THESIS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY IN THERMO AND FLUID DYNAMICS

Ventilation Flow Field Characteristics of a Hydro-Generator Model

An Experimental and Numerical Study

HAMED JAMSHIDI

Department of Applied Mechanics
Division of Fluid Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2017

Ventilation Flow Field Characteristics of a Hydro-Generator Model
An Experimental and Numerical Study
HAMED JAMSHIDI
ISBN 978-91-7597-539-9

Doktorsavhandlingar vid Chalmers tekniska högskola
Ny serie nr. 4220
ISSN 0346-718X

Department of Applied Mechanics
Division of Fluid Dynamics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Cover:
Simulated streamlines of hydro-generator model.

Typeset in LaTeX

Printed by Chalmers Reproservice
Göteborg, Sweden 2017

Ventilation Flow Field Characteristics of a Hydro-Generator Model
An Experimental and Numerical Study
Thesis for the degree of Doctor of Philosophy in Thermo and Fluid Dynamics
HAMED JAMSHIDI
Department of Applied Mechanics
Division of Fluid Dynamics
Chalmers University of Technology

Abstract

Hydro-generators are complex machines used to convert the mechanical energy of the water turbine into electrical energy. Electromagnetic and mechanical losses accompany this energy conversion process which will cause heat generation and temperature increase. Cooling systems are needed to remove this excess heat from hydro-generators. Cooling system should control temperature increase and its temporal and spatial uniformity. An efficient cooling and ventilation must be considered during the electro-mechanical design of a generator. Having a complete picture of the losses, the ventilation flow field characteristics and the temperatures inside a generator is essential for an optimal design of cooling system for it.

The present work provides experimental and numerical tools essential for investigating ventilation flow attributes inside hydro-generators and also a comprehensive studies of flow based on these tools. An extensive knowledge of flow distribution inside the stator ventilation channels in different operational conditions and geometrical configurations are achieved. The obtained knowledge can be used for improvement in design of generator cooling system.

A hydro-generator model was designed and manufactured taking into consideration the needs of both the experimental and numerical methodologies. An inlet section is designed to deliver a uniform flow distribution into the machine and also to facilitate a direct and accurate measurement of the inlet flow rate. A CFD-based procedure is utilized for its design. The intake flow can either be supplied by a specifically designed radial fan connected to the rotor and co-rotating with that, or by an external centrifugal fan. Stators with three different ventilation channel geometrical configurations are used. Total pressure rake, 5-hole probe and hot-wire anemometer are used for taking measurements at stator ventilation channels outlets and generator inlet. Particle image velocimetry is carried out to reveal the flow field inside the ventilation channels.

The computational fluid dynamics simulations are performed using the FOAM-extend CFD toolbox. A block-structured mesh is generated using the ANSYS ICEM CFD mesh generator. The steady-state multiple frames of reference method is used for the numerical simulations. The frozen rotor and mixing plane approaches are used to handle the rotor-stator interaction. The flow is assumed axisymmetric, so just a section of generator model is simulated numerically. Periodic boundary conditions are imposed at the two sides of the computational section. Turbulences in the flow are modeled with Reynolds-averaged Navier-Stokes (RANS) formulation. The flow and pressure field in the generator model are analyzed in detail. The numerical and experimental results show a good agreement, which indicates the applicability of both methods.

Another aspect of hydro-generator ventilation which is important for designers is the convective heat transfer coefficients. An alternative way to indirectly obtain the convective heat transfer coefficients is to conduct mass transfer experiments such as the naphthalene sublimation technique. In the present work this technique is evaluated for analysis of the local heat transfer distribution when a circular air jet impinges normal on a flat surface. The local sublimation rate from the naphthalene surface subjected to the air jet is measured and reduced to the heat transfer that would occur on the surface under analogous thermal conditions. The indirectly obtained local heat transfer distributions and its local Nusselt numbers are compared to the results of numerical simulations and other experiments. The results show that the naphthalene sublimation technique can be used to accurately estimate the local heat transfer coefficients.

Keywords: Hydro Power Generator, Ventilation, CFD, Experiment, Naphthalene Sublimation Technique

The knowledge of anything, since all things have causes, is not acquired or complete unless it is known by its causes.

-Avicenna (Ibn Sina)

Acknowledgements

The research presented was carried out as a part of ”Swedish Hydro-power Center - SVC”. SVC has been established by the Swedish Energy Agency, Energiforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University, www.svc.nu. The simulations were performed on resources provided by the Swedish National Infrastructure for Computing (SNIC) at C^3SE and NSC.

I would like to thank my supervisors, Håkan Nilsson and Valery Chernoray. Their supportive involvement and assistance in every step throughout of my PhD made it possible for me to accomplish this.

List of Publications

This thesis consists of an extended summary and the following appended papers:

Paper A H. Jamshidi, H. Nilsson, and V. Chernoray. "CFD-based Design and Analysis of the Ventilation of an Electric Generator Model, Validated with Experiments". *International Journal of Fluid Machinery and Systems*, **8(2)**, pp.113-123 (2015).

Paper B H. Jamshidi, H. Nilsson, and V. Chernoray. "Ventilation Air Flow Field Asymmetry and Unsteadiness in Stator Channels of a Generator Model". *Submitted for journal publication* (2016).

Paper C H. Jamshidi, H. Nilsson, and V. Chernoray. "The Effect of Inlet Flow Rate on the Air Distribution in Ventilation Channels of a Hydro Generator Model". *Manuscript in preparation for publication*(2017).

Paper D H. Jamshidi, M. Liljemark, H. Nilsson, and V. Chernoray. "Assessment of Naphthalene Sublimation Technique in Jet Impingement Heat Transfer Characterization". *Manuscript in preparation for publication*(2017).

Division of Work

Paper A-C
Hamed Jamshidi did the work on modification of test rig and assembling the set-up with contributions and feedback from Håkan Nilsson and Valery Chernoray. Hamed Jamshidi also did all the work on running the measurements, experimental data post processing and analysing, mesh generation, numerical simulation, analysis and writing of papers. Håkan Nilsson gave feedback in the numerical simulation set-up and writing the paper. Valery Chernoray mainly gave feedback and helped with experimental part of the study.

Paper D
Hamed Jamshidi did the work on design of test rig and assembling the set-up with contributions and feedback from Håkan Nilsson and Valery Chernoray. Marcus Liljemark did the surface measurements of the naphthalene specimen. Hamed Jamshidi also did all the work on mesh generation, numerical simulation, post processing experimental data, analysis and writing of papers. Håkan Nilsson and Valery Chernoray also gave feedback in writing the paper.

List of Additional relevant publications

Relevant publications which are not included in this thesis are listed as the following:

Publication I H. Jamshidi, H. Nilsson, and V. Chernoray. "Experimental and Numerical Investigation of Hydro Power Generator Ventilation". *IOP Conference Series: Earth and Environmental Science 22.1 , 012007, Presented at 27th IAHR Symposium, Montreal* (2014).

Publication II H. Jamshidi, H. Nilsson, and V. Chernoray. "Ventilation Air Flow Field Characteristics in a Hydro Generator Model". *Presented at 19th International Seminar on Hydropower Plants, Vienna* (2016).

Contents

1 Introduction

Hydropower produces around 20% of the world's electricity and over $^4/_5$ ths of the world's renewable electricity today [1]. It is the largest renewable energy source which is based on the natural water cycle. Hydropower plants use a relatively simple concept which make them highly reliable, flexible and cost effective[2]. A hydraulic turbine converts the energy of flowing water into mechanical energy. A hydro-generator converts this mechanical energy into electricity. The operation of a generator is based on the principle discovered by Faraday. His law states that *any change in the magnetic environment of a coil of conductor will induces a current in the coil.* No matter how the change is produced, the electric current will be generated. This change, typically, is produced by rotating an electromagnetic field relative to a stationary conductor coil. Figure 1.1 depicts the main part of a hydro-generator. The stator and the rotor are the main components of the generator. A cooling system is essential for generator and has a significant impact on its efficiency. The stator, the rotor and the cooling system will be described in the coming sections.

1.1 Stator

A stator consists of windings, an iron core and a frame. The stator windings, or coils, are composed of electrically insulated copper conductors. It is in the stator windings where the mechanical energy is converted to electrical energy, by an interaction with the rotating electromagnetic flux provided by the rotor. The stator winding insulation is one of the most critical sub-components of any generator. Insulation is quite sensitive to high temperature therefore can easily affect reliability of generator. Electrical insulation systems are rated by standard NEMA (National Electrical Manufacturers Association) classifications according to maximum allowable operating temperature. It is intended that the hottest-spot total temperature should not exceed 130°C for Class B and 155°C for Class F insulation systems. Class B and F are the most common insulation systems used in generators.

