ENCIT 2018

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

NUMERICAL SIMULATION TO ANALYZE THE RADIATOR OF A POWER TRANSFORMER

Mara Nilza Estanislau Reis
Pontifícia Universidade Católica de Minas Gerais, Av. Dom José Gaspar, 500, Coração Eucarístico, Belo Horizonte, Minas Gerais, Brasil – Cep.30535-610
mara.nilza@terra.com.br

Marcos Vinícius Brasil
Pontifícia Universidade Católica de Minas Gerais, Av. Dom José Gaspar, 500, Coração Eucarístico, Belo Horizonte, Minas Gerais, Brasil – Cep.30535-610
contato@marcosbrasil.net

Paulo Henrique Souza Costa
Pontifícia Universidade Católica de Minas Gerais, Av. Dom José Gaspar, 500, Coração Eucarístico, Belo Horizonte, Minas Gerais, Brasil – Cep.30535-610
paulo.henrique.souza.costa@gmail.com

Thiago Henrique Brum de Paula
Pontifícia Universidade Católica de Minas Gerais, Av. Dom José Gaspar, 500, Coração Eucarístico, Belo Horizonte, Minas Gerais, Brasil – Cep.30535-610
thiago.brum@ymail.com

Abstract. *Power transformers are vital equipment for the electrical system and its radiators are fundamental for its proper functioning. This work is a study of these radiators having as target the companies that construct these equipments and with the main objective is develop computational model of the radiator of a power transformer using numerical simulation by CFD to validate its results through experimental data. The computational fluid dynamics (CFD) software STAR-CCM+ (v.11) was used to perform the simulations and evaluate the influence of the changes made on the radiator.*

Keywords: *Heat transfer, Radiator, Power transformer, STAR-CCM+, CFD*

1. INTRODUCTION

The transformer is an electric device that works from the principles of electromagnetic induction. During the conversion of electric energy from high to low voltages (or the opposite), part of the energy is lost as heat in the coils and cores and soon must be dissipated through the heat exchangers (radiators) of the machine.

In most of power transformers, the cooling occurs through the circulation of oil between ducts in the active parts and radiators outside the tank of the transformer. The heat coming out of the coils is transported to the heat exchanges by the oil and goes to the cooling channels of the parallel plates and then returns to thank of transformer.

The internal flow of oil in the radiator and the external flow of air over the radiator result in heat transfer by natural or forced convection. Thus, the cooling modes in transformer are: oil natural and air natural (ONAN), oil natural and air forced (ONAF), oil forced and air forced (OFAF), oil directed and air natural (ODAN) and oil directed and air forced (ODAF). (KIM, CHO, KIM, 2012; PARAMANE et al., 2014; GARELLI et. al., 2016).

The cooling capacity of the radiators for different cooling configurations needs to be known for reliable cooling system. Transformer manufactures are keen to reduce the number of radiators and fans – in order to reduce the weight of the system – without compromising on the thermal performance (PARAMANE, VEKEN, SHARMA, 2016).

Manufactures of power transformers have been following the industrial trend of improving the efficiency of their products mainly due to increase of technology in the industry and in the houses and the respective increase of consumption of electrical energy. Another Strong feature of today´s industry is the reduction in weight and size of products in order to save on raw material and unnecessary spending, a process also known as miniaturization.

2. THEORETICAL FOUNDATION

Heat can be defined as the energy that is transferred by virtue of a temperature difference. The heat flux occurs from regions of higher temperature to those of lower temperature. Heat transfer can occur through three basic mechanisms:

conduction, convection and radiation; between the three heat transfer mechanisms, convection is the mechanism of greatest influence on the transformer and responsible for the heat exchange almost in its totality.

2.1 First Law of Thermodynamics and Conservations Equations

The first law of thermodynamics, also known as principle of energy conservation, states that energy can´t be created or destroyed during a process, it can only changes its shapes. (ÇENGEL, GHAJAR, 2009)

The first law of thermodynamics is simply a statement that the total energy of a system is conserved, and consequently the only way in which the amount of energy in a system can change is if the energy crosses its boundary. In a closed system (a region of fixed mass) there are only two forms: heat or mass transfer across borders or work performed by the system (INCROPERA, 2015).

