Applied Thermal Engineering 124 (2017) 855–864

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Reduced model for the thermo-fluid dynamic analysis of a power transformer radiator working in ONAF mode

Luciano Garelli [a], Gustavo Ríos Rodriguez [a,*], Mario Storti [a], Daniel Granata [b], Mauro Amadei [b], Marcelo Rossetti [b]

[a] Centro de Investigación de Métodos Computacionales, CIMEC (UNL – CONICET), 3000 Santa Fe, Argentina
[b] Tadeo Czerweny S.A, 2252 Gálvez, Santa Fe, Argentina

HIGHLIGHTS

- Semi-analytical reduced model for ONAF power transformers.
- Forced and natural heat convection equations are coupled to treat fan blown region.
- Complete 3-D radiator convective heat dissipation simulated with LES turbulence model.
- Jointed semi-analytical model/CFD simulations provide an accurate analysis tool.
- Experimental measurements of different key variables on a power workbench.

ARTICLE INFO

Article history:
Received 5 January 2017
Revised 13 June 2017
Accepted 17 June 2017
Available online 19 June 2017

Keywords:
Power transformer
Computational Fluid Dynamics
Experimental measurements
Heat dissipation
ONAF

ABSTRACT

This work introduces an extension of the semi-analytical reduced model presented in Ríos Rodriguez et al. (2016), with the aim of analyzing the thermo-fluid dynamic behavior of a power transformer radiator working in ONAF (Oil Natural Air Forced) mode, considering vertical blowing of the fans. The objective of this research paper is the development of a numerical tool to be used during the design process of power transformers in order to obtain more detailed information about the radiators performance, either they are working in ONAF or ONAN (Oil Natural Air Natural) mode. Since it is observed both in the experimental measurements and in the numerical simulations that not all the radiator panels are blown by a single fan, the reduced model here presented considers a mixed situation wherein the heat is removed by natural convection in a fraction of the radiator while in the rest the heat is removed by forced convection. As a consequence, besides adding to the set of conservation equations introduced in Ríos Rodriguez et al. (2016) those corresponding to the momentum and energy balance for considering forced heat convection transfer, there appears the need of introducing additional equations for coupling the natural and forced heat convection transfer models. The reduced model developed in this work is applied to estimate different characteristic parameters of a power transformer radiator working in ONAF mode, like the oil flow rate, dissipated power, outlet and inlet oil temperatures, among others. The computed results are compared to those obtained by CFD simulations and experimental measurements carried out on an ad-hoc workbench for validation purposes. It is found that the reduced model reproduces with acceptable accuracy the values of the most important design variables at a very low computation cost. In this manner, it can be considered as a reliable and valuable tool in the design of power transformers, allowing to carry out parametric studies.

© 2017 Published by Elsevier Ltd.

1. Introduction

It is known that several sources of heat generation (losses) exists in power transformers ([2]), i.e. hysteresis and eddy currents among others, but the most important comes from the cooper (resistive) losses caused by the main currents due to Joule heating or $I^2 \cdot R$, where I is the current intensity and R the winding resistance. Wherewith, for a given current intensity and environmental conditions the radiators must be able to dissipate the generated heat without exceeding the permissible temperature limits imposed in the certification process of the machine. If this limits are exceeded, damage in the insulation may occur thus reducing the operative life of the transformer. One of the most common

* Corresponding author.
E-mail address: gusadrr@santafe-conicet.gov.ar (G. Ríos Rodriguez).

http://dx.doi.org/10.1016/j.applthermaleng.2017.06.098
1359-4311/© 2017 Published by Elsevier Ltd.

ways to enhance the heat dissipation is to use fans for increasing the air flow between the radiator fins or panels, providing better cooling capacity than that supplied by natural convection. The fans are mounted near the radiators and when the temperature exceeds a prescribed limit they are turned on. Also, under the same environmental conditions a power transformer working in ONAF mode can be used under a higher current load than if it were working in ONAN, without exceeding the design temperature limits. This increase in the transformer load can arise from particular situations like the failure of another transformer, the increase in the current consumption due to hot weather conditions, etc. In this context, several studies have been conducted in the last years with the aim of evaluating and improving the cooling capacity of power transformers working in ONAF mode.

In [3] it is presented a numerical prediction and experimental studies about the cooling capacity of an oil-filled power transformer working both in ONAN and OFAN (Oil Forced Air Natural), considering that the excessive heat generation is the major cause of insulation deterioration. The experimental results show a cooling capacity increase of 7% when the oil flow rate is four times that in the ONAN mode, whereas the CFD (Computational Fluid Dynamics) results show an increase of 17%. These results are also in agreement with those presented in [4], where it is mentioned that the application of forced oil flow is only feasible in some justifiable conditions and only if it is combined with an efficient convective heat transfer in the radiators. In order to improve the cooling capacity, it is also important to consider the blowing direction of the fans. This is studied in detail in [5], where numerical and experimental analysis are carried out both for horizontal and vertical blowing directions. Therein, it is found for a specific transformer, that the horizontal blowing direction has higher efficiency than the vertical one due to less sideways leakage of the air. The cooling capacity increases from 19.7 [Kw] in ONAN to 65–85 [Kw] in ONAF, depending on the configuration and blowing direction of the fans. More recently, [6] studied both the internal oil flow and the external air flow using a coupled thermo-fluid dynamic model for numerical simulations as well as experimental data. They consider for the analysis a power transformer with a five radiator arrangement, each radiator fitted with 27 fins of 2.5 [m] height and 0.52 [m] width, with a spacing of 0.05 [m] between fins. The oil inlet conditions (mass flow rate and temperature) are obtained experimentally and different (vertical and horizontal) blowing directions are studied. The numerical simulation of each coupled thermo-hydraulic problem requires approximately 500 h ($\sim$21 days) on eight parallel processors. It is found, in the case of vertical blowing, that the central radiator dissipates 15% less heat compared to the end radiators, whereas in horizontal blowing there is a monotonic decrease in heat dissipation from the first towards the last radiator. For that configuration, it is concluded that horizontal blowing helps to dissipate 6.1 % more heat than vertical blowing [6].

As can be seen, an important effort has been put into studying the operation conditions of radiators in ONAF mode using CFD simulations and experimental measurements. CFD simulations based on the Finite Volume Method (FVM) or the Finite Element Method (FEM) have shown to be reliable and accurate tools to describe the behavior of the radiator and the windings for different working conditions [3,5–11]. A detailed description of the temperature distribution on each radiator panel, as well as the air and oil flows, can be obtained. However, these simulations are usually very expensive from the computational point of view and they also need accurate information from experimental measurements in order to properly set the initial/ boundary conditions, like the inlet oil mass flow rate and the inlet oil temperature. Also, the simulation of the coupled thermo-hydrodynamics of the complete transformer (oil tank, windings, radiators, etc.) is a very difficult task when considering the processes of building the geometric model with all its details and thereafter the mesh generation. For this reason, in this research article a reduced model is presented with the objective of being able to evaluate the performance of a power transformer during the design and development stages without requiring the generation and modification of complex FVM or FEM models, which can be generated later on an advanced preliminary design.

In this work an extension of the semi-analytical reduced model presented in [1] is introduced, with the aim of providing a low cost computational tool able to analyze the coupled thermo-fluid dynamic behavior of a power transformer radiator working in ONAF mode and considering vertical blowing of the fans. The objective of the reduced model is to be used as a numerical tool during the design process of a power transformer, supplying crucial information of global design parameters at a relatively low cost, thus allowing to test different working conditions and configurations which can also be used as input data for subsequent CFD analysis. The paper presents a thermal and fluid flow analysis of a 30 MVA – 132/34.5/13.8 kV power transformer radiator manufactured by Tadeo Czerweny S.A. company. The study is mainly focused on the heat dissipated within the radiators assuming that the power transformer is working in ONAF mode. The analysis is carried out by first considering analytical calculations, then CFD simulations and finally experimental data acquired from measurements using an ad-hoc transformer workbench. As in [1], the computation of various flow and thermal characteristic parameters of the cooling circuit is addressed by means of an extended reduced model. The solution of this coupled system of equations delivers information about several variables (oil temperature at the inlet and outlet ducts of the radiator, the oil flow rate, the air temperature and its mean velocity, among others).

