26th International Congress of Mechanical Engineering
COBEM 2021
ABCM Associação Brasileira de Engenharia e Ciências Mecânicas
November 22-26, 2021, Virtual Congress, Brazil

COB-2021-0804
PERFORMANCE EVALUATION OF AN ELLIPTICAL CROSS SECTION TUBE POWER TRANSFORMER RADIATOR USING A CFD MODEL

Guilherme Ladeira Pinho
João Marcos Oliveira Condé
Mara Nilza Estanislau Reis
Gustavo Fonseca de Freitas Maia
Pontifícia Universidade Católica de Minas Gerais, Instituto Politécnico da PUC Minas, Departamento de Engenharia Mecânica.
gladeirap@gmail.com; joaomarcosconde@gmail.com; mara.nilza@terra.com.br; gustavomaia@hotmail.com

Abstract. *Power transformers are mainly used in electrical energy generation and distribution systems. Radiators are attached to its structure for the purpose of promoting a cooling procedure of the electrical equipment, ensuring that it can function properly with a minimum energy loss caused by heat interference. Recent studies showed that, at the exit flange of the equipment, the temperature of the fluid, which flows internally, could present a significant variation on its value, depending on the radiator cross section area. One of the studies presented three main subjects: a geometric modeling of an elliptical cross section tube radiator, the calculation procedure of convection coefficient for external fluid flow and the execution of a fluid dynamic simulation of the model, using Siemens Star CCM+ software. The fluid inlet temperature, outlet temperature and mass flow were obtained through experimental observation of a commercial model, manufactured by a renowned company from the industry, with similar design characteristics. These data could be used as boundary conditions for the setting configuration in simulation process. With these parameters, the simulation resulted in the obtention of radiator internal fluid flow velocity and temperature gradients, which could be used to validate the model, in comparison with the experimental results. The mean fluid outlet temperature obtained from simulation was 4.7 ºC higher, if compared to the experimental method. From this result, it could be concluded that the final value for outlet fluid temperature, obtained from computational fluid dynamic simulation, was a good approximation to the experimental result. This paper proposed an improvement of the radiator design, which involved the reduction of the number of vertical plates, from 30 to 28 and 28 to 26, in order to reduce manufacturing costs, without causing a negative impact to the heat exchange process. Another proposal was made to modify the position of the radiator outlet flange, in order to improve fluid internal circulation. The fluid outlet temperatures obtained from simulation, considering the proposed modifications, were similar if compared to the previous study results, indicating the viability of the improvement. Although the temperature did not decrease with the fluid outlet position change, but an enhancement of the fluid flow distribution at the inlet tube could be observed.*

Keywords: *Power transformers, Radiator, Elliptical cross section, StarCCM+.*

1. INTRODUCTION

A power transformer is an electrical equipment used in electrical energy transmission. It is used to transfer power through electromagnetism, usually to increase or decrease the voltage between two circuits. Inside the power transformer, energy losses occur from different sources, such as coil resistance, eddy current flow change or leakage current in its structural parts. Such energy losses are proportional to the equipment electric load or to its physical dimensions. The power transformer energy losses usually generate heat and, consequently, an increase in temperature. In this case, its temperature must be decreased in order to maintain the equipment integrity.

A radiator is an equipment used to cool down the power transformer, allowing the heat transfer between different fluids with different temperatures. The heat is removed from the power transformer windings and transported to the radiator by the oil, which flows internally through the gaps between the winding disks. After leaving the disks, the oil drains through the main tube, located on the top of the radiator, and then into the plate channels. After losing heat, the oil flows from the plate channels directly to the main bottom tube of the equipment, making a return path to the windings. According to Rodriguez *et al.* (2016), an Oil-Natural Air-Natural radiator configuration consists in a specific cooling system mainly used in power transformers which, both the oil, that flows through the cooling circuit, and the air, that flows through radiator fins, occurs from natural convection due to buoyancy effects. The radiator heat transfer efficiency is related to the fluid thermal and mechanical properties, the thermal and flow conditions and the equipment structural design. Special devices, such as fins and baffles, can also be installed on the equipment surface or inside its structure in order to improve the heat exchange efficiency. A numerical method can be used to analyze the radiator thermal efficiency through a CFD (Computational Fluid Dynamics) software. With this method, developers become able to adjust structural and thermophysical parameters, in order to improve the thermo engineering project development, allowing better design proposals and machine size and weight reduction, resulting in a better efficiency, lifetime and manufacturing costs. Due to computational performance restrictions occurrence, which are more related to the geometry complexity, the CFD simulation methods become less implemented in studies involving models with irregular tube cross sections and in which are submitted to complex boundary conditions. According to Radakovic and Sorgic (2010), thermo-hydraulic or thermal network models are usually considered to study the dynamic behavior 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. 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, CFD techniques are usually employed to analyze specific components of a power transformer rather than the complete machine. In this work, a radiator with elliptical channel plates based on a model from a renowned manufacturer was used and some design changes were proposed for it. One of the proposals made was related to the reduction in the number of vertical plates, in order to reduce equipment manufacturing costs, without causing a negative impact to the heat exchange efficiency. Another proposal was associated with the radiator fluid outlet tube position change, in order to improve fluid circulation inside the equipment.

