# CFD ANALYSIS OF CORE TYPE POWER TRANSFORMERS

Carlos M. FONTE
FEUP – Portugal
fonte.carlos@fe.up.pt

José Carlos B. LOPES
FEUP – Portugal
lopes@fe.up.pt

Madalena M. DIAS
FEUP – Portugal
dias@fe.up.pt

Renato G. SOUSA
BOSCH Termotecnologia – Portugal
renato.sousa@pt.bosch.com

Hugo M. CAMPELO
EFACEC – Portugal
hugo.campelo@efacec.com

R. Castro LOPES
EFACEC – Portugal
ricardol@efacec.com

## *ABSTRACT*

*To avoid recurring to experimental verification of hot-spots each time a new power transformer is designed, mathematical models can be used in the project of new devices to control temperature distributions inside the transformer.*

*In the present work, CFD simulations were carried out to analyse the flow distribution and heat removal in core type power transformers. Experimental measurements are complex and intrusive and can only measure a limited number of discrete points. On the other hand, CFD has been shown to be a suitable tool of analysis and it can play a major role on the optimization and design of new geometries by giving detailed information of flow and temperature distribution, including the magnitude and position of hot-spots.*

## INTRODUCTION

Power transformers dissipate large amounts of energy while converting the electric current and voltage from primary to secondary windings. The heat generated in the coils is removed by circulating a fluid, like mineral oil, in the space around the coils. An inefficient heat removal may decrease the lifetime expectancy of power transformers since high temperatures enhance the degradation of the paper used in the windings insulation. Under normal conditions, a power transformer can work correctly for up to 20 years, but an increase of 6°C over the specified limit of the hot-spot maximum temperature can decrease the lifetime by a half [1]. Therefore, it is of great importance to know the detailed description of the flow field inside the power transformer, between the coils, and the information of the heat removal efficiency from the copper, including the position and magnitude of hot-spots.

Experimental measurements are complex and intrusive and can only measure a limited number of discrete points. For this reason, Computational Fluid Dynamics (CFD), can be used to predict the oil flow and temperature distribution in power transformers, giving the necessary detailed information.

Commercial CFD codes have been previously applied to disc type windings to estimate temperature distributions, prediction of the hot-spot position, and respective parametric studies of the influence of the Reynolds and Prandtl numbers, eddy current losses and geometrical arrangements [2-4]. Another work focused on vortex formation and flow distribution, considering layer type windings with different geometrical configurations to optimize the cooling [1]. Global simulations of core type transformers have also been the subject of CFD analysis [5-7].

In this work CFD was applied to evaluate the performance of core type power transformers on which the heat removal is achieved either by natural convection (ONAF) or forced directed cooling fluid circulation (ODAF).

## CFD MODEL

Detailed three-dimensional computational grids of a representative section of the power transformer were obtained with the commercial software Gambit 2.3 from Ansys Inc. Two-dimensional axis-symmetric simulations were also conducted as a simplified approach. In order to solve the governing equations of flow and heat transfer, the commercial CFD code Fluent 6.3 from Ansys Inc was used.

### Transformers geometry

A large core type power transformer comprises a magnetic iron core surrounded by co-axial cylindrical windings being the most common the disc type. The discs can be axially separated by spacers between them, originating radial ducts, where the cooling oil can flow. As a way to direct the flow throughout the radial ducts, washers are inserted periodically along the axial direction to impose a zigzag flow pattern through the winding.

In this work, two different power transformer designs were simulated: one unit with ODAF cooling and the other with ONAF cooling.

The power transformer operating in ODAF has 4 windings located, with increasing diameter, as follows: Tertiary (T), Low Voltage (LV), High Voltage (HV) and Regulation (Reg). The power transformer operating in ONAF has a different geometry comprising three windings: Low Voltage (LV), High Voltage (HV) and Regulation (Reg).

## Computational grid

### ODAF cooled transformer

Two geometrical models were generated: two-dimensional (2D) and three-dimensional (3D). The 2D model is axis-symmetric, representing a cutting plane along the axial direction. Although the windings have a cylindrical shape which is nearly axis-symmetric, the hydrodynamic and thermal effects of radial spacers (especially by reduction of cooling area) are not taken into account in the 2D model. The 3D model represents the smallest repetitive section (volume limited by two axial planes, one in the centre of a column of radial spacers and another in the middle of two columns of radial spacers).

A 2D computational grid with 800 000 elements of variable size was created, using a more refined grid in critical zones with the smallest size of 0.7 mm. This grid was subjected to several sensitivity tests and it was found to be the one with the best precision/computational effort ratio.