The iron core of stator is composed of a stack of thin lamination. Each lamination has a thin coating of insulating that electrically insulates it from the adjacent lamination. These are necessary to minimize eddy current and hysteresis losses generated by alternating magnetism. The laminations are stacked as full rings or segments, in accurate alignment, having ventilation spacers inserted periodically along the core length. The completed core is compressed and clamped axially. The main functions of hydro generator stator core are to conduct magnetic flux, provide space and support for stator winding and serve as a part of machine cooling circuit. The stator winding is contained and supported within stator core slots, and its end windings are supported by the mechanisms mounted on the core end flanges. The stator core also assists with the winding thermal expansion forces. As a part of the machine cooling circuit, the stator core provides heat transfer surface area, and ventilation channels.

The frame of the stator is divided into an inner and an outer section, both of which mount on a single base. The inner frame is a structure designed to support the stator core and windings. The outer frame is a simple housing, which supports the air inlets or recirculation ducts.

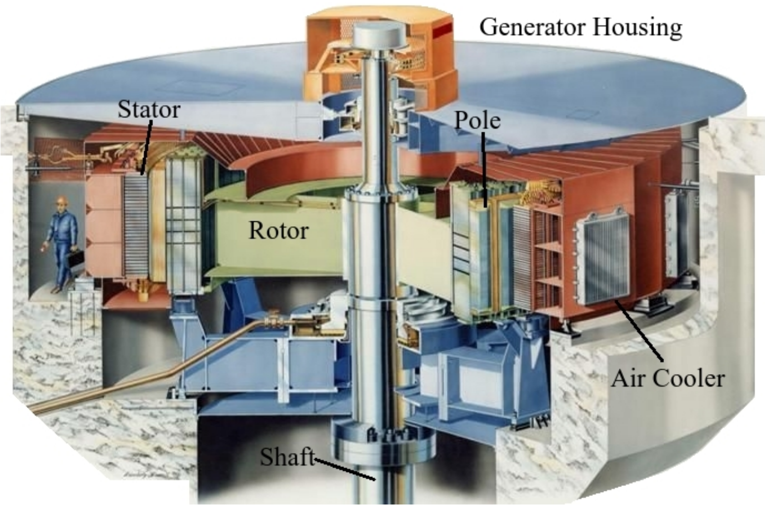


Figure 1.1: *Sketch of Hydro-generator [3].*

1.2 Rotor

The primary function of the generator rotor is to produce the rotating electromagnetic field necessary for excitation of the stator winding. The electromagnetic field is induced in the poles by an electric current passing through the exciting winding. Rotors of electric generators are classified as cylindrical pole rotors and salient pole rotors, see Figure 1.2. A cylindrical rotor, also known as non-salient, is a solid steel shaft with slots that run lengthwise along the outside of its cylindrical shape. Laminated conductor bars are inserted within the slots and kept stationary with metal wedges. These copper bars form the exciting winding of the rotor. The number of poles in these rotors are usually 2 or maximum 4. The ratio of diameter to axial length in these kind of rotor is small. Cylindrical rotors are used in high speed generator, used mainly on turbine generators. According to

$$RPM_{Rotor} = 120 \times f/P, \tag{1.1}$$

the speed of rotor, RPM_{Rotor}, is related to electrical system frequency, f, and the number of poles in the rotor, P.

In salient pole type the rotor consist of large number of poles protrude from the rotor support structure. The structure part is connected to shaft and transmit torque from shaft to the poles. The poles are made up from lamination of steel. There is exciting winding around these poles. Salient pole rotors have large diameter to axial length ratio and they are generally used in lower speed systems as the number of poles are more in this kind of rotor. Hydro-generators usually have a salient pole rotor.

1.3 Cooling System

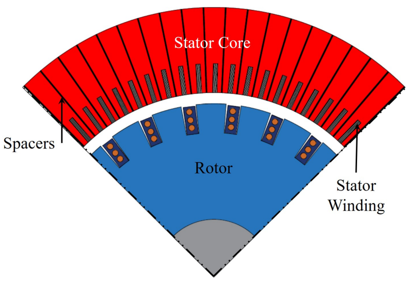
The generator cooling system removes the heat from the machine. An adequate heat removal process is vital to keep the generator at a stable, uniform, and limited temperature, while keeping the losses caused by ventilation flow (windage loss) to a minimum. Improvements in the cooling system increase the generator life expectancy and offer means to run the machines at higher efficiency and rating without any major investments and modifications in the structure.

Hydro-generators can be classified by the cooling method used for the rotor and stator. Indirectly-cooled generators are those in which the heat generated within the windings must flow through the ground insulation before reaching the cooling medium. Cooling air is circulated through passages in the generators by either of the following methods:

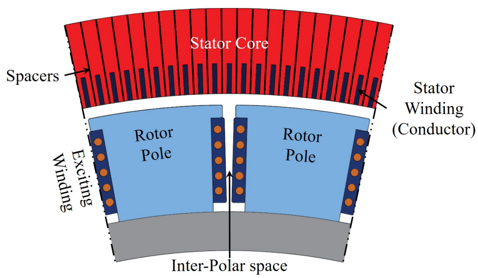
(a) Self-ventilated generator. A self-ventilated generator has cooling air circulated by means integral within the machine.

(b) Separately-ventilated generator. A separately-ventilated generator has cooling air circulated by an independent fan or blower system external to the machine rotor and stator.

Most indirectly-cooled generators over 10 MVA and built since 1930 are totally enclosed. The enclosure allows the internal cooling air to be circulated in a closed loop through the generator and thence through air- to-water heat exchangers located within the enclosure where the heat created by the generator losses is removed from the air before it returns to the machine. Directly-cooled generators are those in which the coolant for the windings flows in close contact with the conductors so that the heat generated within the windings is transferred directly to the coolant without passing through the ground insulation. Directly-cooled generators are totally enclosed. Internal air will be circulated to cool the



(a) *Cylindrical Rotor*



(b) *Salient Pole Rotor*

Figure 1.2: *Schematic Cross Section of an Electric Generator Rotor*

rotor winding (if it is not directly cooled) and perhaps the stator core in a manner similar to indirectly-cooled machines.

Indirectly-cooled generators may be cooled by air or hydrogen. In large machines, no matter what the cooling medium, the heat is transferred to water in heat exchangers that are located within the machine case. Air-Cooling is the standard cooling concept for small to large hydro-generator machines. Air-cooled generators are designed in two basic configurations, open ventilated or totally enclosed with water air cooler. The open ventilated, or self-cooled, configuration is used in small hydro-generators. They depend on ventilating to remove heat. The heated air is exhausted to the air surrounding the machine, the air intake is from the same surroundings. Ducting may be provided to reduce or eliminate re-circulation of the heated exhaust air. As the size of these machines increases, with associated increases in the power rating and losses, ventilation by fresh air alone is not practical. Air-to-water heat exchangers are necessary to adequately control machine temperature. A totally enclosed cooling air system configuration is installed in this case, which requires the input of fresh cool water and the output of heated waste water.

Hydrogen has been applied in the cooling of larger generators. It has a high specific heat, a high thermal conductivity and a low density, so it provides a better heat transfer with lower windage losses than air. Cooling with hydrogen requires additional systems to maintain hydrogen purity at a nonexplosive range, and to keep the hydrogen away from the lubricating oil and shaft seals. Water may be used for direct cooling of stator and/or rotor windings in very large machines with an extreme high output.

In air-cooled generators, rotor-mounted fan blades are usually the only source of power required to force the ventilating air through the machine. The rotor spider-rim assembly may also be used as fan. External electric motor-driven blowers could be used for forcing the cooling air flow. These blowers have the advantage of being able to adjust the air flow to suit the cooling demands. Conceivably, reduced windage losses at low loading would be possible. Disadvantages would include increased costs for the blower drive system, design concerns and dependency of the machine on the blowers for proper operation. Presently, these systems are rarely used in hydro-generators as their disadvantages outweigh the advantages.

Air-cooled generators can have principally radial or axial ventilation system. The usage of one or another ventilation system depends mainly on the rotor type, the size, and the power rating of these machines. In the axial alternative, the air is mainly driven by fans situated at the extremities of the rotor. The air flows through the inter-polar spaces and the air gap before entering the stator ventilation channels. In the case of radial cooling, the rotor is also used as a radial fan which contribute to the air pressure build up. The air flows radially through ducts in the rotor, into the stator channels.

1.4 Energy Losses

The process of conversion of the mechanical energy into electricity in hydro power generators includes three main classes of energy losses;

1. electric losses

2. magnetic losses

3. mechanical losses.

Electric loss is the power lost in the windings of a generator. This loss, which is also known as copper loss, is generated when there is current through a conductor. The electric loss increases as the current or the resistance of the conductor increases, according to

$$L_c = I^2 R, \tag{1.2}$$

where the resistance of the conductor is directly proportional to the temperature change as,

$$R = R_0[1 + \alpha\,(T - T_0)]. \tag{1.3}$$

Here α is temperature coefficient of resistance and T_0 is reference temperature. Copper loss is reduced by using large diameter wires.

Magnetic losses, also known as iron or core losses, include hysteresis losses and eddy current losses. Hysteresis loss is a heat loss caused by the magnetic properties of the stator core when it is exposed to a magnetic field which is changing direction. To compensate for hysteresis losses, heat-treated silicon steel is used in most stators core. The currents that are induced in the generator stator core by a rotating magnetic field are called eddy currents. The power dissipated in the form of heat as a result of these currents is considered as eddy current loss. This loss can be minimized by an insulated lamination of the stator core.