2. THEORETICAL FOUNDATION

According to Incropera *et al*. (2014), the term heat transfer is defined as the thermal energy in transit that flows due to the difference between temperatures. This event can be classified in three different types of heat transfer: conduction, which is the heat transfer that occurs through the solid; convection, which is the transfer that takes place between a surface and a moving fluid, and thermal radiation, which is a phenomenon that encompasses the emission of energy in electromagnetic waves form from surfaces. The main types of heat transfer that occur during the radiator operation and that were considered for this study are the conduction, on the radiator walls, and convection, involving both the part in which the fluid flows internally and the part which the fluid flows externally to the equipment. According to Incropera *et al*. (2014), convection can be classified in two types, depending on the nature of the flow. Forced convection is the type which occurs when the flow is generated by external events such as fans, pumps and atmospheric winds. Natural convection, on the other hand, is the one in which the flow occurs through differences in density caused by variations in the fluid temperature. According to Holman (1983), in natural convection, the fluid in contact with a heated surface undergoes a decrease in density and, as a result, tends to rise. Therefore, the colder fluid tends to occupy the empty space generated by the heated fluid, thus causing convection currents.

The convection heat transfer coefficient calculation is fundamental for heat transfer analysis and to the convection problem solution. According to Holman (1983), the coefficient is defined by fluid transportation properties (thermal conductivity, viscosity, density, among others), flow geometry (which takes place on a surface or inside a tube, for example), surface finishing (smooth or rough) and flow regime (laminar or turbulent). The definition of the Reynolds number becomes important for both internal and external flow, since it determines the fluid flow regime inside and over the equipment, which can be turbulent or laminar. The fluid flow regime is also necessary to know in order to determine the most adequate correlation to be used in the convection heat transfer coefficient calculation. According to Moukalled *et al*. (2016), the Nusselt number is a dimensionless form of the convection heat transfer coefficient and provides a measure of this type of heat transfer on a solid surface. The Prandtl number, in turn, represents the measure of the relative effectiveness of momentum at the velocity boundary layer and the effectiveness of energy transport by diffusion in the thermal boundary layer. This dimensionless is defined by the ratio between the moment diffusivity (kinematic viscosity) and the thermal diffusivity. According to Incropera *et al*. (2014), the Prandtl number has a direct influence in the relative growth of the velocity boundary layer and thermal boundary layer, so that values greater than one for this dimensionless represent a velocity boundary layer thickness greater than the thermal boundary layer thickness in a given flow regime. For values smaller than one, the opposite occurs. The Grashof number is a dimensionless parameter that establish a measure of the ratio between the buoyancy forces and the viscous forces related to the flow velocity boundary layer. The Rayleigh number, on the other hand, is associated with the free convection of a fluid. Its value, being lower than the critical value for a given fluid, represents the occurrence of heat transfer primarily by conduction. While for its value being greater than the critical one, the primary transfer occurs by convection. This dimensionless is defined by the product of the Grashof number and the Prandtl number.

According to Versteeg and Malalasekera (2007), Computational Fluid Dynamics or CFD is a tool that allows to analyze and predict systems involving fluid flow, heat transfer and associated phenomena (such as chemical reactions), through computer-based numerical simulation. The equations that govern fluid flow and heat transfer in the CFD software are the continuity equation, Newton's second law (conservation of momentum) and the first law of thermodynamics (conservation of energy). According to Çengel and Ghajar (2012), most engineering problems related to heat transfer involves geometries with complex boundary conditions, which normally cannot be solved analytically. When there is no analytical solving option, solving by numerical methods can be a good choice. This method is based on replacing the differential equation with a set of algebraic equations. For problems involving heat transfer, the equations are associated with unknown temperatures at selected points from the geometry. The simultaneous solution of these equations results in temperature values at these discrete points.

