

Accepted Manuscript

Numerical and Experimental Thermo-Fluid Dynamic Analysis of a Power Transformer working in ONAN mode

Gustavo Ríos Rodríguez, Luciano Garelli, Mario Storti, Daniel Granata, Mauro Amadei, Marcelo Rossetti

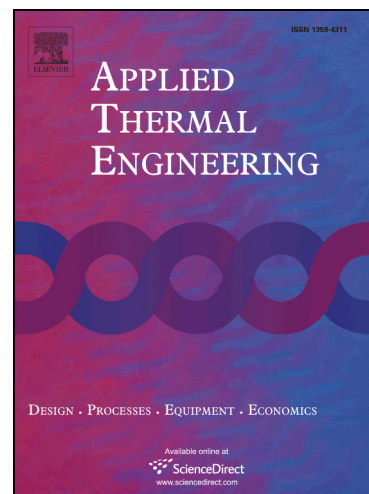
PII: S1359-4311(16)31529-0
DOI: <http://dx.doi.org/10.1016/j.applthermaleng.2016.08.171>
Reference: ATE 8970

To appear in: *Applied Thermal Engineering*

Received Date: 15 June 2016
Revised Date: 19 August 2016
Accepted Date: 27 August 2016

Please cite this article as: G.R. Rodríguez, 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, *Applied Thermal Engineering* (2016), doi: <http://dx.doi.org/10.1016/j.applthermaleng.2016.08.171>

This is a PDF file of an unedited manuscript that has been accepted for publication. As a service to our customers we are providing this early version of the manuscript. The manuscript will undergo copyediting, typesetting, and review of the resulting proof before it is published in its final form. Please note that during the production process errors may be discovered which could affect the content, and all legal disclaimers that apply to the journal pertain.



Numerical and Experimental Thermo-Fluid Dynamic Analysis of a Power Transformer working in ONAN mode

Gustavo Ríos Rodríguez^a, Luciano Garelli^a, Mario Storti^a, Daniel Granata^b, Mauro Amadei^b, Marcelo Rossetti^b

^aCentro de Investigación de Métodos Computacionales, CIMEC (UNL - CONICET), 3000 Santa Fe, Argentina.
Tel.: +54 (0) 342 4511594 Fax: +54 (0) 342 4511595

^bTadeo Czerweny S.A.,
2252 Gálvez, Santa Fe, Argentina.
Tel.: +54 (0) 3404 487200 Fax: +54 (0) 3404 483330

Abstract

This paper presents semi-analytical calculations, computational fluid dynamic simulations and experimental measurements accomplished on a typical 30MVA power transformer. An ad-hoc workbench was devised to carry out the experimental measurements. This facility allowed to estimate the values of different key variables involved in heat dissipation within the radiator, like convective heat transfer coefficients, oil-flow rate, air-flow speed between radiator fins and overall dissipated heat, among others. The main objectives of the study are to analyze the cooling capacity of the current radiator design working in ONAN mode and to validate the numerical simulation and calculation procedures for further design optimizations. The coupled thermo-hydraulic numerical simulations were performed on a computer cluster due to the high computational cost of the resulting models. The results of the simulations show good agreement with the experimentally measured values and with those obtained with the semi-analytical model, thus confirming that this model together with the CFD simulations are an accurate analysis tool. The heat transfer coefficients in the oil, the radiator panel and the air are computed. It is found that convective heat transfer from the panel to the air is approximately 10 times lower than heat transfer in the oil. Possible improvements on the current radiator design are briefly mentioned.

Keywords: Power Transformer, Computational Fluid Dynamics, Experimental Measurements, Heat Dissipation, ONAN

1. Introduction

Power transformers are one of the most valuable components in electrical power networks. During the electric conversion from high to low voltages, part of the energy

Email address: gusadrr@santafe-conicet.gov.ar (Gustavo Ríos Rodríguez)

is lost as heat in the windings and the core, which must be evacuated through the machine radiators. The heat is removed from the windings and transported to the radiators by the oil which flows through the gaps between the winding disks, then through the pipes located at the top of the radiators and into the cooling channels of the fins, then to the pipes at the bottom of the radiators and finally back to the windings. In Oil-Natural Air-Natural (ONAN) power transformers, both the oil flow through the cooling circuit and the air flow through radiator fins are produced by natural convection due to buoyancy effects. It is very important for the transformer manufacturer to have a detailed knowledge about how the heat is conducted and dissipated in the radiators. Identifying the components which comprises the main bottlenecks in the heat dissipation process allows to improve the design where it is most needed. For a given amount of heat to be evacuated, a better design enables to reduce the size and weight of the machine, thus improving its efficiency and lifetime, reducing the manufacturing costs. The aim of this study is the characterization of the heat dissipation performance of a given power transformer radiator of the ONAF (Oil-Natural Air-Forced) class working in ONAN mode, in order to further optimize its current design.

Various approaches have been developed and used, solely or in combination with others, to study and improve different areas of a power transformer design. Thermo-hydraulic or thermal network models are usually considered to study the dynamic behaviour of the complete machine. These models allow to obtain detailed cooling oil-flow and temperature distributions as well as the hot spot temperature in the windings [1]. However, these methods require to compute the heat transfer coefficients with analytical or semi-empirical formulas and cannot consider all the geometrical details of the model. On the other hand, numerical simulations based on the Finite Element (FEM) or Finite Volume (FVM) methods have been widely used to carry out magnetic, thermal and fluid dynamic analysis in different components of power transformers, since they enable a detailed representation of the involved geometry and physics. However, because of the high costs incurred by a 3-D coupled thermo-hydraulic simulation, Computational Fluid Dynamic (CFD) techniques are usually employed to analyze specific components of a power transformer rather than the complete machine. For example, in [2] the 3-D oil flow and temperature distribution in a disk-type winding are solved with CFD techniques employing meshes with hundred of million cells for the fluid and the solid. Also, in [3] an analysis of the heat transfer from the windings and the core to the oil is carried out on six different configurations of a power transformer with optimization purposes. The analysis is based on CFD simulations using the FVM method and assuming axisymmetry of the problem. In [4], an iterative coupled 3-D model is used for solving the thermo-hydraulic equations governing the transformer thermal performance using FEM, while in [5] 3-D CFD simulations are used to study the relationship between the radiator characteristics and its cooling capacity for a power transformer working in ONAN mode. Therein, the oil flow and temperature distributions are computed in the pipes, cooling channels and surfaces of the radiator fins assuming the convective heat transfer coefficient, which is theoretically calculated, is constant and homogeneous for all the surfaces of the radiator. Also, the oil mass flow rate and oil inlet temperature, which are taken from transformer documentation, are imposed as boundary conditions. A recirculation zone at the end of the radiator pipes, which prevents the oil to flow into the cooling channels of the last radiator panel, is found in