The corresponding CFD simulations are presented following the guidelines introduced in [1]. Turbulence of the air flow in the radiator is taken into account in the simulations using a Large Eddy Simulation (LES) model. The dissipated heat, the oil flow rate and the air velocity distribution along the radiator are compared to those experimentally measured.

The experimental measurements are carried out on a workbench specifically dedicated to this end at Tadeo Czerweny S.A. company (a detailed description can be seen in [1]). This workbench enables to validate the calculated values by performing measurements in controlled conditions. Temperature distribution on the fin surface, oil temperature at the inlet and outlet headers of the radiator, oil flow rate as well as air temperature and velocity distributions are measured. The encountered differences among the reduced model, the CFD simulation and the experimental results are discussed. Finally, future research work is briefly described with the aim of improving the heat dissipation in the radiators.

2. Extended reduced model – analytical description

The reduced model here presented is an extension of that introduced in [1] for the ONAN case. The new model allows to calculate the cooling capacity of a radiator working in both ONAN and ONAF operative conditions. It is based on a set of coupled analytical equations of momentum and energy balance for the oil running through the radiator channels, for the air flowing between the radiator panels and also for the heat being conducted through the surface of the panels. In some instances, when a radiator is working in ONAF, and depending on its size, it is observed that not all the fins are being blown by the fan [6]. A certain number of radiator fins are dissipating heat in forced convection condition while the rest are working in natural convection. As a consequence, the extended reduced model introduced in this work will consider both types of heat convection models and will couple them with a suitable set of additional equations. First, for the sake of clarity, we consider each convection condition separately.

2.1. Natural convection

A set of five coupled equations are required in the case of natural convection to obtain the oil flow rate (Q_{oil_N}), the inlet ($T_{oil_{2N}}$) and outlet ($T_{oil_{1N}}$) oil temperatures, the outlet ($T_{air_{2N}}$) air temperature and the air mean velocity (U_{air}) for a given power (P) to be dissipated. This set of equations can be expressed as residuals in the following manner

- Oil momentum balance

$$R_{mom,oil}(T_{oil_{1N}}, T_{oil_{2N}}, Q_{oil_N}) = 0 \tag{1}$$

- Oil energy balance

$$R_{ener,oil}(T_{oil_{1N}}, T_{oil_{2N}}, Q_{oil_N}) = 0 \tag{2}$$

- Air momentum balance

$$R_{mom,air}(T_{air_{2N}}, U_{air}) = 0 \tag{3}$$

- Air energy balance

$$R_{ener,air}(T_{air_{2N}}, U_{air}) = 0 \tag{4}$$

- Energy balance between the oil and the air

$$R_{hex}(T_{oil_{1N}}, T_{oil_{2N}}, T_{air_{2N}}, U_{air}, Q_{oil_N}) = 0 \tag{5}$$

The derivation of the previous equations is described next.

2.1.1. Oil momentum balance

The driving force that moves the oil through the cooling circuit is the result of a change in the oil density ρ_{oil}, which decreases with the increase of the temperature, and generates a pressure difference (Δp_{oil}) that is balanced with the radiator flow resistance. Thereby it can be stated that

$$\begin{aligned}\Delta p_{oil} &= (\Delta\rho)_{oil} g \Delta z,\\ &= (\beta_{oil}\rho_{oil}\Delta T_{oil}) g \Delta z,\end{aligned} \tag{6}$$

where β_{oil} is the oil thermal expansion coefficient, ΔT_{oil} is the temperature difference between the inlet and outlet of the radiator ($T_{oil_2} - T_{oil_1}$), g is the gravitational acceleration and Δz is defined in [4] as

$$\begin{aligned}\Delta z &= (h_1 + h_2),\\ h_2 &= L_p\left(\frac{\Delta T_{oai}}{\Delta T_{oil}} - \frac{\Delta T_{LMTD}}{\Delta T_{oil}} - 0.5\right),\end{aligned} \tag{7}$$

where L_p is the radiator height, h_1 is the difference between the middle level of the radiator and that of the winding, $\Delta T_{oai} = T_{oil_2} - T_{amb}$ is the oil-ambient air temperature difference at the radiator inlet, T_{amb} the air temperature far enough of the radiator and ΔT_{LMTD} is the logarithmic-mean temperature difference between oil and air in the radiator.

The oil flow rate due to the pressure difference given by Eq. (6) will be inversely proportional to the hydraulic resistance of the oil circuit. The channels of the radiator are modeled as rectangles of width w_{oil} and height L_y using the equation derived by [24] for an arbitrary shape

$$Q_{oil} = -\frac{dp}{dz}\frac{1}{\mu_{oil}}\frac{32A_{cha}^3}{\xi Pe_{cha}^2} = -\frac{\Delta p_{oil}}{L_p}\frac{1}{\mu_{oil}}\frac{32A_{cha}^3}{\xi Pe_{cha}^2} \tag{8}$$

where μ_{oil} is the oil dynamic viscosity, A_{cha} is the channel area, Pe_{cha} is the channel perimeter and ξ is a non-dimensional constant that depends only on the shape of the channel section. The aspect ratio of the rectangular oil channel is $w_{oil}/L_y = 0.07$ and from [24] it can be obtained $\xi \approx 88$.

For a radiator with N_p panels and N_{cha} channels by panel, the overall passage area A is calculated as $A = A_{cha}N_{cha}N_p$ and a similar calculation can be done for the perimeter Pe. Finally, combining Eqs. (6) and (8), it can be obtained the following equation

$$Q_{oil}/A - \frac{(\beta_{oil}\rho_{oil}\Delta T_{oil}) g \Delta z 32A_{cha}^2}{L_p \mu_{oil}\xi Pe_{cha}^2} = 0 \tag{9}$$

which can be written in the form

$$R_{mom,oil}(T_{oil_1}, T_{oil_2}, Q_{oil}) = 0 \tag{10}$$

2.1.2. Oil heat transfer

For a given power (P) to be dissipated by the radiator to the surrounding air, an energy balance in the oil can be established

$$\begin{aligned}P &= (\rho C_p Q \Delta T)_{oil},\\ &= F_{oil}\Delta T_{oil},\end{aligned} \tag{11}$$

where $F = \rho C_p Q$ is the calorific capacity of the flow (W/K). Eq. (11) can be written as

$$F_{oil}\Delta T_{oil} - P = 0 \tag{12}$$

or

$$R_{ener,oil}(T_{oil_1}, T_{oil_2}, Q_{oil}) = 0. \tag{13}$$

2.1.3. Air momentum balance

If the radiator is working in ONAN or the radiator panels are not blown by the fan, it can be assumed that the air flows by natural convection. Assuming that the flow between the faces of two panels is fully developed, the Dean's [25,26] correlation can be used. Using this correlation, the friction coefficient (C_f) is defined as follows

$$C_f = \frac{\tau_w}{0.5\rho_{air}U_{air}^2} = 0.073 Re_{air}^{-0.25} \tag{14}$$

where ρ_{air} is the air density and τ_w is the shear stress on the wall surface. The Reynolds number (Re_{air}) is given by

$$Re_{air} = \frac{U_{air}(w_{air}/2)}{\nu_{air}} \tag{15}$$

The air flow rate through the radiator (Q_{air}) can be expressed like

$$Q_{air} = U_{air} w_{air} W_p N_p \tag{16}$$

and it is proportional to the spacing between panels (w_{air}), the panels width (W_p) and the numbers of panels (N_p). On the other hand, the momentum balance of the air establishes that the buoyancy force in the air channel (Δp_{air}) has to be balanced with the wall friction, thereby

$$\begin{aligned}\Delta p_{air} W_p w_{air} &= 2\tau_w L_p W_p,\\ \Delta p_{air} &= 2\tau_w L_p / w_{air},\end{aligned} \tag{17}$$