The mechanical losses are due to windage loss and friction. The windage loss is the power lost due to internal friction in the ventilation flow when passing through any part of cooing system. The windage loss increase the temperature and decreases the overall efficiency of the machine. The friction happens in bearings and brushes. It should be emphasised again that a well design ventilation and cooling system can reduce the windage losses.

1.5 Modeling of The Ventilation Process

Simulations methods for generator ventilation and cooling can basically be divided into analytical methods, such as lumped-parameter networks, and numerical methods, such as computational fluid dynamics (CFD). Analytical network methods typically include flow-network and thermal-network analysis [4]. In its most basic form, these network are analogous to an electrical network. The analytical approach has the advantage of being very fast to calculate, however the developers and users of the network model must invest an effort in defining a circuit that accurately models the main fluid flow and heat transfer paths [5].The calculation of the cooling airflow of the machine can be done with flow-networks. Flow-network analyses is the fluid mechanics counterpart to electrical-network analysis with the following equivalences: pressure to voltage, volume flow rate to current, and flow resistance to electrical resistance. The amount of cooling air, the overall velocity distribution, and the heat transfer coefficients in the different parts of the machine can be derived from the results. These parameters can be used in thermal network simulation, to determine the temperature distribution for a given loss distribution. Empirical dimensionless formulations are used to predict pressure drops in different flow paths in the generator by assuming them as flow restrictions, such as vents, bends, contractions, and expansions. The governing equation that relates the pressure drop, P (equivalent to an electrical voltage) to the volume flow rate, Q (equivalent to electrical current) and fluid-dynamic resistance, R is

$$P = Q^2 R. \tag{1.4}$$

In 1.4, the formulation is in terms of Q^2 rather than Q due to the turbulent nature of the flow. Two types of flow resistance exist. The first exists where there is a change in the flow condition, such as expansions, contractions and bends. The second is due to fluid friction at the duct wall surface; in electrical machines, this is usually negligible compared with the first resistance type due to the comparatively short flow paths. The flow resistance is calculated for all changes in the flow path using

$$R = \frac{k\rho}{2A^2}, \tag{1.5}$$

where ρ is the air density, A is the flow area, and k is the dimensionless coefficient of local fluid resistance whose value depends upon the local flow condition such as expansion, contraction, etc. Many empirical formulations are available to calculate the k factor for all changes in the flow section within the generator [6]. It is crucial to select the most appropriate formulation for all the flow paths involved. For example a sudden contraction when air enters the stator rotor air gap or a 90° bend where the air passes around the end winding.

In a thermal-network, it is possible to lump together components that have similar temperatures and to represent each as a single node in the network. Nodes are separated by thermal resistances that represent the heat transfer between components. Inside the generator, a set of conduction thermal resistances represents the main heat-transfer paths. In addition, convection and radiation resistances are used for heat transfer from the generator to the cooling air [7]. Conduction thermal resistances can be simply calculated using

$$R = \frac{L}{kA}, \tag{1.6}$$

where L is the path length, A is the path area, and k is the thermal conductivity of the material. In most cases, L and A can simply be gained from the components' geometry. The only complication is in assigning a correct value to L for thermal resistances due to the interface gap between components [8].Experimental factors are very important for a correct prediction of this thermal resistance.

Convection thermal resistances for a given surface can be simply calculated using

$$R = \frac{1}{h_C A}, \tag{1.7}$$

where A is the surface area and h_C is convection heat transfer coefficient. Empirical heat transfer correlations and results from flow-networks are used to predict convection heat transfer coefficient for all convection surfaces in the generator [9].

The network models make it possible to analyze heat transfer and flow field in generators rapidly, and are, therefore, the primary choice in the design process. However, the accuracy of the network models is strongly dependent on the underlying empirical parameters. In most cases, experimental data must be used to calibrate the thermal network models in order to get sufficiently accurate results. These models lack the ability to predict the detailed local heat transfer and flow field characteristics. It is hard to examine and understand the underlying physical phenomena of flow field inside a generator, such as turbulence or rotation and transients effects, only based of network methods.

Without a proper understanding of the fluid flow in and around the generator, a continuation in the trend of increasing power density, increasing energy efficiency, and cost reduction, will not be possible. The ability to provide such understanding, and knowledge is the main strength of numerical analysis. CFD offers a good potential to fully predict the flow, and the heat transfer in generators even in complex regions [10]. However, it is relatively demanding in terms of model setup and computational time. CFD primarily aims at determining the coolant flow rate, velocity, and pressure distribution inside the generator. CFD can also determines the surface heat transfer for analysis of the temperature in the active material and the solid structures. Accurate CFD results can even be used for improving the thermal network correlations without the need for costly experiments. This can provide the possibility to optimize the thermal design of machines at an early stage.

Boglietti et al. [4] presented an extended survey on the evolution of the modern approaches of thermal analysis of electrical machines.They compared the most common methods used for thermal analysis and discussed their strengths and weaknesses. They predicted an increasing trend towards scientific CFD studies of ventilation in electric generators. Changming et al. [11] identified and addressed some challenges in the application of computational fluid dynamics modeling in thermal management of electric machines.They found the whole motor CFD model to be a powerful engineering visualization tool and an accurate predictor of electric machine temperatures. The Fluent was used as CFD software in their work. They mentioned the need of a good quality mesh to get reasonable answer from CFD was one of the greatest challenges. Pickering et al. [12] summarized the results of experiments to validate the CFD modeling of large salient pole machines. They concluded in their report that the CFD demonstrates a good ability to predict the air flow and heat transfer of the rotor of a salient pole machine. They developed full-scale experimental facility representing a $1MVA$ synchronous 4-pole generator to measure air flow and local heat transfer coefficients over the surface of rotor poles. A steady state model with constant flow rate at inlet and constant heat fluxes at walls was used in their simulations. They found that in general the heat transfer coefficients predicted by CFD were lower than the measured values.

Shanel et al. [13] investigated the heat transfer and ventilation of an air-cooled generator by using a general purpose CFD code in a simplified generator design. They encountered a lots of convergence problem in the mesh areas with tetrahedral cells especially for temperature field. The authors recommend using of 70% of hexahedral cells. They shown that the coil internal temperature distribution can be approximated by considering the coil as made of a homogeneous material with equivalent thermal conductivity. Fujita et al [14] investigated the the windage loss reduction with the multiple-pitched ventilation ducts in the stator core of an air-cooled turbine generators. They concluded that one of the schemes to adjust the gas amount is the multi-pitched ventilation duct system in which the ventilation ducts are arranged in different pitches in the axial direction. This would help to have an equalized conductor temperature and a more efficient cooling of stator which results 32% of windage loss reduction compared with the conventional uniform duct distribution.However, non-uniform duct distribution leads to the circulating currents in the stator which increase coil losses. They examined an optimal design of ventilation ducts which limited the circulating current loss increase and made it possible to reduce windage and electrical losses.

Ujiie et al. [15] demonstrated that CFD is a valuable addition to the network method for the design optimization of electrical machines, if the accuracy was confirmed on a model test. They proved that CFD can be used as a standalone tool to compute flow, heat transfer, temperatures and even windage losses. They discussed that the insight of flow and heat transfer resulted from CFD could be used to optimize the geometry, to obtain flat temperature profiles, which avoids life-time decreasing temperature peaks. Further more results of CFD could be used to improve assumptions in the network model. Traxler-Samek et al. [16] presented a new method which was based on an iterative numerical coupling of power losses, airflow and temperature calculations. In fact they combined CFD and network models in an iterative manner. Their calculation proceeded in the following way:

1. Calculate the airflow distribution.
2. Calculate power losses at a reference temperature.
3. Calculate the distribution of the cooling-air temperatures from the calculated airflow distribution and the losses.
4. Solve the three-dimensional thermal networks of the stator.
5. Solve the rotor network.
6. Adjust the losses to the computed temperatures.
7. Iterate by continuing with step 3 and check for convergence.

Toussaint et al. [17] presented several simulation strategies to numerically compute the flow field in hydro-generators. They presented 2D and 3D simulation results for the 1:4 scale model obtained from CFD and addresses the challenges associated with the application of CFD to hydro-generators. In particular, they analyzed effect of rotor-stator interface (RSI) types and configuration to determine the approach that best suits this application. Pasha et al. [18] did experimental and CFD analysis on a 1:1 scale partial model of stator. They simulated different air flow velocities in the model duct from $5m/s$ to $30m/s$ in the wind tunnel to calculate the overall pressure drop and the overall hydraulic loss factor.They observed that major losses takes place in the wedge zone and at the leading edge of windings and any modification in this zone would improve the performance of the ventilation system. From CFD analysis they observed that eddies is formed and negative pressure distribution is found in the core zone and at the spacer tip of the exit zone. Schrittwieser et al [19] described the analysis of the fluid flow in the stator ducts of a hydro generator using CFD and defined permissible simplifications of the model in order to speed-up the simulation. They used two possibilities to connect rotor-stator reference models for steady state simulations, the Frozen Rotor and the Stage model. Based on their research they concluded that the wall heat transfer coefficients are constrained by the quality of the mesh near the wall. They suggested that it is also significant to check the boundary conditions of constant temperatures on the heat transport surfaces. These conditions have been simplified, because during operation of the generator the heat losses are not constant along the duct. Therefore the conjugate heat transfer becomes important and has to be considered.