[5] as the explanation of low cooling efficiency. On the other hand, analytical and experimental studies about cooling performance of radiators used in power transformers working both in ONAN and ODAN (Oil-Directed Air-Natural) modes are present in [6]. Therein, the temperature distribution and cooling performance of radiators are predicted with theoretical calculations and then are validated using 3-D CFD simulations. Empirical equations are used in the analytical calculations to predict the convective heat transfer coefficients for the radiator fins and the temperature distribution is obtained from the energy balance differential equation for different oil mass flow rates. The effects of blowing direction and offset of fans on the thermal performance of a power transformer radiator block working in ONAF mode are studied in [7] by means of 3-D simulations. A Shear Stress Transport (SST) turbulence model is used for the air flow and the internal oil flow simulation is not considered. Therein, it is concluded that horizontal blowing is more efficient in dissipating heat than vertical blowing, since a 7.2% increase in the heat transfer rate is attained. Also, it is shown that a particular offset of the fans blowing in the horizontal direction increases the heat transfer rate in 3%. The numerical results are confirmed by experimental measurements performed on a five radiator workbench. More recently in [8], 3-D coupled internal oil and external air flow simulations together with a conjugate heat transfer model is used for analyzing the effect of blowing direction on the thermal performance of a five radiators block. In this work it is found that average oil temperature distribution along the height of the panels follows an exponential law instead of the widespread linear one. Also, it is confirmed a 6.1% increase in the dissipated heat if horizontal instead of vertical blowing fans are employed.

This paper describes a coupled thermal and fluid flow analysis of a 30MVA - 132/34.5/13.8kV power transformer radiator manufactured by the company Tadeo Czerweny S.A. The study is mainly focussed on the heat dissipation within the radiators assuming that the power transformer is working in ONAN 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. The computation of various flow and thermal characteristic parameters of the cooling circuit is addressed by means of an analytical reduced model, which couples the governing equations of the oil and the air flows with those of the heat transfer. The solution of this coupled system of equations delivers the values of the oil temperature at the inlet and outlet ducts of the radiator, the oil flow rate, the air temperature and its mean velocity. The equations are derived from the momentum and energy balance in the oil and the air, and the heat transfer from the oil to the air through the radiator steel fin surface. On the other hand, experimental measurements of the meaningful variables are carried out on a workbench specifically dedicated to this end at Tadeo Czerweny S.A. This workbench enables to validate the calculated values by performing measurements in controlled conditions, since it easily allows to modify the relative position (height) of the windings to that of the radiators, to change the separation distance between radiators, to accurately measure the air speed between radiator fins since the complete workbench is enclosed by protection walls to avoid spurious air currents and to accurately control the power supplied to the windings. 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 experimen-

tal setup is shown and briefly described. Finally, the corresponding CFD simulations are presented in detail. Due 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 solution of the coupled thermo-hydraulic problem is carried out using the computer cluster “Seshat” from CIMEC, equipped with a server Intel®Xeon®CPU E5-2640 v2 @2.00GHz and 69 computing nodes Intel®Xeon®CPU E5-1620 v2 @ 3.7GHz with 16GB RAM Micron®DDR3 1600 Mhz, interconnected with and Infiniband®network. To reduce the computational cost of the analysis, the heat transfer from the oil to the panel, the heat conduction through the steel and the convective heat transfer from the surface of the panel to the air are first simulated for a single panel. From this coupled simulation, a temperature distribution on the panel surface is obtained. This is used as initial temperature distribution on the panels in the complete radiator simulation, wherein only the convective heat transfer from the panels to the air is simulated. A convective heat transfer coefficient for the complete radiator is obtained in the simulation, which is later used to update the calculations done with the reduced model, avoiding the usage of semi-empirical formulas. Turbulence of the air flowing between the radiator fins is taken into account in the simulations with a Large Eddy Simulation (LES) model. The dissipated heat, temperature distribution on the panel surface, oil flow rate and air velocity distribution are compared to those experimentally measured. The encountered differences among the analytical, the CFD simulations and the experimental results are discussed.

2. Reduced model - analytical description

In order to estimate the cooling capacity of a radiator, a reduced model of the fluid dynamic and heat transfer in a power transformer is presented. This model is based on a set of coupled analytical equations (momentum and energy). As result, the oil flow rate (Q_{oil}), the inlet (T_{oil_2}) and outlet (T_{oil_1}) oil temperatures, the inlet (T_{air_1}) and outlet (T_{air_2}) air temperatures and the air mean velocity (U_{air}) will be established for a given power (P) to be dissipated (see figure 1).

2.1. Oil momentum balance

The driving force that impels 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, wherewith

$$\begin{aligned}\Delta p_{oil} &= (\Delta\rho)_{oil} g \Delta z, \\ &= (\beta_{oil}\rho_{oil}\Delta T_{oil}) g \Delta z,\end{aligned}\quad (1)$$

where β_{oil} is the oil thermal expansion coefficient, ΔT_{oil} is the oil 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 [9] as

$$\begin{aligned}\Delta z &= (h_1 + h_2), \\ h_2 &= L_p \left(\frac{\Delta T_{oai}}{\Delta T_{oil}} - \frac{\Delta T_{LMTD}}{\Delta T_{oil}} - 0.5 \right),\end{aligned}\quad (2)$$

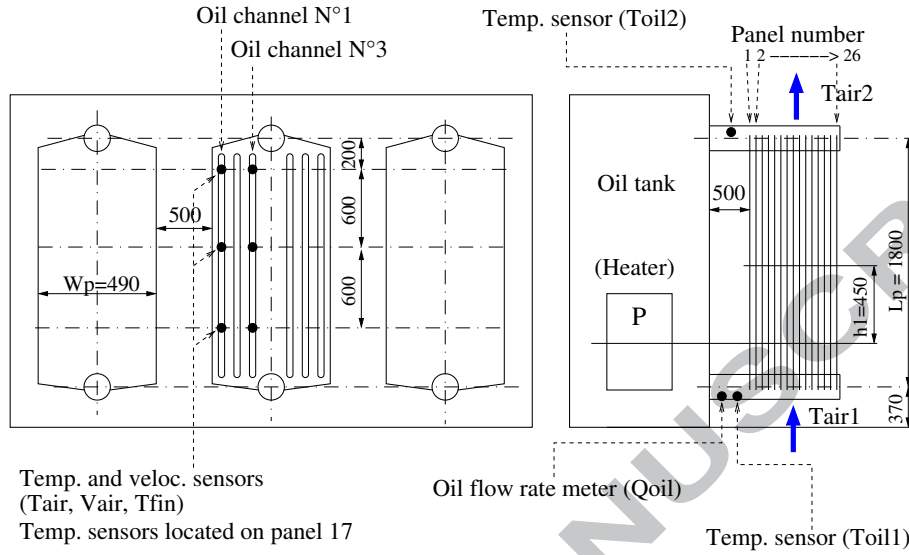


Figure 1: Schematic description of the experimental workbench and setup installation for ONAN flow tests (lengths are in mm and out of scale).