For the 3D simulation, a tetrahedral/hexahedral hybrid mesh with 2 450 000 elements was created. The mesh size is in average 2.5 mm, although in critical zones, the mesh refinement is 0.7 mm.

### ONAF cooled transformer

For the ONAF geometry, only a 2D computational grid with 535 000 elements and an average size of 0.7 mm was used to simulate the flow and heat transfer inside the power transformer.

## Oil physical properties

Within the operation temperature range of power transformers, variations on the density do not exceed 5%, making appropriate the use of a Boussinesq model for the natural convection flow. This model is of great importance when the oil flow is imposed by natural convection. The temperature dependence of the density naphthenic oil is

$$\rho = 868\left[1 - 6.40\times10^{-4}\left(T - 293\right)\right] \quad (1)$$

where $\rho$ is the oil density ($\mathrm{kg\ m^{-3}}$) at temperature $T(\mathrm{K})$.

The oil viscosity temperature dependence was obtained using a cone/plate rheometer within the temperature range of 20°C to 80°C

$$\mu = 1.43\times10^{-7}\exp\left(\frac{3480}{T}\right) \quad (2)$$

where $\mu$ is the viscosity of the oil in $\mathrm{Pa\cdot s}$ and $T$ is the temperature in K. Within this range of temperatures, viscosity varies by one order of magnitude.

The oil thermal conductivity and heat capacity were assumed constant since its variation is negligible in the studied temperature range: $K_{oil} = 0.126\ \mathrm{W\ m^{-1}\ K^{-1}}$ and $C_p = 2016\ \mathrm{J\ kg^{-1}\ K^{-1}}$, respectively.

## Disc heat conduction model

A simulation comprising the individual insulated conductors inside the disc would require a large computational effort. For this reason, the total disc volume was considered a homogeneous medium where an equivalent thermal resistance is calculated in the radial and axial directions including the contribution of each individual component (copper and insulation paper). For the azimuthal direction (for 3D models), the thermal conductivity was assumed to be equal to the copper thermal conductivity.

To validate this heat conduction model, two different types of conductors were simulated. Type 1 with rectangular bar and type 2 with copper transposed cable. Figure 1 shows the detailed and simplified versions.

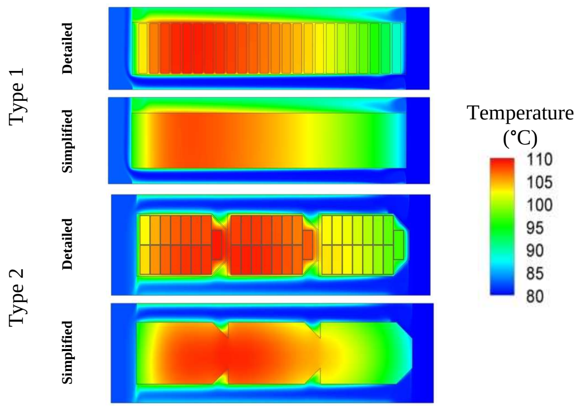



**Figure 1.** Comparison of the thermal behavior of the detailed and simplified disc model for the two types of conductors studied.

Deviations to the average temperature are less than 10% in both types when comparing the real geometry to the simplified version.

## Boundary conditions

### ODAF cooled transformer

In order to solve the governing equations of the flow and heat transfer, boundary conditions must be imposed on the studied domain.

For the 2D model the system has boundary conditions of equal pressure at the inlet and also at the outlet and the pressure difference is set to provide the necessary flow rate of operation. External walls are considered to be adiabatic and uniform energy dissipation sources are imposed to the windings.

For the 3D simulation, the boundary conditions are similar but an azimuthal thermal conductivity has to be considered and the pressure drop required to ensure the flow rate of operation has to be higher due to the presence of the spacers that obstruct the geometry, causing an additional resistance. The boundary conditions for both 2D and 3D simulations are summarized in Table 1.