The buoyancy force in the air channel can also be written as

$$\begin{aligned}\Delta p_{air} &= \Delta\rho_{air} g L_p,\\ &= \frac{1}{2}\beta_{air}\rho_{air}(T_{air_2} - T_{air_1}) g L_p,\\ &= \frac{1}{2}(\beta\rho\Delta T)_{air} g L_p,\end{aligned} \tag{18}$$

where β_{air} is the thermal expansion coefficient of the air. The momentum balance for the air flow rate between the panels can finally be expressed by the following equation

$$2\tau_w(U_{air})L_p/w_{air} - \frac{1}{2}(\beta\rho\Delta T)_{air} g L_p = 0 \tag{19}$$

or

$$R_{mom,air}(T_{air_2}, U_{air}) = 0, \tag{20}$$

2.1.4. Air heat transfer

In a steady state condition, the heat absorbed by the air has to be equal to the power (P) generated in the winding

$$\begin{aligned} P &= (\rho C_p Q \Delta T)_{air}, \\ &= F_{air} \Delta T_{air}, \end{aligned} \tag{21}$$

where $F_{air} = (\rho C_p Q)_{air}$. This equation can be written in the form

$$F_{air} \Delta T_{air} - P = 0. \tag{22}$$

or

$$R_{ener,air}(T_{air_2}, U_{air}) = 0. \tag{23}$$

2.1.5. Heat transfer between oil-air

To evaluate the heat transfer from the oil to the air ($H_{oil \to air}$), it is assumed that in each panel

$$H_{oil \to air} = h(T_{oil}(z) - T_{air}(z)) \tag{24}$$

where z is the vertical coordinate along the panel and h is the heat transfer coefficient between the oil and the air, and it includes the conductivity of the oil, the steel of the panel and the air. Therefore, this coefficient can be written as

$$h^{-1} = h_{oil}^{-1} + h_{steel}^{-1} + h_{air}^{-1} \tag{25}$$

The heat transfer coefficient in the oil channel is computed as follows, considering the convective model proposed in [4]

$$\begin{aligned} h_{oil} &= Nu_{oil}\, k_{oil}/D_h, \\ Pr_{oil} &= \mu_{oil} Cp_{oil}/k_{oil}, \\ Re_{oil} &= \rho_{oil} U_{oil} D_h/\mu_{oil}, \\ Gr_{oil} &= \frac{g\beta_{oil}\Delta T_{os} D_h^3}{(\mu_{oil}/\rho_{oil})^2}, \\ Nu_{oil} &= 0.85\left(0.74 Re_{oil}^{0.2} (Pr_{oil} Gr_{oil})^{0.1} Pr_{oil}^{0.2}\right); \end{aligned} \tag{26}$$

where $D_h = 4A_{chan}/Pe_{chan}$ is the hydraulic diameter of the oil channel, ΔT_{os} is the surface temperature drop from the oil to the cooler wall, k_{oil} is the thermal conductivity of the oil, Pr_{oil}, Gr_{oil}, Re_{oil} and Nu_{oil} are the Prandtl, Grashof, Reynolds and Nusselt numbers for the oil flow in the channel.

On the other hand, the heat transfer in the steel is due to conduction, thereby the corresponding coefficient can be calculated like

$$h_{steel} = \frac{k_{steel}}{t}, \tag{27}$$

where k_{steel} is the thermal conductivity of the steel and t is the thickness of the steel sheet of the panel.

The convective heat transfer of the panel to the air can be modeled by considering a vertical parallel flat plate. Using an empirical equation proposed by [12]

$$\begin{aligned} h_{air} &= Nu\, k_{air}/L_p, \\ Pr_{air} &= \mu_{air} Cp_{air}/k_{air}, \\ Gr_{air} &= \frac{g\beta_{air}|\overline{T_{oil}} - \overline{T_{air}}|L_p^3}{(\mu_{air}/\rho_{air})^2}, \\ Ra_{air} &= Gr_{air}\, Pr_{air}, \\ Nu_{air} &= \left(0.825 + \frac{0.387 Ra_{air}^{1/6}}{\left(1 + (0.492/Pr_{air})^{(9/16)}\right)^{(8/27)}}\right)^2; \end{aligned} \tag{28}$$

where $Pr_{air}, Gr_{air}, Ra_{air}$ and Nu_{air} are the Prandtl, Grashof, Rayleigh and Nusselt numbers for the air, respectively. $\overline{T_{oil}}$ and $\overline{T_{air}}$ are the mean temperatures of the oil and the air, k_{air} is the air thermal conductivity and Cp_{air} is the air constant pressure specific heat.

Finally, the energy balance in the panel can be stated as follows

$$\frac{d}{dz}(F_{air} T_{air}) = h(T_{oil} - T_{air}) \frac{d}{dz}(F_{oil} T_{oil}) = h(T_{oil} - T_{air}), \tag{29}$$

therefore

$$\begin{aligned} \frac{du}{dz} &= -\gamma(T_{oil} - T_{air}), \\ u &= T_{oil} - T_{air}, \\ \gamma &= h\left(\frac{1}{F_{air}} - \frac{1}{F_{oil}}\right) \end{aligned} \tag{30}$$

The solution of Eq. (30) is

$$u = u_1 e^{-\gamma z} \tag{31}$$

$$T_{oil_2} - T_{air_2} = (T_{oil_1} - T_{air_1}) e^{-\gamma L_p} \tag{32}$$

being the last equation in the system

$$T_{oil_2} - T_{air_2} - (T_{oil_1} - T_{air_1}) e^{-\gamma L_p} = 0 \tag{33}$$

which can also be expressed like

$$R_{hex}(T_{oil_1}, T_{oil_2}, T_{air_2}, U_{air}, Q_{oil}) = 0. \tag{34}$$

2.2. Forced convection

If the heat dissipation in forced convection condition is considered, the resulting system of equations is reduced to four since the air momentum balance equation, Eq. (20), is removed. This is so because the air velocity (U_{air}) is a prescribed value for the ONAF model. Then, the following set of equations allows to obtain the oil flow rate (Q_{oil_F}), the inlet ($T_{oil_{2F}}$) and outlet ($T_{oil_{1F}}$) oil temperatures and the outlet ($T_{air_{2F}}$) air temperature for a given power (P) to be dissipated

- Oil momentum balance
$$R_{mom,oil}(T_{oil_{1F}}, T_{oil_{2F}}, Q_{oil_F}) = 0 \tag{35}$$
- Oil heat transfer
$$R_{ener,oil}(T_{oil_{1F}}, T_{oil_{2F}}, Q_{oil_F}) = 0 \tag{36}$$
- Air heat transfer
$$R_{ener,air}(T_{air_{2F}}) = 0 \tag{37}$$
- Heat transfer between oil-air
$$R_{hex}(T_{oil_{1F}}, T_{oil_{2F}}, T_{air_{2F}}, Q_{oil_F}) = 0 \tag{38}$$

Also, another correlation equation to that used for the ONAN model, i.e. Eq. (28), has to be introduced in order to estimate the heat transfer coefficient for the fins working in forced convection. For the radiator which is analyzed in this work, an average air velocity $\bar{U}_{air_F} \approx 3.27[\text{m/s}]$ is adopted for the region blown by the fan, which is calculated as the average of the vertical air velocities measured in that region during the experiments (see Section 4). This air velocity is an input data in the reduced model and it depends on the technical characteristics of the fan. Then, the average Reynolds number based on the length of the panel (L_p) is given by

$$Re_{air_{L_p}} = \frac{\rho_{air} \cdot \bar{U}_{air_F} \cdot L_p}{\mu_{air}} \approx 3.84 \cdot 10^5, \tag{39}$$

for $L_p = 1.8$ [m], $\rho_{air} = 1.1949$ [kg/m^3] and $\mu_{air} = 1.83 \cdot 10^{-5}$ [Pa s]. In [12,13] it is presented the following correlation for a mixed boundary layer (initially laminar followed by a turbulent boundary layer), where the average Nusselt number is written as

$$\overline{Nu}_{air_F} = 2 \cdot \frac{0.3387 \cdot Re_{air_{Lp}}^{1/2} \cdot Pr_{air}^{1/3}}{\left(1 + (0.0468/Pr_{air})^{2/3}\right)^{1/4}}, \tag{40}$$

and Pr_{air} is the air Prandtl number.