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.(1) is inversely proportional to the hydraulic resistance of the oil circuit. The head loss in the system is mainly given by the flow resistance within the channels of a radiator panel. In this work, a fully developed parallel oil flow is assumed [10]. This means that the flow pattern is the same for all sections along the channel and that it has only an axial component (Poiseuille flow). The channels of the radiator are modeled as rectangles of width w_{oil} and height L_y using the equation derived by [10] for an arbitrary shape

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

where μ_{oil} is the oil dynamic viscosity, A_{cha} is the channel area, Pe_{cha} is the channel perimeter and ξ is a nondimensional 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 [10] 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 Eq.(1) and Eq.(3), it can be obtained an equation of

the form

$$R_{mom,oil}(T_{oil_1}, T_{oil_2}, Q_{oil}) = 0. \quad (4)$$

2.2. 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

$$\begin{aligned} P &= (\rho C_p Q \Delta T)_{oil}, \\ &= F_{oil} \Delta T_{oil}, \end{aligned} \quad (5)$$

where $F = \rho C_p Q$ is the calorific capacity of the flow (W/K). Eq.(5) can be written as

$$R_{ener,oil}(T_{oil_1}, T_{oil_2}, Q_{oil}) = 0. \quad (6)$$

2.3. Air momentum balance

The radiator under analysis is working in ONAN condition, wherewith the air flows through the panels of the radiator by natural convection. Assuming that the flow between the faces of two panels is fully developed the Dean's [11, 12] correlation can be used. Using this correlation the friction coefficient (C_f) is defined as

$$C_f = \frac{\tau_w}{0.5 \rho_{air} U_{air}^2} = 0.073 \text{Re}_{air}^{-0.25} \quad (7)$$

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

$$\text{Re}_{air} = \frac{U_{air}(w_{air}/2)}{\nu_{air}} \quad (8)$$

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

$$Q_{air} = U_{air} w_{air} W_p N_p \quad (9)$$

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

$$\begin{aligned} \Delta p_{air} W_p w_{air} &= 2 \tau_w L_p W_p, \\ \Delta p_{air} &= \tau_w L_p / w_{air}, \end{aligned} \quad (10)$$

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

$$\begin{aligned} \Delta p_{air} &= \Delta \rho_{air} g L_p, \\ &= \frac{1}{2} \beta_{air} \rho_{air} (T_{air_2} - T_{air_1}) g L_p, \\ &= \frac{1}{2} (\beta \rho \Delta T)_{air} g L_p, \end{aligned} \quad (11)$$

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

$$R_{mom,air}(T_{air2}, U_{air}) = 0, \quad (12)$$

2.4. 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

$$\begin{aligned} P &= (\rho C_p Q \Delta T)_{air}, \\ &= F_{air} \Delta T_{air}, \end{aligned} \quad (13)$$

This equation can be written in the form

$$R_{ener,air}(T_{air2}, U_{air}) = 0. \quad (14)$$

2.5. Heat transfer between oil-air

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

$$H_{oil \rightarrow air} = h(T_{oil}(z) - T_{air}(z)) \quad (15)$$

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, the heat transfer coefficient can be written as

$$h^{-1} = h_{oil}^{-1} + h_{steel}^{-1} + h_{air}^{-1} \quad (16)$$

The heat transfer coefficient in the oil channel is computed as follows, considering the convective model proposed in [9] and laminar flow

$$\begin{aligned} h_{oil} &= \text{Nu}_{oil} k_{oil} / D_h, \\ \text{Pr}_{oil} &= \mu_{oil} C_{p_{oil}} / k_{oil}, \\ \text{Re}_{oil} &= \rho_{oil} U_{oil} D_h / \mu_{oil}, \\ \text{Gr}_{oil} &= \frac{g \beta_{oil} \Delta T_{os} D_h^3}{(\mu_{oil} / \rho_{oil})^2}, \\ \text{Nu}_{oil} &= 0.85 (0.74 \text{Re}_{oil}^{0.2} (\text{Pr}_{oil} \text{Gr}_{oil})^{0.1} \text{Pr}_{oil}^{0.2}); \end{aligned} \quad (17)$$

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

$$h_{steel} = \frac{k_{steel}}{t}, \quad (18)$$

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 [13]

$$\begin{aligned} h_{air} &= \text{Nu}_{air} k_{air} / L_p, \\ \text{Pr}_{air} &= \mu_{air} C_{p_{air}} / k_{air}, \\ \text{Gr}_{air} &= \frac{g \beta_{air} |T_{oil} - T_{air}| L_p^3}{(\mu_{air} / \rho_{air})^2}, \\ \text{Ra}_{air} &= \text{Gr}_{air} \text{Pr}_{air}, \\ \text{Nu}_{air} &= \left(0.825 + \frac{0.387 \text{Ra}_{air}^{1/6}}{(1 + (0.492 / \text{Pr}_{air})^{9/16})^{(8/27)}} \right)^2; \end{aligned} \quad (19)$$

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

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

$$\frac{d}{dz}(F_{air} T_{air}) = h(T_{oil} - T_{air}) \frac{d}{dz}(F_{oil} T_{oil}) = h(T_{oil} - T_{air}), \quad (20)$$

therefore

$$\begin{aligned} \frac{du}{dz} &= -\gamma(T_{oil} - T_{air}), \\ u &= T_{oil} - T_{air}, \\ \gamma &= h \left(\frac{1}{F_{air}} - \frac{1}{F_{oil}} \right) \end{aligned} \quad (21)$$

The solution of Eq.(21) is

$$u = u_1 e^{-\gamma z} \quad (22)$$

$$T_{oil_2} - T_{air_2} = (T_{oil_1} - T_{air_1}) e^{-\gamma L_p} \quad (23)$$

being the last equation in the system

$$R_{hex}(T_{oil_1}, T_{oil_2}, T_{air_2}, U_{air}, Q_{oil}) = 0. \quad (24)$$

The system of equations (4), (6), (12), (14) and (24) has to be solved to obtain T_{oil_1} , T_{oil_2} , T_{air_2} , U_{air} and Q_{oil} .

$$\begin{aligned} R_{mom,oil}(T_{oil_1}, T_{oil_2}, Q_{oil}) &= 0, \\ R_{ener,oil}(T_{oil_1}, T_{oil_2}, Q_{oil}) &= 0, \\ R_{mom,air}(T_{air_2}, U_{air}) &= 0, \\ R_{ener,air}(T_{air_2}, U_{air}) &= 0, \\ R_{hex}(T_{oil_1}, T_{oil_2}, T_{air_2}, U_{air}, Q_{oil}) &= 0. \end{aligned} \quad (25)$$

The thermal conductivity (k_{oil}), density (ρ_{oil}) and specific heat ($C_{p_{oil}}$) of the oil are calculated using the following approximations presented by [6],

$$\begin{aligned} k_{oil} &= 0.15217 - 7.16 \times 10^{-5} T_{oil_2}, [\text{W}/(\text{m K})] \\ \rho_{oil} &= 1067.75 - 0.6376 T_{oil_2}, [\text{Kg}/\text{m}^3] \\ C_{p_{oil}} &= 821.19 + 3.563 T_{oil_2}, [\text{J}/(\text{Kg K})] \end{aligned} \quad (26)$$

The oil kinematic viscosity is $\nu_{oil} = 3.25 \times 10^{-6} [\text{m}^2/\text{s}]$ and its thermal expansion coefficient is $\beta_{oil} = 7.9 \times 10^{-4} [1/\text{K}]$. The temperature of the surrounding air is assumed to be $T_{air_1} = 303 [\text{K}]$ so its physical properties are: kinematic viscosity $\nu_{air} = 1.56 \times 10^{-5} [\text{m}^2/\text{s}]$, $\rho_{air} = 1.17 [\text{Kg}/\text{m}^3]$, $\beta_{air} = 3.3 \times 10^{-3} [1/\text{K}]$, $C_{p_{air}} = 1005 [\text{J}/\text{Kg K}]$ and $k_{air} = 0.0281 [\text{W}/(\text{m K})]$.

