Claudinei de Moura Altea

**Computational Determination of Convective
Heat Transfer and Pressure Drop
Coefficients of Hydrogenerators Ventilation
System**

Dissertation presented to Escola Politécnica
da Universidade de São Paulo for the partial
requirement for the degree of Master of
Science in Mechanical Engineering

Supervisor: Prof. Dr. Jurandir Itizo Yanagihara

July 2016

Este exemplar foi revisado e corrigido em relação à versão original, sob responsabilidade única do autor e com a anuência de seu orientador.

São Paulo, _____ de _____ de _____

Assinatura do autor: _____

Assinatura do orientador: _____

Catálogo-na-publicação

Altea, Claudinei de Moura

Computational determination of convective heat transfer and pressure drop coefficients of hydrogenerators ventilation system / C. M. Altea -- versão corr. -- São Paulo, 2016.
154 p.

Dissertação (Mestrado) - Escola Politécnica da Universidade de São Paulo. Departamento de Engenharia Mecânica.

1.SISTEMAS DE VENTILAÇÃO I.Universidade de São Paulo. Escola Politécnica. Departamento de Engenharia Mecânica II.t.

Dedictory

I dedicate this work to
my wife *Nathalia* and
our son *Raul*.

Acknowledgments

Above all, I thank God for having blessed me with a wonderful family, great opportunities and the provision of all my needs to walk his proposed ways for me, including the accomplishment of this work.

To my supervisor, Prof. Dr. Jurandir Itizo Yanagihara, for the thrust and belief from the earliest and all the support along the whole work.

To the company Voith Hydro and all the friends I have by been working there. For the knowledge acquired working for years with engineering teams, their support and encouragement for this work, as well the support from the workshop, field service and commissioning (HPP Teles Pires) colleagues for their support with field measurements development.

To my family, especially to my wife Nathalia, for the understanding and patience in the moments of my familiar absence, and all the support in difficult times.

Resumo

O objetivo do presente trabalho é determinar os coeficientes de perda de carga e transferência de calor, normalmente aplicados nos cálculos analíticos de design térmico de hidrogeradores, obtido pela aplicação de cálculo numérico (Computacional Fluid Dynamics - CFD) e validado por resultados experimentais e medições de campo. O objeto de estudo é limitado à região mais importante do sistema de ventilação (os dutos de ar de arrefecimento do núcleo do estator) para obter resultados numéricos dos coeficientes de transferência de calor e de perda de carga, que são impactados principalmente pela entrada de dutos de ar.

Os cálculos numéricos consideraram escoamentos tridimensionais, em regime permanente, incompressíveis e turbulentos; e foram baseados no método dos volumes finitos. Os cálculos de escoamento turbulento foram realizados com procedimentos baseados em equações médias (RANS), utilizando o modelo k-omega SST (Shear-Stress Transport) como modelo de turbulência. Métricas de qualidade de malha foram monitoradas e as incertezas devido à erros de discretização foram avaliadas por meio de um estudo de independência de malha e aplicação de um procedimento de estimativa de incertezas com base na extrapolação de Richardson.

A validação do método numérico desenvolvido pelo presente trabalho (especificamente para simular o comportamento dinâmico do escoamento e obter numericamente o coeficiente de perda de carga do escoamento ao entrar no duto de ar e atravessar o núcleo do estator de um hidrogerador) é realizada comparando os resultados numéricos com dados experimentais publicados por Wustmann (2005). Os dados experimentais foram obtidos como referência por um teste de modelo. A comparação entre os resultados numéricos e experimentais mostra que a diferença da perda de carga para números de Reynolds mais elevados do que 5000 é no máximo de 2%, enquanto que para números de Reynolds inferiores, a diferença aumenta significativamente e atinge 10%. A hipótese mais razoável para a maior discrepância para número de Reynolds menores é a possível influência de instabilidades do escoamento no experimento, fazendo com que o regime seja não-permanente. Conclui-se que o método numérico proposto é validado para a região superior do intervalo analisado. Além da validação pelo ensaio de modelo, medições de campo foram executadas, a fim de confirmar os resultados numéricos. As medições de perda de carga no núcleo do estator de um hidrogerador real era um desafio. No entanto, apesar de todas as dificuldades e consideráveis incertezas da medição campo, o comportamento das curvas de tendência ficou alinhado com resultados numéricos.

Finalmente, uma série de cálculos numéricos, variando parâmetros geométricos do design da entrada do duto de ar e dados operacionais, foram executados a fim de se obter curvas de tendência para coeficientes de perda de carga (resultados deste trabalho) a serem aplicadas diretamente à rotinas de cálculos analíticos de sistemas completos de ventilação de hidrogeradores. Paralelamente à isso, o cálculo térmico numérico foi executado na simulação do protótipo, a fim de se definir o coeficiente de transferência de calor por convecção.

Palavras-chave: Hidrogerador. Sistema de Arrefecimento. Perdas de Ventilação. Troca de Calor. Cálculo Numérico. CFD. Medições de Campo.

Abstract

The objective of the present work is to determinate the pressure drop and the heat transfer coefficients, normally applied to analytical calculations of hydrogenerators thermal design, obtained by applying numerical calculation (Computational Fluid Dynamics – CFD) and validated by experimental results and field measurements. The object of study is limited to the most important region of the ventilation system (the cooling air ducts of stator core) to get numerical results of heat transfer and pressure drop coefficients, which are impacted mostly by the entrance of air ducts.

The numerical calculations considered three-dimensional, steady-state, incompressible and turbulent flow; and were based on the Finite Volume methodology. The turbulent flow computations were carried out with procedures based on RANS equations by selecting k-omega SST (Shear-Stress Transport) as turbulence model. Grid quality metrics were monitored and the uncertainties due to discretization errors were evaluated by means of a grid independence study and application of an uncertainty estimation procedure based on Richardson extrapolation.

The validation of numerical method developed by the present work (specifically to simulate the flow dynamics behavior and to obtain numerically the pressure drop coefficient of the airflow to enter and pass through the Stator Core Air Duct in a hydrogenerator) is performed by comparing the numerical results to experimental data published by Wustmann (2005). The reference experimental data were obtained by a model test. The comparison between numerical and experimental results shows that the difference of pressure drop for Reynolds numbers higher than 5000 is 2% at maximum, while for lower Reynolds numbers, the difference increases significantly and reaches 10%. It is presented that the most reasonable hypothesis for higher discrepancy at lower Reynolds numbers can be assigned to the experiment's non-steady-state condition. It is to conclude that the proposed numerical method is validated for the upper region of the analyzed range. Additionally to the model test validation, field measurements were executed in order to confirm numerical results. Measurements of pressure drop in the stator core of a real hydrogenerator were a challenge. Nevertheless, despite all the difficulties and considerable high field measuring uncertainties, trend curves behavior are similar to numerical results.

Finally, series of numerical calculation, varying geometrical parameters of the air-duct inlet design and operational data, were done in order to obtain pressure drop coefficients trend curves to be directly applied to analytical calculation routines of whole hydrogenerator

ventilation systems. Parallel to it, thermal numerical calculation was executed in the prototype simulation in order to define the convective heat transfer coefficient.

Keywords: Hydrogenerator. Cooling System. Ventilation Losses. Heat Transfer. Numerical Calculation. CFD. Field Measurements.

Table of Contents

List of Figures	11
List of Tables	16
List of Symbols	17
List of Abbreviations and Acronyms	20
1 Introduction	21
1.1 Hydrogenerators	21
1.1.1 Hydrogenerators Cooling System	22
1.1.2 Hydrogenerators Ventilation Systems.....	23
1.1.3 Impact of Ventilation on Hydrogenerators Design	24
1.1.4 Verifications of Ventilation System	25
1.1.5 Model Tests	28
1.1.6 Field Measurements.....	29
1.1.7 Numerical Calculation	29
1.2 Objective	30
1.3 Object of Study.....	30
2 Literature Review	33
2.1 Experimental Analysis	33
2.2 Numerical Analysis and Experimental Validation	35
3 Computational Procedure.....	41
3.1 Solution Strategy.....	41
3.2 The Numerical Method.....	41
3.2.1 Governing Equations.....	42
3.2.2 The Finite Volume Method	43
3.2.3 Spatial Discretization Scheme	46
3.2.3.1 Taylor Series Expansion.....	47
3.2.3.2 Upwind Interpolation (UDS)	48
3.2.3.3 High Resolution Scheme.....	50
3.2.4 Shape Functions	50
3.2.5 Pressure-Velocity Coupling	52
3.2.6 Solution of Linear Equation Systems	54
3.2.7 Turbulence	56
3.2.7.1 Reynolds-Averaged Navier–Stokes (RANS) Equations	57
3.2.7.2 The $k - \varepsilon$ Model	61
3.2.7.3 The $k - \omega$ Model	62

3.2.7.4	The SST $k - \omega$ Model	63
3.2.7.5	Near-Wall Treatments	64
3.2.7.6	Summary of Turbulence Models	65
3.2.8	Discretization Grid.....	68
3.2.8.1	Grid Quality	69
3.2.8.2	Grid Quality Metrics.....	69
3.2.8.3	Grid Quality Metrics – Aspect ratio.....	70
3.2.8.4	Grid Quality Metrics – Orthogonality and Skewness	70
3.2.8.5	Grid Quality Metrics – Others Individual Cell Metrics	72
3.2.8.6	Grid Quality Metrics – Metrics for Domain Regions	72
4	Reference Laboratory and Field Measurements Data	74
4.1	Reference Laboratory Experimental Data from the Literature	74
4.2	Reference Field Measurement Data of Prototype	80
4.2.1	Development of Measurement Procedures and Devices	80
4.2.2	Installation of Measurement Devices at Site.....	81
4.2.3	Field Measurements.....	85
4.2.3.1	Measurement Instruments and Devices Arrangement.....	87
4.2.3.2	Field Measurements Uncertainty Analysis	89
4.2.3.2.1	Uncertainties of Evaluation Type A	89
4.2.3.2.2	Uncertainties of Evaluation Type B	90
4.2.3.2.3	Uncertainties Combination	91
4.2.3.3	Field Measurements Results	93
5	Application to the Model Test and Prototype Geometries and Conditions	96
5.1	Numerical Calculation Applied to the Model Test.....	96
5.1.1	Computational Domain.....	96
5.1.2	Boundary Conditions.....	98
5.1.3	Discretization Grid.....	99
5.1.3.1	Verification of Grid Quality Metrics	101
5.1.3.2	Grid Independence Study	103
5.1.3.3	Estimation of Uncertainty due to Discretization Errors	104
5.2	Numerical Calculation Applied to the Prototype.....	106
5.2.1	Computational Domain.....	106
5.2.2	Boundary Conditions.....	111
5.2.3	Discretization Grid.....	112
5.2.3.1	Verification of Grid Quality Metrics.....	114

6	Results	116
6.1	Numerical Method Validation by Model Test	116
6.1.1	Flow Field Results	116
6.1.2	Experimental and Numerical Data Comparisons	118
6.2	Numerical Results Applied to the Prototype.....	121
6.2.1	Flow Field Results	121
6.2.2	Field Measurements and Numerical Data Comparison	125
6.3	Determination of Coefficients	126
6.3.1	Pressure Drop Coefficient	127
6.3.2	Convective Heat Transfer Coefficient	136
7	Conclusions	142
7.1	Scope for Future Work	144
	References	145
	Appendix	150
	Appendix A: Uncertainty of Pressure Measurements.....	150
	Appendix B: Uncertainty of Velocity Measurements.....	151
	Appendix C: Numerical Results of Pressure Standard Uncertainties	152
	Appendix D: Numerical Results of Velocity Standard Uncertainties	153
	Appendix E: Numerical Results of Pressure Difference Standard Uncertainties	154

List of Figures

Figure 1.1 - Example of hydrogenerator – Sanxia (Kramer, Haluska Jr., & Namoras, 2001).	21
Figure 1.2 - Axial-Radial ventilation systems (Meyer, 2006).	23
Figure 1.3 - Radial-Radial ventilation system (Meyer, 2006).	24
Figure 1.4 - Fluid Dynamics Network model of ventilation system (Meyer, 2006).	26
Figure 1.5 - Example of fluid dynamics calculation result (Meyer, 2006).	27
Figure 1.6 - Application of calculation methods (Ujiie, Arlitt, & Etoh, 2006).	30
Figure 1.7 - Typical stator core including stator winding bar.	31
Figure 1.8 - Costs of hydrogenerator components (Kramer, Haluska Jr., & Namoras, 2001).	32
Figure 2.1 - Comparison of pressure drop coefficients – experimental and numerical results (Wustmann, 2005).	34
Figure 2.2 - PIV measurements on stator core air ducts (Hartono, Golubev, Moradnia, Chernoray, & Nilsson, 2012a).	35
Figure 2.3 - Design proposed by Yanagihara & Rodrigues (1998).	36
Figure 2.4 - Velocity profile measurements on stator core air ducts outlet (Moradnia, Chernoray, & Nilsson, 2012a).	39
Figure 3.1 - Typical control volume definition (ANSYS CFX, 2012).	44
Figure 3.2 - Typical grid element (ANSYS CFX, 2012).	45
Figure 3.3 - Typical control volume and notation for Cartesian 2D grid (Ferziger & Peric, 2002).	48
Figure 3.4 – Left: Discretized solution. Right: Smeared solution (numerical diffusion) by first order UDS. (ANSYS CFX, 2012).	49
Figure 3.5 – Left: Discretized solution. Right: Numerical oscillation by second order UDS. (ANSYS CFX, 2012).	50
Figure 3.6 – Hexahedral Element. (ANSYS CFX, 2012).	51
Figure 3.7 - 1D grid example of co-located arrangement (Versteeg & Malalasekera, 2007).	52
Figure 3.8 - 1D grid example of staggered arrangement (Versteeg & Malalasekera, 2007).	54
Figure 3.9 – Behaviour of Algebraic Multigrid method (ANSYS CFX, 2012).	56
Figure 3.10 – Example of time averaging of turbulence used on RANS models (ANSYS CFX, 2012).	58

Figure 3.11 – Subdivisions of the Near-Wall Region (ANSYS CFX, 2012).....	64
Figure 3.12 - Skewness on a hexaedrical cell (ANSYS ICEM, Inc., 2012)	71
Figure 3.13 - Improvement of orthogonality (Ferziger & Peric, Computational Methods for Fluid Dynamics, 2002).....	71
Figure 3.14 - Left: warping on hexaedrical cell (Ferziger & Peric, 2002); Right: warping on tetraedrical cell (Tu, Yeoh, & Liu, 2013).....	72
Figure 3.15 - Dragged vertex on hexaedrical cell. Left: Ferziger & Peric (2002); Right: ANSYS ICEM CFD 14.5 (ANSYS ICEM, Inc., 2012).....	72
Figure 4.1 – Experiment layout. Based on Wustmann (2005).....	74
Figure 4.2 – Rotor applied on model test (Wustmann, 2005).....	76
Figure 4.3 – Stator applied on model test (Wustmann, 2005).....	77
Figure 4.4 – Model test wedge applied to the numerical simulation (Wustmann, 2005).....	77
Figure 4.5 – Experimental complete model (Wustmann, 2005).....	78
Figure 4.6 – Bypass circuit for flow volume rate measurement (Wustmann, 2005).....	79
Figure 4.7 – Pressure measurements on model test (Wustmann, 2005).....	79
Figure 4.8 – Workshop tests of pressure wall tapping installation.	80
Figure 4.9 – Instruction of pressure wall tapping installation at site.....	81
Figure 4.10 – Scheme of pressure tapping installation.	81
Figure 4.11 – Teles Pires UG05 Stator at generator's pit.	82
Figure 4.12 – Stator frame inspection window.	83
Figure 4.13 – Axial location of pressure tapping installation.	83
Figure 4.14 – Tangential location of pressure tapping installation.	84
Figure 4.15 – Example of identified pressure tapping installation.....	85
Figure 4.16 – Example of field measurement execution.	86
Figure 4.17 – Delimited areas prepared for measurements of air volume flow passing through one air cooler.	87
Figure 4.18 – Measurement instruments and devices arrangement.	89
Figure 4.19 – Measured results for pressure drop of nine working measuring devices.....	94
Figure 4.20 – Measured results for pressure drop of nine working measuring devices, mean curve and uncertainties.	94

Figure 4.21 – Measured results for total flow volume rate with uncertainties.	95
Figure 5.1 – Computational domain.	96
Figure 5.2 – Geometric parameters of stator core air duct model.....	97
Figure 5.3 – Boundary conditions applied to stator core air duct model.	99
Figure 5.4 – Grid applied to the numerical simulations.	100
Figure 5.5 – O-Grid blocks applied for boundary layers cells.	101
Figure 5.6 – Grid of air duct entrance region (lowest aspect ratio colored).	102
Figure 5.7 – Air computational domain of numerical calculation applied to the prototype. ...	106
Figure 5.8 – Detail of axial connection between air ducts in the region of frontal wedges. ..	107
Figure 5.9 – Solid bodies computational domain (assimetric quarter) of numerical calculation applied to the prototype.	108
Figure 5.10 – Air volume geometric parameters.	109
Figure 5.11 – Boundary conditions applied to the air volume of prototype simulation.	112
Figure 5.12 – Grid applied to the prototype numerical simulations.	113
Figure 5.13 – O-Grid blocks applied for boundary layers cells.	113
Figure 5.14 – Grid of air duct region.....	115
Figure 6.1 – Graphhic Output – Static pressure.....	117
Figure 6.2 – Graphhic Output – Volume flow streamlines colored by velocities.	117
Figure 6.3 – Comparisons between experimental and numerical results of flow volume rate.	118
Figure 6.4 – Comparisons between experimental and numerical results of pressure drop...119	
Figure 6.5 – Comparisons between experimental and numerical results of pressure drop coefficient.....	120
Figure 6.6 – Prototype simulation grapghic output – Static pressure.....	122
Figure 6.7 – Prototype simulation grapghic output – Streamlines based on core vortex colored by velocity.	123
Figure 6.8 – Prototype simulation grapghic output –Volume flow streamlines colored by velocities – Detail.....	123
Figure 6.9 – Prototype simulation grapghic output – Air temperature.	124
Figure 6.10 – Prototype simulation grapghic output – Heat flux.	125

Figure 6.11 – Comparisons between field measurements and numerical results of pressure drop.....	126
Figure 6.12 – Pressure drop coefficient as function of velocity ratio for 60° frontal wedge without relief.....	127
Figure 6.13 – Velocity vectors for 60° frontal wedge without relief ($V_r/V_t = 0.37$).	128
Figure 6.14 – Velocity vectors for 60° frontal wedge without relief ($V_r/V_t = 1.00$).	128
Figure 6.15 – Velocity vectors for 60° frontal wedge without relief ($V_r/V_t = 1.67$).	129
Figure 6.16 – Volume flow streamlines colored by velocities for 60° frontal wedge without relief ($V_r/V_t = 0.37$).	130
Figure 6.17 – Volume flow streamlines colored by velocities for 60° frontal wedge without relief ($V_r/V_t = 1.00$).	130
Figure 6.18 – Volume flow streamlines colored by velocities for 60° frontal wedge without relief ($V_r/V_t = 1.67$).	131
Figure 6.19 – Pressure drop coefficient as function of velocity ratio for 60° frontal wedge with 45° relief.....	132
Figure 6.20 – Velocity vectors for 60° frontal wedge with 45° relief ($V_r/V_t = 0.07$).....	133
Figure 6.21 – Velocity vectors for 60° frontal wedge with 45° relief ($V_r/V_t = 0.28$).....	133
Figure 6.22 – Velocity vectors for 60° frontal wedge with 45° relief ($V_r/V_t = 1.55$).....	134
Figure 6.23 – Volume flow streamlines colored by velocities for 60° frontal wedge with 45° relief ($V_r/V_t = 0.07$).....	135
Figure 6.24 – Volume flow streamlines colored by velocities for 60° frontal wedge with 45° relief ($V_r/V_t = 0.28$).....	135
Figure 6.25 – Volume flow streamlines colored by velocities for 60° frontal wedge with 45° relief ($V_r/V_t = 1.55$).....	136
Figure 6.26 – Convective heat transfer coefficient as function of radial velocity in air channels.	137
Figure 6.27 – Temperature of air and solid bodies ($V_r = 1.00 \text{ m.s}^{-1}$).....	138
Figure 6.28 – Temperature of air and solid bodies ($V_r = 5.66 \text{ m.s}^{-1}$).....	138
Figure 6.29 – Temperature of air and solid bodies ($V_r = 12.00 \text{ m.s}^{-1}$).....	139
Figure 6.30 – Temperature of insulation surfaces ($V_r = 1.00 \text{ m.s}^{-1}$).	140
Figure 6.31 – Temperature of insulation surfaces ($V_r = 5.66 \text{ m.s}^{-1}$).	140

Figure 6.32 – Temperature of insulation surfaces ($V_r = 12.00 \text{ m.s}^{-1}$).	141
--	-----

List of Tables

Table 1.1 – Hydrogenerators ventilation system according to IEC 34-6	24
Table 3.1 – Classification of turbulence models (Versteeg & Malalasekera, 2007).	60
Table 3.2 – Turbulence models applied in the reference works.	66
Table 3.3 – Comparisons between $k - \varepsilon$ and SST $k - \omega$ turbulence models.	67
Table 4.1 – Reported constant values for experimental data (Wustmann, 2005)	75
Table 4.2 – Slot location of pressure tapping installation	84
Table 4.3 – Field measurement instruments and devices for pressure.....	88
Table 4.4 – Others field measurement instruments.....	89
Table 4.5 – Field measurement instruments standard uncertainties.....	90
Table 4.6 – Extra standard uncertainties	91
Table 5.1 – Geometric data of numerical calculation.....	97
Table 5.2 – Grid quality metrics obtained for validation simulation	102
Table 5.3 – Grid quality metrics obtained for validation simulation (only air duct)	103
Table 5.4 –Summary of the grid independence study	104
Table 5.5 – Estimation of uncertainty due to discretization	105
Table 5.6 – Material data of numerical simulation applied to the prototype.	108
Table 5.7 – Geometric data of air volume domain.....	110
Table 5.8 – Grid quality metrics obtained for prototype simulation.....	114
Table 5.9 – Grid quality metrics obtained for prototype simulation (only air duct).....	114

List of Symbols

English Symbols

a	acceleration
A	area
c_p	specific heat constant pressure
c_v	specific heat constant volume
d, D	diameter
D_h	duct hydraulic diameter
E	energy
f	function
F	force
g	acceleration of gravity
h	duct width, heat-transfer coefficient by convection, enthalpy
I	electrical current
k	thermal conductivity
L	characteristic length
m	mass
$\dot{m}$	mass rate of flow
M, N	amount of elements
p, P	pressure
p_i	power index
q	heat-transfer rate
q''	heat-transfer rate per unit area
Q	volume flow rate
R	gas constant
s	entropy, experimental standard deviation
S	source or sink of scalar quantity
t	time
T	temperature
u, v, w	Cartesian velocity components
U	relative air humidity
v	specific volume
V	velocity, volume

x, y, z	Cartesian coordinates
X, Y, Z	generic variables

Dimensionless Groups

Ec	Eckert number, $V^2/c_p \Delta T$
Fr	Froude number, $V^2/g L$
Ma	Mach number, V/a
Nu	Nusselt number, $q_w L/k \Delta T$
Pe	Peclet number, $RePr$
Pr	Prandtl number, $\mu c_p /k$
Re	Reynolds number, $\rho VL/\mu$

Greek Symbols

α	thermal diffusivity
β	volumetric thermal expansion coefficient
Δ	variation
γ	specific-heat ratio
δ	velocity boundary layer
δ^*	displacement boundary layer
δ_T	temperature boundary layer
δ_{ij}	Kronecker delta, $\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$
ϵ_{ij}	strain-rate tensor
λ	second viscosity coefficient
η	similarity variable, efficiency, rotational speed
μ	dynamic viscosity, standard uncertainties
μ_t	turbulent viscosity
ν	kinematic viscosity
ν_t	kinematic turbulent or eddy viscosity
π	3.14159...
$\emptyset$	scalar quantity
Γ	diffusivity of scalar quantity
Γ_t	turbulent or eddy diffusivity
Φ	dissipation function
ψ	stream function
ρ	density

τ_{ij}	stress tensor
ω	vorticity
Ω	angular velocity
ζ	pressure drop coefficient

Subscripts

∞	far field
0	initial or reference value
c	cross section
$comb$	combined
$cond$	conduction
$conv$	convection
$crit$	critical
e	freestream, boundary-layer edge
f	fluid
gen	generated
i, j, k	Cartesian coordinates index, counter index
$loss$	lost
m	mean
n	normal
rad	radiation
w	wall
x	at position x

Superscripts

—	time mean
'	differentiation
*	dimensionless variable
+	law-of-the-wall variable

List of Abbreviations and Acronyms

AR	Aspect Ratio
BDS	Backward Difference Scheme
CDS	Central Difference Scheme
CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
CTA	Constant Temperature Anemometry
DNS	Direct Numerical Simulation
FDS	Forward Difference Scheme
HPP	Hydro Power Plant
HWA	Hot-Wire Anemometry
IEC	International Electrotechnical Commission
IC	International Code
LDA	Laser Doppler Anemometry
LES	Large Eddy Simulation
LPTN	Lumped Parameter Thermal-Network
MME	“Ministério de Minas e Energia” (Ministry of Mines and Energy)
MG	Multigrid
MT	Mato Grosso (Brazilian State)
PA	Pará (Brazilian State)
PIV	Particle Image Velocimetry
RANS	Reynolds-Averaged Navier–Stokes
RAM	Random Access Memory
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations
SST	Shear-Stress Transport
UDS	Upwind Differencing Scheme
UG	“Unidade Geradora” (Generation Unit)
VDE	“Verband der Elektrotechnik, Elektronik und Informationstechnik” (Association for Electrical, Electronic and Information Technologies)
VDI	“Verein Deutscher Ingenieure” (The Association of German Engineers)

1 Introduction

In 2013, with 88,966 MW of installed capacity, hydropower represents 67% of Brazilian power grid. According to the Brazilian Decennial Plan for Energy Expansion for 2020 (MME, 2011) this value shall be increased to 115,123 MW and will account for 67.3% of power generation.

