Applied Thermal Engineering 63 (2014) 347–353

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Hydrodynamics and heat transfer in cooling channels of oil-filled power transformers with multicoil windings

Vitaly A. Yatsevsky*

National Academy of Sciences of Ukraine, Institute of Engineering Thermophysics, Department of Heat-and-Mass Transfer Modelling, St. Zhelyabova, 2a, 03680 Kyiv, Ukraine

HIGHLIGHTS

- The CFD model of the transformer with external cooling circuit has been developed.
- The model also includes all the cooling channels in the space between coils.
- The analysis of hydrodynamic processes has revealed self-organizing oil flow.
- The flow features have an influence on the thermal state of windings coils.
- Zones with the maximum temperature shift along height and toward the radial velocity.

ARTICLE INFO

Article history:
Received 21 March 2013
Accepted 26 October 2013
Available online 7 November 2013

Keywords:
Power transformers
CFD modelling
Self-organizing flow
Natural cooling
Oil flow
Thermal and hydrodynamic processes

ABSTRACT

The analysis of heat transfer and oil flow in power transformers at natural convection of cooling oil is carried out by numerical simulation of the fluid dynamic problem in axisymmetrical formulation (CFD-approach). The effects of self-organization structure of oil flowing in the form of the unidirectional flow in the groups of horizontal channels between winding coils have been revealed.

The flow features have an influence on the thermal state of winding coils of the transformer with the natural cooling system. At such oil flowing, in different channels, the heat transfer coefficient varies within the limits of 50–100 W/(m^2 K), and the radial component of velocity is changed over the range of $2.1 \cdot 10^{-4}$–$2.2 \cdot 10^{-3}$ m/s. Thus the region with maximum temperature values is removed towards oil flowing along the coil radius.

1. Introduction

The key factor in ensuring the high reliability and long-term operation of a power transformer is the effective removal of the energy which is inevitably released as heat in basic constructive elements – in the magnetic system, windings and other active parts (Fig. 1). One of the most important technical parameters is the temperature level of hot spots in windings, exceeding of which (more than 98 °C) causes the thermal destruction of the winding insulation. Location and temperature values of hot spots are unpredicted owing to the complex structure of oil circulation in numerous interconnected vertical and horizontal channels inside the oil-filled tank.

The problem related to CFD modelling of thermal processes in power transformers as well as special features of cooling fluid flowing in channels has been studied enough well in enough scientific works. For the last ten years more than a hundred and fifty publications have presented the models of different level of detailization and accuracy. The previous works have revealed that the modification and redistribution of fluid flow influence directly and largely on the temperature distribution in transformer component, and the distribution of winding temperature is more uniform when the flow rate of cooling fluid increases.

Authors of many works put an emphasis on horizontal channels critical zones that require a particular consideration.

The majority of CFD studies are concentrated on examining only separate parts of transformer winding, for example considering one pass.

In the present article, the general formulation has been carried out taking into account complete two windings, the core, the

* Tel.: +38 0681268268.
E-mail address: yatsevsky_v_a@ukr.net.

1359-4311/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.applthermaleng.2013.10.055

Nomenclature

b	width of vertical channels [m]
C_p	specific heat capacity [J/kg K]
g	gravity acceleration [m/s^2]
Gr	Grashof number [−]
h	width of horizontal channels [m]
k	empirical coefficient k in Eq. (4) [−]
P	pressure [Pa]
Pr	Prandtl number [−]
Q	heat source density (heating power per unit volume) [W/m^3]
R	radius [m]
r, z	radial and axial cylindrical coordinates [m]
Re	Reynolds number [−]
T	temperature [K or °C]
V_r	velocity components in radial direction r [m/s]
V_z	velocity components in axial direction z [m/s]

Greek symbols

τ	time [s]
ρ	density [kg/m^3]
μ	dynamic viscosity [kg/m s]
φ	tangential cylindrical coordinate [rad]
λ	thermal conductivity [W/(m K)]

Indices

amb	ambient
in	inner
oil	oil
out	outer
v	per unit volume

transformer tank and the external cooling radiators. Latest similar research works are considered below.

Work [1] gives CFD model for thermal investigation of air ventilation in underground transformer substations and presents the patterns of air flow and temperature distribution inside the substation as well as shows a stagnant zone as a zone with still (static) air.

