Predicting Hot Spots Using **Multiphysics**

Software coupling improves longevity and decreases power loss for electrical transformers.

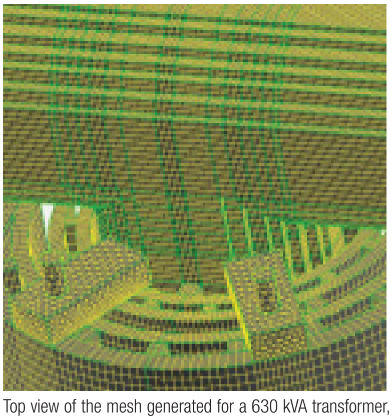
By Jacek Smołka and Andrzej J. Nowak, Silesian University of Technology, Gliwice, Poland

Electrical transformers transfer current from one circuit to another. Their primary purpose is to reduce the voltage in the electrical energy that is being delivered in order to supply devices at appropriate voltage level. Transformers normally consist of multiple sets of coils, also known as windings, which wrap around a ferromagnetic core. The coils are responsible for either inducing a magnetic field or assuming a transferred, induced current created by that magnetic field. This process results in high temperatures (hot spots) in the core and coils, which directly influence power losses and durability. While these components normally are cooled using natural convection, in the case of three-phase Y-Y dry-type power transformers used in mining applications, the transformer is further hermetically sealed in a tank. This creates a challenging situation when trying to encourage heat dissipation from the transformer. Simulation can be used to predict not only the high temperatures, or hot spot locations, that occur in the core and coil regions, but also to address cooling of the overall unit.

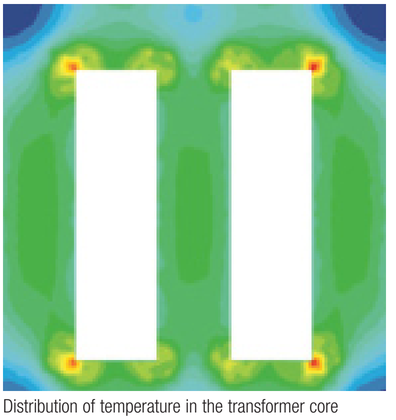
To accurately predict the temperature field, one must take into account the multidisciplinary nature of the physics involved. The fluid flow around the components is responsible for thermal management, whereas the electromagnetic field interaction with the components is both affected by and a primary cause of heating. To address these challenges, an effective numerical analysis must involve both computational fluid dynamics (CFD) and electromagnetic solutions. Combining FLUENT software with ANSYS Emag technology in an iterative, coupled manner provides that solution.

The coupling starts with a FLUENT calculation of the flow and temperature field around the device and a thermal field in the solid regions. The mathematical model uses the appropriate governing equations, source terms, boundary conditions and material properties specific to the device in question. The thermal conductivities are defined as anisotropic, since they have a preferred direction, for the core and coil elements. In addition, the coils' thermal conductivities were determined in a previous numerical analysis containing just the coils to be utilized in this main CFD model. After the FLUENT solver converges, the resulting temperature field is transferred to the electromagnetic solver to define the winding's temperature-dependent electric resistivities for the next solution stage.

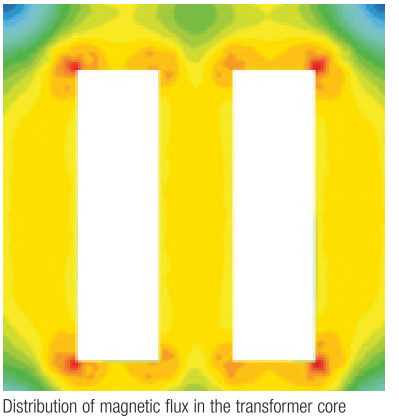
In the ANSYS Emag electromagnetic solver, the governing equations and the boundary conditions are specified to compute both the coil and the core transformer losses. The local core losses are calculated using a Steinmetz-based equation approach, while the coil losses, also known as