The radiator has $N_p = 26$ panels. Each one is assumed to be of rectangular shape, having an equivalent length $L_p = 1.67 [\text{m}]$ and equivalent width $W_p = 0.521 [\text{m}]$, in order to have the same total wetted area $A_p = 1.74 [\text{m}^2]$ of the real panel. Each panel has six oil channels ($N_{cha} = 6$), each one with cross sectional area $A_{cha} = 3.248 \times 10^{-4} [\text{m}^2]$. The spacing between panels is $w_{air} = 0.045 [\text{m}]$ and the distance between the middle level of the radiator and that of the winding is $h_1 = 0.45 [\text{m}]$. The radiator is made of carbon steel with thermal conductivity $k_{steel} = 54 [\text{W}/(\text{m K})]$ and thickness $t = 1.25 \times 10^{-3} [\text{m}]$.

Solving the set of equations (25) with the above parameters and considering that the heat power dissipated by a single radiator measured in the experiments is approximately $P = 9.5 [\text{kW}]$ (see section 5), the computed average heat dissipated by each panel is $365 [\text{W}]$, the oil outlet and inlet temperatures are $T_{oil_1} = 343 [\text{K}]$ and $T_{oil_2} = 353.6 [\text{K}]$, the outlet air temperature is $T_{air_2} = 314.15 [\text{K}]$ and the oil flow rate is $Q_{oil} = 0.527 \times 10^{-3} [\text{m}^3/\text{s}] = 31.6 [\text{l}/\text{min}]$ with an average velocity in the oil channel equal to $U_{oil} = 0.0115 [\text{m}/\text{s}]$ and a mean air velocity $U_{air} = 0.92 [\text{m}/\text{s}]$. The total heat transfer coefficient is $h = 5.1 [\text{W}/(\text{m}^2\text{K})]$, which is composed by the air $h_{air} = 5.4 [\text{W}/(\text{m}^2\text{K})]$, the oil $h_{oil} = 110 [\text{W}/(\text{m}^2\text{K})]$ and the steel $h_{steel} = 43\,200 [\text{W}/(\text{m}^2\text{K})]$ heat transfer coefficients.

3. Three-dimensional thermo-fluid dynamic simulation of the radiator

Nowadays, numerical simulations provide a new methodology to characterize and evaluate the performance of different systems and equipments, reducing the cost and time of building an experimental facility. In order to have a detailed description of the fluid dynamic in the interior of the panel and the corresponding temperature distribution and heat transfer, a numerical model is generated. The results obtained from the CFD simulations can also be considered as input data in the reduced model previously presented, with the aim of reducing the hypothesis and assumptions made therein.

The numerical simulation is performed by solving the incompressible Navier-Stokes equations using a Finite Volume Method (FVM) [14, 15] with a transient analysis. A 3D segregated solver is used with a SIMPLEC coupling between velocity and pressure [16]. A Second Order Linear Upwind (SOLU) method is used for the spatial discretization and a second order scheme is also used for the temporal discretization. It is known that the heat transfer is very sensitive to the level of turbulence developed

within the air channels between radiator panels and also to the thickness of the boundary layer [13]. Thereby the mesh was generated in such a way as to use Large Eddy Simulation (LES) [17] with a Smagorinsky's [18] eddy-viscosity using the Germano's model [19]. A Boussinesq approximation is used to compute the driving force due to a temperature-dependent density.

The simulation was carried out in two stages in order to reduce the complexities and the computational cost of the resulting models. The first stage consists in evaluating the conjugated heat transfer in a radiator panel for a given oil flow rate. With this simulation the oil temperature and velocity distributions inside the channels and the temperature pattern on the panel surface are obtained. Then, in the second stage, a simplified geometry of the panel is used to generate a model of the radiator with 26 panels and the temperature pattern previously obtained is applied to the surfaces of the panels, avoiding to solve a conjugated heat transfer model for the complete radiator.

3.1. Conjugated heat transfer in a panel

To carry out the conjugated heat transfer simulation of a radiator panel, both the oil channels and the surrounding air have to be discretized. With the aim of obtaining reliable results and a good resolution of the shear stresses and boundary layers, a very fine mesh is needed near the walls. However, due to the symmetry, only a quarter of the panel can be modeled, helping to reduce the computational cost. In a computational domain of size $(2.2 \times 0.345 \times 0.05)$ [m] a quarter of the panel under study is placed. The inner part of the panel is discretized with 1.2 MCell (millions of cells) and the outer part requires 6.3 MCell, using a combination of structured and non-structured meshes.

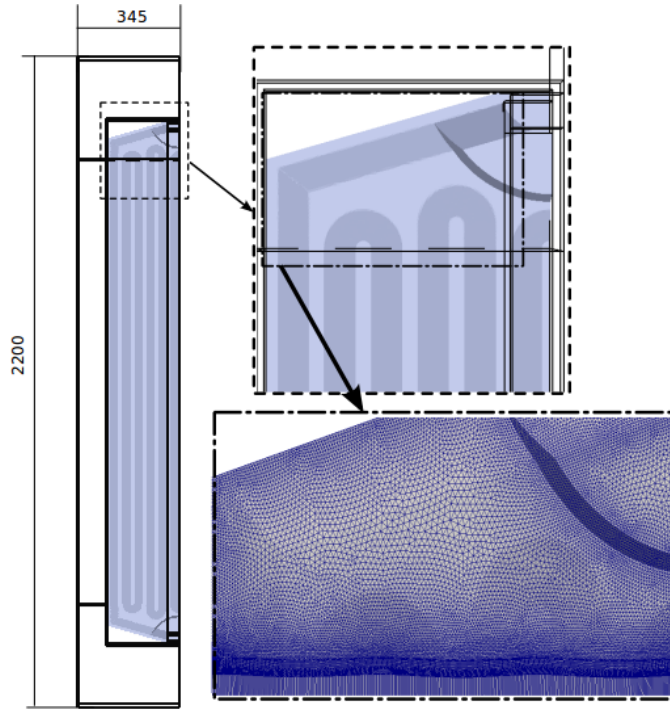


Figure 2: Panel model, computational domain and mesh detail at the top of the panel.

At the oil inlet section of the panel, a flow rate equal to $Q_{oil} = 2.2 \times 10^{-5} \text{ [m}^3/\text{s]}$ is imposed, which is computed as the average per panel of the oil flow rate obtained with the reduced model. The inlet temperature of the oil is $T_{oil_2} = 343 \text{ [K]}$ and the surrounding air is set at $T_{ref} = 303 \text{ [K]}$. At the oil outlet section a reference pressure is prescribed. A free inlet / outlet (opening boundary condition) is set at the top, bottom and sides of the fluid domain, allowing the natural flow of the air through these regions. The Reynolds number in the air channel estimated with the analytical calculations is $Re_{air} \approx 10^4$, so that in order to have enough spatial resolution to use a LES model, following the guidelines proposed in [20, 21], 50 grid points are placed in the normal direction to the wall inside of the air channel. An exponential mesh size growth rate from the wall to the bulk is considered, with a mesh size at the wall equal to $h_{wall} = 0.25 \text{ [mm]}$. The time step size used for the simulation is $\Delta t = 2.5 \times 10^{-3} \text{ [s]}$ and 10^4 time steps are solved until a steady heat transfer value is reached.

