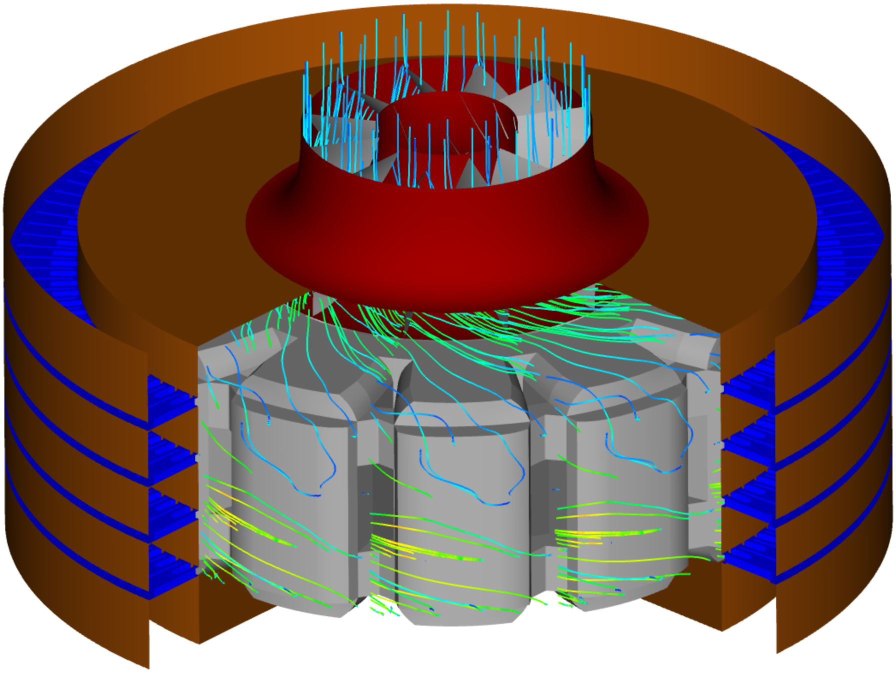
CHALMERS



Ventilation of a Model Hydro-Generator

An Experimental and Numerical Study

HAMED JAMSHIDI

Department of Applied Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2015

THESIS FOR THE DEGREE OF LICENTIATE OF ENGINEERING IN THERMO AND FLUID DYNAMICS

Ventilation of a Model Hydro-Generator

An Experimental and Numerical Study

HAMED JAMSHIDI

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

Göteborg, Sweden 2015

Ventilation of a Model Hydro-Generator
An Experimental and Numerical Study
HAMED JAMSHIDI

© HAMED JAMSHIDI, 2015

Thesis for the degree of Licentiate of Engineering 2015:11
ISSN 1652-8565
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 model generator.

Chalmers Reproservice
Göteborg, Sweden 2015

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

ABSTRACT

Generators are sophisticated machines used to convert the mechanical energy of a prime mover into electrical energy. Electromagnetic and mechanical losses accompany this energy conversion process. Cooling systems remove the heat, generated by losses, from these machines. An efficient cooling and ventilation must be considered during the electromechanical design of a generator. Having a complete picture of the losses, the ventilation flow and the temperatures inside the machine is a key prerequisite for an optimal design of cooling system.

The present work aims at providing experimental and numerical tools essential for a better understanding of ventilation flow attributes inside hydro-generators and coming up with comprehensive study of flow based on these tools. In particular, flow inside the stator ventilation channels is addressed. The obtained knowledge can be used for improvement in design of generator cooling system. A generator model was manufactured taking into consideration the needs of both the experimental and numerical methodologies. A new intake section and fan is designed for the existing generator model to increase the volume flow rate and make it possible to measure it accurately. A CFD-based procedure is utilized for their design. The experimental results point out the effectiveness of the new parts. A new stator with modified ventilation channel configuration is also designed with better optical experimental access. Total pressure rake, 5-hole probe and hot-wire anemometer are used for taking measurements. Particle image velocimetry (PIV) is carried out inside the ventilation channels.

The computational fluid dynamics simulations are performed using the FOAM-extend CFD toolbox. 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 effect of three different stator channel configurations on the ventilation flow properties is studied. The flow and pressure field in the model generator are analyzed. The numerical and experimental results show a good agreement, which indicates the applicability of both methods.

