Applied Energy 114 (2014) 644–653

Contents lists available at ScienceDirect

Applied Energy

journal homepage: www.elsevier.com/locate/apenergy

Flow of cooling air in an electric generator model – An experimental and numerical study

Pirooz Moradnia, Maxim Golubev, Valery Chernoray, Håkan Nilsson *

Applied Mechanics, Chalmers University of Technology, SE-412 96 Gothenburg, Sweden

HIGHLIGHTS

- Cooling air flow in a generator model investigated numerically and experimentally.
- Fully predictive numerical approach, without the use of experimental data.
- High quality PIV and pressure measurements.
- Similar flow characteristics in numerical predictions and experimental validations.
- Close numerical flow rate predictions to the experimental data.

ARTICLE INFO

Article history:
Received 14 May 2013
Received in revised form 16 September 2013
Accepted 13 October 2013
Available online 9 November 2013

Keywords:
Electric generator
Air flow
CFD
Measurements

ABSTRACT

The need for electric power is continuously increasing. The power output of existing electric generators is forced to its limit, and the new intermittent electric energy sources increase the variations of the operating conditions of the electric generators in the power system. This requires better cooling of the heat that is generated by the electric losses in generators. New experimental and numerical techniques need to be developed and validated, to increase the knowledge of the cooling processes and to improve the accuracy of the design tools. The present work focuses on the flow of air through electric generators, as a necessary and important first step towards future accurate and detailed convective heat transfer analysis.

A half-scale model of an electric generator is designed and manufactured specifically for detailed experimental and numerical studies of the flow of cooling air through the machine. The model is slightly simplified compared to the original geometry, to benefit from the use of geometry parameterization, and with numerical mesh quality in mind already at the design of the experimental set-up. Special care is taken to provide optical access for accurate and detailed Particle Image Velocimetry (PIV) measurements inside the machine.

The experimental measurements include PIV measurements at the inlet and inside the machine, and total pressure measurements at the outlet of the stator channels.

Computational Fluid Dynamics (CFD) simulations are performed using two approaches. In 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, yielding predictions differing by 2–7% compared to the experimentally estimated values. The results of both approaches capture the experimental flow details to a high level of accuracy. Mesh sensitivity studies highlight the need of a specific resolution of the baffle edges.

1. Introduction

Electric power generation most of the time involves the use of electric generators. Electric and magnetic losses generate heat that raises the temperature inside the machines. Intermittent operation of electric generators, due to interaction between new energy sources, causes fluctuating temperatures and thermal stresses. A detailed knowledge of convective heat transfer in electric generators is of great importance for the design of efficient machines, and for reducing their maintenance requirements. An elevated temperature increases the electrical resistance of the stator coils, and may cause hotspots that can damage certain components and shorten their lifetimes. Convective cooling of electric generators introduces air flow losses which may account for about 30%

* Corresponding author. Tel.: +46 317721414.
E-mail addresses: p.moradnia@gmail.com (P. Moradnia), maxim.golubev@chalmers.se (M. Golubev), valery.chernoray@chalmers.se (V. Chernoray), hakan.nilsson@chalmers.se (H. Nilsson).

0306-2619/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.apenergy.2013.10.033

of the total generator losses. Important goals in the design of electric generators are thus to obtain a low and uniform temperature distribution, and to keep the losses due to the flow of cooling air at a minimum.

An analysis of convective heat transfer primarily requires an accurate representation of the flow field. There are very few available studies performed on the flow of cooling air in electric generators. Toussaint et al. [2] presented different simulation strategies to numerically compute the flow field in electric generators. Their comparisons of the steady-state and transient simulations showed that the frozen-rotor results are highly dependent on the relative rotor–stator position and the radial position of the Rotor–Stator Interface (RSI) in the air gap. The mixing-plane simulations were shown to be independent of the relative rotor–stator position, but averaging the fluxes at the RSI removed the flow structures at the interface. Moradnia et al. [1] performed steady-state and transient simulations for 2D representations of a simplified electric generator, and discussed different simulation parameters and setups. The time-averaged results of the transient simulations were shown to be more accurate than the steady-state results. However, the high computational effort required to perform the simulations in transient mode was shown to be a significant drawback. Gunabushanam et al. [4] experimentally and numerically investigated the cooling air flow losses in an electric generator model. Their results demonstrated 10–30% error in the prediction of the flow losses. Pickering et al. [3] experimentally and numerically investigated the air flow and heat transfer in a salient pole machine. The simulations were performed in steady-state mode, with inlet boundary conditions taken from the experimental data. Although generally following a similar trend as the experimental data, their numerical results differed from the experimental data by more than 30% and 20% for the local heat transfer coefficients and the volume flow distributions between the stator cooling channels, respectively. The heat transfer coefficient is strongly dependent on the wall shear stresses, and the volume flow distribution between the channels is governed by the flow losses. An accurate prediction of the flow losses and the wall shear stresses are thus extremely important in predicting the correct flow field. The heat transfer coefficient depends on the temperature gradient at the surfaces, which has a strong relation to the velocity gradient at the surfaces. An error in the prediction of the heat transfer coefficient is thus accompanied by an error in the flow losses. A specification of the inlet volume flow rate may thus yield a large error in flow losses, which is however seldom discussed in literature.