Considering the growing demand for electricity and the representativeness of the hydroelectricity generation for the country, it is natural that studies of ways to optimize the use of hydraulic resources earn more importance. The study of thermal design of hydrogenerators is included in this context.

1.1 Hydrogenerators

Generator and Moto-generator differ between them according to its functionality (Wiedemann & Kellenberger, 1967). The first one converts mechanical energy into electrical energy and the second one does the same, but also the opposite. When applied in hydroelectric power plants, they are known as Hydrogenerators and Moto-Hydrogenerators. The object of study of this work is the ventilation system of those machines, which will be called just Hydrogenerators from now on.

The components of a hydrogenerator which impact the ventilation design are located both in the stator as in the rotor of the machine, as presented by Kramer, Haluska Jr., & Namoras (2001).

Figure 1.1 - Example of hydrogenerator – Sanxia (Kramer, Haluska Jr., & Namoras, 2001).

- 1) Rotor Hub: structure made of welded carbon steel which, due to its geometry, contributes in boosting and directing of required air for hydrogenerator cooling.
- 2) Rotor Rim: structure resulting from stacked and pressed plates of high strength steel which, due to its geometry (creating radial ducts), also contributes in boosting and directing of required air for hydrogenerator cooling, by its rotational movement.
- 3) Poles: compound of a steel core (stacked and pressed plates) wrapped in copper coil and insulating material. Due to the electric current flowing through the coil, it dissipates heat by Joule effect, which must be removed by the ventilation system. Considering the fact they are salient poles fixed on the rotor (with air occupying the spaces between them), they also contribute in boosting and directing of required air for hydrogenerator cooling.
- 4) Stator Winding: compound of conductors (bars or coils) of copper and insulating material. Due to the electric current flowing through the conductors, it also loses heat by Joule effect, which must be removed by the ventilation system.
- 5) Stator Core: structure resulting from stacked and pressed sheets of silicon electrical steel, which is designed to create ducts for the airflow in the radial direction.
- 6) Stator Frame: structure made of welded carbon steel designed to guide the air from the stator core to the heat exchangers.
- 7) Heat Exchangers: air-to-water type, responsible for removing heat from the warm air and returning it cooled to the system.
- 8) Air Guides: carbon steel structure (or fiberglass) responsible for guiding the air in the generator housing and delimiting the boundaries of suction and pressure regions of ventilation system.

Depending on the ventilation system type, components may exist with the sole and exclusive function of boosting the air of the ventilation system: e.g. radial or axial fans (fixed to the hydrogenerator shaft), or motor-fans, which operate independently mounted on stationary structure of the hydrogenerator.

1.1.1 Hydrogenerators Cooling System

The electromagnetic losses of a hydrogenerator are responsible for the heat generation, which shall be removed from the machine mainly by convection promoted by cooling system, which uses a fluid acting on surfaces. The cooling system, when compound of an

air circulation system, driven by the hydrogenerator's shaft (or external motor-fans) is called: "Ventilation System".

There is also the possibility of cooling the hydrogenerator partially with application of pure water as cooling fluid, which flows through internal channels into most critical components: the electrical conductors. However, this kind of cooling system is applied very rarely due to its high costs, and it is not covered by this study.

1.1.2 Hydrogenerators Ventilation Systems

Ventilation systems for medium and large size hydrogenerators can be classified into two groups (Meyer, 2006):

- Axial-Radial Ventilation
- Radial-Radial Ventilation

The names refer to the main direction in which the air is forced to flow in the rotor and the stator, respectively.

In Axial-Radial ventilation systems, the air is introduced into the rotor (space between poles) axially driven by fans, which are fixed to the rotor shaft (radial or axial fans) or work independently (motor-fans), and follows in radial direction through air ducts of stator.

Figure 1.2 - Axial-Radial ventilation systems (Meyer, 2006).

In Radial-Radial ventilation system (also known as ventilation through rotor rim or easily as "RIM Ventilation"), both the rotor hub and rim have the function of fans. The air has to change from axial to radial direction as soon as it pass through the openings of rotor hub and follows in radial direction through rotor rim air ducts, spaces between poles and air ducts of the stator.

Rim Ventilation

Figure 1.3 - Radial-Radial ventilation system (Meyer, 2006).

Wiedemann & Kellenberger (1967) presented a generic classification of cooling systems for electrical machines according to the German standard VDE 0530-6 (1996). Considering the corresponding international standard, IEC 34-6 (1991), the presented ventilation systems above can be classified as shown in Table 1.1.

Table 1.1 – Hydrogenerators ventilation system according to IEC 34-6

Ventilation System	IC – According to IEC 34-6
Axial Fan	IC8A1W7
Radial Fan	IC8A1W7
Motor - Fan	IC8A5W7
Rim Ventilation	IC8A1W7

1.1.3 Impact of Ventilation on Hydrogenerators Design

Several materials used in the construction of hydrogenerators can withstand only a certain maximum temperature (especially insulating materials) without jeopardizing its properties or affecting its lifetime. Therefore, the detailed study of the ventilation system is essential to prevent damages arising from improper thermal effects.

To work, the ventilation system needs energy. When driven by the shaft of generator, the mechanical energy required to the air circulation is provided by the turbine coupled to the hydrogenerator. The same occurs when the ventilation system is driven by external motor-fans, whose power shall be considered in the calculation of hydrogenerator efficiency. The total amount of friction losses due to ventilation is relatively low, accounting for only a few tenths of percent of hydrogenerator efficiency, but considering that a

hydrogenerator is designed to last for several decades, the ventilation study becomes relevant for the return of capital employed in a hydroelectric project.

The design of the ventilation system must then ensure that the temperature limits for the respective applied materials are not exceeded, and, at the same time, that the total amount of ventilation losses are minimized in order to maximize the efficiency of the hydrogenerator. An appropriated and adequate ventilation system design is a classic optimization problem, where high precision on calculation procedures is essential.

The complexity of optimization of a ventilation system increases with the size of the hydrogenerator. The electromagnetic losses, which generate heat to be removed by the ventilation system, become greater with the size of the machine.

As a first approximation, it can be asserted that the losses generated in the hydrogenerator (E_{loss}) are proportional to its physical size. Keeping certain dimensional proportionality, the machine physical size and hence the generated losses are proportional to its volume, or to the cube of a machine reference dimension.

$$\sum E_{loss} \propto L^3 \quad (1.1)$$

As the ventilation system works primarily by convection, analogously one can imagine that the external surface and the dissipated heat of this volume are proportional to the square of a machine reference dimension.

$$q_{Conv} \propto L^2 \quad (1.2)$$

Thus, for large hydrogenerators, the ratio of losses by heat generation, to the dissipation through convection at surfaces, increases linearly with a machine reference dimension (Ujii, Arlitt, & Etoh, 2006).

$$\frac{\sum E_{loss}}{q_{Conv}} \propto \frac{L^3}{L^2} = L \quad (1.3)$$

1.1.4 Verifications of Ventilation System

Usually the verification of a ventilation system is performed by applying one-dimensional analytical calculation tools, which rely on a LPTN (Lumped Parameter Thermal-Network). Analytical LPTN analysis can be subdivided into two main calculations (Boglietti,

Cavagnino, Staton, Martin Shanel, & Mejuto, 2009): the first one applied to the airflow (Fluid Dynamics Network) and, the second one, to the heat flow (Thermal Network).

The Fluid Dynamics Network is based on the first law of thermodynamics where the balance of mechanical energy (kinetic and potential) can be applied. Considering the constant gravitational field and incompressible fluid, it is concluded that the sum of pressure in Fluid Dynamics Network is constant.

$$\Delta p_{gen} = - \sum \Delta p_{loss} \quad (1.4)$$

The Fluid Dynamics Network is then idealized with pressure sources (Δp_{gen}) and flow resistances (Δp_{loss}).

Figure 1.4 - Fluid Dynamics Network model of ventilation system (Meyer, 2006).

Applying the continuity equation at each node of the network, we have:

$$\sum \dot{m} = 0 \quad (1.5)$$

The Figure 1.5 shows the variation of pressure in a ventilation system of a hydrogenerator. As commonly made for the global calculation of hydrogenerators ventilation systems, the diagram shows only a discretized evolution of pressure along the flow path. Nevertheless,

localized and distributed pressure drops are accounted on the discretized terms of each component of the whole ventilation system.

Figure 1.5 - Example of fluid dynamics calculation result (Meyer, 2006).

The pressure drop for each component of the system is defined as a ratio of the loss ($\Delta p_{loss}/(\rho g)$) and the velocity head ($V^2/(2g)$), which give us:

$$\Delta p_{loss} = \frac{\rho}{2} V^2 \zeta \quad (1.6)$$

The pressure drop coefficients (ζ) commonly applied on hydrogenerators analytical calculations are consolidated on available literature (Idelchik & Fried, 1989) and defined by model tests of standardized geometries. However, these coefficients refer to settings of well-behaved flows (considering only preferential directions of fully developed flows) and cannot describe internal vortices, such as those occurring within a real hydrogenerator.

Just as for the fluid dynamics calculation, the network method is also usually applied for the thermal calculation, where a Thermal Network is idealized with thermal heat sources and thermal resistances.

For the determination of the temperature, it is taken into account the heat conduction through solid bodies, heat transfer by convection and the heating of the cooling fluid itself:

$$\Delta T = \frac{q_{cond} L}{k A_c} + \frac{q_{conv}}{h A_w} + \frac{q}{2 \rho c_p Q} \quad (1.7)$$

From the variables shown in the Equation (1.7), the coefficient of heat exchange by convection (h) has the most complex determination, which can be done by applying the not necessarily linear function of dimensionless numbers:

$$Nu = \frac{h L}{k} = f(Re, Pr) \quad (1.8)$$

Thus, the coefficient of heat exchange by convection (h) is a function of several parameters (e.g. velocity, density, fluid dynamics viscosity, geometry, etc.) with a not necessarily linear relationship:

$$h \approx h(V, \rho, \eta, \text{geometry, etc.}) \quad (1.9)$$

The determination of those coefficients is usually performed empirically or, more recently, by application of numerical calculation (as described in chapter 1.1.7).

By the written above, it is easy to realize the importance of the pressure drop and heat exchange coefficients accuracy to get good results when applying analytical Fluid Dynamics and Thermal Networks calculations (LPTN) for a hydrogenerator ventilation system. Researchers and the industry have been studying the determination of those coefficients for some hydrogenerator components and different operational conditions, applying analytical, experimental and numerical methods, as detailed described in the literature review (chapter 2). Nevertheless, this subject is far to be considered as an overcome issue and there is still a huge lack of accurate data. This work aims to be a contribution on this matter, presenting the determination of these coefficients for a specific component of the machine (described in chapter 1.3) and a range of operational conditions, applying a numerical method, validated by experimental data.

1.1.5 Model Tests

In order to defined and calibrate pressure drop and heat transfer coefficients more appropriately to the geometries, operational characteristics and boundary conditions commonly encountered in hydrogenerators, manufacturers and researchers have applied specific models tests.

Researchers from universities in Germany, Sweden and Canada have conducted model tests especially dedicated to typical particularities of hydrogenerators, such as internal volumes of the rotor hub, air ducts of rotor rim, volume between poles, air ducts of stator core, volume of stator winding overheads region and volumes of generator housing (chapters 2.1 and 2.2).

There are also some records of model tests, which simulate the airflow within a whole hydrogenerator, usually ordered and financed by manufacturers or investors of megaprojects on hydropower business, who wanted to minimize the technical risk of ventilation system of large hydrogenerators (chapter 2.1).

1.1.6 Field Measurements

After installation, a hydrogenerator shall be commissioned by the manufacturer, who, among others, typically performs measurements in the ventilation system, getting results like flow rates, pressures and temperatures.

Nevertheless, these results are limited to be used for further studies, since: 1) the temperature can be measured only in discrete locations; 2) air volume flow results show the integral values of the whole system, not reflecting valuable information about the airflow distribution; 3) the values of ventilation losses can be calculated only indirectly (e.g. calorimetric method).

1.1.7 Numerical Calculation

Due to the time and cost constraints, model tests are performed only for special researches or during development of megaprojects in hydropower business. The direct application of specific tests results of hydrogenerator singularities (mentioned in chapter 1.1.5) faces deviations of similarities regarding geometry and Reynolds number. Therefore, extrapolations of definitions obtained from model tests to different geometries of real projects are limited.

Thus, the application of numerical calculation (CFD - Computational Fluid Dynamics) appears as a valid alternative. It is based on physical principles and the equations of conservation of mass, momentum and energy. However, despite of CFD being an efficient computational tool, it is a method where the equations are prepared and solved iteratively for a discrete system with a finite number of elements. In addition, the equations must be approximated to the reality by mathematical models (e.g. turbulence models, as discussed in chapter 3.2.7). Therefore, a proper matching between CFD and experimental results is of utmost importance, as done in previous studies.

There are some CFD studies, where the authors were aiming to get an overview of the ventilation system. Models of entire hydrogenerators were applied considering high level of simplifications and the results were compared with experimental data, field measurements or analytical results (chapter 2.2). Therefore, numerical, analytical and experimental methods can complete each other in the study of hydrogenerators ventilation systems.

Method \ Target	Field Calculation Methods	Analytical Methods First Principles
Calculation of Losses	Loss Calculation	
Calculation of Flows	Computational Fluid Dynamics	Volume flow rates using pressure loss coefficients
Calculation of Temperatures	Computational Fluid Dynamics	Heat Source Networks using Heat Transfer Coefficients

Figure 1.6 - Application of calculation methods (Ujiie, Arlitt, & Etoh, 2006).

1.2 Objective

The purpose of present work is to determinate the pressure drop and the heat transfer coefficients, normally applied in analytical calculations of hydrogenerators thermal design, of Rim Ventilation machines. These coefficients shall be obtained by applying numerical calculation (CFD) and validated by experimental results and field measurements.

1.3 Object of Study

As described in chapter 1.1, a hydrogenerator consists of several components that affect the ventilation system. Some of them are heat sources (e.g. pole and stator windings) and others have the function of driving and/or guiding the circulating air (e.g. rotor hub, rotor rim and stator core respectively).

The study of the entire ventilation system, to determinate the pressure drop and heat transfer coefficients of all components, demands an enormous effort, what becomes unfeasible for a work as the present one. Thus, the present work only focuses on a certain region: the stator core including the winding conductors.

Figure 1.7 - Typical stator core including stator winding bar.

The selection of stator core and winding as object of study was based on the following reasons:

1. **Heat Source:** due to the electromagnetic losses, the highest amount of heat comes from the stator winding and it is dissipated through the stator core. In an optimized machine, the components located in this region are the ones working at temperatures very close to their material thermal limits.
2. **Pressure Drop:** depending on the design of a ventilation system, the stator core can represent the greatest resistance for the airflow in the machine. The energy dissipated by the airflow that leaves the machine rotor and gets into the air ducts of stator core is the main cause of the high pressure drop that is achieved in this region. To minimize significantly the ventilation losses of a hydrogenerator, the stator core (specially the inlet of air ducts) shall be necessarily improved.
3. **Material Costs:** the material applied to the stator winding and stator core, whose amount is defined by the machine power, are especial when compared to the rest of the machine. The stator core is made of silicon electrical steel sheets while the stator winding is made of conductors (bars or coils) made of copper and insulating

material. The high material costs represent at the same time the highest potential of cost reduction to be achieved by the improvements of ventilation system.

Figure 1.8 - Costs of hydrogenerator components (Kramer, Haluska Jr., & Namoras, 2001).

4. Experimental Data: stator core with its air ducts have been studied by some researchers over the last years. Consequently, some reliable model test results are available and can be used for validation of the numerical approach, which will be developed by the present work.
5. Geometry: the geometry of a stator core air duct is relatively simple, what makes the numerical calculation be less time-consuming, while the discretization becomes simplified and the meshes can be well oriented.

2 Literature Review

A literature review about the application of numerical and experimental analysis of hydrogenerators ventilation systems between 1995 and 2012 was done for the present work. During this period, studies including CFD simulations and experimental results of hydrogenerator ventilation systems have been published by researchers and manufacturers mostly in Germany, Sweden, Canada and Brazil.

2.1 Experimental Analysis

Wustmann (2005) reported a model test considering the entire circumference (360° , to include periodical oscillations) of a complete hydrogenerator ventilation system, applying variations of air duct inlet design. Different from previous model tests performed in TU Dresden (Technischen Universität - Dresden, Germany), a bypass was installed in the model in order to improve the measurements of volume flow by means of an orifice plate. The tests were performed by varying the rotational speed and the frontal wedge shape. The following experimental data were obtained:

- 2D and 3D velocity field plots of air duct sections by means of interpolated HWA (Hot-Wire Anemometry) measurements;
- Pressure values along air duct (from inlet to outlet);
- Pressure pulsation along air duct (from inlet to outlet);
- Volume flow, by means of an orifice plate in a bypass circuit;
- Pressure drop coefficients calculated by measured pressure difference between air duct inlet and outlet.

At the final, the experimental results were compiled and the measured pressure drop coefficients were compared to analytical formulation derived from numerical simulations obtained by TU Dresden previous studies.

Figure 2.1 - Comparison of pressure drop coefficients – experimental and numerical results (Wustmann, 2005).

Hartono, Golubev, Moradnia, Chernoray, & Nilsson, (2012a) and (2012b), reported experiments where PIV (Particle Image Velocimetry) was applied to measure the velocity into the stator core air ducts (two-dimensional) and in the generator housing (three-dimensional with consecutive vertical shifting of horizontal measurement planes and interpolation). A significant amount of efforts was spent for improving the quality of the PIV measurements, by applying geometric modifications in order to enable optical access to the measured air ducts. Improvements of field of view were achieved by using specially designed channels and channel baffles, whose sharpened trailing edges reduced the shadow regions up to 85%. Two geometrical variations for the air ducts were tested (curved and straight baffles).

Finally, measurements inside the channels captured the recirculation region very well. Results showed a recirculation region due to the flow separation, consuming half of the duct width for both duct designs. The flow became complex and three-dimensional, with clear dominance of the tangential velocity component just after it leaves the air duct. Flow separation near the inlet of curved baffles was detected. Besides that, highly uneven flow distribution between the stator channels in both configurations was observed.

Figure 2.2 - PIV measurements on stator core air ducts (Hartono, Golubev, Moradnia, Chernoray, & Nilsson, 2012a).

Torriano, et al. (2014) studied the effect of rotation on heat transfer mechanisms in rotating poles of hydrogenerators. Using a simplified scale model with heated poles was possible to measure the temperature distribution on the pole surface and to deduce, through numerical simulations, the heat transfer coefficients. The results show an asymmetric profile in the tangential direction since lower coefficient values were found closer to the trailing edge due to the presence of a flow recirculation zone. Furthermore, the heat transfer profiles indicated that, although fans improve cooling at the top and bottom ends of the pole, the highest coefficient values were found in an intermediate region. This is due to the flow from the fans that enters the inter-pole space and, only after penetrating a certain distance in the axial direction, exits through the air gap and goes around the pole face. By post-processing the numerical results from ANSYS CFX and SYRTHES 4.0, the heat transfer coefficient profile along the pole face has been obtained at six different axial positions along the pole.

2.2 Numerical Analysis and Experimental Validation

CFD codes applied to three-dimensional problems were first developed in the early 1970s, and are still being developed today. First versions of general codes were established in the 1980s, but used mainly by researchers at the time. Commercial codes became available for the hydropower industry in the 1990s, and the application of this tool in hydrogenerator ventilation systems started only at the end of nineties.

