbillon. Cependant, ce modèle utilise un temps de calcul très important. Comme nous considérons qu'une convergence optimale des simulations et une représentation extrêmement détaillée de la fonction objectif ne sont pas indispensables pour assurer la convergence du processus d'optimisation, nous avons choisi d'utiliser le modèle standard, qui a montré sa fiabilité pour modéliser les écoulements avec une structure tourbillonnaire.

Évaluation de l'algorithme d'optimisation.

L'algorithme d'optimisation doit être capable de générer une variété de profils de vitesse pour explorer l'espace de design et ainsi minimiser les pertes d'énergie dans l'aspirateur. Pour réaliser cette exploration, on a sélectionné une méthode globale et stochastique de

recherche, l'algorithme génétique, parce que :

- Il a été utilisé dans beaucoup de problèmes d'optimisation de conception de turbomachines.
- La recherche de l'optimum se déroule dans un grand espace, c.à.d. l'algorithme est un générateur de profils de vitesse.
- L'algorithme est applicable à une très grande variété de problèmes, y compris les problèmes de simulation.

Mais, dans ce processus d'optimisation, le calcul de la fonction objectif consomme la plus grande quantité du temps parce qu'elle est le résultat d'une simulation tridimensionnelle. En plus, le processus nécessite une très grande quantité d'évaluations à cause de la recherche globale de l'algorithme. Par conséquent, on a développé une structure de parallélisation pour distribuer les calculs sur différentes machines. Le logiciel iSIGHT fournit un algorithme d'optimisation distribué qui a l'avantage d'être capable d'exécuter chaque tâche sur une machine à distance et d'exécuter plusieurs tâches en parallèle.

Malheureusement, l'efficacité du *Multi Island Genetic Algorithm* (MIGA) est tributaire d'autres paramètres qui doivent être testés. Il y a deux fonctions qui ont été utilisées pour calibrer ces paramètres. La première est la fonction Rosenbrock qui est unimodale. En plus, cette fonction nous a intéressé parce qu'il y a une grande interdépendance entre les variables, comme dans notre cas. Un autre test a été fait en utilisant la fonction Rastrigin. Cette fonction est multimodale, c'est à dire qu'elle a plusieurs minimums et elle peut tester la capacité de l'algorithme à trouver le minimum global.

Une partie très importante du processus d'optimisation a été la définition de la fonction objectif. Cette définition consiste à évaluer la réponse de l'aspirateur à travers des quantités globales de performance tels que, le coefficient de récupération de pression moyenne, le coefficient de récupération de pression sur la paroi et le coefficient de perte d'énergie à des modifications des paramètres du profil de vitesse. Pour ce faire, on a utilisé le design factoriel qui a été mis en oeuvre dans le contexte des aspirateurs par Cervantes & Eng-

ström (2004). Les résultats montrent que le coefficient moyen de récupération de pression est moins sensible aux variations des paramètres. Par contre, le coefficient évalué sur la paroi de l'aspirateur est dominé localement par les paramètres du deuxième vortex. Enfin, le coefficient de perte d'énergie semble être plus sensible aux paramètres du profil de vitesse. Ainsi, ce dernier coefficient sera utilisé comme fonction objectif pendant le processus d'optimisation.

Application de la méthodologie d'optimisation à un diffuseur.

Cette section présente la validation du processus d'optimisation, lequel a été appliqué à un diffuseur. La géométrie et les conditions aux frontières, de même que les données expérimentales, ont été fournies par General Electric Hydro. Pour la validation du modèle du diffuseur, les résultats ont été comparés de façon quantitative et qualitative à des valeurs numériques et expérimentales.

Trois systèmes de vortex d'entrée différents ont été employés, dans le but d'évaluer l'impact de la distribution radiale de vitesse sur la performance du diffuseur. Ceci a permis de générer des configurations d'écoulement le long du cône qui ont aidé à comprendre les effets dominants des vortex d'entrée. Il semble que pour minimiser les pertes d'énergie dans le diffuseur, il faut trouver un niveau optimum d'intensité de tourbillon et une distribution radiale qui ne provoque ni la séparation de l'écoulement à la paroi ni de flux inverse. La vitesse radiale, qui n'a pas été considérée dans les études précédentes, a une influence importante sur la séparation de l'écoulement à la paroi. En outre, le profil optimal a produit de la turbulence dans la région près de la paroi plutôt que dans la région du coeur de l'écoulement. Il convient de noter que quand l'uniformité globale de l'écoulement à la sortie du cône augmente, la performance du diffuseur est améliorée. Ainsi, optimiser l'intensité du tourbillon en aval est une question importante pour pousser l'écoulement vers la paroi tout le long du diffuseur.

Un autre essai a été effectué pour déterminer l'influence du débit pendant le développe-

ment du processus d'optimisation. Les résultats ont montré une petite influence du débit sur le comportement du coefficient de perte d'énergie. L'extrapolation des caractéristiques du profil d'entrée le long du domaine, nous permet d'obtenir des valeurs semblables à celles obtenues au cours du processus d'optimisation sans restriction sur l'écoulement. Ainsi, une augmentation du nombre de Reynolds n'affecte pas de manière significative les résultats, ce qui montre la nature non visqueuse des instabilités provoquées par les profils de vitesse d'entrée.

Cette étude du processus d'optimisation, en utilisant un diffuseur précédemment validé, nous a permis d'établir les paramètres de l'algorithme d'optimisation, le système d'équations et l'intensité du débit pour le processus d'optimisation de l'aspirateur de la turbine 99.

Optimisation du profile de vitesse d'entrée de la Turbine 99.

L'optimisation du profil de vitesse d'entrée vise à démontrer la capacité d'améliorer l'efficacité de l'aspirateur d'une centrale hydroélectrique de basse chute. D'abord, le profil expérimental de vitesse d'entrée produit par une roue de type Kaplan a été approché aux données expérimentales en utilisant une équation modifiée pour mieux représenter l'écoulement près de la paroi. Puis, le maillage fin fourni lors du colloque Turbine 99 a été remplacé par un maillage grossier afin de réduire d'une manière considérable le temps de calcul. Ce maillage a été montré se trouver dans la limite de la région asymptotique de convergence pour le problème. Les résultats ont prouvé que même s'il y a une différence numérique parmi les quantités globales de performance, ils ont la même tendance. Ainsi, le maillage grossier pouvait être employé pour le processus d'optimisation.

La boucle d'optimisation employée pour étudier le comportement du profil de vitesse d'entrée dans le diffuseur a ensuite été appliquée à l'aspirateur du colloque Turbine-99. Cette boucle a permis de déterminer le profil de vitesse pour lequel le facteur de perte d'énergie a été réduit. Ceci a permis de conclure que l'utilisation d'un algorithme d'op-

timisation est nécessaire pour produire automatiquement différentes formes de profils de vitesse d'entrée qui minimisent les pertes d'énergie dans l'aspirateur. Finalement, on a étudié la performance du flux dans l'aspirateur provoqué par les profils optimisés de vitesse d'entrée. Les résultats sont semblables à ceux obtenus avec le diffuseur, en ce qui concerne l'importance de l'uniformité d'écoulement à la sortie. Si le paramètre de quantité de mouvement peut être réduit à l'extrémité de l'aspirateur, sa performance augmente considérablement. Un autre aspect important est le blocage qui produit un gradient défavorable de pression au centre de l'écoulement de la roue. Ceci détériore l'effet de récupération de pression. Par contre, la structure topologique du champ de vitesse change avec l'uniformité de l'écoulement, ce qui provoque la disparition de l'effet du blocage.

Conclusions.

Ce travail a permis de minimiser les pertes d'énergie dans l'aspirateur, provoquées par l'écoulement induit par la roue. En utilisant une méthodologie d'optimisation, il a été possible d'optimiser l'écoulement d'entrée, et par conséquent, de réduire les pertes d'énergie de 78% et ainsi d'améliorer la performance de l'aspirateur de 5.6%.

Entre autres, il a été possible de connaître les principales caractéristiques de l'écoulement qui provoquent l'amélioration de la performance. Quantitativement, ces caractéristiques sont la non-uniformité et l'intensité du tourbillon à la sortie de l'aspirateur et qualitativement, le blocage au centre du cône et la symétrie du flux secondaire. Aussi, il a été démontré que le débit n'est pas un facteur important pour améliorer la performance si on maintient la même forme de profil d'entrée dans une plage de débit. D'autre part, on a trouvé qu'il y a une seule intensité du tourbillon d'entrée pour laquelle il est possible de minimiser les pertes d'énergie. Finalement, on a démontré qu'un maillage grossier peut donner des résultats comparatifs fiables pendant l'optimisation pourvue qu'il soit situé dans le domaine de convergence du maillage.

Les contributions spécifiques de ce travail sont :

- Le développement d'une méthodologie d'optimisation pour minimiser les pertes d'énergie dans l'aspirateur.
- L'identification d'une paramétrisation adéquate des profils à l'entrée des aspirateurs et son extension pour les roues de type Kaplan.
- La réduction du temps total du processus d'optimisation par l'utilisation d'algorithmes distribués et la réduction du maillage associé au modèle numérique de l'aspirateur.
- L'étude qualitative et quantitative de l'écoulement qui permet d'identifier les causes d'amélioration de la performance de l'aspirateur.
- Le développement de programmes qui relient les différents logiciels et modifient les conditions aux frontières en se basant sur la réponse de la performance de l'aspirateur.
- L'établissement d'une méthodologie pour obtenir un profil cible qui pourra être utilisé par les concepteurs comme une première approximation pour obtenir une meilleure performance de l'aspirateur.

Finalement, sur la base de ce travail, nous pouvons émettre les recommandations suivantes pour la recherche à venir :

- Inclure toutes les conditions aux frontières d'entrée de l'écoulement comme les profils d'énergie cinétique et la dissipation turbulente.
- Réduire le temps de calcul en optimisant la génération du maillage minimum de l'aspirateur.
- Étudier l'interaction entre les paramètres du profil de vitesse et ceux du profil de l'aube de sortie de la roue avec les contraintes de design.
- Optimiser sur différents points d'opération de la turbine pour essayer d'éviter les chutes brusques de pression dans l'aspirateur, (Mauri et al., 2000, Arpe & Avellan, 2002).

TABLE OF CONTENTS

DEDICATION iv

ACKNOWLEDGMENTS v

RÉSUMÉ vi

ABSTRACT vii

CONDENSÉ EN FRANÇAIS viii

TABLE OF CONTENTS xix

LIST OF TABLES xxiv

LIST OF FIGURES xxvi

NOTATIONS AND SYMBOLS xxxiii

LIST OF APPENDICES xxxvi

INTRODUCTION 1

CHAPTER 1 STATE OF THE ART. 8

1.1 Hydropower 8

1.1.1 The Francis Turbine. 9

1.1.2 The Kaplan Turbine. 9

1.1.3 The Draft Tube. 10

1.1.4 Flow characteristics downstream of the runner. 14

1.2 The inlet swirling flow effects. 15

1.3 Numerical studies of the inlet swirling flow on draft tube performance. . . 20
1.4 Turbo-machinery optimization 23
1.5 Representation of the inlet velocity profile. 25
1.5.1 Angular momentum method. 26
1.5.2 Swirl intensity. 28
1.5.3 Solid-body swirl ratio. 28
1.5.4 Inlet radial velocity component. 28
1.5.5 Parametric inlet swirling flow representation. 30
1.6 Summary. 31

CHAPTER 2 OPTIMIZATION METHODOLOGY 32
2.1 Problem formulation. 33
2.2 First step: Inlet flow parameterization. 35
2.2.1 Approximating data. 35
2.2.2 Parametrization methods. 36
2.2.3 Parametrization goals. 38
2.3 Second step: Flow field simulation. 39
2.3.1 Computational Fluid Dynamics. 40
2.3.2 Governing equations. 40
2.3.3 Turbulence modeling. 41
2.3.4 Near wall treatment. 44
2.3.5 Flow solver. 45
2.3.5.1 Boundary conditions. 46
2.3.5.2 Numerical accuracy. 48
2.4 Third step: optimization method. 48
2.4.1 Optimization algorithms. 50
2.4.2 Multi-Island Genetic Algorithm. 54
2.4.3 Objective function 56

2.5 Summary. 58

CHAPTER 3 SHAPE PARAMETRIZATION METHODS FOR THE DRAFT TUBE INLET VELOCITY PROFILE. 59

3.1 Geometric representation of the meridional and tangential flow field. . . . 59

3.2 Non linear regression fitting. 66

3.2.1 Approximation of experimental profiles. 67

3.3 Analytical representation of the swirling flow at the runner draft tube interface. 69

3.3.1 Francis inlet velocity profiles approximation. 71

3.3.2 Kaplan inlet velocity profiles approximation. 72

3.4 Summary. 75

CHAPTER 4 VALIDATION OF THE T99 DRAFT TUBE MODEL 78

4.1 Computational aspects. 79

4.1.1 Geometry and grid. 79

4.1.2 Boundary conditions. 81

4.2 $k - \epsilon$ turbulence models. 82

4.2.1 Standard Model (STD). 82

4.2.2 Renormalization Group Model (RNG). 83

4.2.3 Realizable Model (REA). 85

4.3 Discretization schemes. 87

4.3.1 The four FLUENT discretizaton schemes. 87

4.3.2 Results of the discretization scheme study. 88

4.4 RNG swirl modifications. 96

4.4.1 Modifications to the model. 96

4.4.2 Results of the swirl modifications. 96

4.5 Near-wall treatments. 97

4.5.1 Near-wall modeling. 102
4.5.2 Results of the near wall treatment. 103
4.6 Convergence criteria. 111
4.7 Summary. 113

CHAPTER 5 OPTIMIZATION PROCESS. 116
5.1 Optimization set-up. 117
5.2 Parallel DGA. 119
5.3 Optimization scheme validation. 121
5.3.1 The Rosenbrock function 121
5.3.2 The Rastrigin function. 123
5.4 Definition of the objective function. 124
5.4.1 A parametric study of the objective function. 125
5.5 Summary. 127

CHAPTER 6 STEADY SWIRLING FLOW IN A DIFFUSER 130
6.1 The Numerical model 130
6.1.1 Numerical comparison. 132
6.1.2 Modeling of the inlet velocity profile. 133
6.2 Results of optimization. 135
6.2.1 Optimization history. 137
6.2.2 Best optimization points without flow constraints. 141
6.2.3 Best optimization points with flow constraints (FQ=0.5, FQ=1.0, FQ=1.5). 143
6.3 Vortex structure. 147
6.3.1 Velocity and pressure contours. 149
6.3.2 Engineering quantities. 151
6.4 Summary. 152

CHAPTER 7 DRAFT TUBE INLET OPTIMIZATION VELOCITY PROFILE T99. 157

7.1 Inlet velocity profile adaptation. 157

7.2 The Numerical model. 159

7.2.1 Coarse mesh modeling. 160

7.2.2 Wall treatment. 164

7.2.3 Comparison to experimental data. 165

7.2.4 Flow visualization. 166

7.3 Optimization process. 170

7.4 Draft tube flow field analysis. 171

7.5 Summary. 183

CONCLUSION 185

ANNEXES 202

LIST OF TABLES

Table 3.1 Equations used to approximate the inlet velocity components using the Polyfit program, Ruiz (2004). 68

Table 3.2 Parameters for the swirling flow approximation on draft tube inlet for the experimental inlet velocity values from La Grande 3 (LG3), Huites best efficient point (HUI)(bep) and Huites rated output point (rop). 74

Table 3.3 Parameters for the swirling flow approximation on draft tube inlet using three and four vortices. 76

Table 4.1 Global performance variation for the discretization schemes. . . . 89

Table 4.2 Global performance quantities for RNG model options. 97

Table 4.3 Global performance quantities for wall treatment functions. . . . 108

Table 5.1 Basic tuning MIGA parameters 117

Table 5.2 Advanced tuning MIGA parameters 118

Table 5.3 Fractional factorial design on Turbine 99 at two levels of each design parameter. 128

Table 5.4 Global performance quantities obtained for run 1 to 16. 128

Table 5.5 Main effects of the inlet swirling factors. 128

Table 6.1 Comparison results between FLUENT and CFX. 133

Table 6.2 Comparison of the results. 133

Table 6.3 Inlet swirl original parameters. 136

Table 6.4 Inlet swirl parameters for each vortex equation at the best optimization point. 142

Table 6.5 The global performance quantities obtained for the best optimization points. 142

Table 7.1 Inlet swirl original parameters. 158

Table 7.2 Iterative convergence parameters for the five different grids using a total draft tube mass flow of $521.06\ kg/s$. 161
Table 7.3 Grid error estimated in percent for the five grid size and the scheme order; first e_1 and second e_2. 161
Table 7.4 Extrapolated value ϕ_{ext} of the engineering quantities using both discretization schemes order; first $p = 1$ and second $p = 2$. . . . 162
Table 7.5 Mean y^+ value for all grids. 164
Table 7.6 Inlet swirl parameters obtained from the optimization process. . . 172
Table 7.7 Comparison of the global performance quantities obtained from the best inlet velocity profiles, between grid 5 and grid 1. 173

LIST OF FIGURES

Figure 1.1 The fundamental parts of a typical hydropower installation 9

Figure 1.2 Radial flow francis turbine with fixed blades, (Nilsson, 2002) . . 10

Figure 1.3 Kaplan axial-flow turbines with adjustable pitch runner blades, (Nilsson, 2002) 11

Figure 1.4 Draft tube expanding sections. a) side view, b) top view, c) sectional view and d) tri-dimensional view. 12

Figure 1.5 The distribution of losses depending on the flow rate in a Kaplan turbine, (Engström, 2003) 13

Figure 1.6 Distribution of losses in a Francis turbine depending on its specific speed, (Drtina & Sallaberger, 1999) 14

Figure 1.7 Geometry and flow conditions at the inlet to runner 27

Figure 1.8 Circumferential velocity distribution Muntean et al. (2005) 31

Figure 2.1 Inlet velocity profile optimization flow chart 35

Figure 2.2 Tree of turbulence modeling, (Memarzadeh, 1998) 43

Figure 2.3 Overview of segregated solution method. 46

Figure 2.4 Conceptual flow of the MIGA. 55

Figure 3.1 Network used to determinate the flow conditions at the blade runner trailing edge. 62

Figure 3.2 Velocity components at the interface runner-guide wheel. 63

Figure 3.3 Velocity components at the trailing edge blade runner. 64

Figure 3.4 Stream line sketch of the essential number of points point to create the draft tube inlet velocity profile. 65

Figure 3.5 Inlet velocity profiles for being approximated by mathematical equations established in table 3.1. 67

Figure 3.6 Root-mean-square error. 70

Figure 3.7 HUITES swirling flow on draft tube inlet at best efficient operating point (bep) up, and rated output operating point (rop) down. . 73
Figure 3.8 LG3 swirling flow on draft tube inlet at best efficient operating point. 73
Figure 3.9 Turbine 99 cone geometry with the inlet section survey at Cs1b. . 74
Figure 3.10 Velocity profiles approximations to experimental one at Cs1b section. 75
Figure 4.1 Geometry and measurement sections of the draft tube. 80
Figure 4.2 Pressure coefficients along the centerline of the upper and lower walls for the discretization schemes using the $y^+ = 1$ grid. 90
Figure 4.3 Pressure coefficients along the centerline of the upper and lower walls for the discretization schemes using the $y^+ = 50$ grid. . . . 91
Figure 4.4 Experimental variation of the axial velocity contours and horizontal velocity vectors at the Cs2, Andersson (2001). 92
Figure 4.5 Axial velocity contours and horizontal velocity vectors, caused by different discretization schemes, at the Cs2 section for the $y^+ = 1$ grid. 92
Figure 4.6 Axial velocity contours and horizontal velocity vectors, caused by different discretization schemes, at the Cs2 section for the $y^+ = 50$ grid. 93
Figure 4.7 Experimental variation of the horizontal velocity contours at the Cs3, Kurosawa et al. (2001). 93
Figure 4.8 Horizontal velocity contours, caused by different discretization schemes, at the Cs3 section for the $y^+ = 1$ grid. 94
Figure 4.9 Variation of the horizontal velocity contours, caused by different discretization schemes, at the Cs3 section for the $y^+ = 50$ grid. . 95

Figure 4.10 Pressure coefficients along the centerline of the upper and lower walls for the RNG model options using the $y^+ = 1$ grid. 98
Figure 4.11 Pressure coefficients along the centerline of the upper and lower walls for the RNG model options using the $y^+ = 50$ grid. 98
Figure 4.12 Axial velocity contours and horizontal velocity vectors, caused by RNG model options, at Cs2 section for $y^+ = 1$ grid. 99
Figure 4.13 Axial velocity contours and horizontal velocity vectors, caused by RNG model options, at Cs2 section for $y^+ = 50$ grid. 99
Figure 4.14 Horizontal velocity contours, caused by RNG model options, at Cs3 section for the $y^+ = 1$ grid. 100
Figure 4.15 Horizontal velocity contours, caused by RNG model options, at the Cs3 section for the $y^+ = 50$ grid. 101
Figure 4.16 Pressure coefficients along the centerline of the upper and lower walls for different wall treatment functions using the $y^+ = 1$ grid. 104
Figure 4.17 Pressure coefficients along the centerline of the upper and lower walls for different wall treatment functions using the $y^+ = 50$ grid. 105
Figure 4.18 Axial velocity contours and horizontal velocity vectors, caused by the wall treatment functions, at Cs2 section for the $y^+ = 1$ grid. 106
Figure 4.19 Axial velocity contours and horizontal velocity vectors, caused by the wall treatment functions, at Cs2 section for the $y^+ = 50$ grid. 107
Figure 4.20 Horizontal velocity contours, caused by the wall treatment functions, at the Cs3 section for the $y^+ = 1$ grid. 109
Figure 4.21 Horizontal velocity contours, caused by the wall treatment functions, at the Cs3 section for the $y^+ = 50$ grid. 110
Figure 4.22 Comparison of grid quality via Standard Deviation (SD) of the global performance quantities. 111

Figure 4.23 Residual reduction factor (RRF) of the second iteration and the last iteration for the different runs using the $y^+ = 1$ and $y^+ = 50$ grids. 113
Figure 4.24 Solver CPU time in hours used to compute 500 iterations for the different runs using the $y^+ = 1$ and $y^+ = 50$ grids. 114
Figure 5.1 Data communication between parallel MIGA and the calculating network 120
Figure 5.2 Full definition range of the Rosenbrock function 122
Figure 5.3 Convergence history for the Rosenbrock test function with $n = 10$ design variables. MR= Migration rate, P = Population size, MI= Migration interval, DF= default values. 122
Figure 5.4 Full definition range of the Rastrigin function 124
Figure 5.5 Fitness value history of the Rastrigin function at each generation with $n = 2$ and $n = 10$ variables. 125
Figure 6.1 The computational domain of the cone. 131
Figure 6.2 Comparison of the contours of different magnitudes on meridional sections. 132
Figure 6.3 Original velocity profile. 136
Figure 6.4 Optimization history of the energy loss factor (ζ) using the three vortex equations. 138
Figure 6.5 Optimization behavior of the energy loss factor (ζ) using the three vortex equations in terms of a) the inlet swirling flow and b) flow rate factor. 139
Figure 6.6 Optimization development of the wall pressure recovery factor for the three inlet velocity equations in terms of a) the inlet swirling flow S and b) the flow rate factor FQ. 140

Figure 6.7 Optimization progress of the mean pressure recovery factor for the three inlet velocity equations in terms of (a) the inlet swirling flow S and (b) flow rate factor FQ. 141
Figure 6.8 Comparison of the optimized inlet velocity profiles obtained from different vortex equations. 143
Figure 6.9 Best optimization points (BOP) obtained at different flow rates FQ for each vortex system. 144
Figure 6.10 Comparison of the inlet velocity profiles obtained without flow rate constraint (QF) and those obtained with flow rate constraint $FQ = 0.5$, $FQ = 1.0$ and $FQ = 1.5$. 145
Figure 6.11 Extrapolation points obtained from the best optimization points at different flow rates FQ. 146
Figure 6.12 Comparison among the best optimization points and its suitable extrapolation value obtained at different flow rate factor FQ. . . . 148
Figure 6.13 Contours of the axial velocity component on a meridional plane obtained for the different vortex systems. 153
Figure 6.14 Contours of the kinetic energy on a meridional plane obtained for the different vortex systems. 153
Figure 6.15 Absolute pressure behavior as a result of inlet velocity profile on inlet, middle and outlet cross sections. 154
Figure 6.16 Engineering quantities behavior along the diffuser. 155
Figure 7.1 Original inlet velocity profiles at best efficient point. 159
Figure 7.2 Plot of the global performance quantities using the Richardson extrapolation method ϕ, with a first scheme order $\phi(1)$ and a second order scheme $\phi(2)$. 163
Figure 7.3 Pressure at the lower and upper centerline for the draft tube with the finest grid (dt1) and with the coarser one (dt5). 166

Figure 7.4 Global performance quantities evaluated along the daft tube sections for the draft tube with the finest grid (dt1) and with the coarser one (dt5). 167
Figure 7.5 Engineering quantities evaluated along the daft tube sections for the draft tube with the finest grid (dt1) and with the coarser one (dt5). 168
Figure 7.6 The axial (contours) and horizontal (vectors) velocity components at Cs2 (down stream view). 169
Figure 7.7 Horizontal velocity contours at Cs3. 169
Figure 7.8 Optimization history of objective function during the optimization process with respect to the swirl intensity and the flow rate factor. 171
Figure 7.9 Variation of the energy loss factor during the optimization process using coarser and finest grid for mass flow $FQ = free$ and $FQ1 = 1$. 172
Figure 7.10 Inlet velocity profiles to show the flow behavior along the draft tube. 174
Figure 7.11 Axial velocity contours and secondary flow vectors at the draft tube sections provoked by the optimized inlet velocity profiles. . 175
Figure 7.12 Streamlines and vectors of the secondary flow on draft tube cross-sections from the inlet to the outlet. 177
Figure 7.13 Total pressure contours at the draft tube mid-plane provoked by the optimized inlet velocity profiles. 178
Figure 7.14 Streamlines and vectors at the draft tube symmetry mid-plane provoked by the optimized inlet velocity profiles. 180
Figure 7.15 Engineering quantities behavior along the draft tube. 181
Figure 7.16 Engineering quantities behavior along draft tube. 182

Figure III.1 Task diagram. 213
Figure III.2 Parameters display. 214

NOTATIONS AND SYMBOLS

Notation

σ	Thoma number
h_s	Suction head
n_s	Specific speed of the machine
1	Similarity inmediately in back of inlet into runner
2	Similarity inmediately in front of outlet from runner
3	Similarity inmediately after discharge from runner and at the begenning of draft tube
4	Similarity at the end of the draft tube
N	Kinematics vorticity number
$\overline{\rho u_i' u_j'}$	Reynols stress
μ_t	Turbulent (eddy) viscosity
κ	Turbulent kinetic energy
δ_{ij}	Kronecker delta
u_i	Velocities in x_i coordinate
p	Static pressure
ρ	Water density
τ_{ij}	Viscous stress tensor
ϑ	Turbulent characteristic velocity
ι	Turbulent characteristic lenght
ω	Turbulent frecuency
C_μ	Constant (= 0.09)
u'	fluctuation from the x-component
$\overline{u}$	Mean velocity

u_τ	Wall tangential velocity
τ_ω	Wall shear stress
u^+	Non-dimensional velocity
y^+	Non-dimensional distance from the wall
k	Von Kármán constant ($= 0.4187$)
B	Empirical constant ($= 5.5$)
Q	Flow rate
A	Cross section area
Cp_m	Mean pressure recovery factor
Cp_w	Wall pressure recovery factor
ζ	Energy loss factor
l_s	Scale longitud
$\widehat{\nu}$	turbulent kinematic viscosity
e_i	iterative convergence error
η_h	Hydraulic efficiency

Acronyms

bep	Best efficient point
CFD	Computational Fluid Dynamics
MDO	Multidisciplinary Design Optimization
RSM	Reynolds Stress Model
AMG	Algebraic Multigrid Method
GA	Genetic Algorithm
DGA	Distributed Genetic Algorithm
$PDGA$	Parallel Distributed Genetic Algorithm
$MIGA$	Multi-Island Genetic Algorithm
LES	Large Eddy simulation

RNG	Renormalized Group Model
$ERCOFTAC$	European Research Community On Flow, Turbulence And Combustion
REA	Realizable Group Model
FVM	Finite Volume Method
FOU	First Order Upwind-scheme
SOU	Second Order Upwind-scheme
POL	Power Law scheme
QUI	Quadratic Upstream Interpolation scheme
dvm	differential viscosity modification
sdf	swirl-dominated flow
nwt	near wall treatment
ewf	enhanced wall function
$newf$	Non-equilibrium wall function
pge	pressure gradient effects option
MSR	mass source residual
TMF	total mass flux
RRF	Residual Reduction Factor
$STDEV$	Standard Deviation
$GRMIAO$	Groupe de Recherche en Mathématiques de l'Ingénierie Assistée par Ordinateur
QF	Flow rate free
FQ	Flow rate factor
GE	General Electric
BOP	Best Optimization Point

LIST OF APPENDICES

APPENDIX I UNIX SHELL SCRIPT 202

I.1 Zeta 202

I.2 Echidna 203

APPENDIX II MATLAB PROGRAMS 206

II.1 Boundary conditions 1 206

II.2 Boundary conditions 2 208

II.3 Engineering quantities 210

APPENDIX III ISIGHT USER INTERFACE 213

III.1 Task diagram 213

III.2 Parameters display 214

INTRODUCTION

Flowing water creates energy that can be captured and turned into electricity. This is called hydroelectric power or hydropower. Of the renewable energy sources that generate electricity, hydropower is the most often used and is one of the least expensive. Hydropower plants are complex systems where competence in civil, mechanical and electrical engineering are combined. The aim is to convert potential energy contained in an elevated body of water into rotational mechanical energy to drive a generator. Hydropower plants are generally composed of a reservoir delimited partially by the nature environment and a dam, where water is stored. As a control gate is opened, the water flows from the reservoir through the penstock to the turbine, where it transmits its energy. The electrical power generated is a function of the total efficiency of the power plant, the water density, the acceleration due to gravity, the flow rate and the static head.

Despite hydropower being an economical and efficient method of generating electricity, the possibilities of further development depends much on the degree to which new projects can be accepted because of their effect on the environment, in consequence, the demands on rehabilitation and modernization are increasing. This improves the potential for carrying out changes in the design for improved efficiency and associate power output as well as greater operating stability. Therefore, renovation and rehabilitation projects of hydroelectric power station are frequent and represent an important part of the world market of hydraulic turbines.

Because of manufacturing costs, usually only the runner and the guide vanes are replaced but the spiral case and the draft tubes are rarely modified. The runner design is mostly based on the runner energy transformation, from hydraulic to mechanical, in particular: the head losses between runner inlet and outlet, the pressure distribution between both

sides of the vanes and the runner outlet velocity distribution. However, previous studies indicate that the power gained due to the vane runner design could be distorted by the flow delivered to the draft tube, compromising the overall flow stability in the machine. Thus, unfavorable behavior occurs if the runner and draft tube have not been suitably matched. In consequence, it is important to supervise the adequate coupling between runner and draft tube flow, otherwise, flow instabilities and reduction in optimal operation range of the machine will occur.

Problem description. It is common knowledge that the flow leaving the runner in hydraulic turbines losses its velocity in the draft tube, where kinetic energy of the flow is transformed into pressure. This energy conversion significantly impacts on the efficiency and power of the turbines and mostly those that present low head. In these machines, the energy losses that can be related to draft tube performance may reach up to 50% of the total turbine losses. Therefore, the draft tube flow is of great interest for both designer of new machines as well as for those who carry out the rehabilitation of existing ones. This has lead to many numerical studies of draft tube flow with diverse challenges to overcome. Namely the swirling flow at the runner outlet, the secondary flow caused by the draft tube disposition and the pressure gradients caused by its cross section changes.

In recent years, numerical and experimental investigations on the draft tube performance have confirmed the importance of the real inlet swirling flow created by the runner vanes. Masse (1999) corrected the draft tube inlet flow through the vane runner outlet profile modifications. This flow change resulted in an improvement of the draft tube efficiency since it eliminated a large eddy in the draft tube elbow.

Avellan (2000) has exposed that the machine performance according to the Thoma number σ often shows an increase with development of the cavity rope. This efficiency increase could be considered as a measure of the potential gain in the improvement of the matching of the runner with the draft tube.

Ruprecht et al. (2000b) performed an unsteady simulation of an entire Francis turbine for part load and optimum point of operation. It was observed that a flow distribution in the draft tube depends on the point of operation. This is explained by the swirl rate behind the runner bringing about an unsymmetrical flow behavior in one channel, which increases the discharge whereas the others channel shows a reduced flow rate.

Mauri et al. (2002b) carried out numerical simulations to study the sudden pressure recovery drop in the draft tube as the flow discharge is increased. The origin of static pressure recovery drop is localized in the bend zone and is the result of flow separation which blocks one channel leading to a flow acceleration in the other channel. He presumes that the separation is due to the change of the flow rotation direction at the inlet, imposed by the runner blades and the increase of secondary flow introduced by the bend.

Iliescu et al. (2002) performed experimental measurements of the turbulence and velocity fields at the outlet of the draft tube with a single pier with two channels. It was found that at high operational points, the swirling flow structure induced by the runner changes rotation direction, this results in a non-uniform distribution of the flow rate between the two draft tube channels. This is explained by a vortex development on a channel having as a result a back flow at the outlet. The other channel recovers the flow and the velocity increases, leading to a low pressure recovery along the draft tube.

These results indicate that it is still a challenge to get the optimal flow distribution at the draft tube inlet, which gives the best machine performance over a range of operating points. There is, consequently a need to couple the runner and the draft tube to minimize the losses arising from the inlet flow distribution.

Motivation and Objective. The flow characteristics at the inlet of the draft tube has a direct impact on its efficiency. The question is if it is possible to find an inlet velocity profile to minimize the losses for a given draft tube. Is it possible to find this inlet velocity

profile for the machine operating range? What are the inlet flow characteristics which provoke an unfavorable flow, reducing its performance? This research will explore the design space to yield a target velocity field which should be taken into consideration during the preliminary runner design. This velocity profile generated by the runner should aim to guarantee a better match with a given draft tube.

Scientific Motivation. The modeling of the flow in the draft tube is a big challenge, because of the swirling flow generated by the runner. This swirl gives complex inflow boundary conditions for the draft tube analysis. This feature introduces difficult flow inlet conditions which are hard to predict with the help of numerical simulations as a result of the streamlines changes and inverse pressure gradients caused by the curvature effects and the change of section through the diffuser. For this reason, a more detailed knowledge of the boundary conditions influence on the diffuser performance become fundamental. This will constitute a starting point for a better control on the output of this component of the turbine.

