21, rue d'Artois, F-75008 PARIS
http : //www.cigre.org

CIGRE US National Committee
2016 Grid of the Future Symposium

CFD Simulation on Forced Air Cooled Dry-type Transformers

W. WU
ABB Inc.
USA

SUMMARY

Dry-type transformer applications are growing in transformer market since the technology is non-flammable, safer and environmental friendly. Since in a dry-type transformer there is no oil present as dielectric insulation as well as cooling medium, the unit size is normally larger and material cost therefore becomes higher. On the other hand, for more compact dimensions, solid insulation is applied instead, which will furthermore constrain thermal convection and lower down cooling performance. In this context how to design a dry-type transformer with well-balanced dimensions and cost, dielectric and thermal performances becomes greatly important for a transformer manufacturer. In particular, fans are often used to enhance the cooling of the transformers especially during overloading period. In order for optimizing the cooling design of a dry-type transformer with forced air cooling fans, the present paper introduces computational fluid dynamics ("CFD") simulation tool that offers the potential of greater flexibility and accuracy for transformer cooling design compared with traditional calculation approaches. By validating the simulation results with the measurement values obtained from laboratory experiments, the CFD models show acceptable accuracies; thus the verified technology can be employed for the transformer products' cooling design optimization.

KEYWORDS

CFD, simulation, thermal model, dry-type transformer, forced air cooling, temperature rise

wei.wu@us.abb.com

1. **Introduction**

Transformers are key components in electric transmission and distribution networks and they convert electricity from one voltage to anther voltage, either of lower or higher values. Among various transformer technologies, dry-type transformers avoid using oil as dielectric insulation or cooling medium and as such becomes a non-flammable, more environmental friendly alternative to liquid-immersed technologies. Because of the advantages, dry-type transformers gradually gained market share in special environments and applications which require high standards of safety and reliability, such as urban areas, buildings and marine ships etc.

On the other hand, one of the disadvantages of a dry-type transformer is that it is typically of larger dimensions and more material cost, compared with its liquid-immersed competitors, in order for having sufficient cooling and keeping the insulation thermal degradation lower than a required limit. IEEE and IEC standards both have winding temperature rise requirements for dry-type transformers and according to the standards products manufactured need to pass heat-run tests before they can be equipped for operation. Therefore one of the primary tasks of a manufacturer is to predict temperature rises at design stage and deliver products with sufficient cooling to ensure successful heat-run tests.

Dry-type transformers are usually cooled by either natural or forced air flow; transformers that will withstand potential overloading are often equipped with fans to drive forced air flow that provides better cooling. Figure 1 shows a forced air cooled transformer unit with fans equipped below the three-phase coils. In order to optimize these products, tools to predict the temperature rises become of great necessity. Along with the computer technology development, numerical modelling is gradually applied to predict the transformer temperature rises. There are generally two numerical approach categories; either thermal network models [1-5] or numerical solutions which incorporate a degree of computational fluid dynamics (CFD) [6-10]. Thermal network models, based on the analogy with electric circuits, abstract the phenomena of thermodynamics and fluid dynamics in the computation domain into interconnected so-called ‘lumped elements’ and physic principles are encapsulated into the elements for numerical solutions. In comparison, CFD employs much higher level of discretization and, consequently, consumes more computational resources (CPU and memory) and time. Considering the flexibility of CFD simulations and that they can reveal more details of the air and heat flow phenomena, their application for transformer design and optimization is expanding.

Figure 1: A forced air cooled dry-type transformer.

2. **Forced Air Cooling**

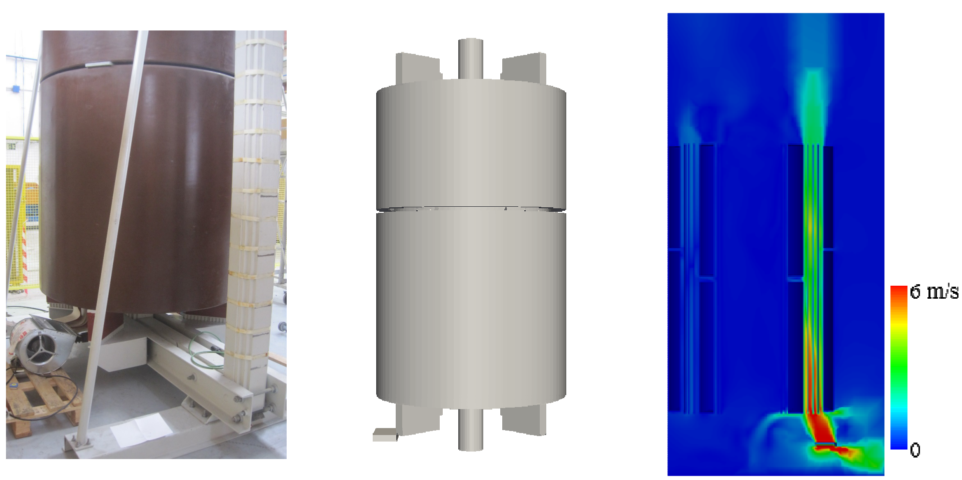
Figure 2 shows a typical configuration of forced air cooling; fans are positioned below the bottom of the coils, along the two side of the yoke, blowing air upwards. Due to the fans’ layout the air blown to the coil is not uniformly distributed along the circumferential direction, which potentially causes local hot-spot. Besides, the air portion flowing through the cooling channels inside the coils will dissipate heat more efficiently; the portion of air passing through the 'by-pass' space outside the outermost windings can only cool the outermost windings to a limited extent and therefore should be restricted. Some technologies such as a horizontal guiding plate can be applied to force the majority of the air to flow through the coil channels [10]. Hence for the purpose of cooling design optimization it is valuable to predict the air flow quantity towards the cooling channels and the by-pass space and the uniformity of the flow distribution along the channel circumferential direction.