Computational Fluid Dynamics simulations for stator core air ducts were performed by Yanagihara & Rodrigues (1998), when the code PHOENICS V2.0 was applied to investigate the influence of geometrical parameters and operating conditions on the pressure drop coefficients. The steady state and bi-dimensional analyzes were performed

considering the $k-\epsilon$ turbulence model with the set of constants suggested by the computational code. The validation of numerical method was performed comparing simulation and experimental results of a simple “T” bifurcation model, which showed differences of pressure drop below 15% for almost all the cases and 24% when working with lower flow rates. Several design and dimensional configurations were simulated and it was concluded that the radial positioning of baffles has the highest impact on the pressure drop (the closer to the stator core inner diameter, the better the air can be guided and the less will the pressure drop). In order to improve the air guidance into the duct, it was proposed to adopt baffles with inclined ends near to stator core inner diameter.

Figure 2.3 - Design proposed by Yanagihara & Rodrigues (1998).

Advantages of CFD simulations applied to a hydrogenerator ventilation system are endorsed by Ujiie, Arlitt, & Etoh (2006). An example of a numerical simulation of complete hydrogenerator ventilation system, validated by experimental data of a scaled model test, is briefly presented. Different mesh resolutions and turbulence models were compared, and results were obtained with a standard $k-\epsilon$ turbulence model. To summarize the agreement of numerical and experimental data, a deviation of global flow rate of 7%, pressure and velocity into the air duct at the average of 5%, were obtained. Finally, it is suggested that numerical results can be used to improve the empirical correlations of analytical calculation methods.

A CFD simulation of complete ventilation system of a hydrogenerator in Brazil (Corumbá IV) was presented by Altea & Meyer (2007). The analysis was performed by applying the solver ANSYS CFX considering a convergence of the global flow rate into the hydrogenerator with a maximum residual at 2×10^{-3} . High level of simplifications was applied and the stator core (object of study of the present work) was assumed as a porous

volume. The comparison between final numerical result (total volume flow) and field measurements showed a deviation of 26%.

Houde, Vincent, & Hudon (2008) published fluid dynamics numerical calculations performed in iterative processes. Global simulations of rotor and stator interaction (with boundary conditions defined by field measurements) were performed applying geometrical simplifications (e.g. porous volume for stator core air ducts and stator winding overhangs) to provide boundary conditions for local simulations of stator core air ducts region. Local simulations were done with detailed geometry, providing results to calibrate the global simulations. The simulations applied the solver ANSYS CFX 11, using RANS (Reynolds-Averaged Navier–Stokes) $k-\epsilon$ turbulence model using wall function. The applied pressure-velocity coupling was done using a Rhie-Chow type interpolation scheme. The simulations were done with upwind scheme and the iterative process used to solve the resulting matrix was based on a first order Euler time marching scheme. The maximum residual at 10^{-4} was considered sufficiently low to provide a good resolution. Results for the volume flow axial distribution behind the stator core had good agreement with field measurements, which did not happen with velocity results due to incompatibilities between simulation and measurement method. The non-uniform axial distribution of recirculation zones into the air ducts can be explained by the axial variation of inlet velocity profile. The same behavior shall occur with the heat transfer coefficients.

A 10 MW hydrogenerator Axial-Radial (with axial fans) ventilation system was numerically simulated by Gregorc (2011). Field measurements (temperature rises, air and water volume flows), performed during the machine commissioning, showed that the ventilation system was not cooling the machine properly. CFD simulations were done in an iterative process, in two steps: first from axial fans, trough poles up to the air gap (for determination of boundary conditions on stator core) and then locally within stator core air ducts including heat transfer effects. The used solver was ANSYS CFX 11, and $k-\epsilon$ (with a wall-function) was the applied RANS (Reynolds-Averaged Navier–Stokes) turbulence model. The iterative process used to solve the resulting matrix was based on the first order Euler time marching scheme. The maximum residual at 10^{-4} was considered sufficiently low. Local fluid dynamics and thermal simulations within stator core air ducts showed highly asymmetrical temperature distribution within the stator's winding-bars: 1) intensified heat transfer at the pressure side of the cooling-duct due to high-intensity vortex within the gap between the rotor and stator; 2) the flow of cooling-air is substantially reduced on the suction-side of the cooling-duct, which remains uncooled.

A fluid dynamics study of an Axial-Radial ventilation system was also reported by Moradnia & Nilsson (2010). A block-structured mesh and the Frozen Rotor were applied.

The convective terms in the momentum equations were discretized using a second-order upwind scheme, while those of the turbulence equations were discretized using the first-order upwind scheme. A detailed study of many RANS models available in OpenFOAM was performed (that is, standard $k-\epsilon$, realizable $k-\epsilon$, RNG $k-\epsilon$, $k-\omega$ SST, Launder-Sharma $k-\epsilon$, Lam-Bremhorst $k-\epsilon$, Launder-Gilson RSTM, Lien cubic $k-\epsilon$, Non-linear Shih $k-\epsilon$, LRR and Spalart-Almaras), applying a backward facing step as a test case. The results of Launder-Sharma $k-\epsilon$ model were quite consistent with the experimental results. Therefore, Launder-Sharma $k-\epsilon$ model was select to perform the simulations. The ability of the numerical code to predict the pressure build-up due to the rotation of the air in the space between rotor poles and stator core was validated by the simulation of a theoretical Couette flow. Four design variations were simulated, showing that the inclusion of air guides and radial blades improved the flow dynamics performance (the initial design was considering the airflow built up only by rotor salient components: e.g. poles). The numerical method could not be validated with experimental data from prototypes or hydrogenerator model tests.

In a following publication (Moradnia P. , 2010), a sequence of a previous study was presented with more detail and some additional design variations applied to the rotor poles. These last variations did not present so much difference between them. The same work was republished (Moradnia & Nilsson, 2011) where some numerical convergence problems seems to have been solved.

A model test of a small-scale electric generator (numerically studied by Moradnia before) was reported by Moradnia, Chernoray, & Nilsson (2011). Volume flow measurements by means of pressure probes and flow distribution by means of smoke visualizations were executed. The previous numerical results for airflow distribution were confirmed by the experimental observations. However, the quantitative results presented less agreement with the numerical simulations, whose deviations may be related to some small geometric differences. The same model test was reported in a following paper (Moradnia, Chernoray, & Nilsson, 2012a) whose the experimental results were compared with further developed simulations, where the rotational speed was varied. Numerical results showed the volume flow increasing almost linearly and the shaft power increasing cubically with the rotational speed. The rotational direction did not affect neither the flow nor the shaft power. In Figure 2.4, right side, the numerical results (black lines) are compared to the range of the scattered measured velocities for all available channels (vertical lines) for four channel rows (horizontal lines).

Figure 2.4 - Velocity profile measurements on stator core air ducts outlet (Moradnia, Chernoray, & Nilsson, 2012a).

Fluid Dynamics numerical simulations applied to ventilation systems of hydrogenerators were performed by Moradnia, et al. (2012b), comparing the codes OpenFOAM and ANSYS CFX. The simulations used 2D and 3D models of a sector of a hydrogenerator region (rotor poles, air gap and stator deflectors) by defining boundary conditions for airflow velocities. The comparisons were applied between the simulation strategies and parameters such as steady and unsteady simulation, boundary conditions, meshes and resolution strategies. The results (rotor torque, radial and tangential velocities, pressure drop and computational time) did not show any significant difference between both codes. Regarding to the rotor-stator interface method, simulations with frozen rotor were always dependent on the interface position. The problem vanishes in sliding grid solutions, as the transient flow in each time step were simulated with new boundary conditions.

Vogt & Lahres (2013) presented an overview of the application of Computational Fluid Dynamics (CFD) and Conjugate Heat Transfer (CHT) to a large external fan-ventilated motor-generator, applying the code ANSYS CFX and $k-\epsilon$ as RANS (Reynolds-Averaged Navier–Stokes) turbulence model. The paper describes the fluid dynamics results of the overall flow into the motor-generator, like flow rate, pressure distribution and ventilation losses, which are subsequently applied as boundary conditions to a CHT model of the stator core and winding conductors. With the CHT localized calculation, it was possible to get thermal results like temperature in the solid bodies. A parametric study about the benefits of static and rotating air guides concerning the ventilation losses was also performed. By the parametric study results and the assumption of a monetary value per power, the financial advantage of the static and rotating air guides application could be proved. At the end of the paper, a brief validation of the presented calculations and methods with field measurements has proven their accuracy and confidentiality, with excellent agreements ($\sim 0.8\%$ for loss, $\sim 2.2\%$ for flow rate and $\sim 5\%$ for temperatures).

A half-scale model of an electric generator was designed and manufactured specifically for detailed experimental and numerical studies of the flow of cooling air through the machine by Moradnia, Golubev, Chernoray, & Nilsson (2013). The experimental measurements include PIV measurements at the rotor inlet, inside the machine, and total pressure measurements at the outlet of the stator channels. Computational Fluid Dynamics (CFD) simulations are performed using two approaches. In the first one, the flow rate at rotor inlet and stator outlet boundary conditions were specified from the experimental data. In the second one, the flow rate is determined from the numerical simulation, independently of the experimental results, yielding predictions differing by 2–7% compared to the experimentally estimated values. The numerical simulations were performed with the OpenFOAM CFD tool, using the cyclic steady-state frozen-rotor concept and Launder-Sharma $k-\epsilon$ model. The numerically predicted flow features agreed very well with the experimental results, for both numerical approaches. Large recirculation regions at the rotor inlet and inside the machine were captured similarly by the experimental measurements and the numerical simulations with a high level of accuracy.

In a following publication, CFD calculation with fully predictive computational fluid dynamics approach was assessed for the flow of cooling air in an axially cooled electric generator (Moradnia, Chernoray, & Nilsson, 2014). In this case, the flow is driven solely by the rotation of the rotor, as in the real application. A part of the space outside the generator is included in the computational domain to allow for the flow of air into and out of the machine. The numerical simulations were performed with the OpenFOAM CFD tool, using the cyclic steady-state frozen-rotor concept and Launder-Sharma $k-\epsilon$ model, as occurred in previous works by the same researcher's team. The numerically predicted flow was compared with experimental data obtained from test model: flow visualizations at the rotor inlet, velocity distribution and total pressure probes. It was found that the numerical approach qualitatively, and largely quantitatively, predicts the same velocity distributions as in the experiment. The numerically predicted flow rates were however lower than that estimated from the experimental data. The differences were considered small, given the many uncertainties in both the numerical and experimental studies, and that they are performed completely independently.

3 Computational Procedure

3.1 Solution Strategy

The numerical method is deeply analyzed and applied in the determination of the pressure drop and the heat transfer coefficients of a hydrogenerator ventilation system. In order to validate the numerical method, the first calculations simulate an experiment (model test) published by TU Dresden (Technischen Universität - Dresden, Germany) and reported by Wustmann (2005), and the numerical results are compared to the experimental ones.

Once numerical method is validated, it can be applied to the prototypes (real hydrogenerators), as long as geometrical design differences between model and prototypes are small and the similarity rules are considered. Results of numerical simulations of prototype can also be compared to field measurements, despite all the difficulties it represents. For the present work, results of numerical simulations of prototype were compared to field measurements performed in a power unit of HPP Teles Pires.

Parametric studies, varying the geometrical parameters of the air-duct inlet design (first without relief, based on the reference model test; and second with 45° relief, based on HPP Teles Pires prototype) and operational data, are done. The goal of such studies is to identify optimized air-duct inlet designs for the hydrogenerator ventilation system by the reduction of pressure drop coefficient (ventilation losses).

The results of numerical simulations for pressure drop coefficients are equated as function of geometrical and operational parameters. The obtained trend curves (results of this work) can be directly applied to analytical calculation routines of whole hydrogenerator ventilation systems.

Parallel to the studies for definition of pressure drop coefficients, thermal numerical calculation was executed in the prototype simulation in order to define the heat transfer coefficient.

3.2 The Numerical Method

The numerical calculation method (CFD), applied to the thermal-fluid problem of this work, solves the governing equations iteratively, for a grid with a finite number of elements, based on the Finite Volume Methodology (Maliska, 2004).

3.2.1 Governing Equations

As was detailed described by Ferziger & Peric (2002), the generic equation for a scalar quantity conservation can be written as follows:

$$\frac{\partial(\rho\phi)}{\partial t} + \text{div}(\rho\phi v) = \text{div}(\Gamma \text{grad}\phi) + S_\phi \quad (3.1)$$

In Cartesian coordinates and tensor notation, the differential form of the equation is:

$$\frac{\partial(\rho\phi)}{\partial t} + \frac{\partial(\rho u_j \phi)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \phi}{\partial x_j} \right) + S_\phi \quad (3.2)$$

The equation above is the starting point for computational procedures in the finite volume method (Versteeg & Malalasekera, 2007), since conservation equations are defined as following:

- Mass conservation, by setting $\phi = 1$, $\Gamma = 0$ and $S_\phi = 0$:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \quad (3.3)$$

- Momentum conservation, by setting $\phi = u_i$, $\Gamma = \mu$ and $S_\phi = -(\partial p / \partial x_i)$:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} \right) - \frac{\partial p}{\partial x_i} \quad (3.4)$$

- Energy conservation, by setting $\phi = T$, $\Gamma = k$ and $S_\phi = -p(\partial u_j / \partial x_j) + \Phi$:

$$\frac{\partial(\rho T)}{\partial t} + \frac{\partial(\rho u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) - p \frac{\partial u_j}{\partial x_j} + \Phi \quad (3.5)$$

The dissipation function is defined by:

$$\Phi = \tau'_{ij} \frac{\partial u_i}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} + p \frac{\partial u_j}{\partial x_j} \quad (3.6)$$

For the present work, the following assumptions are made:

- Incompressible fluid (constant density);

- Other constant fluid properties: viscosity, specific heat and thermal conductivity;
- Steady state dynamic flow and heat transfer;
- Body force and dissipation terms neglected.

Thus, simplifications of conservation equations can be done as following:

- Mass conservation:

$$\frac{\partial(\rho u_j)}{\partial x_j} = 0 \quad (3.7)$$

- Momentum conservation:

$$\frac{\partial(\rho u_j u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} \right) - \frac{\partial p}{\partial x_i} \quad (3.8)$$

- Energy conservation:

$$\frac{\partial(\rho u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{k}{c_p} \frac{\partial T}{\partial x_j} \right) \quad (3.9)$$

3.2.2 The Finite Volume Method

ANSYS CFX uses an element-based finite volume method, which first involves discretizing the spatial domain using a grid. The grid is used to construct finite volumes, which are used to conserve relevant quantities such as mass, momentum and energy.

Despite this work applies a three-dimensional grid, the finite volume method process can be understood by the typical two-dimensional grid shown in the figure below.

Figure 3.1 - Typical control volume definition (ANSYS CFX, 2012).

All solution variables and fluid properties are stored at the nodes (vertices). A control volume (the shaded area) is constructed around each mesh node using the median dual (defined by lines joining the centers of the edges and element centers surrounding the node).

The governing equations (chapter 3.2.1), namely the conservation equation of mass (3.3), momentum (3.4) and energy (3.5), are integrated over each control volume, and Gauss' Divergence Theorem is applied to convert volume integrals involving divergence and gradient operators to surface integrals. If control volumes do not deform in time, then the time derivatives can be moved outside of the volume integrals and the integrated equations become:

- Mass conservation:

$$\frac{d}{dt} \int_V \rho dV + \int_S \rho u_j dn_j = 0 \quad (3.10)$$

- Momentum conservation:

$$\frac{d}{dt} \int_V \rho u_i dV + \int_S \rho u_j u_i dn_j = \int_S \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) dn_j - \int_S p dn_j \quad (3.11)$$

- Energy conservation:

$$\frac{d}{dt} \int_V \rho T dV + \int_S \rho u_j T dn_j = \int_S k \left(\frac{\partial T}{\partial x_j} \right) dn_j + \int_V S_T dV \quad (3.12)$$

Where V and S respectively denote volume and surface regions of integration and dn_j are the differential Cartesian components of the outward normal surface vector. The volume integrals represent source or accumulation terms, and the surface integrals represent the summation of the fluxes (finite volume balance).

Once the conservation equations are integrated for each control volume, the next step in the numerical algorithm is to discretize the volume and surface integrals of each grid single element, like the one shown below.

Figure 3.2 - Typical grid element (ANSYS CFX, 2012).

Volume integrals are discretized within each element sector and accumulated to the control volume to which the sector belongs. Surface integrals are discretized at the integration points (ipn) located at the center of each surface segment within an element

and then distributed to the adjacent control volumes. Because the surface integrals are equal and opposite for control volumes adjacent to the integration points, the surface integrals are guaranteed to be locally conservative.

After discretizing the volume and surface integrals, the integral equations become:

- Mass conservation:

$$V \left(\frac{\rho - \rho_0}{\Delta t} \right) + \sum_{ip} \dot{m}_{ip} = 0 \quad (3.13)$$

- Momentum conservation:

$$V \left(\frac{\rho u_i - \rho_0 u_{i0}}{\Delta t} \right) + \sum_{ip} \dot{m}_{ip} (u_i)_{ip} = \sum_{ip} \left(\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \Delta n_j \right)_{ip} - \sum_{ip} (P \Delta n_i)_{ip} \quad (3.14)$$

- Energy conservation:

$$V \left(\frac{\rho T - \rho_0 T_0}{\Delta t} \right) + \sum_{ip} \dot{m}_{ip} T_{ip} = \sum_{ip} \left(k \left(\frac{\partial T}{\partial x_j} \right) \Delta n_j \right)_{ip} + \overline{S_T} V \quad (3.15)$$

Where $\dot{m}_{ip} = (\rho u_j \Delta n_j)$, V is the control volume, Δt is the time step, Δn_j is the discrete outward surface vector, the subscript ip denotes evaluation at an integration point, summations are over all the integration points of the control volume, and the subscript 0 refers to the old time level. Note that the First Order Backward Euler scheme has been assumed in these equations, although a second order scheme (discussed in chapters 3.2.3.2 and 3.2.3.3) is usually preferable.

3.2.3 Spatial Discretization Scheme

Several schemes for the spatial discretization have been developed over the years and they are presented in detail by the available literature, like Ferziger & Peric (2002), Versteeg & Malalasekera (2007) and Maliska (2004). In the code ANSYS CFX (ANSYS CFX, 2012), a High Resolution Scheme by application of UDS (Upwind Differencing Scheme) and first and second order Taylor Series Expansion approximations are applied. These topics are briefly reproduced in the next chapters (3.2.3.1, 3.2.3.2 and 3.2.3.3). Since the steady state assumption is applied to the present work, temporal discretization scheme will be not discussed here.

3.2.3.1 Taylor Series Expansion

Any continuous differentiable function $\phi(x)$ can, near to x_i , be expressed as a Taylor series:

$$\phi(x) = \phi(x_i) + \frac{(x - x_i)}{1!} \left(\frac{\partial \phi}{\partial x} \right)_i + \frac{(x - x_i)^2}{2!} \left(\frac{\partial^2 \phi}{\partial x^2} \right)_i + \frac{(x - x_i)^3}{3!} \left(\frac{\partial^3 \phi}{\partial x^3} \right)_i + \frac{(x - x_i)^n}{n!} \left(\frac{\partial^n \phi}{\partial x^n} \right)_i + H \quad (3.16)$$

From the Taylor series above, one can obtain approximate expression for the first derivative by changing x for x_{i+1} :

$$\left(\frac{\partial \phi}{\partial x} \right)_i = \frac{\phi_{i+1} - \phi_i}{x_{i+1} - x_i} - \frac{(x_{i+1} - x_i)}{2!} \left(\frac{\partial^2 \phi}{\partial x^2} \right)_i - \frac{(x_{i+1} - x_i)^2}{3!} \left(\frac{\partial^3 \phi}{\partial x^3} \right)_i + H \quad (3.17)$$

By truncating the approximation just after the first term, the formulation for FDS (Forward Difference Scheme) is presented as:

$$\left(\frac{\partial \phi}{\partial x} \right)_i \approx \frac{\phi_{i+1} - \phi_i}{x_{i+1} - x_i} \quad (3.18)$$

From the Taylor series, one can also obtain approximate expression for the first derivative by changing x for x_{i-1} :

$$\left(\frac{\partial \phi}{\partial x} \right)_i = \frac{\phi_i - \phi_{i-1}}{x_i - x_{i-1}} + \frac{(x_i - x_{i-1})}{2!} \left(\frac{\partial^2 \phi}{\partial x^2} \right)_i - \frac{(x_i - x_{i-1})^2}{3!} \left(\frac{\partial^3 \phi}{\partial x^3} \right)_i + H \quad (3.19)$$

On truncating the approximation just after the first term, the formulation for BDS (Backward Difference Scheme) is presented as:

$$\left(\frac{\partial \phi}{\partial x} \right)_i \approx \frac{\phi_i - \phi_{i-1}}{x_i - x_{i-1}} \quad (3.20)$$

From the Taylor series, one can also obtain approximate expression for the first derivative by changing x for x_{i+1} and x_{i-1} :

$$\left(\frac{\partial \phi}{\partial x} \right)_i = \frac{\phi_{i+1} - \phi_{i-1}}{x_{i+1} - x_{i-1}} - \frac{(x_{i+1} - x_i)^2 - (x_i - x_{i-1})^2}{2! (x_{i+1} - x_{i-1})} \left(\frac{\partial^2 \phi}{\partial x^2} \right)_i - \frac{(x_{i+1} - x_i)^3 - (x_i - x_{i-1})^3}{3! (x_{i+1} - x_{i-1})} \left(\frac{\partial^3 \phi}{\partial x^3} \right)_i + H \quad (3.21)$$

On truncating the approximation just after the first term, the formulation for CDS (Central Difference Scheme) is presented as:

$$\left(\frac{\partial \phi}{\partial x}\right)_i \approx \frac{\phi_{i+1} - \phi_{i-1}}{x_{i+1} - x_{i-1}} \quad (3.22)$$

Approximations of the second derivate, for cases with constant spacing, can be defined by applying Taylor series forward and backward (FDS with x_{i+1} and BDS with x_{i-1}) and summing the equations. The result is the Equation (3.23), which is applicable for FDS, BDS and CDS, with constant spacing (Δx):

$$\left(\frac{\partial^2 \phi}{\partial x^2}\right)_i \approx \frac{\phi_{i+1} + \phi_{i-1} - 2\phi_i}{(\Delta x)^2} \quad (3.23)$$

3.2.3.2 Upwind Interpolation (UDS)

The Figure 3.3 shows a typical control volume for a Cartesian 2D grid, which is applied to exemplify the upwind interpolation method.

Figure 3.3 - Typical control volume and notation for Cartesian 2D grid (Ferziger & Peric, 2002).