Industrial Motivation. Modern turbines have a high efficiency which can reach up to 95%. The draft tube is only capable of recovering a fixed proportion, around 85%, of the energy put into it. In consequence, gains in turbine efficiency could lie in the range of $0.5 - 2.0\%$, (Hothersall, 1988). In spite of this, one must recognize that an increase of some points could mean a substantial improvement of profits, thus, the gain in the draft tube performance as a result of the good coupling between runner and draft tube flow could increase electric production.

Objective. Before considering any process for the improvement of the inlet flow to the draft tube, one should identify the main parameters that define this flow and that could affect its performance. In this respect, this work will be dedicated to evaluate an approximation to the draft tube inlet flow profile, and subsequently, the influence of the profile parameters over the draft tube flow field will be quantified. Finally, the optimization of

the inlet profile minimizing the energy loss factor will be attempted. With this effort one could identify the parameters that serve to get an optimal inlet velocity distribution to increase the draft tube efficiency.

To sum up, the purpose of the study is the modeling, simulation and characterization of a complex steady flow in a draft tube created by different inlet velocity profiles. In this context, the numerical optimization of the velocity field at the runner's exit is proposed to improve the development of the draft tube flow and increase the machine output. The flow optimization process must be defined using a flexible set of inlet data parameters in order to build the best field velocity flow for each particular draft tube. Numerical simulation is an economical and effective possible way to fulfill this. However, as both the velocity inlet and the flow through the draft tube are modeled by equations and parameters, their selection and values will influence in a very important way the simulation results.

This thesis investigate the combination of tools based on Computational Fluid Dynamics (CFD) and optimization algorithms in the study of an industrial flow to improve the draft tube performance. This methodology has not been developed until recently due to the high CPU requirements of CFD simulations. The methodology thus requires to verify and validate both CFD simulations and computer optimization techniques to reach reliable results in reasonable time.

Thesis organization

This thesis starts by describing the operation of the draft tube in hydroelectric power plant and its importance in recovering energy. It takes into account a bibliographic review about the capacity of the numerical simulations to reproduce the flow in the draft tube caused by a variable inlet flow condition. This review presents the problems provoked by the swirling flow at the runner's outlet, how it has been modeled and the turbo-machinery optimization process which could be analogously applied to this project.

The second chapter presents the methodology to optimize the draft tube inlet velocity field. Here, each part of the process is described: the draft tube flow numerical modeling, the inlet velocity profile parametrization, and the optimization algorithm.

In the third chapter, a study on parametrization methods for inlet velocity profile shape is presented. The objective is to examine a different approach to experimental data set representation at the draft tube inlet.

The forth chapter is devoted to the draft tube flow modeling. The flow analysis of the steady computations and the comparisons with measurements are carried out. The three elements involved in CFD simulations recognized as critical have been studied: the grid, the numerical discretization scheme and the turbulence model. The flow is analyzed to assess the influence of all these parameters on the optimization process.

To efficiently apply optimization techniques, it is necessary to gain an adequate understanding of the theory as it applies to the improvement of very expensive cost functions. Thus, in the fifth chapter, optimization technique implementations used are described and tested by resolving some standard optimization problems.

The inlet profile velocity optimization process applied to a conical diffuser is presented in the sixth chapter. This optimization is a validation of the inlet velocity parametrization, numerical model and the optimization algorithm process.

In the seventh chapter, the validated optimization process is applied to the Turbine 99 draft tube. The numerical results are obtained by the optimization algorithm in the complete design space. Then, a study of the swirling velocity parameters influence on the draft tube flow performance is developed. To analyze the flow behavior, a study of the different global variables along centerline is carried out. This study is aimed at understanding the energy losses caused by the different inlet flow patterns obtained during the optimization

process.

Finally, a short summary, the contributions of this work and some perspectives are discussed. In the annexes are presented the different codes developed to obtain the coupling among MATLAB, FLUENT and iSIGHT softwares to achieve the optimization process.

Along these lines, it has been showed that an optimal flow distribution at the runner outlet which leads to the best draft tube performance is still a challenge. Consequently, we propose as a principal subject of this thesis, a new attempt to improve the draft tube performance through the control of the inlet flow conditions by an optimization algorithm.

CHAPTER 1

STATE OF THE ART.

This chapter introduces the description and importance of the draft tube in the energy conversion process of an hydraulic turbine. It is composed essentially by a diffuser combined with an elbow and it may cause up to 50% of the total losses of the machine. Besides, the flow stability and cavitation are also affected by this component which has an impact on the entire turbine performance. Many hydro-power turbines will also be refurbished in the future and the hydropower industry needs to understand and enhance the performance of the draft tubes. In this part, different numerical and experimental studies about the swirling flow as principal factor of draft tube losses are presented. Finally, it exposes an historical review of the numerical flow simulation as a tool for modeling the inlet velocity flow and the methods proposed to try to parameterize it.

1.1 Hydropower

Hydropower plants, as illustrated in Fig. 1.1, are composed essentially of a reservoir where the water is stored and a runner coupled to a generator by means of a shaft, which converts the potential energy of water into work, producing electricity. It exists two groups of machines, according to the way to extract the hydraulic energy; on the one hand the impulse machines that take advantage of the kinetic energy and on the other hand the reaction machines, that use mainly the pressure energy. In this last group the Kaplan and Francis turbines present a draft tube added on the runner exit.

Figure 1.1 The fundamental parts of a typical hydropower installation

1.1.1 The Francis Turbine.

The majority of Francis Turbines are arranged so that the axes is vertical, but some small machines can have horizontal axes. Figure 1.2 illustrates a section through a vertical shaft Francis turbine. Water enters via a spiral case called a volute or scroll which surrounds the runner. The area of cross-section of the volute decreases along the flow path in such a way that the flow velocity remains approximately constant. From the volute, the flow enters a ring of stationary guide vanes which direct it onto the runner. In flowing through the runner, the angular momentum of the water is reduced and work is supplied to the turbine shaft. At the design condition, the flow leaves the runner axially into the draft tube and finally the flow enters the tailrace.

1.1.2 The Kaplan Turbine.

This type of turbine is based on the need to generate power from much lower pressure heads than are normally employed with a Francis turbine. To satisfy large power demands,

Figure 1.2 Radial flow francis turbine with fixed blades, (Nilsson, 2002)

very large volume flow rates need to be accommodated. Figure 1.3 is a part sectional view of a Kaplan turbine in which the flow enters from a volute into inlet guide vanes which impart a degree of swirl to the flow determined by the needs of the runner. The flow leaving the guide vanes are forced by the shape of the passage into an axial direction to arrive to the vanes of the runner. The Kaplan turbine incorporates one essential feature not found in others turbine rotors and that is the setting angle of the rotor blade can be controlled. The advantage of using adjustable runner blades are that the efficiency is higher over a wider operating range, which allows the available water power to change without losing a great deal in efficiency.

1.1.3 The Draft Tube.

The energy conversion in the draft tube significantly impacts on the efficiency and power of the turbines and mostly those that present low head (high specific velocity) and which operate away from their best efficiency point (bep). The diffuser, as a device to im-

Figure 1.3 Kaplan axial-flow turbines with adjustable pitch runner blades, (Nilsson, 2002)

prove the efficiency of hydraulic turbines, was patented by Jonval in 1843, with which he reached a turbine efficiency of 75% while maintaining a more stable flow than the one presented by B. Fourneyron in 1827, (Larsson, 2005). The draft tube is an essential feature of the reaction turbines. It has two principal functions: first, after the runner, there is still some kinetic energy left in the water flow which can be partly recovered. Second, it uses the head provided by the elevation difference between the turbine centerline and the tailwater, h_s in Fig. 1.1.

In those turbines, a conical diffuser follows directly after the runner. This first diffuser is followed by an elbow, which typically has a constant or decreasing cross-sectional area. The elbow redirects the vertical flow after the runner to a horizontal flow. After the elbow follows a second diffuser, either conical or rectangular, that leads to the tailrace and into the natural waterways. Figure 1.4 shows a typical elbow draft tube used in a vertical axis turbine.

Figure 1.4 Draft tube expanding sections. a) side view, b) top view, c) sectional view and d) tri-dimensional view.

At the transition from the draft tube to the tailrace, a large part of the kinetic energy present after the runner should have been converted into additional head. This can be seen applying the Bernoulli's theorem to the water entering and leaving the draft tube, subscripts 3 and 4 respectively, and using level y of the draft tube exit as datum, Fig. 1.1:

$$\frac{p_3}{\rho} + \frac{V_3^2}{2g} + h_s + y = \frac{p_4}{\rho} + \frac{V_4^2}{2g} + \Delta h \tag{1.1}$$

Where Δh represents the sum of the losses along the draft tube, which are due to friction, shocks, and whirling and to the conversion of kinetic energy into pressure. Thus the energy gained by the water passing through the draft tube is :

$$\Delta E_{3-4} = \frac{p_3 - p_4}{\rho} + \frac{V_3^2 - V_4^2}{2g} + h_s - \Delta h \tag{1.2}$$

The value of the second fraction is the theoretical regain due to flaring of the draft tube. Consequently the potential difference h_s, the static draft head, is also used by the turbine

but at the same time the theoretical gains will be reduced by the losses Δh.

Figure 1.5 presents a typical losses distribution in a model Kaplan turbine. The losses magnitude and distribution in this type of turbine depend mostly on operating conditions. A major part is due to hydraulic losses throughout the waterways and viscous friction on solid surfaces, kinetic losses due to the production of the turbulence and exit losses. The higher the flow rate, the more important the draft tube losses are.

Figure 1.5 The distribution of losses depending on the flow rate in a Kaplan turbine, (Engström, 2003)

Figure 1.6 illustrates the hydraulic losses in a Francis turbine machine as a function of the specific speed. This is one of the main parameters in hydraulic turbines and is used to class turbine head (H), flow rate (Q) and speed (n). This dimensionless parameter is defined by:

$$n_s = \frac{\omega}{\pi^{\frac{1}{2}}} \frac{Q^{\frac{1}{2}}}{(2\,g\,H)^{\frac{3}{4}}} \tag{1.3}$$

Small hydropower plants generally have tight cost demands and only a small amount of available head to make power. This low head results in a high specific speed runner design. With increasing specific speed, the inlet velocity to the draft tube is also increased.

This means that more non-converted energy steadily enters the draft tube, where it must be used. Therefore, the function of the draft tube in such machines must be as perfect as possible to obtain a good overall turbine efficiency.

Figure 1.6 Distribution of losses in a Francis turbine depending on its specific speed, (Drtina & Sallaberger, 1999)

1.1.4 Flow characteristics downstream of the runner.

The amount of power produced by a turbine is equal to the change in angular momentum across the runner. Leaving the runner, this quantity is theoretically zero at the maximum efficient point. That is, the flow exiting the runner enters the draft tube without any swirl. However, practical loss in efficiency in the runner caused by a small amount of swirl is compensated by increasing the efficiency in the draft tube. This improved performance is primarily due to the energizing effect of the swirl on the boundary layer, which tends to delay separation in the cone. At partial load, the flow entering the diffuser co-rotates with the runner. At full load, the flow rotates opposite to the direction of rotation of the runner.

Dynamic characteristics of the flow downstream of the runner have a complicated structure. The complexity is, on the one hand, due to non-uniformity of velocity and pressure

fields distribution and, on the other hand, to instability of velocity and pressure values in the domain. If the swirl entering the draft tube is too strong and especially with a non-uniform axial velocity, a stagnation bubble of recirculation flow can occur at the centerline, developing a visible cavitating vortex which pulses. The pulsation causes synchronous pressure fluctuations. When this frequency coincides with the natural frequency of the system, the instability produced is known as draft tube surge which severely limits the unit performance. Draft tube surging can occur during both partial load and full load operation and it has been recognized as a problem in the operation of hydropower plants for several decades because these pressure fluctuations lead to power swings, noise, penstock resonance, and vibrations (O'kelly, 1989).

For years, the vortex flow phenomenon has interested researches all over the world and has been subject of numerous publications. Ancient developments of experimental methods (Zolotov et al., 1970, Mollenkopf & Raabe, 1970) and recent numerical simulations techniques (Ruprecht et al., 2002, Vu et al., 2004) show detailed analysis of this flow. Also, some mathematical models have been developed trying to approach the vortex rope (Fanelli, 1998, Goulet, 1997). However, any attempt for modeling the hydrodynamic phenomena leading to the development of the rope and its interaction with all the turbine and hydraulic system components needs further investigations and it is beyond the scope of the present research.

1.2 The inlet swirling flow effects.

The result of the inlet swirling flow in hydraulic turbine draft tube is a very complex phenomenon, which has been extensively investigated both theoretically and experimentally. These investigations have been mainly focused on the effects of the inlet velocity flow on the inception of draft tube surging phenomenon and on its performance.

Falvey & Cassidy (1970) presented the angular momentum method for predicting draft tube surging. This method relates turbine dimensions or quantities affecting the value of the swirl, and the amount of draft tube inlet swirl, which is the difference between the swirl introduced by the wicket gates and the swirl extracted by the runner. For a given draft tube, they found that the surge characteristic of frequency and amplitude are a function of the angular momentum parameter, Eq. 1.8. A significant finding was that surging begins for an angular momentum intensity of about 0.4. A similar study was reported by Padle (1974) using this method to identify surge-free and surging areas on model and prototype turbine performance diagrams. Wahl (1994) used this method to identify a twin vortex surging mode on a model and prototype Francis turbine.

In conical diffusers, an important experimental investigation was conducted by McDonald et al. (1971) to determine the effect of swirling inlet flow on the performance and outlet flow profile. He tested 24 diffusers with different characteristics, diverging angles and area ratios. The swirl generator produced a uniform axial velocity and the tangential profile was a linear function of the radius. His experiments showed that the increase in performance with swirl is very important in flows where the separation from walls is considerable. The optimum swirl angle depends on the particular diffuser and it exists a range of swirl angles where the performance is maximized rather than an asymptotic behavior. He assumed that the swirl is used as a boundary control device to improve the performance of the diffuser but an excess of it could provoke a surge phenomenon. Thus, this results suggest that each diffuser must be analyzed considering its design characteristic and inlet axial velocity distribution in order to maximize the pressure recovery in terms of the inlet swirl flow.

Five conical diffusers with different divergence angles to investigate the influence of swirling flow on the pressure recovery coefficients were tested by Senoo et al. (1978). The inlet radial distribution of circular component of velocity was represented by the

Rankine vortex for a momentum ratio (MR), given by Eq. 1.11, of 0.07, 0.12 and 0.18. The stronger the swirl, the lower the radial distribution of axial velocity near the center is, and a reverse flow begins at the center at 0.18. It is demonstrated that the swirl intensity and the cone divergence angle are factors which determine the axi-symmetric flow characteristic at the inlet section. This research presented an important flow analysis at a downstream sections. It was found the the radius of the solid vortex core increases quickly in the axial direction and the stronger the swirl, the larger the core will be. This reduces the effective cross-sectional area and a low pressure recovery coefficient results. This study was performed based mainly on the swirl intensity and it was not given detailed account of parameter of the momentum ratio number.

With relatively uniform inlet flows, the free stream turbulence level is not an extremely important factor. But a non uniform velocity combined with a swirling flow, can provoke important variations of the turbulence levels. This effect has been difficult to quantify because it is usually associated with other non-ideal conditions (upstream bends and obstructions and radial variations in energy exchange with a turbomachine rotor). It has not been made clear what characteristics are needed to describe the turbulence, but at least the intensity and scale should be determined. At this time, the information about the interaction of these inlet velocity conditions is limited in many diffuser experiments.

Shárán (1976) studied the influence of high intensity turbulence at the diffuser inlet on its performance. He presented an interesting way for evaluating the static pressure recovery in a conical diffuser using the following equation:

$$C_p = \alpha_3 \left[1 - \frac{\frac{\alpha_3}{\alpha_4}}{AR^2} \right] - \frac{\overline{P_3} - \overline{P_4}}{\overline{q_3}} \tag{1.4}$$

$$\overline{P} = \frac{\int PVdA}{\int VdA} \tag{1.5}$$

Where the index 3 describes the inlet while the index 4 indicates the outlet. $\overline{q}$ denotes the dynamic pressure, AR is the area ratio of the diffuser, V the local time-averaged velocity, $\overline{P}$ the mean total pressure and α the kinetic energy flow parameter, defined as:

$$\alpha = \frac{1}{A}\int^{A}\left(\frac{V}{\overline{V}}\right)^{3} dA \tag{1.6}$$

Equation 1.4 shows that the static pressure recovery depends upon two factors; the inlet and exit velocity profile shapes α_3/α_4 and the total pressure loss across the diffuser. Thus, an increase of the static pressure may be obtained either by increasing the amount of diffusion (i.e decreasing any non-uniformity in the profile) or by making the diffusion more efficient (i.e. reducing overall losses across diffuser) or both. In this way, it is likely to result in an increased pressure recovery by increasing the inlet turbulence level to induce better flow uniformity across the diffuser, thereby increasing diffusion but also increasing $\frac{\Delta \overline{P}}{\overline{q}}$. In consequence, the inlet turbulent intensity was varied from about 1 to 8% by inserting turbulence generators at the entrance of the pipe, resulting in a significant improvement of Cp, mainly at higher Reynolds number.

Monacelli & Cooper (1987) evaluated the draft tube performance using several conical and elbow-type draft tubes with varying degrees of upstream swirl. This swirl was generated by variable geometry vanes at angles of $0° - 30°$ where θ is the angle between the velocity vector and the area normal to the longitudinal axis. This swirl generates non-uniformity in the inlet velocity profile. The addition of inlet swirl caused an important improvement in the draft tube efficiency, from 87% without swirl, up to a maximal efficiency of 95% with 15° swirl angle. In this case, the radial velocity was not taken into

account. An interesting fact would have been to know the non-uniformity value at the draft tube outlet and establish its flow field correction effect on the performance. This research confirmed that the addition of inlet swirl caused a tremendous improvement in draft tube efficiency.

Kubota et al. (1996) extracted the draft tube hydraulic energy loss from the performance diagrams of the a Francis turbine. They established that the tube losses reach a maximum at the swirl-free runner outlet discharge. Thus the swirl energy at draft tube inlet increases when the discharge leaves the swirl-free discharge and suppresses the flow separation at the bend, resulting in a decrease loss, irrespective of the direction of swirl. There were two optimum swirl discharges: one at a smaller and the other a larger than the swirl-free discharge where the separation loss were suppressed drastically. From these results, they concluded that in a well designed bend draft tube, where the separation loss in the bend is not so large, only a single peak turbine efficiency (swirl-free at runner outlet) must appear on the lower side of the optimum swirl discharge.

Sharma et al. (1998) related the phenomenon of surging mainly to the angular momentum parameter as a representative of swirl. They used the angular momentum method as function of the flow characteristics at the runner outlet, in contrast to Falvey & Cassidy (1970) and Padle (1974) who used the inlet runner parameters. They related the angular momentum parameter, Eq. 1.10, given by the velocity components, to unit speed outlining the inlet velocity profiles for various unit speeds. This representation supplies inlet velocity profiles with a radial variation but no comparison to experimental profiles is mentioned. This test showed that the critical angular momentum parameter depends not only of its magnitude but also on the details on the way the swirl is distributed along the radius. It is worth mentioning that the variation of the unit speed was made using a generator for different swirl distribution and intensities, but in an actual turbine the unit speed is constant through all operational points.

1.3 Numerical studies of the inlet swirling flow on draft tube performance.

An additional approach to flow field and performance determination of draft tubes has been the development of computational fluid dynamics. Though started much earlier, it is only in the last decade or so that rapid developments in computational hardware have given great impetus to the development of the computational approaches through codes capable of handing complex flow geometries. The principal reason is that there are simply too many variables involved for a direct experimental approach to optimization.

The influence of the inlet boundary conditions on the draft tube flow performance, has recently been treated at length both numerically and experimentally. These studies provided valuable information on the validations of the simulations to improve the numerical modeling accuracy. The principal concern has been the influence of the inlet swirl flow on the draft tube pressure recovery factor.

Shyy & Braaten (1986) studied numerically the effects of the inlet swirl ratio on the flow field and the pressure recovery factor. The addition of inlet swirl, which is given by Eq. 1.12, $S = 0 - 1$, shows a strong influence on the overall flow structure because it eliminated the stalling characteristics in the downstream region and modified the behavior of the flow in the elbow region. The best recovery factor was established to $S = 0.75$.

In Vu & Shyy (1988) and Vu & Shyy (1990) pressure recovery factor and flow behavior in the draft tube with varying different inlet swirling flow intensity were predicted. The inlet flow profile was obtained from various Francis runners at different operating conditions. Solid body swirl to a free vortex were applied to predict pressure recovery factor and were found to be optimal from 25% to 50% of the swirl intensity.

Vu (1989) presented the influence of different geometrical draft tube parameters such as curvature radius of the bend, cross-sectional area-variation and inlet cone angle ratio

variation on the draft tube flow behavior and energy loss. The most important subject related to this thesis was the inlet flow profile optimization. He found that to improve the pressure recovery factor required an inlet solid body swirl with moderate intensity. He tried to find the optimal inlet swirl profile for the existing draft tube geometry. The parameters to define the solid body swirl profile at the inlet were: the solid body swirl ratio, the radial component ratio and the axial profile uniformity. It was assumed that the three components vary linearly in the radial direction. He found the efficiency to be highly sensitive to all three parameters. This study concluded that the inlet flow profile needed to be optimized to achieve the best static pressure recovery factor when the existing draft tube can not be modified.

Agouzoul et al. (1990) studied, in a way similar to Shyy & Braaten (1986) and Vu & Shyy (1988), the flow configuration related to different inlet swirls. The inlet velocity condition was modeled by a plug flow combined with a solid body rotation. The swirl intensities were established between $0.0 - 1.0$. The best pressure recovery factor was found for $S = 0.5$.

Song et al. (1991) tested numerically the effect of swirl from the turbine runner on the draft tube flow. The inlet swirl was defined as maximum tangential velocity over the averaged bulk velocity. His results clearly showed the effect of the inlet flow conditions reducing the flow separation generated at the elbow. However, the pressure recovery factor was not compared to any experimental result to observe the possible improvements displayed on the velocity field.

The influence of inlet swirl over the surging and over the pressure recovery in the draft tube and conical diffusers had been studied in detail until now. However, there is no information in terms of flow parameters, like swirl distribution, on the occurrence of vortex breakdown or the pressure recovery in the water turbine system, due to the complex nature of representing the swirl leaving the runner. Some research has tried to take into

account this inlet flow characteristic.

Longatte et al. (1999) analyzed numerically the flow in the draft tube according to the evolution of different global variables as a function of the curvilinear abscissa. They also introduced the kinematic vorticity number W_k given by the Eq. 1.7 to measure the quantity of rotation and to isolate the swirling structure in the flow. They made a quantitative evaluation of the flow using the global performance variables. Qualitatively, it was noticed that the flow structures for the highest flow rate is different from that at lower flow rates which have two contra rotating large vortices in the elbow.

$$W_k = \left[\frac{\|\Omega\|}{\|\mathbf{S}\|}\right]^{\frac{1}{2}} \tag{1.7}$$

Where $\mathbf{S}$ is the strain rate and Ω is the angular velocity.

Engström & Gebart (1999) studied the possibility of using a simple model based on the flow gyroscopic effect to calculate a new geometry (centerline curvature) of the draft tube that avoid distortion of the vortex core. It was known that there were strong gradients in the bend generated by the displacement of the core of the swirling flow towards one of the sides of the draft tube. They assumed that the losses would be reduced significantly if these gradients could be reduced by a modified geometry. Their results show that regardless of the faster disappearance of the vortex in the new draft tube, the losses are not significantly reduced. It must be mentioned that this study was carried on using two swirling numbers, however the axial velocity was set to be uniform and the tangential velocity was set to be linearly increasing from the center of the draft tube.

Mauri et al. (2002a) investigated the influence of different parameters on the draft tube performance as independent variables for simplicity even thought the result depend on the interaction between them. The near wall velocity profile and the inlet velocity radial

component influence on the pressure recovery are our interest. He carried out the numerical study for three operational points. The inlet radial component was represented as a geometrical distribution and a best linear fit to the experimental points. It was observed that this component can affect considerably the flow in the draft tube in spite of its small magnitude. With regard to the near wall velocity profile, it was considered as a factor of the nearby interior measurement point value, which was 0.076 inlet diameter. He found an almost linear variation of the pressure recovery factor when varying this parameter. He observed also that the angles of the skin friction on the cone are affected especially at the highest flow rate.

Susan-Resiga et al. (2006) presented a swirling flow structure downstream of a Francis runner. Besides, they proposed an outlet runner swirl criteria to avoid an unexpected sudden efficiency drop at certain discharge. The eigenvalue analysis of the swirling flow revealed that for discharge parameter φ larger than 0.365, the flow ingested by the draft tube is called supercritical, while at lower discharge it is subcritical. The critical state (vortex break down) occurs quite close to the discharge value where a sudden variation in the draft tube pressure recovery, as well as in the overall turbine efficiency, is experimentally observed. For this particular turbine tested this discharge value happens to correspond to the best efficiency point, leading to a negative impact on the turbine regulation.

We reviewed the effect of the inlet velocity profile on the draft tube performance and the strategies used to represent this velocity profile. We have still to describe the optimization strategies used with CFD to study the performance of the draft tube.

1.4 Turbo-machinery optimization

Modern computational and experimental fluid dynamics tools have arrived to a stage where they can provide an important insight into engineering process involving fluid

flows. This can help analyze the fluid physics as well as improve the design of practical devices. Objective and efficient evaluation of the designs can be facilitated by the development and implementation of systematic optimizations methods.

Recently new tools based on CFD and optimization algorithms are being introduced to evaluate the performance of industrial flows. Part of this research has been focused on shape optimization of turbo-machinery components like compressor blades profiles, (Fan, 1998, Zangench et al., 1999, Léonard et al., 1999, Burman et al., 2000, Oyama et al., 2002, Mengistu & Ghaly, 2003).

With respect to the design of runner blades profiles (Lipej & Poloni, 2000) used a genetic algorithm in connection with a numerical flow analysis to develop a multiobjective optimization. (Swiderski et al., 2001) used a Results Oriented Solution (ROS) to produce runner blade designs to create desired flow patterns. (Tomas et al., 2002) coupled a design, a CFD and an optimization software with an global algorithm to compare the "optimized design" to a "human defined design". (Peng et al., 2002) combined a multi-variable and multi-objective constrained optimization method with an improved three dimensional inverse computational model of performance for Kaplan runner. (Enomoto et al., 2004) used an optimization system coupling CFD and design tool of Francis turbine runner shape to a Multi-Objective Genetic Algorithm on the condition of 100% and 60% flow rate obtaining a great improvement in turbine efficiency.

For the draft tube sections (Ruprecht et al., 1999, Madsen et al., 2000, Eisinger & Ruprecht, 2001, Ruprecht et al., 2000a, Lindgren, 2002, Puente et al., 2003, Marjavaara & Lundström, 2004, 2005) have used the flow information obtained from CFD prediction and detailed measurements to systematically improve in the geometry design. None of these works have tried to improve the turbine performance by modifying the inlet velocity profiles

To date, the majority of the effort in design optimization of the inlet velocity profile has been applied to casting processes (Hofmann et al., 1997, Berg et al., 1995) and chemical processes, (Cho et al., 1999a,b, Grau et al., 2001). This research presents how the design inlet velocity profile is manipulated by a numerical optimization algorithm until a process with a better performance is achieved.

The state-of-the art presented here about the draft tube performance and the numerical optimization, allows us to assume that if we can parameterize the draft tube inlet velocity profile, we will be able to modify the distribution of the radial inlet velocity to minimize the energy losses using a numerical algorithm as an inlet velocity profile generator.

1.5 Representation of the inlet velocity profile.

It has been postulated that the draft tube surging, "vortex break down" in fluid mechanics, arises as a result of a rotational velocity component remaining in the water as it leaves the turbine runner and enters the draft tube. The drop in efficiency in the draft tube has been attributed to hydraulic losses caused mainly by flow separation. The analysis of the vortex breakdown phenomenon has been studied in connection with the swirling flows in pipes and aircraft wings, but its analysis in the draft tube is difficult compared to simple pipes and diffusers because of several factors are not take in account: water vaporization, varying geometry or even penstock dynamic characteristics. Added to this, in draft tube performance studies, the method used to generate the inlet velocity flow has not used a real turbine runner as generator of inlet profile. In both cases, the representation of the inlet swirling flow has resulted in the neglect of the influence of some parameters which give a real axial and tangential velocity distribution.

At present, there is no information available in terms of the inlet velocity components parameters, on the occurrence of vortex breakdown and performance in the water turbine

draft tube system due to the complex nature of the flow leaving the runner. Nevertheless, specific definitions of swirling structures have been proposed to represent the draft tube inlet flow. Thus, the present section focuses on the review of the structure of the swirl produced by a turbine runner blade and further ingested by the draft tube.

1.5.1 Angular momentum method.

This method was developed in the late 60's at the US Bureau of Reclamations. It is based on the fact that the flow phenomena in the draft tube can be produced without a full turbine model, provided the velocity field in the draft tube intake is reproduced with sufficient similarity. The method assumes that such similarity is given when a dimensionless ratio between the angular momentum and the axial momentum entering the draft tube is reproduced. The turbine draft tube surge results from excessive swirl in the flow entering the draft tube. A quantitative measure of the swirl is obtained by computing the dimensionless "momentum parameter" of the swirl.

$$\Omega = \frac{\rho\, Q^2\, R\, sin(\alpha)}{B\, N\, S} \tag{1.8}$$

Where Ω is the angular momentum, B is the height of the wicket gate, N is the number of wicket gates, R is the radius of the centerline of the gate opening, S is the minimum width of flow passage through the wicket gates, α is the angle between the flow vector and the radius, ρ is the fluid density, Q is the discharge given by:

$$Q = \frac{V}{BNS} \tag{1.9}$$

Fig. 1.7 illustrates these basic dimensions.

The surging phenomenon was studied by Falvey & Cassidy (1970), Falvey (1993), Padle (1974), Doerfler (1985), using this theory. Sharma et al. (1998) used this method, but changed the inlet parameters as:

$$\Omega = \int_0^{R_3} 2\pi V_a \, V_t \, r^2 \, dr \tag{1.10}$$

Where r is the local radius, R_3 is the radius of the draft tube inlet, 0 represents the hub position and V_a and V_t are the velocity components.

The immediate advantage of this method is to produce results without having to construct a complete homologous model. Both of these both analysis have shown that the swirl at the runner exit is fully responsible of the draft tube surging, and the vortex core results from it.

Figure 1.7 Geometry and flow conditions at the inlet to runner

1.5.2 Swirl intensity.

This parameter is defined as the ratio of the angular momentum flux to the axial momentum flux and, for this reason, is also called momentum ratio. It is given by:

$$MR = \frac{\int V_t \, V_a \, r^2 \, dr}{R_3 \int V_a^2 \, r \, dr} \tag{1.11}$$

The pressure surge in the draft tube is closely related to the magnitude of the momentum ratio, as was experimentally studied by Kubota & Aoki (1980). Other researchers have used this parameter to study the draft tube flow via CFD. Engström & Gebart (1999) evaluated the gyroscopic effects due to swirling flows and found that the centrifugal effects appear to be more important in the energy loss factor.

1.5.3 Solid-body swirl ratio.

This parameter has been used for characterizing the swirling inlet flow in numerical studies by Shyy & Braaten (1986), Agouzoul et al. (1990), Song et al. (1991). It is defined as the ratio of the maximum tangential velocity V_t to the bulk axial velocity V_a at the inlet.

$$S = \frac{V_t}{V_a} \tag{1.12}$$

1.5.4 Inlet radial velocity component.

Mauri et al. (2002a) represents the radial velocity component like a geometrical distribution by:

$$V_r = V_a \, tan(\theta \, \frac{r}{R}) \tag{1.13}$$

θ being the cone half opening angle. The inclination of the velocity vector in the radial direction is therefore determined by the geometry of the cone.

Several alternatives were used by the participants in the Turbine 99 Workshop II to handle the problem of unknown inlet radial velocity. Page & Giroux (2001) used the geometrical distribution to demonstrate computationally the high influence of this velocity component on pressure distribution and thus pressure recovery. Cervantes & Gustavsson (2005) used a two dimensional non-viscous description of the swirling flow given by the Squire-Long (Eq. 1.14) for the stream function ψ:

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial \sigma^2} - \frac{1}{\sigma}\frac{\partial \psi}{\partial \sigma} = \sigma^2 \frac{dH(\psi)}{d\psi} - C(\psi)\frac{dC(\psi)}{d\psi} \tag{1.14}$$

Where (x, σ, ϕ) are the cylindrical polar coordinates. The Squire-Long equation has the feature of depending on boundary conditions, since the Bernoulli function H and the circulation C are constants on streamlines. To obtain the radial velocity profile, the method takes advantage of this dependency by computing the equation related to the domain with an iterative process. After some iterations, the derived profiles differ considerably form the first guess which consists of a linear profile. The computed solution is very different from those obtained experimentally. The difference is due to initial guess on the axial velocity at the outlet of the domain. The guess was necessary due to the lack of experimental data. Thus, the detailed mechanism allowing the convergence to correct radial velocity is far from clear. Further analysis seems necessary to obtain a detailed understanding of the process.

1.5.5 Parametric inlet swirling flow representation.

Although the flow structure at the runner inlet draft tube interface appears to be quite complex, Muntean et al. (2005) managed to develop analytical representation for both tangential and axial velocity profiles. This representation was based on the Rankine vortex which presents a simple model for rotating flow. The circumferential velocity V_t is defined as a solid body rotation core followed by a r^{-1} which decays in the radial direction. It is worth to note that the model does not take in account the finite thickness of the shear layer region at $r = R_c$. The characteristic vortex radius R_c measures the vortex core radial extent. These two parameters define the Rankine vortex circumferential velocity.

$$V_t = \begin{cases} \Omega r & 0 \leq r \leq R_c \text{ forced vortex} \\ \frac{\Omega R_c}{r} & r \geq R_c \quad \text{free vortex} \end{cases}$$

Where r is the radial distance from the vortex axis. This simplified model provides a continuous function for $V_t(r)$, but the derivative is discontinuous. A more suitable model for developed swirling flows is the Batchelor's vortex. The circumferential velocity field is described via similarity solution applied to wakes in the far field.

$$V_t = \frac{\Omega R_c^2}{r}\left[1 - exp\left(-\frac{r^2}{R_c^2}\right)\right] \tag{1.15}$$

Equation 1.15 is an exact solution for a viscous vortex produced by radial inflow and axial outflow. It is well known that the Rankine vortex is built using the asymptotic behavior of the Burgers' vortex for large and small radius with respect to the vortex characteristic radius R_c. This idea is shown in Fig. 1.8.

