International Journal of Heat and Fluid Flow 69 (2018) 62–72

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Numerical and experimental thermofluid investigation of different disc-type power transformer winding arrangements

F. Torriano*,[a], H. Campelo[b], M. Quintela[b], P. Labbé[a], P. Picher[a]

[a] Institut de recherche d'Hydro-Québec, Varennes, Canada
[b] EFACEC Energia, Porto, Portugal

ARTICLE INFO

Keywords:
CFD
CHT
Disc-type windings arrangement
ON cooling
Power transformer
Experimental scale model

ABSTRACT

In this paper, measurements are carried out on four different washer arrangements of an ON disc-type power transformer winding scale model. The experimental setup comprises a closed cooling loop with all the main components generally found on a power transformer and it is equipped with both thermal and flow sensors. Moreover, 3D Conjugate Heat Transfer simulations of the entire cooling circuit are performed using a commercial CFD solver and the computed oil flow rates and winding temperatures are compared with the experimental data for both uniform and non-uniform heat loss distributions. The experimental results show that the reduction of the number of washers in the tested scale model winding increases the total oil flow rate but this effect is overridden by a higher flow maldistribution in the radial ducts of a pass. Thus, the discs temperatures increase with the removal of washers and this effect is particularly marked for a non-guided winding arrangement where an almost stagnant flow is observed in several radial cooling ducts. The CFD results show the same trend but the numerical model consistently underpredicts the total oil flow rate circulating in the closed cooling circuit. This underestimation by the CFD model causes, for certain winding arrangements, significant errors in the evaluation of the average and hot-spot temperatures. For this reason, numerical simulations with a reduced computational domain (i.e., winding region only) are also performed by specifying the measured oil flow rate and temperature as inlet boundary conditions. In this case, the accuracy of the numerical model is significantly improved as the predicted average and hot-spot winding temperatures are within 3 °C of the corresponding measured values. This result is reassuring since the majority of published numerical thermofuid studies on transformer windings are performed on the windings region only and boundary conditions are specified at the inlet, thus avoiding the simulation of the entire cooling loop.

1. Introduction

Power transformers are critical components of electrical grids and for this reason both manufacturers and utilities constantly aim to improve their performance and life expectancy. Although transformers are very efficient machines ($\eta > 99\%$), heat is still generated by their active parts, mainly due to the resistivity of the conductors and to eddy currents. These losses cause a thermal heating of the active components and in order to ensure a proper functioning of the transformer, their temperature must be kept below a critical level by circulating a dielectric fluid (usually naphthenic mineral oil) through the windings. This fluid is then directed towards a series of fin-plate radiators which dissipate the heat to the ambient air. Since the flow dynamics in the transformer is one of the main factors dictating its performance, it is essential to correctly evaluate it as well as the inherent heat transfer mechanisms. In order to do so, engineers have used in the past two approaches, namely numerical simulations and experimental measurements. The numerical approach mainly relies on two techniques that are referred to as Thermal Hydraulic Network Models (THNM) and Computational Fluid Dynamics (CFD)(Cig, 2016). THNM methods generally rely on empirical and analytical expressions to evaluate flow losses and convection heat transfer coefficients, while CFD is based on the solution of the governing differential equations for a fluid flow. Furthermore, CFD provides more detailed information on the pressure, velocity and temperature fields but it is more computationally expensive and time consuming compared to THNM. Therefore, it is generally used by manufacturers in the last steps of the design process to validate the data previously obtained with simpler numerical methods or to provide databases from which empirical formulae can be extracted and implemented in THNM codes.

Even though advanced numerical methods such as CFD have reached a good level of accuracy, experimental validation is still needed

* Corresponding author.
E-mail address: torriano.federico@ireq.ca (F. Torriano).

https://doi.org/10.1016/j.ijheatfluidflow.2017.11.007
Received 16 June 2017; Received in revised form 20 October 2017; Accepted 25 November 2017

Nomenclature

c_p = specific heat at constant pressure ($J{\cdot}kg^{-1}{\cdot}K^{-1}$)
D_h = hydraulic diameter (m)
h = heat transfer coefficient ($W{\cdot}m^{-2}{\cdot}K^{-1}$)
H_{disc} = axial length of disc (m)
H_{duct} = axial length of horizontal duct (m)
g = gravity vector ($m{\cdot}s^{-2}$)
k = thermal conductivity ($W{\cdot}m^{-1}{\cdot}K^{-1}$)
$L_{ext.\ duct}$ = radial length of outer duct (m)
$L_{int.\ duct}$ = radial length of inner duct (m)
$\dot{m}$ = mass flow rate ($kg{\cdot}s^{-1}$)
$\dot{m}_0$ = experimental mass flow rate in winding configuration with no pass ($kg{\cdot}s^{-1}$)
$\dot{m}_1$ = experimental mass flow rate in winding configuration with 1 pass ($kg{\cdot}s^{-1}$)
$\dot{m}_2$ = experimental mass flow rate in winding configuration with 2 passes ($kg{\cdot}s^{-1}$)
$\dot{m}_3$ = experimental mass flow rate in winding configuration with 3 passes ($kg{\cdot}s^{-1}$)
p_{avg} = average pressure (Pa)
r = radial coordinate (m)
Ra = Rayleigh number (m)
R_{ic} = inner cylinder radius (m)
S_E = heat source term ($W{\cdot}m^{-3}$)
t = time (s)
T = temperature (°C)
$T_{avg\ oil\ inlet}$ = mean oil temperature at the inlet of the winding (°C)
$T_{avg\ wdg}$ = average winding temperature (°C)
$T_{hot-spot}$ = maximum winding temperature (°C)
$T_{oil\ inlet}$ = oil temperature at the inlet of the winding (°C)
U = velocity vector ($m{\cdot}s^{-1}$)
U_b = bulk velocity ($m{\cdot}s^{-1}$)
U_r = radial velocity component ($m{\cdot}s^{-1}$)
U_z = axial velocity component ($m{\cdot}s^{-1}$)
z = axial coordinate (m)

