ARAI Journal of Mobility Technology
Volume 2 • Issue 2 • April–June, 2022, pp. 228-232
Research Paper
https://doi.org/10.37285/ajmt.1.2.9

Comparison of Numerical Methods for Thermal Performance Evaluation of Circuit Protection Devices in EV Application

Avadhoot Kittur[1], Dikhsita Choudhary[1] and Robert Michael Slepian[2]

[1]*Modelling and Simulation CoE, Eaton Technologies, Survey No. 77, Plot No. 1, Cluster C, Wing 1, EON Free Zone, Kharadi, Pune – Maharashtra.*

[2]*Eaton Corporation, Bloomfield, New Jersey, United States.*

ABSTRACT

With the growing demand of electric vehicles, design of circuit protection devices is now an important consideration in automobile industry. Modern day circuit protection devices have been constantly undergoing miniaturization due to requirement of minimizing the foot print for use in electrical vehicles and aerospace applications. This size reduction makes thermal management one of the most important aspects of their design. Use of numerical model to predict heat transfer can significantly reduce the cost and time required in testing physical prototypes.

In this paper, three different approaches for numerically predicting temperature rise of circuit breakers are discussed and compared from the point of view of accuracy and computational effort. The three methods are 1) Finite volume based analysis in which conjugate heat transfer inside and outside the breaker is modelled by solving Navier-Stokes equations 2) Finite element based heat conduction model in which convection is modelled as boundary condition instead of solving for fluid motion, and 3) Thermal network based model which uses electrical analogy of heat transfer to solve a thermal resistance network.

In the first two iterative models mentioned above, heat generation from current-carrying parts is calculated by solving Maxwell's equations of electromagnetics by Finite element method. Eddy current losses and temperature dependence of electrical conductivity is considered in the calculation of heat loss. In all three methods, electrical and thermal contact resistances are added at appropriate locations based on analytical calculations. All three methods have been validated with temperature rise test results.

In this paper, the heat loss and temperature of a molded case circuit breaker have been predicted by all three methods discussed above. It is observed that the Finite volume-based method is the most accurate amongst the three methods. It can computationally predict air motion and air temperature at critical locations. However, this additional accuracy comes at the cost of added effort in terms of additional mesh count and computation. The Finite element-based method gives good accuracy but does not predict air temperature. The analytical network-based model is less accurate compared to other methods and relies on product expertise and experience.

Based on the study, the following recommendations are made: 1) The finite element-based method is best suited to evaluate designs which do not alter flow pattern significantly 2) The finite volume method is recommended to evaluate effect of flow altering design changes 3) The network-based model is recommended for initial evaluation of correct cross sections of current carrying members.

KEYWORDS: Thermal management, Computational Fluid Dynamic, Thermal Network Analysis, Contact resistance, electrical contacts

Introduction

In present-day world, circuit protection devices form an integral part of all electrical systems including electric vehicles. From single-wire fuses to large switchgears, circuit protection devices find their way into every electrical installation, isolating the circuits from overcurrent conditions due to short circuit or overload.

Power loss from current carrying devices is converted into heat. Hence, they are required to satisfy strict temperature rise requirements by certificating bodies like IEC and UL. As technology is progressing, one of the major development areas of these devices is reduction of material hence increasing their ampacity. To satisfy the temperature regulations, the designs of these current carrying parts need to be optimized.

Experimental setups for every design iteration being a costly affair, numerical and analytical models are widely adopted nowadays to predict temperature rise. Lot of research has gone into building numerical models for heat load and temperature prediction.

For DC applications heat loss from conductors can be calculated as a multiplication of electrical resistance and squared current value. For AC applications, due to electromagnetic effects, there is an additional heat generation. For such cases, a set of electromagnetic equations need to be solved for calculating heat loss. Resistance offered to current is a direct function of the temperature rise in the components hence electromagnetic and thermal model needs to be coupled.