2.3. Coupling natural and forced heat convection

If it is considered that mixed heat convection conditions exist in the complete radiator, wherein N_1 fins work in natural convection and N_2 fins work in forced convection, as indicated by the respective pink and cyan zones in the radiator of Fig. 1, the sets of equations corresponding to the natural and forced heat convection have to be coupled with a proper set of additional equations under the following assumptions. First, the total dissipated power must be the sum of the heat dissipated by the panels working in natural (P_N) and forced (P_F) convection

$$P = P_N + P_F \tag{41}$$

Also, the inlet oil temperature is assumed to be the same for all the fins, because they share the same pipe located at the top of the radiator and the temperature variation is small along the header [10], wherewith

$$T_{oil_2} = T_{oil_{2N}} = T_{oil_{2F}} \tag{42}$$

Another condition to be satisfied is that the overall oil mass flow rate should be the sum of the mass flow rates of both parts

$$Q_{oil} = Q_{oil_N} + Q_{oil_F} \tag{43}$$

and finally, it is considered that the outlet oil temperature is a weighted average of the outlet oil temperatures computed with the natural and forced convection models separately

$$T_{oil_1} = \frac{T_{oil_{1N}} \cdot Q_{oil_N} + T_{oil_{1F}} \cdot Q_{oil_F}}{Q_{oil}} \tag{44}$$

Under the previous constraints, the natural and forced convection subsystems are coupled and a set of nine non-linear equations with nine unknowns has to be solved. Finally, the extended reduced model considers the air properties to depend on the temperature. These properties are taken from [14] and they are interpolated using cubic splines [15].

2.4. Reduced model evaluation

In order to evaluate the reduced model proposed in this work, a radiator manufactured by Tadeo Czerweny Company for its power transformers is analyzed. The radiator has $N_p = 26$ panels and it is considered that 14 panels (N_1) are working in forced convection and the other 12 panels (N_2) are dissipating heat in natural convection condition. Each panel is assumed to have rectangular shape, with equivalents length $L_p = 1.67$ [m] and width $W_p = 0.521$ [m], in order to have the same total wet area $A_p = 1.74$ [m^2] of the real panel. Each panel has six oil channels ($N_{cha} = 6$) with cross sectional area $A_{cha} = 3.248e - 4$ [m^2] each one. The gap between the panels is $w_{air} = 0.045$ [m] and the distance between the centerline of the radiator and that of the winding is $h_1 = 0.45$ [m]. The radiator is made of carbon steel with thermal conductivity $k_{steel} = 54$ [W/(m K)] and thickness $t = 1.25e - 3$ [m]. The thermal conductivity (k_{oil}), density (ρ_{oil}) and specific heat ($C_{p_{oil}}$) of the oil are calculated using the following approximation presented in [3]

$$\begin{aligned} k_{oil} &= 0.15217 - 7.16e - 5T_{oil_2}, \text{[W/(m K)]} \\ \rho_{oil} &= 1067.75 - 0.6376T_{oil_2}, \text{[Kg/m}^3\text{]} \\ C_{p_{oil}} &= 821.19 + 3.563T_{oil_2}, \text{[J/(Kg K)]} \end{aligned} \tag{45}$$

The oil kinematic viscosity and its thermal expansion coefficient are taken from [4] and they are interpolated using cubic splines. The temperature of the surrounding air is assumed to be $T_{air_1} = 289$ [K]. On the other hand, it is found in the experiments that the power dissipated by a single radiator in ONAF is approximately $P = 14.9$ [kW] (see Section 4). Solving the set of coupled non-linear equations with the above parameters the results shown in Table 1 are obtained.

The first column (Natural) refers to the results computed with the coupled reduced model for the radiator part that is dissipating heat in natural convection condition, the second column (Forced) refers to for the part of the radiator which is working in forced convection condition and the third column (Global) shows the results for the complete radiator.

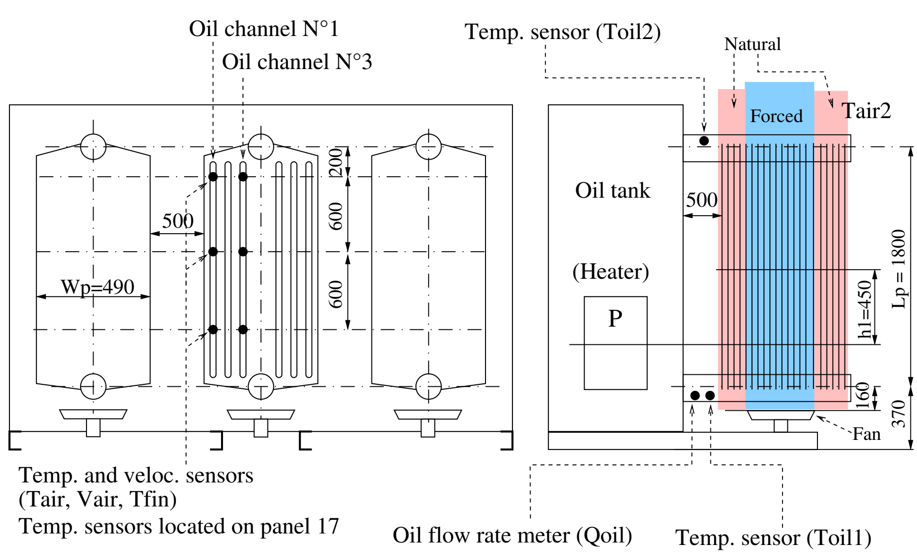

Fig. 1. Schematic description of the experimental workbench and setup installation for ONAF flow tests (lenghts are in mm and out of scale).

Table 1
Characteristic parameters of the radiator computed with the ONAF reduced model.

	Natural	Forced	Global
T_{oil_2} [K]	328.75	328.75	328.75
T_{oil_1} [K]	314.1	308.25	310.35
T_{air_2} [K]	302.65	297.05	–
Q_{oil} [lts/min]	10.4	18.1	28.5
P [kW]	4.34	10.56	14.9
U_{air} [m/s]	0.93	3.27	–
h [W/(m² K)]	7.52	16.1	–

3. Three-dimensional simulation of the radiator in ONAF mode

The CFD simulation is carried out with the HPC CFD Code Saturne ([16,17]) by solving the incompressible Navier-Stokes equations using a Finite Volume Method (FVM) [18,19] with a transient analysis. A 3-D segregated solver is used with a SIMPLEC (Semi-Implicit Method for Pressure-Linked Equations Consistent) algorithm for the coupling between velocity and pressure [20] is used. A Second Order Linear Upwind (SOLU) method is considered for the spatial discretization and a second order scheme is used for the temporal integration. Since the heat transfer is very sensitive to the turbulence level developed within the air channels between the radiator panels and also to the thickness of the boundary layer [12], special care is taken to generate the finite volume mesh in such a way as to use Large Eddy Simulation [21] with a Smagorinsky's [22] eddy-viscosity using the Germano's model [23]. Due to this fact and also to the geometrical complexity of the radiator, an hybrid structured - unstructured mesh with millions of cells is required for the discretization of the problem domain. As a consequence, the simulation is carried out using the computer cluster "Seshat" from CIMEC, equipped with an Intel®Xeon®CPU E5-2640 v2 @2.00 GHz server and 69 Intel®Xeon®CPU E5-1620 v2 @ 3.7 GHz computing nodes with 16 GB RAM Micron®DDR3 1600 MHz, interconnected with an Infiniband® network.

