Second International Conference on Computational Methods for Thermal Problems
THERMACOMP2011, September 5-7, 2011, Dalian, China
X.K.Li, N.Massarotti and P.Nithiarasu (Eds.)

Thermal modeling of power transformer radiators using a porous medium based CFD approach

Rebei Bel Fdhila, Jurjen Kranenborg, Tor Laneryd, Carl-Olof Olsson, Bertil Samuelsson
ABB Corporate Research, Västerås, Sweden, {rebei.bel.fdhila, jurjen.kranenborg, tor.laneryd, carl-olof.olsson, bertil.samuelsson}@se.abb.com

Andreas Gustafsson, Lars-Åke Lundin
ABB Power Products Division / Transformers, Ludvika, Sweden, andreas.gustafsson@se.abb.com

ABSTRACT

In power transformer cooling applications, the heat transfer in large fan-cooled radiator batteries and their immediate surroundings can be successfully studied using a CFD model in which the mixed convection based oil and air flows and the heat transfer between these media in the radiators is modelled using an anisotropic porous medium approach and the area surrounding the radiators using a standard turbulent heat transfer model. The resulting model appears fast enough to allow a large number of realistic simulations to be run to study the effect of radiator design parameters like fan position, fan size and oil flow rate on the cooling capacity of the radiators. As an example the analysis of the effect of fan size on radiator cooling capacity is shown.

Key Words: *Heat Transfer, Radiators, Power Transformers, CFD, Porous Medium.*

1. INTRODUCTION

Power transformers are large products that are often cooled using large assemblies of radiators with fans. The footprint of these radiators is generally of similar size as the power transformer itself. The radiator configuration is tailor-made for each transformer and can be described using a number of design parameters like height and width, number of panels per radiator, number of radiators per assembly, the number and size of fans used as well as their position (horizontal, vertical) on the radiators. As a consequence, a radiator assembly is a complex geometrical entity with a cooling oil flow in small ducts in the radiator panels, a cooling air flow in the small sections between the panels as well as an air flow in a large area surrounding the radiator. Such a setup is well worth studying using a CFD model, an approach that has already been successfully used for the simulation of mixed convection in power transformer windings [1, 2].

In order to allow a large number of simulations as required for studying practical cooling scenarios, a porous medium approach has been chosen to model the radiators on a fine numerical grid, coupled with a turbulent heat transfer model on a much coarser grid for the heat and mass transport in the air surrounding the radiators. To this end the background literature on heat transfer in porous media has been reviewed [3, 4, 5] and published anisotropic models were used for porosity and thermal conductivity [6] to account for the radiator panel orientation.

2. TEST CASE AND CFD RADIATOR POROUS MEDIUM MODEL DESCRIPTION

A specific transformer design as shown in Figure 1 has been chosen as the reference case, serving as a starting point for many of the model variants studied. The thermal parameters that are relevant for radiator cooling (top oil and bottom oil temperatures, thermal losses and net oil flow rate through the radiators) were derived from the heat run test data and the design model. In the same figure the geometry of the CFD model is depicted. Only half of the radiator domain including two radiator groups (each group having three radiators) needs to be modeled due to the symmetry that is present in this particular design.

