15th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2011, Prague, Czech Republic, 12-18 September 2011

INVESTIGATING THE MALFUNCTION OF A HYDRO-GENERATOR'S COOLING-SYSTEM

Aleš Hribernik, Matej Fike
University of Maribor, Faculty of Mechanical Engineering
Smetanova 17, Maribor
Slovenia

Boštjan Gregorc
Dravske elektrarne Maribor
Obrežna ulica 170, Maribor
Slovenia

ABSTRACT
Electro-magnetic losses within a hydro-generator are sources of heat which heat the generator, and then have to be conducted away. This may, however, cause a temperature rise of the stator bars thus reducing the life-span of the insulation materials. Therefore, any temperature rise of the stator windings is limited. In order to inspect the operation of a cooling-system, several temperature sensors have to be integrated into the stator core in order to measure its temperature, which should not exceed 80 ^{0}C. Temperature measurements performed on a generator showed some deviations from the design temperatures, especially at full-power operation. Thus, an attempt was made to analyse the operation of a cooling-system, and to eventually increase its cooling capacity. It was impossible to perform any complex measurements during the generator's operation, due to safety reasons. Therefore, our work was limited to 3D CFD simulations using ANSYS CFX 11 solver. The simulations were partly supported by the measurements of some main integral parameters, such as mass-flow rate and pressure drop across the cooling ducts and radiators, as performed during generator idling. The results are presented and discussed in this paper.
Keywords: Hydro-generator, cooling-system

1. INTRODUCTION
Electro-magnetic losses within a hydro-generator are sources of heat which heat the generator and then have to be conducted away. This may cause, however, a temperature rise of the stator bars thus reducing the life-span of the insulation materials. Therefore, standardization such as DIN VDE 0530 limits the temperature rises of stator-windings [1]. There are three possible cooling concepts for hydro-generator cooling: air-cooling, water-cooling, and evaporation-cooling. Air-cooling is a standard cooling concept for small to large hydro-generators. It is the more favoured because of its simplicity, maintenance, and reliability. Its only disadvantage is its large volume. Water-cooling is used for an extreme high-output generator, whilst evaporation-cooling, which uses the effect of heat-absorption by gasified liquid, has rarely been applied.
This paper, presents an investigation into the malfunction of a 10 MW hydro-generator's air-cooling-system. This generator cooling-system is schematically shown in Figure 1. Air is used as one cooling-media within a closed cooling-system. It takes the heat from the mechanical parts of generator and transfers it to the water of a secondary water-cooling system. Two axial fans are used, one at the bottom and one at the top of the generator, in order to transport the cooling-air through the system. The air-flow is separated immediately after passing the fan. Some 25 % of the air-flow leaving the fan

does not enter the rotor region, but flows radially through the stator's end-windings. The main bulk of the flow (about 75 %) continues its way in an axial direction through the openings between the rotor poles, which act as radial fan blades and gradually direct the air in a radial direction towards the stator-core. After-leaving the rotor at a high tangential velocity through the air-gap (10 to 20 mm space between rotor and stator), the air enters the cooling-ducts of the stator-core. After leaving the core the main air-flow is then mixed with the air-flow that cools the end-windings within the stator retaining-frame, and is then directed to the water-radiators where it is cooled and returned to the circuit at the inflow-sides of both axial fans. In order to inspect the operation of this cooling-system, several temperature sensors had to be integrated into the stator-core in order to measure its temperature, which should not exceed 80 ^{0}C for the reasons already stated in the first paragraph. The temperature measurements performed on the generator showed some deviations from the design temperatures, especially at full-power operation. Thus, an attempt was made to analyse the operation of the cooling-system, and to eventually increase its cooling capacity.