Figure 1.8 Circumferential velocity distribution Muntean et al. (2005)

1.6 Summary.

This chapter has described the draft tube importance in the introduction of energy using a hydraulic turbine. As the draft tube performance affects the overall unit efficiency, it must be considered as an integral part of the hydro-turbine. Besides, through a state of the art revision, past numerical and experimental research has shown that the draft tube inlet velocity has a critical importance in the hydropower plants. However, there is a lack of information in terms of flow parameters and in particular of swirl distribution. In this respect the optimization of the inlet velocity profile is a new attempt to control the inlet flow conditions in order to improve the flow through the draft tube and consequently get a better performance. To achieve this goal, the optimization task will be guided by iSIGHT code which will combine CFD with an optimization algorithm. The detailed description of this methodology is the subject of the next chapter.

CHAPTER 2

OPTIMIZATION METHODOLOGY

To respond to the requirements of the current design competitions, both in terms of quality and time, CFD and optimization tools are increasingly being used to tackle a variety of flow problems. With these tools, engineers can explore different design alternatives in less time and at lower cost. Their use allows to model and simulate real world processes avoiding the construction of prototypes and the task of performing experiments. For hydraulic turbines, these tools can be used to predict the performance of power plant components during the design process or to propose modifications to the original design in rehabilitation or upgrading projects.

Numerical optimization methods have been successfully used for a variety of design problems. However, the optimization of velocity profiles in hydraulic turbines is a new challenge. First, the flow field inside a turbine draft tube is highly three-dimensional and complex. Therefore three-dimensional flow computations are essential despite their intensive and still expensive cost. Second, the draft tube performance is very sensitive to the inlet boundary conditions and swirling flow which requires that the inlet velocity profiles must be correctly parameterized. However, there is, at present, no satisfactory method for predicting the quantitative performance of a draft tube with arbitrary inlet flow. As the flow field is described by a system of nonlinear partial differential equations, the results of the mathematical evaluation of the draft tube performance will be multi-modal (several solutions) and non-linear. Finally, inlet flow optimization must be subject to some required constraints, such as a mass flow rate, swirling intensity, inlet back flow, and also subject to an extensive objective function evaluation.

The optimized profile may or not be of the shape that can be easily produced by a runner, but our goal is to obtain the optimal profile, rather than its technical feasibility. Consequently, the approach proposed to resolve this numerical optimization can be described in the following steps:

- Inlet flow parameterization,
- Flow field simulation,
- Optimization process.

This chapter will focus on establishing an optimization method to improve the swirling flow characteristics at the draft tube inlet. The coupling of iSIGHT, FLUENT and MATLAB commercial softwares has been performed and is presented to determine the best combination of the inlet flow parameters to yield improvement of the draft tube performance. In the following, details of these three stages will be described.

2.1 Problem formulation.

The optimization problem considered here is to seek an inlet velocity profile which optimizes the draft tube performance. This problem has been separated in there major stages. In the first stage, a suitable parameterizations with an appropriate number of variables should be chosen to approximate a real draft tube inlet velocity profile. The second stage considers the actual CFD calculations, where the principal goal is to obtain the pressure and flow fields. The results from this calculations are then sent to the third stage, the numerical optimization, where an objective function and the optimization algorithm have been specified.

The optimization evaluation will then result in some new values of the design variables, which in turn leads to a modification on the inlet velocity profile and a CFD simulation if the result is inadequate. On the other hand, if the result is satisfactory, an optimum design has been found.

The design strategy will be built using the iSIGHT software, where simulation codes of different disciplines can be coupled. This is configured through a graphical interface with which one can set up, monitor and analyze a design problem. Since there is not a single optimization method for all kind of problems, the software provides 13 different optimization techniques and the choice of the technique will depend mainly on the linearity of the problem and the time of search.

The optimization loop used in this process is shown in the Fig.2.1 and the simulations codes are run via a UNIX shell script I, as follows:

The control variables which describe the inlet velocity profile are introduced by means of a data file (1). The inlet velocity profile is parameterized using MATLAB (2) II.1 and II.2, and a profile file with the boundary conditions is obtained (3). The CFD code (FLUENT) reads this file (4) and computes the velocity and pressure fields which are specified by means of a data file (5). MATLAB runs to processes these values (6) and the objective function is transfered via a data file (7) II.3. In (8) the optimization algorithm evaluates the results according to the constraints values and if the objective function does not change, the maximum iteration number is reached or convergence criteria are met (9), the optimization is terminated. Otherwise, the process will continue to point (1) with a new set of swirl parameters proposed.

Figure 2.1 Inlet velocity profile optimization flow chart

2.2 First step: Inlet flow parameterization.

The choice of a parameterization method when coupled with optimization techniques to find desirable shapes has a major effect on the final results, efficiency and effectiveness of particular search strategies and it is the first step of the optimization methodology. Given the same CFD model, the parametrization effectively defines the optimization problem formulation, the topology of the design space, and the landscape of the objective function. Although it is vitally important, it is also very difficult to come up with a set of effective criteria that can be readily used to evaluate the pros and cons of different parameterization methods.

2.2.1 Approximating data.

In the draft tube inlet flow, it is often the case that an experiment produces a set of data points $(r_1, v_1)....(r_n, v_n)$ where the velocities v_k along the radius r_k are distinct. The

goal is to determine a function that relates these variables. Thus, the attempt will be to find a class of allowable functions and then to determine the coefficients. There are many different possibilities for the type of function that can be used, however, we will investigate an underlying mathematical model, based on the physical situation, that will determine the form of the function.

The data being considered which is known only at a set of discrete points and not as a continuous function will be approximated by fitting an approximating function. Three of the more common approximating functions are: polynomials, trigonometric and exponentials. In fact, any analytical function can be used as an approximate function. The approximating functions should be easy to determine, to evaluate, to differentiate and to integrate.

There are fundamentally two different ways to fit an approximating function to a set of discrete data: exact fits and approximate fits. An exact fit yields an approximating function that passes exactly through all discrete points. An approximate fit yields an approximating function that passes through the set of data in the best manner possible, without being required to pass exactly through any of the data points. While an exact fit can produce a curve that contains the given data points, it may oscillate or wiggle its way through every point. Approximation fit can overcome this problem so that the curve still captures the shape of the data points without containing all of them. Furthermore, the use of the exact or approximate fit procedure will arise from accuracy level of all experimental inlet velocity values.

2.2.2 Parametrization methods.

Depending on whether the goal is to improve on an existing design or to create a completely new one, different parametrization methods are required. If the new design only

demands small changes to the initial form, a localized parametrization approach is often used. But when conducting a study of a radically new concept, the parametrization method needs to accommodate a wider range of new shapes.

Three principal methods which can be used to model a curve (Lépine, 1999) are:

- Analytical equation : It describes the curve via a mathematical function.
- Discretisation :It works only with selected points on the curve.
- Interpolation : It passes the curve through selected points.

Thus, when a parametrization model is built around an existing shape profile, the model must be selected according to three important considerations given by Song & Keane (2004): first is the flexibility of the model, i.e. how many different shapes can this model represent. The second consideration is the robustness of the model i.e., the model could generate the desired shape for a large number of different designs. In general models with more design variables will be able to represent more complex shapes and will be more likely to produce novel designs using optimization. However, that will yield more expensive searches as the design space will have higher dimensions, and chances of failure or not generating desired shapes will be higher. The third consideration is the accuracy of the optimum objective function that the final shape can achieve either in an inverse design study (to solve for a geometry that produce a prescriber pressure distribution) or a direct optimization work (through an iterative process produce optimum profile shape). Finally, the model accuracy should be measured in both, geometric and fluid dynamic sense.

2.2.3 Parametrization goals.

Many optimization approaches need a parametric representation of the problem, which is usually the subject of separate studies. Since the complexity of an optimization problem rises exponentially according to the number of variables, it becomes apparent that the inlet velocity representation is crucial to the success of the optimization method. Thus the principal goals to represent the profile of the velocity components are;

- To find the relevant number of variables for a better approximation of the inlet velocity profile,
- To keep the continuity properties of a curve to guarantee smooth radial variation,
- To fit to experimental values given by a Francis and Kaplan runner,
- To explain physically its structure based on the velocity field in the draft tube inlet.

The means to reach these objectives consists in finding a method permitting to approach a shape profile. This method must consist of a compromise between the necessary precision to the complex profile definition and the quantity of information used for its description. First, it is necessary to determine what type of curve agrees best with the geometric shape that one wishes to approach, then to find parameters of the curve permitting the best approximation.

To approach a curve, one uses interpolation methods, generally with the use of polynomial methods of high degree. In an objective of optimization of velocity profiles, the interpolation would not be a satisfactory solution for modeling profiles, because of the large number of control points required which turn out to increase the number of design variables, (Lépine et al., 2001). In addition, one of the serious problems generated by

high degree polynomials is the oscillation and the propagation of fluctuations on the chosen interval. Moreover, using a polynomial expression, the vortex structure provoked by the profiles will not be physically related to any inlet parameter.

One of the key issues in deciding the parametrization method is the balance between requirements of robustness and flexibility, and these decisions are strongly dependent on the goal of the design activity. Although free-form parameterizations may well be able to generate radically new shapes, this is not suitable for designs where the aim is to meet a recovery pressure, due to the poor efficiency caused by the large search space that arise in the optimization process. Another disadvantage of free form parametrization is the inherent difficulties encountered when trying to generate profile-like shapes. Usually additional shape constraints need to be imposed.

Thus, in the following, it will be necessary to compare different parametrization approaches, to analyze their flexibility and robustness in producing optimal shapes when used in the optimization process. Approximate fit will be used to analyze the accuracy that the parametrization can achieve when used to model target inlet velocity profiles. The investigation of the best suited inlet velocity parameterization is the topic of chapter 3.

2.3 Second step: Flow field simulation.

The draft tube flow analysis is an indispensable step to obtain the objective function in the optimization of the inlet velocity profile process. To be able to describe a flow field in the draft tube, a mathematical model must be defined. This step is known as modeling and depending on the goal of the analysis, models of different complexity can be formulated. When the conceptual model has been determined, it will approximated via CFD which will resolve the equations that govern fluid flows on computers.

2.3.1 Computational Fluid Dynamics.

Hydropower flow predictions is one of the applications of Computational Fluids Dynamics (CFD) where most of the flow is turbulent. Essentially, CFD involves the setting up of partial differential equations and boundary conditions describing fluid flow over a physical domain. These equations are then solved numerically to generate the pressure and velocity fields in the flow domain. A critical point to keep in mind while exploring the utility of CFD codes is that because of the complexity of the physical problem, a description of fluid flows that is at the same time computationally economical and physically complete is not possible. For the purpose of numerically solving the Navier-Stokes flow equations, a number of simplifications are made, all of which affect the accuracy of the final solution.

Advanced CFD software is today available as a commercial tool due to its versatility in industrial applications. Since the code has been developed and used in a research environment, a high degree of familiarity with the setup and use of CFD problems is required. We have selected the control-volume-based commercial code FLUENT, as the package to perform our analysis.

2.3.2 Governing equations.

The flow in the draft tube is represented by the Navier-Stokes equations, i.e. by the continuity equation;

$$\frac{\partial u_i}{\partial x_j} = 0 \tag{2.1}$$

and the momentum equation;

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} = \frac{1}{\rho}\left(-\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}\right) \tag{2.2}$$

Where u_i represents the velocities in the x_i coordinate directions, p is the static pressure, ρ the constant density, and τ_{ij} the viscous stress tensor.

For a Newtonian fluid :

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{2.3}$$

Where μ is the dynamic viscosity.

2.3.3 Turbulence modeling.

Draft tube flow includes a number of challenging flow phenomena: swirling flows, streamline curvature, cross section changes, flow separation. Thus, an established and reliable approach to turbulence modeling is required for analyzing this kind of flow. The most important difference between the turbulent flow and laminar one is the appearance of various sizes of eddies and ranges of time. Because of this complexity, turbulence is very difficult to describe and to predict theoretically. So, the process of transportation of the turbulence in the draft tube will not be calculated with an exact method. As an alternative, it will be calculated by a turbulence model that uses empirical information and that permits to link the average quantities of transportation of the turbulence to the mean flow field.

Consequently, the Navier-Stokes equations are averaged over a time period. The physical variables are decomposed into mean and fluctuating components. Only the mean values are solved and therefore it is necessary to express the fluctuating values according to these ones.

$$\begin{aligned} \frac{\partial u_i}{\partial x_j} &= 0 \\ \frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} &= -\frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j}\left(\tau_{ij} - \overline{\rho u_i' u_j'}\right) \end{aligned} \tag{2.4}$$

The resulting equations where all variables are now averaged, describes the mean flow field and present an additional unknown called the Reynolds stresses $\overline{\rho u_i' u_j'}$. This term describes the influence of the turbulent field on the mean flow. Due to the nature of turbulence, it is impossible to model this term in a general way. It can not be expressed exactly as a function of the mean flow variables and it must be related to know quantities using a turbulence model.

There have been a wide range of methods used to model turbulence, using a reduced number of unknowns and of corresponding equations and they can be classified as:

- Zero equation model,
- One equation models,
- Two equations models,
- Reynolds stress models.

In the first three models, the Reynolds stress is assumed to be proportional to the local mean velocity gradients, in analogy to the viscous stress in laminar flow (Boussinesq's eddy viscosity concept).

$$\overline{\rho u_i' u_j'} = -\mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \frac{2}{3}\rho \delta_{ij} \kappa \tag{2.5}$$

where

μ_t - the turbulent (eddy) viscosity

κ - turbulent (fluctuating motion) kinetic energy

δ_{ij} - the Kronecker delta. $\delta_{ij} = 1$ for $i = j$ and $\delta_{ij} = 0$ for $i \neq j$

$\kappa = \frac{1}{2}\sqrt{\overline{u_i' u_i'}}$

Instead of employing the eddy viscosity assumption, which assumes an equal eddy viscosity in all three spatial directions, a Reynolds stress model has an equation for each of the six Reynolds stresses themselves. This allows the modeling of the transport of each of these individual stresses. Fig. 2.2 shows how these turbulent models relate to each other.

Figure 2.2 Tree of turbulence modeling, (Memarzadeh, 1998)

2.3.4 Near wall treatment.

Turbulent flows are significantly affected by the presence of walls. The near wall modeling impacts the fidelity of numerical solutions, inasmuch as walls are the main source of mean vorticity and turbulence. Fluid velocity at the wall surface is zero, which is known as the no-slip condition. The type of flow between the wall and the bulk flow is known as a shear layer, in this case, a wall boundary layer. The boundary layer is a very complex region of high velocity gradient, diffusion dominated development and where the viscous effects dominate. This region is not resolved in most practical calculations with $k - \epsilon$ model and RSM.

Some of the alternatives developed to model the viscosity-affected near-wall region are:

- The wall function approach: It uses semi-empirical formulas called "wall functions" to bridge the viscosity-affected region between the wall and the fully-turbulent region,
- The near-wall modeling approach: It modifies turbulence models to enable the viscosity-affected region to be resolved with a mesh all the way to the wall.

The wall function approach is popular because it is economical, robust and reasonably exact. It is a practical option for the near wall treatment for industrial flow simulations.

The connecting zone is described by a logarithmic relation between the near wall tangential velocity and the wall shear stress and can be expressed as:

$$u^+ = \frac{1}{k} ln(y^+) + B \tag{2.6}$$

With $u^+ = \frac{u}{u_\tau}$, $y^+ = \frac{yu_\tau}{\nu}$, $u_\tau = \sqrt{\frac{\tau_\omega}{\rho}}$.

Where u_τ is the wall tangential velocity, τ_ω the wall shear stress, k and B are empirical constants. The logarithmic zone corresponds to $30 < y^+ < 400$, The near wall region is divided in two additional zones: the viscous sub-layer for $0 < y^+ < 3$ and the buffer zone $3 < y^+ < 30$.

2.3.5 Flow solver.

The entire CFD analysis process can start when the draft tube geometry under investigation is subdivided into a number of cells shaped as hexahedral structured grid. One node in each cell holds the average value over the cell volume for each quantity. The differential equation are integrated over each volume to yield a set of algebraic equations, where the node values are the unknowns, in this case, the flux quantities over cell faces that need to be estimated.

FLUENT allows to choose either of two numerical solution methods: Segregated solver and coupled solver. In both solution methods, the discrete non-linear governing equations have been linearized to produce a system of equations for the dependent variables in every computational cell. The resulting linear system is then solved to yield an updated flow-field solution. In the segregated method, which is used in this work, the discrete governing equations are linearized implicitly with respect to that equation's dependent variable. By implicitly it is meant that the unknown value of a given variable in each cell is computed using a relation that includes both, "existing" and "unknown" values from neighboring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities. Because there is only one equation per cell, this is sometimes called a "scalar" system of equations. A point implicit linear equation solver (Gauss-Seidel) is used in conjunction with an al-

gebraic multigrid method (AMG) to solve the resulting scalar system of equations for the dependent variable in each cell.

This method, illustrated in Fig. 2.3, solves for a single variable field by considering all cells at the same time. It then solves the next variable field by again considering all cells at the same time, and so on. An iteratively converged solution is assumed to be reached when the largest normalized residual of the momentum equations, the continuity equation and the turbulence equations is reduced to 10^{-5}, (Cervantes & Engström, 2004).

Figure 2.3 Overview of segregated solution method.

2.3.5.1 Boundary conditions.

The implementation of the experimental boundary conditions in CFD often requires interpolation to the computational grid. In most cases the boundary conditions can be defined on every type of zone as functions of profile instead of constant values. Either a profile

contained in an external file or a function defined by the user (UDF) can be used. The code provides a mechanism for defining the boundary profiles. This device permits to use data, be it experimental, calculated data by an external program or extracted from a previous solution using a file for the profile as boundary condition for a variable.

Dirichlet conditions are set for velocities and turbulent quantities are imposed as profiles at the inlet. The influence of the details in the boundary conditions at inlets depends on the geometry following the inlet. Transport processes are important and the solution becomes sensitive to the boundary conditions if the geometry changes rapidly downstream of the inlet. With each velocity profile proposed, some boundary conditions however need to be either estimated, such as the radial velocity, or kept constant, such as the turbulence profiles. These quantities have demonstrated to have an important influence on both the flow features and the global performance quantities.

The outlet must be placed at the end of the diffuser. Inadequate outlet boundary conditions may also have an undesirable influence on the simulation. Here, a constant static homogenous pressure is set and Neumann conditions are imposed for other quantities. The velocity is an implicit result of the computation which must remain consistent with a fully developed flow, the normal viscous force component is assumed to be zero.

Walls in the draft tube are assumed to be smooth and no-slip conditions are imposed. The fluid velocity normal to the wall is zero; each boundary integration point mass flow is set to zero. The boundary wall momentum flow includes pressure and viscous forces. The pressure force is computed by interpolation of the control volume pressures. The normal viscous forces are set to zero and consequently the viscous force is equal to the tangential component.

Of course, when boundary conditions or the geometry are changed, a new validation model will be necessary.

2.3.5.2 Numerical accuracy.

The errors that are associated with CFD are different from those obtained in experiments. A CFD simulation will give exactly the same result for the same set of input parameters. The errors in CFD are associated with the numerical solution of a set of differential equations and are associated with the validity of the equations as a representation of a real flow. Numerical solutions are approximations and will always contain some degree of uncertainty and error, i.e. a deficiency. Uncertainty indicates that deficiencies may exist provoked by the lack of knowledge about the physical process that go into building the model. Error implies that the deficiency is identifiable upon examination.

Of the acknowledged errors, the assumptions and approximations in the mathematical model of the physical phenomenon should be validated against experimental data. Also it must be verified that the numerical solution of a set of partial differential equations is correct by means of the iterative convergence and discretization errors. Full validation of the numerical model is the topic of the chapter 4.

2.4 Third step: optimization method.

Mathematical optimization is the format title given to the branch of computational science that seeks to answer the question "What is the best" for problems in which the quality of any answer can be expressed as a numerical value. The goal can be formulated invariably to find the combination of parameters which maximize or minimize a given quantity possibly subject to some restrictions on the allowed parameter ranges. The quantity to be optimized is termed the *objective function*; the parameters which can be changed in the quest for the optimum are called *control variables*, the restriction on the allowed parameter values are known as *constraints*.

Optimization problems are classified according to the characteristics of the objective function, the constraints and the control variables. Probably the most important characteristic of the objective function is the linearity or not of its relationship with the control variables. The constraints reduce the space of solutions which is the feasible region and represents some knowledge about the simulated system. And the number and type of control variables should be able to permit to explore the search space. These control variables will be manipulated by a mathematical algorithm.

This is a maximizing or minimizing process of a desired objective function through the combination of independent variables that satisfy some constraints. Thus, the general optimization procedure may be stated mathematically as :

$$\begin{aligned} \text{minimize} \quad & f(\mathbf{x}), \quad \mathbf{x} = (x_1, x_2, \ldots, x_n)^T \\ \text{subject to} \quad & C_i(\mathbf{x}) = 0, \quad \mathbf{x} = 1, 2, 3, \ldots, m' \\ & C_i(\mathbf{x}) \geq 0, \quad i = m' + 1, \ldots, m. \end{aligned} \tag{2.7}$$

Where $f(\mathbf{x})$ is the objective function, $\mathbf{x}$ is the column vector of the n independent design variables and $C_i(\mathbf{x})$ is the set of constraint functions. Equations of the form $C_i(x) = 0$ are termed equality constraints, and those of the form $C_i(x) \geq 0$ are inequality constraints.

2.4.1 Optimization algorithms.

From the preceding discussion of the special characteristics of the problem we conclude that such an optimization strategy needs to:

1. optimize functions where no analytical expression is available;
2. apply a search technique;
3. incorporate practical constraints emanating from engineering knowledge of the problem.

There are several optimization algorithms available which can be selected for different types of problems. Nevertheless, a few methods are appropriate for a specific problem; to obtain the optimal inlet velocity profile to minimize the draft tube loss without a flow limits. So, first it is important to be able to recognize the characteristics of the optimization strategies in order to identify an appropriate solution technique.

We can separate the optimization methods in two categories: those that permit to determine a local minimum, these methods are called local methods, and those that endeavor to determine a global optimum, these methods are called global search methods. A global search permits to really explore the design space; this phase is called *diversification*; and a local search permits to really exploit a "promising" zone (susceptible to contain a global minimum), localized at the time of the exploration of the domain of search; this phase is called *intensification*.

The term local refers both to the fact that only information about a function from the neighborhood of the current approximation is used in updating the approximation as well as that we usually expect such methods to converge to whatever local extremum is closest to the starting approximation.

In general, most real-world optimization problems have several challenging properties. Nearly all problems have a significant number of local optima, and the search space can be so huge that the exact global optimum cannot be found in reasonable time. Additionally, the problems may have multiple conflicting objectives that should be considered simultaneously. Moreover, there may be a number of non-linear constraints to be fulfilled by the final solution. But then, why choose a global optimization method?

1. They do not require calculation of the local sensitivity of each design variable,
2. They can utilize the information collected from various sources and by different tools,
3. They can offer multi-criterion optimization,
4. They can handle the existence of multiple design points and trade-offs,
5. They can easily perform their task in parallel

Furthermore, the problem may have dynamic components altering the location of the optimum during the optimization process. For some problems, variants of the local search approach have proven to be very efficient, however, deterministic local search algorithms, such as steepest decent, do not allow a decrease in the solution's quality during the search. For this reason, these algorithms often stagnate at a local optimum, which makes local search less desirable for many real-world problems. The exploration using the traditional methods, deterministic and enumerative, are not efficient for all kinds of problems. The stochastic methods are able to resolve high dimensional problems where the calculation time is increased.

The major stochastic search methods are: simulated annealing, tabu search, Artificial Neural Networks and evolutionary algorithms. Among these techniques, the last (EAs)

seem to be a particularly promising approach for several reasons. Evolutionary algorithms are stochastic optimization methods which are particularly well suited for hard problems where little is known about the search space. They maintain a population of candidate solutions which are iteratively transformed by some nature-inspired operators: mutation, recombination and selection. The evolutionary process tends to find globally satisfactory, even if not optimal, solutions to optimization problem.

The Genetic Algorithms (GA) (Coley, 1999), belongs to the evolutionary algorithms which strategy is inspired by the evolutionary process encountered in nature. The GA method has had a lot of applications in shape optimization aerodynamic as it has been described in chapter 1. This method has proved to be efficient in optimization problems which are dominated by a combinatorial element and to seek optimum solution in wide space of objective function, as happens to be the case of the problem presented here. In addition, this method is applicable to complicated optimization problems and has an increased chance of obtaining a globally optimal solution in general. Nevertheless, its robustness is achieved at the expense of the need for greater computational power or parallel computing system.

A typical genetic algorithm might consist of the following steps:

1. a number, or population, of guesses of the solution;
2. a way of calculating how good or bad the individual solutions within the population are;
3. a method for mixing fragments of the better solutions to form new, on average even better solutions; and
4. a mutation operator to avoid permanent loss of diversity within the solutions.

In this manner, several factors has been considered to choose this model:

- it requires only information concerning the quality of the solution produces (objective function) by each parameter set. The GA does not require information on derivatives or complete knowledge of the problem structure and parameters.
- It does not require unreasonably good initial guesses but it considers many points in the search space simultaneously and therefore it should have a reduced chance of converging to a local optimum.
- It differs from many optimization methods by its flexibility and robustness and therefore the objective function could be a result of an numeric simulation.
- It is suitable for parallel computing because it is a population-based search algorithm and all design candidates in each generation can be evaluated in parallel.

By distributing parts of the genetic algorithm to several processing elements which work in parallel, it should be possible to speedup the calculations. A parallel model describes how different parts of the task can be calculated independently of the other parts. Thus, the parallel efficiency is extremely important, if objective function evaluation consumes most of the computational time as is the case for draft tube numerical simulation.

According to Miki et al. (1999) and Eklund (2004), parallel models have been categorized by the way they handle the population and researches in this area has separated into three categories: global population models, island models and the diffusion model.

The global population models have a single large population where each processor is assigned to treat some individuals and each individual has global interaction with each other. This parallel model is often referred to as the farming model or master-slave model.

The island models, also known as the regional model, multideme model (Multiple-population model), have multiple and smaller populations and exchange information among the subpopulations; this exchange is performed by moving some individuals from one population

to another and is know as migration. Genetic algorithms with divided sub-populations and the migration operation are called distributed GA (DGA) and DGA when they are performed on parallel computers are called parallel DGA.

The diffusion model, often referred to as the cellular GA or massively parallel GA, distributes its individuals evenly over a topology of processing elements. It can be interpreted as a global population laid out on a structure of processing elements, where the spatial distribution of the individuals defines the subpopulation or more exactly, the neighborhoods.

The island model is a very popular parallel model, partly because it is relatively easy to implement on a local network with standard work stations with one processor per deme or subpopulation.

2.4.2 Multi-Island Genetic Algorithm.

This exploratory technique is implemented within the iSIGHT software. The Multi-Island Genetic Algorithm (MIGA) technique corresponds to an advanced version of traditional genetic algorithm (GA) approaches. The main advantage of this approach is the possibility to start an optimization loop with a high chance of success avoiding extensive preliminary sensitivity analysis associated with gradient based methods. Furthermore, the MIGA technique is well suited for solving highly non linear problems, like the present one. This section gives a description of the major features of the MIGA technique. The reader is referred to the specific literature for a more detailed description of the optimization algorithms used, (Cardamone, 2006).

The major feature distinguishing a MIGA approach from a traditional GA is the fact that each population of individuals is divided into sub-populations called islands. The usual genetic operations (selection, reproduction and mutation) are performed separately on

each island. A further operation called migration is used to transfer some individuals from an island to another, as is illustrated in Fig. 2.4. The migration process is controlled by two major parameters which are the rate of migration and the interval of migration. The rate of migration corresponds to the percentage of individuals of each island which will be migrated to another island. The interval of migration indicates the number of generations between each migration process. Three other major parameters control the optimization process. These are the size of sub-populations, the number of islands, and the number of generations. These parameters may be adjusted so that the total number of evaluations of the objective function be the desired.

Figure 2.4 Conceptual flow of the MIGA.

In this work, the objective is not to obtain an absolute optimum profile, but only a solution which guarantees the nonexistence of an appreciably better solution. To reach this objective at the end of a reasonable calculation time, it is essential to have recourse to the parallelization of the distribution genetic algorithm.

2.4.3 Objective function

To establish the link between the CFD analysis code and the MIGA, it is necessary to introduce an objective function. This is a mathematical model which evaluates the draft tube performance based on direct flow analysis. In the present case, a number of parameters, such as the energy loss coefficient, mean pressure recovery and wall pressure recovery can be used. These parameters depend on the inlet boundary conditions (non-uniform and swirling inlet flow) and change according to the point of operation of the turbine.

The energy loss factor is the usual quantity to measure the efficiency of the draft tube. A general term applied to the energy that is converted to a form that can not be used (loss) during the operation of an energy producing, conducting, or consuming system. It considers the total pressure loss from the inlet to the outlet compared to the inlet kinetic energy. This factor is defined as:

$$\zeta = \frac{\frac{1}{A_{in}} \int_{in} P_t dA - \frac{1}{A_{out}} \int_{out} P_t dA}{\frac{1}{2}\rho \left(\frac{Q}{A_{in}}\right)^2} \tag{2.8}$$

Where A_{in} and A_{out} refer to the inlet and outlet areas and Q is the flow rate. The total pressure is $P_t = p + 0.5\,\rho\,(u^2 + v^2 + w^2)$, with u, v, w the cartesian velocity components. The total loss in the flux normalized with an energy flux estimator is independent of the swirl component at the inlet. It is not used in experimental work since it requires knowledge about the variables over the whole inlet and outlet cross sections. However in CFD, it is easily calculated and a lower value means less loss of the draft tube.

The wall pressure recovery factor is defined as;

$$Cp_w = \frac{P_{out:wall} - P_{in:wall}}{\frac{1}{2}\rho \left(\frac{Q}{A_{in}}\right)^2} \tag{2.9}$$

where $P_{out:wall}$ is the averaged static wall pressure across the outlet section and $P_{in:wall}$ is the averaged static wall pressure across the inlet section. The inlet averaged static wall pressure, $P_{in:wall}$, is a direct result of the whole field solution for all variables (if the pressure is set at the outlet). The pressure recovery factor indicates the degree of conversion of kinetic energy into static pressure where a higher value means higher efficiency for the draft tube. This value is considered at different points on the wall at the inlet and the outlet sections. The exact value of the pressure recovery factor depends on the whole field solution and can be seen as an integral property of the solution.

The mean pressure recovery factor is defined as;

$$Cp_m = \frac{\frac{1}{A_{out}} \int_{out} PdA - \frac{1}{A_{in}} \int_{in} PdA}{\frac{1}{2}\rho \left(\frac{Q}{A_{in}}\right)^2} \tag{2.10}$$

where P is the static average pressure. This pressure recovery factor indicates also the degree of conversion of kinetic energy into static pressure where a higher value means higher efficiency for the draft tube. The pressure is taken as a mean value at the inlet and outlet sections.

The choice of the objective function is a very important task for the successful application of the optimization method. In order to reduce the computational efforts, a single-value objective function approach should be chosen. This would reduce greatly the code running time but would require a higher knowledge of the significant variation gradients of the parameters in order to tailor the related terms of the objective function and balance in this way the weighting of the various goals.

In extensive preliminary investigations about draft tube, the mean pressure recovery fac-

tor formulation of the terms of performance have been tested in order to determine the objective function. However, these researches have been dedicated to a design optimization in which the inlet boundary condition has not been modified. In our case, since the inlet velocity flow field will be modified at each evaluation, a more complete evaluation at the inlet section must be taken into account. For this reason, in the following chapters the global performance quantities mentioned above will be evaluated in order to choose the most appropriate objective function.

2.5 Summary.

Despite the fact that numerical optimization methods have been successfully used for a countless number of design problems, an application of numerical optimization to the draft tube inlet flow is a challenge. This is because the possible objective function landscape of the flow field optimization is multi-modal (there may exist more than one optimal solution or the objective function may exhibit more than one maximum or minimum) and nonlinear (the flow field is governed by a system of nonlinear partial differential equations expressing the conservation of mass, momentum). Besides, calculation using a CFD code, especially in 3D is very expensive and this will require a very robust and efficient optimization tool.

In this respect, the present chapter has established an optimization methodology explaining each step of the optimization process of the draft tube inlet velocity profile: the inlet velocity parametrization, the CFD simulation and the numerical optimization. However, to have confidence in the results in this research, it remains before the optimization process be applied, to validate each step of the methodology. Thus, as a first step, various inlet velocity profile parametrization methods will be studied. This is the subject of the next chapter.

CHAPTER 3

SHAPE PARAMETRIZATION METHODS FOR THE DRAFT TUBE INLET VELOCITY PROFILE.

One of the most difficult challenges in this project is the determination of the velocity profiles that need to be generated to maximize the draft tube performance. In analogy with airfoil representations, an inlet velocity profile could be represented in a number of different ways. For example, coordinates might be directly used to fit the profile shape using B-splines and Bezier curves via interpolation methods or, analytical functions could be derived to represent the shape profile. In this chapter we present a study on parametrization methods for velocity profile shape optimization. The objective is to examine different approaches to represent experimental data sets at the inlet draft tube. This study begins by considering a purely geometric representation of velocity components along the runner's trailing edge, and subsequently by fitting different purely mathematical analytical equations to experimental data. Then, an analytical equation set presented recently by Susan-Resiga et al. (2006), shows to be an excellent option to fit experimental data. In the following we will detail each of these 3 different approaches to represent the inlet velocity profile: Geometric, through approximation and analytical.

3.1 Geometric representation of the meridional and tangential flow field.

Some statements have been made regarding the dependence of the inlet draft tube flow to the runner blade. Back in the 50's Nechleba (1957) already mentioned that the meridional flow field may be determined to a certain extent by the shape of the runner blade. More recently, Masse (1999) modified the runner outlet blade to reduce the hydraulic losses

in the draft tube. Nowadays Susan-Resiga et al. (2006) revealed that the relative flow angle of a swirling flow generated by a turbine runner, depends only on the blade exit angle provided that the flow remains attached. Thus, assuming an attached flux along the vane, the influence of the trailing edge geometry on the outlet flux will be accounted for representing the inlet velocity to the draft tube.

In order to parameterize geometrically the inlet velocity, we must link the runner trailing edge and the leading edge angles to the velocity triangles of the turbine by means of Euler equation 3.1.

$$H\eta_h = \frac{1}{g}\left(U_1 V_{t1} - U_2 V_{t2}\right) \tag{3.1}$$

Where H is the net head, U is the peripheral runner velocity and V_t is the tangential fluid velocity and η_h the hydraulic efficiency.