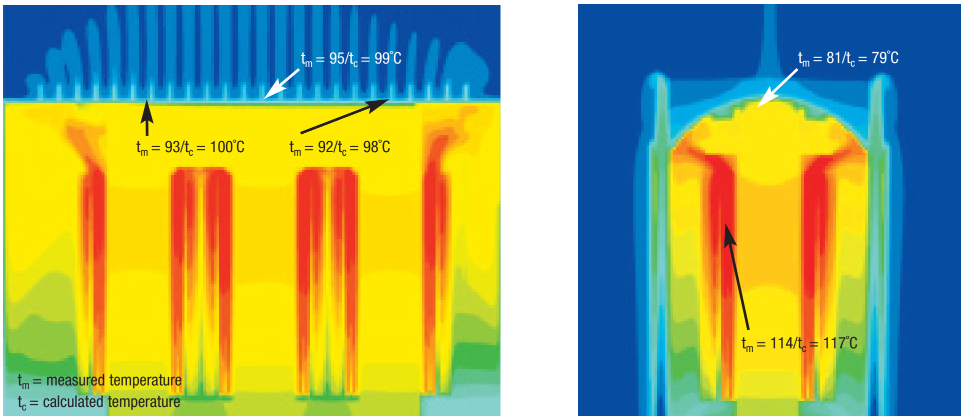
Top view of the mesh generated for a 630 kVA transformer, including the core, coils, screws and locating pads



Distribution of temperature in the transformer core (Locations of highest temperatures are shown in red.)



Distribution of magnetic flux in the transformer core (Areas of highest flux are shown in red.)



Simulation results for temperature field within a 630 kVA transformer station for a short-circuit test scenario are shown along the center of the core (left) and across the core (right).

Joule heat, are determined on the basis of a local current density and a local electrical resistivity. The electrical resistivity is defined as a temperature-dependent quantity using the FLUENT temperature field for its determination. As a result of this calculation, a local field of the transformer losses from the coils and core is obtained.

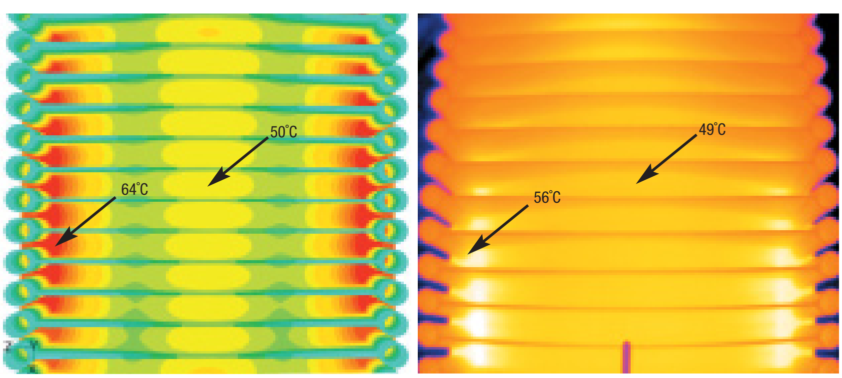
The calculated losses on an individual mesh cell basis are then transferred back to the CFD analysis, through the use of FLUENT user-defined functions (UDFs), as heat sources in the energy equation and the entire analysis process is reiterated. The cycle is repeated until the variations of the updated quantities become negligible. The typical case took two to three iterations between the FLUENT solver and the ANSYS Emag solver to reach convergence.

The largest transformer considered was a 630 kVA unit with a 2.13 cubic meter tank. The tank and transformer itself were naturally cooled by the surrounding air only. To effectively remove the heat from inside the tank, natural convection on the external walls of the station was enhanced by the addition of 25 flat fins welded to the top surface. Additionally, the surface area of the side-walls of the station was increased by the addition of 50 cooling pipes.

As was expected, free convection played a dominant role in the heat removal process. However, in the real device, the heat from the coils and the core was also dissipated via conduction to the transformer tank. In order to investigate the influence of the heat conduction on the total heat removal, the geometrical model included very small elements, such as the core screws and the coil locating pads. In order to accurately represent these components in the simulation, the generation of a fine mesh in GAMBIT software near these geometries was required. For the CFD computations, a mesh with 7.6 million elements was used in conjunction with parallel computing, which enabled the calculation of a solution in a practical amount of time.

To validate the numerical model, experimental transformer temperature tests for short-circuit, open-circuit and underrated parameter cases were performed according to the current European Standards for dry-type transformers. During the tests, wall temperatures were measured at selected points on transformer elements as well as on the overall external tank surfaces. In addition, information on the temperature field was captured for internal and external air surrounding both the device and the tank. The numerical results obtained confirmed that the prediction of the temperature distribution for the analyzed transformers and their surroundings was accurate.

The multiphysics mathematical model and procedures developed using coupled FLUENT and ANSYS Emag software have been applied to transformer units of different internal construction, positions in the tank and cooling system configurations. For each of the dry-type transformers considered, a number of potential changes were suggested to enhance the most effective heat transfer mode and lead to hot spot reduction. Moreover, the procedures developed for this group of devices can be easily extended to the more common oil-cooled devices by simply redefining the coolant material properties. ■



Computational (left) and experimental (right) results show temperature distribution on the external fins and top surface of a 630 kVA transformer tank for a short-circuit scenario.