The temperature distribution on a cutting plane through the middle of the panel is shown in Fig.(3) right. Each of the six oil channels can be clearly identified in this figure because of the oil high temperature. Also, the low temperature at the edges and lower regions of the panel are apparent.

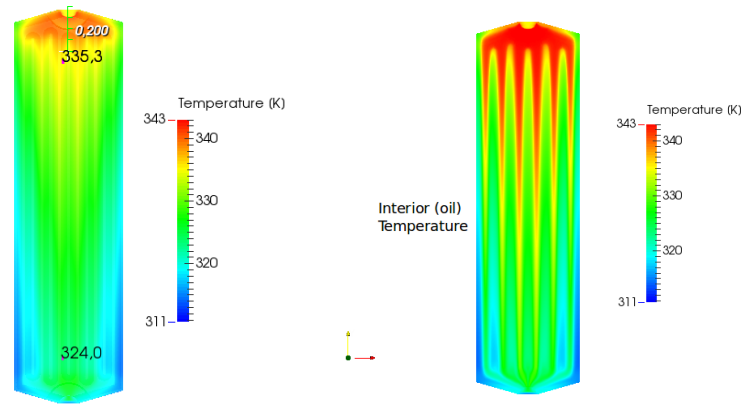


Figure 3: Exterior temperature on the panel surface (left). Interior temperature on a middle plane through the panel (right).

Another important result from this simulation is the temperature distribution on the exterior surface of the panel, which is shown in Fig.(3) left. This temperature distribution is also measured during the experimental tests at the locations marked in Fig.(1). From the numerical simulation, the temperatures at those locations are, for the oil channel number 3: 335.3 [K] at 200[mm], 331.0 [K] at 800[mm] and 324.0 [K] at 1400[mm]. On the other hand, for the oil channel number 1 they are: 332.0 [K] at 200[mm], 322.5 [K] at 800[mm] and 319.0 [K] at 1400[mm].

Also, this simulation gives detailed information about the oil flow distribution for each cooling channel. It can be seen in Fig.(4) that there is not a significant velocity difference between the channels. The oil flows mainly by the center of the channel with a maximum velocity of 0.0276 [m/s] which is 2.45 times the average velocity. This result validates the analysis done with the reduced model, wherein the obtained relation between the average and the maximum velocities is 2.4.

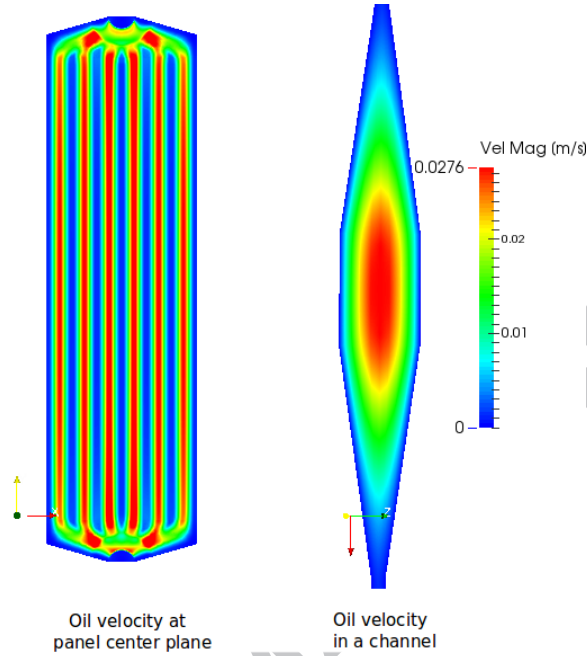


Figure 4: Internal oil velocity magnitude distribution

Assuming that $\Delta T_{oai} = (T_{oil_2} - T_{ref}) = 40$ [K], the heat dissipated by the isolated panel is $P_p \approx 580$ [W], with variations of $\pm 5\%$ during the transient simulation. It is known that, in real life, this dissipated power per panel will be lower due to the presence of the other radiator panels. This is why, during the design process of power transformers, manufactures have empirical correction factors that allow them to take into account the reduction in heat transfer due to the number of panels in the radiator, the separation distance between radiators and the separation distance between the panels, among others.

3.2. Radiator simulation

The temperature distribution on a single panel computed with the reduced model is used for setting the temperature distribution on each one of the 26 panels of the complete radiator. In order to reduce the complexities in generating the mesh, it is assumed that the panels have no thickness. 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] as shown in Fig.(5). The radiator relative position in the domain is the same that it has in the experimental facility with respect to the oil tank and the ground (see Fig.1). The domain is discretized with 8.21 Mcell using a hybrid mesh composed by tetrahedrons, hexas and pyramids in order to attain a good quality mesh. A cut of this mesh at the top of the radiator is also shown in Fig.(5). As was considered in the conjugated heat transfer simulation (section 3.1), the top and the sides of the fluid domain are set as free inlet / outlet (opening boundary condition), allowing the air to flow naturally through these regions. On the other hand, a wall boundary condition is set at the bottom in order to take into account the floor.

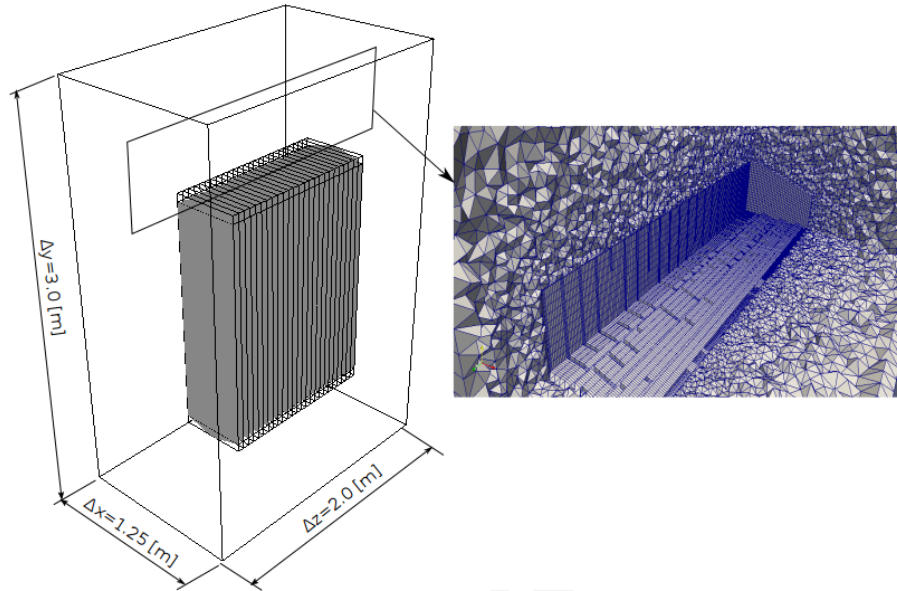


Figure 5: 3D domain for the complete radiator simulation and mesh detail around the radiator and between the panels.