A Boussinesq approximation is used to compute the driving force due to the temperature dependency of the density. The physical properties of the air are those used in the reduced model. As a result of the numerical simulation the total dissipated heat, the air velocity, the air temperature between the panels and the global and local heat transfer coefficients are obtained. In order to reduce the computational cost of the simulation avoiding the solution of a conjugated 3-D heat transfer problem for the complete radiator, the temperature distribution computed on a single radiator panel with the coupled 3-D thermo-fluid dynamic model presented in [1] is used for setting that on each radiator panel.

The mesh is generated assuming that the panels have no thickness and the radiator is placed in a domain of size $\Delta x = 1.25$ [m], $\Delta y = 3.0$ [m] and $\Delta z = 2.0$ [m]. The radiator and the fan relative positions with respect to the oil tank and the ground are the same than those existing in the experimental facility (see Fig. 1). To attain a good quality mesh the domain is discretized with 6.5 Mcell hybrid mesh composed of tetrahedrons, hexas and pyramids. The mesh size exponentially grows from the wall to the bulk of the flow, with a mesh size at the wall equal to $h_{wall} = 0.25$ [mm]. The time step size used in the simulations is $\Delta t = 1.25e-3$ [s] and 10^4 time steps were required to reach a steady heat transfer condition.

A cut of the mesh on a symmetry plane of the radiator is shown in Fig. 2. The top and lateral boundaries of the fluid domain are set as free inlet/outlet (opening boundary condition), allowing the air to escape through these regions. On the other hand, a wall boundary condition is set at the bottom to take into account the presence of the floor.

The fan is modeled as a boundary region wherein it is allowed to set up the experimental fan curve as input through a polynomial with pressure curve coefficients which relate the pressure jump and velocity across the fan boundary. This kind of model predicts the global flow rate impelled by the fan but does not provide a detailed description of the flow through the fan blades. Additional

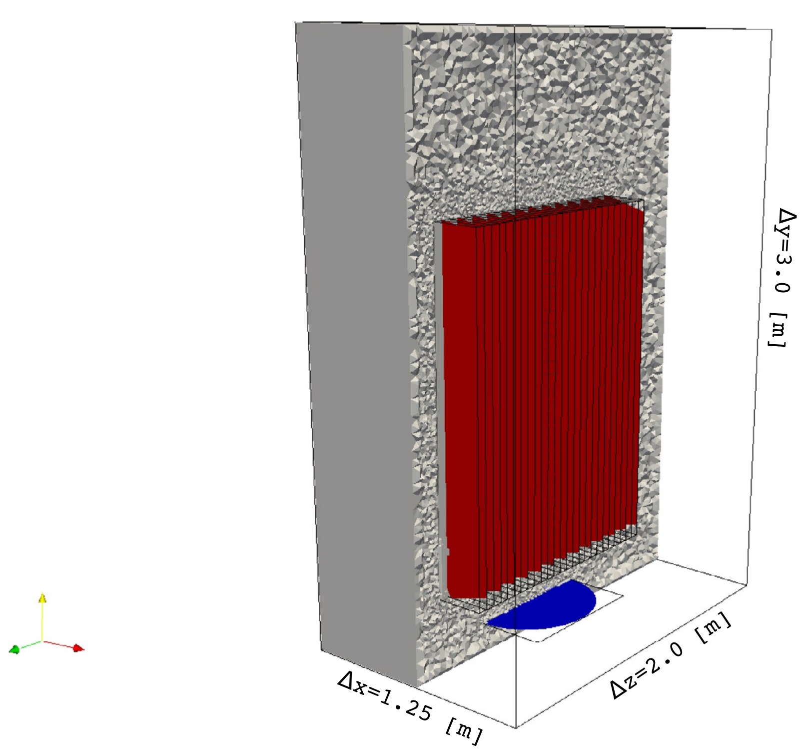

Fig. 2. 3D domain for the complete radiator simulation and mesh detail around the radiator and between the panels. Fan boundary (blue), radiator panels (red) and fluid domain (grey). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

parameters to be provided are the fan and hub radius. The pressure jump Δp across the fan can be expressed as

$$\Delta p = \sum_{n=1}^{N} C_n v^{n-1}, \tag{46}$$

where C_n are the pressure curve coefficients and v^{n-1} are the magnitudes of the local air velocity. This curve is provided by the fan manufacturer and it is fitted with a fourth degree polynomial (See Fig. 3). The characteristics of the fan are: rotation speed $\Omega = 900$ [RPM], fan radius $R = 350$ [mm] and hub radius $H_r = 71$ [mm].

The time averaged dissipated heat computed during the last 600 time steps is $P_{CFD} = 13.4$ [Kw] with a standard deviation of 0.2 [Kw]. In Fig. 4 it is shown a time averaged distribution of the heat transfer on each panel. It can be clearly observed the influence of the fan only in the middle section of the radiator, but it almost has no effect on the panels outside of that region. Only 14 panels are effectively blown by the fan. This conclusion will be also confirmed with the experimental measurements shown in Section 4. The power dissipated by the 12 panels working in ONAN condition is approximately 4.8 [Kw] and the power dissipated by the 14 panels blown by the fan is 8.6 [Kw].

The increase in both the heat transfer coefficient and the total dissipated power are due to an increase in the air velocity, as it is shown in Fig. 5. At the outlet of the fan the magnitude of the velocity is approximately 4.0 [m/s] and the average air speed at the outlet of the radiator is about 2.6 [m/s]. This average velocity is slightly smaller than that experimentally measured.

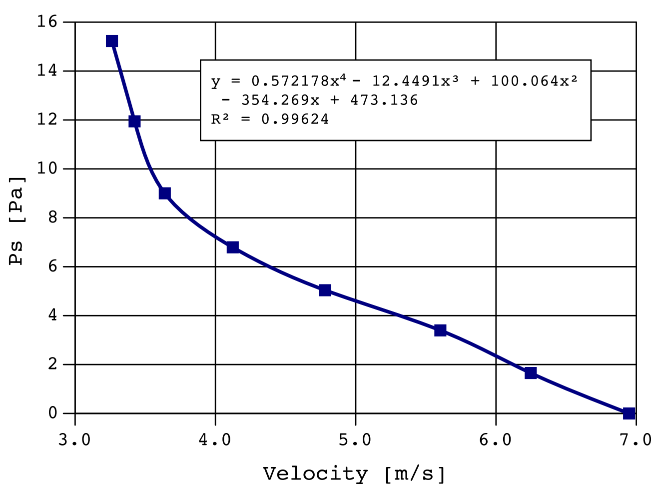

Fig. 3. Fan curve for rotation speed $\Omega = 900$ RPM.

Plotting the vertical air velocity at 200[mm] from the center of the top header it is also possible to determine the influence of the fan. In Fig. 6 it can be seen the time-averaged mean velocity between the panels (air channels) with square markers and the time-averaged minimum and maximum values in red for the CFD simulations. It is evident that the blown fins are those located right above the fan (see Fig. 7).

Also, it is important to compare the air temperature distribution between the radiator panels. From the reduced model the average air temperature in the channels not blown by the fan is 302.65 [K] and that obtained from the CFD simulation is 300.5 [K]. The reduced model estimates a temperature of 297.05 [K] for the blown panels, whereas the CFD simulation value is 293.7 [K]. Although the CFD simulation provides a detailed description of the air temperature inside each air channel, the reduced model predicts very well the average temperature at a very low computational cost, which is helpful during the design process.

Finally, Table 2 summarizes the results already mentioned for a better comparison with those computed with the reduced model (Table 1) and measured in the experiments (Table 3).