Liang et al. [20] studied the influence of stator ventilation channel cutting length on the multi-physics in air-cooled hydro-generator. Their results shown that the appropriate cutting length of the ventilation channel can improve the fluid flow inside the ventilation groove and lower stator temperature. Zhang et al. [21] simulated flow field of dual radial ventilation system without fan for a hydro-generator. The multiple rotating frames (MRF) method was used to simulate the rotating motion of the generator and a porous media model was used to simulate the pressure loss of the air cooler. Their results shown that local pressure loss of stator entrance is dominant. There are leeward and windward areas for the air flowing with circumferential velocity component. This study suggested that a rational design of stator ducts entrance with some diversion effect, can reduce the pressure loss of the stator ventilation and improve the cooling of stator ducts. Han et al. [22] studies the flow field of the stator and the air gap for a large air-cooled generator. Their results demonstrated that There are different flow characteristics in different regions of the air gap because of the influence of stator and rotor radial air injection and axial through flow.

Li et al. [23] studied the influence of rotation on the flow and the temperature distribution of the rotor of an air-cooled $250MW$ hydro-generator. They discussed that CFD can accurately simulate the fluid field of the rotor ventilation system, which will provide the theoretical basis for the design and improvement of the generator ventilation system. They concluded that in the premise of the measured circulation flow rate, the maximum temperature was found at the excitation winding leeward side and the temperature is within the maximum temperature rise permissible for class F insulation. The highest temperature decreases with the increase of entrance velocity. But the position of the maximum temperature is not changed. Schrittwieser et al. [24] presented two different methods for simulating the heat transfer along the stator ducts of a hydro generator. Their investigations were focused on the fluid flow in an air gap between the insulation of the winding bars and the stator iron. They shown that it is also possible to define a thermal resistance in the interface simulating only the heat conduction without modeling the air gap resulting in faster simulations.

Torriano et al. [25] focused on the effect of rotation on heat transfer mechanisms in rotating machines, with the purpose of improving the understanding of thermal phenomena and cooling of hydro-generators. They measured the temperature distribution on the pole surface and deduced the heat transfer coefficients through numerical simulations. Their results shown an asymmetric heat transfer coefficient profile in the tangential direction. Their study also shown that the heat transfer coefficients along the pole face at $300rpm$

are in average about four times those at $50rpm$. They also observed a strong temperature gradient in the axial direction due to the presence of fans that improve cooling at the top and bottom ends of the pole. Yanagihara et al. [26] presented the validation of a numerical calculation method applied to the ventilation system of hydro-generators, by comparing the numerical results to experimental data obtained from specific model tests using hydrogenerator's geometrical and operational characteristics. Their study was limited to the most important region of the ventilation system, the cooling air ducts of stator core to get numerical results of pressure drop coefficient, which are impacted mostly by the entrance of air ducts.

Moradnia et al. [27, 28, 29] performed steady-state frozen rotor simulations of a simplified electric generator and validated the results with experimental measurements. They designed and manufactured a half-scale model of an electric generator. The model was slightly simplified compared to the original geometry. They performed CFD simulations using two approaches. In one approach the inlet flow rate is specified from the experimental data, as is commonly done in the literature. In the other approach the flow rate is determined from the numerical simulation, independently of the experimental results. Their Mesh sensitivity studies highlighted the need of a specific resolution of the baffle edges. Klomberg et al. [30, 31, 32] investigated different methods of analyzing a large hydro generator with computational fluid dynamics, using two transient and two steady-state approaches. Their studies focused on the end winding area of a large hydro generator and the influence of different ventilation schemes on the heat transfer and fluid flow at this area. They presented the reduced model slot sector model, which is smaller and faster to analyze than a standard pole sector model.

1.6 Aims and Scope

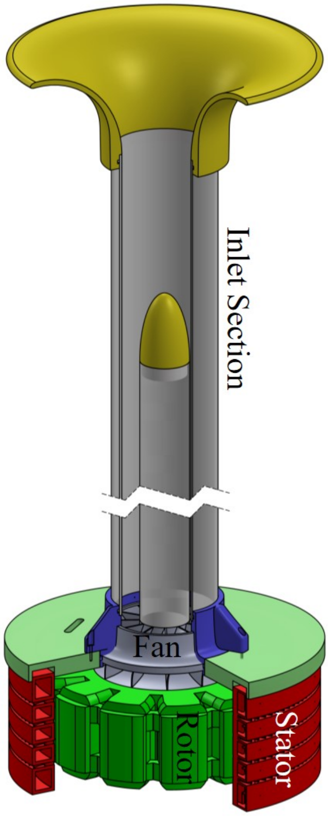
The preliminary aim of this work is to develop experimental and numerical tools for detailed studies of ventilation flow characteristics in generators. A CFD-based hydro-generator model test rig is designed and manufactured. Also, three different stators with various ventilation channels configuration are designed and manufactured. These provide the necessary comprehensive data to study the effects of spacers shape and ventilation channels geometry on ventilation flow characteristics of hydro-generator. An external fan can be connected to the test rig to control the inlet volume flow rate precisely. Overall the present set-up reasonably characterizes and resembles a scaled hydro-generator, as well as providing an accurate and relatively simplified geometry for numerical simulation. This apparatus enables fast and reliable measurements of different flow field attributes. A steady state numerical model is set up based on MRF method and validated by the experimental data. This numerical model can provide detail information and data of flow and pressure field in all parts of hydro-generator model.

The main objective of this work is to expand the knowledge of the underlying physical phenomena of ventilation in hydro-generators or any electrical machine with similar geometries. This objective is fulfilled by examining the detailed effects of various parameters on the flow field and discussing the cause of these effects based on experimental and verified numerical results. Among the studied parameters the effect of ventilation channels geometry, the effect of axial to tangential velocity ratio in rotor-stator air gap and the effect of inlet flow rate can be mentioned. The effects of these aforementioned parameters on the flow distribution and pressure field are comprehensively discussed. Finally the knowledge of ventilation flow characteristics can provide guidelines to improve the cooling in generators. A secondary aim is to provide and validate an experimental method to study the heat transfer inside the present hydro-generator model. The naphthalene sublimation method is assessed for this purpose.

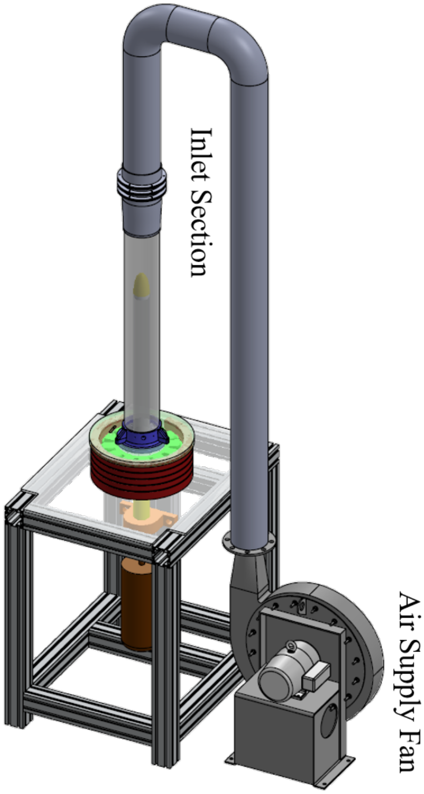
The present work is dedicated solely to the study of the ventilation air flow. Therefore both the experimental and numerical studies are in cold condition. Accordingly it is assumed that there is no heat generation or heat transfer in the system. The experimental model represent an axially ventilated hydro-generator with a salient pole rotor.

2 Procedures and Methodologies

This study includes an experimental and a numerical investigation. The experimental apparatus is designed according the purpose and the scope of this research. The numerical simulations are done by FOAM-extend CFD tool. In the following sections the experimental and the numerical investigation procedures and methodologies will be described.



(a) *Generator Model with Co-rotating Fan*



(b) *Complete Test-rig with External Fan*

Figure 2.1: *Computer-aided Design of Generator Model*

2.1 Experimental Set-up

The generator model is built taking into consideration both the CFD requirements and the ability to acquire the desired measurements data. It has a rotor with 12 poles, a stator with 4 rows of ventilation channels along the axis of rotation, and each row has

108 channels and windings, see Figure 2.1. The stator height is 0.175m. The height of the stator channels is 4.7mm. The rotor tip radius and the stator inner and outer radii are 0.178, 0.1825 and 0.219m, respectively. The rotational speed of the rotor is 2000rpm. The air flow can either be driven exclusively by the rotor rotation and its co-rotating fan ,see Figure 2.1a, or by an external air supply fan which provides the system with required flow rate, see Figure 2.1b.