The numerical simulation is carried out with a high performance parallel code considering the same methodology that was used in the coupled thermo-fluid dynamic problem (section 3.1). The physical properties of the air are the same than those used in the reduced model. As result of this simulation the total dissipated heat, the air velocity, the temperature between the panels and the global and local heat transfer coefficients are obtained. These results are compared in section 5 to those experimentally obtained and also are used as new input data to enhance the computations in the reduced model.

After reaching a quasi-stationary regime in terms of dissipated heat power, the temperature and velocity in the air channels are analyzed. The results are shown in Fig.(6) for three different y-coordinates in height (i.e., 200 [mm], 800[mm] and 1400 [mm] from the center of the top header, the same coordinates that are considered for making the experimental measurements, see section 4). At each height the velocity magnitude is represented as an elevation map and it is coloured with the temperature and velocity magnitude, respectively.

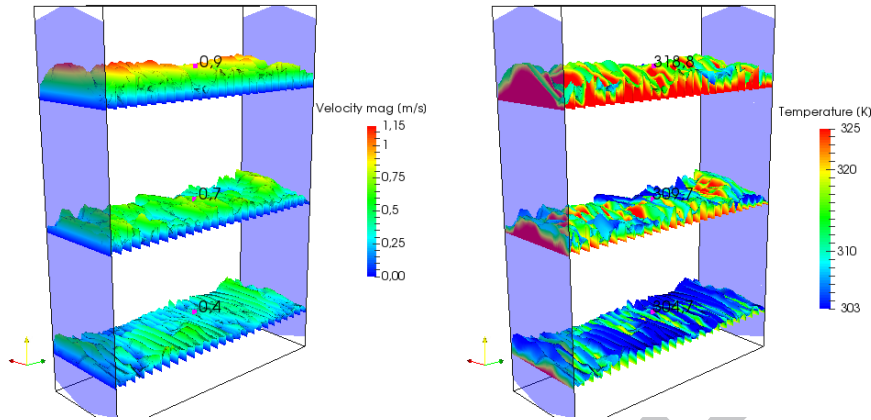


Figure 6: Air temperature and velocity magnitude at 200[mm], 800[mm] and 1400[mm] from the center of the top header.

At the lower position the air velocity fluctuates between 0.15 [m/s] and 0.5 [m/s]. After increasing its temperature, the air also increases its velocity. At the middle height section (800[mm]), the velocity ranges between 0.2 [m/s] and 0.8 [m/s] and at the higher section it fluctuates between 0.25 [m/s] and 1.15 [m/s]. The air velocity between two panels is highly dependent on the air flow entering to the radiator through the sides. This lateral flow also modifies the air temperature and the heat transfer coefficient. On the other hand, if the air temperature behaviour is analyzed, at the lower position it has almost the reference value but at the middle section it ranges between 309 [K] and 315 [K]. At the higher section it fluctuates between 316 [K] and 325 [K] with an average value of 318 [K], wherewith $\Delta T_{air} = (T_{air_2} - T_{ref}) = 15$ [K].

The colormaps of the local heat flux and heat transfer coefficients on panels 1, 6, 11, 16, 21, 26 are shown in Fig.(7) for a particular time instant. From these figures it can be stated that higher values of heat transfer correlate with the fluctuating streamlines of the flow in the air channels. This clearly shows the importance of considering an appropriate turbulence modelization.

The average air heat transfer coefficient and dissipated power for each panel in the radiator are plotted in Fig.(8). In this figure it can be observed that the exterior panels dissipate approximately 15% to 20% more energy than the inner panels. The average heat power dissipated by the radiator is 12.9 [kW]. Additionally, a time-averaged air heat transfer coefficient $h_{air} = 6.99$ [W/m² K] is calculated for the entire radiator over the last 200 time steps of the simulation. This value is 29.5% greater than that obtained with the reduced model.

The average value of the air heat transfer coefficient computed in the complete radiator simulation can be used as input data for the reduced model, instead of considering the value obtained with Eq.(19). In this manner, the new steady state condition obtained with the reduced model is: oil inlet and outlet temperatures $T_{oil_1} = 332.3$ [K] and $T_{oil_2} = 342.55$ [K], outlet air temperature $T_{air_2} = 314.45$ [K], oil flow rate $Q_{oil} = 0.541 \times 10^{-3}$ [m³/s] = 32.05 [l/min], average velocity in the oil channel $U_{oil} = 0.0105$ [m/s] and mean air velocity $U_{air} = 0.93$ [m/s].

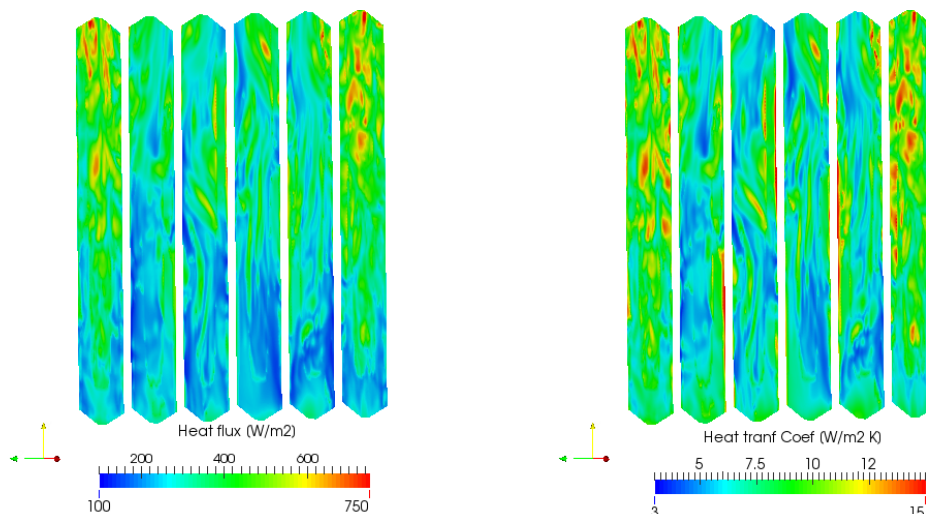


Figure 7: Heat flux and air heat transfer coefficient distributions for panels {1, 6, 11, 16, 21, 26}.

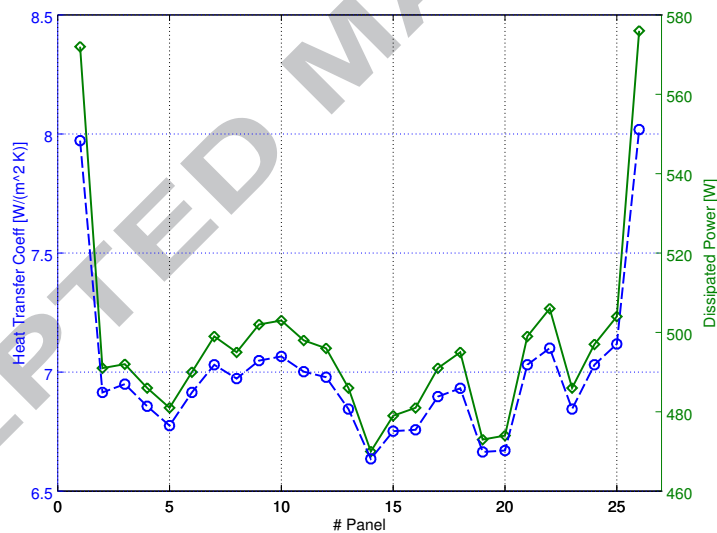


Figure 8: Dissipated power and air heat transfer coefficient for the radiator panels.