4. Experimental setup and measurements

The experimental measurements were realized on the workbench provided by Tadeo Czerweny, which is schematically shown in Fig. 1. This workbench is equipped with three heaters, a 3000 Liters oil tank and three radiators, each one fitted with 26 panels of 1800 [mm] height and 490 [mm] width. More details about the workbench are described in [1]. For the ONAF test three fans were installed at 160 [mm] under the center of the radiators bottom header. The fans have no casing and the rotation speed is 900 [RPM]. The dimensions and technical specifications of the fans are those described in Section 3.

K-type thermocouples are installed at the top and bottom headers of the central radiator, halfway between the radiator and the oil tank (see Fig. 1), with the aim of measuring the oil temperature. The thermocouples have an accuracy of 2.2 °C or 0.75% for the temperature range > 0 °C to 1250 °C. On the other hand, the oil flow rate is registered at the return header, after the oil has cooled in the radiator. Finally, the air speed and temperature are measured at 200 [mm] from the center of the top header, widthwise of the

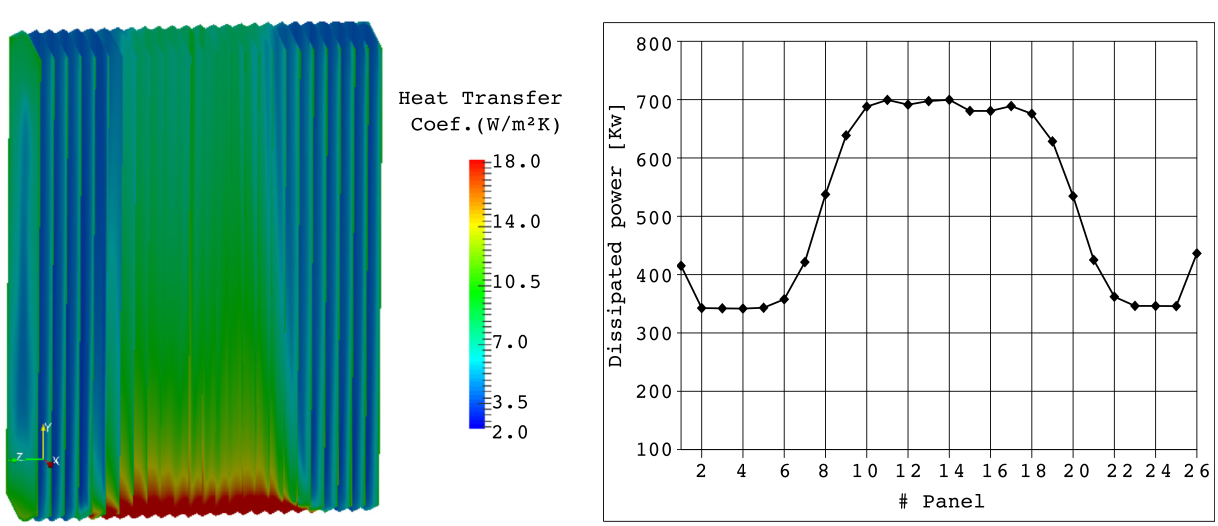

Fig. 4. Time averaged heat transfer coefficient and dissipated power per panel.

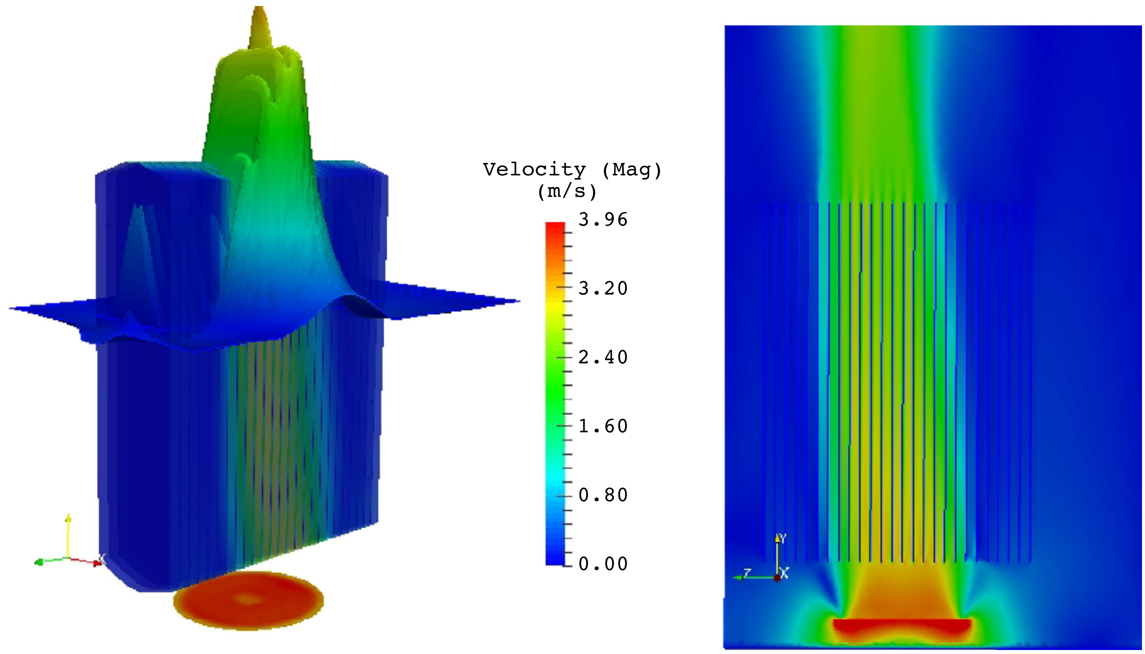

Fig. 5. Magnitude of the air velocity.

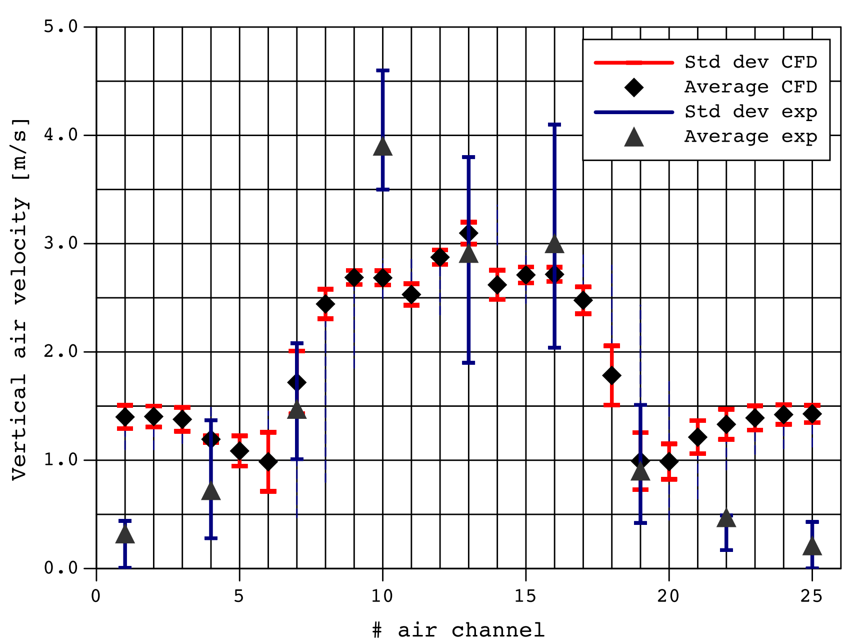

Fig. 6. CFD and experimental time averaged air velocity magnitude along the radiator.

radiator, with a hot-wire anemometer fitted with an 8[mm] diameter probe. The anemometer has a measuring range of 0.15 [m/s] to 3[m/s] with an accuracy of 3% and 3.1 [m/s] to 30 [m/s] with an accuracy of 3% and an air temperature range of −20 °C to 80 °C with an accuracy of 0.4%.

The workbench is surrounded by walls, not drawn in Fig. 1 in order to avoid air currents to influence the air speed measurements and the oil tank is wrapped with a thermal insulation to avoid heat losses.