3. METHODOLOGY

3.1 Geometry Design and Simulation Parameters

For the proposed radiator numerical analysis, a similar methodology used by Reis *et al.* (2018) was applied to this study, resulting in the creation of a three-dimensional model, which represents the equipment structure, based on the dimensions provided by the manufacturer drawing. The radiator model is composed by a set of plates with elliptical channels that, through convection with the ambient air, promote heat dissipation from the hot oil that flows through the channels. The three-dimensional radiator developed model is represented by Figure 1 and its technical drawing is represented by Figure 2.

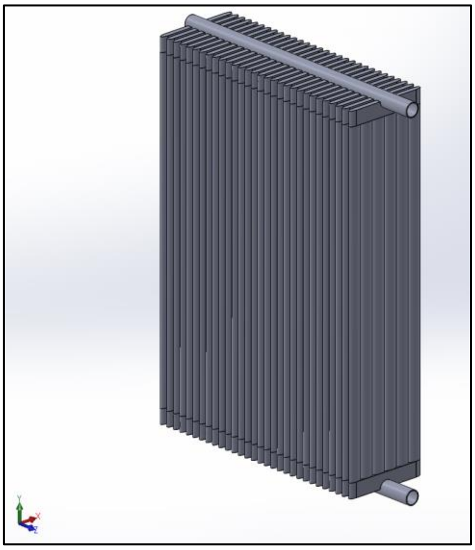

Figure 1. Radiator CAD model.

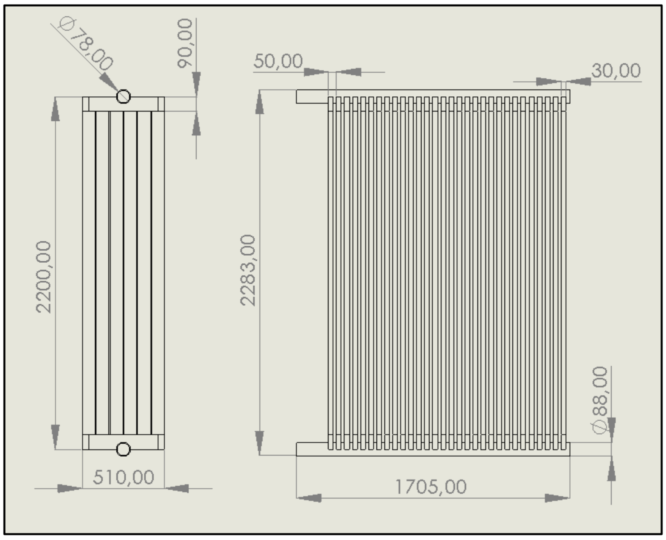

Figure 2. 30 vertical plates radiator dimensions.

Each one of the plates arranged in the developed model has six branches, whose cross sections have an elliptical shape and allow the oil to flow through the entire radiator. These branches are connected to the horizontal oil inlet and outlet ducts.

The dimensions of the vertical plate elliptical channels can be seen in more detail in Figure 3.

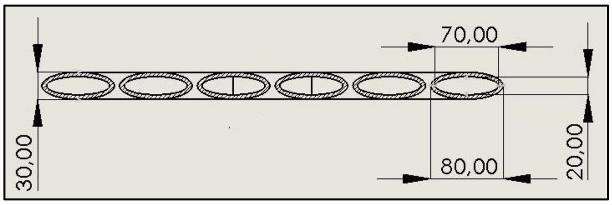

Figure 3. Cross-section view from the radiator vertical plate.

Siemens Star CCM + 2019.2.1 Build 14.04.013 software was used to perform the model heat transfer simulation. After modeling the geometry and importing it into StarCCM+, it was necessary to perform some modeling treatments. These treatments include extracting the internal volume filled by the radiator fluid and changing model settings in order to simplify the geometry. The simplification was done through the creation of a vertical symmetry plane, whose purpose was to reduce the computational cost of the simulation. The simulation was performed using only the fluid volume, due to the fact that the computational resources available were not sufficient to simulate the solid and the fluid volume in an adequate time.