Approximating ϕ_e , by its value at the node upstream of "e" is equivalent to using a backward or forward difference approximation for the first derivative (depending on the flow direction), hence the name upwind differencing scheme (UDS) for this approximation. In UDS, ϕ_e is approximated as:

$$\phi_e = \begin{cases} \phi_P & \text{if } (v_e > 0); \\ \phi_E & \text{if } (v_e < 0). \end{cases} \quad (3.24)$$

Applying Taylor series expansion about P gives (for Cartesian grid and $v_e > 0$):

$$\phi_e = \phi_P + (x_e - x_P) \left(\frac{\partial \phi}{\partial x} \right)_P + \frac{(x_e - x_P)^2}{2} \left(\frac{\partial^2 \phi}{\partial x^2} \right)_P + H \quad (3.25)$$

In ANSYS CFX code (ANSYS CFX, 2012) the advection term requires the integration point values of ϕ to be approximated in terms of the nodal values of ϕ . Its implemented schemes can be cast as following:

$$\phi_e = \phi_P + \beta \nabla \phi \cdot \Delta \vec{r} \quad (3.26)$$

Where β is the blend factor and $\Delta \vec{r}$ is the vector from the upwind node (ϕ_P) to the ϕ_e . Particular choices for β and $\Delta \vec{r}$ yield different schemes.

A value of $\beta = 0$ yields a first order UDS, which is a very robust scheme. However, it will introduce diffusive discretization errors that tend to smear steep spatial gradients as shown below:

Figure 3.4 – Left: Discretized solution. Right: Smeared solution (numerical diffusion) by first order UDS. (ANSYS CFX, 2012).

By choosing a value for $\beta > 0$ and $\beta < 1$, and by setting $\nabla \phi$ equal to the average of the adjacent nodal gradients, the discretization errors associated with the UDS are reduced. The quantity $\beta \nabla \phi \cdot \Delta \vec{r}$, called the Numerical Advection Correction, may be viewed as an anti-diffusive correction applied to the upwind scheme.

The choice $\beta = 1$ is formally second-order-accurate in space, and the resulting discretization will more accurately reproduce steep spatial gradients than first order UDS. However, it is unbounded and may introduce dispersive discretization errors that tend to cause non-physical oscillations in regions of rapid solution variation as shown below.

Figure 3.5 – Left: Discretized solution. Right: Numerical oscillation by second order UDS. (ANSYS CFX, 2012).

3.2.3.3 High Resolution Scheme

The applied spatial discretization the High Resolution Scheme uses a special nonlinear recipe for the blend factor (β) at each node, computed to be as close to 1 as possible (second order UPS). The advective flux is then evaluated using the values of β and $\nabla\phi$ from the upwind node. The recipe for β is based on the boundedness principles used by Barth and Jespersen (1989).

This methodology involves first computing a ϕ_{min} and ϕ_{max} at each node using a stencil involving adjacent nodes (including the node itself). Next, for each integration point around the node, the Equation (3.33) is solved for β to ensure that it does not undershoot ϕ_{min} or overshoot ϕ_{max} .

The nodal value for β is taken to be the minimum value of all integration point values surrounding the node. The value of β is also not permitted to exceed 1.

3.2.4 Shape Functions

Solution fields and other properties are stored at the mesh nodes. However, to evaluate many of the terms, the solution field or solution gradients must be approximated at integration points. ANSYS CFX uses finite-element shape functions perform these

approximations. Finite-element shape functions describe the variation of a variable ϕ varies within an element, as follows:

$$\phi = \sum_{i=1}^{N_{node}} N_i \phi_i \quad (3.27)$$

Where N_i is the shape function for node i and ϕ_i is the value of ϕ at node i . The summation is over all nodes of an element. Key properties of shape functions include:

$$\phi = \sum_{i=1}^{N_{node}} N_i = 1 \quad (3.28)$$

$$\text{At node } j, N_i = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases} \quad (3.29)$$

The shape functions used in ANSYS CFX are linear in terms of parametric coordinates. They are used to calculate various geometric quantities as well, including ip coordinates and surface area vectors. The tri-linear shape functions for the hexahedral grid element nodes (applied to the discretization grid of the present work, as shown in chapter 5.1.3) are given below:

Figure 3.6 – Hexahedral Element. (ANSYS CFX, 2012).

$$N_1(s, t, u) = (1 - s) (1 - t) (1 - u) \quad (3.30)$$

$$N_2(s, t, u) = s(1 - t) (1 - u)$$

$$N_3(s, t, u) = st (1 - u)$$

$$N_4(s, t, u) = (1 - s) t (1 - u)$$

$$N_5(s, t, u) = (1 - s) (1 - t) u$$

$$N_6(s, t, u) = s (1 - t) u$$

$$N_7(s, t, u) = s t u$$

$$N_8(s, t, u) = (1 - s) t u$$

3.2.5 Pressure-Velocity Coupling

The grid association with the finite volume discretization method can be performed in two different ways: Co-located or Staggered Grid arrangement. ANSYS CFX 14.5 (ANSYS CFX, 2012) uses the Co-located Grid arrangement, wherein the control volumes are identical for all transport equations, ensuring the correct pressure-velocity coupling, by means of Rhie & Chow (1982) interpolation method. That is the most common practice in Co-located unstructured grids and in curvilinear structured meshes (Versteeg & Malalasekera, 2007).

In order to identify the differences between Co-located or Staggered Grid arrangement, the 1D examples presented by Versteeg & Malalasekera (2007), are briefly presented below.

Figure 3.7 - 1D grid example of co-located arrangement (Versteeg & Malalasekera, 2007).

In the 1D co-located arrangement shown in Figure 3.7, the velocity u is stored and calculated at the location P (center of control volumes), and the discretized u -momentum equation at node P is:

$$a_{p_p} u_p = \sum a_{nb} u_{nb} + (p_w - p_e) A_p + b_p \quad (3.31)$$

Pressures are also stored at other nodes (W , P , E , etc.), therefore it is needed to interpolate to find the required face pressures p_w and p_e . If linear interpolation is applied, we obtain:

$$(p_e - p_w) = \frac{(p_E + p_P)}{2} - \frac{(p_P + p_W)}{2} = \frac{(p_W - p_E)}{2} \quad (3.32)$$

The discretized Equation (3.31) now becomes:

$$a_{P_P} u_P = \sum a_{nb} u_{nb} + \frac{1}{2} (p_W - p_E) A_P + b_P \quad (3.33)$$

$$u_P = \frac{\sum a_{nb} u_{nb} + b_P}{a_{P_P}} + \frac{d_P}{2} (p_W - p_E) \quad (3.34)$$

Where:

$$d_P = \frac{A_P}{a_{P_P}} \quad (3.35)$$

Similarly, the discretized u -momentum equation at node E is:

$$u_E = \frac{\sum a_{nb} u_{nb} + b_E}{a_{P_E}} + \frac{d_E}{2} (p_P - p_{EE}) \quad (3.36)$$

Where:

$$d_E = \frac{A_E}{a_{P_E}} \quad (3.37)$$

However, in case of a “checker-board” problem, as shown in Figure 3.7, where $(p_W - p_E)$ is zero, linear interpolation leads to non-physical solutions. Rhie and Chow (1982) suggest that a higher-order term should be included to prevent this effect. In this case, the interpolation practice is to obtain the cell face velocity u_e as:

$$u_e = \frac{u_P + u_E}{2} - \frac{1}{2} (d_P + d_E) (p_P - p_E) - \frac{1}{4} d_P (p_W - p_E) - \frac{1}{4} d_E (p_P - p_{EE}) \quad (3.38)$$

Figure 3.8 - 1D grid example of staggered arrangement (Versteeg & Malalasekera, 2007).

In the staggered arrangement (Figure 3.8), the u -momentum equation is solved at location e (the face of the scalar control volume centered at P), and using standard notation the discretized form at velocity node e is:

$$a_{Pe}u_e = \sum a_{nb}u_{nb} + (p_P - p_E)A_e + b \quad (3.39)$$

The continuity equation is applied to the scalar control volume to derive the pressure correction equation of the resolution algorithm, which is applied in an iterative method (e.g. SIMPLE: Semi-Implicit Method for Pressure-Linked Equations). Those algorithms will not be described here, whereas staggered grid will not be used by this work.

3.2.6 Solution of Linear Equation Systems

ANSYS CFX 14.5 (ANSYS CFX, 2012) uses a Multigrid (MG) technique for solving the discrete system of linearized equations. It is an iterative solver whereby the exact solution of the equations is approached during the course of several iterations. The linearized system of discrete equations described above can be written in the general matrix form:

$$[A] [\phi] = [b] \quad (3.40)$$

Where $[A]$ is the coefficient matrix, $[\phi]$ the solution vector and $[b]$ the right hand side.

The above equation can be solved iteratively by starting with an approximate solution, ϕ^n , that is to be improved by a correction, ϕ' , to yield a better solution, ϕ^{n+1} , that is:

$$\phi^{n+1} = \phi^n + \phi' \quad (3.41)$$

Where ϕ' is a solution of:

$$A \phi' = r^n \quad (3.42)$$

With the residual r^n obtained from:

$$r^n = b - A \phi^n \quad (3.43)$$

Repeated application of this algorithm will yield a solution of the desired accuracy. However, iterative solvers tend to decrease rapidly in performance as the number of grid elements and their aspect ratio increases, as shown in chapter 0.

The Multigrid process involves carrying out early iterations on a fine grid and later iterations on progressively coarser virtual ones. The results are then transferred back from the coarsest grid to the original fine one. From a numerical standpoint, the MG approach offers a significant advantage. For a given grid refinement level, iterative solvers are efficient only at reducing errors that have a wavelength of the order of the grid spacing. Therefore, while shorter wavelength errors disappear quite quickly, errors with longer wavelengths, of the order of the domain size, can take an extremely long time to disappear. The Multigrid method bypasses this problem by using a series of coarse meshes such that longer wavelength errors appear as shorter wavelength errors relative to the grid spacing. To prevent the need to mesh the geometry using a series of different grid refinement, Algebraic Multigrid is applied.

Algebraic Multigrid forms a system of discrete equations for a coarse grid by summing the fine grid equations. This results in virtual coarsening of the grid during the course of the iterations, and then re-refining to obtain an accurate solution. This technique significantly improves the convergence rates and it is less expensive than other Multigrid methods because the discretization of the nonlinear equations is performed only once for the finest grid. ANSYS CFX uses a particular implementation of Algebraic Multigrid called Additive Correction. This approach is ideally suited to the CFX-Solver implementation since it takes advantage of the fact that the discrete equations are representative of the balance of conserved quantities over a control volume. The coarse grid equations can be created by merging the original control volumes to create larger ones as shown below:

Figure 3.9 – Behaviour of Algebraic Multigrid method (ANSYS CFX, 2012).

3.2.7 Turbulence

All flows encountered in engineering problems become unstable above a certain Reynolds number, when they change from laminar to turbulent state. The Reynolds number studied for the present work is higher than 3000, which means that the flows shall be always turbulent. Turbulence causes the appearance in the flow of eddies with a wide range of length and time scales that interact in a dynamically complex way. Given the importance of this subject in the flow dynamics studies, it is no surprise that a substantial amount of research effort is dedicated to the development of numerical methods to capture the effects due to turbulence, whose some results are available on the literature,

e.g. Ferziger & Peric (2002) and Versteeg & Malalasekera (2007). The methods can be grouped into the following three categories:

- Direct Numerical Simulation (DNS): the mean flow and all turbulent velocity fluctuations are computed. The unsteady Navier–Stokes equations are solved on spatial grids that are sufficiently fine that they can resolve the Kolmogorov length scales at which energy dissipation takes place and with time steps sufficiently small to resolve the period of the fastest fluctuations. These calculations are highly costly in terms of computing resources, so the method is not used for industrial flow computations nor for the present work.
- Large Eddy Simulation (LES): intermediate form of turbulence calculation that tracks the behavior of larger eddies only, by filtering them according to their sizes. Despite of working with model flow for the smallest unresolved eddies, it still demands a lot on computing resources for the large ones, so it is not used for the present work.

Reynolds-Averaged Navier–Stokes (RANS): it is focused on the mean flow and the effects of turbulence on mean flow properties. Prior to the application of numerical methods, the Navier–Stokes equations are time averaged (if the flow is statistically steady), over a coordinate in which the mean flow does not vary, or over an ensemble of realizations (an imagined set of flows in which all controllable factors are kept fixed). Extra terms appear in the averaged flow equations due to the interactions between various turbulent fluctuations. These extra terms are modelled with classical turbulence models (e.g. k– ϵ model and the Reynolds stress model). The computing resources required for reasonably accurate flow computations are modest, so this approach has been the mainstay of engineering flow calculations and it is used for the present work.

3.2.7.1 Reynolds-Averaged Navier–Stokes (RANS) Equations

In a statistically steady flow, every scalar quantity can be written as the sum of a time-averaged value and a fluctuation about that value:

$$\phi(x_i, t) = \bar{\phi}(x_i) + \phi'(x_i, t) \quad (3.44)$$

The time averaging of the scalar quantity is defined as:

$$\bar{\phi}(x_i) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_0^T \phi(x_i, t) dt \quad (3.45)$$

Where T is the averaging time of the simulation. It must be large compared to the typical timescale of the turbulent fluctuation for statistically steady flow. The sum of a time-averaged value (e.g. velocity component, u) can graphically be represented according to Figure 3.10:

Figure 3.10 – Example of time averaging of turbulence used on RANS models (ANSYS CFX, 2012).

Averaging a quadratic nonlinear term, results in the product of the average and a covariance.

$$\overline{u_i \phi} = \overline{(\bar{u}_i + u'_i)(\bar{\phi} + \phi')} = \bar{u}_i \bar{\phi} + \overline{u'_i \phi'} \quad (3.46)$$

The last term is zero only if the two quantities are uncorrelated, what rarely happens in turbulent flows. Actually, the conservation equations contain terms such as $\overline{\rho u'_j u'_i}$, called Reynolds stress, and $\overline{\rho u'_j \phi'}$, known as the turbulent scalar flux, which cannot be represented uniquely in terms of the mean quantities.

The averaged continuity, momentum and energy equations can (for the simplification assumptions: incompressible flow, constant properties and no body forces) be written in tensor notation and Cartesian coordinates as:

- Mass conservation:

$$\frac{\partial(\rho \bar{u}_j)}{\partial x_j} = 0 \quad (3.47)$$

- Momentum conservation:

$$\frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{u}_j \bar{u}_i + \overline{\rho u'_j u'_i}) = \frac{\partial \bar{\tau}_{ij}}{\partial x_j} - \frac{\partial \bar{p}}{\partial x_i} \quad (3.48)$$

Where $\bar{\tau}_{ij}$ is the mean viscous stress tensor component:

$$\bar{\tau}_{ij} = \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (3.49)$$

- Energy conservation:

$$\frac{\partial(\rho \bar{T})}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{u}_j \bar{T} + \overline{\rho u'_j T'}) = \frac{\partial}{\partial x_j} \left(\frac{k}{c_p} \frac{\partial \bar{T}}{\partial x_j} \right) \quad (3.50)$$

Finally, the equation for the mean of a scalar quantity can be written:

$$\frac{\partial(\rho \bar{\phi})}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{u}_j \bar{\phi} + \overline{\rho u'_j \phi'}) = \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \bar{\phi}}{\partial x_j} \right) \quad (3.51)$$

The consequence of having Reynolds Stresses and turbulent scalar flux in the conservation equations is that the latter are not closed, in other words they have more variables than equations. Thus, it is required the application of some approximations, which usually take the form of prescribing the Reynolds stress tensor and turbulent scalar fluxes in terms of the mean quantities and are called Turbulence Models.

The most common RANS turbulence models are classified according to the number of additional transport equations that need to be solved along with RANS flow equations. The models listed in Table 3.1 form the basis of standard turbulence calculation procedures currently available commercial CFD codes.

Table 3.1 – Classification of turbulence models (Versteeg & Malalasekera, 2007).

Number of extra transportation equations	Name
Zero	Mixing length model
One	Spalart-Allmaras model
Two	$k - \varepsilon$ model $k - \omega$ model Algebraic stress model
Seven	Reynolds stress model

From the models above, mixing length and $k - \varepsilon$ model are at present by far the most widely used and validated. They are based on the assumption that there exists an analogy between the action of viscous stresses and Reynolds stresses on the mean flow.

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3.52)$$

There are experimental evidences, which show that turbulence decays unless there is shear in isothermal incompressible flows. Furthermore, turbulent stresses are found to increase with the mean rate of deformation. Boussinesq proposed in 1877 that Reynolds stresses might be proportional to mean rates of deformation (for incompressible flows):

$$\tau_{ij} = -\overline{u'_i u'_j} = \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (3.53)$$

Where $k = 1/2 \left(\overline{u'_i u'_i} \right)$ is the turbulent kinetic energy.

By analogy, turbulent transport of a scalar is taken to be proportional to the gradient of the mean value of the transported quantity:

$$-\rho \overline{u'_i \phi'} = \Gamma_t \frac{\partial \phi}{\partial x_i} \quad (3.54)$$

Since turbulent transport of momentum and heat or mass is due to the same mechanism – eddy mixing – we expect that the value of the turbulent diffusivity Γ_t is fairly close to that of the turbulent viscosity μ_t . This assumption is better known as the Reynolds analogy. The turbulent Prandtl/Schmidt number, whose value after several flows experiments is established constant and approximately unit, is introduced as follows:

$$\sigma_k = \frac{\mu_t}{\Gamma_t} \quad (3.55)$$

3.2.7.2 The $k - \varepsilon$ Model

The standard $k - \varepsilon$ model (Launder & Spalding, 1974) has two model equations, one for k and one for ε , based on our best understanding of the relevant processes causing changes to these variables. The first one, the turbulent kinematic energy (k), defines the velocity scale and it can be derived as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{u}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial k}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left(\frac{\rho}{2} \overline{u'_j u'_i u'_i} + \overline{p' u'_j} \right) - \overline{\rho u'_i u'_j} \frac{\partial \bar{u}_i}{\partial x_j} - \mu \frac{\partial \bar{u}'_i}{\partial x_k} \frac{\partial \bar{u}'_i}{\partial x_k} \quad (3.56)$$

The second term on the right-hand side represents turbulent diffusion of kinetic energy (which is actually transport of velocity fluctuations by the fluctuations themselves); it is usually modeled by use of a gradient diffusion assumption:

$$- \left(\frac{\rho}{2} \overline{u'_j u'_i u'_i} + \overline{p' u'_j} \right) \approx \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \quad (3.57)$$

The third term of the right-hand side of Equation (3.56) represents the rate of production of turbulent kinetic energy by the mean flow, a transfer of kinetic energy from the mean flow to the turbulence. Applying the eddy-viscosity hypothesis, Equation (3.53), to estimate the Reynolds stress, it can be written:

$$P_k = - \overline{\rho u'_i u'_j} \frac{\partial \bar{u}_i}{\partial x_j} \approx \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_j} \quad (3.58)$$

The last term represents the product of the density (ρ) and the dissipation (ε), the rate at which turbulence energy is irreversibly converted into internal energy. A number of equations have been used for this purpose and the most popular one is based on the observations that the dissipation is needed in the energy equation and, in so-called equilibrium turbulent flows, i.e., ones in which the rates of production and destruction of turbulence are in near-balance and related by:

$$\varepsilon \approx \frac{k^{3/2}}{L} \quad (3.59)$$

The equilibrium turbulent flows is the main character of the $k - \varepsilon$ model and the factor that ultimately determines whether the model is appropriate for a particular application or not. The final equation for the dissipation derived from the Navier-Stokes equations and assumptions above is most commonly found in the following form:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_j \varepsilon)}{\partial x_j} = C_{\varepsilon 1} P_k \frac{\varepsilon}{k} - \rho C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) \quad (3.60)$$

The eddy energy is expressed as:

$$\mu_t = \rho C_\mu \sqrt{k} L = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3.61)$$

The equations contain five adjustable constants, whose values are defined by comprehensive data fitting for a wide range of turbulent flows:

$$C_\mu = 0.09; \quad C_{\varepsilon 1} = 1.44; \quad C_{\varepsilon 2} = 1.92; \quad \sigma_k = 1.00; \quad \sigma_\varepsilon = 1.30 \quad (3.62)$$

3.2.7.3 The $k - \omega$ Model

The second most commonly used model is the $k - \omega$ model, popularized by Wilcox (1998). In this model an equation for a turbulence frequency scale (ω) is applied. The $k - \omega$ model also uses the turbulent kinetic energy equation from $k - \varepsilon$, Equation (3.56), but it has to be modified as following:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{u}_j k)}{\partial x_j} = P_k - \rho \beta^* k \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k^*} \right) \frac{\partial k}{\partial x_j} \right] \quad (3.63)$$

The ω equation is also given:

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \bar{u}_j \omega)}{\partial x_j} = \gamma_1 \frac{\omega}{k} P_k - \rho \beta_1 k \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega^*} \right) \frac{\partial \omega}{\partial x_j} \right] \quad (3.64)$$

In this model, the eddy viscosity is expressed as:

$$\mu_t = \rho \frac{k}{\omega} \quad (3.65)$$

As for $k - \varepsilon$, $k - \omega$ model equations also contain adjustable constants, whose values are defined by comprehensive data fitting for a wide range of turbulent flows:

$$\gamma_1 = 0.553; \quad \beta_1 = 0.075; \quad \beta^* = 0.09; \quad \sigma_k^* = \sigma_\omega^* = 2.00; \quad \varepsilon = \beta^* \omega k \quad (3.66)$$

3.2.7.4 The SST $k - \omega$ Model

It was noted by Menter (1994) that the results of the $k - \varepsilon$ model are much less sensitive to the (arbitrary) assumed values in the free stream, but its near-wall performance is unsatisfactory for boundary layers with adverse pressure gradients. This led him to suggest a hybrid model using a transformation of the $k - \varepsilon$ model into a $k - \omega$ model in the near-wall region and the standard $k - \varepsilon$ model in the fully turbulent region far from the wall. The Reynolds stress computation and the k -equation are the same as original $k - \omega$ model, but the ε -equation is transformed into an ω -equation by substituting $\varepsilon = k\omega$. This yields to the following equation:

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho\bar{u}_j\omega)}{\partial x_j} = \gamma_2 \frac{\omega}{k} P_k - \rho\beta_2\omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega,1}^*} \right) \frac{\partial\omega}{\partial x_j} \right] + 2 \frac{\rho}{\omega} \frac{\partial k}{\partial x_k} \frac{\partial\omega}{\partial x_k} \quad (3.67)$$

Summarizing a series of modifications to optimize the performance of SST $k - \omega$ model based on experience with the model in general-purpose computation (Versteeg & Malalasekera, 2007), resulting in the following revised constants:

$$\gamma_2 = 0.44; \quad \beta_2 = 0.083; \quad \beta^* = 0.09; \quad \sigma_k^* = 1.0; \quad \sigma_{\omega,1}^* = 2.00; \quad \sigma_{\omega,2}^* = 1.17 \quad (3.68)$$

Blending functions are introduced in the equation to modify the cross-diffusion term and are used for model constants that take value C_1 for the original $k - \varepsilon$ model and value C_2 in transformed $k - \varepsilon$ model:

$$C = F_C C_1 + (1 - F_C) C_2 \quad (3.69)$$

Typically, a blending function $F_C = F_C(l_t/y, Re_y)$ is a function of the ratio of turbulence $l_t = \sqrt{k}/\omega$ and distance y to the wall and of a turbulence Reynolds number $Re_y = y^2\omega/\nu$. The functional form of F_C is chosen so that it is zero at the wall, tends to unity in the far field and produces a smooth transition around a distance half way between the wall and the edge of the boundary layer.