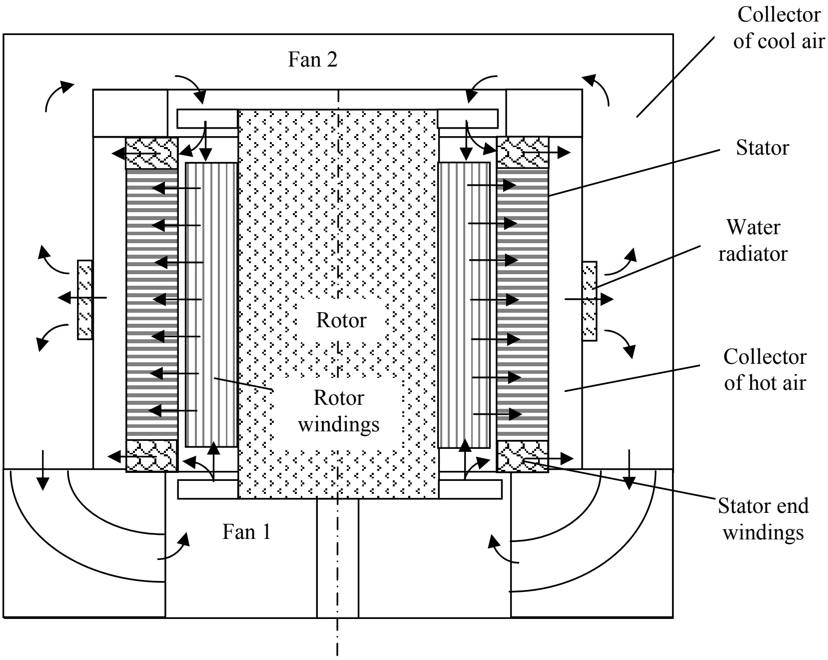


Figure 1. Hydro-generator's cooling-system

It was impossible to perform any complex measurements during the generator's operation, due to safety reasons. Therefore, our work was limited to numerical simulations supported with measurements from certain main integral parameters, such as mass-flow rate and pressure drop across the cooling ducts and radiators, as performed during generator idling.

2. PRELIMINARY MEASUREMENTS

The operation of a set of water radiators was checked during the first step Both the water and air-flow rates were measured, and the inflow and outflow temperatures acquired. Calculations showed that the actual heat-flow transferred from the cooled air to the water covered only 70 % of the generator's losses, which is obviously the reason for the generator overheating. The question then arose as to whether the radiators operated well or not. Increasing the water-flow rate did not bring any improvement, therefore it was concluded that the flow-rate had to be increased on the cooling-air side. In order to analyse the complex air-flow within the hydro-generator, numerical simulation was applied using Computational Fluid Dynamics (CFD).

CONTENTS
AUTHORS

3. NUMERICAL SIMULATION

The application of CFD to a hydro-generator is relatively recent; Ujiie *et al.* [2] and Schlemmer *et al.* [3] are good examples of what can be done. Both of these studies used a computational-domain covering both the rotor and the stator, by treating the junction between the two with a frozen rotor approach.

The turbulent flow which prevails within a generator forces the use of turbulence-models in order to solve Navier-Stokes equations in their averaged form (Reynolds Averaged Navier-Stokes - RANS). This kind of simulation, also called global-simulation, provides a good overview of flow-distribution within a generator, however, it lacks those detailed flow-patterns within the cooling-channels of the stator-core necessary for predicting local heat-transfer conditions. Therefore, an iterative process was applied involving multiple RANS simulations; global and local simulations. The idea was to use the results of simplified global-simulation as boundary conditions for local – detailed simulation.

The general CFD methods used by commercial software packages provide powerful tools for analyzing flow within complex geometries. The solver used for the presented project was ANSYS CFX 11. All the simulations presented in this paper were done using a k-ε model. (along with a wall-function). The iterative process used to solve the resulting matrix was based on the first order Euler time marching scheme. The maximum residual at 10^{-4} was deemed to be sufficiently low to provide good resolutions of the system of equations on the specified grid with the given boundary conditions.

3.1. Global simulation

Firstly, the main part of the cooling-air stream, which flows in the axial direction through the openings between the rotor poles, was simulated in order to gain more-realistic boundary conditions for a detailed-simulation of flow within the cooling-ducts of the stator-core, where very acute heat-transfer takes place. The computational domain, together with some of the stream lines, is presented in Figure 2. A structured hexahedral mesh with 2 million elements was used. Application of the periodical boundary condition allows only one fan-blade and four rotor poles to be simulated. The air-stream enters the axial fan at the top of the domain and flows through the fan, which accelerates it towards the rotor-poles. An abrupt flow-area change takes place here and the air is squeezed into narrow channels between the rotor poles. Eventually the cooling air leaves the rotating channel in the radial direction and enters the cooling ducts of the stator-core. Averaged five components outflow velocity profile (*u, v, w, k, ε*) was computed, and applied to the detailed simulation.