Keywords: Hydro Power Generator, Ventilation, CFD, Experiment

Absence of understanding does not warrant absence of existence.
-Avicenna (Ibn Sina)

Acknowledgements

First and foremost, I would like to thank my supervisors, Håkan Nilsson and Valery Chernoray. Their supportive involvement and assistance in every step throughout the last two and a half years of my PhD made it possible for me to accomplish this.Moreover, I would like to thank all colleagues and friends at Chalmers. Most importantly, none of this could have happened without having my lovely wife and son by my side.

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 Kraftnt together with Lule 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.

PREFACE

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

Paper A	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 (2014), 012007.
Paper B	H. Jamshidi, H. Nilsson, and V. Chernoray. "CFD-based Design and Analysis of the Ventilation of an Electric Generator Model, Validated with Experiments". *Manuscript in preparation for publication* (2015)
Paper C	H. Jamshidi, H. Nilsson, and V. Chernoray. "Analysis of Air Flow Field Attributes in an Axially Ventilated Generator Model with a Salient-pole Rotor". *Manuscript in preparation for publication* (2015)

CONTENTS

1 Introduction

Faraday's 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 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 affecting reliability of generator, which is quite sensitive to high temperature. The stator windings are recessed in and supported by the slots formed by the assembly of the laminated stator iron core. The core is composed of a stack of thin lamination. Each lamination has a thin coating of insulating that electrically insulates it from the adjacent lamination. 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.

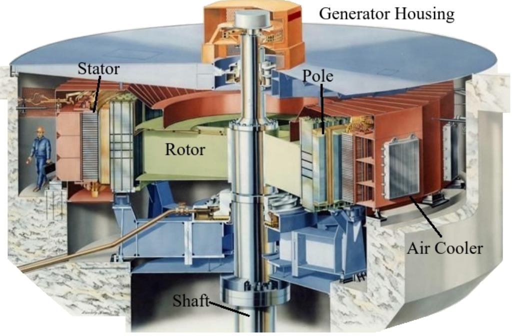
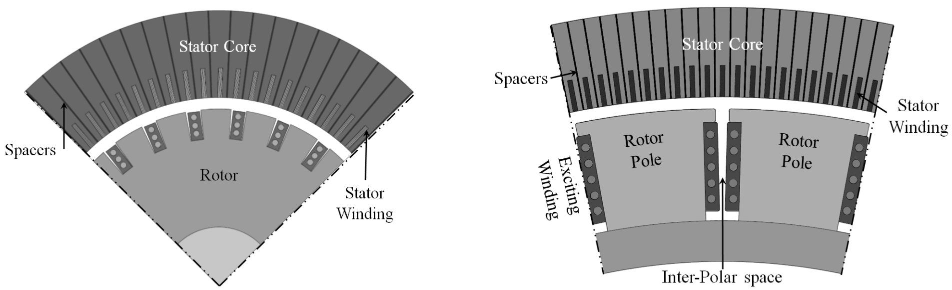


Figure 1.1: *Sketch of Hydro-generator [1].*

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. In salient pole type the rotor consist of large number of poles protrude from the rotor support structure. The structure part which also called spider-rim assembly, is connected to shaft and transmit torque from shaft to the poles. The poles are made up from laminations 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. Hydro-generators usually have a salient pole rotor.

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. Generators may be cooled by air, hydrogen or water. 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.



(a) *Cylindrical Rotor* (b) *Salient Pole Rotor*

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

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.

1.3 Energy Losses

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

$$L_c = I^2 \times R, \tag{1.1}$$

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

$$R = R_0[1 + \alpha\,(T - T_0)]. \tag{1.2}$$

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 in bearings. The windage loss is the power lost due to internal friction in the air flow. The windage loss increase the temperature and decreases the overall efficiency of the machine. The

1.4 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 and thermal analyses [19]. The calculation of the cooling airflow of the machine can be done with flow networks. 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. The network models are able to analyze complex cases 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.

Without a proper understanding of the fluid flow in and around the machines, 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 [15] even in complex regions. However, it is relatively demanding in terms of model setup and computational time. CFD primarily aims at determining coolant the velocity, and pressure distribution in the cooling passages or around the machine. CFD can also determine the surface heat transfer for subsequent analysis of the temperature in the active material and the remaining solid structures. Accurate CFD results can also 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.