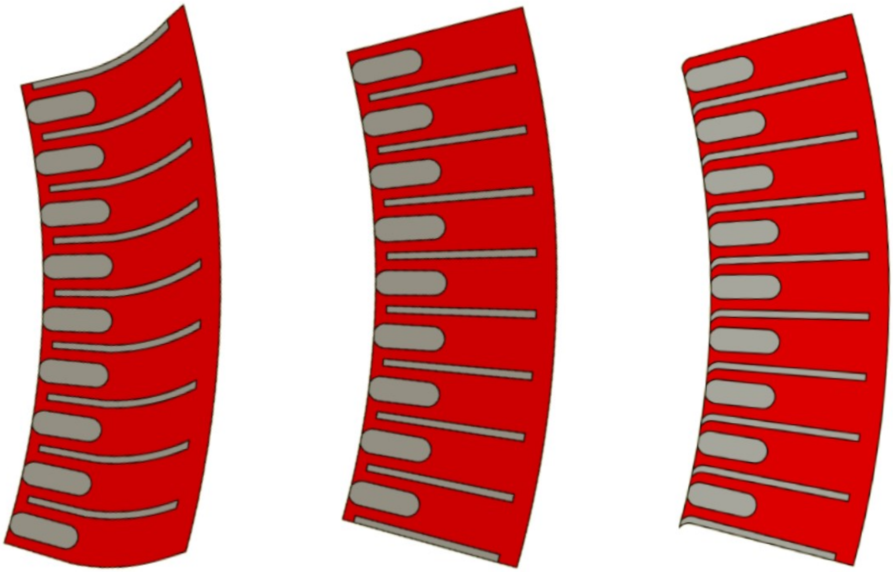
The hydro generator model test rig includes a fan connected on top of the rotor or an external fan to supply air flow, an inlet section, a rotor, a stator and an electric motor to rotate the rotor. The inlet section in this case is designed to deliver a uniform flow distribution into the machine and also to facilitate a direct and accurate measurement of the inlet flow rate. The inlet section in case of using external fan, respectively from fan to generator, consists of a pipe with two bends, honeycombs and series of mesh screens, a nozzle, two concentric pipes and a nose cone. Honeycombs and mesh screens control the free stream turbulence entering the concentric pipes. The nose cone is connected to the internal pipe and provides a contraction in the flow direction. This contraction acts as pressure difference flow meter, which is designed and calibrated for the flow rate measurements in this study. The geometry of the rotor and stator are slightly simplified with the purpose of improving the accuracy of the experimental results, and for preparing for the high-quality CFD simulations. The rotor and stator are manufactured using a rapid prototyping method. The rotor is manufactured using a Stereo Laser Sintering (SLS) process, since it is less fragile and will not break in small pieces. To suppress the intensive radiation scattered from the laser sheet by dust particles on the surfaces, the rotor is coated by reflective and absorbing paint. The stator is manufactured using a Stereo Lithography Apparatus (SLA) process as it has better surface finish, lower tolerance and higher accuracy. Stator is adapted for Particle Image Velocimetry (PIV).A small section of the stator is manufactured from acrylic parts in order to have a transparent column for PIV measurements.

Three stator layouts with different spacers are studied in this work, see Figure 2.2. The first stator has curved spacers, the second stator has straight spacers, and the third stator has straight spacers that extende to inner side of the stator, with a so-called *pick-up* edge curvature.

Figure 2.3 shows the manufactured stator of hydro-generator model. It is made of one piece to increase the axisymmetrical accuracy of the geometry. In addition, a PIV optical access section is considered for this stator. It is made of separately milled acrylic pieces, see Figure 2.3b.

2.2 Measurement Procedures

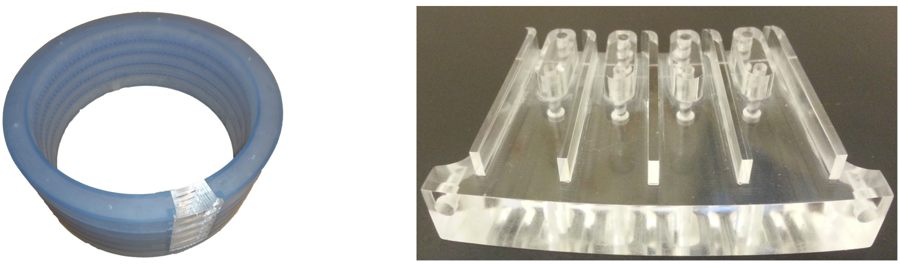
The total pressure at the outlet of the stator channels is measured by a custom made pressure rake. It consists of 14 pipes with a tip diameter of 0.5mm. These pipes are fixed side by side, forming a rake. By measuring the total pressure, the velocity and flow distribution of the channel outlet can be estimated. During the measurement, the rake is located at the middle of the channel height, and at the end wall of the baffles. The rake angle follows the angle of the outer part of spacers, see Figure 2.4. The rake



(a) *Curved Spacers* (b) *Straight Spacers* (c) *Spacers with Pick-up*

Figure 2.2: *Three Stator Configurations*

measurements have been done using a 16 channel *PSI 9116* digital pressure scanner from *Pressure Systems Inc.* The sample rate is 500 samples per second with a duration of 2 seconds, corresponding to 800 rotor pole passes. The samples are then averaged.

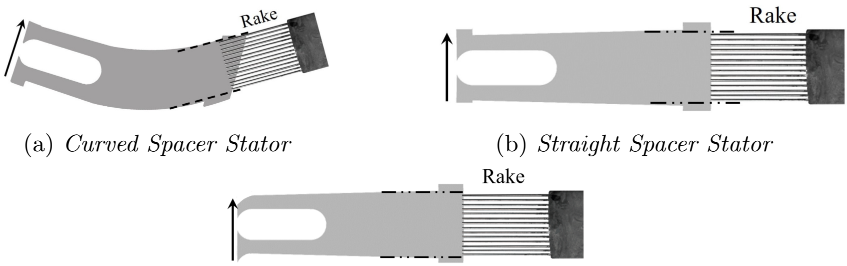


(a) *Stator* (b) *Piece of Stator Made of Acrylic*

Figure 2.3: *Stator with Optical Access Section*

5-hole pressure probe measurements are done upstream the fan, radially between two pipe of intake section. The 5-hole probe used in this work is manufactured and calibrated by the *Aeroprobe Corporation.* The calibration is performed for 2563 angular positions at a velocity of 20m/s, and for pitch and yaw angles varied within $\pm$ 55°. The measurement accuracy is better than 1% for the velocity magnitude and 0.5° for the flow angles. The diameter of the probe tip is 1.6mm, with individual distances between the holes of 0.5 mm, and a tip half cone angle of 30°.

Planar two-component PIV measurements are done in a symmetry plane of two stator channels in the 1st row, in a section of the stator which is specially constructed for optical access. The optical access for the camera is provided via the top cover of the generator, see Figure 2.5. The camera is of the type *Imager ProX 4M*. A double-pulsed laser of the type *EverGreen 200* is used. The camera and laser are synchronized via a programmable timing unit (PTU) which can create flexible trigger sequences. The image capturing is synchronized to the rotor position. The PIV images are acquired and processed in the *DaVis* software. In the PIV processing, a multi pass mode was used with window size of 64×64 pixel^2 at the first pass and 32×32 pixel^2 at the last pass with 50% overlapping. Averaging is done over 100 velocity fields to get the mean flow distribution in the symmetry plane of stator channels. The error in the PIV velocity field is estimated to 0.17 m/s due to the 0.1 pixel error in the sub-pixel interpolation.



(a) *Curved Spacer Stator* (b) *Straight Spacer Stator*

(c) *Pick-up Spacer Stator*

Figure 2.4: *Alignment of Total Pressure Rake During Measurements*

Hot-wire measurements are done upstream the fan and at the outlet of the ventilation channels. The hot-wire measurements in this study are based on constant temperature anemometry (CTA). In CTA the temperature (resistance) of the sensor is kept constant by an advanced feedback control loop that contains an electronic bridge circuit. This way, the anemometer produces a continuous voltage that is proportional to the instantaneous flow velocity. The output signal is sampled with a high resolution so that the flow velocity is determined accurately both in the amplitude domain and in the frequency domain. The hot-wire probe is driven by *DISA type 56C17* CTA bridge and a *DISA type 56C01* CTA unit mounted in a *DISA type 56B10* main frame. A *DISA type 56N20* signal conditioner provides a signal filtering and a gain selection. The measurements are done at data acquisition frequency of 20kHz during 2000 rotor revolutions. The hot-wire probe is a platinum-plated tungsten wire with a diameter of $5\mu m$. The system in calibrated in a free low-turbulence jet with an accurately controlled velocity range of 0.5-50m/s. The accuracy of the calibration polynomial was better than 0.1 %.