The energy flow in the system is determined by the Eq. (1).

$$\Delta E_{Acum.}^{Tot} = Q - W \tag{1}$$

Where:
$\Delta E_{Acum.}^{Tot}$ is the total energy accumuated by the system (W);
Q is the net heat value transferred to the system (W);
W is the net work performed by the system (W).

In a transformer there is neither work nor the entry of heat into the system and if the process occurs in steady state, that is, without varying the total energy of the system, the heat that is generated must be equal to the heat that is dissipated by the radiators. Then the energy balance Eq. (1) can be summarized according to Eq. (2): (ÇENGEL, GHAJAR, 2009)

$$Q_{ger} = Q_{sai} \tag{2}$$

Where:
Q_{ger} is the value of the heat generated by the system through the losses (W);
Q_{sai} is the value of the heat dissipated through the radiators (W);

The governing equations of the problem expressed in rectangular coordinates for the flow are equations of mass conservation, momentum and Navier-Stokes equations. The first physical principle for which the relationship between system and control volume formulations is applied is the principle of mass conservation: the mass of the system remains constant. The equation of mass conservation is also called the equation of continuity expressed by eq. (3): (FOX, PRITCHARD, MCDONALD; 2014)

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} + \frac{\partial \rho}{\partial t} = 0 \tag{3}$$

For steady state flows, where the specific mass variations with time are negligible, that is, $\partial\rho/\partial t \approx 0$, we can write eq. (4) as:

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0 \tag{4}$$

Where:
ρ is the specific mass (kg/m³);
x, y and z are the components of the physical coordinates (m);
u, v and w are the velocity components corresponding to the physical coordinates (m/s).

This equation indicates that the sum of the rate of mass change within the control column with the net rate of mass flow across the control surface is zero. (FOX, PRITCHARD, MCDONALD; 2014)

A dynamic equation describing the motion of the fluid can be obtained by applying Newton´s second law to a particle. In a differential form of the momentum equation, Newton´s second law is applied to a fluid particle of infinitesimal mass of mass dm. (FOX, PRITCHARD, MCDONALD; 2014)

The eq. (5), (6) and (7) determine the conservation equation of the momentum in differential form expressed in rectangular coordinates:

$$\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = \rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) \tag{5}$$

$$\rho g_y + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} = \rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) \tag{6}$$

$$\rho g_z + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} = \rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) \tag{7}$$

Where:
ρ is the specific mass (kg/m³);
g is gravity acceleration (m/s²);
τ is the shear stress (N/m²);
σ is the normal stress to the flow plane (N/m²);
u, v and w are the velocities in the axes x, y and z respectively (m/s);
x, y and z are coordinates of Cartesian system (m);
t represents the portion of time (s).

The conservation equations, applied to a Newtonian fluid, where the viscous stress is directly proportional to the rate of shear deformation, are called Navier-Stokes equations. When applied to incompressible flow and with constant viscosity are greatly simplified and can be expressed by eq. (8), (9) e (10):

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho g_x - \frac{\partial \rho}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \tag{8}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho g_y - \frac{\partial \rho}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \tag{9}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho g_z - \frac{\partial \rho}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \tag{10}$$

Where:
ρ is the specific mass (kg/m³);
g is gravity acceleration (m/s²);
μ is the coefficient of viscosity of the fluid (N.s/m²);
x, y e z are the components of the Cartesian coordinates (m);
u, v e w are the velocity components corresponding to the Cartesian coordinates (m/s).

2.2 Convection

Convection heat transfer occurs with the contact between a moving fluid and a solid surface, both at different temperatures and involving the effects of conduction and movement of a fluid. (ROHSENOW, HARTNETT, CHO, 1998).

When fluid movement is forced on the surface by the use of external means such as fan or pump, convection is said to be forced and is called natural convection when fluid motion is free, occurring only by fluctuating forces due to differences in density.