As a next step for the modeling treatment, the radiator surfaces were then divided between the fluid inlet and outlet surfaces, walls (where heat exchange occurs) and the symmetry surface. After surfaces definition, it was possible to proceed with the mesh generation process. For this process, polyhedral elements were used in sets with layers of prismatic elements. These layers were defined to be used in areas near the fluid where the heat exchange with the solid part of the radiator occurs.

The vertical plate channel region was refined due to the importance of representing the flow and heat transfer from the fluid in this region, and also because of its dimensions and curvatures, which were much smaller than the rest of the model. The generated mesh was composed of 18424224 elements.

Figure 4 presents a volumetric mesh in the fluid inlet region, while Figure 5 presents the mesh elements from the vertical plate section.

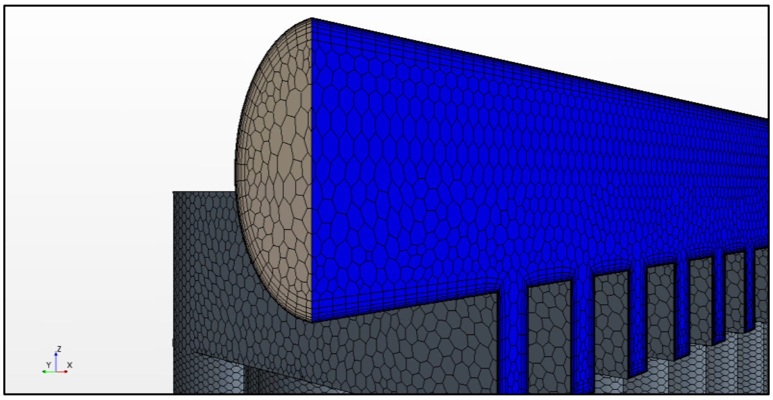

Figure 4. Volumetric mesh of the inlet region.

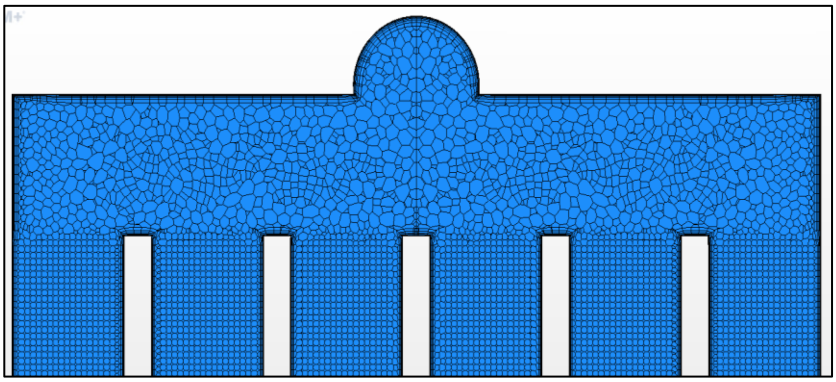

Figure 5. Vertical plate volumetric mesh.

The same process was used to generate the models with the proposed modifications, including the radiator with 26 vertical plates, the radiator with 28 vertical plates and the one with the outlet tube in an inverted position.

3.2 Boundary Conditions and Convection Coefficients

It was necessary to define the boundary conditions of the problem in order to perform the simulation. Boundary conditions can be considered as the basis for CFD software calculations. The temperature and speed values of the oil at the inlet radiator tube and the ambient temperature could be obtained through experimental tests carried out by the manufacturer and were used as initial parameters. Table 1 gathers the boundary conditions obtained through the experimental result.

Table 1. Boundary conditions experimentally obtained.

Variable	**Value**
Inlet velocity (v)	0.2 m/s
Inlet temperature (T)	71.5 °C
Environment temperature (T_∞)	22 °C

The vertical plate external convective coefficient and the horizontal tube external convective coefficient were calculated and determined as boundary conditions for the model. For this calculation process, different correlations available in the literature were used, so that each one of them was associated with a specific geometrical part of the equipment and its flow condition. Incropera *et al*. (2014) says that, considering a situation of free convection over a long horizontal cylinder, it is possible to establish a value for the average Nusselt number through the correlation provided by Eq. (1). In this equation, this dimensionless is related to the entire circumference of an isothermal cylinder and its value is influenced by the development of the outer fluid flow boundary layer.

$$\overline{Nu}_D = \left\{0{,}60 + \frac{0{,}387 Ra_D^{1/6}}{\left[1+(0{,}559/Pr)^{9/16}\right]^{8/27}}\right\}^2 \tag{1}$$

where $\overline{Nu}_D$, Ra_D and Pr are the average Nusselt number, Rayleigh number and Prandtl number respectively.