The runner blades have shapes that vary along the leading and trailing edges and consequently, the velocity components are not equal and the velocity triangles change in each point of the edges. From an energy point of view, the trailing edge is the most important section of the runner because the velocity not used will leave the runner and thus it must be considered as lost unless that a part of the energy is recovered in the draft tube. This is essentially kinetic energy which depends on absolute velocity V_2. This velocity has two components, the meridional velocity V_m and the tangential velocity V_t. In order to obtain dimensionless velocity values using a real fluid where viscosity and the internal friction exist, a space network must be used, Fig. 3.1. The flow may be characterized according to following assumptions:

First, in low speed turbines where the transition of the flow from the radial into the axial direction takes place at large radii, it is assumed that the centrifugal force, created by this

deflection of flow is so small that is has no influence on the distribution of the pressure in the streaming liquid. Under this assumption, we admit only the possibility of a velocity change along the traces of the flow surfaces. But, in turbines of higher speed, where the curvature of the flow in the meridional plane is greater, this assumption is not valid. Thus, as a second assumption, we must admit that the water particles streaming on the meridional curved paths generate centrifugal forces with which they act upon adjacent particles, so that the flow along the inner contour (i) proceeds under a higher pressure than the flow along the outer contour (e), Fig. 3.1. For this reason the flow surfaces are closer spaced towards the outer contour.

To find the major blade parameters, Kazan (1962) admit the use of the constant meridional velocity along the outer edge blade i.e. based on the one-dimension blade design theory. Therefore according to fig. 3.1 it must be established, $r_{2(x)}\Delta B_{2(x)} = cte.$ valid along the trailing edge.

Thus, the meridional velocity can be calculated by:

$$V_{m2} = \frac{\Delta Q}{2\pi r_{2(x)}\Delta B_{2(x)}} = cte. \tag{3.2}$$

Where $\Delta B_{2(x)}$ is the distance between two adjacent streamlines and $r_{2(x)}$ the radius to the trailing edge runner.

The other velocity components are obtained from fig. 3.2, which illustrate the velocity components at the distributor runner interface and from fig. 3.3, which illustrate the velocity components at the runner leading and trailing edges. Thus, the relative velocity of flow at any point of the trailing edge and any operational point is determined by:

Figure 3.1 Network used to determinate the flow conditions at the blade runner trailing edge.

$$w_{2(x)} = \sqrt{u_{2(x)}^2 + h_i - \left[\left(h_i - w_{2(m)}^2 + u_{2(m)}^2\right)\left(\frac{v_{m0(x)} sin\gamma_{0(x)}}{tg\alpha_0}\right)\right]} \tag{3.3}$$

and the tangential velocity by:

$$v_{t2(m)} = u_{2(m)} - \sqrt{u_{2(x)}^2 - v_{m2}^2 + h_i - \left[\left(h_i - w_{2(m)}^2 + u_{2(m)}^2\right)\left(\frac{v_{m0(x)} sin\gamma_{0(x)}}{tg\alpha_0}\right)\right]} \tag{3.4}$$

Where h_i is the specific indicated velocity and it is independent of the streamline:

$$2u_0 v_{t0(x)} + w_{2(x)}^2 - u_{2(x)}^2 = h_i \tag{3.5}$$

Figure 3.2 Velocity components at the interface runner-guide wheel.

For the best efficient point (bep) there is no shock velocity loss and $w_2 = u_2$.

With the relative flow velocity we can find the angle at which the flow comes out from the runner blades:

$$sin\beta_{2(x)} = \frac{v_{m2}}{w_{2(x)}} \tag{3.6}$$

The angle at which the water enters the runner is calculated from Fig. 3.2 for any streamline (x):

$$ctg\beta_{1(x)} = \frac{2u_{1(x)}^2 - \left[\left(h_i - w_{2(m)}^2 + u_{2(m)}^2\right)\left(\frac{v_{m0(x)}cos\gamma_{0(x)}}{tg\alpha_0}\right)\right]}{v_{m1(x)}} \tag{3.7}$$

Figure 3.3 Velocity components at the trailing edge blade runner.

Finally we obtain from the meridional velocity, the axial component:

$$sin\gamma_{2(x)} = \frac{v_{r2(x)}}{v_{m2}} \tag{3.8}$$

And the radial component:

$$cos\gamma_{2(x)} = \frac{v_{a2(x)}}{v_{m2}} \tag{3.9}$$

Where γ is the angle of inclination of the meridional streamline at the runner trailing edge.

According to this analysis, to obtain the circumferential component and the relative flow velocity in Eqs. 3.4 and 3.3, the variables along the trailing edge must be estimated. In this case the number of degrees of freedom given to the parameterized profile must be large

enough to explore a wide range of shapes, but small enough to limit as much as possible the number of parameters involved in the problem. Unfortunately we need several points along the runner trailing edge where we calculate each velocity component, because we do not obtain equal outlet triangles on the other streamlines, as shown in Fig. 3.4.

Figure 3.4 Stream line sketch of the essential number of points point to create the draft tube inlet velocity profile.

Hence conditions for entry into the draft tube will not be uniform on all stream lines. This should be taken into account to simultaneously obtain on all streamlines a velocity triangle with a shock-less entry into the draft tube which is a necessary condition for attaining good draft tube efficiency.

In this way, a multi variable optimization problem becomes very complex because of the large number of points along the trailing edge where the velocity triangles should be built. Thus, this constraint turns the inlet flow optimization into an expensive numerical

problem.

3.2 Non linear regression fitting.

As it was explained in the previous section, a geometric representation method was investigated in order to build the inlet velocity profile. However, this parametrization would use a very large number of variables. Another option is to find an analytical expression which is capable to approximate the experimental curve, namely to determine the formula $y = f(x)$ that relates these variables.

Usually, a class of allowable formulas is chosen and then coefficients must be determined. There are many different possibilities for the type of function that can be used. Often there is an underlying mathematical model, based on the physical situation, that will determine the form of the function.

While analyzing experimental data, it is often required to represent such data using a model based on mathematical equations. Regression analysis gives the ability to summarize a collection of sampled data by fitting it to a model that will accurately describe the data. These models may be empirical, semi-empirical or even phenomenological. Regression analysis can turn the sampled data points into a smooth continuous function that may be used analytically or utilized by a computer program to return expected values for certain values of the independent variable. Similar to linear regression, the goal of nonlinear regression is to determine the best-fit parameters for a model by minimizing a chosen merit function. Where nonlinear regression differs is that the model can have a nonlinear dependence on the unknown parameters, and the process of merit function minimization is an iterative approach. The process is to start with some initial estimates and incorporate algorithms to improve the estimates iteratively. The new estimates then become a starting point for the next iteration. These iterations continue until the merit function effectively

stops decreasing.

Thus, in the present study, a *Levenberg-Marquardt* nonlinear regression has been used to fit experimental data to various semi-empirical and phenomenological equations.

3.2.1 Approximation of experimental profiles.

The experimental inlet velocity profiles considered for the test are showed in Fig. 3.5. These represent draft tubes of Huites (HUI) , La Grande 3 (LG3) power stations in Mexico and Canada respectively, and those given by Mauri et al. (2002a) (MAU). These profiles have been normalized by the velocity magnitude. Table 3.1 shows the different equations used to find the best representation of the inlet velocity profiles. Figure 3.6 presents the root-mean-square error E_f, calculated by Eq. 3.10 achieved by means of the Polyfit program, Ruiz (2004), for each proposed equation. Where the $f(x_k)$ indicates the values obtained by the equation, while y_k denotes the values obtained by experiments.The best fitting line is the one which minimizes these error. The domain of the inlet velocity profiles has been divides in three sections: at the center $0.0 \leq R \leq 0.2$, the section to be approximated $0.2 \leq R \leq 0.8$ and for the boundary layer $0.8 \leq R \leq 1.0$.

Figure 3.5 Inlet velocity profiles for being approximated by mathematical equations established in table 3.1.

Number	Equation
Y (2.2)	$A + B * X + C * X^2$
Y (2.3)	$A + B * X + C * X^2 + D * X^3$
Y (2.4)	$A + B * X + C * X^2 + D * X^3 + E * X^4$
Y (3.1)	$A * X^B$
Y (4.1)	$A * exp(B * X)$
Y (4.2)	$A * exp(B/X)$
Y (4.3)	$A * ln(B * X)$
Y (4.4)	$A * ln(B/X)$
Y (4.5)	$A * exp(B * X) + C * X$
Y (4.6)	$A * log(B * X) + C * X$
Y (4.7)	$A/(1 + exp(B + C * X))$
Y (4.8)	$A + B/(1 + exp(C + D * X))$
Y (4.9)	$A * ln(B * X^C) + D$
Y(4.10)	$A * exp(B * X^C) + D$
Y(4.11)	$A * exp(B * X^C) * X^D + E$
Y (5.1)	$A * sin(B + C * X)$
Y (5.2)	$A * cos(B + C * X)$
Y (5.3)	$A * sinh(B + C * X)$
Y (5.4)	$A * cosh(B + C * X)$
Y (5.5)	$A * Tang(B + C * X)$
Y (6.1)	$A/(sinh(B * X)^C$
Y (6.2)	$A/(cosh(B * X)^C$
Y (6.3)	$A/(Tgh(B * X)^C$
Y (7.1)	$(A + B * X + C * X^2 + D * X^3) * (1 - X)^3/X^2$
Y (7.2)	$A * (1 - X)^B/X^C$
Y (8.1)	$A * exp(-B * (X - C)^2)$
Y (8.2)	$A * exp(B + C * X + D * X^2)$
Y (8.3)	$A * exp(-B * (X - C)^2) + D - E * X$
Y (8.4)	$A * exp(-B * (X - C)^2) * (D * X - E * X)$
Y (8.5)	$A * exp(-B * (X - C)^2) + (D * X - E * X)$
Y (8.6)	$A * exp(-B * (X - C)^2) + D * exp(-E * (X - F)^2)$
Y (9.1)	$((A - X) * (B - X))^2 + ((C - X) * (D - X))^2$
Y (9.2)	$A * exp(B/X) * (C/(C - X))^D$
Y (9.3)	$A * exp(B/X) * (C/(C - X))^(D + E * X)$
Y (9.4)	$A * exp(-B * (C - X)^2) + D * exp(-E * (F - X)^2)$
Y (9.5)	$A * (1 - X)^B * exp(-C/X) + D * (1 - X)^E * exp(-F/X)$

Table 3.1 Equations used to approximate the inlet velocity components using the Polyfit program, Ruiz (2004).

$$E_f = \left(\frac{1}{N} \sum_{k=1}^{N} |f(x_k - y_k)|^2 \right)^{1/2} \tag{3.10}$$

The results reveal a bad approximation of these equations to the experimental results even when using only part of the section profiles, with $0.2 \leq R \leq 0.8$. This technique has the disadvantage that tangent and axial velocities are not mathematically related. The analytical equations seem not to be physically appropriate. Furthermore they were not fitted through the proposed non-linear regression technique.

3.3 Analytical representation of the swirling flow at the runner draft tube interface.

The swirling flow structure at the interface runner-draft tube is determined by the runner design and the operating point. The flow structure can be represented by curves which can be described directly by a mathematical function. The analytic equation then permits get all the information on the shape in an exact way. However, no analytical representation of the inlet velocity profile at the draft tube interface runner had been presented considering the non linear radial variation of the axial and tangential components.

The current alternative proposed by Susan-Resiga et al. (2006), is based on the formulas employed in the experimental study of the vortex breakdown phenomenon. This study included the axial and circumferential velocity components at the runner outlet for 17 operating points. This study concluded that the swirling flow at the runner outlet for a Francis turbine can reasonably be represented using a superposition of elementary vortices, incorporated into the following equation set:

Figure 3.6 Root-mean-square error.

For tangential velocity:

$$V_t(r) = \Omega_0 r + \Omega_1 \frac{R_1^2}{r}\left[1 - exp\left(-\frac{r^2}{R_1^2}\right)\right] + \Omega_2 \frac{R_2^2}{r}\left[1 - exp\left(-\frac{r^2}{R_2^2}\right)\right] \tag{3.11}$$

And for the axial velocity:

$$V_a(r) = U_0 + U_1 exp\left(-\frac{r^2}{R_1^2}\right) + U_2 exp\left(-\frac{r^2}{R_2^2}\right) \tag{3.12}$$

Where the discharge coefficient can be reached by:

$$\varphi = U_0\, r + U_1 R_1^2 \left[1 - exp\left(-\frac{R_s^2}{R_1^2}\right)\right] + U_2 R_2^2 \left[1 - exp\left(-\frac{R_s^2}{R_2^2}\right)\right] \tag{3.13}$$

Where the design parameters are:

V_a dimensionless axial component of velocity.

V_t dimensionless tangential component of velocity.

R_s dimensionless radius of the survey section.

R_1, R_2 dimensionless vortex core radii.

$\Omega_0, \Omega_1, \Omega_2$ dimensionless angular velocities.

U_0, U_1, U_2 dimensionless axial velocities.

r dimensionless radius.

φ discharge coefficient.

The velocity is made dimensionless by the runner angular speed while a length is transformed into a dimensionless quantity with respect to the runner outer radius. The swirling flow model described by Susan-Resiga et al. (2006) and Muntean et al. (2004) with the Eqs. 3.11 and 3.12 has eight parameters for the three elementary vortices, as follows: Two Batchelor vortices, one co-rotating and the other counter-rotating with respect to $V_t = \Omega_0 r$, and co-flowing/counter-flowing with respect to $V_a = U_0$.

3.3.1 Francis inlet velocity profiles approximation.

The parameters $\Omega_0, \Omega_1, \Omega_2, U_0, U_1, U_2, R_1, R_2$ are found by fitting both the tangential and axial velocity profiles. Figs. 3.7 for the HUITES model and Figs. 3.8 for the LG3 model turbine show the experimental data, as well as their analytical representation using Eqs. 3.11 and 3.12. By integrating the axial velocity profile, one obtains the theoreti-

cal discharge coefficient φ_{th} by the Eq. 3.13. The analytical inlet velocity in the radial direction was calculated according Mauri et al. (2002a) by $Vr = Va \cdot tan(\gamma)$.

Table 3.2 shows the swirl parameters for the two machines (LG3) and (HUI). The quality of the analytical fit can be appreciated also by the root-mean-square error for the inlet velocity components which it was calculated using the Eq. 3.10. This error is often used because it is much easier to minimize computationally than the maximum or average error. This error shows that the analytical approximation correctly represents the experimental data for the operating points. The theoretical φ_{th} and experimental φ_{ex} discharge coefficient are also compared for both turbines. The experimental discharge coefficient has been calculated by:

$$\varphi_{ex} = \frac{Q}{\omega \pi R_s^3} \tag{3.14}$$

Where: ω is the angular speed of the runner.

In this preliminary study the agreement between numerics and experiment is quite good. Therefore the vortex equation can be applied to approximate other experimental inlet velocity data produced by a Francis runner.

3.3.2 Kaplan inlet velocity profiles approximation.

In order to apply the Eqs. 3.11, 3.12 and 3.13 on the Turbine 99 draft tube, which operates with a Kaplan runner, the survey section has been changed to the section Cs1b as shown in the Fig.3.9. This was mainly done because at the Cs1a center the hub of the runner breaks the completeness of the swirl. A second modification was carried out for the characterization of the current case. First it was noted that the axial velocity profile for

Figure 3.7 HUITES swirling flow on draft tube inlet at best efficient operating point (bep) up, and rated output operating point (rop) down.

Figure 3.8 LG3 swirling flow on draft tube inlet at best efficient operating point.

this Kaplan turbine near the wall was higher than the one developed by a Francis runner. Next, the angular velocity for the second vortex Ω_2, at the inlet of a Francis runner showed a higher value relative to the other parameter vortex. So, looking for a better approach to represent the experimental inlet velocity, the equation proposed by Burgers (Alekseenko et al., 1999), with a different modeling was tested for the third vortex as:

Swirl Para	LG3	HUI bep	HUI rop
R_1	0.2783	0.1300	0.0873
R_2	0.9955	0.4218	0.5635
U_0	0.0709	0.7476	0.7026
U_1	-0.1153	-1.0940	-0.3730
U_2	0.3125	0.6176	0.8863
Ω_0	0.1791	0.3355	0.2182
Ω_1	0.2789	6.0530	-4.847
Ω_2	-0.1866	-1.085	-0.4579
rms Va	0.0098	0.0522	0.0359
rms Vr	0.0002	0.0076	0.0034
rms Vt	0.0065	0.0193	0.0165
φ_{th}	0.3130	1.1052	1.2275
φ_{ex}	0.3174	1.135	1.275

Table 3.2 Parameters for the swirling flow approximation on draft tube inlet for the experimental inlet velocity values from La Grande 3 (LG3), Huites best efficient point (HUI)(bep) and Huites rated output point (rop).

$$V_t = \frac{\Omega_1 R_1^2}{r}\left[1 - exp(-R_1 r^2)\right] \tag{3.15}$$

and

$$V_a = U_0 + U_1 exp(-R_1 r^2) \tag{3.16}$$

Figure 3.9 Turbine 99 cone geometry with the inlet section survey at Cs1b.

The model parameters Ω, U and R are found by fitting the experimental data and the results are shown in Table 3.3 with a experimental discharge coefficient $\varphi_{exp} = 0.1706$.

The Root-mean-square approximation for axial ($R - sqV_a$) and tangential velocity ($R - sqV_a$) was calculated for Batchelor (Ba) and Burgers (Bu) equations using three (3v) and four (4v) vortex.

The graphical results of the approximation are shown in Fig.3.10(a) for the axial velocity and 3.10(b) for the tangential velocity, for both the Batchelor's and Burger's three and four vortex profile specifications. In the present preliminary study, the three Batchelor vortex parameterization (Ba-3v) were used to represent inlet profiles because of the reduced number of parameters involved. The approximation using the four Batchelor vortex representation (Ba-4v) seems the most accurate even though the number of design variables is increased.

(a) Axial velocity (b) Tangential velocity

Figure 3.10 Velocity profiles approximations to experimental one at Cs1b section.

3.4 Summary.

This chapter has explained the parametrization methodology developed to obtain the best draft tube inlet velocity representation. In selecting a shape representation for the inlet

Swirl Par	Bu-4v	Ba-4v	Ba-3v	Bu-3v
R_1	0.1602	0.1634	0.1709	0.1797
R_2	0.9719	0.9418	0.5205	1
R_3	0.3061	1.07	-	-
U_0	-0.4261	0.0363	0.1667	0.1797
U_1	-0.1748	-0.1764	-0.1818	-0.1894
U_2	-0.4492	-0.9034	-0.03005	-0.0318
U_3	1.0	0.9935	-	-
Ω_0	-1.374	-0.3267	0.1149	0.2283
Ω_1	0.5148	0.5061	0.4604	0.3922
Ω_2	-1.273	-3.259	-0.1965	-0.2646
Ω_3	8.413	3.51	-	-
φ_{th}	0.6079	0.1718	0.1740	0.1743
R-sq Va	0.9494	0.9695	0.9543	0.9546
R-sq Vt	0.9602	0.9638	0.9728	0.8329

Table 3.3 Parameters for the swirling flow approximation on draft tube inlet using three and four vortices.

velocity, a first issue has been to reduce the resulting number of design variables as much as possible, because the complexity of the optimization process can be directly linked to this factor. Although parametric curves (Bezier, Splines, NURBS) have been used to represent complicated geometries in turbo-machinery optimization shape, they will not give any physical meaning of the draft tube inlet flow structure.

Moreover, the use of the geometric parameters such a flow angle of the blades has shown to require a large number of variables to represent the three velocity profile components along the runner trailing edge. Another approximation was made using the Polyfit program which has a set of different analytical equations. This program uses a *Levenberg-Marquardt* nonlinear regression algorithm to find the parameters values. The results were not suitable. Finally, an approximation was developed using the equation set proposed recently by Susan-Resiga et al. (2006). The approximation results show that these equations match the inlet velocity profiles produced by a Francis runner of the Huites and La Grande 3 hydroelectric power plants draft tubes.

Then, these equations were used to approximate the experimental data given by a Kaplan runner fixing the swirl parameter values through fitting. It was found that when more

vortices are added more exact is the approximation to the experimental values, mainly near the wall. However the number of design variables is increased. That allows the possibility to add to vortex equations a near-wall velocity profile, (Mauri et al., 2002a).

In this way, having found reliable parametrization methodology to approximate the draft tube inlet velocity profiles, the first step of the optimization process has been completed. The next step will be the validation and verification of the draft tube numerical model.

CHAPTER 4

VALIDATION OF THE T99 DRAFT TUBE MODEL

In the application of industrial computational fluid dynamics (CFD), the quality of simulations and the level of reliability of the results have emerged as major issues of concern. Three elements involved in CFD have been recognized as critical: grid generation, algorithm development and turbulence modeling. While for the first two elements mathematical theories exist, turbulence modeling is far less developed because of the complex nature of the turbulence phenomenon. As a representative of high-order turbulence models, large eddy simulations (LES), as well as the stress and $k - \epsilon$ models can be identified. Among these, the $k - \epsilon$ models have become popular because of their short computational times and their simplicity. Moreover, as the strengths and weaknesses of these models have become apparent, improvements have been made to enhance their performance. In the commercial code FLUENT, two of these variants are available: the RNG model and the realizable $k - \epsilon$ models. The RNG model is derived from instantaneous Navier-Stokes equations, using a statistical technique called renormalization group theory, while the realizable $k - \epsilon$ models satisfy certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows (FLUENT, 2003). The flow in the draft tube is an example of problems which are a challenge for industrial CFD applications, due to the highly rotational, fluctuating and non-uniform flow that leaves the runner.

In this chapter, the commercial code FLUENT (version 6.1.22) is used to study various options of the $k-\epsilon$ turbulence models and their interaction with the discretization schemes of the equations governing fluid flow. The grid, discretization and turbulence combined choices are discussed through the accuracy of $k - \epsilon$ turbulence models for the swirling flow in the Turbine 99 draft tube. The influence of the first-order upwind, second-order

upwind, third-order upwind (QUICK) and Power Law schemes over these models has been studied. As the turbulent flows are significantly affected by the presence of walls, the wall function and the near-wall models were tested for modeling the near-wall region. Two different grids with $y^+ = 1$ and $y^+ = 50$ at the wall adjacent cell were used to study the flow behavior. Discussion is based on graphical results and by comparing numerical simulations and experiments in the *operational mode T* (best efficiency point). In this study, modeling includes geometrical data, as well as boundary and initial conditions, as will be described in the next section.

4.1 Computational aspects.

The Hölleforsen Kaplan model draft tube, located in Indalsälven Sweden, was used to carry out this study. This geometry has previously been used in two ERCOFTAC workshops (Engström et al., 2001, Cervantes & Gustavsson, 2005). The configuration is a $1 : 11$ scale model of the real turbine including the spiral case and the draft tube. The model runner has a diameter of 0.5 m and has five blades and 24 guide vanes. The tip clearance between the runner blades and the shroud is 0.4 mm. The computations are made at a head of $H = 4.5$ m, a runner speed of $N = 595$ rpm and a volume flow rate of $Q = 0.522\ m^3/s$. This operation condition is close to the best efficiency point (T).

4.1.1 Geometry and grid.

Detailed velocity and pressure measurements has been made by (Andersson, 2001) at a number of measurement sections. These measurements are used both to set the correct boundary conditions and to validate the computational results. A perspective view of the draft tube is shown in Fig. 4.1(a). Figure 4.1(b) presents a vertical symmetrical plane

which includes six cross sections where the results are compared. The origin of the coordinate system is placed at the concentric center of the inlet (Cs1a) with the z axis in the vertical direction. The grids used in this study are predefined block-structured hexahedral with approximate first-node wall distances of $y^+ = 50$ for the first grid and of $y^+ = 1$ for the second. The same number of points (1,002,360) is used for both grids.

(a) Geometry

(b) Measurement sections

Figure 4.1 Geometry and measurement sections of the draft tube.

4.1.2 Boundary conditions.

The axial and tangential velocities boundary conditions were imposed along a radial line at Cs1a, yielding steady axi-symmetric inlet boundary conditions. Due to the absence of the experimental radial component, this velocity distribution was assumed to be attached to the runner cone and the draft tube wall and it was obtained by:

$$V_r = V_a \cdot tan(\theta) \tag{4.1}$$

and:

$$\theta = \theta_{cone} + \frac{\theta_{wall} - \theta_{cone}}{R_{wall} - R_{cone}}(r - R_{cone}) \tag{4.2}$$

Where $R_{cone} \leq r \leq R_{wall}$, and $\theta_{cone} = -12.8^o$ and $\theta_{wall} = 2.8^o$. The velocity profiles was scaled in order to get the correct volume flow. From the measurements, the root mean square values of the U and W component velocities were available, corresponding to the Reynolds stresses $\overline{u^2}$, $\overline{w^2}$ and $\overline{uw}$. The rest of the Reynolds stresses were estimated as $\overline{v^2} = \overline{w^2}$ and $\overline{uv} = \overline{vw} = \overline{uw}$. The turbulence inlet boundary conditions were set using the measurements along the radial line at Cs1a for the inlet kinetic energy as:

$$k = 0.5\left(\overline{u^2} + \overline{v^2} + \overline{w^2}\right) \tag{4.3}$$

and for the inlet dissipation rate of kinetic energy:

$$\epsilon = \frac{k^{3/2}}{l_s} \tag{4.4}$$

where $l_s = 0.1$ m is a constant length scale. An outlet boundary condition is set at the

end of the draft tube. The static pressure is here a constant value. At the runner hub a rotating wall boundary condition is set with the runner speed. For all other walls, cone and side-wall, no-slip wall condition was set.

4.2 $k - \epsilon$ turbulence models.

The standard, RNG and realizable $k - \epsilon$ models have similar transport equation forms for k and ϵ. The RNG model incorporates modifications over the standard $k - \epsilon$ model. The analytical derivation results in a model with constants different from those in the transport equations for k and ϵ. The general form of the equations is similar to the standard $k - \epsilon$ model. This includes an additional term in the equation that significantly improves the accuracy for rapidly strained flows, including the effect of swirl on turbulence. Besides, it provides an analytical formula for turbulent Prandtl numbers, while the standard model uses constant values. Finally the effective viscosity is derived from an analytical differential equation that accounts for low-Reynolds number effects.

The realizable $k - \epsilon$ model is a relatively recent development and differs from the standard model in two important ways. First, the model uses a different formulation for turbulent viscosity. Second, it includes a new transport equation for the dissipation rate, ϵ. These special features ensure that this model satisfies certain mathematical constraints on the normal stresses, consistent with the physics of turbulent flows.

4.2.1 Standard Model (STD).

This is a semi-empirical model based on the model transport equation for turbulent kinetic energy k and its dissipation rate ϵ. In deriving this model, which was developed by Launder & Spalding (1974), it is assumed that the flow is fully turbulent and that the effects

of molecular viscosity are negligible. These quantities are obtained from the following transport equation for k:

$$\rho\frac{\partial k}{\partial t} + \rho U_i\frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \tau_{ij}\frac{\partial U_i}{\partial x_j} - \rho\epsilon \tag{4.5}$$

and for ϵ:

$$\rho\frac{\partial \epsilon}{\partial t} + \rho U_i\frac{\partial \epsilon}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}\tau_{ij}\frac{\partial U_i}{\partial x_j} - C_{2\epsilon}\rho\frac{\epsilon^2}{k} \tag{4.6}$$

Where $C_{1\epsilon} = 1.44$ and $C_{2\epsilon} = 1.92$ are constants, $\sigma_k = 1.0$, $\sigma_\epsilon = 1.3$ are the turbulent Prandtl number for k and ϵ respectively. The turbulent viscosity is calculated by:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{4.7}$$

with $C_\mu = 0.09$.

4.2.2 Renormalization Group Model (RNG).

The RNG-based $k - \epsilon$ model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called "renormalization group theory", and its turbulent

kinetic energy and its dissipation are acquired from the following transport equations:

$$\rho\frac{\partial k}{\partial t} + \rho U_i\frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\alpha_k\mu_{eff}\frac{\partial k}{\partial x_j}\right) + \tau_{ij}\frac{\partial U_i}{\partial x_j} - \rho\epsilon \tag{4.8}$$

and

$$\begin{aligned}\rho\frac{\partial \epsilon}{\partial t} + \rho U_i\frac{\partial \epsilon}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\alpha_\epsilon\mu_{eff}\frac{\partial \epsilon}{\partial x_j}\right) + \\ C_{1\epsilon}\frac{\epsilon}{k}\left(\tau_{ij}\frac{\partial U_i}{\partial x_j} + C_{3\epsilon}\right) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} - R_\epsilon\end{aligned} \tag{4.9}$$

The turbulent viscosity. The effective turbulent viscosity μ_{eff} is modeled by a differential equation:

$$d\left(\frac{\rho^2 k}{\sqrt{\epsilon\mu}}\right) = 1.72\frac{\hat{\nu}}{\sqrt{\hat{\nu}^3 - 1 + C_\nu}}\,d\hat{\nu} \tag{4.10}$$

where $\hat{\nu} = \mu_{eff}/\mu$ is the turbulent kinematic viscosity and $C_\nu \approx 100$.

Equation 4.10 is integrated to obtain an accurate description of how the turbulent transport varies with the effective Reynolds number. By default, the effective viscosity is computed using the high Reynolds number to obtain Eq. 4.7, with $C_\mu = 0.0845$.

The Prandtl numbers. The effective inverse Prandtl numbers α_k and α_ϵ involved in Eqs. 4.8 and 4.9 are computed using the following formula, derived analytically through RNG theory:

$$\left|\frac{\alpha_{k,\epsilon} - 1.3929}{\alpha_0 - 2.3929}\right|^{0.6321} \left|\frac{\alpha_{k,\epsilon} - 1.3929}{\alpha_0 - 2.3929}\right|^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}} \tag{4.11}$$

where $\alpha_0 = 1.0$ refers to molecular properties. In the high-Reynolds-number limit, $\frac{\mu_{mol}}{\mu_{eff}} \ll 1$, and $\alpha_k = \alpha_\epsilon = 1.393$.

The R term in the ϵ equation.

The additional R_ϵ term in Eq. 4.9 is given by:

$$R_\epsilon = \frac{C_\mu \, \rho \, \eta^3 (1 - \eta/\eta_0)}{1 + \beta \, \eta^3} \frac{\epsilon^2}{k} \tag{4.12}$$

.

where $\eta \equiv \mathbf{S} \, k/\epsilon$, $\eta_0 = 4.38$ and $\beta = 0.012$. $\mathbf{S}$ is a scalar measure of the deformation tensor (strained flows) and is based on the magnitude of the vortices.

4.2.3 Realizable Model (REA).

The realizable model satisfies certain mathematical constraints on the Reynolds stresses consistent with the physics of turbulent flows. The model transport equations for k and ϵ in the realizable model are:

$$\rho \frac{\partial k}{\partial t} + \rho U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} \frac{\partial U_i}{\partial x_j} - \rho \epsilon \tag{4.13}$$

and

$$\rho\frac{\partial\epsilon}{\partial t}+\rho U_i\frac{\partial\epsilon}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right]+$$
$$\rho C_1 S\epsilon-\rho C_2\frac{\epsilon^2}{k+\sqrt{\nu\epsilon}}+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon} \tag{4.14}$$

Where $C_1 = max\left[0.43, \frac{\eta}{\eta+5}\right], \eta \equiv \mathbf{S}\,k/\epsilon, \mathbf{S} = \sqrt{2\mathbf{S}_{ij}\mathbf{S}_{ij}}, C_{1\epsilon} = 1.44, C_2 = 1.9, \sigma_k = 1.0,$ $\sigma_\epsilon = 1.2$

The realizable model differs from the standard and RNG models in the following respects:

The turbulent viscosity.

A variable C_μ, which is a function of the mean strain, the rotation rate, the angular velocity of the system rotation and the turbulent field k and ϵ, being computed from:

$$C_\mu = \frac{1}{A_0 + A_s\frac{kU^*}{\epsilon}} \tag{4.15}$$

Where

$$U^* = \sqrt{\mathbf{S}_{ij} + \widetilde{\Omega}_{ij}} \tag{4.16}$$

Where $\widetilde{\Omega}$ is the mean rate of rotation tensor and $\mathbf{S}$ the mean rate of strain tensor A_0 and A_s are model constants.

The new dissipation ϵ equation. A new model for the dissipation equation 4.14, where the production term (the second from the right) does not involve the production of k. Also, the denominator on the destruction term (the third from the right) never vanishes, even if k vanishes or becomes smaller than zero.

4.3 Discretization schemes.

When the set of differential equations has been chosen, including the equations for the turbulence model, they must be transferred to a discretized space of nodes and cells in a computational grid. The method used by FLUENT for this purpose is the finite volume method (FVM).

The geometry under investigation is subdivided into a large number of cells, and each cell holds the average value over the cell volume for each quantity. The differential equations are integrated over each volume to yield a set of algebraic equations where the node values are the unknowns. The algebraic equations are then solved. A differential scheme is used to discretize the partial derivatives. It is therefore of interest to see what the penalty is for the use of each of these transport equation discretization methods on the turbulence model.

4.3.1 The four FLUENT discretizaton schemes.

FLUENT allows choose several upwind schemes:

The First-Order Upwind scheme (FOU). The simplest FOU scheme was originally developed from a comparison with boundary layer flow with scalars being transported downstream by the flow. Thus, the face value of the scalars is set equal to the cell-centered value in the upstream cell.

Power Law scheme (POL). This scheme uses an exponential approximation of the scalar field, and is called the exponential scheme, although it is sometimes referred to as the exact scheme. This scheme is only accurate for one-dimensional flow, and exhibits numerical diffusion in multidimensional flows.

The Second-Order Upwind scheme (SOU). In SOU scheme, the value of a scalar is approximated by an SOU extrapolation. In this approximation, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid.

The Quadratic Upstream Interpolation scheme (QUI). The QUI for convective kinetics (QUICK) scheme uses a three-point upstream-weighted quadratic interpolation to compute cell face values. The face value of the scalar is obtained from a quadratic function passing through two bracketing nodes (one on each side of the face) and a node on the upstream side.

4.3.2 Results of the discretization scheme study.

This part of the study presents the results of our investigation of the influence of the discretization schemes over the $k - \epsilon$ turbulence models. The global performance quantities used as a basis of comparisons are summarized in Table 4.1. The pressure recovery Cp_w, based on the wall pressure, is used for comparison with the experimental value, which is 1.12. The energy loss factor ζ is not used in experimental work, and a lower value means less loss for the draft tube. The pressure recovery Cp_m based on the mean values of the pressure over the inlet and outlet areas is a more relevant value when the performance of the draft tube is evaluated.

Figures 4.2 and 4.3 present the results of the pressure coefficients along the centerline wall. The most significant difference in this parameter is presented by the RNG model on both grids. There are only small variations between the other two models, standard and realizable. Nevertheless, the QUI and SOU schemes seem to approximate the experimental results in the same way for all the models with both grids.

As a qualitative illustration of fluid flow characteristics, Figs. 4.5 and 4.6 show the results of the axial velocity contours at the Cs2 section. It can be observed that the RNG model is more sensitive to discretization scheme changes. The POL scheme for the $y^+ = 50$ grid presents almost the same details as the experimental results shown in Fig. 4.4.