3.2.7.5 Near-Wall Treatments

During the selection of the turbulence model to be applied in a CFD simulation, the near-wall treatment method shall also be considered. In this section, a brief summary about the near-wall treatments implemented by the code ANSYS CFX is presented. Therefore, the theory guide provided by ANSYS CFX 14.5 (ANSYS CFX, 2012) is conveniently reproduced here to turn it an easy reference to current reader. All information and credit shown in this section should be given to ANSYS CFX 14.5 (ANSYS CFX, 2012).

Near a no-slip wall, there are strong gradients in the dependent variables. In addition, viscous effects on the transport processes are large. The representation of these processes within a numerical simulation raises the following problems:

- How to account for viscous effects at the wall?
- How to resolve the rapid variation of flow variables that occurs within the boundary layer region?

Experiments and mathematical analysis have shown that the near-wall region can be subdivided into two layers. In the innermost layer, the so-called viscous sub-layer, the flow is almost laminar-like, and the (molecular) viscosity plays a dominant role in momentum and heat transfer. Further, away from the wall, in the logarithmic layer, turbulence dominates the mixing process. Finally, there is a region between the viscous sub-layer and the logarithmic layer called the buffer layer, where the effects of molecular viscosity and turbulence are of equal importance. The figure below illustrates these subdivisions of the near-wall region.

Figure 3.11 – Subdivisions of the Near-Wall Region (ANSYS CFX, 2012).

Assuming that the logarithmic profile reasonably approximates the velocity distribution near the wall, it provides a means to compute numerically the fluid shear stress as a function of the velocity at a given distance from the wall. This is known as a 'wall function' and the logarithmic nature gives rise to the well-known 'log law of the wall.'

Two approaches are commonly used to model the flow in the near-wall region:

- The wall function method uses empirical formulas that impose suitable conditions near to the wall without resolving the boundary layer, thus saving computational resources. All turbulence models in CFX are suitable for a wall function method. The major advantages of the wall function approach is that the high gradient shear layers near walls can be modeled with relatively coarse meshes, yielding substantial savings in CPU time and storage. It also avoids the need to account for viscous effects in the turbulence model.
- The Low-Reynolds Number method resolves the details of the boundary layer profile by using very small mesh length scales in the direction normal to the wall (very thin inflation layers). Turbulence models based on the ω -equation, such as the SST or SMC- ω models, are suitable for a low Reynolds method. The computations are extended through the viscosity-affected sub-layer close to the wall. The low-Re approach requires a very fine mesh in the near-wall zone and correspondingly large number of nodes. Computer-storage and runtime requirements are higher than of Wall-Function approach. Care must be taken to ensure good numerical resolution in the near-wall region to capture the rapid variation in variables. To reduce the resolution requirements, an automatic wall treatment was developed by CFX, which allows a gradual switch between Wall Functions and low-Reynolds number grids, without a loss in accuracy.

Wall Functions are the most popular way to account for wall effects. In CFX, Scalable Wall Functions are used for all turbulence models based on the ε -equation. For ω -based models (including the SST model), an Automatic Near-Wall Treatment method is applied.

3.2.7.6 Summary of Turbulence Models

Since time-averaged properties of the flow satisfy the needs of the present work, the turbulent flow computations were carried out with procedures based on the Reynolds-Averaged Navier–Stokes equations (RANS).

The compilation of the information regarding to the turbulence models applied in the reference works presented in the literature review (chapter 2.2), results in the table below:

Table 3.2 – Turbulence models applied in the reference works.

Reference	Scope	Code	Turbulence Model	Accuracy	Result
(Yanagihara & Rodrigues, 1998)	stator core	PHOENICS V2.0	$k - \varepsilon$	-	15% - 24%
(Ujii, Arlitt, & Etoh, 2006)	complete ventilation system	-	$k - \varepsilon$	-	5% - 7%
(Altea & Meyer, 2007)	complete ventilation system	ANSYS CFX	-	2×10^{-3}	26%
(Houde, Vincent, & Hudon, 2008)	complete ventilation system	ANSYS CFX	$k - \varepsilon$ (with wall function)	1×10^{-4}	5% - 20%
(Gregorc, 2011)	complete ventilation system	ANSYS CFX	$k - \varepsilon$ (with wall function)	1×10^{-4}	-
(Moradnia & Nilsson, 2010)	complete ventilation system	OpenFOAM	Launder - Sharma $k - \varepsilon$	-	-
(Vogt & Lahres, 2013)	complete ventilation system	ANSYS CFX	$k - \varepsilon$	-	0.8% - 2.2%
(Moradnia, Golubev, Chernoray, & Nilsson, 2013)	complete ventilation system	OpenFOAM	Launder - Sharma $k - \varepsilon$	-	2% - 7%
(Moradnia, Chernoray, & Nilsson, 2014)	complete ventilation system	OpenFOAM	Launder - Sharma $k - \varepsilon$	-	-

From the table above, there is only one reference with the same scope of simulation of this work: the stator core (Yanagihara & Rodrigues, 1998). Besides, the numerical results showed itself not so coincident with the reality (comparison of numerical results with

experimental data or analytical formulation with differences from 15% to 24%). It shall be understood as a reason to avoid applying directly the same turbulence model without deeper investigations.

In order to identify the best turbulence model applicable for the studied problem, preliminary simulations with $k - \varepsilon$ and SST (Shear-Stress Transport) $k - \omega$ models were executed. The results are presented and compared in the table below:

Table 3.3 – Comparisons between $k - \varepsilon$ and SST $k - \omega$ turbulence models.

Rotational Speed [Hz]	6.8		8.8		10.8		12.8	
Turbulence Model	$k - \varepsilon$	SST $k - \omega$	$k - \varepsilon$	SST $k - \omega$	$k - \varepsilon$	SST $k - \omega$	$k - \varepsilon$	SST $k - \omega$
y_{max}^+	2.90	4.61	3.96	5.49	4.83	6.26	5.69	6.97
$\overline{y^+}$	0.83	0.91	1.09	1.14	1.33	1.37	1.56	1.60
# of Interactions	195	209	256	176	277	177	279	178
Computational time [min]	60	65	82	61	83	64	89	52
Δp [Pa]	95.4	92.4	157.5	156.7	239.7	238.3	339.0	338.0

The values of the table above show that, besides the simulations with $k - \varepsilon$ turbulence model took longer and more interactions to converge, the results (Δp) do not differ much from those simulations with SST $k - \omega$ turbulence model. Furthermore, some simulations with $k - \varepsilon$ turbulence model achieved the convergence only when started with initial conditions defining a minimum radial velocity in the air ducts (0.1 m/s).

By the selection of a turbulence model, it is also very important to consider the wall treatment to be applied. The present work simulation refers to a Low-Reynolds Number flow and boundary layers with high adverse pressure gradients. Therefore, the application Low-Reynolds Number methods are more recommended over wall function methods. As described in chapter 3.2.7.5, Low-Reynolds Number methods can be applied on ANSYS CFX 14.5 only by the selection of a ω -based model (including the SST). The Automatic Near-Wall Treatment of those models in ANSYS CFX 14.5 are able to switch automatically between Wall Functions and Low-Reynolds Number methods based on local grid refinement.

Thus, the turbulence model SST (Shear-Stress Transport) k - ω (Menter, 1994) was selected for the simulations of this work. It is a hybrid model, which automatically switches

to the k- ω model in the near-wall region and the k- ϵ model away from the walls, by means of blending function that is one near the wall surface and zero far away from the wall. Thus, SST k- ω results in a robust model to calculate flow properties in the free-stream turbulent flow region (far from the wall) and boundary layers with adverse pressure gradients, at the same time.

3.2.8 Discretization Grid

Since numerical solutions are obtained over a discretized domain in an iterative process, they always have errors, whose significant portion comes exactly from the discretization itself. A proper selection of grid type and controlling of its quality is essential to maximize the accuracy of CFD results.

Different types of grid according to the level of complexity of the geometry, algorithm and flow characteristics, have been developed over the years and published by several authors, like Maliska (2004), Ferziger & Peric (2002) and Versteeg & Malalasekera (2007).

Ferziger & Peric (2002) classified numerical solution errors as *Modeling Errors*, *Discretization Errors*, *Iteration Errors* and *Programming and User Errors*. The first one, also named by Versteeg & Malalasekera (2007) as “*Uncertainty*”, represents a potential deficiency in a CFD model, caused by a lack of knowledge. The main sources of those errors can be geometry, initial and boundary conditions, material properties, physical or chemical processes (e.g. turbulence, combustion, multi-phase flow) or due to simplifying assumptions in the modeling process (e.g. incompressible flow, steady flow).

Discretization Errors are related to the fact that the governing equations are approximated and integrated in the finite volume method on a certain time and space discretization. The smaller the spatial and temporal discrete elements, the more accurate these approximations become, but it requires increasing amounts of memory and computing time. Since the flow may vary strongly in both space and time, uniform discretization is seldom optimal, considering that the errors increase with the changes in variables.

The discretization process produces a coupled set of non-linear algebraic equations, which are normally linearized and solved by an iterative method. The iterative process will be stopped when the level of residual has been reduced by a predefined value (convergence criterion), however, the exact solution will never be obtained, and the difference between the exact and iterative solutions defines the *Iteration Errors*.

Discretization Errors are directly related to the grid refinement, which can minimize and deliver reliable estimation of these errors, when applied systematically. However, the refinement level also affects the effort to reduce the *Iterative Errors*, and its definition shall be done considering the level of *Iteration Errors* (convergence criteria).

3.2.8.1 Grid Quality

Although grid systematic refinements reduce *Discretization Errors*, better results can be achieved by optimization of a grid with a given number of grid points (Ferziger & Peric, 2002). It is therefore important to pay attention to the *Grid Quality*.

One of the few definitions for *Grid Quality* available in the literature is presented by Knupp (2007): “*Grid Quality* concerns the characteristics of a grid that permit a particular numerical PDE simulation to be efficiently performed, with fidelity to the underlying physics, and with the accuracy required for the problem”. It means that a satisfactory grid quality is only achieved when the grid: i) is suitable for the performed calculation and its physical problem, ii) does not harm the numerical calculation (convergence and stability issues) and iii) results in sufficiently accurate values.

3.2.8.2 Grid Quality Metrics

To measure *Grid Quality*, traditional metrics can be found in the literature and in user manuals of some commercial codes like FLUENT (ANSYS FLUENT, Inc., 2012) and ANSYS ICEM CFD (ANSYS ICEM, Inc., 2012). A metric is a real number assigned to one geometric characteristic of a finite element, finite volume, boundary element, or other type of discretization of the solution to partial differential equations.

Over the years, some specific technical communities have developed their own certain grid quality metrics as published by Field (2000) for crash simulation in automotive industry, by Alter (2004) for hypersonic flows in Aerospace Sciences and Mavriplis, et al. (2008) for aerodynamic drag prediction.

There are also some available algorithms, which are able to evaluate the metrics of each mesh entity or combined characteristics, like VERDICT from Stimpson, et al. (2007). Others, like MESQUITE, from Freitag, et al. (2002), goes even further and can improve and smooth a generated grid by applying specific schemes like vertex relocation and topology modification.

3.2.8.3 Grid Quality Metrics – Aspect ratio

Known as AR, this metric shows how narrow a cell is, by comparing the length of its edges. Large aspect ratios should always be avoided in the flow domain; otherwise, the pressure gradient on the larger side must be computed from much more widely spaced nodes, what may result in velocity vectors with high values and oscillations. It cannot be removed by the pressure-velocity coupling algorithm and results in poor iterative convergence.

Tu, Yeoh, & Liu (2013) recommend maintaining AR within the range of $0.2 < AR < 5$ in the interior region of the flow. For near wall boundaries, these limits can, however, be relaxed. Considering 2D quadrilateral cells with the flow mainly in the y direction there is the need to choose small Δx mesh spacing, which generally yields $AR > 5$. In such a case, first- and second-order gradients indicates a flow biased in the y direction, behaving almost as one-dimensional, since:

$$\frac{\partial^2 \phi}{\partial x^2} \ll \frac{\partial^2 \phi}{\partial y^2} \quad \text{and} \quad \frac{\partial \phi}{\partial x} \ll \frac{\partial \phi}{\partial y} \quad (3.70)$$

The equation above helps with convergence difficulties. It is properly used to solve the wall boundary layers, where Ferziger & Peric (2002) suggest the application of hexahedra or prisms layers.

For three-dimensional grids, the problem is even more pronounced. In ANSYS ICEM CFD 14.5 (ANSYS ICEM, Inc., 2012) AR of hexahedral cells is defined by the size of the minimum element edge divided by the size of the maximum element one, while AR of tetrahedral cells is calculated by the ratio between the volume of the cell and the radius of its circumscribed sphere power three.

3.2.8.4 Grid Quality Metrics – Orthogonality and Skewness

In most finite volume methods, it is not important that the grid lines be orthogonal at cells corners (Ferziger & Peric, 2002); the quality of a cell is determined mainly by two other important angles.

First, the orthogonality between the vector from cell centroid to the centroids of each adjacent cells and the vector from the cell centroid to each of its faces center. It maximize the accuracy of convective fluxes with linear interpolation or the midpoint rule, since the vector connecting each adjacent cells pass closer to the center of the common face.

In addition to the condition above, a second angle should be maintained as close as possible to orthogonal, to maximize the accuracy of diffusive fluxes. The angle between the vector from the cell centroid to the centroids of each adjacent cells and the vector normal to each face. Orthogonality at this second angle increases the accuracy of central-difference approximations to the derivative in the direction normal to the cell faces.

Figure 3.12 - Skewness on a hexahedral cell (ANSYS ICEM, Inc., 2012)

For some complicated geometries, there is a high probability that the generated grid contains cells with unfavorable orthogonality and skewness. Especially near to the walls, where the orthogonality is even more essential, since most automatic grid generators usually create layers of hexahedra or prisms at boundaries. When the poor orthogonality is unavoidable, the accuracy of convective fluxes can be improved by the refinement of the grid, which increases the effort for numeric resolution and computation time. By applying higher-order approximations with derivatives using more nodes, skewness can be compensated and the accuracy of diffusive fluxes can be improved. It may lead, however, to convergence problems.

Figure 3.13 - Improvement of orthogonality (Ferziger & Peric, Computational Methods for Fluid Dynamics, 2002).

3.2.8.5 Grid Quality Metrics – Others Individual Cell Metrics

Warping is measured by the rotation of a cell face around its normal. Tu, Yeoh, & Liu (2013) recommend special care with unstructured grids, where the warp angle should not be greater than 75° , or large deviations can lead to serious convergence problems. Therefore, it is suggested to apply prismatic or hexahedral cells instead of tetrahedral elements in wall boundary layers, considering they have regular shapes and ability to adjust in accordance with the near-wall turbulence model requirements.

Figure 3.14 - Left: warping on hexahedral cell (Ferziger & Peric, 2002); Right: warping on tetrahedral cell (Tu, Yeoh, & Liu, 2013).

Other kinds of distortions, like shared faces or dragged vertexes, are also undesirable. For hexahedral cells, those distortions can be measured by code algorithms (ANSYS ICEM, Inc., 2012) comparing opposite faces with special metrics such as Taper (area ratio) and Parallelism.

Figure 3.15 - Dragged vertex on hexahedral cell. Left: Ferziger & Peric (2002); Right: ANSYS ICEM CFD 14.5 (ANSYS ICEM, Inc., 2012).

Sometimes the grid generation results in such high distortion, that the centroid is actually situated outside the cell. To achieve approximations by the midpoint rule with second-order accuracy, that must be avoided. For the volume, integrals are computationally calculated on the centroid nodes.

3.2.8.6 Grid Quality Metrics – Metrics for Domain Regions

As an alternative to measuring each grid cell individually, there are some metrics used to measure the overall quality of a combined group of cells in a domain region.

Characteristics representing cell size (e.g. edge length, face areas or volumes) can be compared, and the results can be used as control parameters, when a uniform discretization is desired. However, when considering that optimal discretizations are seldom uniform (due to critical regions of high-flow gradients), another metric becomes very useful: *Size Expansion Ratio*.

By controlling the size expansion, local refinement and adaptive grids can be created avoiding discontinuities and sudden changes in the grid size. Otherwise, the numerical procedure can be destabilized due to the accumulation of truncation errors in the critical flow regions.

4 Reference Laboratory and Field Measurements Data

4.1 Reference Laboratory Experimental Data from the Literature

The reference experimental data used for numerical method validation by this work are results of a model test performed at Institute for Aerospace Engineering of TU Dresden (Technischen Universität - Dresden, Germany) and reported by Wustmann (2005).

The applied test layout is shown in Figure 4.1 and is based on the design of a ventilation system of a real RIM Ventilation Hydrogenerator (schematically shown in Figure 1.3).

Figure 4.1 – Experiment layout. Based on Wustmann (2005).

As described in chapter 1.1, the airflow inside of the hydrogenerator is boosted by the rotating components of the machine. In real generators, components with this hole are the Rotor Hub, the Rotor Rim and Poles (Figure 1.1). In the test rig, this function is performed by a unique rotating part (the Rotor, labeled in orange color in Figure 4.1), where the airflow enters axially through the upper opening Inlet Blades and turns to radial direction in the Interpolar Spaces.

Due to the shaft rotational speed, the air leaves the Rotor and enters into the Air Gap (between Rotor and Stator) with a high tangential velocity component. At this point,

despite the high tangential velocity, the airflow must change its direction and enter radially into the Stator Core Air Ducts.

In the Stator Core of the prototype machines, the airflow faces a real heat source: the Stator Winding conductors. Analogically, the Stator Core of the model test has some Air Ducts equipped with sensors of heat transfer coefficient, which are compound of a heat sources and temperature sensors.

By leaving the Stator Core of the test rig, the airflow finds a very simplified environment, namely a cylindrical housing with an outlet on the upper side, which leads the air to the bypass circuit for volume rate measurement. The bypass circuit guides the air finally back to the Rotor openings. Differently to the model test, by leaving the Stator Core of a real machine, the airflow has a path with more obstacles. It is guided by the Stator Frame; it passes through the Heat Exchangers; it is released in Generator Housing and it is finally guided back to the Rotor openings by the Air Guides.

Parameters regarding to the environment conditions and other constants applied to the experimental data post-processing, were reported as shown in Table 4.1.

Table 4.1 – Reported constant values for experimental data (Wustmann, 2005)

Parameter	Symbol	Value
Normal static pressure	p_n	101.325 kPa
Density	ρ	1.188 kg·m ⁻³
Kinematic viscosity	ν	15.300 x 10 ⁻⁶ m ² ·s ⁻¹
Gas constant	R_n	289.000 J·(kg·K) ⁻¹

The rotor applied to the model test was designed and constructed with axial inlet openings on upper side and four radial outlet openings on the circumference, as shown in the Figure 4.2.

Figure 4.2 – Rotor applied on model test (*Wustmann, 2005*).

On the stator side, a model with forty-four radial air ducts on the circumference and seven ducts on the axial direction was constructed. On upper side of the stator, the model includes the winding overhangs.

Figure 4.3 – Stator applied on model test (Wustmann, 2005).

By means of exchangeable wedges, four different configurations of air duct inlet could be tested. For the numerical simulations, only the symmetric wedge (Figure 4.4) was considered, since it is the most applied design on real machines.

Figure 4.4 – Model test wedge applied to the numerical simulation (Wustmann, 2005).

The other dimensions of the stator core air ducts are the same to the ones applied to the geometry of the numerical model and shown in Figure 5.2 and Table 5.1.

The rotor was placed into the stator, while keeping an air gap of 33 mm. It was driven by an electrical motor and the rotational speed was controlled by a measuring instrument with rotating coded ring and photocell.

Figure 4.5 – Experimental complete model (Wustmann, 2005).

The flow volume rate was measured in a bypass circuit with an orifice plate. The impact of the additional flow circulation path to the ventilation system was calculated and considered by Wustmann (2005) in compiled results. The airflow inside of the stator core air ducts is pulsing, since it is driven by discretized air ducts located on the rotor.

Figure 4.6 – Bypass circuit for flow volume rate measurement (Wustmann, 2005).

The reported pressure drop coefficients were calculated by the pressure difference between air duct inlet and outlet. Static pressure measurements were performed with a PSI® pressure scanner equipped with piezoresistive pressure sensors, linked to measurement point by flexible plastic hoses. Uncertainties for pressure measurements were reported (Wustmann, 2005) as limited to 1%.

Figure 4.7 – Pressure measurements on model test (Wustmann, 2005).

4.2 Reference Field Measurement Data of Prototype

An extra reference data to be used for numerical method validation, are the results of field measurements performed in a generator of HPP Teles Pires.

The generators in HPP Teles Pires are designed with 96 rotor salient poles, 720 stator-winding slots, grid frequency of 60 Hz and rated speed of 75 rpm. The UG05 ("Unidade Geradora" - Generation Unit number 05) of this power plant was equipped during its erection so that it was possible to measure the air pressure drop between its stator core inner and outer diameters.

4.2.1 Development of Measurement Procedures and Devices

Due to the tight schedule of site assembly and machine commissioning, which is normally required for the payback of this kind of enterprise, all the special procedures for the measurements must have minimal time impact, using some short schedule windows in the machine erection plan.

Therefore, the measurement plan and devices (pressure wall tapplings) were developed and previously tested in workshop using the stator core sheets of similar project (Belo Monte), as showed in Figure 4.8 and Figure 4.9.

Figure 4.8 – Workshop tests of pressure wall tapping installation.

Figure 4.9 – Instruction of pressure wall tapping installation at site.

The pressure wall tapplings were defined with flexible hoses and a plastic thin plate (terminal) glued in its end. The fixation of the terminal on the stator core inner diameter was done with an industrial tape. Silicone paste was used to eliminate any protrusion of the hose ends, which would cause interference on the measured values (Figure 4.10).

Figure 4.10 – Scheme of pressure tapping installation.

4.2.2 Installation of Measurement Devices at Site

At site, the pressure tapplings were installed in the stator of UG05, during its assembly process on erection area, before being placed into the generator's pit as showed. In Figure 4.11, the stator is already installed in the generator pit and the rotor is being placed.

Figure 4.11 – Teles Pires UG05 Stator at generator's pit.

The stator frame is designed with 12 columns, each one with an inspection window with access to the outer diameter of stator core. Therefore, the measuring hoses were installed in the stator winding slots aligned with those windows, as showed in Figure 4.12, Figure 4.13 and Figure 4.14 and listed in Table 4.2.

Figure 4.12 – Stator frame inspection window.

Figure 4.13 – Axial location of pressure tapping installation.

Figure 4.14 – Tangential location of pressure tapping installation.