2.3 CFD Methodology

The numerical simulations of the generator model are performed using the FOAM-extend CFD tool, and the multiple reference frame (MRF) concept. MRF is a steady state

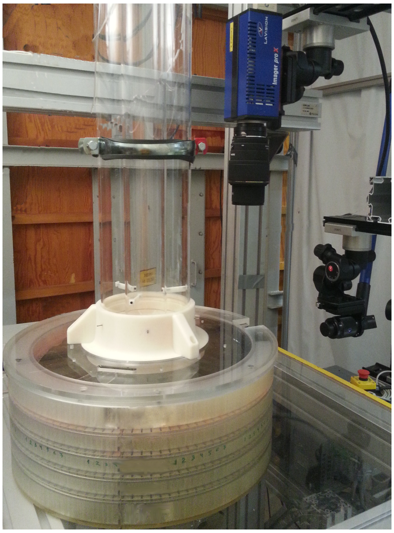


Figure 2.5: *2D PIV Set-up*

approximation where the fluid zone in the rotor region is modeled as a rotating frame of reference and the stator zones are modeled in a stationary frame. The MRF method does not require a sliding mesh, but instead applies source terms in the momentum equations for the rotation. The MRF allows individual cell zones to rotate or translate with different speeds. This is achieved by dividing the whole problem domain into separate zones where the flow is solved in stationary or rotating coordinate systems. While the MRF approach is clearly an approximation, it can provide a reasonable model of the flow for many applications. The MRF model can be used for simulation of flow in hydro-generator, in which rotor-stator interaction is relatively weak, and the flow is relatively uncomplicated at the interface between the moving and stationary zones. Figure 2.6 shows the stator and rotor zone and the interface between them. The governing equations of steady incompressible fluid flow for the rotating frame when convected velocity is the absolute velocity can be written as

$$\nabla \cdot \vec{V} = 0 \qquad \text{Conservation of mass} \tag{2.1a}$$

$$\nabla \cdot (\vec{V_r} \otimes \vec{V}) + \vec{\Omega} \times \vec{V_r} = -\nabla(\frac{p}{\rho}) + \nu \nabla \cdot \nabla(\vec{V}) + S \qquad \text{Conservation of momentum} \tag{2.1b}$$

where $\vec{V}$ is absolute velocity and $\vec{\Omega}$ is whirl velocity. Relative velocity is defined by

$$\vec{V_r} = \vec{V} - \vec{\Omega} \times \vec{r} \tag{2.2}$$

The MRF method requires that stationary and rotating zones are defined separated from each other. Two different approaches are used to handle the rotor-stator interaction (RSI), frozen rotor (FR) and mixing plane (MP). The frozen rotor approach retains the relative position of the rotor and stator and thus transfers the flow parameters in fixed positions. The mixing plane method averages the flow parameters in the circumferential

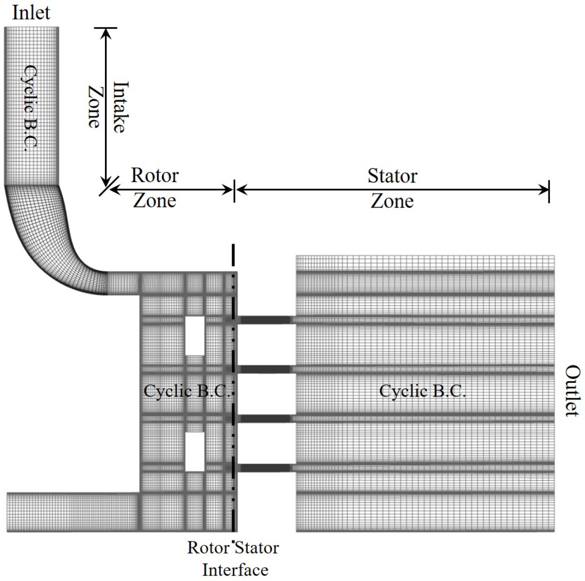
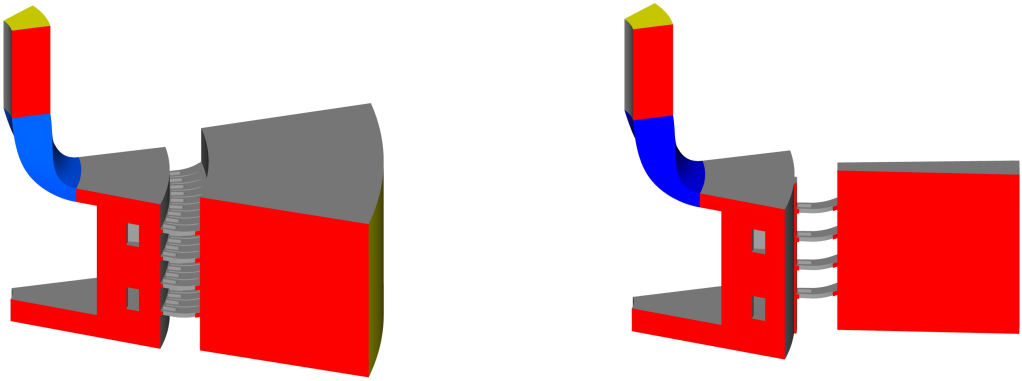


Figure 2.6: *MRF Mesh*

direction and transfers the averaged values to the adjacent interface. The relative position between the rotating and stationary parts is thus not taken into account in mixing plane method and thus avoids flow field non-uniformities. In another word the major difference between FR and MP is that the FR directly translates the properties of the flow at the interfaces between the rotating and stationary zones, whereas the MP averages the properties of the flow circumferentially.

A low Reynolds number (low-Re) turbulence model is required for modeling the effect of turbulence, due to the relatively low velocities and small dimensions in the stator channels. The Launder-Sharma $k-\varepsilon$ [33] low-Re model and four equation V^2f [34] turbulence model are used in the present study. In low-Re $k-\varepsilon$ model, the algebraic damping functions and extra source terms introduced to correct the behavior of turbulent quantities close to the wall. These models take into account the importance of viscous effects In the proximity of solid walls. The V^2f turbulence model is an alternative to the standard $k-\varepsilon$ model and was introduced to model the near-wall turbulence without the use of exponential damping or wall functions, as it is valid up-to solid walls. The model requires the solution of four differential equations. The two basic equations for k and ε and two additional equation; the transport equation for $\overline{v^2}$ and the elliptic equation for the relaxation function f.

A block-structured mesh is generated using the *ANSYS ICEM CFD* mesh generator software, keeping the wall y$^+$ values at about 1. The number of cells in case with FR and MP interfaces are 30×10^6 and 22×10^6, respectively.The mesh is block structured, covering a $1/12^{th}$ sector in the tangential direction, employing cyclic boundary conditions on the two sides. The computational domain with FR interface includes one rotor pole, one fan blade passage, and nine cooling channels in each channel row, see Figure 2.7a. The computational domain with Mp interface includes one rotor pole, one fan blade passage, and one cooling channels in each channel row, see Figure 2.7b. The second order central difference scheme is used to discretize the diffusion terms, and the second order linear upwind difference scheme is used to approximate the convection term. A mass conservative interpolation scheme is used for velocity field at mixing plane interface. For pressure term at MP interface an interpolation scheme which basically behaves like a zero-gradient scheme, while keeping the average pressure level the same on both sides of the interface is used. Area-averaging interpolation scheme is used for other fields at mixing plane interface. The CFD code in parallelized using message passing interface (MPI) library. The simulations are performed using 320 Intel Xeon $E5-2660$ processors. The results are considered converged when the residuals are small and stabilized and the rotor axial torque is stabilized.



(a) *Frozen Rotor* (b) *Mixing plane*

Figure 2.7: *Computational Domain*

2.4 Naphthalene Sublimation Technique

Convective heat transfer coefficients are often determined by complex experiments involving advanced instruments and difficult procedures. These procedures are even more difficult when the local heat transfer distribution over an entire test section in a practical engineering model with a complex geometry in needed. An alternative way to indirectly obtain the convective heat transfer coefficients is to conduct mass transfer experiments, which have better accuracy and are usually easier to perform. One of the most convenient mass transfer methods is the well-developed naphthalene sublimation technique that can be used to study mass and heat transfer for a variety of applications [35].

There are many advantages of using the naphthalene sublimation technique rather than doing direct heat transfer experiments. The naphthalene sublimation technique is particularly useful in complex flows and complex geometries with restrictions in the visual access [36]. Large gradients in transfer rate are difficult to capture in heat transfer measurements, but the naphthalene sublimation technique can be used to determine them even at corners and edges. Mass transfer boundary conditions analogous to isothermal and adiabatic walls in convective heat transfer can be easily imposed in this method. The surface that is coated or made of naphthalene acts as an isothermal surface while the other surfaces are adiabatic. Even if the mass/heat transfer coefficient is obtained with an isothermal condition it can be applied to other boundary conditions, because the mass/heat transfer coefficient is a weak function of the wall temperature distribution in turbulent flows. Furthermore, the nature of mass transfer allows one to impose these boundary conditions such that errors analogous to conductive and radiative losses in a wall are not present. The method thus readily distinguishes pure convection from conduction and radiation in conjugate heat transfer situations.