Figures 4.8 and 4.9 display the velocity magnitude contours for the horizontal velocity for the Cs3. These results confirm the sensitivity of the RNG model to the discretization scheme and to grid quality. Otherwise, the standard and realizable models seem to be rather insensitive to the discretization schemes, but not to grid quality.

	$y^+ = 1$			$y^+ = 50$		
TEST	Cp_m	ζ	Cp_w	Cp_m	ζ	Cp_w
FOU std	0.8724	0.1798	0.9663	0.8852	0.1623	0.9956
SOU std	0.8841	0.1704	1.0034	0.8956	0.1546	1.0310
POL std	0.8749	0.1780	0.9733	0.8880	0.1619	1.0036
QUI std	0.8842	0.1703	1.0040	0.8956	0.1545	1.0318
FOU rng	0.8765	0.1782	0.9680	0.8714	0.1760	0.9807
SOU rng	0.9128	0.1117	1.0415	0.9929	0.0840	1.1273
POL rng	0.8298	0.2162	0.9274	0.8760	0.1563	0.9934
QUI rng	0.9037	0.1031	1.0328	0.9888	0.0874	1.1243
FOU rea	0.8554	0.1786	0.9551	0.8661	0.1646	0.9787
SOU rea	0.8591	0.1725	0.9866	0.8634	0.1675	1.0043
POL rea	0.8602	0.1771	0.9659	0.8653	0.1652	0.9854
QUI rea	0.8588	0.1727	0.9872	0.8725	0.1620	1.0142

Table 4.1 Global performance variation for the discretization schemes.

(a) Upper centerline wall (b) Lower centerline wall

Figure 4.2 Pressure coefficients along the centerline of the upper and lower walls for the discretization schemes using the $y^+ = 1$ grid.

(a) Upper centerline wall (b) Lower centerline wall

Figure 4.3 Pressure coefficients along the centerline of the upper and lower walls for the discretization schemes using the $y^+ = 50$ grid.

Figure 4.4 Experimental variation of the axial velocity contours and horizontal velocity vectors at the Cs2, Andersson (2001).

Figure 4.5 Axial velocity contours and horizontal velocity vectors, caused by different discretization schemes, at the Cs2 section for the $y^+ = 1$ grid.

Figure 4.6 Axial velocity contours and horizontal velocity vectors, caused by different discretization schemes, at the Cs2 section for the $y^+ = 50$ grid.

Figure 4.7 Experimental variation of the horizontal velocity contours at the Cs3, Kurosawa et al. (2001).

Figure 4.8 Horizontal velocity contours, caused by different discretization schemes, at the Cs3 section for the $y^+ = 1$ grid.

Figure 4.9 Variation of the horizontal velocity contours, caused by different discretization schemes, at the Cs3 section for the $y^+ = 50$ grid.

4.4 RNG swirl modifications.

Turbulence is, in general, affected by rotation or swirl in the mean flow, and, in order to account for its effects, the turbulent viscosity must be appropriately modified.

4.4.1 Modifications to the model.

This modification takes the following functional form:

$$\mu_t = \mu_{t0} f\left(\alpha_s, \Omega, \frac{k}{\epsilon}\right) \tag{4.17}$$

This choice allows the model to better handle low Reynolds numbers and near-wall flows. In Eq. 4.17, μ_{t0}, the value of the turbulent viscosity is calculated without the swirl modification, using either Eq. 4.7 or 4.10. The parameter Ω indicates the characteristic swirl number and α_s is a swirl number that assumes different values, depending on whether the flow is swirl-dominated (*sdf*) or swirls only mildly.

4.4.2 Results of the swirl modifications.

Table 4.2 presents the engineering quantities obtained using the RNG options for both grids. The FOU scheme was used to discretize the equations governing fluid flow.

Figures. 4.10 and 4.11, show the variation of the pressure coefficients along the centerline, where (*dvm*) is the differential viscosity modification, (*dvm-sdf*) refers to the swirl-dominated flow and (*sdf*) (0.007, 0.07, 0.7) represents the swirl intensity of the fluid. It can be seen that the numerical results on the wall's centerline depend on the intensity of

the swirl at the inlet for both grids. If the intensity is increased, the results will move further away from the experimental values. The swirl-dominated flow parameter (*sdf*) with 0.07 seems to be a good approximation, but, using the differential viscous equations (*dvm*) for a low Reynolds number at the same time, the Cp depends on the grid used.

Figures. 4.12 and 4.13 show the normal velocity contours and axial velocity vectors at the section Cs2. These results display the sensitivity of the flow to the grid quality. For this grid, the most important parameter to take care of is the intensity of the swirl.

Figures. 4.14 and 4.15 show the contours of horizontal velocity using the different options of the RNG model at section Cs3. The grid is an important parameter only when the swirl intensity is modified, and it defines the quality of the flux obtained in this section of the draft tube.

	$y^+ = 1$			$y^+ = 50$		
TEST	Cp_m	ζ	Cp_w	Cp_m	ζ	Cp_w
rng	0.8765	0.1782	0.9680	0.8714	0.1760	0.9807
dvm	0.8982	0.1542	1.0022	0.8786	0.1705	0.9874
dvmsdf	0.9142	0.1462	1.0172	0.8267	0.2179	0.9344
sdf007	0.8793	0.1753	0.9701	0.8915	0.1603	0.9990
sdf07	0.9061	0.1550	0.9961	0.8923	0.1576	0.9985
sdf7	0.7968	0.2287	0.8818	0.7435	0.2394	0.8459

Table 4.2 Global performance quantities for RNG model options.

4.5 Near-wall treatments.

Traditionally, there are two approaches for modeling the near-wall region. In the first approach, formulas called "wall functions" are used to bridge the viscosity-affected region between the wall and the fully turbulent region. In the second formulation, called "near-wall modeling", the turbulence models are modified to enable the viscosity-affected

(a) Upper centreline wall (b) Lower centreline wall

Figure 4.10 Pressure coefficients along the centerline of the upper and lower walls for the RNG model options using the $y^+ = 1$ grid.

(a) Upper centreline wall (b) Lower centreline wall

Figure 4.11 Pressure coefficients along the centerline of the upper and lower walls for the RNG model options using the $y^+ = 50$ grid.

Figure 4.12 Axial velocity contours and horizontal velocity vectors, caused by RNG model options, at Cs2 section for $y^+ = 1$ grid.

Figure 4.13 Axial velocity contours and horizontal velocity vectors, caused by RNG model options, at Cs2 section for $y^+ = 50$ grid.

Figure 4.14 Horizontal velocity contours, caused by RNG model options, at Cs3 section for the $y^+ = 1$ grid.

Figure 4.15 Horizontal velocity contours, caused by RNG model options, at the Cs3 section for the $y^+ = 50$ grid.

region to be resolved with a fine mesh all the way to the wall, including the viscous sublayer.

4.5.1 Near-wall modeling.

FLUENT offers two choices for the wall function approaches:

Standard wall function. The ideal conditions that has been considered in their derivation are a constant shear and a local equilibrium hypothesis which assume that the production of k and its dissipation rate are equal in the wall-adjacent volume.

Non-equilibrium wall function. The key elements here are that the log-law for mean velocity is sensitive to pressure-gradient effect. Besides, a two-layer concept is adopted to compute the amount of turbulence kinetic energy in the wall-neighboring cells.

The near wall modeling method combines a two-layer method with enhanced wall functions. In this model, the viscosity-affected near-wall region is completely resolved all the way to the viscous sublayer. In the two-layer approach, the whole domain is subdivided into a viscosity-affected region and a fully-turbulent region and it is used to specify both ϵ and the turbulent viscosity in the near-wall cells. The enhanced wall functions methods can extend its applicability throughout the near wall region by blending linear (laminar) and logarithmic (turbulent) laws-of-the-wall. This consideration allows the turbulent law to be extended to take into account other effects such as pressure gradients or variable properties.

4.5.2 Results of the near wall treatment.

Table 4.3 shows the values of the engineering quantities for the wall treatment functions. In this section, the FOU scheme was used to discretisize the equations governing fluid flow. Figures 4.16 and 4.17 show the pressure recovery factor along the centerline wall using options like (*swf*) standard wall functions, (*ewf*) enhanced wall function, (*newf*) non equilibrium wall function and (*ewf pge*) enhanced wall function with pressure gradient effects for the two grids. This quantitative evaluation shows the highly significant sensitivity of Cp using the $y^+ = 1$ grid. The (*newf*) option is shown to be the most inexact approximation of the experimental results for the three turbulence models. The use of the pressure gradient effects option (*pge*) has only a slight effect for the RNG model. The standard wall function option (*swf*) seems to be the closest approximation of the experimental results. By contrast, these options do not display any effect for the pressure recovery factor using the $y^+ = 50$ grid. Figures 4.18 and 4.19 display the axial velocity contours and horizontal velocity vectors at section Cs2. These figures confirm that the *newf* option produces the most important changes in the flow quality for each turbulence model. For the $y^+ = 50$ grid, the wall function treatment does not have any effect in terms of representing the quality of flow. Figure 4.20 for the $y^+ = 1$ grid and Fig. 4.21 for the $y^+ = 50$ grid present the results for the horizontal flow at section Cs3.

(a) Upper centreline wall (b) Lower centreline wall

Figure 4.16 Pressure coefficients along the centerline of the upper and lower walls for different wall treatment functions using the $y^+ = 1$ grid.

(a) Upper centreline wall (b) Lower centreline wall

Figure 4.17 Pressure coefficients along the centerline of the upper and lower walls for different wall treatment functions using the $y^+ = 50$ grid.

Figure 4.18 Axial velocity contours and horizontal velocity vectors, caused by the wall treatment functions, at Cs2 section for the $y^+ = 1$ grid.

Figure 4.19 Axial velocity contours and horizontal velocity vectors, caused by the wall treatment functions, at Cs2 section for the $y^+ = 50$ grid.

TEST	$y^+ = 1$			$y^+ = 50$		
	Cp_m	ζ	Cp_w	Cp_m	ζ	Cp_w
std swf	0.8724	0.1798	0.9663	0.8852	0.1623	0.9956
std ewf	0.8871	0.1644	0.9984	0.8876	0.1620	0.9974
std ewfpge	0.8871	0.1644	0.9985	0.8893	0.1595	0.9993
std newf	0.8524	0.1959	0.9214	0.8876	0.1607	1.0000
rng swf	0.8765	0.1782	0.9680	0.8714	0.1760	0.9807
rng ewf	0.9102	0.1511	1.0169	0.8753	0.1726	0.9830
rng ewfpge	0.9071	0.1454	1.0136	0.8839	0.1664	0.9912
rng newf	0.8580	0.1994	0.9249	0.8839	0.1664	0.9912
rea swf	0.8554	0.1786	0.9551	0.8661	0.1646	0.9787
rea ewf	0.8683	0.1670	0.9824	0.8717	0.1622	0.9841
rea ewfpge	0.8701	0.1668	0.9841	0.8717	0.1622	0.9841
rea newf	0.8580	0.1994	0.9250	0.8707	0.1627	0.9859

Table 4.3 Global performance quantities for wall treatment functions.

Figure 4.20 Horizontal velocity contours, caused by the wall treatment functions, at the Cs3 section for the $y^+ = 1$ grid.

Figure 4.21 Horizontal velocity contours, caused by the wall treatment functions, at the Cs3 section for the $y^+ = 50$ grid.

(a) Energy loss coefficient

(b) Wall pressure recovery factor

(c) Mean pressure recovery factor

Figure 4.22 Comparison of grid quality via Standard Deviation (SD) of the global performance quantities.

4.6 Convergence criteria.

The convergence criteria for the simulations were controlled by the iterative convergence error, by the residual factor and by the flux imbalance. The iterative convergence error is defined as:

$$e_i = \frac{\text{MSR}}{\text{TMF}} \tag{4.18}$$

where MSR is the mass source residual (the residual for the pressure correction equation) and TMF is the total mass flux into the domain. The mean value of $e(i)$ for all simulations carried out using the $y^+ = 1$ grid was estimated to be $1.07^{-05}\%$ and $1.31^{-04}\%$ for simulations using the $y^+ = 50$ grid. The residual reduction factor (RRF) was computed as a function of the mass source residual obtained at the second and at the last iteration. Figure 4.23 presents the value reached for each simulation. Generally, better convergence is achieved using the $y^+ = 1$ grid, as measured through this factor, than using the $y^+ = 50$ grid.

The results obtained using the RNG turbulence model only arrived at a residual factor of 1.0^5 with great difficulty, the convergence criterion established by Cervantes & Engström (2004). Also, to avoid false convergence, the flux imbalance was monitored, and this value needed to remain stable to at least four digits during several iterations. The RNG turbulence model always presented large oscillations and the highest imbalance values. The results of the numerical simulation carried out with the SOU scheme and the $y^+ = 50$ grid for the three $k - \epsilon$ turbulence models was selected for comparison with the organizers' experimental data, (Cervantes & Gustavsson, 2005). These results were chosen over similar results obtained using the $y^+ = 1$ grid, because, even though they present a lower residual factor, their approximation to the experimental Cp_m is better. In this way, it will only be necessary to check the relation between the poor convergence obtained using the RNG model and the approach to the experimental results.

The solver CPU time is another important issue to review, mainly if we want to choose a model for iterative simulations. Figure 4.24 shows the time in hours used by the solver for 500 iterations using the various numerical schemes. This result confirms us that the standard model is computationally the least expensive and the grid quality does not seem to have any influence on this condition.

Thus, by economical, robust and most specially because in an optimization process the

objective function must be evaluates quantitatively, the standard $k - \epsilon$ turbulence model will be used in the optimization process.

Figure 4.23 Residual reduction factor (RRF) of the second iteration and the last iteration for the different runs using the $y^+ = 1$ and $y^+ = 50$ grids.

4.7 Summary.

This chapter presented a numerical study of the flow through the Turbine 99 model draft tube using several variations of the $k - \epsilon$ turbulence model. The assessment was carried

Figure 4.24 Solver CPU time in hours used to compute 500 iterations for the different runs using the $y^+ = 1$ and $y^+ = 50$ grids.

out in the *operational mode T*. It was observed that the discretization schemes of the equations governing the flow through the draft tube are not greatly influenced by the standard and realizable $k-\epsilon$ turbulence models. Nevertheless, the RNG model is affected by these schemes. This can be confirmed by the results shown in Fig. 4.22, which summarizes the sensitivity of the flow to the three turbulence $k-\epsilon$ models using the standard deviation statistical parameter of the engineering quantities. The qualitative and quantitative results show that the near-wall grid is an important parameter for representing the flow characteristics in sections Cs2 and Cs3. Using the RNG options, the swirl intensity is the most important factor impacting the numerical results. In our case, the inlet flow, which has a moderate swirl intensity, seems to give the best approximation of the experimental data. In this model, the flow quality at different sections is influenced by the near-wall grid $y^+ = 1$.

Also, it has been observed that the wall treatment options do not have any influence on the flow using the $y^+ = 50$ grid. Flow visualization of the flow field simulations has shown that the RNG model would be able to more exactly represent the principal details of the draft tube flow, but a deeper sensitivity study of the intensity swirl should be conducted. However, if a large number of flow characteristic details are not required, for instance during an optimization process where the numerical convergence of the residuals is not crucial, this is not the most highly recommended model. So, the standard model is again confirmed to be a reliable and robust turbulence model that can be used to study the effects of a swirling flow through the turbine draft tube.

In this way, having validated and verified the draft tube numerical model, it remains to evaluate the numerical algorithm before starting the optimization processes. The numerical optimization algorithm will be applied to two standard problems previously validated which are appropriate for testing different GA implementations. This will be the subject of the next chapter.

CHAPTER 5

OPTIMIZATION PROCESS.

The inlet velocity profile optimization process must allow to explore the design space with the objective of minimizing the energy losses in the draft tube. As aforesaid, the optimization approach will be based on a more efficient GA technique called distributed GA (DGA). This method has been chosen because it is effective to seek an optimum solution in a wide design space. However, the optimization based on GA needs a large number of evaluations being better-suited for a parallel computing system. This is an important issue because in the optimization process, the CFD simulations consume excessive computing time. Thus, a distributed processing system will allow to execute tasks on remote machines, that is other than the host on which iSIGHT (the optimization manager) session is running and executing the designs at the same time (i.e. parallel).

However, one disadvantage of DGA is that there exist several parameters which affect both the accuracy of the solution and the calculation time. In order to test the DGA implementations, in the following, two standard optimizations problems are solved. The test functions vary in difficulty, in number of local minima and in number of design variables. They have a local extrema that is hidden among many local extrema, therefore they are appropriate for testing different parameters of the DGA provided by iSIGHT, such as the Multi Island Genetic Algorithm (MIGA). Finally, as mentioned before, to establish the link between the CFD analysis and the MIGA, an objective function is required. For this reason a parametric study will be carried out to identify the most representative global performance quantity which will be used as objective function.

5.1 Optimization set-up.

Efficiency of MIGA depends largely on its parameter values, for example; ways of selecting the next generation, ways to crossover, rates of mutation in genes and so on. In general it is difficult to adjust manually or to choose a proper combination of those parameters.

Basic Parameters Tuning. There are three major parameters which control the optimization process. These are the size of sub-population (fixed to a default value of 10), the number of islands (also fixed to a default value of 10) and the number of generations (100). The values of the general parameters used for this optimization process are summed up in table 5.1.

Size of subpopulation	Number of islands	Number of generations
10	3	100

Table 5.1 Basic tuning MIGA parameters

The size of sub-population is the size of each island population. The size population depends on the number of island (3) and size of sub-population (10). In this case it will be 30. the number of evaluations or designs depends on the number of generations that the algorithm evaluates. Thus if we set-up 100 generations, the total number of designs is $10 \times 3 \times 100 = 3000$.

Advanced Parameters Tuning. Table 5.2 presents the advanced parameters tuning for the MIGA optimization technique.

The gene size parameter controls the number of bits N in all genes used for encoding the value of each inlet velocity profile variable. As every bit of the gene can switch its value between 0 and 1 the total number of possible combinations in every gene is then $2^{16} = 65536$ for $N = 16$. And the minimum change in the value of any design variable

Gene size	16
Rate of crossover	1.0
Rate of mutation	0.02
Rate of migration	0.3
Interval of migration	3
Elite size	1
Rel. Tournament size	0.5

Table 5.2 Advanced tuning MIGA parameters

during the genetic operation will be given by:

$$x = \frac{b_j - a_j}{2^{N_j}} \tag{5.1}$$

Where N is the gene size and (b_j, a_j) is the domain of variable j.

In the optimization process, only the relative precision of the unknowns is important. For this reason, to establish a correct gene size parameter will allow a very large space to be searched without the need for excessive string lengths.

New designs are created by the genetic crossover operation and the rate of crossover, which is a fraction in each generation that will be subjected to this operation, was maintained to its default value. By contrast, the probability of change of each gene (0 and 1) of every individual, which increases the variability of the population and avoid stagnation in the evolution process, was changed from 0.01 to 0.02.

The operation called migration is used to transfer some individuals from an island to another one periodically. This operation is controlled by two parameters which are 1) the rate of migration and 2) the interval of migration. The latter value is the number

of generations between each migration and the former is the percentage of individuals transferred from an island to another one at the time of migration. The iSIGHT default values for these parameters are 5 and 0.5 respectively, but they were changed to 30 % of the individuals contained in a sub-population every 3 generations.

MIGA allows to preserve by means of elitism the best individuals from the previous generation without alteration and carry over the child generation. The number of best individuals was set by default value to 1. In the tournament selection, the best individuals are selected to be duplicated for the child generation. Reducing the relative tournament size will increase the randomness in the selection process. By increasing it, more duplicates of the best individuals in the child population will result. The default value of 0.5 was maintained.

5.2 Parallel DGA.

DGA's, (Distributed GA's) are GA's with multiple subpopulations and a migration procedure which produces a chromosome exchange between sub-populations. Its principal role is to promote genetic diversity, and in essence, to allow the sharing of possible solutions. DGA's are suitable for parallel processing since each subpopulation can be assigned to a processor of a parallel computer, and inter-processor communication occurs only at migration time. There are two main reasons for parallelizing a DGA: one is to achieve time savings by distributing the computational effort, and the second is to preserve the diversity due to the semi-isolation of sub-populations, which may help prevent premature convergence.

iSIGHT offers a way to distribute the execution of tasks and/or semicodes to remote machines different than the host machine on which the iSIGHT session is running. This capability, known as distributed processing, does not necessarily mean that the tasks/semicodes

will execute simultaneously (i.e. in parallel) they are simply executing on different machines.

Parallel execution is a separate but related issue. The optimization process can sometimes be very expensive due to the time associated with executing the CFD code. Therefore, the process cost is multiplied many times due to the number of system analyzes that are often required. Using MIGA, a number of data points can actually be executed simultaneously, since they are independent. In this research, the computational resources are available and the capabilities exist to coordinate the execution of multiple design points simultaneously (i.e. in parallel) to get great time and cost savings.

The GRMIAO (Groupe de Recherche en Mathématiques de l'Ingénierie Assistée par Ordinateur) has a calculating network with two master nodes which permit to have access to 14 calculating nodes. Each node has two processors with 2.0GHz and 8Go of RAM memory. The data exchange between the CFD process (design) and the iSIGHT host is shown in Fig 5.1. The number of parallel designs was limited by the number of FLUENT licences available.

Figure 5.1 Data communication between parallel MIGA and the calculating network

5.3 Optimization scheme validation.

Efficiency of DGA depends largely on the parameter values used to control operations such as mutation and crossover. In general, it is difficult to adjust these parameters manually, since this require extremely large computation cost. In order to test the GA implementations, two standard problems were chosen from the literature, (Mengistu & Ghaly, 2003). The tests include the Rosenbrock banana function and the Rastring function. The test functions vary in difficulty, in number of local minima, and number of design variables. These functions are appropriate for testing different implementations of a global search optimizer such a DGA.

5.3.1 The Rosenbrock function

The Rosenbrock function is expressed by Eq. 5.2 and its landscape is shown in Fig. 5.2:

$$f(x) = \sum_{i=2}^{n} \left(100(x_i^2 - x_{i+1})^2 + (x_i - 1)^2\right) \tag{5.2}$$

Rosenbrock's valley is a classic optimization problem, also known as the Banana function. The global optimum is inside a long, narrow, parabolic shaped flat valley. To find the valley is trivial, however convergence to the global optimum is difficult and hence this problem has been repeatedly used to assess the performance of optimization algorithms. The convergence to the global optimum is difficult and is a uni-modal function which has a strong dependency between its variables. For this reason it is a suitable test to assess the performance of the GA.

The function has a global minimum value of zero at the point $x_i = 1$. Different ranges of the design variable can be selected. As establish by Mengistu & Ghaly (2003), the range

Figure 5.2 Full definition range of the Rosenbrock function

was defined as $x_i = 10.12$. Figure 5.3 shows the convergence history for the problem of minimizing the Rosenbrock function with $n = 2$ and $n = 10$ variables.

Figure 5.3 Convergence history for the Rosenbrock test function with $n = 10$ design variables. MR= Migration rate, P = Population size, MI= Migration interval, DF= default values.

Fig. 5.3 shows the minimum fitness values found by the MIGA respect to the migration rate, migration interval and population size. According to Hiroyasu et al. (1999) and Miki et al. (1999), these parameters have an important influence on the search ability of a MIGA. It is clear from these data that the algorithm can not find an optimal value for this function, even though there is only a peak value. This result is not unexpected, since there is a relationship among the design variables in this function. The minimum value is shown with respect to migration rate of $MR = 0.3$ for $n = 10$ and for $n = 2$ the computation time is reduced dramatically. When the population size is too small, as in our case, solutions are dependent on the migration rate and migration interval. It is difficult to find the optimal migration rate and migration interval in these systems.

Mengistu & Ghaly (2003) show that the GA had difficulty in reducing the convergence history for the Rosenbrock test function with 40 design variables. It is noted here that the test function achieved a value of 10^1 for 10^6 evaluations. This is because the GA is inherently not good for this function.

5.3.2 The Rastrigin function.

The Rastrigin's function is given by the following expression:

$$f(x) = 10n + \sum_{i=2}^{n} \left((x_i^2 - 10cos(2\pi x_i)) \right) \tag{5.3}$$

Rastrigin's function, illustrated in Fig. 5.4, is based on a function with the addition of cosine modulation to produce many local minima. Thus, the test function is highly multi-modal i.e. with several local extrema. However, the location of the minima are regularly distributed. The function global minimum value is 0 and it occurs at $x_i = 0$, i.e. the global minimum is $f(0) = 0$. In the optimization search all the $x_i's$ are defined on the

range ± 5.12 which gives 11 local optima as each integer corresponds to a local minimum of the Rastrigin function.

Figure 5.4 Full definition range of the Rastrigin function

Fig. 5.5 gives the optimization history for the MIGA using the parameters establish in table 5.2. In this case for $n = 2$ and $n = 10$ the GA was able to find the global minimum accurate to 10^{-5} with 15000 and 10^{-4} with $40,000$ iterations respectively. Mengistu & Ghaly (2003) reported results for the Rastrigin function with 30 variables where the best function value 10^{-6} was achieved in 88,100 evaluations and a function value less than 10^{-14} was achieved with 278,800 function evaluations using 10 variables.

5.4 Definition of the objective function.

As it is known, in order to formulate the numerical optimization problem, we must define an objective function and a set of constraint functions (if it exists). An essential component of the optimization algorithm is the sensitivity analysis of the objective and constraint functions with respect to the design parameters. Thus as a study of the objective function, we will evaluate the response of the draft tube performance indexes: the

Figure 5.5 Fitness value history of the Rastrigin function at each generation with $n = 2$ and $n = 10$ variables.

mean pressure recovery, Eq. 2.10, the wall pressure recovery, Eq. 2.9 and the loss energy coefficient, Eq. 2.8, to modification of the inlet velocity profile design parameters given in the Eqs.3.11, 3.12 and 3.13.

5.4.1 A parametric study of the objective function.

In the context of optimization, sensitivity analysis could be used for problems exhibiting a small number of parameters to compute gradients. Generally speaking, sensitivities can be grouped into two categories, shape and value sensitivities. In the first, we seek the dependence of flow variables on the shape parameters. In the second, the parameters may describe boundary conditions, fluids properties and functions. In our case, the velocity profile is represented by parametric equations and the selection of these parameters may influence in a very important manner the flow performance through the draft tube. This

situation requires a systematic approach in order to identify the dominant parameters which most affect the inlet draft tube flow.

Fractional Factorial Design (FFD) is a statistical method for experimental design where a set of experimental runs is determined by varying the factors in a certain pattern and has been applied to CFD by Cervantes & Engström (2004). The results obtained using this method allow to estimate the influence of one factor (main effect) and the interaction of different factors (join effect) on quantities representative of the process. However, this procedure sacrifices high order interactions among factors to economize experimental runs. This is based on the main effects postulate which states that single-factor effects tend to dominate two-factor interactions, and these in turn dominate three-factor interactions. In our case, a fractional factorial design at two levels was performed, i.e. 2^{k-p} runs, where k are the variables to be studied and p the variables to be introduced using interactions from base designs. Nevertheless, a significance parameter level for an effect is a specific challenge in the application of FFD to CFD.

The present design involves $2^{8-4} = 16$ runs and it was conducted to determine the most important swirl parameters in the optimization process, within reasonable computational time. The application of factorial design to CFD differs from its application to experiments, since repeated simulations yield the same result each time, assuming the same residual is reached. Therefore, a standard error analysis based on replicated runs is not possible in CFD. Then, the quantity under evaluation contains an iterative error, if the simulation is not fully converged. The convergence criteria was achieved with a mean iterative convergence error e_i, given by Eq. 4.18, of $3.4406^{-7}\%$, where the mean mass source residual (MSR) was established by $1.7728^{-4}\%$. Due to the lack of reference values to establish the higher and lower levels, the variation of each swirl parameter was limited by the discharge coefficient $\pm 10\%$.

To perform computations in the Turbine 99 draft tube, the computational model and the

inlet boundary conditions were set as specified in chapter 4. FLUENT has been used to represent the governing equations using a cell-centered finite volume scheme. The flow in the draft tube is assumed to be turbulent, stationary and incompressible. The standard $k-\epsilon$ model has shown to be a robust model with adequate approximating capacity of the experimental results in this draft tube as was previously tested in chapter 4, when treating the transport momentum equations with a second order upwind scheme. A converged solution obtained using the first order upwind scheme and original inlet flow profile is used as initial solution for all simulations. Discretization of the pressure used the standard scheme and the standard wall-function condition has been imposed on all walls.

The high and low levels of these parameters are represented by a $(+)$ and $(-)$ respectively in Table 5.3. Table 5.4 shows the variation of the global performance quantities (mean and wall pressure recovery factor and loss energy factor) as the swirl parameters were modified for each run. The inlet velocity profile was set using the Eqs. 3.11, 3.12 and 3.13. Table 5.5 presents the results, in percent of the original value, of the main effects of the inlet swirl parameters on the global performance quantities. The pressure recovery based on the mean value at the inlet and outlet sections Cp_m seems to be the less sensitive to the inlet parameters variation whereas the pressure recovery the based on the wall pressure Cp_w is impacted locally by the second vortex parameters, as R_2 and Ω_2. ζ may be relevant when evaluating the performance of the draft tube since it shows to be the most sensitive to the variations of the inlet swirl parameters.

5.5 Summary.

In this chapter we have tested the performance of the MIGA via two types of test functions which require a good local and global research from the algorithm. Although, the selection of the adequate model depends on the problem particularities, it can be said that

Simulation	U_0	U_1	R_1	U_2	R_2	Ω_0	Ω_1	Ω_2
ana01	-	-	-	-	-	-	-	-
ana02	+	-	-	-	-	+	+	+
ana03	-	+	-	-	+	-	+	+
ana04	+	+	-	-	+	+	-	-
ana05	-	-	+	-	+	+	+	-
ana06	+	-	+	-	+	-	-	+
ana07	-	+	+	-	-	+	-	+
ana08	+	+	+	-	-	-	+	-
ana09	-	-	-	+	+	+	-	+
ana10	+	-	-	+	+	-	+	-
ana11	-	+	-	+	-	+	+	-
ana12	+	+	-	+	-	-	-	+
ana13	-	-	+	+	-	-	+	+
ana14	+	-	+	+	-	+	-	-
ana15	-	+	+	+	+	-	-	-
ana16	+	+	+	+	+	+	+	+

Table 5.3 Fractional factorial design on Turbine 99 at two levels of each design parameter.

Simulation	C_{pm}	Cpw	ζ
exp	0.859	0.7686	0.1867
ana00	0.8459	0.7072	0.204
ana01	0.8466	0.7006	0.2025
ana02	0.8477	0.7046	0.1951
ana03	0.8417	0.721	0.188
ana04	0.8476	0.7114	0.1936
ana05	0.8375	0.6761	0.2307
ana06	0.8372	0.7274	0.1807
ana07	0.8496	0.6851	0.2258
ana08	0.8442	0.7029	0.208
ana09	0.8447	0.7089	0.2008
ana10	0.8394	0.7204	0.186
ana11	0.8442	0.6688	0.2336
ana12	0.8459	0.7245	0.1835
ana13	0.8419	0.6989	0.2154
ana14	0.8405	0.6832	0.2211
ana15	0.8437	0.7076	0.2134
ana16	0.8427	0.711	0.2053

Table 5.4 Global performance quantities obtained for run 1 to 16.

Eng. Quant.	CP_m	CP_w	ζ
U_0	0.03	1.05	4.89
U_1	0.18	0.11	0.63
R_1	0.15	0.60	1.30
U_2	0.07	0.05	1.13
R_2	0.19	1.02	0.16
Ω_0	0.10	1.36	3.32
Ω_1	0.12	0.40	3.00
Ω_2	0.06	0.98	0.14

Table 5.5 Main effects of the inlet swirling factors.

the effectiveness of the algorithm can be described by these two types of test functions.

In order to develop this work, a parallelization model was implemented on a network computation system, where the population was divided into a small sub-populations that are assigned to different processors. This parallel execution has been capable to speed up the optimization process significatively and the global optimum has been found with a high accuracy even if there exist many similar sub-optima. Improvements on the convergence have been obtained through the tuning of the migration parameters and its ability to preserve the population density avoiding premature convergence. It was a way to try to gain the best solution under a given population size and uncertainty of the appropriate crossover and mutation rates. Thus, the fitness level achieved for both test functions ensures that the algorithm is able to find the global minimum with a reliable accuracy.

Also, the sensibility of the engineering quantities to the inlet velocity parameters was studied, in order to define the objective function of this optimization process. The results indicate that the energy loss factor is more sensible than the mean pressure recovery factor, which is customarily used as measure of performance on the draft tube. The wall pressure recovery factor is more sensitive to local variations and is not recommended as an objective function in the optimization process. The major challenge has been to choose the effective swirl parameter levels to generate different shapes of inlet velocity profiles. As a result of this, the optimization algorithm is expected to be capable of conveniently manage the inlet swirl parameters to generate a wide range of velocity profile shapes. Thus, a deeper study of the interaction effects on the flow performance in the draft tube will be conducted after the optimization procedure.

Having tested separately the three essential steps in a optimization process; inlet velocity profile parametrization, draft tube numerical model, and the algorithm implementation, the next chapter will be devoted to test the optimization process which will be applied to a straight diffuser.

CHAPTER 6

STEADY SWIRLING FLOW IN A DIFFUSER

This chapter presents a numerical validation and the assessment of the inlet velocity profile optimization process for a swirling flow in a cone. The model geometry and the initial data have been provided by GE Hydro. First, the numerical model characteristics of the cone are presented. Numerical results are quantitatively and qualitatively compared to the available data. After this step, the optimization methodology is tested using the three vortex mathematical representation to verify whether we can improve the initial velocity profile or confirm that it is nearly optimal. Finally, a discussion concerning the development of a flow structure caused by the inlet swirl parameters is included.

6.1 The Numerical model

The computational domain is based on the geometry used by GE Hydro for experiments on swirling flow through a diffuser. This geometry is shown in Fig. 6.1. The conical diffuser has an area ratio of 3.087, with a half-angle of 12 degrees.

In this work an incompressible fluid with constant properties is considered. The Navier-Stokes equations describing this incompressible, viscous, turbulent and steady flow are solved using the commercial program FLUENT which is based on the finite volume method, and the pressure velocity coupling. The geometry and the computational grid was edited using GAMBIT for boundary specification. A coarse grid with 47,124 cells designed for near wall treatment was used. Experimental and numerical data were available for validation at the inlet and outlet sections.

Figure 6.1 The computational domain of the cone.

