Applied Thermal Engineering 36 (2012) 96–105

Contents lists available at SciVerse ScienceDirect

Applied Thermal Engineering

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

Numerical prediction of local hot-spot phenomena in transformer windings

Alex Skillen [a,*], Alistair Revell [a], Hector Iacovides [a], Wei Wu [b]

[a] Computational Fluid Dynamics Group, Modelling and Simulation Centre, School of MACE, The University of Manchester, UK
[b] Electrical Energy and Power Systems Group, School of EEE, The University of Manchester, UK

ARTICLE INFO

Article history:
Received 18 July 2011
Accepted 26 November 2011
Available online 6 December 2011

Keywords:
Power transformer
CFD modelling

ABSTRACT

In this paper, an open source CFD code is used for predicting the mixed-convection flow of coolant in a low-voltage transformer winding. A significant feature of these flows is the high Prandtl number of the dielectric coolant fluid. It is found that for the geometry under consideration, the CFD model predicts the presence of hot-plumes in some of the horizontal cooling ducts. This appears to be due to the highly non-uniform mass flow distribution around the winding, which arises as a result of an inhomogeneous temperature profile across the individual vertical ducts (a direct consequence of high fluid Prandtl number). These hot-plumes are particularly problematic since local heating can cause thermal degradation of the paper insulation. It is also found that there is a strong coupling between the flow in different passes, which is communicated via persistent hot streaks in the fluid being convected from one pass to the next. The importance of these hot streaks is apparent from the CFD results, yet they are unresolved by the lumped parameter based 'network models' preferred by industry, which assume perfect mixing and homogeneous flow quantities in all cooling ducts. In particular, these methods are unable to account for the persistence of these inhomogeneous features from one pass to the next. While it is expected that CFD approaches should provide higher detail than network modelling, the insight gained from the present simulations would suggest that significant changes to current industrial approaches might be necessary in order to correctly account for the observed phenomena.

1. Introduction

Power transformers incur energy losses during their operation due to the Joule heating effect. Both the primary electrical current, as well as secondary magnetic flux induced eddy currents, contribute towards this heating. Although the losses amount to only a small fraction (around 1%) of the transformer's power rating, they are sufficient to necessitate the use of assisted cooling. Coolant oil circulates through a closed-loop network of ducts with the aim of transferring heat energy from the electrically active discs (the heat source) to the ambient air, via radiators. Inadequate cooling can lead to the existence of regions of localised peaks in temperature known as 'hot-spots'. These hot-spots are particularly problematic for the layers of paper insulation, which can undergo rapid thermal ageing, drastically shortening transformer lifespan. Accurate prediction of temperature distribution throughout the transformer winding is therefore of great importance, so as to locate and understand the characteristics of local hot-spot regions during not only the operational period of a in-service transformer, but also the design phase of a new transformer.

Traditional methods of transformer winding thermal design have utilised lumped parameter (semi-empirical) models such as 'thermal-electric analogy' [1–3] and 'network models' [4–7]. These approaches provide fast-to-use approximations by introducing assumptions in order to simplify the analysis. However, details of oil flow and temperature patterns cannot be expected with these methods due to the low discretisation applied and the suitability of the underlying assumptions and empirical relations upon which they are based. Despite these obvious drawbacks, the network modelling approach has been proven useful for cases where an oil pump is used (as the flow rate is greater and the flow is generally monotonic). However, in the more general situation with moderate or low oil velocities, i.e. with 'Natural Oil' cooling modes (when a pump is not used), buoyancy forces become more influential and natural convection co-exists with, or even dominates over, forced convection. In these situations, the flow often becomes non-uniform, giving rise to pockets of local flow recirculation. The equations currently applied in the lumped parameter models may not be suitable in these scenarios and would thus need further development and calibration.

In the present study, Computational Fluid Dynamics (CFD) tools have been utilised in order to obtain a complete depiction of the oil flow and temperature distribution throughout a complete winding of 5 passes. The majority of previous CFD studies on transformer

* Corresponding author.
E-mail address: alex.skillen@manchester.ac.uk (A. Skillen).

1359-4311/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2011.11.054