The heat transfer coefficient, which is often desired, can be readily determined from the measured mass transfer results with good confidence via a mass/heat transfer analogy. Although the analogy requires that the Schmidt (Sc) and Prandtl (Pr) numbers are similar, the naphthalene sublimation method is still applicable to many heat transfer problems under certain conditions. The Schmidt number for naphthalene vapor in air is 2.28 at 25°C, which is sufficiently close to the Prandtl number of numerous gases and liquids to apply the analogy with good confidence. A mass transfer test section of naphthalene can easily be fabricated and handled. Also, it does not require any complex heating and integrated measuring system, such as insulating material and thermocouple junctions. The naphthalene sublimation technique needs a naphthalene-coated specimen and an accurate geometry or weight measurement system. The test specimen can be easily prepared by several methods, including dipping, machining, spraying, and casting. It is also possible to make a complex geometry including moving parts with the exact representations of the desired boundary conditions. The local transfer coefficients can be determined with high accuracy and in detail by automated measurement systems that eliminate most human errors. However, the naphthalene sublimation method cannot generally be used in certain flow situations, such as high-velocity flows due to aerodynamic and viscous heating (recovery effect) and natural convection due to the thermal buoyancy effects of the sublimation latent heat [35, 36].

The naphthalene sublimation mass/heat transfer technique can be summarized as following steps:

1. Provide a specimen with naphthalene.

2. Measure the initial naphthalene surface shape.

3. Conduct the experiment with the naphthalene specimen.

4. Measure the naphthalene surface shape again and obtain the time-averaged local mass transfer rates from the sublimation depths (change in naphthalene surface height) and the time the specimen was subjected to the flow.

5. Reduce the data to obtain the mass/heat transfer coefficient.

3 Summary of Papers

This chapter gives a short summary of the main contents and results reported in the papers.

3.1 Paper A

CFD-based Design and Analysis of the Ventilation of an Electric Generator Model, Validated with Experiments

Summary and Outcomes: In this paper the air flow inside a generator model is analyzed, with an emphasis on the flow distribution inside the stator channels. A major part of the work is focused on the design of a new intake section and a new fan impeller. A CFD-based approach is used for the design of these new parts. The new fan provides a sufficient flow rate, that is driven solely by the rotating parts of the generator. Experimental results of the total volume flow rate, the outlet total pressure, and velocity distributions are presented. Steady-state CFD simulations are performed using the FOAM-extend CFD toolbox. The multiple rotating reference frames method is used, together with the frozen rotor and mixing plane rotor-stator coupling approaches. The experimental and numerical results agree to a large extent, making all the results reliable. The flow distribution is not even through different stator ventilation channel rows and in each channel. There is more flow at the upstream side of windings.

3.2 Paper B

Ventilation Air Flow Field Asymmetry and Unsteadiness in Stator Channels of a Generator Model

Summary and Outcomes: The flow field attributes of an axially ventilated generator model with a salient pole rotor are investigated both experimentally and numerically. The experimentally quantified total pressure and velocity distributions are presented. PIV is applied to reveal the details of the flow inside the stator channels. Hot wire measurements are performed to show the effects of the salient pole rotor on the unsteadiness of the flow field. Steady-state CFD simulations are performed, based on the multiple rotating reference frames method. The rotor-stator coupling is handled by both the frozen rotor and mixing plane approaches. The numerical results are validated with the time-averaged experimental data. Both rotor-stator approaches capture the time-averaged properties well, within their limitations. These numerical approaches may thus be considered useful for providing an overall picture of the ventilation flow attributes in electric generators. The flow distribution is not uniform through different channel rows, and in each channel. The axial and tangential velocity components in the rotor-stator gap affect both the uniformity of the flow distribution and the mean static pressure at the channel entrances.

The frozen rotor approach qualitatively captures the effects of the asymmetry of the rotor on the velocity field in different channels, but to an exaggerated level.

3.3 Paper C

The Effect of Inlet Flow Rate on the Air Distribution in Ventilation Channels of a Hydro Generator Model

Summary and Outcomes: In this article the ventilation flow distributions inside a hydro generator model that was reduced in complexity compared with a real generator are investigated both experimentally and numerically. The experiments and simulations conducted for three different operating conditions of air supply rates: Q_0, $1.1Q_0$ and $1.25Q_0$.The experimentally measured total pressure and velocity distributions are presented. PIV is applied to reveal the details of the flow inside the stator channels. The multiple rotating reference frames computational fluid dynamic method with mixing plane approach is used for numerical simulation. The numerical results are validated with the time-averaged experimental data. The numerical results are used to analyse the tangential and axial distribution of volumetric flow rate in the stator channels. It is found the flow distribution inside the stator channels will be more even in the axial direction as the air supply rate increased. But the excess flow pass more through the upstream side of conductors in each ventilation channel. The so called pick-up part of the spacers is good when the ratio of axial to tangential velocity at the inlet of ventilation channels in stator is high. The ventilation air flow distribution of a one sided axially ventilated hydro generator model with a salient pole rotor are investigated both experimentally and numerically in three different air supply rate. Mean total pressure and velocity distributions are measured at the outlet of the stator ventilation channels. The non-uniformity of the flow distribution in different channel rows from the experimental data. The velocity field inside the first row stator ventilation channel is obtained by PIV measurements. The experimental data is also used to validate numerical simulations of the flow field. The CFD simulations are performed using the steady-state MRF method with the mixing plane rotor-stator coupling approaches. The numerically predicted flow features agree very well with the experimental results. The numerical results thus provide reliable complementary information of the ventilation flow characteristics. The flow is not uniformly distributed between the channel rows, and inside each channel. BY increasing the inlet flow rate the air flow distribute more uniformly in axial direction of stator. The pickup has an inverse effect on the flow distribution in 1st row where the axial to tangential velocity ratio is high. In the 3rd and 4th row pickup is effective in distribution of quite uniform air flow.

3.4 Paper D

Assessment of Naphthalene Sublimation Technique in Jet Impingement Heat Transfer Characterization

Summary and Outcomes: The naphthalene sublimation technique is an experimental method for indirectly determining convective heat transfer coefficients. In this paper this technique is evaluated for analysis of the local heat transfer distribution when a circular air jet impinges normal on a flat surface. The air flow is turbulent, and the velocity profile of the jet is fully developed due to a large length to diameter ratio of the circular straight nozzle. Two Reynolds numbers based on the nozzle exit condition, 15,000 and 23,000, and two nozzle diameter distances from the jet exit to the surface, 6 and 8, are considered. The local sublimation rate from the naphthalene surface subjected to the air jet is measured under each condition. This mass transfer relates to the heat transfer that would occur on the surface under analogous thermal conditions. The indirectly obtained local heat transfer distributions and its local Nusselt numbers are compared to the results of numerical simulations and other experiments. The results show that the naphthalene sublimation technique can be used to accurately estimate the local heat transfer coefficients. The local heat transfer coefficient of a jet impinging normal to a flat surface is measured experimentally and determined numerically. The naphthalene sublimation technique is used for obtaining the mass transfer coefficients. Through the mass and heat transfer analogy this data is reduced to the heat transfer coefficient. Four different cases were considered, varying the nozzle distance to the impingement surface and the jet Reynolds number. The results show that the naphthalene sublimation technique is capable of analyzing the heat transfer characteristics of jet impingement.

4 Conclusion

To expand the knowledge of ventilation flow inside hydro-generators numerical and experimental tools are required. The current work provides and validate such tools. The ventilation air flow fields of a one sided axially ventilated hydro generator model with a salient pole rotor are investigated both experimentally and numerically through this work. A generator model is designed and modified based on the CFD and experimental requirements. The intake section is designed to serve as an accurate flow-meter, and nicely guide the flow into the machine. Optionally, an external fan can be connected to the inlet section of test rig to provide a controlled inlet flow rate. Three different inlet flow rate and three different stator ventilation channels shapes are tested. The results provide a valuable sets of data. Various flow attributes are measured using 5-hole probe, total pressure rake, and hot-wire. Also 2D PIV is done inside the channels. Mean total pressure and instantaneous velocity distributions are measured at the outlet of the stator ventilation channels. The non-uniformity of the flow distribution in different channel rows and the inherent transient nature of the flow are deduced from the experimental data. The experiments show that the flow at the outlet of stator ventilation channels is not evenly distributed. Hot-wire measurements show the transient nature of ventilation flow due to rotor stator interaction and somewhat asymmetric rotor. The velocity field inside the first row stator ventilation channel is obtained by PIV measurements. The experimental data are also used to validate numerical simulations of the flow field.

The simulations are performed using the steady-state multiple reference frame method, with the frozen rotor and the mixing plane rotor-stator coupling approaches. The numerically predicted flow features agree very well with the experimental results at all experimental sections. The numerical results could capture the mean flow distribution inside channels reasonably. The numerical results thus provide reliable complementary information of the ventilation flow attributes. The flow is not uniformly distributed between the channel rows, and inside each channel. This highlights the necessity to optimize the hydro-generators from a fluid dynamics point of view, to get an even flow distribution between different stator channel rows and inside each stator channels. It is shown that the axial and tangential velocity components in the rotor-stator gap affect both the uniformity of the flow distribution and the mean static pressure at the channel entrances. The frozen rotor approach qualitatively captures the effects of the asymmetry of the rotor and inter-polar spaces on the velocity field in different channels, but to an exaggerated level. The mixing plane provided the opportunity to simulate a finer mesh and decrease the run time. BY increasing the inlet flow rate the air flow distribute more uniformly in axial direction of stator. The pickup has an inverse effect on the flow distribution in 1st row where the axial to tangential velocity ratio is high. In the 3rd and 4th row pickup is effective in distribution of quite uniform air flow.