At the inlet section the velocity field, three velocity components, and constant turbulent quantities are imposed. The radial distribution of the circumferential component is established using a free vortex approach and the axial component has an uniform radial distribution. This profile was obtained through numerical and laboratory experiments and we do not expect the optimum to be far from this profile. The transport and momentum equations are discretized with a second order upwind difference, after the solution has reached a complete convergence with a first order upwind discretization scheme. Discretization of the pressure is done with a second order scheme. The $k - \epsilon$ turbulence model was employed to model the Reynolds stress terms and to close the governing equations. For the solution of the momentum and continuity equations a segregated algorithm is applied. The governing equation are resolved sequentially. The SIMPLE algorithm is used for the pressure-velocity coupling procedure. At the outlet a constant pressure is used.

The standard logarithmic rough wall-function has been imposed at the walls. The surface roughness is set to 10 μm at all cone walls. The fluid density was set to 998.2 kg/m^3 and the dynamic viscosity 1.003^{-3} m^2/s.

6.1.1 Numerical comparison.

The computational results are compared with numerical data provided by GE Hydro obtained using the CFX ANSYS package. The principal variables of the flow are shown on the meridional section.

(a) Total pressure Pa

(b) Velocity magnitude $m \cdot s^{-1}$

(c) Eddy viscosity $Pa \cdot s$

(d) Turbulent kinetic energy $m^2 \cdot s^2$

Figure 6.2 Comparison of the contours of different magnitudes on meridional sections.

The qualitative results of CFX and FLUENT are displayed in Fig. 6.2. The agreement among both softwares for the total pressure, velocity magnitude and turbulence quantities is reasonable and are essentially the same.

The above qualitative analysis is supported by the quantitative assessment presented in Table 6.1:

Area weighted-average	position	CFX	FLUENT	Lab.
Total pressure Pa	inlet/outlet	54.0649/37.3121	55.8184/ 38.1087	54.0648/40.8131
Velocity magnitude $m \cdot s^{-1}$	inlet/outlet	1.0151/0.2574	1.0162/ 0.2549	0.9850/0.2545
Axial velocity $m \cdot s^{-1}$	inlet/outlet	0.9830/0.2466	0.9807/ 0.2461	-
Turbulent viscosity $(Kg/m)s$	inlet/outlet	0.08823/0.3323	0.0976/ 0.3334	-
Turbulent dissipation m^2/s^3	inlet/outlet	0.0153/0.0001	0.0137/ 0.0001	-
Turbulent kinetic energy J/Kg	inlet/outlet	0.0038/0.0005	0.0038/ 0.0005	-
Mass flow rate Kg/s	inlet/outlet	771.3710/771.3750	772.5716/ 772.5716	771.3745/771.3745
y+ $(-)$	wall	78.9244	67.4416	21.41

Table 6.1 Comparison results between FLUENT and CFX.

And by the draft tube global performance quantities in Table 6.2:

source	Cp_m	Cp_w	ς	Q	S	imb
Lab.	0.9516	–	0.0273	0.7713	0.1463	–
FLUENT	0.9509	0.8401	0.0347	0.772754	0.1560	2.35E-05
CFX	–	–	0.0346	0.772754	–	–

Table 6.2 Comparison of the results.

6.1.2 Modeling of the inlet velocity profile.

The distribution of the circumferential velocity in a real swirling flow is mainly due to the method used to generate the swirl. From a mathematical point of view, it is possible to create models to properly approximate the behavior of the flow field from a Rankine vortex model. For this reason, at the inflow section the velocity distribution is prescribed using a superposition of several vortices, (Susan-Resiga et al., 2006), given by the equations:

for one vortex.

For tangential velocity: forced vortex

$$V_t(r) = \Omega_0 r \tag{6.1}$$

And axial velocity: uniform

$$V_a(r) = U_0 \tag{6.2}$$

for two vortices.

For tangential velocity: forced vortex + Batchelor vortices

$$V_t(r) = \Omega_0 r + \Omega_1 \frac{R_1^2}{r}\left[1 - exp\left(-\frac{r^2}{R_1^2}\right)\right] \tag{6.3}$$

And for the axial velocity: uniform + Faller and Leibovich

$$V_a(r) = U_0 + U_1 exp\left(-\frac{r^2}{R_1^2}\right) \tag{6.4}$$

for three vortices.

For tangential velocity: forced vortex + two Batchelor vortices

$$V_t(r) = \Omega_0 r + \Omega_1 \frac{R_1^2}{r}\left[1 - exp\left(-\frac{r^2}{R_1^2}\right)\right] + \Omega_2 \frac{R_2^2}{r}\left[1 - exp\left(-\frac{r^2}{R_2^2}\right)\right] \tag{6.5}$$

And for the axial velocity: uniform + 2 Faller and Leibovich vortices

$$V_a(r) = U_0 + U_1 exp\left(-\frac{r^2}{R_1^2}\right) + U_2 exp\left(-\frac{r^2}{R_2^2}\right) \tag{6.6}$$

Near-wall velocity profile. The nearest measurement point is at 0.076 inlet diameters from the wall. In order to control the velocity profile in this zone, the velocity there is imposed as:

$$V_{near-wall} = V_{max} \cdot \left[\frac{R_{max} - r}{R_{max} - R_{bl}}\right]^{0.1428} \tag{6.7}$$

Radial velocity profile.

$$V_r(r) = tan(\frac{\theta\, r}{R})\, V_a \tag{6.8}$$

θ being the cone half opening angle.

6.2 Results of optimization.

A free vortex (FV) inlet swirling flow through a 12 degree conical diffuser and a uniform axial velocity is considered to represent the inlet distribution of the velocity, Eq. 6.1 and Eq. 6.2 respectively. It is only necessary to establish the Ω_0 parameter for the circumferential and U_0 parameter for the axial component. As shown in the Fig. 6.3, the inlet swirl parameters presented in Table 6.3 fit the original velocity values. Due to the higher sensitivity to the inlet swirl parameters, previously demonstrated in chapter 5, the minimization of the loss coefficient factor ζ was defined as objective function and there is no

restriction on the inlet flow (QF), i.e. the MIGA should find the best results in an wide range of the normalized inlet flow:

$$FQ = \frac{Q_{num}}{Q_{bep}} \tag{6.9}$$

Where Q_{num} is the flow rate obtained from the inlet swirl parameters, and Q_{bep} is the flow rate at the best efficient point of the turbine.

Figure 6.3 Original velocity profile.

Swirl Parameter	value
R_1	0.0
R_2	0.0
U_0	1.0
U_1	0.0
U_2	0.0
Ω_0	0.3
Ω_1	0.0
Ω_2	0.0

Table 6.3 Inlet swirl original parameters.

6.2.1 Optimization history.

This section presents the behavior of the different global performance quantities during the optimization process using the different vortex equations to represent the inlet velocity profile. Figure 6.4 presents the convergence history of the simulation: the value of the objective function is plotted versus the index of each generated individual. This picture shows the stochastic aspect of an evolutionary algorithm since the objective function varies from one individual to the next. This picture indicates that convergence has been reached since the objective function is no longer decreasing. This figure also exposes the fast convergence that the one vortex equation achieves owing to only two parameters being used. The two and three vortex equations takes more computations, roughly 500 and 1000 respectively, to get a valuable result.

Figure 6.5(a) shows the variation of the objective function with respect to the inlet swirling flow intensity. For the one vortex equation formulation, it appears to achieve the optimal at a value slightly smaller than the one obtained by the original profile which has a swirl intensity of $S = 0.1561$. The two vortex equation seems to achieve the lowest energy loss factor with a higher inlet swirl. But with the three vortex equation, there is a domain around $S = 0.15$ where it can minimize the objective function. This picture shows that with the three vortex equation, the MIGA has carried out a better research of the optimum as a result of the increase in the number of variables. However, in spite of the fact that the entire research space has been explored, the objective function reaches the same minimum value which was obtained with the one system equation.

In Fig. 6.5(b) the behavior of the loss energy factor is presented in terms of the flow rate factor (FQ). The MIGA found a value lesser than that found with the original inlet velocity profile, $\zeta = 0.034$. Along the flow domain, the MIGA has found three possible local minimum flow rates where the energy loss has been minimized. These optimal values are

function of different inlet velocity profile shapes generated by the three vortices equation system.

Figure 6.4 Optimization history of the energy loss factor (ζ) using the three vortex equations.

Figure 6.6(a) shows that the behavior of the Cp_w depends on swirl intensity. As it was shown with the energy loss factor, the optimum inlet swirl intensity seems to be around the original value $S = 0.15$. Figure 6.6(b) shows the wall pressure recovery factor during the optimization process in terms of the flow factor. For the one and two vortex equation, the MIGA found a range between 1.0 and 2.0 of the flow rate where the Cp_w value is higher than the original inlet velocity profile. Using the three vortex equation, it has obtained three optimized concentrations or local minimum.

The behavior of the mean pressure recovery factor results in an undefined pattern using two or three vortex equations, as shown in Fig. 6.7(b) in terms of the inlet flow rate and in Fig. 6.7(a) in terms of the inlet swirling flow.

With these graphics we can conclude that the optimum value reached is always near the original swirl intensity. It also can be seen that, no matter the number of vortices used to model the inlet flow, there is only a swirling value for which the objective function is minimized. Besides, the flow rate seems not to affect the energy loss coefficient, which indicate that a test is needed to study its specific influence on the optimization process. In

(a) Inlet swirl intensity

(b) Factor flow rate

Figure 6.5 Optimization behavior of the energy loss factor (ζ) using the three vortex equations in terms of a) the inlet swirling flow and b) flow rate factor.

(a) Inlet swirling flow intensity.

(b) Factor flow rate.

Figure 6.6 Optimization development of the wall pressure recovery factor for the three inlet velocity equations in terms of a) the inlet swirling flow S and b) the flow rate factor FQ.

the following, we will first address the results obtained when no constraint is applied.

(a) Inlet swirling flow intensity.

(b) Flow rate factor.

Figure 6.7 Optimization progress of the mean pressure recovery factor for the three inlet velocity equations in terms of (a) the inlet swirling flow S and (b) flow rate factor FQ.

6.2.2 Best optimization points without flow constraints.

Table 6.4 displays the inlet velocity parameters found by the MIGA which minimized the objective function ζ. Table 6.5 presents the optimized global performance quantities values which were obtained with the inlet swirl parameters of Table 6.4. The objective function for each of the three vortex equations has achieved a final value (0.0243, 0.0233 and 0.0236) that is smaller than the one obtained by the original inlet velocity profile

(0.0347). The objective function reduction has been established around 29.9% for the one vortex equation and 32.8% for the two and three vortex equation. The wall pressure recover factor was increased in a minor extent of 1.39%. The flow rate at the optimum is $FQ = 0.88$ using the one vortex equation and around $FQ = 1.35$ using the two and three vortex equation respectively. The last column of this table shows the flow imbalance between the inlet and outlet section, which ensures that the simulation has reached a good convergence.

Vortex	R_1	R_2	U_0	U_1	U_2	Ω_0	Ω_1	Ω_2
Original	–	–	1.0000	–	–	0.3000	–	–
1 vortex	–	–	0.8807	–	–	0.1803	–	–
2 vortex	0.4781	–	1.3165	0.1404	–	0.8450	-1.2306	–
3 vortex	0.3186	0.9543	1.3577	0.2113	-0.0312	1.0131	1.4373	-1.1410

Table 6.4 Inlet swirl parameters for each vortex equation at the best optimization point.

Vortex	Cp_m	Cp_w	ζ	FQ	S	imb
Original	0.9509	0.8402	0.0347	1.0000	0.1561	-2.30E-05
1 vortex	0.9467	0.8541	0.0243	0.8807	0.1065	-1.50E-05
2 vortex	0.9729	0.8519	0.0233	1.3490	0.1582	1.00E-05
3 vortex	0.9780	0.8504	0.0236	1.3609	0.1728	9.00E-06

Table 6.5 The global performance quantities obtained for the best optimization points.

The radial distribution of axial, radial and tangential components of velocity are presented in Fig. 6.8, where they are represented by different marks. Here are compared the inlet velocity profile components obtained with the best values during the energy loss factor minimization. Using the one vortex equation, the inlet velocity profile generator MIGA produced a profile of a solid body rotation and uniform axial velocity along the radius. Using two and three vortex equations, the MIGA found U_1 values, shown in Table 6.4, which give a radial distribution with a peak in the core region for the axial component. The tangential component presents a different radial distribution for the two and for the three vortex equations with inverse rotation in the core given by the Ω_1 parameter, but

this distribution presents a slightly different inlet swirl number S. The radial distribution of circumferential velocities can be established by the radius of the core R_1 which has a different value for the two and three vortex equation. The radial velocity component is obtained from the geometrical distribution proposed by Mauri et al. (2002a) which considers the inlet cone angle θ and its radial distribution, which depends on the axial velocity component intensity.

Figure 6.8 Comparison of the optimized inlet velocity profiles obtained from different vortex equations.

6.2.3 Best optimization points with flow constraints (FQ=0.5, FQ=1.0, FQ=1.5).

Aforementioned by (Susan-Resiga et al., 2006), the hydrodynamic field at the runner outlet is a direct outcome of the runner design and the operating point. As a result, the diffuser efficiency will depend on the flow rate and the inlet swirling flow intensity.

The best optimization point (BOP) obtained for each vortex system, in the last section, has been found without any constraint on the flow rate and the swirl intensity. In this section, to test the influence of the flow rate on the diffuser performance, the optimization process has been conducted with a constant inlet flow rate and, consequently, a fixed swirl number intensity. The normalized flow has been fixed to $FQ = 0.5$, $FQ = 1.0$ and $FQ = 1.5$ of the original flow rate. At each flow rate point, the MIGA has minimized the objective

function ζ and the results are presented graphically in Fig. 6.9.

For a flow rate of $FQ = 0.5$ the energy loss factor reaches a minimum value that is similar for catch of the three vortex systems. On the contrary, using the flow rates of $FQ = 1.0$ and $FQ = 1.5$, the optimization process reaches, minimums of the objective function that are different for each vortex equation. This emphasizes the fact that depending of the flow rate, the three vortex system can attain as lightly different energy loss factor. The higher the flow rate, the better the radial velocity distribution should be to get small energy loss factor.

Figure 6.9 Best optimization points (BOP) obtained at different flow rates FQ for each vortex system.

Figure 6.10 presents the comparison of the inlet velocity profiles obtained during the optimization process for each inlet flow rate. The principal characteristic of these profiles is the variation at the core center for the axial velocity. This shape characteristic can only be obtained using a two and three vortex system. The vortex characteristic radius R_1 represents the hypothetical border between viscous core region and free vortex region. The tangential velocity profile also withstands strong shape variations near the core center

which means a direction change of the flow rotation.

(a) One vortex equation (1V)

(b) Two vortex equation (2V)

(c) Three vortex equation (3V)

Figure 6.10 Comparison of the inlet velocity profiles obtained without flow rate constraint (QF) and those obtained with flow rate constraint $FQ = 0.5$, $FQ = 1.0$ and $FQ = 1.5$.

Figure 6.11 shows the best energy loss values obtained during the optimization process

without any flow restriction (QF) for the three vortex systems. They have been achieved for $FQ = 0.88$, $FQ = 1.34$ and $FQ = 1.36$ flow rate factors as is shown in Table 6.5. The inlet velocity profiles obtained, have been extrapolated along the flow rate range for $FQ = 0.5$ to $FQ = 1.5$, while maintaining the shape and swirl intensity. The results obtained using two and three vortex equations show that they are reach very similar values, but the energy loss factor using the one system vortex has as lightly larger slope. The energy loss difference calculated using the same inlet velocity profile shape between $FQ = 0.5$ and $FQ = 1.5$ is only 2.9^{-03} for the one vortex system. And the difference of energy loss factor among the different profile shapes generated by the three vortex system is only 3.6^{-03} . With these values we can consider that the energy loss factor has been minimized, achieving almost the same value no matter what vortex system is used.

Figure 6.11 Extrapolation points obtained from the best optimization points at different flow rates FQ.

Figure 6.12 presents the comparison among the extrapolated values and the best optimization ones along the flow rate range. Using the one vortex equation, Fig. 6.12(a), the results for both processes are almost identical for each inlet flow rate. For the extrapolation values, the same inlet swirl intensity was conserved $S = 0.1065$. For the two vortex equation

Fig. 6.12(b) the optimization values do not match with those obtained by extrapolation in each flow rate. This might be attributed to the optimization process which for each flow rate was not able to reach complete convergence. Nevertheless, the extrapolation values obtained with the same inlet swirl intensity $S = 0.1582$ are lower at the different flow rates. In Figure 6.12(c) presents the results for three vortex system. The results are similar to these obtained for two vortex equations with a swirl number of $S = 0.1728$, except that for the highest flow rates, where both procedures achieve the same values.

These last results demonstrate that the optimization process without any flow rate restriction (QF), seems to be a reliable methodology to minimize the energy loss factor instead of using an optimization process with a fixed normalized flow. Besides, it has been shown that the energy loss factor is only slightly affected by the flow rate. Thus, after obtaining the optimized best value, we can extrapolate the shape profile maintaining the same swirl intensity to any inlet flow rate to get almost the same objective function value. It has been mentioned that the distribution of the circumferential velocity in a real swirling flow is mainly due to the method used to generate the swirl, then the MIGA is able to minimize the energy loss factor up to a similar level independently of the shape profile generator.

6.3 Vortex structure.

The flow configuration is the major subject in the present section. The role of the diffuser is to convert kinetic energy into pressure. Depending upon the desired pressure recovery, velocity reduction or flow uniformity, various diffuser parameters such as shape, area ratio, length and inlet flow conditions may be varied to achieve the same goal. For this reason, computations of inlet velocity profiles in a simplified model were made as a first step towards making reliable numerical simulations of inlet boundary condition effects on draft tube performance.

(a) One vortex equation

(b) Two vortex equation

(c) Three vortex equation

Figure 6.12 Comparison among the best optimization points and its suitable extrapolation value obtained at different flow rate factor FQ.

6.3.1 Velocity and pressure contours.

Early research results have shown that certain types of inlet distortions increase the pressure diffuser recovery coefficient. The inlet swirl component pushes the flow towards the wall by centrifugal force and the wall boundary layer is less likely to separate even if the divergence angle of the diffuser is large. This yields a high pressure recovery. However, excessive amounts of swirl reduce too much the axial velocity near the centerline of the diffuser or induces a reversal flow region which results in a low pressure recovery rate, Fig. 6.6(a) and 6.7(a). It is known also that Reynolds number does not affect the performance of a diffuser significantly, provided that the entry boundary layer is thick, which has been confirmed in Fig. 6.6(b) and Fig. 6.7(b).

The inlet velocity profiles used to study the flow structure in the cone are shown in Fig. 6.8. As expected for an solid body rotation (SBR) and an uniform axial inlet radial distribution, the swirl promotes a more rapid axial velocity decay in the core region, (Fig. 6.13(a) and (d)). Fig. 6.13(b) and (c) display the flow behavior of the axial velocity provoked by the two and three vortex systems. The effect of the inlet swirling flow increases the near wall axial velocity, which tends to retard near-wall flow separation. The radial distribution of the axial component at the inlet has a peak in the core and the circumferential component is a profile with one or two Batchelor vortices. This peak in the axial component avoids the stagnation or even the reversal of velocity in the core of the diffuser.

The turbulence quantity shown in Fig. 6.14 indicates an amplification of the near-wall thickness downstream from the diffuser entrance, followed by a subsequent axial decay. The turbulent kinetic energy in the boundary layer along the diffuser wall becomes stronger due to inlet swirl intensity, and it appears to correlate well with large radial gradients of the mean velocity components, since larger gradients of mean velocity com-

ponents lead to a significant production of turbulent energy k. Further downstream, the turbulent kinetic energy peak decays. The k values are higher using the two or three vortex system, As shown in Figs. 6.14(c) and (d), than those obtained for a (SBR) inlet profile (Fig. 6.14(a) and (b)). Under the combined influence of an adverse axial pressure gradient, swirl driven radial pressure gradient and conservation of overall mass flow, the axial velocity component drops everywhere and the boundary-layer thickness grows at a slower rate.

The secondary flow is indicated as vectors and the variation of the total pressure as full contours on each cross section of the diffuser in Fig. 6.15. A common characteristic of the flow though the diffuser is the loss of near-wall radial and tangential velocity intensity. For an inlet flow generated by two and three vortex systems there exists a step pressure gradient in the radial direction and it is conserved downstream where the pressure near the center is lower than the pressure at the wall (Figs.6.15(b) and (c)). For an SBR inlet flow distribution, at a downstream section, the radius of the solid vortex core increases and the circumferential velocity and the radial pressure gradient is small, consequently, the pressure near the center is not much different from that at the wall (Figs.6.15(a) and (d)). Comparing the radial pressure distribution at the inlet section with that at a downstream section, it is clear that the axial pressure gradient near the center is slightly higher than that near the wall. Thus, the fluid in the core does not have enough energy to flow downstream against the pressure gradient and it is pulled forward by the fluid surrounding it. The effective cross-sectional area of the section is reduced and a higher total pressure loss results. On the contrary, in Figs. 6.15(b) and (c) we observe a favorable total pressure gradient at the core center and the total pressure variation near the wall is small which means a better diffuser performance.

6.3.2 Engineering quantities.

Other engineering quantities of interest are the kinetic energy correction factors, one related to the axial velocity α_{ax}, the other to the swirl velocity α_{tg} and the momentum correction factor β plus the swirl intensity S, as illustrated in Fig. 6.16.

Physically α represents the ratio of the actual kinetic-energy flux, at a given cross section of an internal flow stream, to the minimum kinetic-energy flux which could exist at the particular flow rate. It is given by:

$$\alpha = \frac{kinetic\ energy\ actual\ velocity}{kinetic\ energy\ average\ velocity} \tag{6.10}$$

The effect of non-uniformity in the velocity profile is to make α greater than unity, thus the excess kinetic energy flux increasing as the non uniformity increases. Since diffusion requires a reduction in kinetic energy flux, any increase in the non uniformity of the velocity profile, and hence α, represents a reduction in the amount of diffusion that takes place. Figure, 6.16(a) presents the variations of α_{ax} provoked by the axial non-uniformity. The MIGA was able to find, for the three vortex system, an inlet velocity profile which provoked a reduction of this factor downstream of the middle section. However, the original inlet flow (SBR), provoked an augmentation and not a reduction of this factor, due to insufficient diffusion achieved by the excessive swirl intensity. The amount of kinetic energy of the tangential velocity component is quantified with the coefficient α_{tg}, Fig. 6.16(d) and is equal or larger than zero.

Fig. 6.16(c) presents the momentum correction factor also called momentum coefficient or Bousinessq coefficient which is the momentum of water passing through the diffuser. It is defined as the ratio of momentum of the flow per second based on actual velocity to the momentum of the flow per second based on average velocity across the section. It is

denoted by:

$$\beta = \frac{Momentum\ actual\ velocity}{Momentum\ average\ velocity} \tag{6.11}$$

This applies when velocity across a flow area is not uniform.

The assumption of the velocity being approximately equal to the mean velocity across the diffuser is usually sufficiently accurate and it is presented at the inlet and outlet of the diffuser for the three vortex systems. Nevertheless, in the original inlet profile, the jets impinging on the axial flow disturb the velocity distribution across the diffuser cross section and the value of β is no longer one, or even constant, as the axial velocity itself is varying along the flow direction. Thus, any rise in β would lead to a subsequent pressure drop by shear stress over the entire diffuser.

The swirl intensity along the diffuser is presented in Fig. 6.16(d). The three vortices systems increases its swirl level downstream, inversely the original profile reduces its swirl intensity at the end of the cone. The increase of swirl in this part of the cone will avoid fluid separation from the wall which will occasion a lower energy loss.

6.4 Summary.

This chapter has presented the numerical validation of the optimization process on a conical diffuser. This same methodology will be used during the draft tube optimization process. The numerical approach has been satisfactorily compared to previous numerical and experimental values.

Three different inlet vortex systems have been used, with the aim of finding the impact

Figure 6.13 Contours of the axial velocity component on a meridional plane obtained for the different vortex systems.

Figure 6.14 Contours of the kinetic energy on a meridional plane obtained for the different vortex systems.

Figure 6.15 Absolute pressure behavior as a result of inlet velocity profile on inlet, middle and outlet cross sections.

(a) Axial (b) Tangential

(c) Momentum (d) Swirl intensity

Figure 6.16 Engineering quantities behavior along the diffuser.

of the inlet velocity radial distribution on the diffuser performance. This gave rise to flow configuration along the cone which helped to understand the dominance effect of the inlet vortex systems. To find an optimum inlet swirl level and radial distribution which gives neither flow separation at the diffuser wall nor axial reversal at the center line, seems to be the ideal link to obtain the minimum energy loss coefficient. The radial velocity, which has not been considered in previous studies, has an important influence on the wall flow separation. Besides, the optimum has produced mostly turbulence in the near-wall region rather than in the core region. Also, it should be noted that when the overall uniformity of the flow at cone exit increases (hence α and β decrease) the diffuser achieves an improvement in its performance. To maintain or increase the swirl intensity downstream is an important issue to push the swirling flow to the wall.

Another test carried out was to know the influence of the flow rate and swirl intensity in the course of the optimization process. The optimization was made at three different flow rates. The results showed a little influence of the flow rate on the behavior of the energy loss coefficient. The extrapolation of the inlet velocity characteristics, along the flow rate domain, has permitted us to obtain values similar to those obtained during the optimization process with the flow rate as a constant. Thus, an increase in the Reynolds number does not affect significantly the results, showing the inviscid nature of the instabilities provoked by the inlet velocity profiles.

All these tests ensure that the optimization algorithm parameters, the system equation with three vortices and the optimization process without any flow restriction are the correct elements by which the draft tube optimization process will be completed. Thus, as was pointed out for the optimization cone strategy, the optimization step applied to Turbine-99 draft tube will be the subject of the next chapter.

CHAPTER 7

DRAFT TUBE INLET OPTIMIZATION VELOCITY PROFILE T99.

This chapter will focus on carrying out the optimization method previously applied to a cone to improve the flow field characteristics at the draft tube inlet. The coupling of iSIGHT, FLUENT and MATLAB commercial softwares is the base to determine the best combination of draft tube inlet flow parameters which lead to the highest performance. Thus, an analytical representation for the outlet swirling flow of the Kaplan turbine runner is fitted to experimental data based on a modified version of the (Susan-Resiga et al., 2006) equations to account for the near wall axial velocity increase at the draft tube entrance.

Afterwards, as the 3D Navier-Stokes equations demand a large number of computations, the original model has been substituted by an approximate one with a much lower computational cost. The optimization process is developed using as objective function the energy loss factor without any flow restriction in the research space. Finally, a qualitative and quantitative flow field draft tube analysis is presented using optimized inlet velocity profiles. With the results obtained, it is expected that a properly modeled inlet velocity profile can suppress or mitigate the undesirable draft tube flow characteristics such as secondary flow, irregular evolution and excessive velocity through this important component of the turbine.

7.1 Inlet velocity profile adaptation.

As it was mentioned, the vortex equations proposed by Susan-Resiga et al. (2006) were established to represent a swirling flow produced by a constant pitch Francis runner. They

analyzed the flow at the runner outlet using a mathematical model for steady, non symmetric and inviscid swirling flow. It has been depicted using a superposition of three distinct vortices: a solid body rotation motion, a counter-rotating and co-flowing Batchelor vortex and a co-rotating and co-flowing Batchelor vortex by Eqs. 6.5 and 6.6.

To adapt the three velocity equations to the flow generated by a Kaplan runner, they must be modified to better match their characteristics. In consequence, a near-wall velocity profile and a near-cone hub velocity profile have been added at each curve extremity. The nearest measurement point was established at 0.03 inlet diameter form the wall runner hub and 0.036 inlet diameters from the wall by the Eq. 6.7. Without this near-wall improvement, the inlet velocity profile corresponds to a solenoidal vector field. It should be noted, however, that once that the above corrections are added, a divergence free field is not strictly enforced. Nevertheless, this anomaly is immediately corrected by the simulation on the first elements adjacent to the inlet plane.

Eight parameters have been determined by fitting the experimental data and are shown in Table 7.1 and graphically shown in Fig.7.1.

Swirl Parameter	value
R_1	0.0687
R_2	0.3763
U_0	0.2428
U_1	-0.04035
U_2	-0.04036
Ω_0	0.1076
Ω_1	0.8992
Ω_2	-0.009213

Table 7.1 Inlet swirl original parameters.

7.2 The Numerical model.

The model used to obtain reliable numerical data during the optimization process is given by the Navier-Stokes equations which have been previously tested in chapter 4. This model, however, does not indicate what inlet velocity shape is required to improve the draft tube performance. The search for an optimized velocity profile must therefore be guided either by an experienced designer or by an optimization method.

The choice of a global optimization method, as MIGA, requires a large number of Navier-Stokes computations, to evaluate many different inlet velocity shapes before reaching a good solution satisfying field flow requirements. The design process may be considered converged as soon as an acceptable solution has been found or the maximum iteration number has been reached. Although this procedure allows to have the possibility of finding an efficient profile, it is expensive in terms of computational time due to the evaluation of the individual's fitness, which is the most time consuming component of the optimization. Then, it is necessary to find a way to keep the cost and duration of the design process within reasonable limits. Thus an attempt to reduce the CFD time is to substitute the fine draft tube grid size by a coarse one with a much lower computational cost.

Figure 7.1 Original inlet velocity profiles at best efficient point.

7.2.1 Coarse mesh modeling.

To achieve accuracy in a numerical flow prediction, a model able to estimate the numerical errors is needed. The assessment of the turbulent model was analyzed in chapter 4. In this section, the method for estimating the numerical errors for the coarse mesh model will allow to compute the necessary coarse grid size for a reasonable computation time.

The errors in numerical flow simulations can be divided in two parts, iterative and grid convergence errors. The iterative error is the difference between the current and the exact solution on the same grid. The grid convergence error is the difference between the converged solution on a fixed grid and the exact solution. Most important is that the solution should be as close to grid independence as possible. But in this work, it will be necessary to estimate the lower grid size which presents a reasonable grid error.

The draft tube calculations were performed on a sequence of five different grids each having approximately twice the number of elements compared to the previous. The grids were constructed with 100 blocks to properly resolve the sharp corners and avoid degenerated control volumes caused by the boundary layer mesh.

The iterative convergence error e_i was defined by the Eq. 4.18 and are listed in Table 7.2. In these results, there does not seem to be a connection between the iterative convergence error and the number of grid cells because a coarser grid gives a smaller iterative error. However the solver CPU time is an important parameter to estimate the optimization process cost.

Using the values of the Table 7.3, it is possible to estimate the grid convergence error for each engineering quantity, (Bergström, 1999), by means of the equation:

Grid	N	MSR kg/s	e(i) %	Solver CPU time/1000 iter s	imb kg/s
1	1019844	1.09E-07	2.09E-08	20,694.44	2.58E-05
2	559564	5.78E-08	1.11E-08	11,416.91	3.83E-05
3	252784	6.53E-08	1.25E-08	4,275.46	1.41E-05
4	126524	5.45E-08	1.05E-08	3,573.29	5.65E-06
5	56886	5.57E-08	1.07E-08	1,722.82	4.31E-05

Table 7.2 Iterative convergence parameters for the five different grids using a total draft tube mass flow of $521.06\ kg/s$.

			ζ			Cpw			Cpm		
Grid	N	α	ϕ	e_1	e_2	ϕ	e_1	e_2	ϕ	e_1	e_2
1	1019844	1.00	0.1755	12.16	5.87	1.2526	11.33	5.44	0.8856	0.18	0.08
2	559564	1.22	0.1701	14.85	8.75	1.2172	13.84	8.12	0.8852	0.22	0.12
3	252784	1.59	0.1603	19.79	14.04	1.1817	16.35	10.80	0.8843	0.33	0.23
4	126524	2.00	0.1439	28.00	22.84	1.1577	18.05	12.60	0.8895	0.27	0.36
5	56886	2.62	0.1187	40.61	36.36	1.1243	20.42	15.13	0.8903	0.36	0.45

Table 7.3 Grid error estimated in percent for the five grid size and the scheme order; first e_1 and second e_2.

$$\frac{\phi_{\alpha_{2h}} - \phi_{\alpha_{1h}}}{\phi_{\alpha_{2h}} - \phi_{\alpha_{3h}}} = \frac{\alpha_1^p + \alpha_2^p}{\alpha_2^p + \alpha_3^p} \tag{7.1}$$

Where h is the grid cell size, and α is the grid refinement factor, which is defined by:

$$\alpha = \left(\frac{N_1}{N_2}\right)^{\frac{1}{3}} \tag{7.2}$$

N_1 is the grid size number for the finest grid, and N_2 for the coarser grid. The solution of Eq. 7.1 resulted in $p = 1.4$ for the energy loss factor, $p = 1.68$ for the wall pressure recovery factor, and $p = 3.06$ for the mean pressure recovery factor. These are reasonable values of p which is the order of the method, should be between 1 for a first order discretization scheme and 2 for a second one. Thus, the global variable performance

obtained by the Richardson's extrapolation can be expressed as:

$$\phi_{ext} = \frac{\alpha_2^p \phi_h - \phi_{\alpha_2 h}}{\alpha_2^p - 1} \tag{7.3}$$

The results for both schemes are shown in Table. 7.4.

$\phi_{ext}(\zeta)$		$\phi_{ext}(Cp_w)$		$\phi_{ext}(Cp_m)$	
$(p=1)$	$(p=2)$	$(p=1)$	$(p=2)$	$(p=1)$	$(p=2)$
0.1998	0.1865	1.4127	1.3246	0.8871	0.8862

Table 7.4 Extrapolated value ϕ_{ext} of the engineering quantities using both discretization schemes order; first $p = 1$ and second $p = 2$.

The grid error was estimated using;

$$e_r = \frac{\phi_{ext} - \phi_h}{\phi_{ext}} \tag{7.4}$$

The errors e_1 and e_2 for different grid sizes are shown in Table 7.3 if p is assumed to be 1 and 2. Thus, an expected error for ζ, for example, using the finest grid (1) varies between 5-12 % and using the coarser (5) will be between 36-40 %. Deciding what is an acceptable error depends on the circumstances and what the results will be used for. (Bergström, 2000) considers that a "small error" can be anything from 0.1 % to 30 % error.

A plot of each global performance quantity for the five grid sizes is shown in Fig.7.2. The curves in these figures are obtained by solving for η in:

$$\phi(\alpha) = \phi_{ext} + \eta\alpha^p \tag{7.5}$$

Where:

$$\eta = \frac{\phi_{ext} - \phi_h}{h^p} \tag{7.6}$$

(a) Mean pressure recovery factor

(b) Wall pressure recovery factor

(c) Energy loss factor

Figure 7.2 Plot of the global performance quantities using the Richardson extrapolation method ϕ, with a first scheme order $\phi(1)$ and a second order scheme $\phi(2)$.

Here, for Cp_m and Cp_w only the finest grids are in the asymptotic range of convergence. On the other hand, for ζ the five grids are inside this range. Thus, for this quantity, the

coarse grid (5) seems to be in the size limit which would minimize the CFD computations and allow the implementation of an economically feasible optimization process.