4. Experimental setup and measurements

Experimental measurements of different thermodynamic and fluid variables were accomplished on an ad-hoc workbench provided by the Tadeo Czerweny Company. Cooling oil is heated in a 3000 liters tank by three heaters of 10kW each, although it is possible to connect either one or two heaters also. Each radiator is provided with a

valve located at the bottom header which closes the oil circuit of that radiator. This, in conjunction with the possibility of connecting from one to three oil heaters, enables to make the tests with oil flow through only one radiator if no interaction effects between them are desired to be studied. This is useful for making comparisons with CFD simulations of only one radiator. The oil temperature is measured with k-type thermocouples installed at the top and bottom headers of the central radiator, halfway between the radiator and the oil tank (see figure 1). The oil flow rate is measured at the return header, after the oil has cooled in the radiator, as it is also denoted in the same figure.

On the other hand, the temperature on the fin surface is measured with k-type thermocouples at six different locations, denoted by dots in Fig.(1). There are two thermocouples located at 200 [mm] from the center of the top header, the next two are located at 800 [mm] from the same reference position and the latter are at 1400 [mm]. The thermocouples are making contact with the fin surface which comprises the outside of the oil channels number 1 and 3, each fin having six oil cooling channels. The 17th panel is chosen to make the measurements, counting from the closest to the oil tank panel to the outside of the radiator. Also, the same locations are used to measure the air speed and temperature between the radiator fins using a hot-wire anemometer with an 8[mm] diameter probe. In this case, the measurements are taken at the middle of the gap between two fins. Also, the air velocity and temperature are measured at 200 [mm] from the top header for various air channels between the radiator panels. 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 completely surrounded by protection walls (not drawn in figure 1) in order to avoid spurious air currents which could modify the measurements. Also, the oil tank is completely covered with a thermal insulation to minimize the heat losses. For all the tests, the vertical distance between the centerlines of the heater and the radiator is 450mm and the distance between the borders of two radiators is 500 [mm]. Each radiator is fitted with 26 fins, each one of 1800 [mm] height, 490 [mm] width and 1.74m² surface each side.

5. Results

The experimental results were obtained with the three heaters connected and the oil flowing through the three radiators, after the workbench reached a steady regime. The oil temperature difference measured between the inlet and outlet of the radiator is $\Delta T_{oil} = 10.4^\circ\text{C}$ and the oil flow rate is $Q_{oil} = 33$ [l/min]. If these values are compared to those obtained with the reduced model (i.e., $\Delta T_{oil} = 10.3^\circ\text{C}$, $Q_{oil} = 31.6$ [l/min]) after considering the heat transfer coefficient computed in the complete radiator simulation, it can be stated that there is a good agreement among them. Considering the energy balance in the oil given by Eq.(5), the measured dissipated power of one radiator is $P = 9.471\text{kW}$, which is almost 30% lower than that computed in the complete radiator simulation. Part of this discrepancy is due to the fact that the radiator simulation does not consider the influence of the vicinity of the other radiators, an effect that was present during the experimental measurements.

The temperature distribution on the surface of the panel at the locations indicated in Fig.(1) is summarized in table (1). These temperature values can be compared to

those computed by CFD with the conjugated heat transfer model for a single panel (see section 3.1). These values are also summarized in table (1). As it can be seen, the temperatures obtained with the CFD simulation are higher than those experimentally acquired, the major difference being approximately 5.5°C at the top of the panel. It is considered that this difference is acceptable given the uncertainties in the experimental measurements, thus confirming that conjugated heat transfer simulation is valid as an analysis tool.

Position [mm]	200	800	1400
Experimental measurements			
Oil channel N°1	326.5	323.4	321.2
Oil channel N°3	330.0	325.5	321.8
Computed with conjugated heat transfer model			
Oil channel N°1	332.0	322.5	319.0
Oil channel N°3	335.3	331.0	324.0

Table 1: Measured and calculated temperature [K] at different locations on the panel surface (see Fig.1).

On the other hand, the vertical air velocity is measured at 200[mm] for various air channels along the complete radiator (i.e. the air channels are numbered following a the same ordering than that used for the panels - see Fig.1). These velocities are shown in Fig.(9) for air channels number 1, 4, 7, 10, 13, 16, 19, 22, 25, the circles denoting the time-averaged mean values of the vertical air velocities at the center of the channels and the bars representing the time- averaged minimum and maximum values registered at those locations during the time lapse of the measurements, which was approximately 180 seconds.

Also, in the same figure are denoted with boxes the time-averaged vertical air velocities computed by CFD simulation on the complete radiator, widthwise of the same air channels. The diamonds in the boxes represent the time-averaged mean air velocities and the top (bottom) borders of the boxes represent the time-averaged maximum (minimum) values. It can be seen that, except for channel number 1, the air velocities computed by CFD agree rather well with those experimentally measured. On the other hand, the mean air speed recorded in the experiments at 200[mm] for the complete radiator, i.e. the average of the mean time-averaged velocities considering all the air channels and not only the ones shown in figure (9), is $U_{air} = 0.668$ [m/s] and that obtained from the radiator simulation is $U_{air} = 0.67$ [m/s]. Considering the level of turbulence of the flow in the air channels, the velocity fluctuations registered in the experiments and those obtained in the simulations, it can be inferred that both results are similar.

6. Conclusions

In this work it was analyzed the coupled thermo-fluid dynamic behaviour of a power transformer radiator working in ONAN mode. The methodology followed for the analysis consists in using first a reduced model based on analytical and semi-empirical formulas. Then, the results obtained from this model are used as boundary

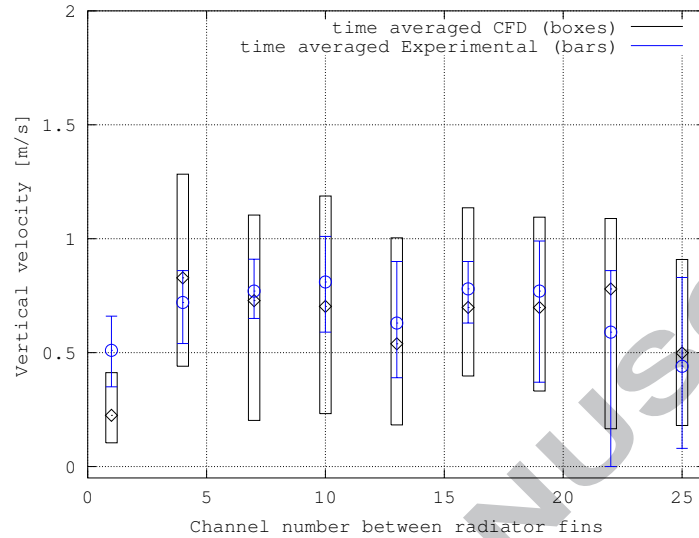


Figure 9: Experimental and CFD simulation vertical air velocity comparison at 200 [mm] from the top header.