The experimental test is carried out in accordance to the following procedure in order to compare the results of the measurements with those obtained with the CFD simulations presented in Section 3 and with those calculated with the reduced model presented in Section 2. First, the workbench is taken to steady state regime in ONAN, considering that the three heathers are supplied with 10 [kW] each. Then, the fans are turned on at 900 rpm and the power supplied by the heaters is adjusted in order to keep constant the oil temperature at the inlet of the radiator to the value registered in steady state ONAN conditions ($T_{oil_2} = 327.51$ K). The oil temperature at the outlet (T_{oil_1}) and the oil flow rate are measured to calculate the dissipated power of the radiator in that steady state ONAF condition. The results obtained with the former procedure are presented in Table 3, where T_{air_2} and U_{air} are mean time averaged quantities.

The vertical air velocity distribution measured for various air channels at 200[mm] from the top header center is shown in Fig. 6 (i.e. the air channels are numbered following the same order as the radiator panels – see Fig. 1). Therein the yellow dots denote

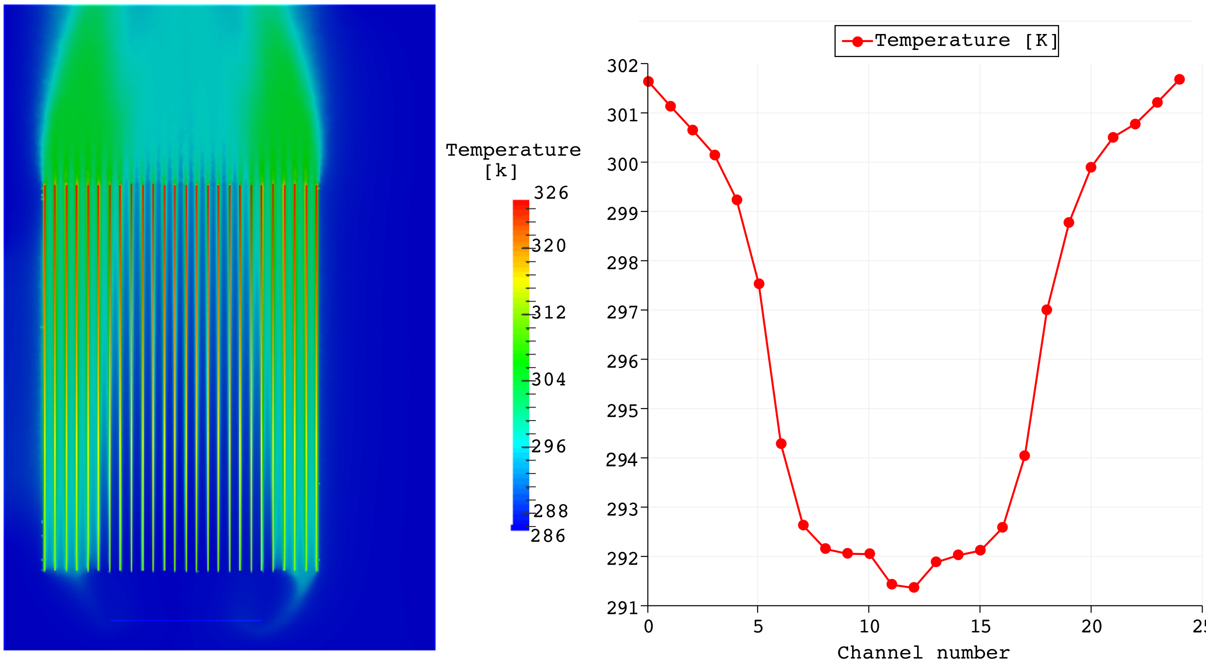

Fig. 7. Radiator mid section air temperature distribution and air temperature at channels center.

Table 2
Characteristic parameters of the radiator computed with the CFD simulation.

	Natural	Forced	Global
T_{air_2} [K]	300.5	293.7	–
P [kW]	4.8	8.6	13.4
U_{air} [m/s]	1.24	2.6	–

Table 3
Experimentally measured parameters of the workbench for steady state ONAF conditions.

	Natural	Forced	Global
T_{oil_2} [K]	–	–	327.5
T_{oil_1} [K]	–	–	311.6
T_{air_2} [K]	297.9	295.9	–
Q_{oil} [lts/min]	–	–	33
P[kW]	–	–	14.9
U_{air} [m/s]	0.68	3.27	–

the time-averaged mean values registered at the center of the channels and the blue bars represent the time averaged minimum and maximum values measured during a time lapse of approximately 180 s. It is apparent that the influence region of the fan is limited only to the 15 panels located directly above, which is the same behavior observed in the numerical simulation results, although there are some differences in the air velocities for the regions in natural convection.

Finally, Table 4 shows a comparison among the results computed with the reduced analytical model against those measured in the experiments and computed with the CFD simulation. Therein, for example, the relative error in the oil temperature ΔT_{oil} is defined as $|e_{\Delta T_{oil}}| = |\Delta T_{oil(RM)} - \Delta T_{oil(Exp)}|/|\Delta T_{oil(Exp)}|$ if it is in the *Reduced vs. Experimental* column or as $|e_{\Delta T_{oil}}| = |\Delta T_{oil(RM)} - \Delta T_{oil(Cfd)}|/|\Delta T_{oil(Cfd)}|$ if it is in the *Reduced vs. CFD* column.

When analyzing the results computed with the reduced model in Table 4 against those obtained in the experiments, it should be considered that there are some uncertainties in the temperature, oil flow rate and air velocity measurements, due to the accuracy of the instruments. On the other hand, the reduced model is based on a set of coupled nonlinear residual equations, and since it uses semi-analytical correlations and also experimental values as input data, it is very difficult to trace back which are the origins of the biggest uncertainties in the computed values. For example, the reduced model considers a correlation for computing the Nusselt number in the forced convection air zone, which depends on the air velocity. Since the air velocity used in the reduced model is the average of that measured in the experiments in the fan region and due to the uncertainties in the experimental values shown in Fig. 6 and also in the accuracy of the correlation given by Eq. (40), it is expected that this value could introduce some errors in the results computed with the reduced model.

On the other hand, if the air velocity in the fan region of the CFD simulation is analyzed, it should be considered that the fan is modeled as a boundary region with the aim of reducing computational costs. This simplification could also introduce errors in the CFD

Table 4
Comparison among the reduced analytical model, experimental and CFD results. (1) Experimental value used as input in the reduced model. (2) Experimental value used as input in the CFD model.

	Reduced model vs. Experimental			Reduced model vs. CFD		
	Natural	Forced	Global	Natural	Forced	Global
$\lvert e_{\Delta T_{oil}}\rvert$	–	–	0.157	–	–	–
$\lvert e_{T_{air_2}}\rvert$	0.016	0.004	–	0.007	0.011	–
$\lvert e_{Q_{oil}}\rvert$	–	–	0.136	–	–	(2)
$\lvert e_P\rvert$	–	–	(1)	0.096	0.228	0.112 (1)
$\lvert e_{U_{air}}\rvert$	0.367	(1)	–	0.456	0.205	–

results because, as it is already mentioned in Section 3, this fan model predicts the global flow rate impelled by the fan but does not provide a detailed description of the flow through the fan blades. As a consequence, the details of the air flowing into the radiator are not considered in the simulations and the exact details of the flow turbulence between the panels might not be accurately predicted. The simulation of such features of the flow should improve the accuracy of the CFD results in that region, but at a very high computational cost. Also, the reduced model equations do not consider local and three dimensional effects in the air flow through the panels and barely take into account the transition from laminar to turbulent flow considering parallel flow between flat plates.