Temperature rise due to this heat generated is predicted by numerous methodologies in literature. In one approach, FEM is used to solve steady-state heat transfer equation to predict temperature rise. In such methods, dissipation of heat due to convection is approximated by assuming values of heat transfer coefficient. Weichart and Steinhauser [1] have used an FEM based tool to evaluate temperature on current conducting parts on a low voltage switchgear. Frei and Weichart [2] later used similar model in their work to predict temperature on non-current conducting parts like plastic casing in addition to current carrying parts. FVM is used solve all the conservation equations of mass, momentum and energy to predict temperature. In such analysis, a detailed study of air motion around the current carrying parts is possible. Heat transfer by convection is predicted accurately, unlike FEA based method. Bedkowski et al. [3] in their work shows a good validation of temperature rise prediction in a low voltage switchgear by a coupled electromagnetic- FVM model with experimental results. Xiao Yu, Fan Yang, Gao Bing *et al.* [5] have used coupled electromagnetic and thermal analysis to predict hot spot in vacuum bottle of vacuum circuit breaker. They observed that contact resistance between current carrying components can have significant effect on temperature rise. Accurate estimation of contact resistance thus, becomes necessary.

Molitor, F., Shoory, A., Sologubenko, O., Kaufmann, P. et al. [6] performed FEA based simulations to predict hotspots in busplates and contacts. They discuss the importance of capturing skin and proximity effect for accurate estimation of current distribution and hence ohmic losses.

The two methods discussed above, although helps in cost saving, require additional time in preparing the numerical models of each design iteration. Many researchers have used analytical models involving thermal network theory to approximate temperatures. Such models are based on the analogy between thermal and electrical resistances. Cherukuri [4] in his work has elaborately described how such a model can be employed to predict temperature rise in the design phase.

Literature has all three of the methods employed extensively in predicting temperature on circuit protection devices. There is very limited work wherein comparison of results predicted by all three methodologies has been made. The present work takes the case of a molded case circuit breaker and studies temperature prediction using all three methodologies described above. Results predicted are compared with actual experimental test results. The advantages and disadvantages of using each model are discussed.

Numerical Methods

Numerical analysis of circuit protection devices involves solving two different physical phenomena. Maxwell's equations for electromagnetics are solved for calculating heat generation due to electromagnetic effects in FEM based tool. The temperature rise due to this heat generated is calculated by solving (a) Steady-state heat transfer equation, in the FEM based approach, (b) equations of momentum, continuity, energy, turbulence and radiation in the FVM based approach. The network-theory based approach calculates analytically the Joule heat and heat due to eddy currents and predicts the temperature rise considering all three modes of heat transfer, viz. conduction, convection and radiation in the form of resistances.

Finite volume method based approach

Geometry and mesh: The circuit protection device used for the study is a Molded Case Circuit breaker (MCCB) which is a low voltage device. Fig. **1** shows the current carrying parts of the breaker.

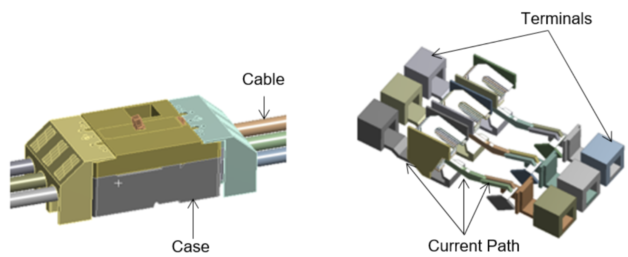


Fig. 1. MCCB Geometry.

The electromagnetic model used to predict heat generation in both the iterative approaches, was developed in ANSYS Maxwell tool. The mesh generation in this tool was carried out by adaptive discretization algorithm which utilizes the principle of energy conservation.