The *Rayleigh* number is defined by Eq. (2).

$$Ra_D = \frac{g\beta(T_s - T_\infty)D^3}{\nu\alpha} \tag{2}$$

where Ra_D, g, β, ν, α, D, T_s and T_∞ are the Rayleigh number, Prandtl number, gravity acceleration, thermal expansion coefficient, kinematic viscosity, thermal diffusivity, tube external diameter, tube superficial temperature and environment temperature respectively.

The calculated values are exposed in Table 2.

Table 2. Calculated values for horizontal tube external convective coefficient determination.

Variable	**Value**
$\beta^{(1)}$	3.17E-03 K^{-1}
$\nu^{(1)}$	2.74E-02 W/(m.K)
$\alpha^{(1)}$	2.47E-05 m^2/s
D	8.8E-2 m
T_s	334.95 K
T_∞	295.15 K
Ra_D	1.97E+06
$\overline{Nu}_D$	1.76E+01
$\bar{h}$	5.5 W/(m^2K)

(1) air proprieties at 315.05 K

According to Incropera *et al*. (2014), vertical or inclined channels formed between parallel plates and opened to the environment at their opposite ends are common geometry found in many equipment submitted to natural convection. For this case, the correlation was developed according to Eq. (3), considering symmetrically heated isothermal plates, where the length of the plates is much greater than the distance between them.

$$\overline{Nu}_S = \frac{Ra_S(S/L)}{24} \quad (3)$$

where $\overline{Nu}_S$, Ra_S, S and L are the average Nusselt number, Rayleigh number, distance between plates and plate length respectively.

Eq. (4) present the Rayleigh number for this correlation.

$$Ra_S = \frac{g\beta(T_S - T_\infty)S^3}{\nu\alpha} \quad (4)$$

where Ra_S, g, β, ν, α, S, T_s and T_∞ are the Rayleigh number, Prandtl number, gravity acceleration, thermal expansion coefficient, kinematic viscosity, thermal diffusivity, distance between plates, plate superficial temperature and environment temperature respectively.

All the calculated values are disposed in Table 3.

Table 3. Calculated values for the vertical plate external convection coefficient determination.

Variable	Value
$\beta^{(1)}$	3.17E-03 K^{-1}
$\nu^{(1)}$	2.74E-02 W/(m.K)
$\alpha^{(1)}$	2.47E-05 m²/s
S	2E-2 m
L	2.2 m
T_s	334.95 K
T_∞	295.15 K
Ra_S	2.3E+04
$\overline{Nu}_S$	8.73
$\bar{h}$	12 W/(m²K)

(1) air proprieties at 315.05 K

4. RESULTS

The simulations were performed using a 6-core Intel® Xeon® E5-1650 v3 computer with 3.5 GHz, 32 GB RAM memory and Windows 8.1 Pro 64bit operational system. The variables used to verify the convergence of the simulation results were: the oil temperature at the radiator outlet tube, the heat flow in three plates (the one closest to the inlet, the one in the middle of the radiator and the one farthest from the inlet region) and residual errors.

4.1 Temperature

After 2000 iterations and about 34 hours of simulation processing, the oil temperature at the base model radiator outlet region stabilized at 56.8 °C, which represents an increase of 4.7 °C (9.0%) compared with the experimental result of the manufacturing company commercial model. Through the analysis of the final result, it was possible to verify that the developed model represented a good approximation of the commercial radiator model, in terms of heat exchange performance. This statement could be inferred due to the acceptable difference between the outlet oil temperatures, one obtained from experimental data and the other from the simulation result.

Figure 6 shows the base radiator model temperature distribution.

