CFD Analysis of a Heat Exchanger for an Electric Machine

A. Curci[1], D. Falchi[2], G. Secondo[1]

[1]ABB S.p.A. Italy
[2]Università degli studi di Pavia, Italy

Abstract

In recent years the thermal behavior of electric machines is an attractive research topic. Due to the complexity of the problem, several approaches that exploit FEM analysis have been developed and presented in literature. In this research a 3D thermo-fluid dynamic simulation of an electric machine equipped with rubber belts directly applied on its shaft has been performed through COMSOL Multiphysics software. Aim of this investigation is the temperature distribution map of the whole motor, especially on the shaft and on the finned heat exchanger that is mounted on it.

The cooling system of the motor consists in two fans with constant airflow rate of 33 m^3/h per each. The first analysis has been done on the original configuration of the machine and, based on this case study, a campaign of optimization in terms of topology of the heat exchanger and variation of the airflow input has been conducted. Figure 1 shows the machine and the belts area with the heat exchanger mounted on the shaft. Laminar Conjugate Heat Transfer is the physic used in this model. The equations of conservation of mass, momentum and energy have been used for the numerical calculation in order to take in account the gravitational forces that are responsible of natural convection (in COMSOL represented by volume force) and the heat sources that, in this case, are represented by the electrical motor losses that have been calculated through an electromagnetic simulation. Moving wall and Translational motion have been implemented in order to consider the airflow motion due to the rotation of the shaft and the heat exchanger. The rotational speed is constant and equal to 114 rad/s. The boundary conditions set consists in: atmospheric pressure equal to 1 atm, airflow domain speed at initial condition equal to 0 m/s and ambient temperature of 27 degC. Figure 2 shows the inlet and outlet surfaces of the airflow in the model. For each steady-state simulation the same losses and rotational speed values have been used. Three different cases studies have been pointed out: the first one is the original configuration, in the second one a different orientation of the inlet airflow which reflects a different positioning of the fans of the motor has been presented and the last one in which the topology of the heat exchanger in terms of slot between fins has been modified. Figure 3 shows the streamlines of airflow that highlight temperature distribution of the model, while figure 4 shows the temperature distribution on the heat exchanger for the three different cases. From the analysis of simulation results, the best configuration seems to be that one in which the slot between the fins of the heat exchanger is increased. In fact this solution allows to reach a good convective heat transfer coefficient between the fins due to the possibility of the airflow to flow

better between them.

Reference

"B. Desmet", "B. Watel", "S. Harmand", Influence of fin spacing
and rotational speed on the convective heat exchanges from a rotating finned tube, International Journal of HEAT AND FLUID FLOW, 21(2):221-227, (1999)

"S. Harmand", "B. Latour", "P. Bouvier", Convective heat transfer on a rotating finned cylinder with transverse airflow. International Journalof HEAT AND MASS TRANSFER, 54(2):4710-4718, (2011)

"K.Morgan", "R.W.Lewis" , "O.C. Zienkiewicz", Numerical Methods in Heat Transfer, Wiley, (1981)

Figures used in the abstract

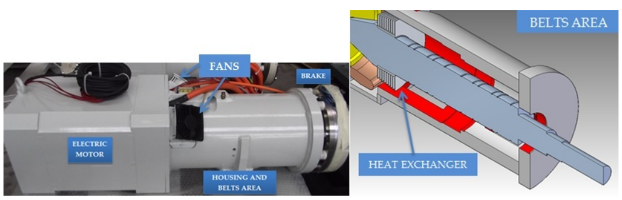

Figure 1: Electric machine and heat exchanger mounted on the shaft

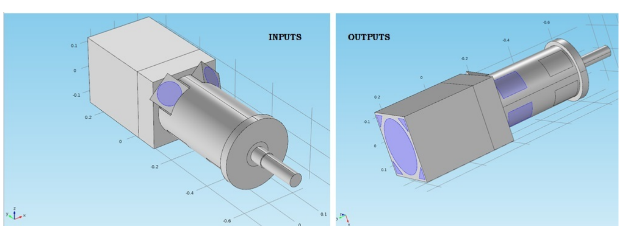

Figure 2: Inputs and outputs of airflow

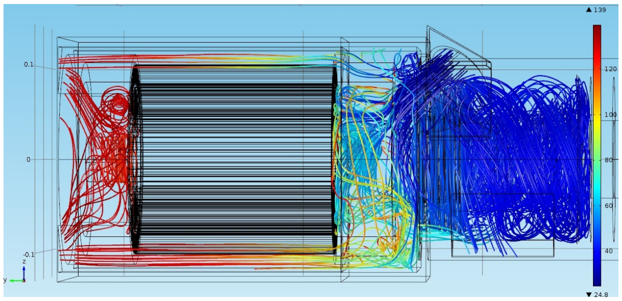

Figure 3: Streamlines and temperature distribution

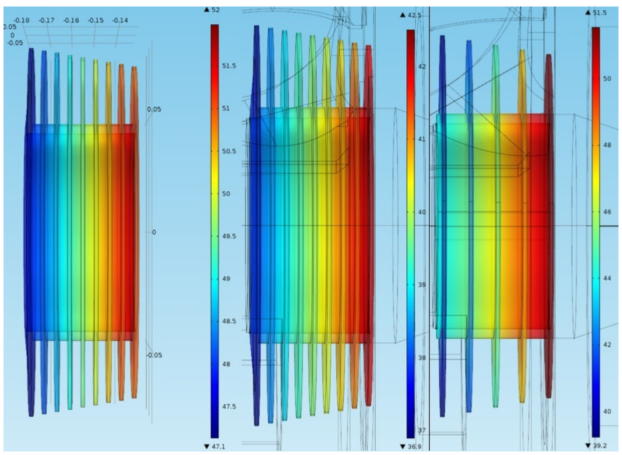

Figure 4: Temperature distribution of the heat exchanger at the different case studies