The rate of heat transfer by convection is expressed by the Newton's Law of Cooling. This rate is proportional to the temperature difference and the surface area in contact with the fluid, as can be observed in Eq. (11). (INCROPERA, 2015)

$$\dot{q}_{conv} = hA(T_s - T_\infty) \tag{11}$$

Where:
$\dot{q}_{conv}$ is the rate of heat transfer by convection(W);
h is the heat transfer convection coefficient (W / (m².K));

A is the surface area in contact with the fluid (m²);
T_S is the surface temperature (K);
T_∞ is the circulation fluid temperature (K);

The convective heat transfer coefficient can be defined as the rate of heat transfer between a solid surface and a fluid per unit area per unit of temperature difference. (ÇENGEL, GHAJAR, 2009)

3. METHODOLOGY

In order to simulate numerically the experimental data, collected and processed from a test of a three-phase 15 MVA and 69 kV oil-cooled power transformer by the ONAN method, a radiator model with a height of 2200 mm, 510 mm wide, 30 elements (plates) and a 45mm spacing there between using the commercial CAD software SolidWorks 2016. Fig. 1 represents the radiator model elaborated with its dimensions:

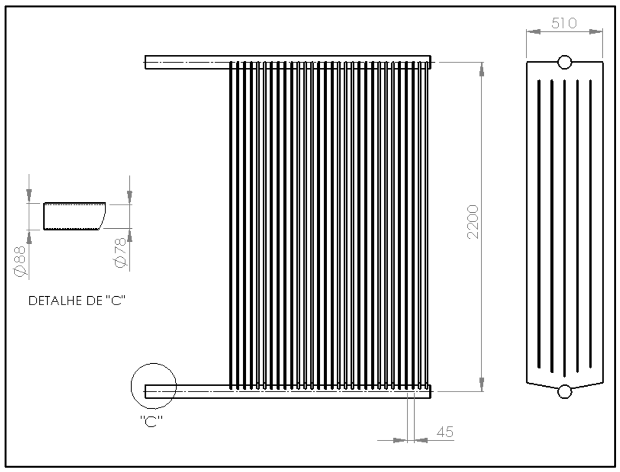

Figure 1. Radiator Dimensions

The radiator geometry elaborated in SolidWorks was then imported into the STAR-CCM+. Fig. 2 shows the imported CAD model for CFD software.

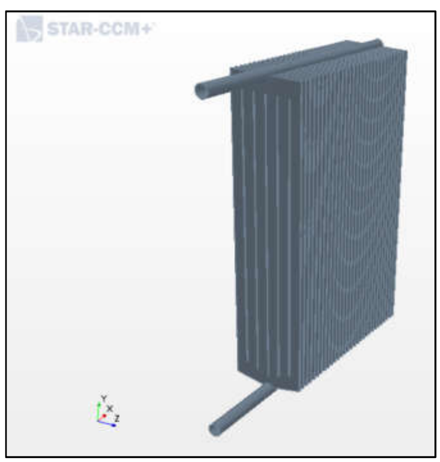

Figure 2. Radiator CAD model

The numerical model was elaborated from the finite volume method, a branch of fluid mechanics that uses numerical methods and algorithms to analyze problems involving mainly heat transfer or mass.

The vertical tubes of the studied radiator are plates composed of stamped plates and welded to each other forming the channels for oil passage. These channels have elliptic geometry. However, for our study and development of the calculations, only a rectangular tube of the same hydraulic diameter and the same external convection area of the added elliptical tubes were considered, as shown in fig. 3.

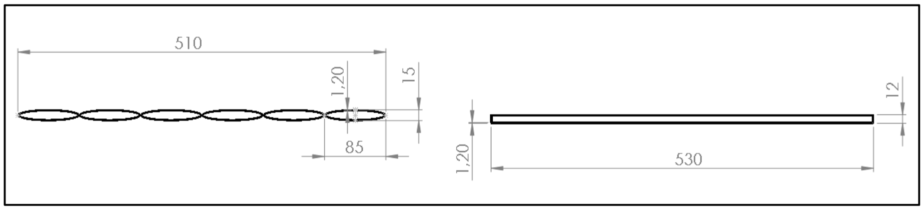

Figure 3. Approximation for mathematical modeling

To determine the hydraulic diameter of a non-cylindrical tube with a rectangular profile, eq. (12): (ÇENGEL, GHAJAR, 2009)

$$D_h = \frac{4A_c}{P} \quad (12)$$

Where:
D_h is the hydraulic diameter (m);
A_c is the area of the inner cross section (m²).
P is the internal perimeter (m).