For Finite Volume Method, an air domain around the MCCB was modeled. Use of hexahedral cells for the CFD model ensured better accuracy and a reduced mesh size. Prism cells with higher aspect ratio were used in the fluid volume near solid walls to accurately capture boundary layer phenomena. The effect of change of density due to change in air temperature is captured by ideal gas equation.

Governing equations: For all types of flows involving heat transfer, ANSYS Fluent solves conservation equations for mass, momentum and energy. The steady-state conservation equations for mass, momentum and energy are:

$$\nabla.(\rho \boldsymbol{v}) = S_m \quad \ldots(1)$$

$$\nabla.(\rho \boldsymbol{v}\boldsymbol{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho g + F \quad \ldots(2)$$

$$\nabla.\left(\boldsymbol{v}(\rho E + p)\right) = \nabla.\left(k_{eff}\nabla T\right) + S_h \quad \ldots(3)$$

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \quad \ldots(4)$$

$$h = \sum_j h_j Y_j \quad \ldots(5)$$

$$h_j = \int_{T_{ref}}^{T} c_{p,j} dT \quad \ldots(6)$$

where,

p is the static pressure

$\bar{\bar{\tau}}$ is the stress tensor

ρg is the gravitational body force

F is the external body force

k_{eff} is the effective conductivity

S_h is the volumetric heat generation due to current flow through the conductors.

Table 1 below shows the values of k_{eff} and for materials used in simulation.

TABLE 1

Values of $\boldsymbol{k_{eff}}$ for materials used in simulation

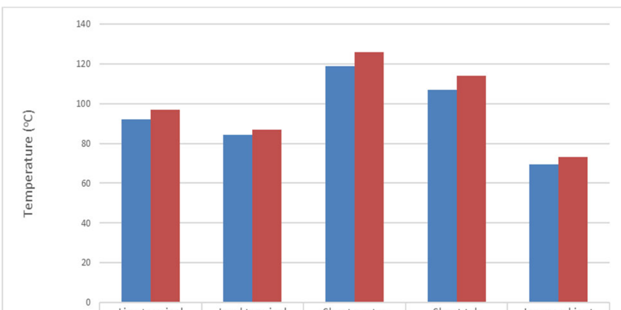
Parameter	Value
k_{eff}	390 W/mK (for copper) 0.3 W/mK (for plastics)

Values of other parameters are computed by the simulation model at various nodes.

The source term S_h in the energy equation is obtained by solving Maxwell's equations in the FEM based solver ANSYS Maxwell. The solver considers skin and proximity effects due to eddy currents set up due to electromagnetic induction.

To capture air flow around intricate shaped bodies inside the circuit breaker, k-ε turbulence model is chosen.

Radiation of heat inside the breaker is captured by the surface-to-surface model which assumes that all walls of the breaker are diffuse.



	Line terminal	Load terminal	Shunt centre	Shunt tab	Inner ambient
Experiment	92.2	84.3	118.7	107.1	69.6
CFD	97	87	126	114	73

Fig. 2 Comparison of temperature predicted by FVM method with experiment.

Fig. 2 shows comparison of temperature predicted by FVM with experiment. This methodology predicts temperatures quite accurately on the current path with a maximum deviation of 80C. Also, fluid temperatures can be accurately predicted as air flow is solved by solving Navier Stokes equations numerically. Fig. 3 shows air velocity vectors colored by temperature on the air domain surrounding the current path. This output can be useful to predict air temperature near the electronic components inside the switchgear. This methodology is best recommended when internal convection has a significant contribution in overall heat dissipation. Problems involving thermal performance evaluation of heat sinks, flow through vents/ louvres, forced convection by fans, blowers etc. can be best studied using this methodology.