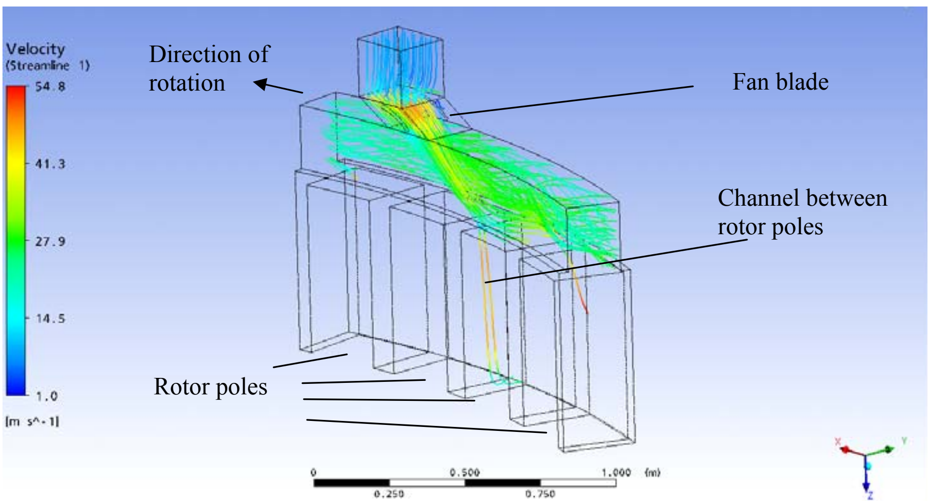
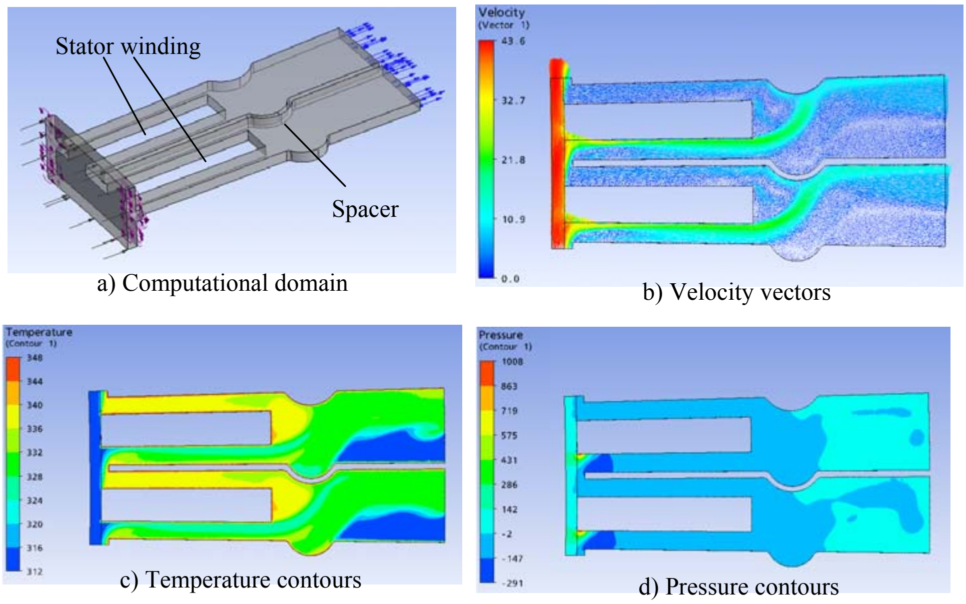


Figure 2. Global simulation

3.2. Detailed simulation

Detailed simulation was used to study the flow and heat transfer within the cooling ducts of the stator-core. The computation grid used structured hexahedral mesh with 1 million elements. Refinement near the walls enabled a good resolution of the wall function used, in conjunction with the k-ε model. The inlet condition used the 5 components averaged velocity profiles (*u, v, w, k, ε*) obtained from the global-simulation. The computational domain, and some of the results, are presented in Figure 3.



a) Computational domain b) Velocity vectors

c) Temperature contours d) Pressure contours

Figure 3. Results of detailed-simulation of flow and heat-transfer conditions within a cooling-duct

There is a very high tangential velocity component at the inlet, which forms a high-intensity vortex within the gap between the rotor and stator, and intensifies the heat-transfer at the pressure side of the cooling-duct. On the other hand, the flow of cooling-air is substantially reduced on the suction-side of the cooling-duct, which remains uncooled. This implies a highly asymmetrical temperature distribution within the stator's winding-bars, and possible local temperature peaks. A high pressure drop at the inflow was observed. It is believed that it is caused by the spacer, which induces a vortex at the suction-side of the cooling-duct, and almost totally blocks the air-flow in the radial direction.

4. CONCLUSION

The study has been presented of a hydro-generator cooling-system's malfunctions. 3D CFD simulations using ANSYS CFX 11 solver were used to analyse the flow and heat-transfer conditions within the cooling-ducts of the stator-core. An iterative process was applied involving multiple RANS simulations; global, and local-simulations. The results from the simplified global simulations were used as boundary conditions for a detailed-simulation of the cooling-duct. The results show highly asymmetrical conditions within the cooling-duct, due to a high tangential velocity-component. The spacer, which is placed within the cooling-duct, increases these uneven conditions even more, and should be relocated.

5. REFERENCES

[1] Shoenjauer, J., Schlemmer, E., Mueller, F., Finite element modeling of the temperature distribution in the stator of a synchronous generator, The International Journal for Computation and Mathematics in Electrical and Electronic Engineering, Vol. 22 No. 4, 2003, pp. 909-921

[2] Ujiie, R., Arlitt, R., Etoh, H., Application of Computational Fluid Dynamics (CFD) on ventilation-cooling optimization of electrical machines, REVIEWENERGY TECHNOLOGIES generation, transmission and distribution of electric and thermal energy, contents 4, pp. 17-22, 2006

[3] Shlemmer, E., Schoenauer, J., Farnleitner, E., Mueller, F., Transient electromagnetic and coolant flow investigation of synchronous generators using numerical approaches, ICEM, Krakow, Poland, 2004