Table 4.2 – Slot location of pressure tapping installation

Installation Point	Between Stator Winding Slots
1	436 – 437
2	496 – 497
3	556 – 557
4	616 – 617
5	676 – 677
6	16 – 17
7	76 – 77
8	136 – 137
9	196 – 197
10	256 – 257
11	316 – 317
12	376 – 377

Before the stator was placed into the generator pit, all the installation of pressure tapping were identified and documented as showed in the example of Figure 4.15.

Figure 4.15 – Example of identified pressure tapping installation.

4.2.3 Field Measurements

The field measurements were executed during the commissioning tests of the fifth power unit of HPP Teles Pires, occurred on March 31st 2016. The generator was not connected to the grid and without excitation current, that is, there was no electromagnetic load.

The unit rotational speed was increased slowly, stopping in steps of 25, 50, 75 and 100% of rated speed (75 rpm), for the measurements of the air pressure drop between its stator core inner and outer diameters.

For each speed step, the air pressure drop was measured in all the installed twelve measurement devices (Chapter 4.2.2), as showed in the Figure 4.16.

Figure 4.16 – Example of field measurement execution.

Parallel to the pressure drop measurements, it was also measured the air volume flow passing through one selected air cooler, the air temperature and the air relative humidity, for each rotational speed step.

The air volume flow passing through the selected air cooler is measured by means of an anemometer, which is placed in each one of the 24 delimited areas (as showed in the Figure 4.17). The velocity results are multiplied by the known areas (defined by constant lengths (L_1) and heights (L_2)) and all flow data are summed together in order to get the total amount of volumetric flow.

Figure 4.17 – Delimited areas prepared for measurements of air volume flow passing through one air cooler.

4.2.3.1 Measurement Instruments and Devices Arrangement

The measurement instruments and the devices applied to the pressure field measurements are listed in the Table 4.3.

Table 4.3 – Field measurement instruments and devices for pressure

Instrument / Device Type	Manufacturer / Model	Characteristics
Pressure Transmitter	Emerson / Rosemount 1151	<u>Max. working pressure:</u> 300 psi (2.06×10^6) <u>Power supply:</u> 12 – 45 VDC <u>Output:</u> 4 – 20 mA
Data Acquisition System	Delphin / AMDT V2.0	<u>Analogic input voltage:</u> 0 – 10 V or ± 10 V <u>Analogic input current:</u> 0 – 20 mA or 4 – 20 mA or ± 20 mA <u>Maximum output rate:</u> 50 Hz (USB or LAN) <u>Power supply:</u> 12 – 28 VAC / 12 – 36 VDC
Power Supplier	Balluff / BAE0005	<u>Input:</u> 100 – 240 VAC, 47 – 63 Hz, 1.5 A <u>Output:</u> 24 VDC, 2.5 A, 60 W

The Pressure Transmitter was ranged using a HART communicator by setting the points as following:

- 4 mA = 0 mmH₂O = 0 Pa;
- 20 mA = 51 mmH₂O = 499.8 Pa.

Others settings and applied values are:

- Power Supplier input = 220 VAC, 60 Hz;
- Acquisition System analogic input = 4 – 20 mA;
- Acquisition System output rate = 1 Hz (using an LAN cable).

The measurement instruments and devices applied to the field measurements were arranged as showed in the Figure 4.18:

Figure 4.18 – Mesasurement instruments and devices arrangement.

The measurement instruments and the devices applied to the air volume flow, temperature and relative humidity measurements are listed in the Table 4.4.

Table 4.4 – Others field measurement instruments

Instrument / Device Type	Manufacturer / Model	Characteristics
Anemometer	Minipa / MDA 11	<u>Temperature range:</u> 0 – 50 °C <u>Velocity range:</u> 0 – 30 m.s ⁻¹
Thermo - Hygrometer	Minipa / MT 242	<u>Temperature range:</u> 0 – 50 °C <u>Relative humidity range:</u> 20 – 90 %

4.2.3.2 Field Measurements Uncertainty Analysis

The uncertainty analysis was executed considering uncertainties of Evaluation Type A and Evaluation Type B (INMETRO, 2008).

4.2.3.2.1 Uncertainties of Evaluation Type A

The uncertainties of Evaluation Type A are estimated from the statistic distribution of values of a measurement set and can be defined by the standard deviation, as described below for a generic quantity X_j and N measured values.

- Mean:

$$X_m = \frac{1}{N} \sum_{j=1}^N X_j \quad (4.1)$$

- Experimental standard deviation:

$$s(X_j) = \sqrt{\frac{1}{N-1} \sum_{j=1}^N (X_j - X_m)^2} \quad (4.2)$$

- Experimental standard deviation of the mean:

$$s(X_m) = \frac{s(X_j)}{\sqrt{N}} \quad (4.3)$$

The uncertainties of evaluation type A were applied to the measured data of pressure, substituting X for the pressure P_1 (Appendix A) and to the measured data of air velocity, substituting X for the velocity V_1 (Appendix B).

4.2.3.2.2 Uncertainties of Evaluation Type B

The first group of standard uncertainties of Evaluation Type B are based on the measuring instrument accuracies as showed in Table 4.5.

Table 4.5 – Field measurement instruments standard uncertainties

Instrument / Device	Measurement	Standard Uncertainty	Source
Pressure Transmitter	P_2 : Pressure	$\pm 0.12 \%$	Calibration
Data Acquisition System	I : Pressure acquisition current	$\pm 0.01 \text{ mA}$ and $\pm 0.05 \%$	Instrument accuracy class
Anemometer	V_2 : Air velocity	$\pm 3 \%$ of full scale (30 m.s^{-1})	Instrument accuracy class
Thermo - Hygrometer	T : Air temperature	$\pm 2 \text{ }^\circ\text{C}$	Instrument accuracy class
Thermo - Hygrometer	U : Air relative humidity	$\pm 5 \%$	Instrument accuracy class

Additionally to the standard uncertainties based on the measuring instrument accuracies, the standard uncertainties of two extra parameters are defined as showed in Table 4.6.

Table 4.6 – Extra standard uncertainties

Parameter	Standard Uncertainty	Source
L_1, L_2 : Length / Height of air passage area	± 0.02 m	Field staff experience
η : Rotational speed	± 5 %	Limits of speed governor control system

4.2.3.2.3 Uncertainties Combination

Assuming that the quantities (X_i) are independents, they are related between them according to the equations below:

$$Y = f(X_1, X_2, \dots, X_M) \quad (4.4)$$

The standard uncertainties, before to be combined, are defined as following:

- Type A:

$$\mu(X_i) = s(X_m) \quad (4.5)$$

- Type B:

$$\mu(X_i) = \text{according to Table 4.5 and Table 4.6} \quad (4.6)$$

The relative standard uncertainties are defined as following:

$$\frac{\mu(X_i)}{X_i} \quad (4.7)$$

The relative standard uncertainties are combined as following:

$$\left(\frac{\mu_{comb}(Y)}{Y} \right)^2 = \sum_{i=1}^M \left(p_i \frac{\mu(X_i)}{X_i} \right)^2 \quad (4.8)$$

The measurand equation is considered according to the equation:

$$Y = X_1^{p_1} \cdot X_2^{p_2} \dots \cdot X_M^{p_M} \quad (4.9)$$

When applied to the pressure, the measurand equation is defined as following:

$$P = P_1 \cdot P_2 \cdot I \quad (4.10)$$

Where:

$$X_1 = P_1 = \text{values of pressure measurements set} \quad (4.11)$$

$$X_2 = P_2 = \text{pressure measured by the Pressure Transmitter} \quad (4.12)$$

$$X_3 = I = \text{pressure acquired by Data Acquisition System} \quad (4.13)$$

$$p_1 = p_2 = p_3 = 1 \quad (4.14)$$

The numerical results of pressure standard uncertainties are available in the Appendix C.

When applied to the air volumetric flow, the measurand equation is defined as following:

$$Q = V_1 \cdot V_2 \cdot L_1 \cdot L_2 \quad (4.15)$$

Where:

$$X_1 = V_1 = \text{values of velocity measurements set} \quad (4.16)$$

$$X_2 = V_2 = \text{velocity measured by the Anemometer} \quad (4.17)$$

$$X_3 = L_1 = \text{length of air passage area} \quad (4.18)$$

$$X_4 = L_2 = \text{height of air passage area} \quad (4.19)$$

$$p_1 = p_2 = p_3 = p_4 = 1 \quad (4.20)$$

The numerical results of air volumetric flow standard uncertainties are available in the Appendix D.

The standard uncertainties of pressure and air volumetric flow are combined (including others standard uncertainties of evaluation Type B) to get the pressure difference, according to the following equation:

$$\Delta P = P \cdot Q^2 \cdot \eta \cdot T^{-1} \cdot U \quad (4.21)$$

Where:

$$X_1 = P = \text{standard uncertainties of pressure according to equation (4.10)} \quad (4.22)$$

$$X_2 = Q = \text{standard uncertainties of air volumetric flow according to equation (4.15)} \quad (4.23)$$

$$X_3 = \eta = \text{rotational speed} \quad (4.24)$$

$$X_4 = T = \text{air temperature measured by the Thermo - Hygrometer} \quad (4.25)$$

$$X_5 = U = \text{air relative humidity measured by the Thermo - Hygrometer} \quad (4.26)$$

$$p_1 = p_3 = p_5 = 1 \quad (4.27)$$

$$p_2 = 2 \quad (4.28)$$

$$p_4 = -1 \quad (4.29)$$

The numerical results of pressure difference standard uncertainties are available in the Appendix E.

4.2.3.3 Field Measurements Results

By the data resulting from the field measurements, it is possible to identify that nine from the twelve installed measuring devices (Chapter 4.2.2) worked properly. The others (three installed measurement devices) seem to be damaged during the machine erection or its own rotating movement.

The measured results for pressure drop (pressure difference between inner and out stator diameter) for each of nine working measuring devices are showed in the Figure 4.19.

Figure 4.19 – Measured results for pressure drop of nine working measuring devices.

Adding a mean curve and the uncertainties to it (according to chapter 4.2.3.2) in the graphic above, the graphic of Figure 4.20 is obtained.

Figure 4.20 – Measured results for pressure drop of nine working measuring devices, mean curve and uncertainties.

The measured results for the total flow volume rate, that is sum of all 24 delimited areas (as showed in Chapter 4.2.3), times 12 (total number of coolers), are showed in the Figure 4.21, with uncertainties (according to chapter 4.2.3.2).

Figure 4.21 – Measured results for total flow volume rate with uncertainties.

As showed in chapter 4.2.3.2, the uncertainties for the pressure drop results are mostly affected by the pressure measurements variation (evaluation type A) while for the flow volume rate results the uncertainties are mostly affected by the poor anemometer's accuracy.

5 Application to the Model Test and Prototype Geometries and Conditions

5.1 Numerical Calculation Applied to the Model Test

5.1.1 Computational Domain

Geometrical parameters of computational domain are 1:1 scaled from a geometry previously considered on model tests reported by Wustmann (2005), as described on chapter 4.1, and used by this work as a reference to validate the numerical calculation. However, instead of an entire hydrogenerator, the numerical calculation considers only the geometry of the region of interest, namely: Stator Core Air Duct, Air Gap (between Rotor and Stator) and Stator Outer Region (Figure 5.1). The computational domain is extended downstream five times to allow the use of outflow boundary condition.

Figure 5.1 – Computational domain.

Due to the axial symmetry of the flow, the geometry of numerical calculation represents only half of the Stator Core Air Duct, and its parameters are shown in Figure 5.2 and listed in Table 5.1.

Figure 5.2 – Geometric parameters of stator core air duct model.

Table 5.1 – Geometric data of numerical calculation

Description	Parameter	Value
Segment angle	Alfa	8.18°
Air gap pitch (half)	Ha	24.6 mm
Downstream to center	DS	1600.0 mm
Rotor diameter	Dr	1044.0 mm
Stator inner diameter	Di	1110.0 mm
Stator outer diameter	Do	1460.0 mm
Air duct width	Wd	59.3 mm
Air duct height (half)	Hd	3.6 mm
Winding bar width (half)	Wb	10.0 mm
Spacers web thickness	Ts	2.0 mm
Spacers radial positioning	Prs	5.0 mm
Spacers tangential positioning	Pts	13.2 mm
Wedge radial thickness	Tw	5.0 mm
Wedge radial positioning	Prw	1.0 mm
Wedge angle	Beta	30.0°

5.1.2 Boundary Conditions

The boundary conditions are based on operational data applied on the reference model test (described on chapter 4.1), restricted to the considered computational domain, shown in chapter 5.1.1.

Due to the modeling of only half Air Duct (axial symmetry), symmetry boundary condition was used at complete upper surface. This same consideration was done for the surfaces that define the connection between the modeled Air Duct to the neighbor one at the lower side. The lower surfaces of Air Gap and Downstream Extended regions were also considered with symmetry boundary condition.

At Inlet surface, tangential and radial velocity components were applied to the fluid, according respectively to the model rotational speed and experimentally measured air volume flow. At Outlet surface, fully developed outflow was considered by the application of uniform reference pressure boundary condition.

Due to high tangential velocity component of the fluid in the Air Gap region and the remaining tangential velocity component in the Downstream Extended Region, tangential surfaces are connected with rotational periodicity boundary condition. At all Walls representing the Stator Core (Inner Diameter, Outer Diameter and Air Duct Walls), “no slip” condition was applied.

The boundary conditions are summarized and showed in the Figure 5.3.

Figure 5.3 – Boundary conditions applied to stator core air duct model.

5.1.3 Discretization Grid

At the present work, the grids applied to the numerical simulations are block-structured with body-fitting edges, created by the application of the code ANSYS ICEM CFD V14.5.

The discretization is non-uniform, with the Air Gap and Air Duct being fine (Air Duct entrance region even finer) while the downstream extension region being as coarse as possible to save computation resources.

Figure 5.4 – Grid applied to the numerical simulations.

O-grid generation method was applied in order get the desired refinement level in the boundary layers without increasing unnecessarily the number of elements far from the walls (consequently increasing the computation effort), especially in the downstream extension region.

Figure 5.5 – O-Grid blocks applied for boundary layers cells.

The O-Grid Block method allows us to modify a single block to a five sub-block topology (seven sub-blocks in 3D). It arranges grid lines into an “O” shape to reduce skew where a block corner lies on a continuous curve or surface.

5.1.3.1 Verification of Grid Quality Metrics

During the grid preparation, the controlling of grid quality was made by monitoring some available metrics in ANSYS ICEM CFD V14.5. For a better understanding and clear comparison, the metrics results were normalized from zero (0), for worst quality, to one (1), for the best possible metric quality value.

The grid prepared for the numerical simulations of selected experimental data (designed with 427053 nodes and 457746 hexahedral cells), achieved the quality metrics shown in the table below:

Table 5.2 – Grid quality metrics obtained for validation simulation

Quality Metric	Minimum	Maximum	Average
Aspect ratio	0.001	0.970	0.079
Orthogonality	0.203	1.000	0.839
Skewness	0.203	1.000	0.840
Warping	0.784	1.000	0.993
Taper	0.203	1.000	0.840
Expansion ratio	0.040	1.000	1.000

The poorest results of grid quality metrics (especially aspect and expansion ratios), appear on downstream-extended region. However, it does not affect the results of proposed work (pressure drop in air duct entrance).

By checking grid aspect ratio only on air duct entrance region, the following results for aspect ratio are found: min. = 0.058, max = 0.970, average = 0.222. As shown in the figure below Figure 5.6, cells with lowest aspect ratio are located on near wall boundaries (grid inflation layers), where this metric can be relaxed (Tu, Yeoh, & Liu, 2013).

Figure 5.6 – Grid of air duct entrance region (lowest aspect ratio colored).

All the quality metrics achieved only in the region of air duct entrance are shown in the table below:

Table 5.3 – Grid quality metrics obtained for validation simulation (only air duct)

Quality Metric	Minimum	Maximum	Average
Aspect ratio	0.058	0.970	0.222
Orthogonality	0.261	1.000	0.888
Skewness	0.261	1.000	0.899
Warping	0.784	1.000	0.993
Taper	0.203	1.000	0.840
Expansion ratio	0.118	1.000	1.000

5.1.3.2 Grid Independence Study

Aiming to ensure the stability and accuracy of the numerical solution, the grid independence was evaluated by applying the same numerical simulation, using grid sets with different refinement levels and checking a resulting key variable: the pressure drop (Δp), which is taken from the maximum difference between static pressure measurements at the inner and outer diameter of stator core.

Since the focus of the present work is the pressure drop of the airflow to enter into the Air Duct and pass through it, the difference of grid refinement levels is applied to the global size of the cells in the region of the Air Duct.

Other important parameters to be controlled are the maximum and average dimensionless distance from the wall (y^+ and $\overline{y^+}$). To be able to assume that shear stress is entirely viscous in origin, avoiding the application of wall function approach, the criteria $y^+ < 11.63$ shall be applied (Versteeg & Malalasekera, 2007). The exact value of 11.63 is the intersection of the linear profile and the log-law, which is obtained from the solution of Equation (5.1) considering Karman's constant $k = 0.4187$ and the smooth walls roughness constant $E = 9.793$.

$$y^+ = \frac{1}{k} \ln(Ey^+) \quad (5.1)$$

Table 5.4 –Summary of the grid independence study

GRID	Number of Cells in the Air Duct	y_{max}^+	$\overline{y^+}$	Δp [Pa]	Mean Deviation [%]
GRID 1	38,344	6.97	1.63	327.154	1.536
GRID 2	75,528	6.97	1.60	337.953	1.487
GRID 3	422,448	6.96	1.56	338.969	1.483

5.1.3.3 Estimation of Uncertainty due to Discretization Errors

Using the three grid sets from the independence study (GRID 1, 2 and 3) and the corresponding pressure drop results, an estimation of uncertainty due to discretization errors was executed according to Celik, et al. (2008). The method for the estimation of discretization errors applies the Richardson extrapolation using results of three significantly different sets of grids. Formulations and results are shown in Table 5.5.

Table 5.5 – Estimation of uncertainty due to discretization

Description	Formulation	Value
Number of nodes of GRID 1	N_1	32,811
Number of nodes of GRID 2	N_2	65,291
Number of nodes of GRID 3	N_3	368,343
Average global cell size of GRID 1	h_1	8.69
Average global cell size of GRID 2	h_2	2.09
Average global cell size of GRID 3	h_3	0.28
Refinement factor	$r_{21} = h_1/h_2$	4.16
Refinement factor	$r_{32} = h_2/h_3$	7.58
Key variable (pressure drop) of GRID 1 [Pa]	Φ_1	327.154
Key variable (pressure drop) of GRID 2 [Pa]	Φ_2	337.953
Key variable (pressure drop) of GRID 3 [Pa]	Φ_3	338.969
Solution deviation from 2 to 1	$\varepsilon_{21} = \Phi_2 - \Phi_1$	1.080
Solution deviation from 3 to 2	$\varepsilon_{32} = \Phi_3 - \Phi_2$	1.016
Oscillatory convergence indicator	$s = 1 \cdot \text{sign}(\varepsilon_{32}/\varepsilon_{21})$	1
Apparent order	$p = \frac{1}{\ln(r_{21})} \cdot \ln \varepsilon_{32}/\varepsilon_{21} + q(p) $ $q(p) = \ln\left(\frac{r_{21}^p - s}{r_{32}^p - s}\right)$	-1.745
Richardson extrapolated value	$\Phi_{ext}^{21} = (r_{21}^p \Phi_1 - \Phi_2)/(r_{21}^p - 1)$	326.974
Approximate relative error	$e_a^{21} = (\Phi_1 - \Phi_2)/\Phi_1 $	3.30%
Extrapolated relative error	$e_{ext}^{21} = (\Phi_{ext}^{21} - \Phi_1)/\Phi_{ext}^{21} $	0.06%
Fine-grid convergence index	$CGI_{fine}^{21} = 1.25 e_a^{21}/(r_{21}^p - 1)$	0.07%

5.2 Numerical Calculation Applied to the Prototype

Numerical calculations were performed for the HPP (Hydro Power Plant) Teles Pires, which has been built in the Teles Pires River (a Tapajós River affluent), located in border of the Brazilian states of Pará (PA) and Mato Grosso (MT), in the cities of Jacareacanga (PA) and Paranaíta (MT). The HPP Teles Pires is being equipped with five power units and shall deliver 1.820 MW of installed capacity to the Brazilian grid.

5.2.1 Computational Domain

As defined for the computational domain applied to the simulation of model test (chapter 5.1.1), the computational domain of numerical calculation applied to the prototype considers also only the geometry of the region of interest, namely: Stator Core Air Duct, Air Gap (between Rotor and Stator) and Stator Outer Region (Figure 5.7).

Figure 5.7 – Air computational domain of numerical calculation applied to the prototype.

Differently from the geometry applied to the simulation of model test (chapter 5.1.1), the geometry of numerical calculation applied to the prototype has no axial symmetry, because of an axial connection between the air ducts in the region of frontal wedges, as shown in Figure 5.8.

Figure 5.8 – Detail of axial connection between air ducts in the region of frontal wedges.

Since the numerical calculation applied to the prototype is a thermal-fluid simulation, it also considers the geometry of solid bodies of calculation domain. It means that stator core sheets, air duct spacers and the conductor bar (with copper and insulation material) were modeled, as shown in Figure 5.9.

Figure 5.9 – Solid bodies computational domain (assimetric quarter) of numerical calculation applied to the prototype.

Data material of air and all the solid bodies applied to the numerical calculation are listed in Table 5.6.

Table 5.6 – Material data of numerical simulation applied to the prototype.

Component	Molar Mass [kg/kmol]	Density [kg/m ³]	Specific Heat Capacity [J/(KgK)]	Dynamic Viscosity [kg/(ms)]	Thermal Conductivity [W/(mK)]
Air	28.96	1.188	1004.4	1.831×10^{-5}	2.61×10^{-2}
Stator Core Sheets	55.85	7854	434	-	60.5
Air Duct Spacers	55.85	7854	434	-	60.5
Conductor Copper	63.55	8933	385	-	401
Conductor Insulation	180	1680	1000	-	0.249
Frontal Wedge	180	1680	1000	-	0.249

Geometry parameters of a whole Air Duct air volume cell computational domain are shown in Figure 5.10 and listed in Table 5.7.

Figure 5.10 – Air volume geometric parameters.

Table 5.7 – Geometric data of air volume domain.