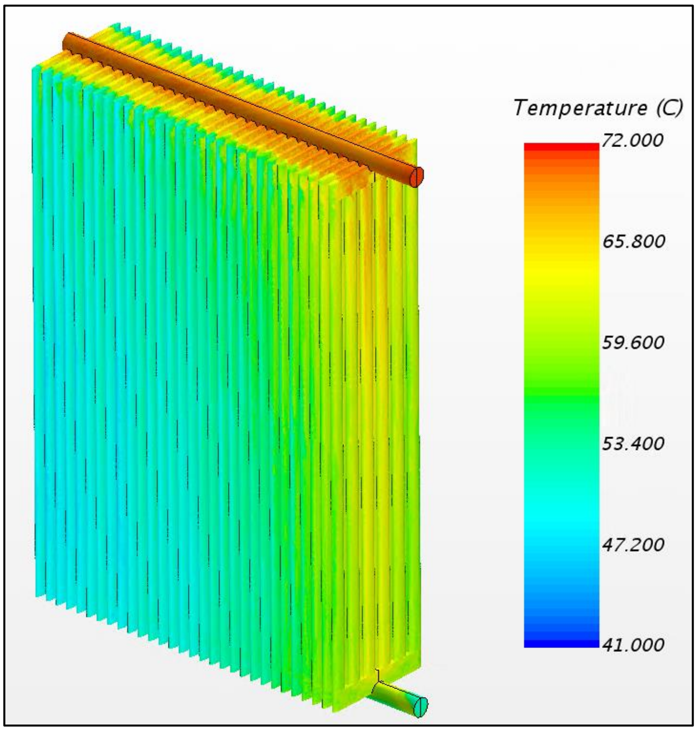

Figure 6. Radiator base model temperature distribution.

Simulation results of the 26 plates model converged to a temperature of 58.3 °C after 1500 iterations and approximately 24 hours duration, showed an increase of 1.5 °C (2.8%) compared to the base model, while the simulation results of the 28 plates model converged to a temperature of 57.8 °C after 1880 iterations and approximately 31 hours of duration, resulted in a 1.0 °C (1.9%) increase over the base model.

The volume of the base radiator is 0.32 m^3 and the volumes of radiators with 26 and 28 plates are 0.28 m^3 and 0.30 m^3 respectively. The reduction in the number of vertical plates, in the case of the 26 plates model, represented a 12.5% reduction in the radiator volume and in the case of the 28 plates model, a 6.3% reduction in relation to the base model.

A comparison of percentage increase in the oil temperature at the radiator outlet with the percentage reduction in the radiator volume, between the two vertical plate reduced models and the base model, showed that despite the reduction in the number of plates presented a worsening in the equipment thermal exchange, it also presented a significant reduction in the amount of material needed for its manufacture, as indicated in Table 4. The percentage reduction in volume is considerably greater than the percentage increase in the oil exit temperature, which justifies the proposal to change the geometry of the equipment if the oil temperature increase does not exceed the limit established for the cooling system operation. The reduction in the number of plates of the radiator consists in less raw materials necessary to its manufacture process, which can be a valuable statement in terms of manufacturing costs. Beyond this fact, reducing the number of plates implies in the radiator size reduction, making it more affordable to be installed in confined spots or in which there are difficult assemblies to be attached.

Table 4. Percentage increase in temperature and percentage reduction in volume of different radiator configurations compared to the base model (30 plates).

Model	Temperature increase (%)	Volume reduction (%)
26 Vertical Plates	2.8	12.5
28 Vertical Plates	1.9	6.3

The model simulation results, considering the inverted outlet position, converged to a temperature of 59.1 °C after 2000 iterations and approximately 35 hours of simulation, representing a 2.3 °C (4.0%) increase over the base model.

Changing the radiator fluid outlet position did not promote a reduction in oil temperature nor a better temperature distribution, as can be seen in Figure 7.

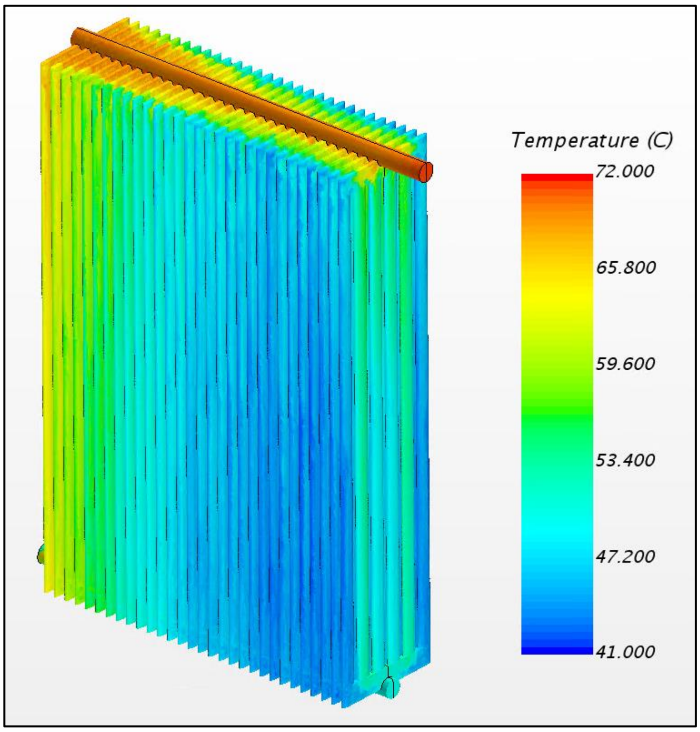

Figure 7. Inverted outlet position radiator temperature distribution.