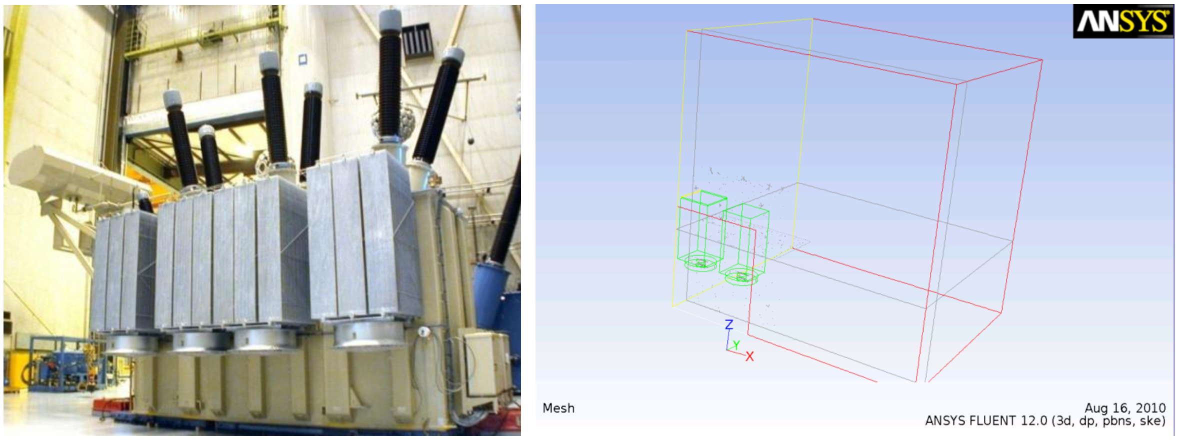

FIGURE 1. Reference transformer design (left) and corresponding CFD model geometry (right) that includes the radiators as well as a large surrounding area to minimize the effect of computational boundaries on the simulation results.

In the radiator domain, general porous medium equations for heat and mass transfer are solved separately for both air and oil (both phases modeled using mixed convection), and the heat transfer from oil to air is modeled using a common heat transfer coefficient. The model coefficients (flow resistance and thermal conductivity) that are used in the radiator domain are anisotropic to represent the parallel-panel structure of the radiators. The fans are modeled as a volume where a known velocity is specified on the boundary of the fans. The outer computational domain boundaries model the floor, the transformer tank wall (adiabatic in our simulations, so only affecting the flow) and open domain boundaries (the latter located far away from the radiators and using pressure outlet conditions, such as to minimize the effect on the computational result). Since the cooling oil flow in the transformer under study is driven by natural convection (as a consequence of the buoyancy difference between the warm oil inside the transformer tank and the colder oil in the radiators), the total oil flow rate through the radiator is fixed based on the reference design estimate and the oil inlet temperature is fixed as well (since the transformer itself is not modeled). In this way the main measure of the radiator cooling capacity we use is top-to-bottom oil temperature difference; for a given thermal loss a larger difference implies a larger cooling capacity.

The CFD model was implemented using ANSYS Fluent 12.0. Special care has been taken for parallelization of the code, and as a consequence the model has shown to be efficient allowing a typical scenario case to be calculated in just a few hours on 32 nodes of the computational cluster at ABB Corporate Research.

3. RESULTS

Figure 2 shows the simulation results for the reference design case used to validate the model. In this figure the computational grid is shown at a horizontal cross-cut through the radiators, revealing the dense, accurate grid inside the radiators (allowed by use of the efficient porous medium approach), the coarse blue regular grid at the outside as well as the intermediate (pink) grid in between. The validation results show good agreement between measured and calculated results.