Equations 7.1, 7.3 and 7.6 can be used to estimate, based on ζ extrapolated 7.4, the necessary grid size, for a grid error of e.g 1%. This results in a necessary grid size of 7.95^{06} using $p = 1$ and 1.83^{09} using $p = 2$.

7.2.2 Wall treatment.

The near wall treatment was modeled using the standard wall function for all computations just presented. This function for the mean velocity is known to be valid for $30 < y^+ < 300$. It is not easy to obtain y^+, because the grid refinement can not be done all the way to the wall since the grid point closest to the wall and the wall shear stress must be known in advance to be in the logarithmic region. However, if the near wall grid points are kept at a constant distance from the wall, the Richardson's extrapolation can be used, (Bergström & Gebart, 1999). In FLUENT, the log-law is employed when $y* > 11.225$. The value of the different grids was approximately $y^+ \simeq 20$ and are presented in table 7.5. These quantities are approximately equal for equilibrium turbulent boundary layers.

Grid	α	Average y^+	Average y^*
1	1019844	19.243073	22.21058
2	559564	19.47376	21.98328
3	252784	19.07108	21.59194
4	126524	18.61353	21.27803
5	56886	17.828278	20.39874

Table 7.5 Mean y^+ value for all grids.

In Chapter 4, the near-wall treatment options provided by FLUENT were evaluated. For the same first near-wall node distance $y^+ = 50$ and grid size, the Wall Function Approach

(standard and non-equilibrium) and the Near-Wall Model Approach (enhanced wall function) did not have any effect for representing the global performance quantities values, as is shown in Table 4.3 and the Figs. 4.19 and 4.21. The drawback is that the near wall behavior of real flows sometimes does not conform to the law of the wall which is the basis of the wall functions.

7.2.3 Comparison to experimental data.

Pressure measurements for the T99 draft tube are available at the upper and lower centerline of the walls, Fig. 7.3. At the lower wall centerline, Fig. 7.3(a) shows that both grids 1 and 5 agree well with experiments except for a wall longitude of 0.2, where the flow abruptly changes direction at the lower back corner. Both grids capture practically the same wall pressure behavior. At the upper wall centerline (Fig.7.3(b)), both grids do not agree with the experimental values between for a distance between 0.2 and 0.6, i.e. between the bend and the Cs3. The reason of this disagreement is probably that the grid is not fine enough to resolve the flow separation. However, the pressure at the upper and lower centerline is captured by the finest and the coarser grid in the same way.

Several parameters have been calculated to evaluate the difference of the two grids in predicting the flow through the draft tube. The results are shown graphically in Fig.7.4 and 7.5. There is a slight difference among the results of the two mesh sizes along the draft tube for the axial kinetic energy, tangential kinetic energy, and momentum factor. The swirl intensity is practically the same for both grids. A good agreement is found with the experimental measurement, with the exception of the momentum factor. The wall pressure recovery Fig.7.4(a) and the energy loss factor Fig.7.4(c) are captured in the same way by both grid sizes but there exists a constant difference along the draft tube. The energy loss factor presents a sudden increase at the station C1b, where the contraction

area caused by the hub ends.

7.2.4 Flow visualization.

Figure 7.6 shows the contours of the axial velocity component and the vectors for the horizontal components at Cs2 location for predicted and measured values. The agreement is qualitatively close to the experiments. The low speed area, at the left hand, is captured by both grids but the high speed zone in the upper right corner does not match closely the experiments. The horizontal cross flow component is quite large in the left half of the bottom region and with an opposite sign at the top.

Figure 7.7 shows the horizontal component as contours at the section Cs3. For this component, the agreement between simulation and experimental data is weak. However, it can be seen essentially that the agreement is the same for fine and coarse grids.

(a) Pressure at the lower centerline (b) Pressure at the upper centerline

Figure 7.3 Pressure at the lower and upper centerline for the draft tube with the finest grid (dt1) and with the coarser one (dt5).

(a) Wall pressure recovery factor (b) Mean pressure recovery factor

(c) Energy loss coefficient

Figure 7.4 Global performance quantities evaluated along the daft tube sections for the draft tube with the finest grid (dt1) and with the coarser one (dt5).

(a) Axial kinetic energy correction factor

(b) Tangential kinetic energy correction factor

(c) Momentum factor

(d) Swirl intensity factor

Figure 7.5 Engineering quantities evaluated along the daft tube sections for the draft tube with the finest grid (dt1) and with the coarser one (dt5).

(a) Experimental (b) Grid 5 (dt5)

(c) Grid 1 (dt1)

Figure 7.6 The axial (contours) and horizontal (vectors) velocity components at Cs2 (down stream view).

(a) Experimental (b) Grid 5 (dt5)

(c) Grid 1 (dt1)

Figure 7.7 Horizontal velocity contours at Cs3.

Thus, the simulations show us that the coarser grid can give reliable results at least to carry out the optimization process knowing that the estimated grid error is 30 % but it could be reduced to 5% using the finest grid in order to revalue the global performance quantities and to study the field flow patterns along the draft tube. With this procedure, it is expected to reduce the CFD time and be able to develop a minimum necessary number of computations to achieve the convergence of the process during a realistic time instead of allowing only a predetermined number of generational evaluations.

7.3 Optimization process.

In this section, the results of the optimization process are presented for the inlet velocity profile generated using the equations proposed above for a Kaplan runner. This process is similar to the one which was applied to the cone presented in Chapter 6. The convergence history is presented in Fig.7.8 where the objective function is plotted for the best individual of each generation versus the index of iteration. This graph indicates that the convergence has been reached after $10 \times 3 \times 100 = 3000$ runs, each population has 30 individuals which were evaluated in the course of 100 generations. The optimization process had an initial value of zero for each variable of the vortex equations. As it can be seen in Fig.7.8, the original value of the objective function obtained with the coarser grid (dt5) was $\zeta = 0.11$ and it was minimized approximatively by a factor of 3.6. This figure also shows the variation of the objective function with respect to the inlet swirling flow. The original swirling value was $S = 0.26$ and the objective function is minimized when this value is $S = 0.0129$. The objective function as function of inlet flux reaches its minimum value when the flow rate factor $FQ = 1.613$, which is the ratio of the analytical mass flux to the experimental one.

Table 7.6 shows the parameter values of the best velocity profiles reached during the

optimization process which have as a principal characteristic a different shape. Table 7.7 shows the comparison of the global performance quantities between grid 1 (dt1) et grid 5 (dt5) obtained from the best inlet velocity parameters. Fig. 7.9 shows the run history during the optimization process using the parameters of the inlet velocity profiles. These values were applied to the finest grid, and the results are presented in the same figure. The behavior of both grids is similar, using the same inlet velocity profiles. This test leads us to believe that the finest grid would have had a behavior similar to that of the coarser grid during the optimization process.

7.4 Draft tube flow field analysis.

Figure 7.10 presents the inlet velocity profiles which gives an important gradient of the objective function $\zeta = 0.1252$, $\zeta = 0.0619$ and $\zeta = 0.0029$ for runs 256, 643 and 2966 respectively. The principal characteristic of these profile with respect to the original is the near wall peak. This increase of the axial velocity provokes the flow attachment induced by the radial velocity. The change of direction in the radial distribution of the tangential velocity causes a reduced swirl intensity number.

Figure 7.8 Optimization history of objective function during the optimization process with respect to the swirl intensity and the flow rate factor.

Run	R_1	R_2	U_0	U_1	U_2	Ω_0	Ω_1	Ω_2
original	0.0687	0.3763	0.2428	-0.0403	-0.0403	0.1076	0.8992	-0.0092
256	0.2365	0.9829	0.9745	0.2378	-0.8360	-0.1093	-0.6798	0.3286
281	0.2381	0.9829	0.9745	0.2378	-0.7891	-0.1093	-0.6798	0.3286
319	0.2694	0.9829	0.9745	0.2378	-0.7891	-0.1093	-0.6798	0.3286
558	0.2967	0.9830	0.9744	0.2627	-0.7145	-0.1250	-0.6835	0.1689
643	0.2967	0.9829	0.9665	0.2647	-0.7891	-0.1250	-0.6835	0.4189
768	0.2960	0.9839	0.9743	0.2627	-0.7891	-0.1249	-0.5734	0.3313
1198	0.3045	0.9839	0.9032	0.2687	-0.7108	-0.1266	-0.4888	0.3318
1261	0.3046	0.9839	0.9813	0.2685	-0.6954	-0.1267	-0.0083	0.3318
2052	0.6759	0.2462	0.5800	-0.2438	0.0943	-0.0016	0.2598	-0.9812
2185	0.8205	0.3086	0.5800	-0.2418	0.0709	-0.0299	0.2597	-0.5189
2681	0.8127	0.1752	0.5805	-0.2468	0.0946	-0.0375	0.1260	-0.5071
2966	0.8132	0.1748	0.5805	-0.2487	0.0953	-0.0390	0.1263	-0.4734

Table 7.6 Inlet swirl parameters obtained from the optimization process.

Figure 7.9 Variation of the energy loss factor during the optimization process using coarser and finest grid for mass flow $FQ = free$ and $FQ1 = 1$.

Run	grid	Cp_m	Cp_w	ζ	FQ	S	imb
original	dt5	0.8903	1.1242	0.1186	1.0	0.2600	4.13^{-05}
	dt1	0.8855	1.2526	0.1755	1.0	0.2600	2.58^{-05}
256	dt5	0.8914	1.3888	0.1252	1.8031	0.0694	6.00^{-05}
	dt1	0.9108	1.5815	0.1291	1.8053	0.0694	1.45^{-04}
281	dt5	0.9065	1.3607	0.0887	1.9430	0.0628	3.00^{-05}
	dt1	0.9235	1.5294	0.0968	1.9445	0.0628	1.90^{-04}
319	dt5	0.9090	1.3518	0.0788	1.9656	0.0402	2.00^{-05}
	dt1	0.9240	1.5145	0.0884	1.9676	0.0402	7.50^{-05}
558	dt5	0.9406	1.2934	0.0416	2.2262	-0.1202	2.00^{-06}
	dt1	0.9524	1.4208	0.0566	2.2276	-0.1202	8.00^{-06}
643	dt5	0.9265	1.3460	0.0619	1.9738	0.0759	5.00^{-05}
	dt1	0.9438	1.5053	0.0664	1.9763	0.0759	4.50^{-05}
768	dt5	0.9234	1.3446	0.0550	2.0032	0.0242	1.00^{-05}
	dt1	0.9393	1.5004	0.0609	2.0056	0.0242	9.30^{-05}
1198	dt5	0.9356	1.3269	0.0339	1.9458	0.0343	7.00^{-06}
	dt1	0.9548	1.4807	0.0361	1.9479	0.0343	1.37^{-04}
1261	dt5	0.9429	1.2834	0.0261	2.3223	0.1128	6.00^{-06}
	dt1	0.9522	1.4003	0.0414	2.3234	0.1128	4.80^{-05}
2052	dt5	0.9158	1.2255	0.0208	1.9891	0.0186	2.00^{-06}
	dt1	0.9229	1.3320	0.0561	1.9859	0.0186	4.00^{-06}
2185	dt5	0.9227	1.2275	0.0121	1.8902	0.0497	1.00^{-06}
	dt1	0.9272	1.3347	0.0488	1.8875	0.0497	1.40^{-05}
2681	dt5	0.9288	1.2462	0.0034	1.8664	0.0113	7.00^{-06}
	dt1	0.9346	1.3587	0.0392	1.8632	0.0113	1.80^{-05}
2966	dt5	0.9294	1.2481	0.0029	1.8613	0.0129	9.00^{-05}
	dt1	0.9352	1.3612	0.0385	1.8581	0.0129	3.00^{-06}

Table 7.7 Comparison of the global performance quantities obtained from the best inlet velocity profiles, between grid 5 and grid 1.

Figure 7.11 presents the axial velocity contours at different draft tube cross sections. At the inlet section Cs1a, the principal difference lies in the radial distribution at the end among inlet velocity profiles. At the section Cs1b, the axial velocity magnitude has been reduced, however, in the original profile, there is a large area of back flow downstream of the rotating hub. Inversely, for the optimized profiles this area is progressively reduced. At the downstream sections, the pictures reveals a reduction of the main flow towards the left side, in the original flow, with back-flow at the lower wall. The optimized flow presents a displacement of the main flow on the lower wall, without back-flow. The axial magnitudes are almost the same in spite of the inlet flow rate difference.

For the original and the optimized inlet flows, the streamlines and vectors at several cross sections are shown in Fig. 7.12. The pictures reveals asymmetric structures in the secondary flow, due to the change of rotational direction at the inlet. While the tangential velocity component exists in all the original sections, in the optimized sections this component is reduced up to practically disappearing except for a strong radial component towards the cone wall.

The streamlines in the section Cs2 of the original flow indicates two counter-rotating vortices on each side of the centreline of this section. The left vortex seems to be stronger

Figure 7.10 Inlet velocity profiles to show the flow behavior along the draft tube.

(a) Original (b) 256 (c) 643 (d) 2966

Figure 7.11 Axial velocity contours and secondary flow vectors at the draft tube sections provoked by the optimized inlet velocity profiles.

than that of the right side. For the optimized flows, the counter-rotating vortices are maintained and have a lower intensity and it is finally displaced to the lower wall. This same pattern continues downstream along the next sections Cs3, Cs4a and Cs4b. At the outlet section the secondary flow seems to be more distributed in the best optimized profile while back-flow zones exist in the other outlet sections.

Thus, while for the original flow the inlet swirl forms a vortex that develops on the left side, the optimized inlet profile provokes two symmetrical vortices which disappear at the outlet section. For this reason the energy losses are reduced because lesser flow symmetries provoke dissipative structures which absorb the amount of excess energy available that the more symmetrical flow is not able to absorb.

In Fig. 7.13, total pressure contours are presented for the original and optimized flows at the middle sections, plane XZ for $y = 0$, for the original and optimized draft tube flow. A stronger decrease of the total pressure is obtained by the optimized profiles. The lowest total pressure is observed below the runner's hub, indicating, back-flow mean velocity and low pressure. This region is due to the separation flow below the hub, which creates a very complex vortex structure below the runner hub. The radial and axial velocity gradients near the wall rise the pressure on the wall, avoiding flow separation and consequently low pressure regions beneath the runner hub which decreases considerably in the optimized flow. This allows downstream, an homogeneous pressure distribution because there is no exist an adverse pressure gradient in the elbow zone. As such, the flow generated by the un optimized profiles affect the draft tube function through a blockage effects.

Figure 7.14 shows the streamlines and vectors at the symmetry mid-plane from inlet to outlet. Due to the changes of rotation direction along the inlet radius and the near-wall axial velocity increase for the optimized profile, the main flow moves to the lower wall and consequent bend blockage and back-flow beneath the hub are suppressed. In these figures it can be appreciated that the main velocity uniformity is increased by the optimized flows

(a) Original (b) 256 (c) 643 (d) 2966

Figure 7.12 Streamlines and vectors of the secondary flow on draft tube cross-sections from the inlet to the outlet.

(a) Original

(b) 256

(c) 643

(d) 2966

Figure 7.13 Total pressure contours at the draft tube mid-plane provoked by the optimized inlet velocity profiles.

and that the last diffuser sections are too big for the discharge flow rate since the back-flow at the outlet section has been changed from the lower to the upper wall.

Figure 7.15 presents the draft tube performance, which has been evaluated along the draft tube by considering engineering quantities. Fig. 7.15(a) shows the effect of the non-uniformity in the velocity profile by the axial kinetic energy factor α_{ax}. Since diffusion requires a reduction in kinetic energy flux, any increase in the non-uniformity represents an increase of α_{ax}. The optimized profiles reduce this factor at downstream section Cs4a. The amount of kinetic energy of the tangential velocity component is quantified in Fig. 7.15(b) and represents the augmentation of actual kinetic energy due to swirl where the optimized profiles achieve a reduced number along all the draft tube length.

Figure 7.15(c) presents the momentum correction factor which is the momentum of water passing through the diffuser and it is different from the unity when the velocity across a flow area is not uniform. The optimized flow shows a better approximation to this value at the end of the draft tube.

Figure 7.15(d) presents the swirl intensity along the diffuser. For the optimized flow, this quantity is maintained almost constant throughout the draft tube except at the end, where it undergoes a slight increase. On the contrary, the other flows suffer a drop of the swirl intensity at the end that provokes a wall flow detachment.

Finally, Fig. 7.16 presents the comparisons between the pressure recovery and energy loss obtained through the optimization process. Using the finest grid, the gain obtained reduces the energy loss by a factor of 78% compared with the original flux. The mean pressure recovery factor Fig. 7.16(a) and the wall pressure recovery factor Fig. 7.16(b) do not present the same performance order generated by the optimized flow because the optimization was based on the energy loss factor, but even with the Cp_m, which has shown a poor sensitivity, the increment is significant 5.6%.

(a) Original

(b) 256

(c) 643

(d) 2966

Figure 7.14 Streamlines and vectors at the draft tube symmetry mid-plane provoked by the optimized inlet velocity profiles.

(a) Axial kinetic energy factor

(b) Tangential kinetic energy factor

(c) Momentum factor

(d) Swirling factor

Figure 7.15 Engineering quantities behavior along the draft tube.

(a) Mean pressure recovery factor.

(b) Wall pressure recovery factor.

(c) Energy loss factor

Figure 7.16 Engineering quantities behavior along draft tube.

7.5 Summary.

In this chapter, the numerical optimization of the velocity profiles at the draft tube inlet has been presented. The procedure can optimize and improve the draft tube efficiency of a low head hydroelectric power station. The goal has been the improvement of the runner-draft tube coupling. First, the experimental inlet velocity profile generated by a Kaplan runner has been conveniently approximated using a near wall modified vortex equation. Then, the fine draft tube grid size was replaced by a coarse one which reduces in a considerable way the computational cost. This coarse grid is in the lowest limit of the asymptotic range of grid convergence. The results showed that even if there exists a numerical difference among the global performance and engineering quantities, they have the same tendency and the coarse grid could be used for the optimization process.

Afterwards, the optimization loop used to investigate the inlet velocity profile behavior in the straight diffuser was applied to the Turbine-99 draft tube. This loop allowed to determine the inlet profile for which the minimum energy loss factor was reached. This yielded the conclusion that the use of an optimization algorithm is necessary to automatically generate different inlet velocity profile shapes. The best inlet velocity profiles were used as boundary conditions with the finest grid and the results showed the same tendency as that obtained with the coarse grid was observed.

Finally, the flux performance in the draft tube provoked by the optimized inlet velocity profiles was studied. The results were similar to those obtained with the straight diffuser regarding the importance of the flow uniformity at the end of the conduit. If the momentum parameter can be reduced at the end of the draft tube, its performance is increased considerably. Another important aspect is the reduced blockage area produced by the optimized velocity profile. This blockage promotes an adverse pressure gradient in the cone which, deteriorates the pressure recovery effect. Thus, the drop in energy loss is

explained by a global stability that has the inlet velocity profile as parameter. The topological structure of the velocity field changes with the flow uniformity leading to flow field that suppress the bend blockage.

CONCLUSION

Past numerical and experimental research has shown that the draft tube inlet velocity has a critical importance in hydropower plants. However, there is a lack of information in terms of flow parameters and in particular of swirl distribution. In this respect the optimization of the inlet velocity profile has been a new attempt to control the inlet flow conditions in order to improve the flow through the draft tube and consequently to reach a better performance. To achieve this goal, the optimization task has been guided by the iSIGHT code, combining CFD analysis with an optimization algorithm.

The optimization methodology defined by three steps has been executed: the inlet velocity parametrization, the CFD simulation and the numerical optimization. Each of them has been previously validated to be confident in the results provided by this research.

Equations were used to approximate the experimental data given by a Kaplan runner fixing the swirl parameter values by fitting. It was found that the more vortices are added the more exact is the approximation to the experimental values, mainly near of the wall. However the number of variable design is increased.

Also, the sensitivity of the inlet velocity parameters to the draft tube performance quantities has been studied. The results indicate that the energy loss factor, is more sensitive than the mean pressure recovery factor which is customarily used as measure of performance of the draft tube. The wall pressure recovery factor is more sensitive to local variations and is not recommended as an objective function to the optimization process.

The three elements involved in CFD which have been recognized as critical were tested to validate the numerical model: grid quality, algorithm development and turbulence modeling. The $k - \epsilon$ models were studied because of their short computational times and their

simplicity. If a large number of flow characteristic details are not required, for instance during an optimization process where the numerical convergence of the residuals is not crucial, the standard model confirmed to be a reliable and robust turbulence model that can be used to study the effects of a swirling flow through the turbine draft tube. It has been observed that the wall treatment options do not have any influence on the flow using the $y^+ = 50$ grid but on the $y^+ = 1$, this option influences the results although they are not closer to the experimental ones.

Also the performance of the optimization algorithm (MIGA) has been tested via two types of test functions which demand to the algorithm a good local and global research. Although, the selection of the adequate model and parameters depends on the problem particularities, the effectiveness of the algorithm, at least for these two classical problems was validated.

A parallelization model was implemented on a computational network, where the population was divided into small sub-populations, each one assigned to different processors. This parallel execution has been capable to speed up the optimization process significatively and the global optimum has been found with a high probability even if there exist many similar sub-optima. Improvements on the convergence have been obtained by tuning the migration parameters due to the ability to preserve the population density and avoiding premature convergence. It was a way to try to gain the best solution under a given population size and uncertainty of the appropriate crossover and mutation rates. Thus, the fitness level achieved for both test functions ensures that the algorithm is able to find the global minimum with a reliable accuracy.

The numerical validation of the optimization process was carried out on a conical straight diffuser. It has been used three different inlet vortex systems, with the aim of finding the impact of the inlet velocity radial distribution on the diffuser performance. This gave rise to flow configurations along the cone which helped to understand the dominance effect of

the inlet vortex systems.

To find an optimum inlet swirl level and radial distribution, which gives neither flow separation at the diffuser wall nor axial reversal at the center line, seems to be the ideal link to obtain the minimum energy loss coefficient. The radial velocity, which has not been considered in previous studies, has an important influence on the wall flow separation. Besides, these vortex systems have produced turbulence mostly in the near-wall region rather than in the core region. Also, it should be noted that when the overall uniformity of the flow at cone exit increases (hence α and β decrease) the diffuser achieves an improvement of its performance. To maintain or increase the swirl intensity downstream is an important issue to push the swirling flow to the wall. The results showed a small influence of the flow rate on the behavior of the energy loss coefficient. The extrapolation of the inlet velocity characteristics, along the flow rate domain, has permitted to obtain similar values to those obtained during the optimization process with flow restriction.

The optimization process has been studied with regards to the improvement of the runner-draft tube coupling. The experimental inlet velocity profile generated by a Kaplan runner was approximated using modified near wall vortex equations. Then, the fine draft tube grid size was replaced by a coarse one, which reduced in a considerable way the computational cost. This coarse grid is in the lowest limit of the asymptotic range of grid convergence. The optimization results were similar to those obtained with the original cone concerning the importance of the flow uniformity at the end of the conduit. If the momentum parameter can be reduced at the end of the draft tube, its performance is increased considerably. Another important aspect is the reduced blockage area produced by the optimized velocity profile. This blockage promotes an adverse pressure gradient in the code which deteriorates the pressure recovery effect.

It is considered that this methodology helps to find a profile shape thanks to the evolutionary optimization strategy. However, further research is still needed to obtain a better

solution, taking into account other physical constraints (cavitation, turbulent intensity) in the objective function definition. It can be noticed that with advances in computer hardware, CFD has become an integral part of the turbo-machinery design process. CFD has contributed to cut analysis cost and time scales by reducing the number of required experimental tests. Although, it is a very useful instrument, the success of the final design still depends on the knowledge and intuition of the designer.

Specific contribution.

As mentioned in the objectives of this work, this thesis contributes to the state of the scientific and industrial knowledge in the following ways;

- The first part of the thesis consists of a literature review of important background knowledge for developing the inlet velocity profile optimization applied to a straight diffuser and a draft tube.
- We identify and explain different flow structures provoked by optimized inlet velocity profiles which will minimize the draft tube energy loss.
- We give an optimization method which assembles three different softwares, iSIGHT, MATLAB and FLUENT through UNIX shell programs and find the best inlet velocity profile for a particular draft tube.
- We demonstrate that the draft tube performance can be still further upgraded by modifying the radial distribution of the inlet velocity components.
- We have achieved a reliable optimized inlet velocity profile which can be used as a target during the runner design or blades modifications and achieve a better coupling between runner and draft tube.
- We propose a reduced grid size numerical model evaluation using the Richardson's

extrapolation for accelerating the computational time during the optimization process.

Perspectives.

In this work, it has been demonstrated that the inlet velocity profile optimization using standard numerical tools can be reached in a reasonable computational time. The optimization technique require a reduction of CPU time as a result of the optimization algorithm used. This algorithm has worked as a profile generator which provokes a wide variety of profiles with different shapes. The way to reduce the computational time of the process has been the grid size model reduction via a grid convergence study to find out the coarse grid with a predetermined error. This issue could require an additional optimization grid size step which might reduce the minimal requirements to initialize the optimization process.

Possible future work should be focused on including operational mode constraints such as flow rate or more advanced objective functions such as cavitation during the optimization process. Both issues have proved to have an important influence on the physical development of the draft tube flow. Radial inlet velocity measurements should be available to allow its approximation via a new equation in order to quantify the prediction capability of the velocity assumption presented in this work. Geometric and structural blade considerations should also be taken into account. The turbulence inlet quantities should change as the inlet velocity profile changes. In consequence, these profiles should be parameterized using analytical equations.

Concretely, there are some issues which must be taken into account for future optimization process. However, it can be said that this work, where the CFD and an optimization algorithm have been combined, might contribute to cut analysis cost and time scales by reducing the number of required experimental tests. However, it is just an instrument for

estimating the performance of a given draft tube configuration. This methodology will be able to display its ability to the full when the runner trailing edge can be connected to the inlet swirl parameters and be able to build an inlet velocity profile in accordance with the runner blade design.

Finally, it is expected that a properly modeled inlet velocity profile can suppress or mitigate the undesirable characteristics such as secondary flow, irregular evolution and excessive velocity through this important component of the turbine.

BIBLIOGRAPHY

Agouzoul, M., Reggio, M., & Camarero, R. (1990). Calculation of turbulent flows in a hydraulic turbine draft tube. *Journal of Fluids Engineering, Transactions of the ASME, 112*, 257–263.

Alekseenko, V., Kuibin, P., Okulov, V., & Shtork, S. (1999). Helical vortices in swirl flow. *Journal of Fluid Mechanics, 382*, 195–243.

Andersson, U. (2001). Test case T- some news results and updates since workshop 1. In *Proceedings of Turbine 99-WS2, The second ERCOFTAC workshop on draft tube flow, Alvkarleby, Sweden.*

Arpe, J. & Avellan, F. (2002). Pressure wall measurements in the whole draft tube: steady and unsteady analysis. In *Proceedings of the XXIst IAHR Symposium on Hydraulic Machinery and Systems, Laussane, Sweden.*

Avellan, F. (2000). Flow investigation in a Francis draft tube: The FLINDT project. In *Proceedings of the Hydraulic Machinery and Systems 20th IAHR Symposium, Charlotte, North Carolina , USA*, DY-03.

Berg, J., Adams, R., Malas-III, J., & Banda, S. (1995). Nonlinear optimization-based design of RAM velocity profiles for isothermal forging. *IEEE Transactions on Control Systems Technology, 3*(3), 269–277.

Bergström, J. (1999). Approximations of numerical errors and boundary conditions in a draft tube. *Proceedings of Turbine 99-Workdhop on draft tube flows, Porjus Sweden.*

Bergström, J. (2000). Modeling and numerical simulation of hydro power flows. Ph.D. thesis, LULEA UNIVERSITY OF TECHNOLOGY.

Bergström, J. & Gebart, R. (1999). Estimation of numerical accuracy for the flow field in a draft tube. *International journal of Numerical Methods for heat and fluid flow*, *9*(4), 472–486.

Burman, J., Gebart, B., & Mårtensson, H. (2000). Developement of a blade geometry definition with implicit design variables. In *38th AIAA Aerospace Sciences Mettingand Exhibit, Reno, N.Y*, 0671.

Cardamone, P. (2006). Aerodynamic optimisation of highly loaded turbine cascade blades for heavy duty gas turbine applications. Ph.D. thesis, Universität der Bundeswehr München.

Cervantes, M. & Engström, F. (2004). Factorial design applied to CFD. *Journal of Fluids Engineering*, *126*, 791–798.

Cervantes, M. & Gustavsson, H. (2005). Estimation of the radial velocity from the Squire-Long equation and experimental data. In *Water Power XIV: Advancing Technology for Sustainable Energy, Austin, Texas*, 14.

Cho, W., Choi, D., & Kim, M.-U. (1999a). Optimization of inlet concentration profile for uniform deposition in a cylindrical chemical vapor deposition chamber. *International Journal of Heat and Mass Transfer*, *42*, 1141–1146.

Cho, W., Choi, D., & Kim, M.-U. (1999b). Optimization of the inlet velocity profile for uniform epitaxial growth in a vertical metalorganic chemical vapor deposition reactor. *International Journal of Heat and Mass Transfer*, *42*, 4143–4152.

Coley, D. (1999). *An introduction to genetic algorithms for scientists and engineers.* World Scientific Publishing.

Doerfler, P. (1985). Francis turbine surge prediction and prevention. *Waterpower 85*, 952–961.

Drtina, P. & Sallaberger, M. (1999). Hydraulic turbines-basic principles and state-of-the-art computational fluid dynamics applications. *Proc. Instn. Mech. Engrs.*, *213*(C), 85–102.

Eisinger, R. & Ruprecht, A. (2001). Automatic shape optimization of hydro turbine components based on CFD. In *Seminar on CFD for turbomachinery applications, Gdansk*, v2001-05. Read Nov. 2006, from http://www.ihs.uni-stuttgart.de/forschung/veroeff/stroem/v2001_05.pdf.

Eklund, S. (2004). A massively parallel architecture for distributed genetic algorithms. *Science@Direct. Parallel Computing*, (30), 647–676.

Engström, F. (2003). Simulations and experiments of turbulent diffuser flow with applications. Ph.D. thesis, Luleå University of Technology, Luleå, Sweden.

Engström, T., Gustavsson, L., & Karlsson, R. (2001). *Proceedings of Turbine 99 Workshop 2 on Draft tube Flow, Lulea University of Technology, Sweden*. Read jan. 2007, from http://www.sirius.ltu.se/strl/Turbine-99/t99_2/Turbine-99_Workshop2_papers.pdf.

Engström, T. & Gebart, B. (1999). Gyroscopic design of swirling flow diffusers. In *Proceedings of FEDSM99, 3er ASME/JSME Joint Fluids Engineering Conference, San Francisco, California USA*.

Enomoto, Y., Kurosawa, S., & Suzuki, T. (2004). Design optimization of a Francis turbine runner using multi-objective genetic algorithm. In *22nd IAHR Symposium on Hydraulic Machinery and Systems, Design methods Turbines Stockholm Sweden*, A01-2.

Falvey, H. (1993). A primer on draft tube surging. *Hydro Review*, (Feruary), 77–86.

Falvey, H. & Cassidy, J. (1970). Frecuency and amplitude of pressure surges generated by swirling flows. In *Proceedings of the International Association for Hydraulic Research.*

Symposium on Hydraulic Machinery and Cavitation. Stockholm, Sweden, E1, (pages 1–12).

Fan, H.-Y. (1998). An inverse design method of diffuser blades by genetic algorithms. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of power and energy*, *112*(4), 261–268.

Fanelli, M. (1998). Mathematical models of the vortex rope surge effects in the draft tube of a Francis turbine working at partial load. In *IAHR 14 th Symposium, Trondheim, Norway*, F1, (pages 341–354).

FLUENT (2003). *User's Guide 6.1. Modelling Turbulence, Chapter 10.4.*

Goulet, S. (1997). Modélisation de la torche dans les turbines hydrauliques. Master's thesis, École Polytechnique de Montréal, Montréal, Québec, Canada.

Grau, M. D., Nougués, J. M., & Puigjaner, L. (2001). Obtention of the optimal feeding profile in a fed-batch reactor using genetic algorithms. *Ind. Eng. Chem. Res.*, *40*, 1488–1494.

Hiroyasu, T., Miki, M., & Negami, M. (1999). Distributed genetic algorithms with randomized migration rate. In *Systems, Man, and Cybernetics, 1999. IEEE SMC '99 Conference Proceedings.*, volume 1, (pages 689–694). Tokyo Japan.

Hofmann, N., Olive, S., Laschet, G., Hediger, F., Wolf, J., & Sahm, P. (1997). Numerical optimization of process control variables for the Bridgman casting process. *Modelling and Simulation in Materials Science and Engineering*, *5*(1), 23–34.

Hothersall, R. (1988). Assessing the design of draft tubes. *Water Power and Dam Construction, April*, 55–57.

Iliescu, M., Ciocan, G., & Avellan, F. (2002). 3D PIV and LDV measurements at the outlet of a Francis turbine draft tube. In *ASME 2002 Fluids Engineering Division Summer Meeting, Montréal, Québec Canada.*

Kazan, M. (1962). Étude de l'écoulement à la sortie d'une roue Francis. Ph.D. thesis, École Polytechnique de l'Université de Lausanne., Lausanne, Switzerland.

Kubota, T. & Aoki, H. (1980). Pressure surge in the draft tube of Francis turbine. In *Practical Experiences with Flow-Induced Vibrations; Proceedings on IAHR/IUTAM symposium*, B8, (pages 279–284). Springer-Verlag.

Kubota, T., Han, F., & Avellan, F. (1996). Performance analysis of draft tube for GAMM Francis turbine. In *Proceedings of the XVIII IAHR Symposium, Hydraulic Machinery and Cavitation. Dordrecht NL.*, 1, (pages 130–139).

Kurosawa, S., Debasish, B., & Nagafuji, T. (2001). Turbulent flow simulation in the draft tube of a Kaplan turbine. In *Proceedings of Turbine 99-WS2, The second ERCOFTAC workshop on draft tube flow, Alvkarleby, Sweden.*

Larsson, C. (2005). Development of hydro turbines. Read Nov. 2006, from http://www.ept.ntnu.no/vk/Forelesn05/Konstr/Forelesning%201/Development%20of%20hydro%20turbines.pdf.

Launder, B. & Spalding, D. (1974). The numerical computational of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, *3*, 269–289.

Lindgren, M. (2002). Automatic shape optimization of hydropower flows: The draf tube. Master's thesis, Luleå University of Technology, Luleå, Sweden.

Lipej, A. & Poloni, C. (2000). Design of Kaplan runner using multiobjective genetic algorithm optimization. *Journal of Hydraulic Research*, *38*(1), 73–79.