In order to remove the dependence of the simulation results on the availability of experimental measurements at the inlet, a fully predictive numerical approach is used in the present work. In this approach the computational domain is extended to include parts of the ambient air surrounding the generator. The inlet and outlet boundaries are thus eliminated, and the cooling air recirculates within the computational domain. In the case of an electric generator, the flow is driven solely by the rotation of the rotor, interacting with the stator. The flow field and the flow rate through the machine are hence determined by the solution, rather than by prescribed values at the inlet. Using this numerical approach, Moradnia and Nilsson [6] investigated the effect of different rotor and stator geometries on the flow field in an electric generator. The results showed that the volume flow through the machine may be significantly increased by confining the inlet of the machine, and by using fan blades on the rotor. Early experimental comparisons by Moradnia et al. [7,8] confirmed the numerically predicted flow behavior at the inlet and outlet. However, it was concluded that more detailed experimental studies and more investigations of the numerical setup were required to verify the approach.

In the present work, a lab rig is designed and manufactured for detailed Particle Image Velocimetry (PIV) and pressure measurement purposes. The geometry is a half-scale model of the electric generator studied by Morandia et al. [7,8]. The Reynolds number is preserved by increasing the rotational speed in the half-scale model. The generator components are manufactured using rapid prototyping techniques, allowing future modifications of the geometrical details at a low cost. The rig is designed exclusively for investigations of the flow of cooling air, which allows modifications of the original geometry that increase the experimental accessibility, increases the geometrical accuracy, and removes unwanted effects from the electromagnetic components. Given the freedom to do small modifications to the geometry, it is also slightly adapted to the requirements from the CFD simulations. The geometry is thus parameterized, and prepared for the generation of high-quality meshes for the CFD simulations.

The experimental results of the present work are used to validate new numerical simulations performed using the fully predictive numerical approach, and the commonly used approach with inlet and outlet boundary conditions. Validation and mesh sensitivity studies are also performed. The studies are carried out in cold conditions without heat transfer, since the present work is dedicated solely to investigating the flow of cooling air in an electric generator, and its associated losses. The study is a step towards future convective heat transfer analyses.

The geometry and operating conditions of the generator model are described in detail in Section 2. The governing equations are presented in Section 3. The experimental details are reported in Section 4. The numerical cases are described in Section 5, and the results and conclusions are discussed in Sections 6 and 7 respectively.

2. Geometry and operating conditions

The geometry used in the present work is a half-scale model of the electric generator studied by Moradnia et al. [7,8]. Fig. 1 shows the experimental rig and the CAD model of the generator used in the present work. The experimental rig is designed and manufactured exclusively for detailed measurements of the flow of cooling air, and is adapted for Particle Image Velocimetry (PIV) and pressure measurements. The geometry of the rotor and stator are slightly simplified compared to the original geometry, with the purpose of improving the accuracy of experimental results, and preparing for high-quality CFD simulations. The rotor and stator are manufactured using a rapid prototyping method, which makes it possible to easily manufacture new designs at a low cost. The rotor is manufactured using a Stereo Laser Sintering (SLS) process, since it has to withstand large centrifugal forces. The stator is manufactured using a Stereo Lithography Apparatus (SLA) process [11]. The generator model has 12 poles, 12 radial fan blades attached on top of the rotor, 4 rows of stator cooling channels along the axis of rotation, and 108 cooling channels and coils in each row. The stator height is 0.175 m. The rotor tip radius and the stator inner and outer radii are 0.178, 0.1825 and 0.219 m respectively, as shown in Fig. 1(c). The stator cooling channels are shown in Fig. 1(d). The height of the stator channels is 4.7 mm. The air flow is driven by the rotating parts of the generator, without external control, as in the original generator. The Reynolds number, based on the rotor tip speed and radius, is kept the same as that in the full-scale generator, i.e. $Re_{1:1} = Re_{1:2} \approx 1 \times 10^5$. The rotational speed of the rotor in the full-scale model, $\omega_{1:1} = 500$ rpm, thus yields $\omega_{1:2} = 2000$ rpm.

The computational geometry is based on the one used in [7,8], as seen in Fig. 1(b) and (c). It thus differs from the experimental rig in that it includes the rotor rings and shaft above the top baffle. These details are exchanged in the rig with a rotating baffle that is aligned with the stationary top horizontal baffle, resembling the lower surface of the rotor rings. The geometric differences are