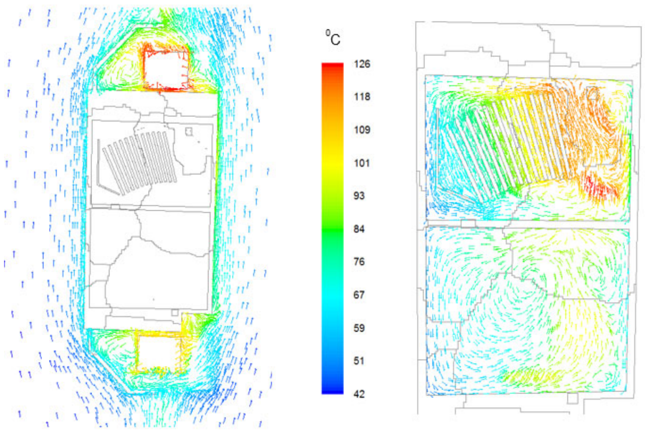


Fig. 3. Velocity vectors colored by temperature inside and outside the MCB

Finite element-based Methodology

Products like MCCBs, MCBs usually have small internal air cavities and no vents for air circulations. Natural convection and radiation from external casing surfaces and cables are the major modes of heat dissipation in these products. If simulation objective is to optimize the dimension of current carrying parts, then FEA based approach proves time effective. In this tool, only the conduction equation is solved while convection is treated as a boundary condition by providing empirically estimated heat transfer coefficients This section gives details of this methodology for the same MCCB discussed above.

Heat generation due to current flowing through MCCB is first predicted by electromagnetic analysis carried out in ANSYS Maxwell like the method discussed above. The heat loads are imported to ANSYS Steady state thermal module where various modes of heat transfers are computed. Temperature from thermal analysis is again fed back into Maxwell where electrical conductivity is modelled as a function of temperature. This iterative process is continued until the temperature changes in two successive iterations are less than 1%.

Boundary Conditions

Convection from external surfaces to ambient: The casing and the cable dissipate heat to ambient by convective heat transfer. Heat transfer coefficients are calculated based on empirical correlations and modelled as function of surface temperature. The reference temperature used to compute heat dissipation by this mode is the ambient temperature.

Radiation: Both internal and external radiation are modelled using radiosity method after computing view factor for all participating surfaces by hemicube method.

Internal convection: surfaces enclosed within the casing exchange heat via convection through internal air. This effect is captured by creating a pilot node with assumed temperature. The internal surfaces dissipate

$$Q_i = h_i A_i (T_i - T_{pilot\ node}) \quad \ldots(7)$$

The pilot node temperature is then obtained iteratively by ensuring the total heat generation in the domain is equal to total heat dissipation.

Results and Discussion

Fig. **4** shows comparison of test results with simulation. As can be seen, the methodology can predict temperatures on current path within 10° C on current path but cannot predict the temperature rise on the casing sides accurately.

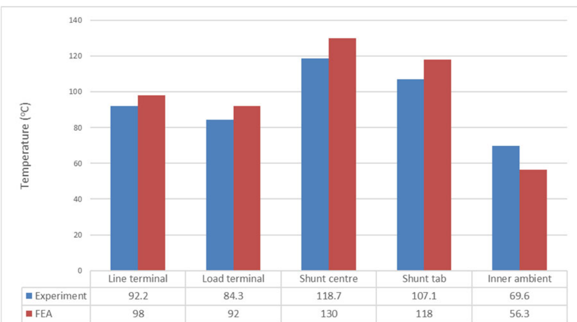


Fig. 4. Comparison of temperature predicted by FEA method and experiment

Another limitation of FEA based methodology is the inability to predict air temperature within the casing. Air temperature within the casing is important to determine to ensure reliability of electronic components inside the breaker.

Considering these limitations FEA methodology is best suited where temperature rise along the current part is only of interest. It is recommended to use CFD based model when accurate estimation of air or casing temperature is required.

Thermal Network Model

During the concept generation stage of product development using a full-fledged FEA or CFD based model may prove unviable as the detailed CAD models and inputs may not be available also the time to evaluate minor changes in dimensions is large using the first two methods. In such scenario, analytical model built using thermal resistances can prove effective.