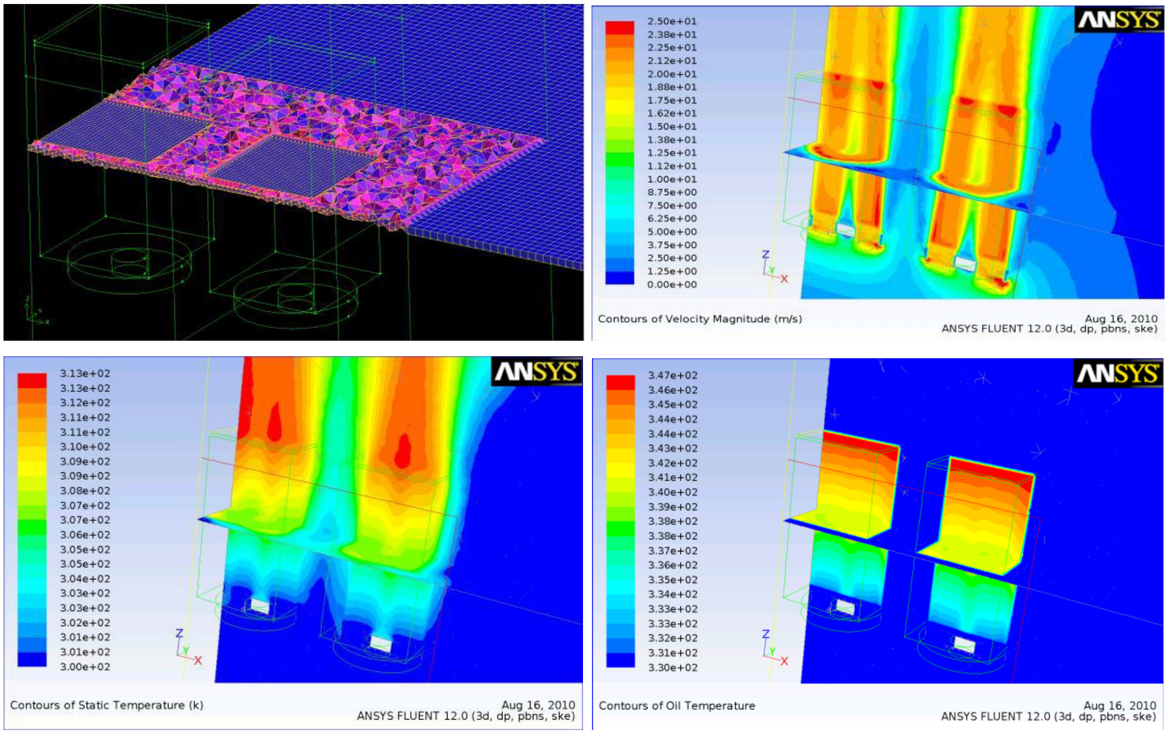

FIGURE 2. validation simulation results. Upper left: impression of computational grid (horizontal crosscut). Upper right: air flow magnitude (m/s). Lower left: Absolute air temperature (Kelvin). Lower right: oil temperature distribution in radiator.

The CFD model has proved to be very useful in generating a large database of solutions for analyzing various design scenarios, in which the effect of a radiator design parameter on cooling capacity is studied. One of these scenarios deals with the effect of radiator fan diameter. We have investigated whether smaller fans have an advantage over less but larger fans (assuming an equal net air flow through the radiators to allow the cases to be compared) since the smaller fans allow a better coverage of the lower radiator air inlet area. The fan diameter has been varied between 0.8 and 1.5 m, allowing two, three or four fans to cover the lower air inlet area of the radiator groups (with the two radiator assemblies as shown in figure 2 now modeled as a single group). As an example for the case with three fans, Figure 3 shows that the inlet coverage has a profound effect on the temperature distribution in the radiators and thus may be expected to affect the cooling capacity as well.

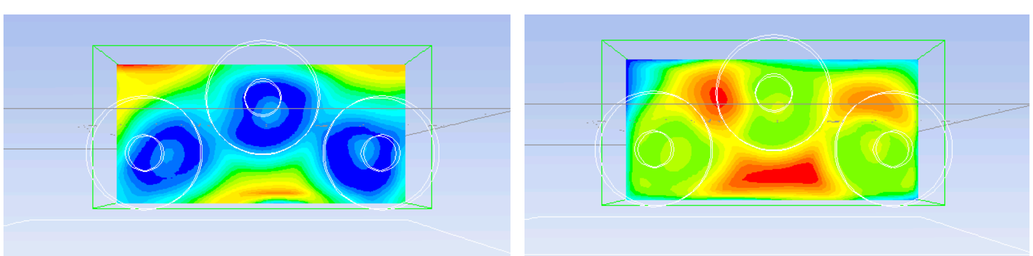

Figure 3. Air (left) and oil (right) relative temperature distribution in the radiators at horizontal mid-plane cross-section (same position as in Figure 2) for a radiator inlet coverage with three fans.