The final outlet fluid temperature of all simulations is indicated in Table 5.

Table 5. Outlet fluid temperatures.

Model	**Outlet fluid temperature (°C)**
Base	56.8
26 Vertical Plates	58.3
28 Vertical Plates	57.8
Inverted outlet position	59.1

4.2 Velocity

The purpose of the radiator outlet position change was to permit a better fluid distribution inside the equipment ducts, creating a more homogeneous velocity profile for the oil flow. The radiator outlet position change presented a better distributed velocity profile along the entire upper horizontal tube. The velocity at the end of the upper horizontal tube, considering the base model, was close to zero. While, considering the model with the inverted outlet tube position, a more homogeneous behavior along the entire length of the tube could be noticed.

Figure 8 shows the oil velocity in the base model upper tube.

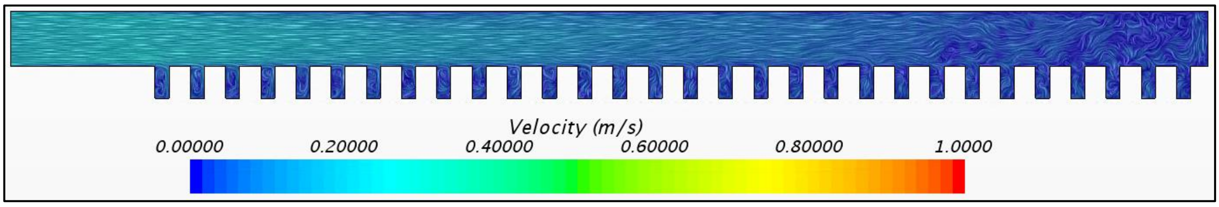

Figure 8. Oil velocity profile from the base radiator horizontal upper tube.

The Figure 9 shows the oil velocity in the inverted outlet radiator model upper tube.

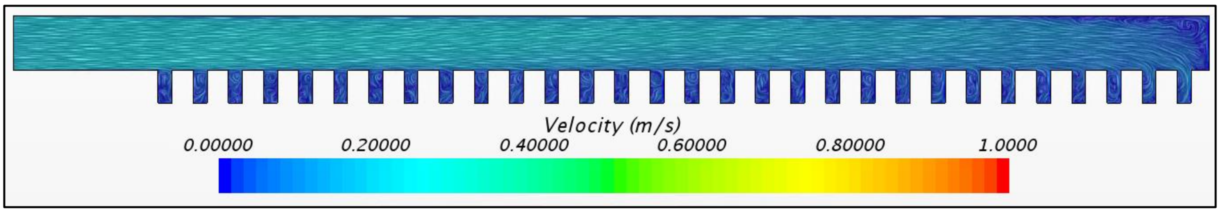

Figure 9. Oil velocity profile from the upper tube of the inverted outlet radiator model.

Fluid recirculation zones could also be observed, in a higher scale at the base model horizontal upper tube, as it gets close to the last vertical plates inlet flanges. According to Nabati, Mahmoudi and Etheran (2009), fluid recirculation occurs inside radiators whenever pressure increase at the end of a radiator block. That phenomena consequently prevents enough fluid flow through the last vertical plates, which can possibly cause a negative impact in the heat exchange process.

Now considering the horizontal tube from the bottom, the base model tube velocity presented a slightly more homogeneous behavior than the inverted outlet tube model. Figure 10 shows the velocity profile at the base model bottom tube.

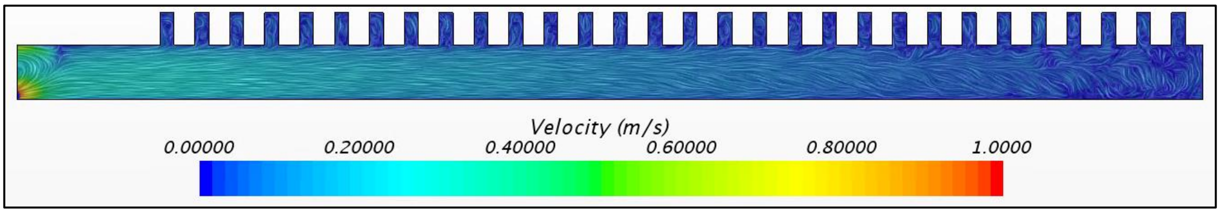

Figure 10. Oil velocity profile from the base radiator model bottom tube.