Using eq. (12), the hydraulic diameter found was $D_h = 1{,}69x10^{-2}m$.

However, the numerical model of simulation was elaborated with several rectangular rather than elliptic sections, and maintaining the same characteristics, in order to make the simulation even closer to the real model, in order to obtain results closer to the experimental ones according to fig. 4.

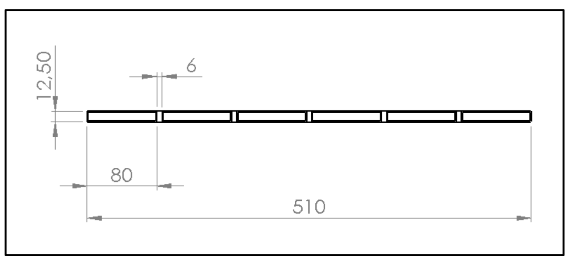

Figure 4. Section of plate for numerical modeling

As the parallel plates along with the oil inlet and return pipes are symmetrical, the simulation was done using only half of the heat exchanger. This technique is used to reduce the computational simulation cost.

Because it is a complex geometry represented in three dimensions and because it is more recommended for CFD simulations, a polyhedral surface mesh has been developed with a prism layer as shown in fig. 5

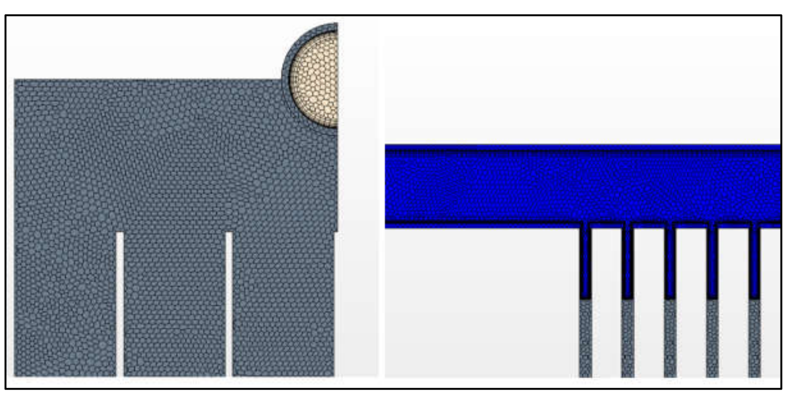

Figure 5. Mesh

The simulations were performed using a CFD software STAR-CCM+ 11. Experimental data obtained during the tests of a power transformer with capacity of 15MVA was used as input to variables and boundary conditions required.

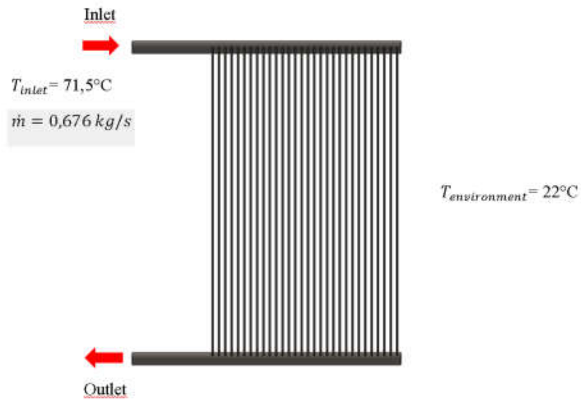

Figure 6. Radiator boundary conditions

The air and oil properties were used to calculate the convective transfer coefficients. In the case of heat transfer inside the radiator fin, considering flat plate and laminar flow, were used the equations (13) and (14) below:

$$Nu = \frac{h \times D_H}{k} \tag{13}$$

Where:
Nu is the Nusselt number;
h is the heat transfer convection coefficient (W / (m².K));
k is the fluid thermal conductivity (W / (m.K));
D_H is the hydraulic diameter for outline flow (m);

$$Nu = \{0{,}453.Re^{1/2}.Pr^{1/3}\} \tag{14}$$

Where:
Re is the Reynolds number;
Pr is the fluid Prandtl number;