Description	Parameter	Value
Segment angle	Alfa	0.5°
Air gap height pitch	Ha	36.0 mm
Downstream to center	DS	10412.5 mm
Rotor diameter	Dr	16487.0 mm
Stator inner diameter	Di	16530.0 mm
Stator outer diameter	Do	17210.0 mm
Air duct height	Hd	6.0 mm
Winding bar slot width (half)	Ws	10.6 mm
Winding bar slot length	Ls	161.5 mm
Winding bar slot radius	Rs	1.0 mm
Winding bar slot wedge thickness	Tws	5.5 mm
Winding bar slot wedge radial positioning	Pws	0.5 mm
Winding bar slot wedge width (half)	Wws	10.8 mm
Winding bar slot wedge angle	Beta	30.0°
Winding bar width (half)	Wb	10.1 mm
Winding bar length (inner)	Lbi	72.5 mm
Winding bar length (outer)	Lbo	72.7 mm
Winding bar round radius (inner)	Rbi	2.0 mm
Winding bar round radius (outer)	Rbo	3.0 mm
Winding bar main insulation thickness	Tbmi	3.2 mm
Winding bar separator thickness	Tbs	6.0 mm
Winding bar fixing spring thickness	Tbf	5.5 mm
Winding bar frontal wedge thickness	Tbw	4.7 mm
Winding bar frontal wedge angle	Gama	45.0°
Winding bar frontal wedge chamfer	Cbw	0.75 mm x 45.0°
Winding bar frontal wedge slot height	Tbwh	14.0 mm

Winding bar frontal wedge slot radius	Tbwr	1.6 mm
Spacers radial positioning (inner side)	Prsi	1.0 mm
Spacers radial positioning (outer side)	Prso	9.0 mm
Spacers tangential positioning	Pts	15.0 mm
Spacers beam web thickness	Ts	1.5 mm
Spacers beam flange width	Wfs	5.0 mm
Spacers beam flange thickness	Tfs	1.0 mm
Spacers beam round radius	Trs	0.5 mm
Spacers beam angle	Delta	75.0°

5.2.2 Boundary Conditions

The boundary conditions are based on operational data found in the prototype hydrogenerator (described on chapter 5.2), restricted to the considered computational domain, shown in chapter 5.2.1.

Due to the modeling of a whole Air Duct volume cell, the surfaces that define the boundaries of modeled Air Duct and the neighborhood at lower and upper sides are connected between them. This boundary condition was applied in the Air Gap and in the Downstream Extended regions.

As previously applied to the simulation of model test (chapter 5.1.2), tangential and radial velocity components were applied to the fluid at Inlet surface, according respectively to the prototype rotational speed and measured air volume flow. At Outlet surface, fully developed outflow was considered by the application of uniform reference pressure boundary condition. Due to high tangential velocity component of the fluid in the Air Gap region and the remaining tangential velocity component in the Downstream Extended Region, tangential surfaces are connected with rotational periodicity boundary condition.

Surfaces of solid bodies (except for copper conductors) located in axial boundaries are defined with Symmetry condition, while the surfaces located in the tangential boundaries and the respective mirrored pairs are connected with rotational periodicity boundary condition. The surfaces of copper conductors located in axial and tangential boundaries are defined as constant temperature surfaces (140°) to simulate the heat source.

At all interfaces between Air Duct volume cell and solid bodies (Stator Core Sheets, Air Duct Spacers, Conductor Insulation and Frontal Wedge) a Fluid x Solid connection condition was applied. Solid x Solid connections were applied between the different solid bodies, including those that compounds the Conductor Bar (copper and insulating materials).

Figure 5.11 – Boundary conditions applied to the air volume of prototype simulation.

5.2.3 Discretization Grid

As previously applied to the simulation of model test (chapter 5.1.3), the grid applied to the numerical simulations of prototype is block-structured with body-fitting edges and O-grid generation method, created by the application of the code ANSYS ICEM CFD V14.5.

Figure 5.12 – Grid applied to the prototype numerical simulations.

Figure 5.13 – O-Grid blocks applied for boundary layers cells.

5.2.3.1 Verification of Grid Quality Metrics

The grid prepared for the numerical simulations of prototype (designed with 847443 nodes and 956680 hexahedral cells), achieved the quality metrics shown in the table below (normalized from zero (0), for worst quality, to one (1), for the best possible metric quality value):

Table 5.8 – Grid quality metrics obtained for prototype simulation

Quality Metric	Minimum	Maximum	Average
Aspect ratio	0.002	0.983	0.123
Orthogonality	0.004	1.000	0.829
Skewness	0.004	1.000	0.844
Warping	0.000	1.000	0.998
Taper	0.001	1.000	0.150
Expansion ratio	0.000	1.000	1.000

By checking quality metrics achieved only in the region of air duct, the following results are found:

Table 5.9 – Grid quality metrics obtained for prototype simulation (only air duct)

Quality Metric	Minimum	Maximum	Average
Aspect ratio	0.003	0.983	0.064
Orthogonality	0.004	1.000	0.799
Skewness	0.004	1.000	0.815
Warping	0.000	1.000	0.998
Taper	0.001	1.000	0.130
Expansion ratio	0.001	1.000	1.000

Figure 5.14 – Grid of air duct region.

6 Results

6.1 Numerical Method Validation by Model Test

Despite of CFD being an efficient computational tool, it is a method where the equations are approximated and solved iteratively for a discrete system with a finite number of elements. In addition, by the application of RANS equations, some equations must be approximated to the reality by mathematical models (e.g. turbulence models) for the closure of the set of equations. Thus, the numerical method requires always a robust validation.

The numerical results presented in this chapter were obtained by simulating the reference experiment described on chapter 4.1. The intermediate tested grid refinement level (GRID 2, as presented in chapter 5.1.3) was applied and a convergence criterion to residuals for the flow equations less than 10^{-5} was considered.

The numerical simulation, as described above, could be executed in 52 minutes and 30 seconds, using a computer with 4.00 GB of RAM (Random Access Memory) and a processor of 2.10 GHz.

6.1.1 Flow Field Results

For a better understanding of the fluid dynamics behavior of the flow, graphic outputs were obtained by post-processing of the numerical results in ANSYS CFX 14.5. The graphics represent the simulation with boundary conditions regarding to 12.8 Hz for the rotor speed.

Analyzing the static pressure result in Figure 6.1, it is possible to identify a pressure side (upper side of air duct inlet) and a suction side (lower side of air duct inlet), caused by the counterclockwise rotational movement of the flow in the air gap region. Since the pressure decreases a lot just after the air duct inlet (except for some recirculation regions), it is clear that the pressure drop is mainly affected by the necessary energy for the air to change from its rotational movement and flow radially into the air duct.

Figure 6.1 – Grapghic Output – Static pressure.

The Figure 6.2 (volume flow streamlines colored by velocities) shows clearly the flow concentration on pressure side of the air duct and, at the same time, some recirculation region on suction side of the air duct. On the outlet of the stator core, the non-homogeneous velocity profile (a not totally developed flow condition) is responsible for rotational movement of the air on downstream-extended region.

Figure 6.2 – Grapghic Output – Volume flow streamlines colored by velocities.

6.1.2 Experimental and Numerical Data Comparisons

The proposed numerical method is validated specifically to simulate the flow dynamics behavior and to obtain the pressure drop coefficient of the airflow to enter into the air duct of a hydrogenerator stator core, and pass through it.

First, the agreement between experimental and numerical results of flow volume rate for different rotational speeds, shown in Figure 6.3, confirms that the boundary conditions applied to the numerical simulation (radial and tangential velocity components) are consistent to the experimental flow volume rate.

Figure 6.3 – Comparisons between experimental and numerical results of flow volume rate.

Comparing numerical and experimental results of pressure drop, the curves in Figure 6.4 are obtained.

Figure 6.4 – Comparisons between experimental and numerical results of pressure drop.

The pressure drop coefficient (ζ) is calculated by Equation (6.1) and is commonly used in general fluid dynamics experimental reports (Idelchik & Fried, 1989) and model tests of hydrogenerator ventilation system (Wustmann, 2005).

$$\zeta = \frac{2\Delta p}{\rho V^2} \quad (6.1)$$

Applying the geometrical parameters of the air duct shown in chapter 5.1.1 (Hd , Wd and Ts), a hydraulic diameter (D_h) for a whole air duct can be calculated as following:

$$D_h = \frac{4 \cdot Hd \cdot (Wd - 2 \cdot Ts)}{6 \cdot Hd + 2 \cdot (Wd - 2 \cdot Ts)} \quad (6.2)$$

From the above air duct hydraulic diameter (D_h), the Reynolds number can be calculated considering the radial component of the airflow velocity (V_r , defined on the inlet boundary condition of the air duct, chapter 5.1.2) and the kinematic viscosity ν of the air, reported in Table 4.1, chapter 4.1:

$$Re = \frac{V_r \cdot D_h}{\nu} \quad (6.3)$$

Comparing experimental and numerical results for pressure drop coefficients as functions of Reynolds number, the curves in Figure 6.5 are obtained.

Figure 6.5 – Comparisons between experimental and numerical results of pressure drop coefficient.

The comparison above shows that the difference between the experimental and numerical pressure drop coefficients for Reynolds numbers higher than 7500 is 2% at maximum. For lower Reynolds numbers, the difference increases significantly and reaches 10% for $Re = 4500$.

The most reasonable hypothesis for higher discrepancy at lower Reynolds Numbers is the non-constant inlet velocity over time, since the inlet flow in the experiment is driven by air ducts located on rotating shaft (seven rotating air ducts in the circumference). By the increase of rotational speed (and consequently the Reynolds number), the inlet condition becomes closer to a steady-state situation, as considered by the numerical simulations.

The second reason can be assigned to the inlet velocity, which may be not homogeneously distributed on inlet surface, as considered on the numerical model.

6.2 Numerical Results Applied to the Prototype

The numerical results of presented in this chapter were obtained by applying the boundary conditions regarding to a generator rotor speed of 75 rpm, which is its rated speed, and the copper conductors at constant temperature of 100°C. The convergence criterion to residuals for the flow equations less than 10^{-4} was considered.

The numerical simulation, as described above, could be executed in 7 hours, 18 minutes and 37 seconds, using a computer with 4.00 GB of RAM (Random Access Memory) and a processor of 2.10 GHz.

6.2.1 Flow Field Results

Similar to previously showed in the simulation of model test (chapter 6.1.1), analyzing the static pressure result in Figure 6.6, it is also possible to identify a pressure side (left side of air duct inlet) and a suction side (right side of air duct inlet), caused by the rotational movement of the flow in the air gap region. The pressure drop mainly affected by the necessary energy for the air to change from its rotational movement and flow radially into the air duct is noted too.

Figure 6.6 – Prototype simulation graphic output – Static pressure.

Differently from the results of model test simulation (chapter 6.1.1), the Figure 6.7 and Figure 6.8 (volume flow streamlines colored by velocities) show flow less sideward concentrated on pressure side of the air duct and, at the same time, less recirculation regions on suction side of the air duct. It happens because of the optimized design of air duct entrance (frontal wedge) when compared to the tested model.

Figure 6.7 – Prototype simulation graphic output – Streamlines based on core vortex colored by velocity.

Figure 6.8 – Prototype simulation graphic output –Volume flow streamlines colored by velocities – Detail.

Thermal results can be observed in the Figure 6.9, which shows the temperature of the air in the axial middle plane of the air duct.

Figure 6.9 – Prototype simulation graphic output – Air temperature.

Heat flux through the solids surfaces in contact with the cooling air can be observed below in the Figure 6.10.

Figure 6.10 – Prototype simulation graphic output – Heat flux.

6.2.2 Field Measurements and Numerical Data Comparison

Since pressure values from the ventilation system of a real hydrogenerator (from inner and outer stator core diameter) could be measured, as showed in chapter 4.2.3, they were used as a reference to be compared to the prototype numerical calculation results.

As the comparison between model test experimental data and its numerical simulation (chapter 6.1.2), the comparison below also aims to validate the proposed numerical method specifically to simulate the flow dynamics behavior and to obtain the pressure drop coefficient of the airflow to enter into the air duct of a hydrogenerator stator core, and pass through it.

Comparing the field measurements and numerical results of pressure drop, the curves in Figure 6.11 are obtained.

Figure 6.11 – Comparisons between field measurements and numerical results of pressure drop.

The comparison above shows that the difference between the field measurements and numerical pressure drop coefficients is high especially for higher rotational speed, reaching 77% at maximum speed. Nevertheless, numerical results are still covered by field measurements uncertainties for all rotational speed range.

The most reasonable hypothesis for the high discrepancy between the field measurements and numerical is the field measurements uncertainties itself. It can be observed by the fact that both, the uncertainties and the discrepancy increase with the rotational speed.

The second reason can be assigned to the quality of measuring devices. Because the measuring devices were not intended during the machine design conception, but installed afterwards with some adaptations, its instability could be noted by the measuring results and the fact that three of them (25%) were clearly damaged.

6.3 Determination of Coefficients

Based on numerical simulations prepared for the model test and the prototype, compared and validated by experimental data (chapter 6.1.2) and field measurements (chapter 6.2.26.1.2), a set of simulations were executed varying the operational parameters in order to prepare trend curves.

The curves presented in the next chapters can be used to improve existing analytical calculation routines of whole ventilation systems of hydrogenerators.

6.3.1 Pressure Drop Coefficient

The airflow pressure drop in the air duct inlet is mostly affected by the incidence angle of the flow in the inlet front wedges, that is, the ratio between radial velocity V_r (defined by the air volume flow) and tangential velocity V_t (defined by the machine rotational speed). Therefore, results curves present the pressure drop coefficients as function of velocity ratio V_r/V_t , which can be directly used by existing analytical calculation routines of whole ventilation systems of hydrogenerators.

As showed in Chapters 5.1.1 and 5.2.1, the designs of air duct inlet of the model test and the prototype numerically simulated by this work are different. Therefore, two different curves were prepared for the pressure drop coefficient.

The first curve (Figure 6.12) is applicable for 60° front wedge without any relief.

Figure 6.12 – Pressure drop coefficient as function of velocity ratio for 60° frontal wedge without relief.

Checking the velocity vectors for three different velocities ratios ($V_r/V_t = 0.37$; $V_r/V_t = 1.00$; $V_r/V_t = 1.67$), the following figures were obtained:

Figure 6.13 – Velocity vectors for 60° frontal wedge without relief ($V_r/V_t = 0.37$).

Figure 6.14 – Velocity vectors for 60° frontal wedge without relief ($V_r/V_t = 1.00$).

Figure 6.15 – Velocity vectors for 60° frontal wedge without relief ($V_r/V_t = 1.67$).

The figures above show the difficulty the flow faces to enter into the air duct at low ratio value V_r/V_t (Figure 6.13), due to its high velocity tangential component. By the increase of the V_r (Figure 6.14 and Figure 6.15), the flow becomes less tangential and more aligned to the air ducts, which makes the flow entrance into the channels easier.

Checking the volume flow streamlines, colored by velocities, for the same three different velocities ratios from above, the following figures were obtained:

Figure 6.16 – Volume flow streamlines colored by velocities for 60° frontal wedge without relief ($V_r/V_t = 0.37$).

Figure 6.17 – Volume flow streamlines colored by velocities for 60° frontal wedge without relief ($V_r/V_t = 1.00$).

Figure 6.18 – Volume flow streamlines colored by velocities for 60° frontal wedge without relief ($V_r/V_t = 1.67$).

The figures of flow streamlines above also show that at low ratio value V_r/V_t (Figure 6.16), the poorly aligned flow in entrance portion of the air duct creates large recirculation areas in this region, which obstructs the airflow path. This phenomenon is minimized by the increasing of the V_r (Figure 6.17 and Figure 6.18), when the recirculation areas are reduced and the flow occupies almost the whole air duct width.

The second curve (Figure 6.19) is applicable for 60° front wedge with 45° relief.

Figure 6.19 – Pressure drop coefficient as function of velocity ratio for 60° frontal wedge with 45° relief.

Checking again the velocity vectors for three different velocities ratios ($v_r/v_t = 0.07$; $v_r/v_t = 0.28$; $v_r/v_t = 1.55$), the following figures were obtained:

Figure 6.20 – Velocity vectors for 60° frontal wedge with 45° relief ($V_r/V_t = 0.07$).

Figure 6.21 – Velocity vectors for 60° frontal wedge with 45° relief ($V_r/V_t = 0.28$).

Figure 6.22 – Velocity vectors for 60° frontal wedge with 45° relief ($V_r/V_t = 1.55$).

The same phenomenon observed with 60° frontal wedges without relief, that is, the entrance of the airflow into the air ducts is improved by the increase of the velocity radial component (V_r) due to a better flow alignment to the channel, also happens for 60° frontal wedges with 45° relief. However, the 45° relief presents an advantage over the frontal wedges without relief: a better alignment of the flow to the air duct and a better usage of the channel width. It can be noted by comparing the first duct (from right to left side) of Figure 6.15 and Figure 6.22.

Checking again the volume flow streamlines, colored by velocities, for the same three different velocities ratios from above, the following figures were obtained:

Figure 6.23 – Volume flow streamlines colored by velocities for 60° frontal wedge with 45° relief ($V_r/V_t = 0.07$).

Figure 6.24 – Volume flow streamlines colored by velocities for 60° frontal wedge with 45° relief ($V_r/V_t = 0.28$).

Figure 6.25 – Volume flow streamlines colored by velocities for 60° frontal wedge with 45° relief ($V_r/V_t = 1.55$).

The benefit of the 45° relief showed by the velocity vectors is confirmed by the figures of flow streamlines above. It is clear that the 45° relief reduces the recirculation areas and promotes a total occupation of the duct, by comparing the first duct (from right to left side) of Figure 6.18 and Figure 6.25.

6.3.2 Convective Heat Transfer Coefficient

In ANSYS CFX code (ANSYS CFX, 2012) the Heat Flux variable is the total heat flux into the domain, including convective and radiative contributions. Since radiation was not considered in the simulations of this work, it represents only convective part.

Considering results of heat flux through all the surfaces in contact with the airflow (q), the total surface area (A) and the global temperature difference (ΔT), the heat transfer (h) can be calculated with equation (6.4) below:

$$h = \frac{q}{\Delta T \cdot A} \quad (6.4)$$

In order to create a trend curve for the convective heat transfer coefficient (h), a set of numerical simulations starting from the prototype simulation, was done, varying the air radial velocity in the air duct channels (V_r). The result curve is showed in Figure 6.26.

Figure 6.26 – Convective heat transfer coefficient as function of radial velocity in air channels.

Checking the temperatures (of air and solid bodies) for three different radial velocities ($V_r = 1.00 \text{ m.s}^{-1}$; $V_r = 5.66 \text{ m.s}^{-1}$; $V_r = 12.00 \text{ m.s}^{-1}$), the following figures were obtained:

Figure 6.27 – Temperature of air and solid bodies ($V_r = 1.00 \text{ m.s}^{-1}$).

Figure 6.28 – Temperature of air and solid bodies ($V_r = 5.66 \text{ m.s}^{-1}$).

Figure 6.29 – Temperature of air and solid bodies ($V_r = 12.00 \text{ m.s}^{-1}$).

The figures above show that the convective heat transfer is intensified with the increase of airflow velocity (V_r), as expected by the behavior of convective heat transfer coefficients results. At lower velocity (Figure 6.27), the air gets warmer and loses its capacity to remove heat of solid bodies (conductor and insulation material). The extent to which the airflow velocity (V_r) increases (Figure 6.28 and Figure 6.29), the temperature of air and insulation materials around copper conductors decreases.

The convective heat transfer improvement by increase of radial velocity can also be identified with reduced temperatures of insulation surfaces, as showed in the following figures sequence (Figure 6.30, Figure 6.31 and Figure 6.32):

Figure 6.30 – Temperature of insulation surfaces ($V_r = 1.00 \text{ m.s}^{-1}$).

Figure 6.31 – Temperature of insulation surfaces ($V_r = 5.66 \text{ m.s}^{-1}$).

Figure 6.32 – Temperature of insulation surfaces ($V_r = 12.00 \text{ m.s}^{-1}$).

7 Conclusions

By the literature review, it can be observed that some experimental and numerical analysis applied to hydrogenerators ventilation systems have been performed over the last years. However, validation studies dedicated to hydrogenerators ventilation systems components, as presented in this work, are still rare. After the importance and impact of the ventilation systems in the hydro business was exposed and the reasons for the selection of the object of study was argued, the objective of this work represents an important contribution for the hydro industry.

There are very few published reports with detailed experimental analysis that can be used for validation of a numerical simulation. Wustmann (2005) is one exception and his reported model test could be numerically simulated by this work, without assuming any hypotheses for any missing data.

Regarding field measurements applied to hydrogenerators ventilation systems, the literature review showed a complete lack of reported scientific works. For the present work, it was done, apparently for the first time, despite all the following difficulties:

- Logistics: like almost all hydro power plants, HPP Teles Pires (where field measurements for this work were performed) is located in very hard accessible site.
- Preparation Planning: due to the complexity, the huge dimensions and long erection time of hydrogenerators, the installation of measuring devices shall be done during erection and well planned much earlier. Once the rotor and the stator are placed in their final positions, nobody can access the air gap any more.
- Measuring Schedule: measurements shall be done during the commissioning works of a hydrogenerator, which is the last step before to deliver a machine to start generating power. It is not a surprise if enterprises of this magnitude have some delays, and this last step occurs under high time pressure.
- Environment: a hydro power plant, especially inside of the generator pit, the environment is not ideal for scientific measurements. Not controlled temperature or air relative humidity, dust, oil mist and electrical interferences are some examples of common problems.
- Measurements Uncertainties: measuring equipment and methods normally used during machine commissioning have not the same accuracy required for

experimental process at laboratory. This, added to low precision in the control of some operation parameters like rotational speed, results in high measurement uncertainties.

The application of proposed numerical method (CFD) on the study of hydrogenerators ventilation system proved to be an efficient tool for a better understanding of ventilation issues (e.g. ventilation losses and heat exchange) and development of the cooling design.

During modeling of numerical simulation, the attention with the grid generation strategy and controlling of grid quality metrics resulted in a stable and accurate numerical solution. For the present work, the uncertainties due to discretization errors were proved very low by means of a grid independence study and application of an uncertainty estimation procedure based on Richardson extrapolation.

The comparison between results achieved by the numerical calculation applied to the model test and available experimental data showed that the proposed numerical method (including boundary conditions, numerical resolution strategy, turbulence model, discretization grid, etc.) is validated for the upper region of the analyzed range (that is, at higher Reynolds Numbers or higher rotational speeds). Reasonable hypotheses for higher discrepancy between numerical and experimental data at lower Reynolds Numbers were presented, which is mostly assigned to the very non-steady-state condition of operation parameters.

The comparison between results achieved by the numerical calculation applied to the prototype and field measurements executed in HPP Teles Pires showed some higher differences than numerical and experimental data comparison. Nevertheless, two points must be highlighted: 1) the numerical results are covered by the considerable uncertainties of the field measurement data; 2) the flow behavior of numerical calculation (pressure drop increasing as function of rotational speed) follows field measurements data tendency.

The result of this work, the trend curves of pressure drop (for both type of frontal wedges, with and without relief) and convective heat transfer (for frontal wedge with relief), could be created from a set of simulations (based on numerical simulations prepared for the model test and the prototype) varying the operational parameters. These curves have directly applicability on the analytical calculations of complete hydrogenerators ventilation systems.