There is an increasing trend towards scientific CFD studies of ventilation in electric generators. Changming et al. [8] identified and addressed some challenges in the application of computational fluid dynamics modeling in thermal management of electric machines. Pickering et al. [12] summarized the results of research work to validate the CFD modeling of large salient pole machines. It was shown 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. Shanel et al. [15] investigated the heat transfer and ventilation of an air-cooled generator by using a general purpose CFD code in a simplified generator design. Ujiie et al. [20] 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. Traxler-Samek et al. [18] presented a new method which was based on an iterative numerical coupling of power losses, airflow and temperature calculations. Boglietti et al. [2] present an extended survey on the evolution of the modern approaches of thermal analysis of electrical machines.

Toussaint et al. [17] presented several simulation strategies to numerically compute the flow field in electric generators. Pasha et al. [11] did experimental and CFD analysis on a partial model of stator. They observed that major losses takes place in the wedge zone and at the leading edge of windings. Schrittwieser et al [13] 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. Liang et al. [22] studied the influence of stator ventilation channel cutting length on the multi-physics in air-cooled hydro-generator. Zhang et al. [23] 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. Han et al. [3] studies the flow field of the stator and the air gap for a large air-cooled generator.

Li et al. [21] studied the influence of rotation on the flow and the temperature distribution. They discuss 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. Schrittwieser et al. [14] 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. Torriano et al. [16] 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. Moradnia et al. [4, 9, 10] performed steady-state frozen rotor simulations of a simplified electric generator and validated the results with experimental measurements. Klomberg et al. [5, 6, 7] 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 and the influence of different ventilation schemes on the heat transfer and fluid flow.

1.5 Aims and Scope

The primary aim of this work is to develop experimental and numerical tools for detailed studies of ventilation flow in generators. A series of CFD-based design modifications are done on an existing generator test rig. The new experimental set-up reasonably characterizes a hydro-generator, as well as provides an accurate and relatively simplified geometry for numerical simulation. This apparatus enables fast and reliable measurements of different flow field attributes. A numerical model is set up and validated by the experimental data. A secondary aim is to use these tools to find the strengths and weaknesses of the system in terms of its flow characteristics and provide guidelines to improve the ventilation in generators.

The present work is dedicated solely to the flow of the cooling air. 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 generator with a salient pole rotor.

2 Procedures and Methodologies

This study includes an experimental and a numerical investigation. The experimental apparatus was originally designed and built by Hartono et al. [4]. It is modified 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.

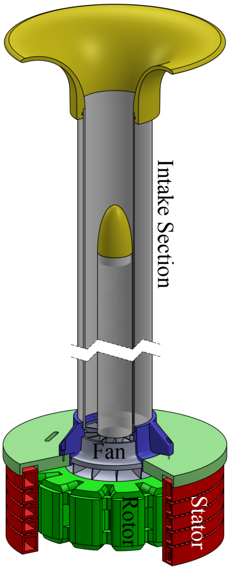


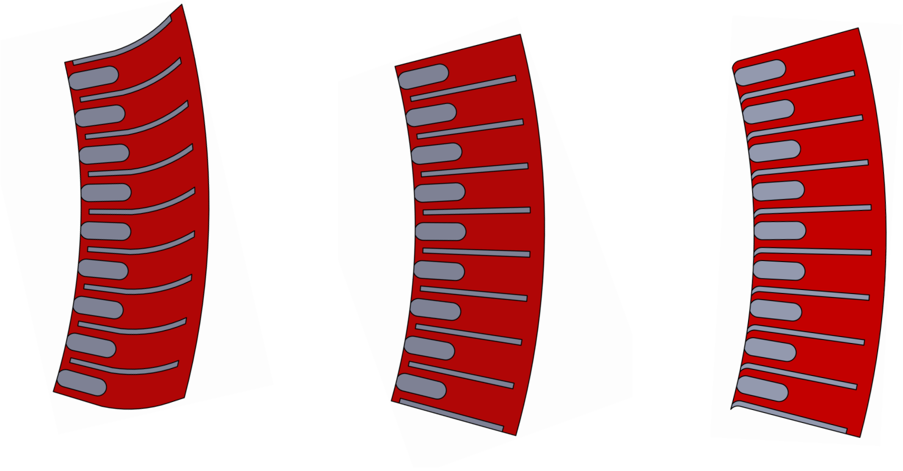
Figure 2.1: *Computer-aided Design of Generator Model*

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 fan attached on top of the rotor, 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 is driven exclusively by the rotor rotation, with its co-rotating fan.