(a) Top view (b) Front view (cross-section)

Figure 2: Layout of cooling fans at coil bottom; grey colour is iron core, red colour is inner and outer windings, yellow colour is insulation barriers and light blue rectangles represent the fans.

3. **CFD Modelling and Test Cases**

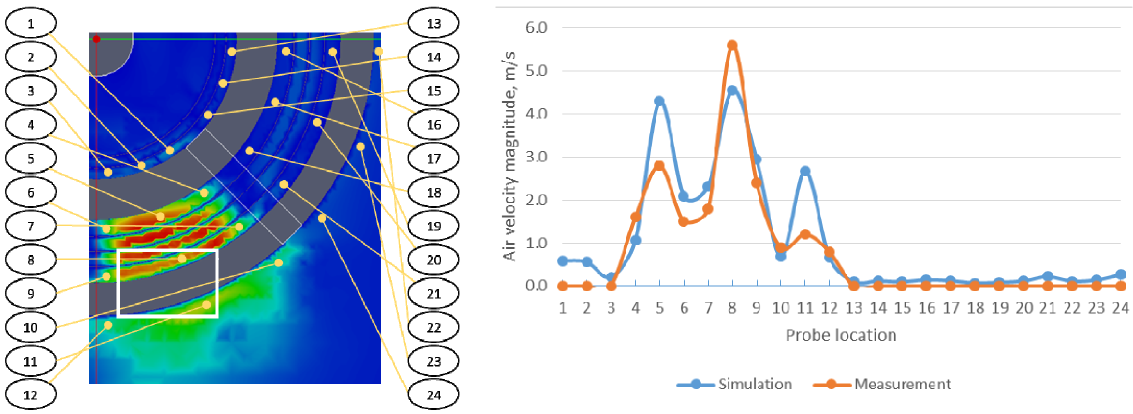
A series of experiments was setup in laboratory to conduct measurement and to validate CFD simulation tools. It starts with a simple test case that includes one phase coil with one fan placed below the coil bottom [9]. The case serves the purpose of investigating both the air flow distribution at the coil bottom going towards the channels or the by-pass space and the distribution along the channel circumferential direction. Figure 3 (a) shows the experimental setup and (b) shows the simplified 3D geometry for CFD modelling; considering that the single fan makes the structure asymmetric, 3D simulation on the full 360^{o} circumferential range was conducted by employing the software package OpenFOAM. The computation domain was meshed into around 3 million of cells and then simulated with steady state k-omega SST turbulence model. Since this study focused on air flow distribution, in the experimental setup or the simulation there is no heat generation considered. The simulated air velocity results at a vertical cross-section are illustrated in Figure 3 (c), from which it can be discovered that majority of the air flows to the cooling channels between the inner and outer windings; some portions are blocked by the winding bottoms, turn their direction and flow towards the central core or through the by-pass space outside the outer winding.



(a) Lab setup (b) 3D geometry (c) Air velocity distribution at vertical cross-section

Figure 3: A single coil, single fan experimental setup and its CFD simulation results.