Greek symbols

θ = angular coordinate (deg)
μ = dynamic viscosity ($kg{\cdot}m^{-1}{\cdot}s^{-1}$)
ν = kinematic viscosity ($m^2{\cdot}s^{-1}$)
ρ = density ($kg{\cdot}m^{-3}$)
ρ_{ref} = reference density ($kg{\cdot}m^{-3}$)

Subscripts

alu = aluminum
f = oil
he = heating element

since the thermofluid analysis of power transformers remains quite complex. In fact, as it has been demonstrated in previous studies (Torriano et al., 2010; 2012; Skillen et al., 2012; Wakil et al., 2006; Mufuta and Bulck, 2000; Wijaya et al., 2012), the modeling of the oil dynamics in a disc-type transformer winding is quite challenging due to the flow high sensitivity to several geometrical and operating parameters (ex.: duct size, number of discs/pass, flow regime, etc.). Moreover, as it has been shown by Kranenborg et al. (2008), for some windings arrangements and flow conditions, the hot streak can play an important role and must be correctly captured by the numerical model in order to accurately predict the hot-spot. A more recent numerical study from the same research group (Gustafsson et al., 2016), has also shown that, for low oil flow rates, relatively small temperature gradients at the inlet of a pass can strongly influence the oil flow rate distribution in the radial cooling ducts. Moreover, this study has emphasized the presence of a recirculating flow around the first disc of a pass for high oil flow rates which causes the hot-spot to be located in the upstream section of the pass. The same observation was made in a previous numerical investigation carried out by Campelo et al. (2016) where it was also shown that classic THNM methods can hardly predict such localized flow inversions.

On the other hand, experimental measurements require a considerable instrumentation effort and access to transformers in-situ is often difficult due to operational constraints. For these reasons, a laboratory scale model appears to be the best compromise to acquire, in a controlled environment, the data necessary to validate the numerical models. This hybrid approach based on numerical modeling and experimental measurements has been embraced by some authors in the past. For example, Rahimpour et al. (2007) equipped an ON cooled transformer disc-type winding with 11 thermocouples and compared the measured discs temperatures with the values obtained from their THNM model. The overall temperature distribution was well computed by the numerical model but some discrepancies were observed locally. A similar investigation was carried out by Mufuta (1999), where a simplified scale model of a disc-type winding consisting of a water-filled box with two columns of six heated blocks arranged in-line inside was built. The comparison between the 2D CFD simulations and the experimental data showed a maximum difference of 3 °C for the discs temperature and a maximum relative error of 20% and 400% for the vertical and horizontal oil velocities respectively. More recently, Zhang et al. (2008) used a disc-type winding setup to derive heat transfer correlations along the discs surfaces and to calibrate their THNM. A similar strategy was adopted in Lee et al. (2010), where the authors first performed 2D CFD simulations of a zig-zag winding to obtain correlations for the pressure drop and heat transfer and then conducted measurements on a 70 heated discs setup to validate their expressions. The comparison showed that the theoretical correlations can predict the pressure drop and winding temperature with 10% accuracy. In Schmidt et al. (2013) the authors computed the conductors temperatures of a disc-type winding with a 2D CFD model and noted a difference of 10% and 25% with the measured values at high and low oil flow rates respectively. Finally, Yatsevsky (2014) has performed a 2D CFD simulation of the windings, the tank and radiators of a 210 MVA ONAN transformer and has compared the windings and oil temperatures with the experimental data.

In 2012, a transformer winding scale model was designed and built at IREQ (Hydro-Québec's Research Institute) in order to acquire valuable experimental data that could be used to validate both CFD models and in-house THNM codes. The main advantages of a scale model compared to a transformer in-situ are that the operating conditions can be better controlled, it can be more easily instrumented and it is more flexible to design modifications. This vision was also shared by EFACEC Energia (a transformer manufacturing company), and for this reason a long-term R&D collaboration was undertaken in 2014 between the two organizations. This paper presents some of the results obtained during this partnership and pursues the work that was presented in Torriano et al. (2016), namely 3D CHT (Conjugate Heat Transfer) simulations of the complete cooling circuit (i.e., winding, tank and radiators) of the IREQ transformer winding scale model as well as experimental data for four different winding arrangements.

2. Experimental setup

Performing in-situ measurements can be quite challenging and it is often quite difficult to properly control the operating conditions. Moreover, the size of the equipment and its associated characteristics