Léonard, O., Rothilde, A., & Duysinx, P. (1999). Compressor and turbine blade design by optimization. In *Proceedings of the Third World Congress of Structural and Multidisciplinary Optimization WCSMO3, Buffalo (NY-USA)*, 43-EOA3-1.

Longatte, F., Kueny, J., & Avellan, F. (1999). Three-dimensional analysis of the flow in the ERCOFTAC draft tube with N3S and FLUENT softwares. In *Proceedings of Turbine 99 - Workshop on Draft Tube Flow, Porjus, Sweden.*

Lépine, J. (1999). Optimisation de la répresentation des profils d'ailes pour application au design aérodynamique. Master's thesis, École Polytechnique de Montréal, Montréal, Québec, Canada.

Lépine, J., Guibault, F., Trépanier, J.-Y., & Pépin, F. (2001). Optimized nonuniform rational B-spline geometrical representation for aerodynamic design of wings. *American Institute of Aeronautics and Astronomics*, *39*(11), 2033–2041.

Madsen, J., Shyy, W., & Haftka, R. (2000). Response surface techniques for diffuser shape optimization. *AIAA journal*, *38*(9), 1512–1518.

Marjavaara, D. & Lundström, T. (2004). Shape optimization of a hydropower draft tube. In *22nd IAHR Symposium on Hydraulic Machinery and Systems. Design methods Turbines. Stockholm Sweden*, volume A03-2.

Marjavaara, D. & Lundström, T. (2005). Redesign of a sharp heel draft tube by a validated CFD-optimization. *International Journal for Numerical Methods in Fluids*, *50*, 911–924.

Masse, B. (1999). Simulation creates potential 5 million revenue gain from hydropower turbines. *Journal Articles by Fluent Software Users*, (JA101), 1–4.

Mauri, S., Kueny, J., & Avellan, F. (2000). Numerical prediction of the flow in a tur-

bine draft tube influence on boundary conditions. In *ASME 2000 Fluids Engineering Division Summer Meeting, Boston, Massachusetts, USA.*

Mauri, S., Kueny, J., & Avellan, F. (2002a). Flow simulation in an elbow diffuser: verification and validation. In *Proceedings of the Hydraulic Machinery and Systems XXIst IAHR Symposium, Lausanne.*

Mauri, S., Kueny, J., & Avellan, F. (2002b). Werlé-Legendre separation in a hydraulic machine draft tube. In *ASME 2002 Fluids Engineering Division Summer Meeting, Montréal, Québec Canada.*

McDonald, A., Fox, R., & Van Dewoestine, R. (1971). Effects of swirling inlet flow on pressure recovery in conical diffusers. *AIAA Journal, 9*(10), 2014–2018.

Memarzadeh, F. (1998). *Ventilation Design Handbook on Animal Research Facilities Using Static Microisolators, I-V.* Read feb. 2007, from http://orf.od.nih.gov/PoliciesAndGuidelines/Bioenvironmental/toc_vent.htm.

Mengistu, T. & Ghaly, W. (2003). Global optimization methods for the aerodynamic shape design of transonic cascades. In *Proceedings of the 11th CFD Conference of the Canadian Society of CFD, Vancouver, BC*, volume 1, (pages 238–243).

Miki, M., Hiroyasu, T., Kaneko, M., & Hatanaka, K. (1999). A parallel genetic algorithm with distributed environment scheme. In *Systems, Man, and Cybernetics, 1999. IEEE SMC '99 Conference Proceedings*, volume 1, (pages 695–700). Tokyo Japan.

Mollenkopf, G. & Raabe, J. (1970). Measurements of fluctuations of velocity and pressure in the draft tube of a Francis turbine. In *IAHR Symposium, Stockholm*, B3, (pages 1–12).

Monacelli, G. & Cooper, P. (1987). Performance of draft tubes for small-hydropower turbines. *WATERPOWER '87*, 1200–1209.

Muntean, S., Ruprecht, A., & Susan-Resiga, R. (2005). A numerical investigation of the 3d swirling flow in a pipe with constant diameter. Part 1 inviscid computation. In *Workshop on Vortex Dominated Flows - Achievements and Open Problems Timisoara, Romania*. Scientific Bulletin of the Politehnica University of Timisoara Transactions on Mechanics Special issue.

Muntean, S., Susan-Resiga, R., Bernard, S., & Anton, I. (2004). 3D turbulent flow analysis of the GAMM Francis turbine for variable discharge. In *22nd IAHR Symposium on Hydraulic Machinery and Systems, Stockholm, Sweden.*

Nechleba, M. (1957). *Hydraulic turbines their design and equipment.* Prague Czechoslovakia: Artia, c1957.

Nilsson, H. (2002). Numerical investigations of turbulent flow in water turbines. Ph.D. thesis, Chalmers University of Technology, Göteborg, Sweden.

O'kelly, F. (1989). Draft tube surging in Francis turbines. *Hydro Review*, *April*, 86–86.

Oyama, A., Liou, M., & Obayashi, S. (2002). Transonic axial-flow blade shape optimization using evolutionary algorithm and three-dimensional Navier-Stokes solver. In *9th AIAA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, Atlanta, Georgia*, volume AIAA 2002-5642.

Padle, U. (1974). Model and prototype turbine draft tube surge analysis by the swirl momentum method. In *Proceedings of the International Association for Hydraulic Research. Symposium on Hydraulic Machinery and Cavitation. Stockholm, Sweden*, III 3, (pages 1–13).

Page, M. & Giroux, A. (2001). Turbulent flow computations in Turbine 99 draft tube with CFX-TASCflow, FIDAP and FINE/Turbo. In *The 2nd ERCOFTAC Workshop on Draft Tube Flow, Vattenfall Utveckling AB, Älvkarleby, Sweden.*

Peng, G., Cao, S., Ishizuca, M., & Hayama, S. (2002). Design optimization of axial flow hydraulic turbine runner; part II - multi - objective constrained optimization method. *Int. J. Numer. Meth. Fluids*, *39*, 533–548.

Puente, L., Reggio, M., & Guibault, F. (2003). Automatic shape optimization of a hydraulic turbine draft tube. In *CFD 2003: The Eleventh Annual Conference of the CFD Society of Canada, Vancouver, BC*. Read Nov. 2006, from http://tetra.mech.ubc.ca/CFD03/papers/paper29PB1.pdf.

Ruiz, E. (2004). *Polyfit Software*. Chaire de Recherche sur le Composite Haute Performance CCHP, École Polytechnique de Montréal.

Ruprecht, A., Eisinger, R., & Göde, E. (2000a). Innovative design environments for hydro turbine components. In *HYDRO 2000, Bern*, v2000-05. Read Nov. 2006, from http://www.ihs.uni-stuttgart.de/forschung/veroeff/stroem/v2000_05.pdf.

Ruprecht, A., Eisinger, R., Göde, E., & Rainer, D. (1999). Virtual numerical test bed for intuitive design of hydro turbine components. In *Hydropower into the Next Century, Gmunden, Austria*, v1999-04. Read Nov. 2006, from http://www.ihs.uni-stuttgart.de/forschung/veroeff/stroem/v1999_04.pdf.

Ruprecht, A., Heitele, M., Helmirich, T., Moser, W., & Aschenbrenner, T. (2000b). Numerical simulation of complete Francis turbine including unsteady rotor/stator interactions. In *Proceedings of the Hydraulic Machinery and Systems 20th IAHR Symposium, Charlotte, North Carolina , USA*, CFD-S03.

Ruprecht, A., Helmrich, T., Aschenbrenner, T., & Scherer, T. (2002). Simulation of vortex rope in a turbine draft tube. In *Proceedings of the Hydraulic Machinery and Systems 21 st, IAHR Symposium, Lausanne*.

Senoo, Y., Kawaguchi, N., & Nagata, T. (1978). Swirl flow in conical diffusers. *Bulletin of the JSME*, *21*(151), 112–119.

Sharma, C., Soundranayagam, S., & Mukund, T. (1998). Dependence of vortex breakdown on angular momentum parameter in draft tubes flows. *Current Science*, *75*(12), 1355–1363.

Shárán, V. (1976). An exponential investigation of the behaviour of conical diffusers in turbulent flow. *Journal of Applied Mathematics and Physics*, *27*, 447–432.

Shyy, W. & Braaten, M. (1986). Three-dimensional analysis of the flow in a curved hydraulic turbine draft tube. *International Journal of Numerical Methods in Fluids*, *6*, 861–882.

Song, C., He, J., & CHen, X. (1991). Calculation of turbulent flow through a Francis turbine runner and elbow draft tube. In *International Power Generation Conference, San Diego, CA, USA*, (pages 1–6).

Song, W. & Keane, A. (2004). A study of shape parametrisation methods for airfoil optimisation. *American Institute of Aeronautics and Astronautics*, 1–8.

Susan-Resiga, R., Ciocan, G., Anton, I., & Avellan, F. (2006). Analysis of the swirling flow downstream a Francis turbine runner. *Journal of Fluids Engineering*, *128*, 177–189.

Susan-Resiga, R., Ciocan, G., & Avellan, F. (2004). Swirling flow downstream a Francis turbine runner. In *The 6th International conference on Hydaulic Machinery and Hydrodynamics Timisoara, Romania*. Scientific Bulletin of the Politehnica University of Timisoara Transactions on Mechanics Special issue.

Swiderski, J., Martin, J., & Norrena, R. (2001). Automated runner blade design optimization process based on CFD verification. In *Waterpower XII, Salt Lake City*.

Tomas, L., Pedretti, C., Chiappa, T., François, M., & Stoll, P. (2002). Automated design

of a Francis turbine runer using global optimization algorithms. In *Proceedings of the XXIst IAHR Symposioum on Hydraulic Machinery and Systems, Lausanne.*

Vu, T. (1989). A design parameter study of turbine draft tube by viscous flow analysis. *WATERPOWER'89*, 557–566.

Vu, T., Nennemann, B., Ciocan, G., Iliescu, M., Braun, O., & Avellan, F. (2004). Experimental study and unsteady simulation of the FLINDT draft tube rotating vortex rope. In *Proceedings of the Hydro 2004 Conference, Porto, Portugal.*

Vu, T. & Shyy, W. (1988). Viscous flow analysis for hydraulic turbine draft tubes. In *Proceedings of 14th I.A.H.R. Symposium on Progress within large and high-specific, Trondheim*, volume 2, (pages 915–926).

Vu, T. & Shyy, W. (1990). Navier-Stokes flow analysis for hydraulic turbine draft tubes. *Journal of Fluids Engineering*, *112*, 199–204.

Wahl, T. (1994). Draft tube surging times two; the twin vortex phenomenon. *Hydro Review*, *February*, 60–69.

Zangench, M., Goto, A., & Harada, H. (1999). On the role of three-dimensional inverse design methods in turbomachinery shape optimization. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, *213*(1), 27–42.

Zolotov, L., Klabukov, V., Ivanov, I., & Ivanova, G. (1970). Flow dynamic characteristics downstream of hydraulic turbine runner and their influence on conditions of turbine units regulation. In *IAHR Symposium, Stockholm*, B2, (pages 1–12).

APPENDIX I

UNIX SHELL SCRIPT

I.1 Zeta

```
# Nom du repertoire de travail (unique)
echo "### Construction des noms de repertoire pour Echidna"
rep_courant = ` pwd `
rep_travail= ècho $rep_courant | gawk ´ BEGIN {FS= "/" } {print $NF} `
rep_travail_base_echidna= "/home/segal/T99/T99_parallel/T4/"
rep_travail_distribue_echidna= "$rep_travail_base_echidna$rep_travail"
# Nettoyage des fichiers
find .-name 'out.dat' -exec rm {} \ ;
# Copie profil de vitesse sur echidna
echo "#### Transfert du fichier in.dat sur echidna:$rep_travail_distribue_echidna"
ssh segal@echidna.gi.polymtl.ca " cd $rep_travail_base_echidna ; mkdir $rep_travail"
scp in.dat segal@echidna.gi.polymtl.ca:$rep_travail_distribue_echidna/.
# Lance execution echo "### Execution sur echidna : ` date` "
ssh segal@echidna.gi.polymtl.ca "cd $rep_travail_base_echidna ; qsub -v
WORK_DIR_ECHIDNA_BASE=$rep_travail_base_echidna,
WORK_DIR_ECHIDNA_DISTRIBUE= $rep_travail_distribue_echidna,
WORK_DIR_RECHERCHE= $rep_courant T99_echidna.pbs"
# Test presence fichier pour confirmer que fluent est termine
until [ -f out.dat ]
do
sleep 1 done
# Suppression des repertoires sur echidna
#DISPLAY=:0.0
#export DISPLAY
#ssh segal@echidna.gi.polymtl.ca "cd $rep_travail_base_echidna ;rm -r
$rep_travail_distribue_echidna"
echo "### Scritp termine : ` date` "
```

I.2 Echidna

```
#cd $WORK_DIR_ECHIDNA
# Redirection des output vers un fichier exec & >
$WORK_DIR_ECHIDNA_DISTRIBUE/logFileEchidna.txt
echo "##############################################"
echo "###### Calculs effectues sur la machine ` hostname` "
echo "##############################################"
cd $WORK_DIR_ECHIDNA_BASE
find . -name 'T99_echidna.pbs.* ' -exec rm {} \ ;
cd $WORK_DIR_ECHIDNA_DISTRIBUE
matlab_test=1
count=1
while [[ $matlab_test ]]
do
/usr/local/bin/matlab -nodesktop -nosplash -r "path ( path,' $ WORK_DIR_ECHIDNA_BASE') ; BC1 ; exit" > BC1.txt &
PIDG= $!
echo "—> $count — MATLAB-1 est lance avec PID = $PIDG : ` date` "
wait $PIDG
echo "—> MATLAB-1 a termine : ` date ` "
matlab_test$=$` gawk ' $1 ~ /License/ && $2 ~ /Manager/ && $3 ~ /Error/ {print 1 ; exit }'BC1.txt `
count= ` expr $count + 1`
sleep 1
done
fluent -r6.2.16 3d -g -i ../T99_setup_1.jou > T99_res_1.txt &
PIDG=$!
echo "#### Fluent-1 est lance pour determiner le debit numerique : ` date` "
wait $PIDG
echo "#### Fluent-1 a termine :` date` "
# Test presence d'un flux inverse
TEST_FQ="0"
TEST_FQ=` tail -1 m.out | gawk '{if ( $2 < 0.000001 ) print "0" }'`
if [ "$TEST_FQ" == "0" ] then
"Engineering quantities" > out.dat
echo "CPm: 0.000000" » out.dat
echo "CPw: 0.000000" » out.dat
echo "Z: 1000.000000" » out.dat
echo "FQ1: 1000.0000" » out.dat
echo "FQ2: 1000.000000" » out.dat
```

```
echo "S: 1000.0000" » out.dat
echo "imb: 1.000000" » out.dat
echo "!!!!!!!!! Flux en sens contraire: Fluent pas relance"
scp $WORK_DIR_ECHIDNA_DISTRIBUE/out.dat
gonzalez@zeta.recherche.polymtl.ca:$WORK_DIR_RECHERCHE/.
exit 0
fi
matlab_test=1 while [[ $matlab_test ]]
do
matlab -nodesktop -nosplash -r "path(path,'$WORK_DIR_ECHIDNA_BASE') ; BC2 ; exit" > BC2.txt &
PIDG=$!
echo "—> MATLAB-2 est lance avec PID = $PIDG : ` date` "
wait $PIDG
echo "—> MATLAB-2 a termine : ` date` "
matlab_test=` gawk ´ $1  /License/ && $2  /Manager/ && $3  /Error/ {print 1 ;exit }' BC2.txt ` sleep 1
done
# Test flux trop important
TEST_FQ="0" FQ1=` gawk '/FQ1/ ' out.dat`
TEST_FQ=` gawk '/FQ1/
{if ( $2 > 2.5 ) print "0" }' out.dat` if [ "$TEST_FQ"== "0" ]
then
echo "Engineering quantities" > out.dat
echo "CPm: 0.000000" » out.dat
echo "CPw: 0.000000" » out.dat
echo "Z: 1000.000000" » out.dat
echo $FQ1 » out.dat
echo "FQ1: 1000.0000" » out.dat
echo "FQ2: 1000.000000" » out.dat
echo "S: 1000.0000" » out.dat
echo "imb: 1.000000" » out.dat
echo "!!!!!!!!! Flux > 2.5 : Fluent pas relance"
scp $WORK_DIR_ECHIDNA_DISTRIBUE/out.dat gonzalez@zeta.recherche.polymtl.ca:$WORK_DIR_RECHERCHE/.
exit 0
fi
fluent -r6.2.16 3d -g -i ../T99_setup_2.jou > T99_res_2.txt &
PIDG=$!
echo "#### Fluent-2 est lance pour determiner la performance : ` date` "
wait
$PIDG echo "#### Fluent-2 a termine : ` date` "
matlab_test=1 while [[ $matlab_test ]]
do
matlab -nodesktop -nosplash -r "path(path,'$WORK_DIR_ECHIDNA_BASE') ; EQ ; exit" > EQ.txt &
```

```
PIDG=$!
echo "—> MATLAB-3 est lance avec PID = $PIDG : ` date` "
wait $PIDG
echo "—> MATLAB-3 a termine : ` date` " matlab_test=` gawk ´ $1   /License/ && $2   /Manager/ && $3   /Error/ {print 1 ; exit}'
EQ.txt`
sleep 1
done
scp $WORK_DIR_ECHIDNA_DISTRIBUE/out.dat gonzalez@zeta.recherche.polymtl.ca: $WORK_DIR_RECHERCHE/.
scp $WORK_DIR_ECHIDNA_DISTRIBUE/T99_res_1.txt gonzalez@zeta.recherche.polymtl.ca: $WORK_DIR_RECHERCHE/.
scp $WORK_DIR_ECHIDNA_DISTRIBUE/T99_res_2.txt gonzalez@zeta.recherche.polymtl.ca: $WORK_DIR_RECHERCHE/.
```

APPENDIX II

MATLAB PROGRAMS

II.1 Boundary conditions 1

```
% Reading of the inlet swirl parameters
fp =fopen('in.dat','r');
fgetl(fp) ;
[ d count ] =fscanf(fp,'%*s % f') ;
fclose (fp) ;
d; count;
[m n]=size (d);
R1=d(1,1); R2=d(2,1); U0=d(3,1); U1=d(4,1); U2=d(5,1); W0=d(6,1); W1=d(7,1); W2=d(8,1);
% Experimental velocity components best efficient point.
R_ex = [ 0.0981 0.1014 0.1041 0.1068 0.1094 0.1161 0.1227 0.1294 0.1427 0.1560 0.1693 0.1826 0.1959 0.2092 0.2168 0.2221
0.2261 0.2301 0.2314 0.2328 0.2341 0.2365]';
Va_ex = [ 0.0000 -2.5983 -2.7154 -2.8964 -3.0455 -3.2691 -3.2585 -3.3437 -3.4821 -3.5992 -3.6312 -3.8015 -3.9293 -3.9506 -3.8867
-3.7802 -3.7802 -3.7696 -3.7057 -3.6205 -3.3650 0]';
Vr_ex= [ 0.0000 -0.5726 -0.5833 -0.6061 -0.6210 -0.6219 -0.5762 -0.5458 -0.4754 -0.3957 -0.3033 -0.2176 -0.1218 -0.0191 0.0393
0.0776 0.1074 0.1368 0.1440 0.1507 0.1487 0]';
Vt_ex= [ -6.5063 -0.3940 -0.6176 -0.7667 -0.7348 -0.6922 -0.6496 -0.6389 -0.6922 -0.8412 -0.9690 -1.1394 -1.3950 -1.3630 -1.3737
-1.5227 -1.6718 -1.6931 -1.6612 -1.6186 -1.5440 0]';
K_ex= [ 0.0000 0.4973 0.5727 0.7158 0.5234 0.3200 0.2887 0.2133 0.2063 0.2082 0.2063 0.3324 0.6556 0.3722 0.3976 0.6154
0.8732 0.9654 1.0099 0.9582 1.0236 0]';
E_ex= [ 0.0000 3.5067 4.3346 6.0566 3.7869 1.8096 1.5512 0.9851 0.9368 0.9499 0.9368 1.9160 5.3087 2.2703 2.5061 4.8274
8.1593 9.4850 10.1475 9.3783 10.3560 0]';
% Experimental constants
thh=-12.8; %center angle
thw=2.8; % wall angle
Rh=0.09809; %center radius
Rw=0.236461014; % Wall radius
D=998.19; %Flow density
Ain=0.14482079; %Area inlet
Q_ex=-0.522; %inlet flow
m_ex=D*Q_ex; %inlet flow mass
% Normalized factors Velocity profiles
```

```
n=595; % runner rotational speed
Rre=(0.5/2); %Runner radius as reference radius
w=2*pi* (n/60); %runner angular speed
Vre=w*(Rre); % reference velocity
Rs=(Rw-Rh)/(Rre-Rh);% Measure section radius
%%%%%%%%Near-wall expeimental velocity profile %%%%%%%%
R_hu = [0.09809 0.100090005 0.103090003]';
R_co = [ 0.107590005 0.11434 0.135514215 0.156688407 0.1778626 0.199036807 0.220211014 0.226961017]';
R_wa = [0.231461003 0.23446101 0.236461014]';
r_hu=(R_hu-Rh)./(Rw-Rh); % Points at the hub radius
r_co=(R_co-Rh)./(Rw-Rh); % Points at the center radius
r_wa=(R_wa-Rh)./(Rw-Rh); % Points at the wall radius
rr = cat(1,r_hu,r_co);
RR = cat(1,R_hu,R_co);
m=length (r_co); o=length (r_hu); p=length (r_wa); q=length (rr);
%%%%%%%%Analytical central velocity profile%%%%%%%%
th_an=thh+(thw-thh)/((Rw)-(Rh))*((RR)-(Rh)); % Flow angle
Vac_an=U0+U1*(exp(-(rr.^ 2/R1^ 2)))+U2*(exp(-(rr.^ 2/R2^ 2)));
Vtc_an=(W0.*rr+((W1.*R1^ 2)./rr).*(1-exp(-(rr.^ 2./R1^ 2)))+((W2*R2^ 2)./rr).*(1-exp(-(rr.^ 2./R2^ 2))));
Vrc_an=-Vac_an.*(tan(th_an*(pi/180)));
PHI_an=(U0*Rs^ 2)+(U1*R1^ 2)*(1-exp(-(Rs^ 2/R1^ 2)))+(U2*R2^ 2)*(1-exp(-(Rs^ 2/R2^ 2)));
%%%%%%%%Flow constraint%%%%%%%%
if any(Vac_an < 0); Vac_an=(Vac_an.*0.000); end
if any(Vtc_an < 0); Vac_an=(Vac_an.*0.000); end
%%%%%%%% Analytical near-wall profile %%%%%%%%
fh=((Rh-R_hu)/(Rh-R_co(1,1))).^ 0.14285714;
i=o+1; xx=Vac_an(i,1); yy=Vrc_an(i,1); zz=Vtc_an(i,1);
Vah_an=xx.*fh; Vrh_an=yy.*fh;
%%%%%%%% Analytical Near wall inlet velocity profile %%%%%%%%
[i,j] = size(R_co); [ii,jj] = size(rr); fw=((Rw-R_wa)./(Rw-R_co(i,j))).^ 0.14285714;
aa=Vac_an(ii,jj); bb=Vrc_an(ii,jj); cc=Vtc_an(ii,jj);
Vaw_an=aa.*fw; Vrw_an=bb.*fw; Vtw_an=cc.*fw;
%%%%%%%% grouping of the inlet velocity profiles %%%%%%%%
Va_an = cat(1,Vac_an,Vaw_an); Vr_an = cat(1,Vrc_an,Vrw_an);
Vt_an = cat(1,Vtc_an,Vtw_an); Va_an(1,1)=Vah_an(1,1); Va_an(2,1)=Vah_an(2,1); Va_an(3,1)=Vah_an(3,1);
Vr_an(1,1)=Vrh_an(1,1); Vr_an(2,1)=Vrh_an(2,1); Vr_an(3,1)=Vrh_an(3,1);
Vt_an(1,1)=[6.5063/Vre]'; r = cat(1,r_hu,r_co,r_wa);
R= cat(1,R_hu,R_co,R_wa); [pp,qq] = size(R); K_an = interp1(R_ex,K_ex,R,'spline'); E_an = interp1(R_ex,E_ex,R,'spline');
K_an(1,1)=0; E_an(1,1)=0; K_an(pp,1)=0; E_an(pp,1)=0;
%%%%%%%% Analytical real inlet boundary conditions %%%%%%%%
va_an=Va_an.*-Vre; vr_an=Vr_an.*-Vre; vt_an=Vt_an.*-Vre;
k_an=K_an; e_an=E_an; r=(r.*(Rw-Rh))+Rh;
```

```
%%%%%%%% Analytical Velocity profile 1 %%%%%%%%
[mm,nn] =size(r); M=mm*nn;
fid = fopen('T99_BC1a_1.prof','w');
fprintf(fid, '((bc_1a radial %0.0f)\n',M);
fprintf(fid, '(r \n');
fprintf(fid, '%6.6f \n' , r);
fprintf(fid, ') \n');
fprintf(fid, '(va \n');
fprintf(fid, '%6.6f \n', va_an);
fprintf(fid, ') \n');
fprintf(fid, '(vr \n');
fprintf(fid, '%6.6f \n', vr_an);
fprintf(fid, ') \n');
fprintf(fid, '(vt\n');
fprintf(fid, '%6.6f \n', vt_an);
fprintf(fid, ') \n');
fprintf(fid, '(k\n');
fprintf(fid, '%6.6f \n', k_an);
fprintf(fid, ') \n');
fprintf(fid, '(e\n');
fprintf(fid, '%6.6f \n', e_an);
fprintf(fid, ') \n');
fclose(fid);
exit
```

II.2 Boundary conditions 2

```
R_ex = [ 0.0981 0.1014 0.1041 0.1068 0.1094 0.1161 0.1227 0.1294 0.1427 0.1560 0.1693 0.1826 0.1959 0.2092 0.2168 0.2221
0.2261 0.2301 0.2314 0.2328 0.2341 0.2365]';
Va_ex = [ 0.0000 -2.5983 -2.7154 -2.8964 -3.0455 -3.2691 -3.2585 -3.3437 -3.4821 -3.5992 -3.6312 -3.8015 -3.9293 -3.9506 -3.8867
-3.7802 -3.7802 -3.7696 -3.7057 -3.6205 -3.3650 0]';
Vr_ex=[ 0.0000 -0.5726 -0.5833 -0.6061 -0.6210 -0.6219 -0.5762 -0.5458 -0.4754 -0.3957 -0.3033 -0.2176 -0.1218 -0.0191 0.0393
0.0776 0.1074 0.1368 0.1440 0.1507 0.1487 0]';
Vt_ex=[ -6.5063 -0.3940 -0.6176 -0.7667 -0.7348 -0.6922 -0.6496 -0.6389 -0.6922 -0.8412 -0.9690 -1.1394 -1.3950 -1.3630 -1.3737
-1.5227 -1.6718 -1.6931 -1.6612 -1.6186 -1.5440 0]';
K_ex=[ 0.0000 0.4973 0.5727 0.7158 0.5234 0.3200 0.2887 0.2133 0.2063 0.2082 0.2063 0.3324 0.6556 0.3722 0.3976 0.6154
0.8732 0.9654 1.0099 0.9582 1.0236 0]';
E_ex=[ 0.0000 3.5067 4.3346 6.0566 3.7869 1.8096 1.5512 0.9851 0.9368 0.9499 0.9368 1.9160 5.3087 2.2703 2.5061 4.8274 8.1593
```

```
9.4850 10.1475 9.3783 10.3560 0]';
Q_ex=0.522; Ain=0.14482; D=998.2; m_ex=Q_ex*D; Rw=0.236461014;
r=dlmread('T99_BC1a_1.prof',' ',[2,0,15,0]) ;
Va_an=dlmread('T99_BC1a_1.prof',' ',[18,0,31,0]) ;
Vr_an=dlmread('T99_BC1a_1.prof',' ',[34,0,47,0]) ;
Vt_an=dlmread('T99_BC1a_1.prof',' ',[50,0,63,0]) ;
k_an=dlmread('T99_BC1a_1.prof',' ',[66,0,79,0]) ;
e_an=dlmread('T99_BC1a_1.prof',' ',[82,0,95,0]) ;
[n m] = textread('m.out','%f %f', 'headerlines',3);
[i j]=size (m); m1_nu = m(i,j);
FQ1=m1_nu/(m_ex); nor=FQ1;%WITHOUT NORMALIZATION FLUX nor=FQ1
ff=FQ1/nor;
%%%%%%%% NORMALIZATION FLUX %%%%%%%%
va_an=Va_an./ff; vr_an=Vr_an./ff; vt_an=Vt_an./ff; k_an=k_an./ff; e_an=e_an./ff;
vt_an(1,1)=[-6.5063]';
%%%%%%%% inlet Swirl Number outer radius %%%%%%%%
[mm,nn] =size(r);
for i=1:1:mm-1; dr(i)=[r(i+1)-r(i)]; end
dr=cat(2,0,dr); dr=dr';
SA=(r.^ 2.*D.*(vt_an.*va_an.*dr));
SB=(Rw.**D.*(r).*(va_an.^ 2).*dr);
SA=sum(SA); SB=sum(SB); S=SA/SB;
M=mm*nn; fid = fopen('T99_BC1a_2.prof','w');
fprintf(fid, '((bc_1a radial %0.0f)\n',M);
fprintf(fid, '(r \n');
fprintf(fid,'%6.6f \n' , r);
fprintf(fid, ')\n');
fprintf(fid, '(va \n');
fprintf(fid,'%6.6f \n', va_an);
fprintf(fid, ')\n');
fprintf(fid, '(vr \n');
fprintf(fid,'%6.6f \n', vr_an);
fprintf(fid, ') \n');
fprintf(fid, '(vt\n');
fprintf(fid,'%6.6f \n', vt_an);
fprintf(fid, ')\n');
fprintf(fid, '(k\n');
fprintf(fid,'%6.6f \n', k_an);
fprintf(fid, ')\n');
fprintf(fid, '(e\n');
fprintf(fid,'%6.6f \n', e_an);
fprintf(fid, ')\n');
```

```
fclose(fid);
%%%%%%%% Preliminary inlet engineering quantities %%%%%%%%
CPm=0.000000; CPw=0.000000; Z=1000.000000; imb=1.000000; FQ2=1000.000000;
fid = fopen('out.dat','w');
fprintf(fid, 'Engineering quantities\n');
fprintf(fid, 'CPm: %6.6f \n', CPm);
fprintf(fid, 'CPw: %6.6f \n', CPw);
fprintf(fid, 'Z: %6.6f \n', Z);
fprintf(fid, 'FQ1: %6.6f \n', FQ1);
fprintf(fid, 'FQ2: %6.6f \n', FQ2);
fprintf(fid, 'S: %6.6f \n', S);
fprintf(fid, 'imb: %6.6f \n', imb);
fclose(fid);
exit
```

II.3 Engineering quantities

```
%%%%%%%% Turbine data %%%%%%%%
Ain=0.144817 ; % Inlet section area D=998.2; % Water density Q_ex=0.522 ; m_ex=Q_ex*D;%inlet flow mass
%%%%%%%% Obtaining from files the engineering quantities %%%%%%%%
fp =fopen('m.out','r');
fgetl(fp); fgetl(fp); fgetl(fp) ;
[d,count] =fscanf(fp,'%f');
fclose (fp) ;
[m n]=size (d);
m2_nu = d(m, n) ;
fp =fopen('Ps_in.out','r') ;
fgetl(fp) ; fgetl(fp); fgetl(fp);
[d,count] =fscanf(fp,'%f');
fclose (fp) ;
[m n]=size (d);
Ps_in=d(m,n) ;
fp =fopen('Ps_out.out','r');
fgetl(fp) ; fgetl(fp); fgetl(fp);
[d,count]=fscanf(fp,'%f');
fclose (fp) ;
[m n] =size (d);
Ps_out=d(m,n) ;
```

```
fp =fopen('Pt_in.out','r');
fgetl(fp) ; fgetl(fp); fgetl(fp) ;
[d,count]=fscanf(fp,'%f');
fclose (fp) ;
[m n]=size (d);
Pt_in=d(m,n)
fp =fopen('Pt_out.out','r');
fgetl(fp) ; fgetl(fp); fgetl(fp);
[d,count]=fscanf(fp,'%f');
fclose(fp) ;
[m n]=size (d);
Pt_out=d(m,n);
fp =fopen('Pw_in.out','r');
fgetl(fp) ; fgetl(fp); fgetl(fp) ;
[d,count]=fscanf(fp,'%f');
fclose (fp) ;
[m n]=size (d);
Pw_in=d(m,n) ;
fp =fopen('Pw_out.out','r');
fgetl(fp); fgetl(fp); fgetl(fp);
[d,count]=fscanf(fp,'%f');
fclose (fp) ;
[m n]=size (d);
Pw_out=d(m,n) ;
fp =fopen('imb.out','r');
fgetl(fp) ; fgetl(fp); fgetl(fp) ;
[d,count]=fscanf(fp,'%f');
fclose(fp) ;
[m n]=size (d);
imb=d(m,n) ;
fp =fopen('out.dat','r');
fgetl(fp) ;
[d count]=fscanf(fp,' %*s %f');
fclose (fp);
d; count;
[m n]=size (d); S=d(6,1);
fp =fopen('out.dat','r');
fgetl(fp) ;
[d count]=fscanf(fp,' %*s %f') ;
fclose (fp) ; d; count;
[m n]=size (d);
FQ1=d(4,1);
```

```
%%%%%%% Calculating Engineering quantities %%%%%%%
Q2_nu=m2_nu/D ;
Vam2_nu=Q2_nu/(Ain);
Pdy_in=(0.5*D*(Vam2_nu)^ 2);
CPm=(Ps_out-Ps_in)/Pdy_in;
CPw=(Pw_out-Pw_in)/Pdy_in;
Z=(Pt_in-Pt_out)/Pdy_in;
FQ2=m2_nu/m_ex;
%%%%%%% Generating out file %%%%%%%
fid = fopen('out.dat','w');
fprintf(fid, 'Engineering quantities\n');
fprintf(fid, 'CPm: %6.6f \n', CPm);
fprintf(fid, 'CPw: %6.6f \n', CPw);
fprintf(fid, 'Z: %6.6f \n', Z);
fprintf(fid, 'FQ1: %6.6f \n', FQ1);
fprintf(fid, 'FQ2: %6.6f \n', FQ2);
fprintf(fid, 'S: %6.6f \n', S);
fprintf(fid, 'imb: %6.6f \n', imb);
fclose(fid) exit
```

APPENDIX III

ISIGHT USER INTERFACE

III.1 Task diagram

Figure III.1 Task diagram.

III.2 Parameters display

Figure III.2 Parameters display.