In Figure 4 the cooling capacity results for the different fan diameters have been collected. It reveals that the top-to-bottom temperature difference and thereby the cooling capacity increases if the same amount of air is spread more evenly with a larger number of smaller fans because of the improved coverage. The figure also shows that the transformer tank wall has some influence on cooling capacity since the two configurations with three fans do not give exactly the same result. The maximum cooling capacity improvement obtained in this way is of the order of 5% – 10%, based on the investigated cases.

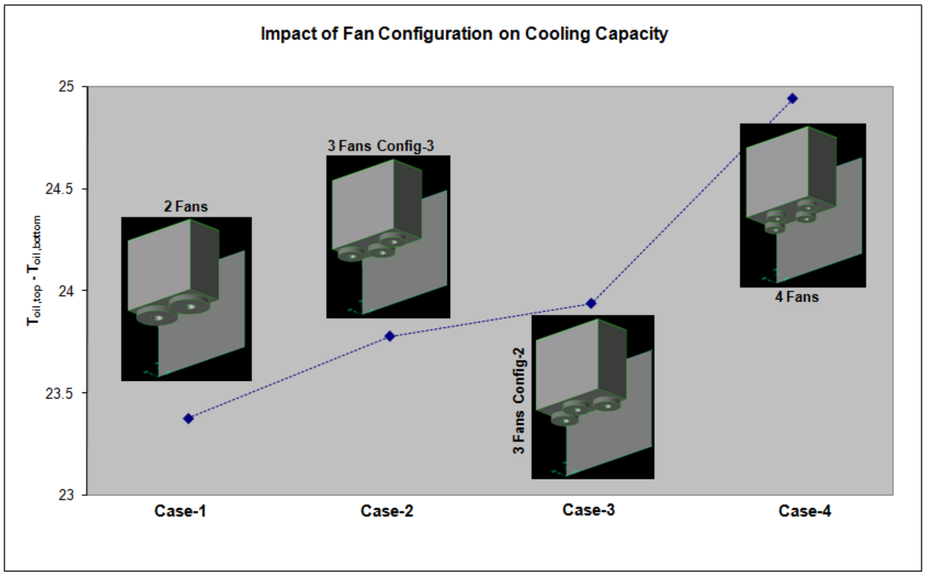

FIGURE 4. Cooling capacity (in terms of top-to-bottom oil temperature difference) for four different fan configurations with different fan diameter.

4. CONCLUSIONS

The CFD model presented here provides a fast yet accurate approach for performing realistic simulations on an important class of radiator cooling problems. Using the porous medium approach the execution time becomes small enough to allow design scenarios to be investigated that require a large number of simulations. Consequently, CFD methods can be applied directly to derive design guidelines for large industrial cooling equipment like fan-cooled transformer radiators.

REFERENCES

[1] E.J. Kranenborg, C. O. Olsson, B. R. Samuelsson, L-Å. Lundin, R. M. Missing, Numerical study on mixed convection and thermal streaking in power transformer windings, *5th European Thermal-Sciences Conference (EUROTHERM 2008)*, The Netherlands, 2008.

[2] P. Picher, F. Torriano, M. Chaaban, S. Gravel, C. Rajotte, B. Girard, Optimization of transformer overload using advanced thermal modeling, *CIGRE Biannual meeting 2010*, Paris, France, 2010

[3] Ergun, S., Fluid Flow Through Packed Columns, *Chemical Engineering Progress*, pp. 89-94, vol. 48, No 2, 1952.

[4] Bejan, A., Designed Porous Media: Maximal Heat Transfer Density At Decreasing Length Scales, *Int. J. Heat Mass transfer*, pp. 3073-3083, vol. 47, 2004.

[5] Bejan, A., Dincer, I., Lorente, S., Miguel, A. F. and Reis, A. H., "*Porous and Complex Flow Structures in Modern Technologies*", Springer-Verlag, 2004.

[6] Morega, A. M., Bejan, A. and Lee, S. W., Free Stream Cooling Of A stack Of Parallel Plates, *Int. J. Heat Mass transfer*, pp. 519-531, vol. 38, No 3, 1994.