**Table 1.** Boundary conditions used in the CFD simulations in the ODAF cooled power transformer.

| | |
|---|---|
| **Inlets** | $P_{static}$ = 248 Pa (2D) and $P_{static}$ = 310 Pa (3D)<br>(to ensure experimental flow rate of 37.4 $m^3$/h)<br>$T_{inlet}$ = 80°C |
| **Windings power dissipation** | **HV** 343 708 W/$m^3$ (48 446 W total)<br>**LV** 312 509 W/$m^3$ (31 341 W total)<br>**T** 4 333 W/$m^3$ (90 W total)<br>**REG** 413 100 W/$m^3$ (7 931 W total) |
| **Outlets** | $P_{static}$ = 0 Pa |

## ONAF cooled transformer

For systems where natural convection is the driving force for the oil flow, boundary conditions of pressure have to be corrected, because when the main oil velocities are low (around 1 mm/s), the total pressure is essentially determined by the hydrostatic pressure head. *Fluent* considers this pressure assuming a constant operation density, $\rho_{op}$. However, it is important to consider for these systems that the local oil density changes implicitly with the height, $h$,

$$\rho = \rho_0 \left[1 - \beta\left(T_{inlet} + \alpha h - T_0\right)\right] \qquad (3)$$

where $\alpha$ is the temperature raise slope, $T_{inlet}$ is the temperature at the inlet of the windings, $\beta$ is the oil thermal expansion coefficient and $\rho_0$ is the oil density at the temperature of reference $T_0$. The temperature raise slope was determined experimentally with fibre optic probe measurements.

In order for the model to be physically consistent, the value of the relative pressure drop, $\Delta P_{relative}$, has to be calculated by the integration of the density equation (3) between the inlet position, $h_{inlet}$, and the outlet position $h_{outlet}$,

$$\Delta P_{relative} = \int_{h_{inlet}}^{h_{outlet}} \rho g dh - \rho_{op} g\left(h_{outlet} - h_{inlet}\right). \qquad (4)$$

Apart from the pressure correction, the other boundary conditions are identical to the ODAF cooled transformer and are given in Table 2.

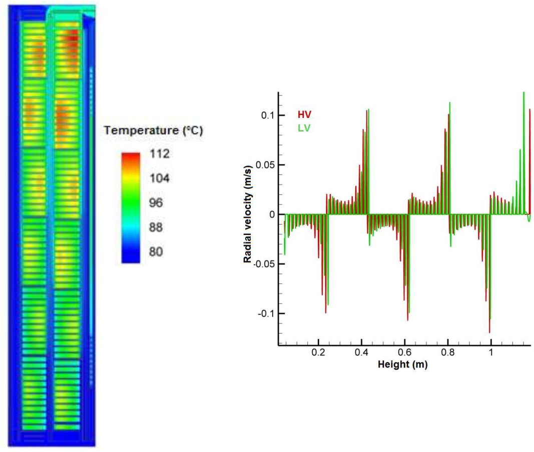
**Table 2.** Boundary conditions used in the CFD simulations in ONAF operating mode.

| | |
|---|---|
| **Inlets** | Pressure will change with height<br>$T_{inlet}$ = 61.6°C |
| **Windings power dissipation** | **HV** 148 213 W/$m^3$ (22 088 W total)<br>**LV** 139 082 W/$m^3$ (13 660 W total)<br>**REG** 206 094 W/$m^3$ (5 063 W total) |
| **Outlets** | Pressure will change with height |

# RESULTS

## ODAF cooled transformer

The results of the CFD simulation are depicted on Figure 2 in terms of the temperature distribution - both in the oil and discs - and the oil velocities along the windings.




(a) (b)

**Figure 2.** Temperature contours (a) and radial velocity (b) for the 2D model.

The oil distributes unevenly in the four windings and flows along the power transformer.

The presence of washers causes a zigzag motion by blocking the axial flow from a section to another and imposes the oil to flow through the radial ducts. This increases the radial velocity and, consequently, the heat transfer rate in the radial ducts.

The position and magnitude of the hot-spot can also be located. For this particular geometry and operating conditions, the hot-spot is localized on the third disc from the top.

Average and hot-spot temperatures in the HV and LV windings are given in Table 3 and are compared with the experimental measurements in order to validate de simulation method.

**Table 3.** Average and hot-spot temperatures in HV and LV windings.

| | $T_{2D}$ (°C) | $T_{3D}$ (°C) | $T_{Experimental}$ (°C) |
|---|---|---|---|
| HV (average) | 99.6 | 102.9 | 106.7 |
| LV (average) | 98.6 | 106.6 | 117.9 |
| HV (hot-spot) | 112.3 | 115.5 | - |
| LV (hot-spot) | 108.7 | 118.4 | - |

Smaller differences between the 3D simulation and the experimental measurements are observed.

The 2D model does not consider the presence of radial and vertical spacers. Consequently, the effective area for heat transfer is higher, leading to an underestimation of the winding temperatures and greater deviations to experiments.