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 has to withstand large centrifugal forces. 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 Calorimetry (PIV).A small section of the stator is manufactured from Plexiglas 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.

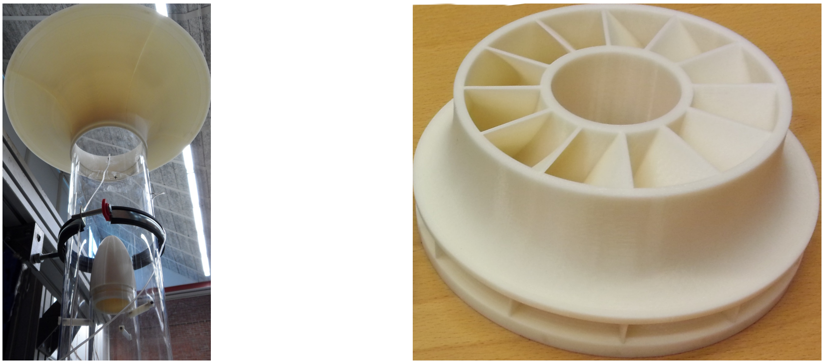


(a) *Curved Spacers* (b) *Straight Spacers* (c) *Spacers with Pick-up*

Figure 2.2: *Three Stator Configurations*

The intake section, see Figure 2.1, is designed to give a controlled inlet velocity distribution with as low losses as possible, and to facilitate direct and accurate flow rate measurements. It consists of two concentric pipes, a bell-mouth at the inlet and a nose cone on top of the inner pipe. The bell-mouth and nose cone are manufactured using PolyJet 3D printing with a rigid opaque material, which leads to smooth surfaces and a high axisymmetric precision, see Figure 2.3a. The pipes are fixed to each other by specially made airfoil-shaped spacers. The airfoil shape of spacers ensure that the flow disturbance and blockage are as low as possible.

The geometry of the bell-mouth consists of a smooth convergent inlet followed by a constant throat area. The convergent part has an elliptical profile. The nose cone on the inner pipe act as an obstruction in the flow stream. It changes the velocity and pressure of the flow gradually like a contraction cone. This part works as a differential pressure type flowmeter device. It allows the calculation of the volume flow rate from the differential static pressure. This pressure difference is acquired from two wall taps located on the wall of the outer pipe, before and after the contraction. The profile of



(a) *Intake Section* (b) *The Radial Fan*

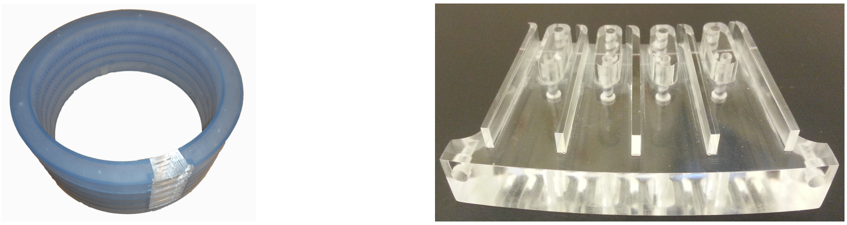
Figure 2.3: *Modified Parts*

this nose is one-half of an ellipse, with the major axis being the centerline and the minor axis being the base of the nose cone which is the inner pipe diameter. The full body of revolution of the nose cone is formed by rotating the profile around the centerline. The proper design of these two parts is verified by CFD simulations before manufacturing. The results show that the inlet flow is passing the bell-mouth and nose cone smoothly without any separation, at the expected flow rates. CFD simulations are also used to determine the proportionality between the measured pressure difference and the flow rate. Several steady-state 2D axisymmetric simulations were done with different inlet flow rate conditions. The comparison of the CFD and experimental volume flow rates shows that the flow rate measurement method is applicable and accurate.

A closed impeller fan is designed in the present work, see Figure 2.3b. The hub and shroud profile are designed so that the flow in the impeller undergoes a gradual deflection in the axial-radial plane, leading to a change in flow direction from axial to radial. The numerical and experimental results show that this fan is able to force reasonable flow into the system, efficiently. A stator is designed for the generator model, see Figure 2.4. 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 Plexiglass pieces, see Figure 2.4b.