The heat exchange from the radiator to the air occurs by natural convection. The radiator fin is a flat plate. To determine the average convective heat transfer coefficient were used equations (15), (16) and (17) below:

$$Nu = \frac{h \times L}{K} \tag{15}$$

Where:
L is the vertical plate leght (m);
k is the steel thermal conductivity (W / (m.K));

$$Nu = \left\{0{,}825 + \frac{0{,}387.R_{aL}{}^{1/6}}{[1+(0{,}492/Pr)^{9/16}]^{8/27}}\right\}^2 \tag{16}$$

Where:
R_{aL} is the Rayleigh number;
Pr is the air Prandtl number;

$$R_{aL} = \frac{g\beta(T_S - T_\infty)L}{\nu\alpha} \tag{17}$$

Where:
g is gravity aceleration (m/s²);
β is thermal expansion coefficient (1/°C);

T_S is the surface temperature (°C);
T_∞ is the environment temperature (°C);
L is the vertical plate length (m);
α is the thermal difusivity (m²/s);
v is the kinematic viscosity (m²/s);

These calculated values are necessary to fill boundary conditions in CFD software.

First, an analysis based on conditions similar those the transformer was tested was performed to validate the simulation with experimental results.

The following simulations were performed by modifying the physical characteristics of the radiator as the tube diameter and dimensions of radiator fin, trying to increase the radiator heat exchange.

4. RESULTS

The simulation was run on a computer with a 12-core Intel® Xeon® processor with 3.5GHz, 32GB of RAM and a Windows 8.1 Pro 64bits operating system in approximately 16 hours. After 1500 iterations, there were no longer significant variations in the monitored variable, in the case of the simulation, the radiator oil outlet temperature, and, thus, the simulation ended with a residual error of 1.0 e-04. To view the results, STAR-CCM+ has the scenes feature, where you can monitor various quantities in real time and different types of scales.

4.1 Oil Temperature in radiator

As shown in Fig, 7, one can observe how the temperature distribuition in the radiator is given.

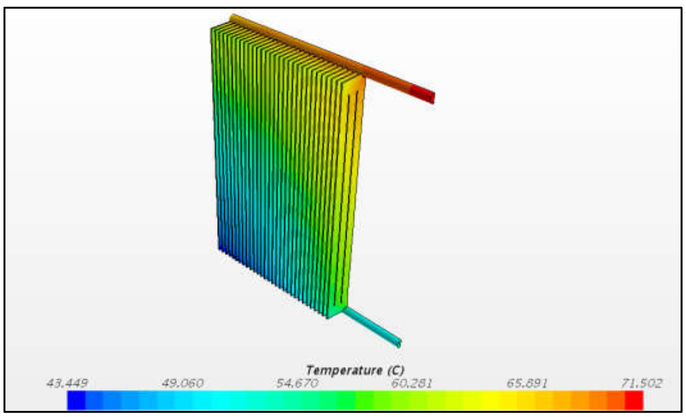

Figure 7. Temperature distribuition in °C

As shown in Fig. 8, it can be seen that temperature in the lower region of the elements (plates) decreases while the plate farthest from the radiator inlet region is evaluated.