## ONAF cooled transformer

Similar data was computed for the ONAF power transformer. The hot-spot is located on the top of the high voltage winding with a value of 118.1°C. It was observed that the flow inverts its direction between two consecutive washers. In order to validate the simulation, experimental measurements with fibre optic probes were performed afterwards. The probes were positioned to measure the temperatures of the hottest spot inside the disc and the adjacent oil in the radial duct. These are shown in Table 4.

**Table 4.** Measured hot-spot temperatures and the total range of temperatures modelled inside the disc and oil duct from the CFD simulations.

| Fibre optic probe | | $T_{Experimental}$ (°C) | $T_{2D}$ (°C) |
|---|---|---|---|
| Disc | HV | 116 ± 2 | 99.1 - 115.1 |
| | LV | 109 ± 2 | 90.8 - 103.7 |
| Oil | HV | 98 ± 2 | 91.0 - 115.5 |
| | LV | 98 ± 2 | 89.8 - 99.6 |

Deviations to the experiments can be either explained by the use of a simplified heat conduction model or overestimation of the effective area for heat transfer since only a 2D axis-symmetric model is used.

## OPTIMIZATION

Flow inversion was observed to occur, in some operating ONAF cases, between two consecutive washers. The heated oil in a radial duct re-enters the upper radial duct, leading to a local increase of average temperature. This causes the disc to overheat since the driving force for the heat transfer decreases leading to a poor heat removal, and hence a poor performance of the power transformer.

A study was performed to the HV winding in order to overcome this inversion problem and several geometries were tested. By adding new washers and repositioning them, it is possible to align the oil flow between consecutive washers. A decrease of 6.2°C in the hot-spot temperature and 1.7°C in the winding average was achieved by this way as reported in Table 5.

**Table 5.** Comparison of average and hot-spot temperature for the normal and modified geometry.

| Geometry | $T_{HV\ average}$ (°C) | $T_{HV\ maximum}$ (°C) |
|---|---|---|
| Normal | 92.3 | 118.1 |
| Modified | 90.6 | 111.9 |

## CONCLUSIONS

CFD simulations can provide a non-intrusive way of analysing thermal performance of power transformers under in-service conditions. Detailed temperature and flow distribution along the windings can be obtained as well as the location and magnitude of hot-spots.

For the 2D axis-symmetric model, the spacers' effect cannot be taken directly into account in hydrodynamic and thermal simulation. A proposed correction on the temperature was introduced by taking into account the heat transfer area reduction.

The 3D model gives the best estimates for the experimental temperatures. Observed deviations can be caused by several uncertainties like on the estimation of the inlet/outlet temperatures or real differences on geometrical parameters since the CFD simulation assumes a perfect construction.

In ONAF cooling, the appropriate corrections must be made to the boundary conditions for a physically consistent model, in order to obtain realistic results. It was detected flow inversions between two consecutive washers. These flow inversions can be the main source of overheating in ONAF cooling. By acting on the placement of washers, it was demonstrated that it is possible to control potential inversions of the oil and to reduce the hot-spot temperature by as much 10°C which has a strong direct impact on the power transformer's lifetime.

## ACKNOWLEDGMENTS

Carlos M. Fonte gratefully acknowledges the PhD grant from FCT (SFRH/BD/39040/2007).

## REFERENCES

[1] N. El Wakil, N.-C. Chereches, J. Padet, 2006, "Numerical study of heat transfer and fluid flow in a power transformer ", *Int. J. Therm. Sci.* vol. 45, 615-626.

[2] R. Hosseini, M. Nourolahi, G.B. Gharehpetian, 2008, "Determination of OD cooling system parameters based on thermal modeling of power transformer winding", *Simul. Modell. Pract. Theory*, vol. 16, 585-596.

[3] J.-M. Mufuta, E. van den Bulck, 2000, "Modelling of the mixed convection in the windings of a disc-type power transformer," *Appl. Therm. Eng.*, vol. 20, 417-437.

[4] F. Torriano, M. Chaaban, P. Picher, 2010, "Numerical study of parameters affecting the temperature distribution in a disc-type transformer winding" *Appl. Therm. Eng.*, vol. 30, 2034-2044.

[5] J. Smolka, A. Nowak, 2008, "Experimental validation of the coupled fluid flow, heat transfer and electromagnetic numerical model of the medium-power dry-type electrical transformer," *Int. J. Therm. Sci.*, vol. 47, 1393-1410.

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

[7] H. Campelo, C.M. Fonte, R.G. Sousa, M.M. Dias, J.C.B Lopes, R. Lopes, J. Ramos, D. Couto, 2009, "Detailed CFD analysis of ODAF power transformers" *Proceedings of the International Colloquium Transformer Research and Asset Management*, Croatia.