Figure 11 shows the velocity profile of the bottom tube from the inverted outlet radiator model.

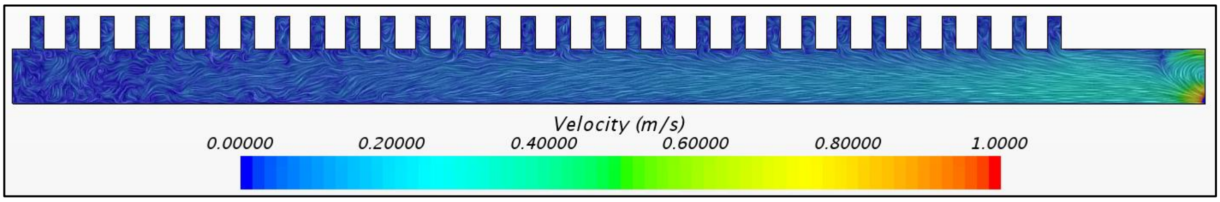

Figure 11. Bottom tube oil velocity profile of the inverted outlet radiator model.

5. CONCLUSION

The base model simulation result showed that the developed model can be considered as a good approximation of the commercial model on which it was based, having achieved a variation of less than 10% of the expected value. The proposal in the reduction of the vertical plates number proved to be viable, as the percentage increase in the oil outlet temperature was less than the percentage reduction in the volume of the radiator structure, both for the 26 plates model and the 28 plates model. A 12.5% reduction, in the case of the 26 plates model, and a 6.3% reduction, in the case of the 28 plates model, presented a considerable cost reduction in the manufacture of the radiator, due to the reduction in the amount of raw material needed. Also, the reduction in the equipment dimensions could permit more possibilities of arrangement and installation of the radiator if more space is necessary.

Changing the radiator fluid output position did not promote a reduction in oil temperature nor a better temperature distribution. This result may be related to the increase in oil velocity in the upper horizontal tube, which despite having made the velocity more homogeneous, negatively impacted in the fluid distribution between the vertical plates, concentrating the largest amount of fluid in the plates near the outlet of the radiator.

6. REFERENCES

Çengel, Y. A. and Ghajar, A. J., 2012. *Heat and Mass Transfer: A Practical Approach (in Portuguese).* AMGH. Porto Alegre, Brazil.

Holman, J. P., 1983. *Heat* Transfer *(in Portuguese)*. McGraw-Hill, São Paulo, Brazil.

Incropera, F. P., Bergman, T L., Lavine, A. and Dewitt, D. P., 2014. *Fundamentals of Heat and Mass Transfer (in Portuguese)*. LTC, Rio de Janeiro, Brazil.

Moukalled, F., Mangani, L. and Darwish, M., 2016. *The Finite Volume Method in Computational Fluid Dynamics*. Springer, Horw, Switzerland.

Nabati, H., Mahmoudi, J. and Ehteram A., 2009. “Heat transfer and fluid flow analysis of power transformer's cooling system using CFD approach”. *Chemical Product and Process Modeling*, Vol 4, No. 1.

Radakovic, Z.R. and Sorgic, M.S., 2010. “Basics of detailed thermal-hydraulic model for thermal design of oil power transformers”. *IEEE Transactions on Power Delivery*, Vol 25, No. 2, pp. 790-802.

Reis, M. N. E., Brasil, M. V., Costa, P. H. S. and Paula, T. H. B., 2018. “Numerical Simulation to Analyze the Radiator of a Power Transformer”. In *Proceedings of the 17th Brazilian Congress of Thermal Sciences and Engineering – ENCIT 2018*. Águas de Lindóia, Brazil.

Rodriguez, G., Garelli, L., Storti, M., Granata, D., Amadei, M. and Rossetti, M., 2016. “Numerical and experimental thermo-fluid dynamic analysis of a power transformer working in ONAN mode”. *Applied Thermal Engineering*, Vol 112, pp. 1271-1280.

Versteeg, H. K. and Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics*. Pearson Education Limited, Harlow, England.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.