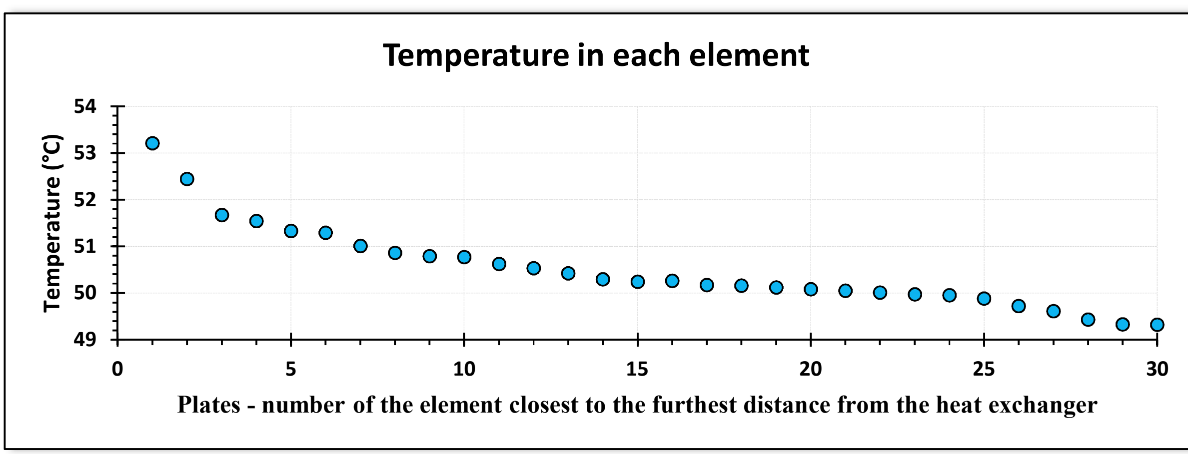

Figure 8. Variation of the oil temperature at the radiator base with the position of the plate

Using the "probe" or probe measurement feature of the STAR-CCM +, the value of the oil outlet temperature on the radiator surface was measured as well as experimentally measured.

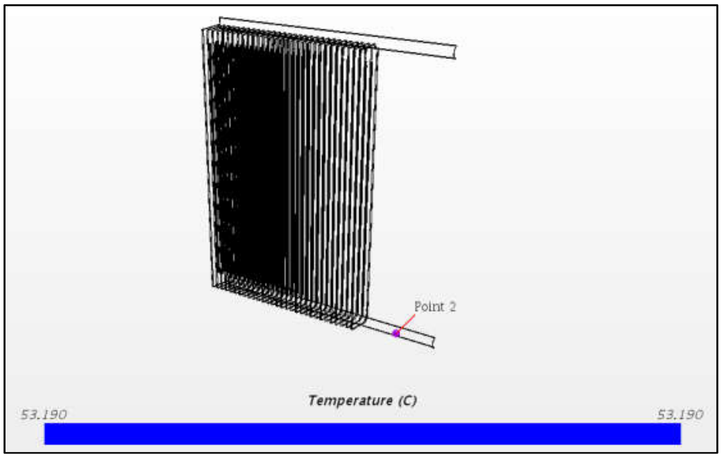

Figure 9. Oil outlet temperature in °C

As shown in fig. 9, it can be observed that the simulation results show behavior close to the experiment, because with the same input temperature and ambient temperature the simulation returns an oil output temperature value of 53.19 ° C, while the value measured during the test is 52.1 ± 1 ° C, with an error of 2.09%.

4.2 Oil speed profile

The results of the simulation in STAR-CCM+ still allow to observe the development of the speed of the oil inside the radiator, as well as the behavior of the speed of the oil along its distribution by the radiator as shown in fig. 10.