To assess naphthalene sublimation technique the local heat transfer coefficient of a jet impinging normal to a flat surface is measured experimentally and determined numerically. The naphthalene sublimation technique is used for obtaining the mass transfer coefficients. Through the mass and heat transfer analogy this data is reduced to the heat transfer coefficient. Four different cases were considered, varying the nozzle distance to the impingement surface and the jet Reynolds number. The results show that this technique is capable of analyzing the heat transfer characteristics of jet impingement, which suggest that this technique can also be used to study the heat transfer in the present hydro-generator model.

References

[1] International Energy Agency. “Energy Technology Perspectives; Scenarios and Strategies to 2050”. 1 (2008).

[2] International Renewable Energy Agency. “Renewable Energy Technologies: Cost Analysis Series, Hydropower”. 1 (2012).

[3] URL: http://www.elforsk.se/SVC/Forskningsprojekt/Elektromekanik/.

[4] A. Boglietti et al. “Evolution and Modern Approaches for Thermal Analysis of Electrical Machines”. *IEEE Transactions on Industrial Electronics* 56.3 (Mar. 2009), pp. 871–882. ISSN: 0278-0046. DOI: 10.1109/TIE.2008.2011622.

[5] G. Traxler-Samek, R. Zickermann, and A. Schwery. “Cooling Airflow, Losses, and Temperatures in Large Air-Cooled Synchronous Machines”. *Industrial Electronics, IEEE Transactions on* 57.1 (2010), pp. 172–180.

[6] Crane Valve Group. “Flow of Fluids through Valves, Fittings and Pipe”. *Crane Technical Paper TP-410M* (2009).

[7] C. Mejuto et al. “Thermal modelling investigation of heat paths due to iron losses in synchronous machines”. *2008 4th IET Conference on Power Electronics, Machines and Drives.* Apr. 2008, pp. 225–229. DOI: 10.1049/cp:20080516.

[8] A. Boglietti, A. Cavagnino, and D. Staton. “Determination of Critical Parameters in Electrical Machine Thermal Models”. *IEEE Transactions on Industry Applications* 44.4 (July 2008), pp. 1150–1159. ISSN: 0093-9994. DOI: 10.1109/TIA.2008.926233.

[9] D. A. Staton and A. Cavagnino. “Convection Heat Transfer and Flow Calculations Suitable for Analytical Modelling of Electric Machines”. *IECON 2006 - 32nd Annual Conference on IEEE Industrial Electronics.* Nov. 2006, pp. 4841–4846. DOI: 10.1109/IECON.2006.348143.

[10] M. Shanel, S. J. Pickering, and D. Lampard. “Application of computational fluid Dynamics to the cooling of salient pole electrical machines”. *ICEM 2000 : international conference on electrical machines.* Aug. 2000, pp. 338–342.

[11] C. Liao, C.-L. Chen, and T. Katcher. “Thermal management of AC induction motors using computational fluid dynamics modelling”. *Electric Machines and Drives, 1999. International Conference IEMD ’99.* May 1999, pp. 189–191.

[12] S. Pickering, D. Lampard, and M. Shanel. “Modelling ventilation and cooling of the rotors of salient pole machines”. *Electric Machines and Drives Conference, 2001. IEMDC 2001. IEEE International.* 2001, pp. 806–808.

[13] M. Shanel, S. Pickering, and D. Lampard. “Conjugate heat transfer analysis of a salient pole rotor in an air cooled synchronous generator”. *Electric Machines and Drives Conference, 2003. IEMDC’03. IEEE International.* Vol. 2. 2003, 737–741 vol.2.

[14] M. Fujita et al. “Air-cooled large turbine generator with multiple-pitched ventilation ducts”. *Electric Machines and Drives, 2005 IEEE International Conference on.* 2005, pp. 910–917.

[15] R. Ujiie, R. Arlitt, and H. Etoh. “Application of Computational Fluid Dynamics (CFD) on Ventilation-Cooling Optimization of Electrical Machines”. *Review Energy Technologies -(ICEMENERG).* 2006, pp. 17–22.

[16] G. Traxler-Samek, R. Zickermann, and A. Schwery. “Advanced calculation of temperature rises in large air-cooled hydro-generators”. *Electrical Machines, 2008. ICEM 2008. 18th International Conference on.* 2008, pp. 1–6.

[17] K. Toussaint et al. “CFD analysis of ventilation flow for a scale model hydro-generator”. *ASME 2011 Power Conference collocated with JSME ICOPE 2011.* American Society of Mechanical Engineers. 2011, pp. 627–637.

[18] A. A. Pasha, M. Hussain, and N. Gunubushanam. “Experimental and CFD analysis of hydrogenerator stator”. *Proc. of the 37th National and 4th Int. Conf. on Fluid Mechanics and Fluid Power, Madras, India.* 2010.

[19] M. Schrittwieser et al. “Numerical analysis of heat transfer and flow of stator duct models”. *International Conference on Electrical Machines (ICEM).* 2012, pp. 385–390.

[20] L. Yan-ping and B. Xu. “Influence of Stator Ventilation Channel Cutting Length on the Multi-Physics in Air-Cooled Hydro-Generator”. *International Conference on Electromagnetic Field Problems and Applications (ICEF).* 2012, pp. 1–4.

[21] Q. F. Zhang et al. “CFD Application on Ventilation System of Hydro-generator”. *Advanced Materials Research.* Vol. 383. Trans Tech Publ. 2012, pp. 3561–3565.

[22] J. D. Han et al. “Numerical Studies on the Flow Field of Stator and Air Gap for Large Air-Cooled Turbo-Generator”. *Advanced Materials Research* 516 (2012), pp. 970–975.

[23] L. Weili, G. Chunwei, and C. Yuhong. “Influence of Rotation on Rotor Fluid and Temperature Distribution in a Large Air-Cooled Hydrogenerator”. *Energy Conversion, IEEE Transactions on* 28.1 (2013), pp. 117–124.

[24] M. Schrittwieser et al. “Validation of Numerical Approaches for Simulating the Heat Transfer in Stator Ducts With Measurements”. *Magnetics, IEEE Transactions on* 50.2 (2014), pp. 261–264.

[25] F. Torriano et al. “Heat transfer coefficient distribution on the pole face of a hydrogenerator scale model”. *Applied Thermal Engineering* 70.1 (2014), pp. 153–162.

[26] J. I. Yanagihara and A. Claudinei de Moura. “Validation of a Numerical Method Applied to the Calculation of a Hydrogenerator Ventilation System”. *15th Brazilian Congress of Thermal Sciences and Engineering.* 2014.

[27] E. Hartono et al. “PIV Measurement of Air Flow in a Hydro Power Generator Model”. *16th Int Symp on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal.* 2012.

[28] P. Moradnia, V. Chernoray, and H. Nilsson. “Experimental assessment of a fully predictive CFD approach, for flow of cooling air in an electric generator”. *Applied Energy* 124 (2014), pp. 223–230.

[29] P. Moradnia et al. “Flow of cooling air in an electric generator model – An experimental and numerical study”. *Applied Energy* 114 (2014), pp. 644–653.

[30] S. Klomberg et al. “Comparison of CFD analyzing strategies for hydro generators”. *International Conference on Electrical Machines (ICEM).* 2014, pp. 1990–1995.

[31] S. Klomberg et al. “Implementation of a new simulation strategy for a cooling study of large hydro generators”. *International Conference on Power Electronics, Machines and Drives (PEMD).* 2014, pp. 1–6.

[32] S. Klomberg, E. Farnleitner, and G. Kastner. “Characteristics of the Convective Heat Transfer Coefficient at the End Winding of a Hydro Generator”. *Journal of Thermal Science and Engineering Applications* 7.1 (2015), p. 011011.

[33] B. Launder and B. Sharma. “Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc”. *Letters in heat and mass transfer* 1.2 (1974), pp. 131–137.

[34] P. A. Durbin. “Separated flow computations with the k-epsilon-v-squared model”. *AIAA journal* 33.4 (1995), pp. 659–664.

[35] R. Goldstein and H. Cho. “A review of mass transfer measurements using naphthalene sublimation”. *Experimental Thermal and Fluid Science* 10.4 (1995), pp. 416–434. ISSN: 0894-1777. DOI: `http://dx.doi.org/10.1016/0894-1777(94)00071-F`.

[36] P. S. Mendes. “The naphthalene sublimation technique”. *Experimental Thermal and Fluid Science* 4.5 (1991), pp. 510–523. ISSN: 0894-1777. DOI: `http://dx.doi.org/10.1016/0894-1777(91)90031-L`.