Thermal network model involves constructing a thermal resistance network of electrical device under consideration using analogy between current flow and heat transfer. The temperature difference is equivalent to potential difference in electrical circuits while heat flow is equivalent to current flow. Thermal resistance is then given by equation

$$R_{th} = \frac{\Delta T}{Q} \quad \ldots(8)$$

Resistance offered by conduction, convection and radiation is then given by equations

$$R_{cond} = \frac{L}{KA} \quad \ldots(9)$$

$$R_{conv} = \frac{1}{hAs} \quad \ldots(10)$$

$$R_{rad} = \frac{1}{h_r As} \text{ where } h_r = 4\varepsilon\sigma {T_{avg}}^3 \quad \ldots(11)$$

Where R_{th}, R_{cond} , R_{conv} and R_{rad} denote the thermal resistance, conduction resistance, convection resistance and radiation resistance respectively. L, A and As denote Length, Cross sectional area and surface area of the component. h and h_r denote the values of convection and radiation heat transfer coefficients respectively. ε,σ and K denote emissivity, Stephen Boltzmann constant and thermal conductivity respectively.

Following Values are used for these parameters

TABLE 2

Parameters for thermal network model

Parameter	Value
K	390 W/mK (for copper) 0.3 W/mK (for plastics)
ε	0.05 for metals, 0.9 for plastics
σ	5.670374419 × 10-8 W/m²K⁴
h	Calculated based on empirical correlations

A network can be built using these resistances in which components are represented by their conduction resistance connected to nodes where temperature is to be determined. The heat dissipation due to convection and radiation can be represented by their respective resistance. Fig. **5** shows one such network constructed for an ACB

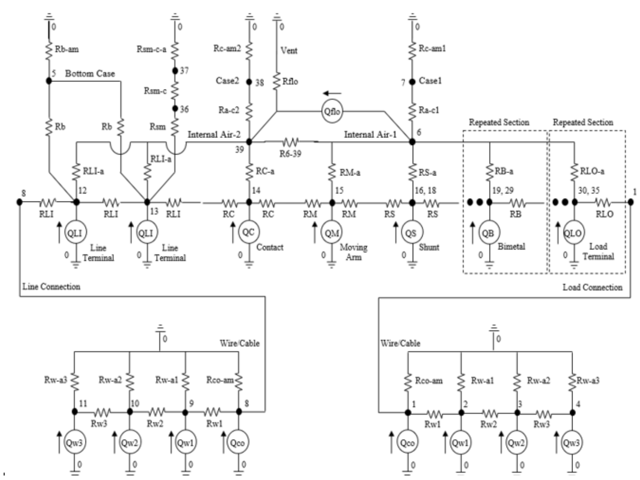


Fig. 5. Schematic of network model for MCCB

By equating total heat flow at each node to zero we can get set of simultaneous equation which can then be solved to get temperatures at the nodes.

Figure 6 shows comparison between temperature rise predicted by network model and test data for an MCCB. Compared to CFD or FEA based model the accuracy is low in this model although the trend of temperature rise across various components is captured correctly. Thus, this kind of analysis can be used for concept evaluations during initial stages of product development.

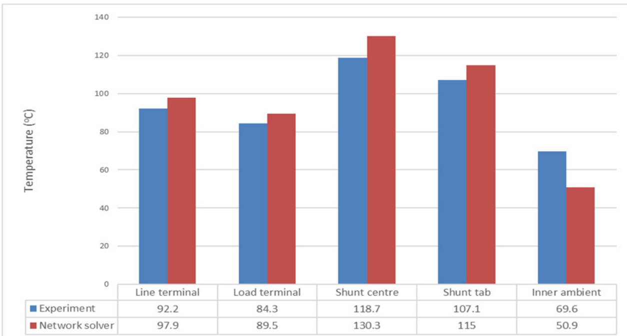


Fig. 6. Comparison of temperature predicted by network model and experiment

Comparison and Comments

Fig. 7 shows comparison of temperature predicted by the three methods discussed above.