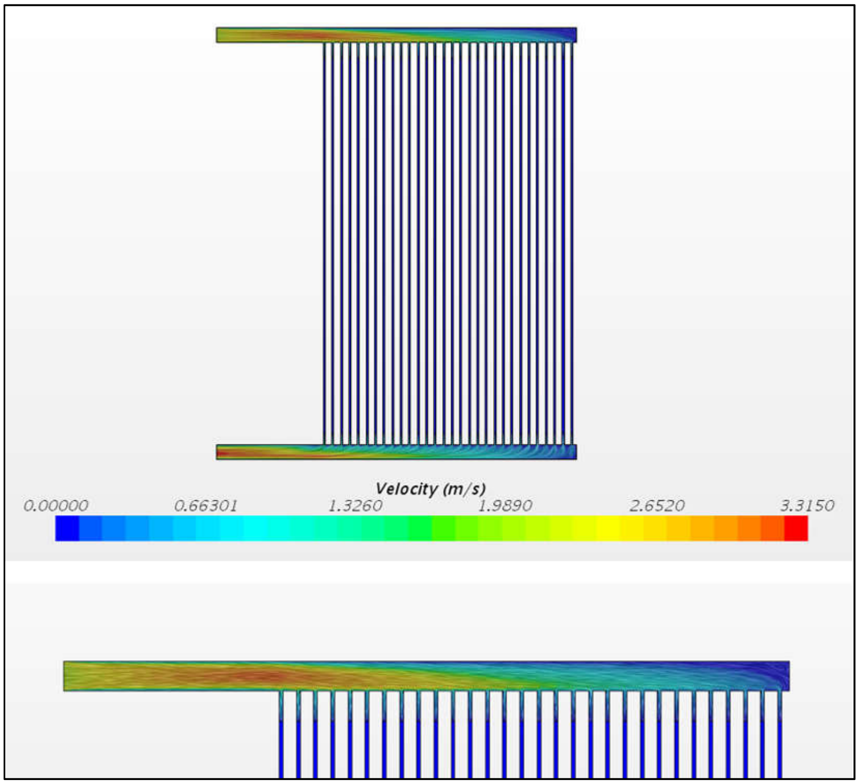

Figure 10. Oil speed profile on radiator

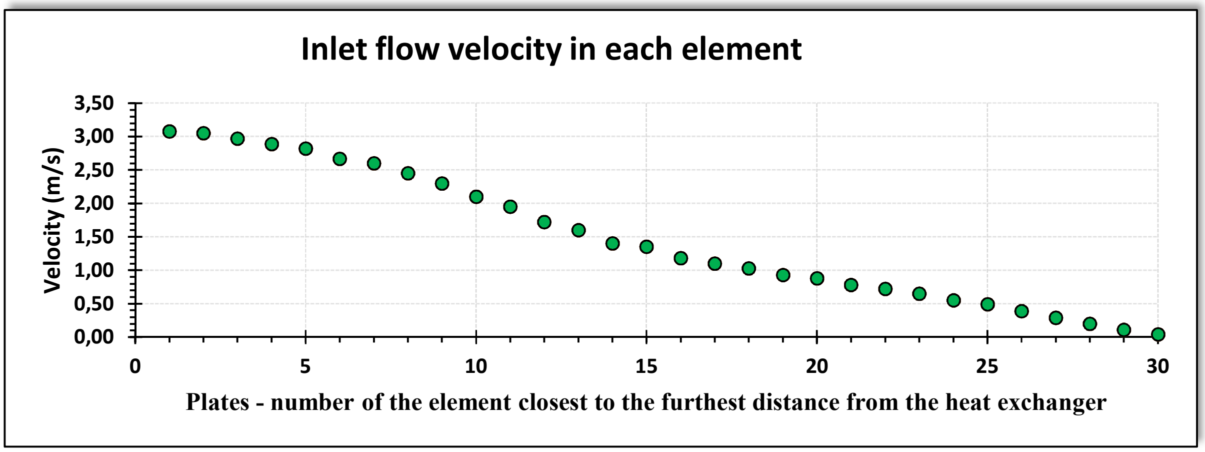

Figure 11. Variation of the oil speed at the entrance of the radiator plates with the position of the plate

Figure 11 shows the variation of the oil inlet velocity in the vertical plates according to the position of the plate, where the plates farthest from the transformer receive the flow at a slower speed. This occurs due to the decrease in the mass of fluid flowing in the conductive duct to the last plates of the heat exchanger, since the flow decreases along the tube as shown in fig. 12.