5. Conclusions

A reduced model to simulate and analyze the thermo-fluid dynamic behavior of a power transformer working in ONAF conditions was introduced in this work. The model assumes a mixed heat convection framework, wherein a fraction of the radiator panels is dissipating heat to the air by forced convection and the other fraction is performing this by natural convection. The model solves a set of coupled non-linear equations for natural and forced heat convection which is obtained from applying the energy and momentum conservation laws to the air and the oil, as well as the heat conduction through the panel. Also, semi-empirical correlations and appropriate coupling conditions between both convection models are applied. The experimental measurements and the CFD results show that the reduced model predicts with acceptable accuracy the values of the most important design variables, i.e., oil flow rate, oil temperature, air temperature and mean velocity, at a reduced computational cost. This feature of the reduced model would allow to run parametric studies or introduce an optimization procedure, for example, in the shape and size of the oil channels or in the shape, size or separation distance between radiator panels. As a consequence, design engineers could consider this model as a valuable calculation tool. Of course, if localized effects like the oil velocity or temperature distributions within a channel are required to be analyzed, only CFD simulations and experiments on a workbench are able to provide detailed information but also at higher costs. Finally, the air velocity distribution along the radiator both computed in the CFD simulations and experimentally measured shows that the cooling produced by the fan which is tested in the workbench is not efficient since approximately half of the radiator panels are not blown. Some kind of diffuser together with a better design of the blades as well as the usage of smaller and evenly distributed fans may improve the efficiency in the cooling if vertical blowing is assumed. On the other hand, the horizontal blowing already explored in [5], considering an offset of the fans, is also worth to be analyzed.

Acknowledgments

This work has received financial support from Consejo Nacional de Investigaciones Científicas y Técnicas (CONICET, Argentina, grant PIP 11220150100588CO), Universidad Nacional del Litoral (UNL, Argentina, grant CAI+D 2011-01-00012-LI, CAI+D-501-201101-00233-LI), Agencia Nacional de Promoción Científica y Tecnológica (ANPCyT, Argentina, grants PICT 2660–14, PICT-E 0191–14, PICT 0938–13, PICT-2015-2904), Secretaría de Ciencia, Tecnología y Producción para la Defensa (grant PIDDEF-4/14), Agencia Santafesina de Ciencia, Tecnología e Innovación (ASACTEI, Argentina, grant 00010-18-2014).

This work was performed with *Free Software Foundation/GNU-Project* resources like GNU–Linux OS, GNU–GFortran, GNU–Octave, GNU–Git and GNU–GIMP, as well as other Open Source resources such as NETGEN, Para-View, Xfig and LaTeX.

Also, the authors made use of the computer cluster "Seshat", which is part of the computer center of CIMEC. This computer center is integrated to the Sistema Nacional de Computación de Alto Desempeo (SNCAD), which is a joint project of Ministerio de Ciencia, Tecnología e Innovación Productiva (MINCyT) and Consejo Interinstitucional de Ciencia y Tecnología (CICyT).

The authors would also like to acknowledge Tadeo Czerweny S. A company, Santa Fe, Argentina for providing human resources, the benchmark and other facilities for realizing the experimental measurements. Finally, a special acknowledgment is dedicated to Engineer José Ríos for giving his support and experience to carry out the experimental measurements.

References

[1] G. Ríos Rodriguez, L. Garelli, M. Storti, D. Granata, M. Amadei, M. Rossetti, Numerical and experimental thermo-fluid dynamic analysis of a power transformer working in ONAN mode, Appl. Therm. Eng. 112 (2016) 1271–1280.

[2] R.M.D. Vecchio, B. Poulin, M.E.F. Feeney, P.T. Feghali, D.M. Shah, R. Ahuja, D.M. Shah, Transformer Design Principles: With Applications to Core-Form Power Transformers, CRC Press, 2001.

[3] M. Kim, S.M. Cho, J. Kim, Prediction and evaluation of the cooling performance of radiators used in oil-filled power transformer applications with non-direct and direct-oil-forced flow, Exp. Therm. Fluid Sci. 44 (2013) 392–397.

[4] K. Karsai, D. Kerenyi, L. Kiss, Large Power Transformers, Elsevier Company, New York, 1987.

[5] S.B. Paramane, K. Joshi, W. Van der Veken, A. Sharma, CFD study on thermal performance of radiators in a power transformer: effect of blowing direction and offset of fans, IEEE Trans. Power Del. 29 (6) (2014) 2596–2604.

[6] S.B. Paramane, W. Van der Veken, A. Sharma, A coupled internal-external flow and conjugate heat transfer simulations and experiments on radiators of a transformer, Appl. Therm. Eng. 103 (2016) 961–970.

[7] N. El Wakil, N.C. Chereches, J. Padet, Numerical study of heat transfer and fluid flow in a power transformer, Int. J. Therm. Sci. 45 (2006) 615–626.

[8] M. Tsili, E. Amoiralis, A. Kladas, A. Souflaris, Power transformer thermal analysis by using an advanced coupled 3D heat transfer and fluid flow FEM model, Int. J. Therm. Sci. 53 (2012) 188–201.

[9] F. Torriano, P. Picher, M. Chaaban, Numerical investigation of 3D flow and thermal effects in a disc-type transformer winding, Appl. Therm. Eng. 40 (2012) 121–131.

[10] H. Nabati, J. Mahmoudi, A. Ehteram, Heat transfer and fluid flow analysis of power transformer's cooling system using CFD approach, Chem. Prod. Process Model. 4 (43) (2009).

[11] J. Gastelurrutia, J.C. Ramos, G.S. Larraona, A. Rivas, J. Izagirre, L. del Río, Numerical modeling of natural convection of oil inside distribution transformers, Appl. Therm. Eng. 31 (4) (2011) :493–505.

[12] S.W. Churchill, H.H.S. Chu, Correlating equations for laminar and turbulent free convection from a vertical plate, Int. J. Heat Mass Transfer (1975) 18.

[13] S.W. Churchill, A comprehensive correlating equation for forced convection from flat plates, AICHE J. 22 (2) (1976) 264–268.

[14] F.J. McQuillan, J.R. Culham, M.M. Yovanovich, Properties of dry air at one atmosphere, in: Microelectronics Heat Transfer Lab, University of Waterloo, 1984.

[15] R.H. Bartels, J.C. Beatty, B.A. Barsky, An Introduction to Splines for use in Computer Graphics and Geometric Modeling, Morgan Kaufmann, 1998, pp. 9–17.

[16] Code Saturne an open source CFD software. <http://code-saturne.org>.

[17] F. Archambeau, N. Mchitoua, M. Sakiz, Code Saturne: a finite volume code for the computation of turbulent incompressible flows-industrial applications, Int. J. Finite 1 (1) (2004) 1–62.

[18] H. Versteeg, W. Malalasekra, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Prentice Hall, 2007.

[19] F. Moukalled, L. Mangani, M. Darwish, The Finite Volume Method in Computational Fluid Dynamics, Springer International Publishing, 2015.

[20] D.S. Jang, R. Jetli, S. Acharya, Comparison of the PISO, SIMPLER, and SIMPLEC algorithms for the treatment of the pressure-velocity coupling in steady flow problems, Numer. Heat Transfer 10 (3) (1986) 209–228.

[21] Large Eddy Simulation for Incompressible Flows: An Introduction. Sagaut, P. Series: Scientific Computation, fourth ed., 2006.

[22] Smagorinsky, General circulation experiment with the primitive equation. I. The basic experiment. J. Month. Weather Rev. 91 (1963) 99–165.

[23] M. Germano, U. Piomelli, P. Moin, W. Cabot, A dynamic subgrid-scale eddy viscosity model, Phys. Fluids A 3 (1991) 1760–1765.

[24] J.A. Fay, Introduction to Fluid Mechanics, MIT Press, 1994, ISBN: 9780262061650.

[25] J. Kim, P. Moin, R. Moser, Turbulence statistics in fully developed channel flow at low Reynolds number, J. Fluid Mech. 177 (1987) 133–166.

[26] R. Dean, Reynolds number dependence of skin friction and other bulk flow variables in two- dimensional rectangular duct flow, J. Fluids Eng. 100 (2) (1978) 215–223.