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 2.5. The rake 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.

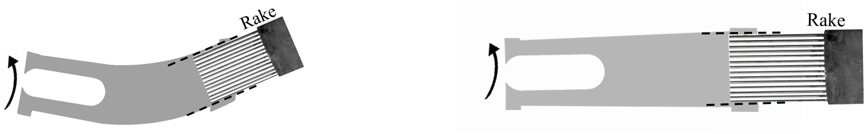


(a) *Stator* (b) *Plexiglass Piece of Stator*

Figure 2.4: *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 ± 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 2.6. 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*

Figure 2.5: *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 %.

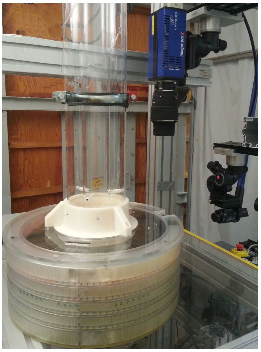


Figure 2.6: *2D PIV Set-up*

2.3 CFD Methodology

The numerical simulations of the generator model are performed using the FOAM-extend CFD tool, and the steady-state multiple reference frame (MRF) concept. The MRF method does not rotate any part of the mesh, but applies source terms in the momentum equations for the rotation. 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 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.

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 $k-\varepsilon$ low-Re and V^2f turbulence model are used in the present study. A block-structured mesh is generated using the *ANSYS ICEM CFD* mesh generator software, keeping the wall y^+ values at about 1. 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 hence includes one rotor pole, one fan blade passage, and nine cooling channels in each channel row. The results are considered converged when the residuals are small and stabilized and the rotor axial torque is stabilized.

3 Summary of Papers

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

3.1 Paper A

Experimental and numerical investigation of hydro power generator ventilation

Aim: To study the effect of the stator channel configuration on their ventilation flow characteristics.
Summary and Outcomes: The flow of cooling air in a lab model of an electric generator is experimentally studied. The experimental data is used to validate numerical simulations of the flow field. The simulations are performed using the steady-state multiple frame of references method, with the frozen rotor approach. The investigations are done under cold conditions, since the present focus is on the flow characteristics of the cooling air inside an electric generator. The numerically predicted flow features agree very well with the experimental results at all experimental sections, and the numerical approach may be considered useful for developing a better understanding of the flow of cooling air in electric generators. The flow field inside the stator channels with pick-up is more even, and the losses are lower, than for the stators without pick-up. The study shows that the flow properties can be improved with small modification of geometry like the addition of pick-up to the spacers. The frozen rotor approach predicts the flow quite different in different channels, according to the relative position of the channels to the pole.

3.2 Paper B

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

Aim: A detailed description of the design modification of a generator model, and an analysis of the ventilation flow distribution in the stator.
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.

3.3 Paper C

Analysis of Air Flow Field Attributes in an Axially Ventilated Generator Model with a Salient-pole Rotor

Aim: Experimental and numerical investigation of velocity and pressure fields in 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.

4 Conclusion

To expand the knowledge of ventilation flow inside hydro-generators numerical and experimental tools are required. The current work aims at providing and validating such tools. A generator model is designed and modified based on the CFD and experimental requirements. The intake section is designed to serve as an accurate flowmeter, and nicely guide the flow into the machine. The fan provides a sufficient inlet flow rate. Different flow attributes are measured using a 5-hole probe, a total pressure rake, and a hot-wire. Also 2D PIV is done inside the channels. 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 asymmetric rotor. 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 presented investigation of the flow properties highlights the necessity to optimize the electric generators from a fluid dynamics point of view, to get an even flow distribution between different stator channel rows and inside each stator channels.

References

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

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

[3] 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.

[4] 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.

[5] 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.

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

[7] 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.

[8] 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.

[9] 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.

[10] 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.

[11] 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.

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

[14] 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.

[15] 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.

[16] 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.

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

[18] 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.

[19] 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.

[20] 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.

[21] 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.

[22] 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.

[23] 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.

Appended Papers

Paper A

Experimental and Numerical Investigation of Hydro Power Generator Ventilation

Paper B

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

Paper C

Analysis of Air Flow Field Attributes in an Axially Ventilated Generator Model with a Salient-pole Rotor