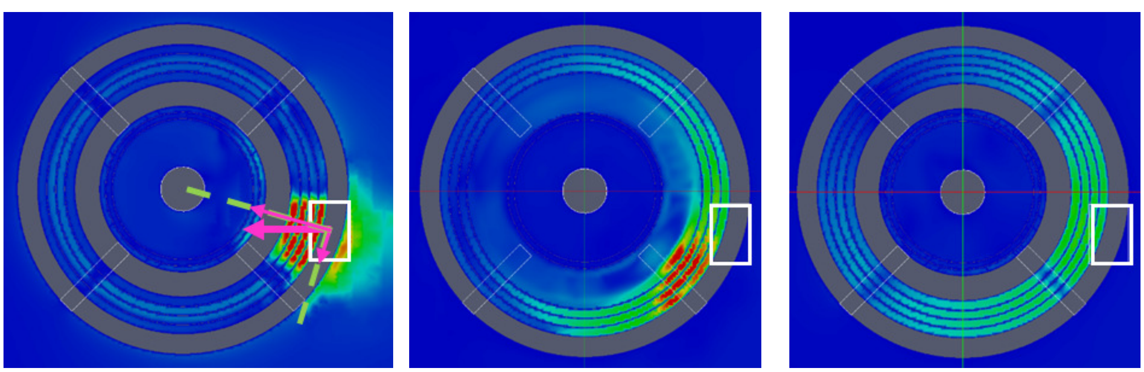
In order to validate the simulation results, air velocity was measured at the coil bottom particularly in the vicinity of the fan; the simulation results were then compared with the measurement values. Figure 4 (a) shows the measurement locations at the coil bottom; (b) compares between the measurement and the simulation numbers. The deviation between both series are 0.4 m/s on average; the distribution patterns match between both as well, implying that the simulation validity to predict the flow distribution going towards the cooling channels or the by-pass space.



(a) Measurement probe locations (b) Air velocity comparison

Figure 4: Air velocity measurement at the coil bottom near the fan (illustrated as the white rectangle) and comparison with CFD results.

Another set of air flow phenomenon details that can be investigated from the CFD simulation is the distribution along the cooling channel circumferential direction, shown in Figure 5. In this single fan asymmetric system, the air flow blown out from the fan outlet starts with a relatively concentrated pattern at the coil bottom and then the flow gradually expands along the channels towards the coil top direction. With the expansion the flow distribution develops its uniformity circumferentially; the evenest pattern happens at the coil top. Besides, because the fan blowing direction is not exactly towards the center of the circular windings, the air velocity has a tangent component which will drive the air shifting circumferentially as well; the radial component is however blocked by the concentric circular barriers. This flow phenomenon is observed more clearly in a single fan system than in a system with multiple fans and supports the validity of the simulation results as well.



(a) Coil bottom (b) Coil middle (c) Coil top

Figure 5: Air flow shift and uniformity development from coil bottom to top.

A single fan case is artificial and practical cases will have multiple fans positioned symmetrically along the two sides of transformer bottom yoke; to evaluate the simulation accuracy of symmetric configurations, a single phase coil equipped with eight fans was setup in laboratory as a next step, i.e. four fans each side of the coil. Because of the geometry symmetry only a quarter of the circumferential range can represent the simulation domain. Modelling using the same CFD technology with the previous single fan case yields simulation results which are then in comparison with the laboratory measurement values in Figure 6. It is more important that the distribution patterns match between the simulation and the measurement numbers, because the deviation between both may also be caused by measurement errors, especially when fluid velocity is relatively high, turbulence becomes greater and the measurement is more sensitive to measuring locations.