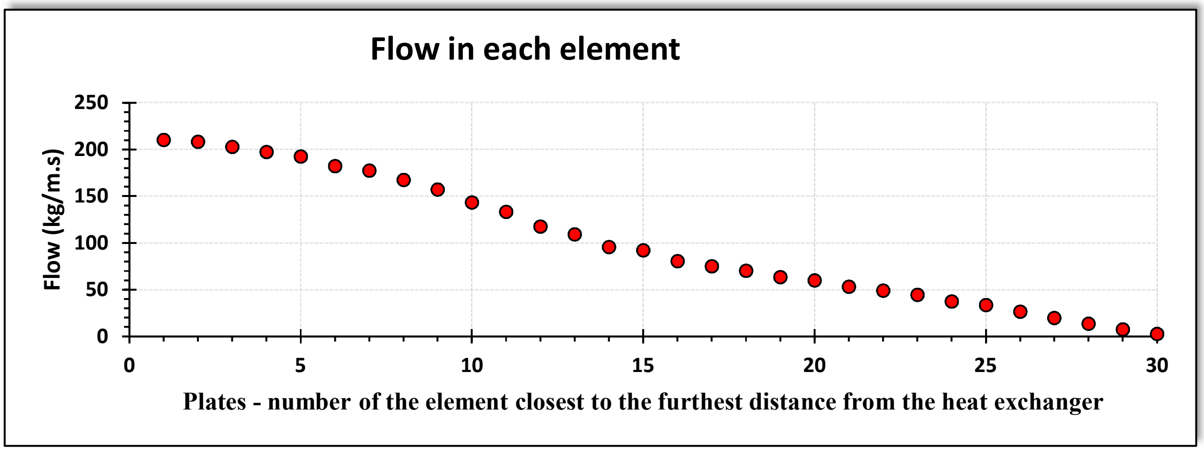

Figure 12. Variation of the oil flow at the radiator plate inlet with the position of the plate

5. CONCLUSION

The results show that the applied methodology, when compared with the experimental data, such as the configurations used in the program and the adaptations made in the oil flow channels of the numerical model, allowed a simulation in a timely manner with less computational requirement, and reflect a satisfactory result and very close to what was expected. While the experimental value for the oil temperature at the radiator outlet was 52.1 ° C, with a measurement error of ± 1 ° C, the simulation showed a value of 53.19 ° C. From these results and their approximation with the test data it is realized that the numerical simulation is a viable alternative for a critical analysis and later changes in a project since the costs of a simulation are much smaller than the prototyping of a transformer in scale. In this sense, it is correct to state that the simulation can be used to change the dimensions of the radiator and to verify the influence of the altered characteristics on the construction and the efficiency of the radiator, as well as to provide the project innovation.

6. REFERENCES

ÇENGEL, Yunus A.; GHAJAR, Afshin J. *Transferência de Calor e Massa: Uma Abordagem Prática.* 4. ed. Porto Alegre: AMGH, 2012

GARELLI, Luciano; RODRIGUEZ, Gustavo R.; STORTI, Mario A.; GRANATA, Daniel; AMADEI, Mauro; ROSSETTI, Marcelo. Análisis termo-fluido dinámico de un transformador de potencia en modo ONAN. Mecánica Computacional, v.34, p. 947-964, 8-11 nov. 2016.

INCROPERA, Frank P. et al. *Fundamentos de Transferencia de Calor e de Massa*. 7. ed. Rio de Janeiro: LTC, 2015.

KIM, Min-gu; CHO, Sang Moon; KIM, Joong-Kyoung. 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, v. 44, 23 jul. 2012.

PARAMANE, Sachin B.; KISHOR, Joshi; VEKEN, Wim Van der; SHARMA, Atul. CFD Study on Thermal Performance of Radiators in a Power Transformer: Effect of Blowing Direction and Offset of Fans. IEEE Transactions on Power Delivery, v. 29, no. 6, dez. 2014

PARAMANE, Sachin B.; VEKEN, Wim Van der; SHARMA, Atul. A coupled internal-external flow and conjugate heat transfer simulations and experiments on radiators of a transformer. Applied Thermal Engineering, v. 103, p. 961-970, 25 jun. 2016.

ROHSENOW, Warren M.; HARTNETT, James P.; CHO, Young I. *Handbook of Heat Transfer*. 3. ed., McGraw-Hill, 1998.

7. RESPONSIBILITY NOTICE