7.1 Scope for Future Work

For future work, the validation process of numerical calculation compared to field measurements must be continued. A new hydrogenerator, which is in its beginning of design process, must be chosen. Instead of installing pressure measuring points in the air duct channels (as done in this work), definitive pressure tapings shall be designed and installed directly on the front wedges. This probably solves instability problems of presented measurements and shall improve measured results. Additionally to this improvement, a measuring method for the convective heat transfer shall be developed. A possible way to do that seems to be by installing temperature sensors over the conductors surfaces, since the temperature inside of the bars is known by existing sensors.

Considering that the highest part of the pressure drop occurs in the inlet of the air ducts of stator core and especially due to its poor aerodynamic design, there is a great potential of improvement by application of an optimization routine in a parametric study changing the geometric design of the frontal wedges and the air duct itself. The goal function of the optimization routine shall be developed in order to include the pressure drop minimization and convective heat transfer maximization, at the same time.

Finally, other components and regions of hydrogenerators ventilation systems can be studied as done by this work and suggested above. In this sense, the following components would be indicated: the rotor rim channels, due to its considerable pressure drop; and the passing air between rotor poles and stator winding overhangs, due to the generated and dissipated heat.

References

- Altea, C., & Meyer, A. C. (2007). Aplicação de simulação CFD no estudo do sistema de ventilação de hidrogeradores. *CIGRÉ - SNPTTE 2007 - 19th National Seminar on Production and Transmission of Electricity*. (p. GGH15). Rio de Janeiro, Brazil: Voith Siemens Hydro Power.
- Alter, S. J. (2004). A Structured-Grid Quality Measure for Simulated Hypersonic Flows. *Proceedings of the 42nd AIAA Aerospace Sciences Meeting and Exhibit* (pp. AIAA 2004-0612). Reno, USA: NASA Langley Research Center.
- ANSYS CFX, I. (2012). ANSYS CFX 14.5 - Theory Guide.
- ANSYS FLUENT, Inc. (2012). *ANSYS FLUENT 14.5 - User's Manual*.
- ANSYS ICEM, Inc. (2012). ANSYS ICEM CFD 14.5 - User's Manual.
- Barth, T. J., & Jespersen, D. C. (1989). The Design and Application of Upwind Schemes on Unstructured Meshes. *AIAA*, pp. 89-0366.
- Boglietti, A., Cavagnino, A., Staton, D., Martin Shanel, M. M., & Mejuto, C. (2009, March). Evolution and Modern Approaches for Thermal Analysis of Electrical Machines. *IEEE TRANSACTIONS ON INDUSTRIAL ELECTRONICS*, pp. VOL. 56, NO. 3.
- Celik, I., Ghia, U., Roache, P., Freitas, C., Coleman, H., & Raad, P. (2008, July). Procedure for Estimation of Uncertainty Due to Discretization in CFD Applications. *Journal of Fluids Engineering*, pp. 1 - 4.
- Ferziger, J., & Peric, M. (2002). *Computational Methods for Fluid Dynamics* (3rd ed.). Berlin, Germany: Springer-Verlag.
- Ferziger, J., & Peric, M. (2002). *Computational Methods for Fluid Dynamics* (3rd Edition ed.). Berlin, Germany: Springer-Verlag.
- Field, D. A. (2000). Qualitative measures for initial meshes. *International Journal for Numerical Methods in Engineering*, pp. 887-906.
- Freitag, L., Knupp, P., Leurent, T., & Melander, D. (2002). MESQUITE Design: Issues in the Development of a Mesh Quality Improvement Toolkit. *Proceedings of the 8th International Conference on Grid Generation Methods for Numerical Field Simulations*, (pp. 159-168). Honolulu, USA.

- Gregorc, B. (2011). Investigating the Malfunction of a Hydrogenerator's Cooling-System. *TMT 2011 15th International Research/Expert Conference "Trends in the Development of Machinery and Associated Technology"* (pp. 861–864). Prague, Czech Republic: University of Maribor.
- Hartono, E., Golubev, M., Moradnia, P., Chernoray, V., & Nilsson, H. (2012a). *A PIV Study of the Cooling Air Flow in an Electric Generator Model*. Gothenburg, Sweden: Chalmers University of Technology.
- Hartono, E., Golubev, M., Moradnia, P., Chernoray, V., & Nilsson, H. (2012b). PIV Measurement of Air Flow in a Hydro Power Generator Model. *16th International Symposium on Applications of Laser Techniques to Fluid Mechanics* (p. Session 4.1.). Lisbon, Portugal: Chalmers University of Technology.
- Houde, S., Vincent, P., & Hudon, C. (2008). Simulations Strategies of the Cooling Flow for Large Hydro-Generator. *Hydropower & Dams*(Issue 6), 93–98.
- Idelchik, I. E., & Fried, E. (1989). *Flow Resistance – A Design Guide for Engineers*. Philadelphia: Taylor & Francis Ltd.
- IEC. (1991). 34-6 - International Standard - Rotating Electrical Machines - Methods of cooling (IC Code). Geneva, Switzerland: International Electrotechnical Commission.
- INMETRO. (2008). *Guia para Expressão da Incerteza de Medição*.
- Knupp, P. M. (2007). Remarks on Mesh Quality. *Proceedings of the 45th AIAA Aerospace Sciences Meeting and Exhibit, 2007*. Reno, USA: Sandia National Laboratories.
- Kramer, C. W., Haluska Jr., C., & Namoras, J. F. (2001). *Geração de Energia - Hidrogeradores*. São Paulo, Brazil: Voith Siemens Hydro Power.
- Launder, B. E., & Spalding, D. B. (1974). The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, pp. 269-289.
- Maliska, C. R. (2004). *Transferência de Calor e Mecânica dos Fluidos Computacional*. São Paulo, Brazil: LTC.
- Mavriplis, D. J., Vassberg, J. C., Tinoco, E. N., Mani, M., Brodersen, O. P., Eisfeld, B., et al. (2008). Grid Quality and Resolution Issues from the Drag Prediction Workshop Series. *American Institute of Aeronautics and Astronautics*, pp. 2008-930.

- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, Vol. 32, pp. 1598-1605.
- Meyer, A. C. (2006). Ventilation Systems for Hydrogenerators. São Paulo, Brazil: Voith Hydro Power Generation.
- MME. (2011). *Plano Decenal de Expansão de Energia 2020*. Brasília, Brazil: MINISTÉRIO DE MINAS E ENERGIA, EMPRESA DE PESQUISA ENERGÉTICA.
- Moradnia, P. (2010). *CFD of Air Flow in Hydro Power Generators. Thesis for the Degree of Licentiate of Engineering*. Gothenburg, Sweden: Chalmers University of Technology.
- Moradnia, P., & Nilsson, H. (2010). CFD of Air Flow in Hydro Power Generators for Convective Cooling, using OpenFOAM. *5th European Conference on Computational Fluid Dynamics, ECCOMAS CFD*. Lisbon, Portugal: Chalmers University of Technology.
- Moradnia, P., & Nilsson, H. (2011). A Parametric Study of the Air Flow in an Electric Generator through Stepwise Geometry Modifications. *ECCOMAS Thematic Conference, CFD and OPTIMIZATION* (p. 040). Antalya, Turkey: Chalmers University of Technology.
- Moradnia, P., Chernoray, V., & Nilsson, H. (2011). Experimental and Numerical Investigation of the Cooling Air Flow in an Electric Generator. *HEFAT 2011, 8th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics* (pp. 242-249). Pointe Aux Piment, Mauritius: Chalmers University of Technology.
- Moradnia, P., Chernoray, V., & Nilsson, H. (2012a). Experimental and Numerical Study of Cooling Air Flow in a Hydroelectric Generator. *ETMM9 European Research Community On Flow, Turbulence and Combustion*. Gothenburg, Sweden: Chalmers University of Technology.
- Moradnia, P., Chernoray, V., & Nilsson, H. (2014, February 24). Experimental assessment of a fully predictive CFD approach, for flow of cooling air in an electric generator. *Applied Energy*, pp. 223–230.
- Moradnia, P., Golubev, M., Chernoray, V., & Nilsson, H. (2013, November 9). Flow of cooling air in an electric generator model – An experimental and numerical study. *Applied Energy*, pp. 644–653.

- Moradnia, P., Nilsson, H., Page, M., Beaudoin, M., Torriano, F., Morissette, J., et al. (2012b). *Transient and steady-state air flow simulations in generators using OpenFOAM*. Gothenburg, Sweden: Chalmers University of Technology.
- Rhie, C., & Chow, W. (1982). A Numerical Study of the Turbulent Flow Past an Isolated Airfoil with Trailing Edge Separation. *AIAA*, pp. 82-0998.
- Stimpson, C. J., Ernst, C. D., Knupp, P., Pébay, P. P., & Thompson, D. (2007). *The Verdict Geometric Quality Library*. Albuquerque, USA: Sandia National Laboratories.
- Torriano, F., Lancial, N., Lévesque, M., Rolland, G., Hudon, C., Beaubert, F., et al. (2014, May 10). Heat transfer coefficient distribution on the pole face of a hydrogenerator scale model. *Applied Thermal Engineering*, pp. 153-162.
- Tu, J., Yeoh, G., & Liu, C. (2013). *Computational Fluid Dynamics - A Practical Approach*. Oxford, UK: Butterworth-Heinemann-ELSEVIER.
- Ujiie, R., Arlitt, R., & Etoh, H. (2006). Application of Computational Fluid Dynamics (CFD) on Ventilation-Cooling Optimization of Electrical Machines. *REVIEW ENERGY TECHNOLOGIES - generation, transmission and distribution of electric and thermal energy (ICEMENERG), contents 4*, pp. 17-22.
- VDE. (1996). 0530-6 - Deutsche Norm - Drehende elektrische Maschine - Einteilung der Kühlverfahren (IC-Code). Berlin, Germany: Deutsche Elektrotechnische Kommission in DIN und VDE (DKE).
- VDI. (2006). *Wärmeatlas*. Berlin Heidelberg, Germany: Verein Deutscher Ingenieure, VDI-Gesellschaft Verfahrenstechnik und Chemieingenieurwesen.
- Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics - The Finite Volume Method* (Second Edition ed.). Harlow, UK: Pearson Education Limited.
- Vogt, B., & Lahres, S. (2013). Calculation of Cooling and Ventilation of Large Hydro Generators with Advanced 3D Numerical Methods. *International HYDRO Conference*. Innsbruck, Austria: Voith Hydro Holding GmbH & Co. KG.
- Wiedemann, E., & Kellenberger, W. (1967). *Konstruktion elektrischer Maschinen*. Berlin / Heidelberg: Springer-Verlag.
- Wilcox, C. D. (1998). *Turbulence Modeling for CFD*. La Canada: DCW Industries, Inc.

- Wustmann, W. (2005). Experimentelle aerodynamische Untersuchungen des Kühlluftkreislaufes eines Generators zur Bestimmung von Verlustbeiwerten. *Master Engineer thesis*. Dresden, Germany: Technical University of Dresden.
- Yanagihara, J. I., & Rodrigues, R. (1998). *Análise do fluxo de ar nos canais de ventilação do estator de hidrogeradores*. São Paulo, Brazil: Siemens Brasil (KWU), Polytechnic School of University of São Paulo.

Appendix

Appendix A: Uncertainty of Pressure Measurements

	Rotational Speed [rpm]											
	18,75			37,50			56,25			75,00		
j	X _j [Pa]	I _j [mA]	(X _j -X _m) ²	X _j [Pa]	I _j [mA]	(X _j -X _m) ²	X _j [Pa]	I _j [mA]	(X _j -X _m) ²	X _j [Pa]	I _j [mA]	(X _j -X _m) ²
1	15,619	5,00	87,820	56,228	6,40	1090,486	121,826	8,60	4361,942	193,673	10,70	10534,456
2	31,238	5,80	39,031	99,960	8,00	114,704	209,291	11,40	458,816	337,365	15,50	1685,513
3	31,238	6,00	39,031	93,713	7,90	19,914	187,425	11,00	0,199	281,138	14,10	230,205
4	34,361	6,00	87,820	124,950	8,80	1274,490	259,271	13,10	5097,960	406,088	17,70	12051,100
5	28,114	5,90	9,758	103,084	8,30	191,373	234,281	12,50	2153,888	359,231	16,50	3959,084
6	15,619	5,30	87,820	74,970	7,20	203,918	146,816	9,40	1685,513	253,024	12,80	1873,699
7	18,743	5,30	39,031	71,846	7,00	302,891	156,188	9,70	1003,860	243,653	12,40	2772,812
	X _m	I _m	s(X _j)	X _m	I _m	s(X _j)	X _m	I _m	s(X _j)	X _m	I _m	s(X _j)
	24,990	5,61	8,07	89,250	7,66	23,09	187,871	10,81	49,60	296,310	14,24	74,28
			s(X _m) [Pa]			s(X _m) [Pa]			s(X _m) [Pa]			s(X _m) [Pa]
			3,05			8,73			18,75			28,08

Appendix B: Uncertainty of Velocity Measurements

				Rotational Speed [rpm]											
				18,75			37,50			56,25			75,00		
j	Passage area length - L_1 [m]	Passage area height - L_2 [m]	Air passage area - A [m ²]	Velocity - X_j [m/s]	Air Flow - Q_j [m ³ ·s ⁻¹]	$(X_j - X_m)^2$	Velocity - X_j [m/s]	Air Flow - Q_j [m ³ ·s ⁻¹]	$(X_j - X_m)^2$	Velocity - X_j [m/s]	Air Flow - Q_j [m ³ ·s ⁻¹]	$(X_j - X_m)^2$	Velocity - X_j [m/s]	Air Flow - Q_j [m ³ ·s ⁻¹]	$(X_j - X_m)^2$
1	0,45	0,40	0,18	0,62	0,112	0,03	1,51	0,272	0,07	2,49	0,448	0,24	3,28	0,590	0,25
2	0,45	0,40	0,18	0,60	0,108	0,02	1,36	0,245	0,02	2,21	0,398	0,04	2,98	0,536	0,04
3	0,45	0,40	0,18	0,61	0,110	0,02	1,38	0,248	0,02	2,07	0,373	0,00	2,85	0,513	0,01
4	0,45	0,40	0,18	0,55	0,099	0,01	1,48	0,266	0,06	2,28	0,410	0,08	3,25	0,585	0,22
5	0,45	0,40	0,18	0,46	0,083	0,00	1,32	0,238	0,01	2,18	0,392	0,03	2,93	0,527	0,02
6	0,45	0,40	0,18	0,47	0,085	0,00	1,44	0,259	0,04	2,34	0,421	0,11	3,22	0,580	0,20
7	0,45	0,40	0,18	0,46	0,083	0,00	1,32	0,238	0,01	2,08	0,374	0,01	2,98	0,536	0,04
8	0,45	0,40	0,18	0,45	0,081	0,00	1,22	0,220	0,00	1,97	0,355	0,00	2,77	0,499	0,00
9	0,45	0,40	0,18	0,43	0,077	0,00	1,20	0,216	0,00	1,93	0,347	0,01	2,65	0,477	0,02
10	0,45	0,40	0,18	0,44	0,079	0,00	1,18	0,212	0,00	1,94	0,349	0,00	2,70	0,486	0,01
11	0,45	0,40	0,18	0,47	0,085	0,00	1,24	0,223	0,00	1,96	0,353	0,00	2,68	0,482	0,01
12	0,45	0,40	0,18	0,53	0,095	0,01	1,35	0,243	0,01	2,22	0,400	0,05	3,07	0,553	0,09
13	0,45	0,40	0,18	0,47	0,085	0,00	1,24	0,223	0,00	1,97	0,355	0,00	2,80	0,504	0,00
14	0,45	0,40	0,18	0,45	0,081	0,00	1,10	0,198	0,02	1,83	0,329	0,03	2,50	0,450	0,08
15	0,45	0,40	0,18	0,43	0,077	0,00	1,05	0,189	0,04	1,79	0,322	0,05	2,40	0,432	0,14
16	0,45	0,40	0,18	0,42	0,076	0,00	1,05	0,189	0,04	1,77	0,319	0,05	2,42	0,436	0,13
17	0,45	0,40	0,18	0,41	0,074	0,00	1,09	0,196	0,02	1,81	0,326	0,04	2,45	0,441	0,11
18	0,45	0,40	0,18	0,40	0,072	0,00	1,17	0,211	0,00	1,94	0,349	0,00	2,68	0,482	0,01
19	0,45	0,40	0,18	0,40	0,072	0,00	1,29	0,232	0,00	2,13	0,383	0,02	3,11	0,560	0,11
20	0,45	0,40	0,18	0,39	0,070	0,00	1,15	0,207	0,01	1,79	0,322	0,05	2,68	0,482	0,01
21	0,45	0,40	0,18	0,37	0,067	0,01	1,05	0,189	0,04	1,90	0,342	0,01	2,45	0,441	0,11
22	0,45	0,40	0,18	0,35	0,063	0,01	1,09	0,196	0,02	1,75	0,315	0,06	2,48	0,446	0,09
23	0,45	0,40	0,18	0,36	0,065	0,01	1,11	0,200	0,02	1,76	0,317	0,06	2,45	0,441	0,11
24	0,45	0,40	0,18	0,43	0,077	0,00	1,31	0,236	0,01	1,95	0,351	0,00	2,88	0,518	0,01
				X_m	Q_m	$s(X_j)$	X_m	Q_m	$s(X_j)$	X_m	Q_m	$s(X_j)$	X_m	Q_m	$s(X_j)$
				0,46	0,082	0,08	1,24	0,223	0,14	2,00	0,360	0,20	2,78	0,500	0,28
				$s(X_m)$			$s(X_m)$			$s(X_m)$			$s(X_m)$		
				0,02			0,03			0,04			0,06		

Appendix C: Numerical Results of Pressure Standard Uncertainties

		i	1	2	3		
		Variable	P ₁	P ₂	I		
		Description	Pressure measurements	Pressure transmitter uncertainty	Data acquisition system uncertainty		
		Unit	[Pa]	[Pa]	[mA]		
		Evaluation Type	A	B	B		
		p _i	1	1	1		
η - Rotational Speed [rpm]	18,75	x _i	24,99	24,99	5,61	Σ	μ _{comb} (Y)/Y
		μ (x _i)	3,05E+00	3,00E-02	1,28E-02		
		μ (x _i)/x _i	1,22E-01	1,20E-03	2,28E-03		
		[p _i · μ (x _i)/x _i] ²	1,49E-02	1,44E-06	5,21E-06		
	37,50	x _i	89,25	89,25	7,66	Σ	μ _{comb} (Y)/Y
		μ (x _i)	8,73E+00	1,07E-01	1,38E-02		
		μ (x _i)/x _i	9,78E-02	1,20E-03	1,81E-03		
		[p _i · μ (x _i)/x _i] ²	9,57E-03	1,44E-06	3,26E-06		
	56,25	x _i	187,87	187,87	10,81	Σ	μ _{comb} (Y)/Y
		μ (x _i)	1,88E+01	2,25E-01	1,54E-02		
		μ (x _i)/x _i	9,98E-02	1,20E-03	1,43E-03		
		[p _i · μ (x _i)/x _i] ²	9,96E-03	1,44E-06	2,03E-06		
	75,00	x _i	296,31	296,31	14,23	Σ	μ _{comb} (Y)/Y
		μ (x _i)	2,81E+01	3,56E-01	1,71E-02		
		μ (x _i)/x _i	9,48E-02	1,20E-03	1,20E-03		
		[p _i · μ (x _i)/x _i] ²	8,98E-03	1,44E-06	1,45E-06		

Appendix D: Numerical Results of Velocity Standard Uncertainties

		i	1	2	3	4		
		Variable	v_1	v_2	L_1	L_2		
		Description	Velocity measurements	Anemometer uncertainty	Passage area length uncertainty	Passage area height uncertainty		
		Unit	$[m.s^{-1}]$	$[m.s^{-1}]$	$[m]$	$[m]$		
		Evaluation Type	A	B	B	B		
		p_i	1	1	1	1		
η - Rotational Speed [rpm]	18,75	x_i	0,46	0,46	0,45	0,40	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	2,00E-02	9,00E-01	2,00E-02	2,00E-02		
		$\mu(x_i)/x_i$	4,35E-02	1,96E+00	4,44E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	1,89E-03	3,83E+00	1,98E-03	2,50E-03		
	37,50	x_i	1,24	1,24	0,42	0,40	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	3,00E-02	9,00E-01	2,00E-02	2,00E-02		
		$\mu(x_i)/x_i$	2,42E-02	7,26E-01	4,76E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	5,85E-04	5,27E-01	2,27E-03	2,50E-03		
	56,25	x_i	2,00	2,00	0,42	0,40	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	4,00E-02	9,00E-01	2,00E-02	2,00E-02		
		$\mu(x_i)/x_i$	2,00E-02	4,50E-01	4,76E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	4,00E-04	2,03E-01	2,27E-03	2,50E-03		
	75,00	x_i	2,78	2,78	0,42	0,40	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	6,00E-02	9,00E-01	2,00E-02	2,00E-02		
		$\mu(x_i)/x_i$	2,16E-02	3,24E-01	4,76E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	4,66E-04	1,05E-01	2,27E-03	2,50E-03		

Appendix E: Numerical Results of Pressure Difference Standard Uncertainties

		i	1	2	3	4	5		
		Variable	P	Q	η	T	U		
		Description	Pressure	Air flow	Rotational speed	Air temperature	Air relative humidity		
		Unit	[Pa]	[m ³ .s ⁻¹]	[rpm]	[°C]	[%]		
		Evaluation Type	A/B	A/B	B	B	B		
		p _i	1	2	1	-1	1		
η - Rotational Speed [rpm]	18,75	x _i	24,99	0,50	75,00	30,00	74,00	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	3,05E+00	9,79E-01	3,75E+00	2,00E+00	3,70E+00		
		$\mu(x_i)/x_i$	1,22E-01	1,96E+00	5,00E-02	6,67E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	1,49E-02	1,53E+01	2,50E-03	4,44E-03	2,50E-03		
	37,50	x _i	89,25	0,50	75,00	29,90	77,00	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	8,73E+00	3,65E-01	3,75E+00	2,00E+00	3,85E+00		
		$\mu(x_i)/x_i$	9,78E-02	7,29E-01	5,00E-02	6,69E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	9,57E-03	2,13E+00	2,50E-03	4,47E-03	2,50E-03		
	56,25	x _i	187,87	0,50	75,00	29,90	74,00	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	1,88E+01	2,28E-01	3,75E+00	2,00E+00	3,70E+00		
		$\mu(x_i)/x_i$	9,98E-02	4,56E-01	5,00E-02	6,69E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	9,96E-03	8,31E-01	2,50E-03	4,47E-03	2,50E-03		
	75,00	x _i	296,31	0,50	75,00	30,60	70,00	Σ	$\mu_{comb}(Y)/Y$
		$\mu(x_i)$	2,81E+01	1,66E-01	3,75E+00	2,00E+00	3,50E+00		
		$\mu(x_i)/x_i$	9,48E-02	3,32E-01	5,00E-02	6,54E-02	5,00E-02		
		$[p_i \cdot \mu(x_i)/x_i]^2$	8,98E-03	4,40E-01	2,50E-03	4,27E-03	2,50E-03		