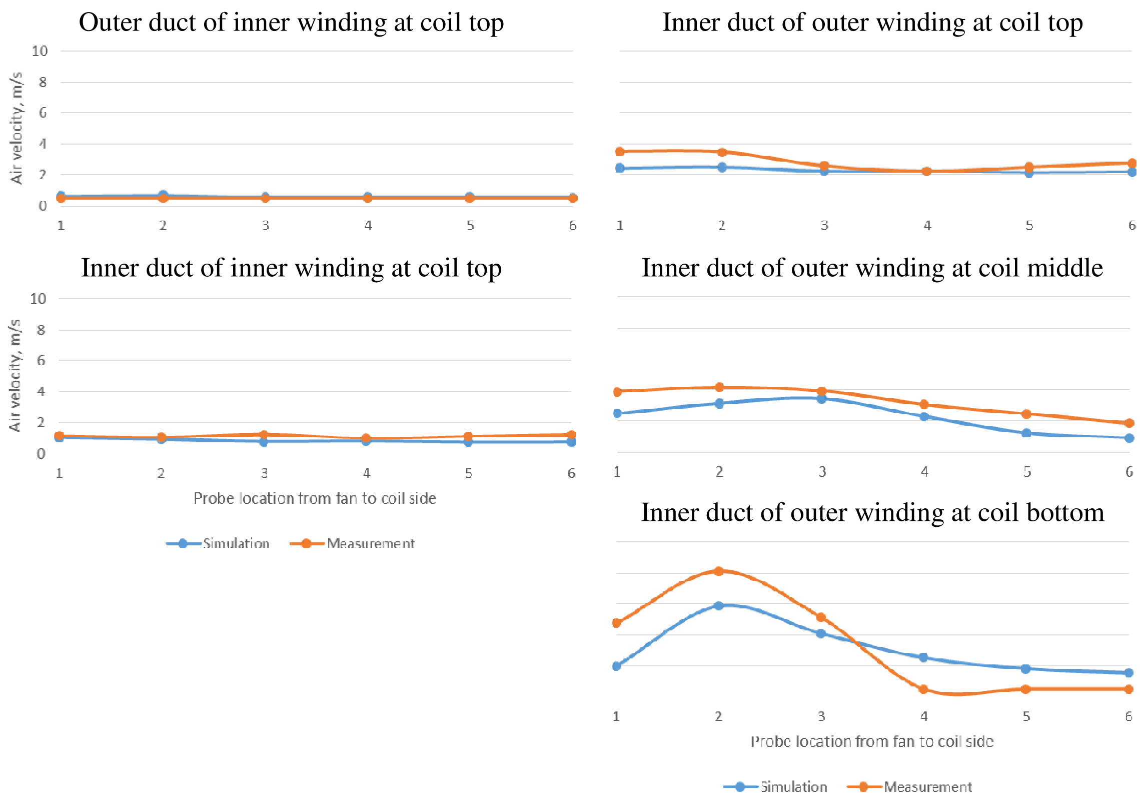


Figure 6: A single coil, eight fans experimental setup, the comparison between CFD and measurement results.

4. **Conclusion**

The CFD models presented in this paper provides a rapid yet accurate numerical approach to perform realistic cooling design evaluation of a forced air cooled dry-type transformer, in particular the forced air flow distributions either in radial or in circumferential directions; in comparison with prototyping and laboratory experiments, the simulation approaches cost lower. Consequently CFD modelling has gained the great potential to be applied in practice either to derive transformer cooling design guidelines or to directly aid the transformer products' design and development.

ACKNOWLEDGMENTS

Due appreciation should be given to our colleague Antonio Nogues in ABB S.A. Spain, who carried out the laboratory experiments in this paper for validating the CFD simulation results.

BIBLIOGRAPHY

[1] W. Wu, Z. Wang, A. Revell, H. Iacovides and P. Jarman. "Computational fluid dynamics calibration for network modelling of transformer cooling oil flows – Part I: heat transfer in oil ducts" (IET Electr. Power Appl., vol. 6, no. 1, pp 19-27, 2012).

[2] W. Wu, Z. Wang, A. Revell and P. Jarman. "Computational fluid dynamics calibration for network modelling of transformer cooling oil flows – Part II: pressure loss at junction nodes" (IET Electr. Power Appl., vol. 6, no. 1, pp 28-34, 2012).

[3] A. Blaszczyk, R. Flueckiger, T. Mueller and C.-O. Olsson. "Convergence behavior of coupled pressure and thermal networks" (Compel Journal, vol. 33, no. 4, pp 1233-1250, 2014).

[4] E. Morelli, P. Di Barba, B. Cranganu-Cretu and A. Blaszczyk. "Network based cooling models for dry transformers" (ARWtr Conference, Baiona, Spain, 2013).

[5] A. Cremasco, P. D. Barba, B. Cranganu-Cretu, W. Wu and A. Blaszczyk. "Thermal simulations for optimization of dry transformers cooling system" (The 10th Scientific Computing in Electrical Engineering (SCEE), Wuppertal, Germany, 2014).

[6] A. Weinlaeder, W. Wu, S. Tenbohlen and Z. Wang. "Prediction of the oil flow distribution in oil-immersed transformer windings by network modelling and computational fluid dynamics" (IET Electr. Power Appl., vol. 6, no. 2, pp. 82-90, 2012).

[7] A. Skillen, A. Revell, H. Iacovides and W. Wu. "Numerical prediction of local hot-spot phenomena in transformer windings" (Applied Thermal Engineering, vol. 36, pp. 96-105, 2012).

[8] R.B. Fdhila, J. Kranenborg, T. Laneryd, C.-O. Olsson and B. Samuelsson. "Thermal modeling of power transformer radiators using a porous medium based CFD approach" (THERMACOMP 2011, Dalian, China, 2011).

[9] W. Wu, A. Nogues and J. Kern. "OpenFOAM CFD modelling on forced air cooling dry-type transformers" (OpenFOAM User Conference 2015, Stuttgart, Germany, 2015).

[10] W. Wu, Y. Wang and Z. Wang. "CFD Simulation on Heat Exchanger Cooled Dry-type Transformers" (to be published in the Proceedings of the 2nd World Congress on Electrical Engineering and Computer Systems and Science (EECSS'16), Budapest, Hungary, 2016).