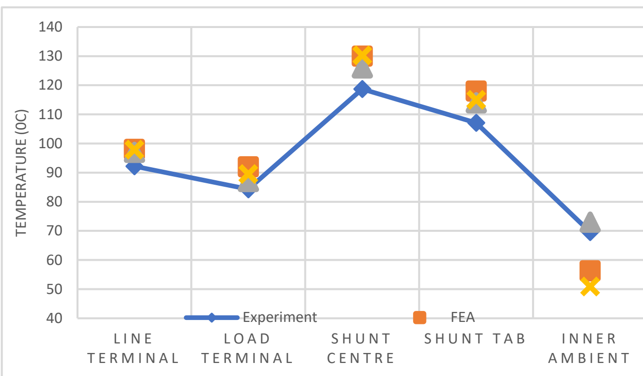


Fig. 7. Comparison of temperature predicted by three methods with experiment

Figure 7 shows comparison of temperature predicted by the three methods discussed above. It can be seen that, for simple switchgears with small air cavities and no vents all the three methods can predict the temperature rise on current carrying components with fairly good accuracy. CFD has a clear advantage in predicting air temperatures near area of interests as air flow is accurately captured. If vents are present or forced convection heat transfer is involved CFD proves to be considerably more accurate than other two methods but at the cost of higher meshing and computational effort. Network-based solver is accurate if the cross section of current carrying parts is constant and does not involve complex curved shapes.

Conclusion

Three methods to predict temperature rise in circuit protection devices were discussed along with advantages and limitations. Following guidelines can be used when deciding simulation approach for switchgears 1) FVM based approach should be used when accurate information about case temperature, air temperature is important or vents and fins are to be optimized 2) FEM based approach should be used when only accurate information about temperature of current carrying parts is needed or sizing of conducting bodies is to be optimized. 3) Network based method should be used during initial approximate sizing of current carrying parts.

References

[1] Paulke, J., Weichert, H. and Steinhaeuser, P., 2002. Thermal simulation of switchgear. *IEEE Transactions on Components and Packaging Technologies*, 25(3), pp.434-439.).

[2] Frei, P. U., & Weichert, H. O. (n.d.). Advanced thermal simulation of a circuit breaker. *Proceedings of the 50th IEEE Holm Conference on Electrical Contacts and the 22nd International Conference on Electrical Contacts Electrical Contacts, 2004.* https://doi.org/10.1109/holm.2004.1353102

[3] Bedkowski, M., Smolka, J., Banasiak, K., *et al.* 2014. Coupled numerical modelling of power loss generation in busbar system of low-voltage switchgear. *International Journal of Thermal Sciences*, 82, pp.122-129.

[4] Checurri, K., 2007. *Thermal Network Theory for Switchgear under continuous current.* PhD. NIT Rourkela.

[5] Yu, X., Yang, F., Bing, G., Liu, H., Li, X., Ai, S. and Wu, X., 2019. Investigation on the Thermal Performance of a 363 kV Vacuum Circuit Breaker Using a 3D Coupled Model. *IEEE Access*, 7, pp.128606-128616.

[6] Molitor, F., Shoory, A., Sologubenko, O., Kaufmann, P., Kraft, K., Janos, J., Vitek, L. and Iordanidis, A., 2021. Thermal Analysis and Design of Laminated Bus Plate Based Low Voltage Switchgear. *IEEE Transactions on Power Delivery*, 36(3), pp.1440-1448.

Address correspondence to *Mr. Avadhoot Kittur, Lead Engineer, Modelling and Simulation COE, Eaton Technologies, Survey No. 77, Plot No. 1, Cluster C, Wing 1, EON Free Zone, Kharadi, Pune – Maharashtra*
Email: AvadhootKittur@Eaton.com
Phone: 09619870759