The approach to optimization of mutual configuration of coils (disks) and cooling channels is developed by Smolka [2] for effective cooling of a dry-type transformer. The optimization procedure combines the methods of computational fluid dynamics (CFD) and genetic algorithm to find optimum diameter of channels and coils. In the computer CFD model proposed by Smolka in Ref. [2], the thermal properties of coils and core are considered as anisotropic and temperature-dependent quantities. The heat sources (heating power) for elements are computed by coupled CFD and EMAG (electromagnetic) models. A similar approach is used in the present article.

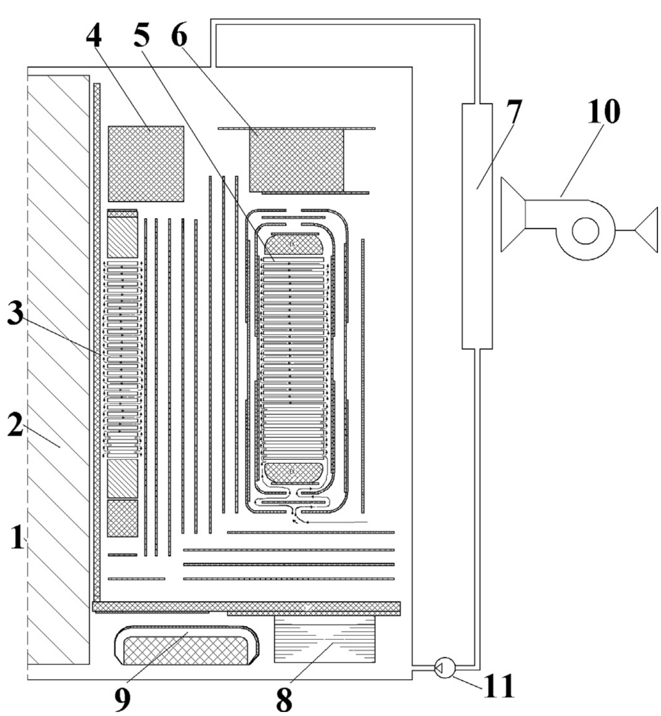

Fig. 1. The scheme of 210 MV A power transformer: 1 – axis of symmetry; 2 – transformer core; 3 – lower voltage winding (LV); 4 – pressing ring above LV winding; 5 – high-voltage winding (HV); 6 – pressing ring above HV winding; 7 – radiator of external cooling subsystem; 8 – shunt under HV winding; 9 – shunt under LV winding; 10 – fan; 11 – oil pump.

The article by Torriano et al. [3] presents the following results. The data obtained with simplified models differ appreciably from the results of the conjugate heat transfer model. The windings having separate copper wires with paper insulation which form a disk are studied. Unfortunately, this approach is extremely time-consuming and complicated for implementing as for many types of complex-structured wires (transposed wires or subdivided wires).

Work [4] (a subsequent paper by the same authors) gives the results of 3D simulation of coupled heat transfer and hydrodynamic flow in a disk-type low-voltage winding of distribution transformer 66 MVA–225/26.4 kV ONAN/ONAF. The low-voltage winding contains 78 disks divided into four passes. All the passes, except the first one, have 19 disks each. The first pass consists of 21 disks, but has an additional block-washer between the second and the third disks. The 3D results are compared with the similar computations by 2D model. It show be noted that 3D model includes all geometrical parts (i.e., sticks, intersticks, duct spacers and oil washers). The comparison is made for the homogeneous and inhomogeneous loss distributions in the winding. The computer models are realized by commercial finite-volume CFD code Ansys-CFX v12.1. As computational results show, the significant three-dimensional effects take place owing to availability of strips and spacers in the channel and, therefore the flows in cooling channels cannot be considered as completely axially symmetric. The authors of the article [4] emphasize that although 3D computations are very valuable, they need high-speed computer facilities. For this reason, the improved 2D models are proposed.

The CFD Code_Saturne developed by Électricité de France (EDF) is used by Skillen et al. in Ref. [5] for studying mixed convection for axisymmetric (2D) problem of cooling flows in a low-voltage transformer winding. Only region with fluid is simulated and the simplified (idealized) 3D model of windings is examined. The computations are realized for both a complete winding with five passes (the winding has 78 disks) and a separate pass. The CFD model discovers hot plumes in some horizontal cooling channels. The radial flows differ in direction for each horizontal channel. As revealed, the solution varies depending even on small deviation of the inlet velocity of the mass flow. That shows a highly unstable flow configuration. It is believed that all flows run in several channels where the cross-flow of fluid are negligible (e.g. less than 1% of inlet flow velocity). This causes the high temperature of disk surface and local hot-spots. The vector of gravitational force and then radial cooling flows in the passes and hot plumes are left out of account.