conditions for carrying out a detailed coupled 3-D thermo-hydraulic CFD simulation for a single radiator panel. From this simulation, accurate oil flow in the cooling channels as well as temperature distribution on the surface of the panel are obtained. This latter is used as initial temperature distribution for all the panels in the complete radiator simulation, wherein only the convective heat transfer from the panels to the air is solved. Finally, a convective heat transfer coefficient is computed for the complete radiator, which is used in the reduced model for replacing the corresponding value calculated from empirical formulas. This procedure allows to improve the accuracy of the results calculated with the reduced model in first instance, since the convective air heat transfer coefficient obtained from the complete radiator simulation takes into account the geometrical details of the panel, the details of the oil flow and heat transfer within the cooling channels of the panel, the air turbulence, as well as the effects of the air flow coming in and out from the radiator laterals and the proximity between the radiator panels. The accuracy of the calculations is evaluated by comparing the numerical results against the experimental ones. This allows to conclude that the proposed numerical methodology is able to predict the values of some characteristic variables of the problem with an acceptable accuracy. However, for another parameters like the air speed and the temperature on the surface of the radiator panels, the uncertainties in the predicted values are higher. Also, the analysis enables to foretell physical phenomena like the air flow between radiator panels or the oil velocity distribution in the cooling channels of the panel. On the other hand, the proposed methodology avoids to solve the coupled 3-D thermo-hydraulic problem for the complete radiator, thus cutting down the costs from the computational point of view.

Finally, from the analysis performed on the current radiator design it is predicted

that the convective heat transfer from the panels to the surrounding air could be improved in order to increase the thermal performance. The calculations presented with the reduced model in subsection (2.5) show that the heat transfer coefficient in the oil is almost ten times higher than that in the air. Even if the oil flow rate were greatly increased the heat dissipated would not increase that much, as it is verified in [6] and [9]. Hence, the bottleneck in the heat dissipation in the radiator is due the low heat transfer from the panels to the air. It is thought that although there is turbulence in the air flow between the panels, this is not enough for improving the mixing of fresh air with the hot air within the thermal boundary layer in contact with the panels. Therefore, as next step in the research, different passive devices for improving the air mixing between radiator panels will 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 (AN-PCyT, 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 L^AT_EX.

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 Desempeño (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 the Tadeo Czerweny S.A company, Santa Fe, Argentina for providing human resources, the benchmark and other facilities for realizing the experimental measurements as well as the permission for publishing this work. Finally, a special acknowledgement is dedicated to Engineer José Ríos for giving his support and experience to carry out the experimental measurements.

References

- [1] Radakovic, Z.R. and Sorgic, M.S.: Basics of detailed thermal-hydraulic model for thermal design of oil power transformers. **IEEE Trans. on Power Delivery**, 25 (2), 790-802, 2010.
- [2] Torriano, F. and Picher, P. and Chaaban, M.: Numerical investigation of 3D flow and thermal effects in a disc-type transformer winding. **Applied Thermal Engineering**, 40, 121-131, 2012.

- [3] El Wakil, N. and Chereches, N.C. and Padet, J.: Numerical study of heat transfer and fluid flow in a power transformer. **International Journal of Thermal Sciences**, 45, 615-626, 2006.
- [4] Tsili, M. and Amoiralis, E. and Kladas, A. and Souflaris, A.: Power transformer thermal analysis by using an advanced coupled 3D heat transfer and fluid flow FEM model. **International Journal of Thermal Sciences**, 53, 188-201, 2012.
- [5] Nabati, H. and Mahmoudi, J. and Ehteram, A.: Heat Transfer and Fluid Flow Analysis of Power Transformer's Cooling System Using CFD Approach. **Chemical Product and Process Modeling**, 4(43), 2009.
- [6] Kim, M. and Cho, S.M. and Kim J.: Prediction and evaluation of the cooling performance of radiators used in oil-filled power transformer applications with non-direct and direct-oil-forced flow. **Experimental Thermal and Fluid Science**, 44, 392 - 397, 2013.
- [7] Paramane, S.B. and Joshi, K. and Van der Veken, W. and Sharma, A.: CFD Study on Thermal Performance of Radiators in a Power Transformer: Effect of Blowing Direction and Offset of Fans **Power Delivery, IEEE Transactions on**, 29 (6), 2596-2604, 2014.
- [8] Paramane, S.B. and Van der Veken, W. and Sharma, A.: A coupled internal-external flow and conjugate heat transfer simulations and experiments on radiators of a transformer **Applied Thermal Engineerig**, 103, 961-970, 2016.
- [9] Large Power Transformers. Karsai, K., Kerenyi, D. and Kiss,L.: Elsevier Company, New York, 1987.
- [10] Fay, J.A.: Introduction to Fluid Mechanics. ISBN: 9780262061650. **MIT Press**, 1994.
- [11] Kim, J., Moin, P., and Moser, R.: Turbulence statistics in fully developed channel flow at low Reynolds number. **Journal of Fluid Mechanics**, 177:133-166, 1987.
- [12] Dean R.: Reynolds number dependence of skin friction and other bulk flow variables in two- dimensional rectangular duct flow. **Journal of Fluids Engineering**, 100(2):215-223, 1978.
- [13] Churchill, S.W., and Chu, H.H.S.: Correlating equations for laminar and turbulent free convection from a vertical plate, **Int. Journal Heat Mass Tran.** 18 1975.
- [14] An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Versteeg,H. and Malalasekra, W. Prentice Hall 2007.
- [15] The Finite Volume Method in Computational Fluid Dynamics. Moukalled, F., Mangani,L., Darwish, M. Springer International Publishing 2015.
- [16] Comparison of the piso, simpler, and simplec algorithms for the treatment of the pressure-velocity coupling in steady flow problems. Jang,D.S., Jetli,R., Acharya, S.: **Numerical Heat Transfer**, 10(3):209-228, 1986.

- [17] Large Eddy Simulation for Incompressible Flows: An Introduction. Sagaut, P. Series: Scientific Computation, 4rd edition, 2006.
- [18] General circulation experiment with the primitive equation. I. The basic experiment. Smagorinsky, J. Monthly Weather Review, 91, 99-165, 1963.
- [19] A dynamic subgrid-scale eddy viscosity model. Germano, M., Piomelli, U., Moin, P., Cabot, W. Physics of Fluids A, 3. 1760-1765, 1991.
- [20] Ugo Piomelli: Wall-layer models for large-eddy simulations, **Progress in Aerospace Sciences**. 44:437-446, 2008.
- [21] Choi, H. and Moin, P.: Grid-point requirements for large eddy simulation: Chapman estimates revisited. **Center for Turbulence Research Annual Research Briefs**, 2011.

Numerical and Experimental Thermo-Fluid Dynamic Analysis of a Power Transformer working in ONAN mode

Highlights

- Coupled thermo-hydraulic CFD simulations of single radiator panel.
- Complete 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.
- Turbulence between panels unable to supply fresh air into thermal boundary layer.

ACCEPTED MANUSCRIPT