ISSN 2347 - 3983
Volume 10. No.5, May 2022
International Journal of Emerging Trends in Engineering Research
Available Online at http://www.warse.org/IJETER/static/pdf/file/ijeter081052022.pdf
https://doi.org/10.30534/ijeter/2022/081052022

A Coupled Multifield Estimation for Temperature Rise in a Gas Insulated Bus Bar Using ANSYS

Polepeddy D P Srilekha[1], Dr. A. Raghu Ram[2], Dr. M. Mohan Rao[3]
[1]M.Tech Research Scholar, JNTUH College of Engineering (A), srilekhapolepeddy@gmail.com
[2]Professor, JNTUH College of Engineering (A), Telangana, India
[3]Additional GM/GSG & Electrical, Bharat Heavy Electricals Limited, Telangana, India

Received Date : April 6 , 2022 Accepted Date : April 28, 2022 Published Date : May 07, 2022

ABSTRACT

Gas Insulated Switchgear (GIS) is an integral part of the urban power systems, which have been developed to address the substantial need for making substations more compact and reliable. The gas insulated bus bar is a crucial component of the power equipment in the substations with voltages over 100kV. In general, the ampacity of the bus bar is constrained by the maximum operating temperature, which has to be predicted, according to the demands of the standard IEC 62271. Therefore, thermal analysis of gas insulated bus bar has to be performed in the design of GIS. However, the mechanisms of heat generation and transfer in the SF6 gas and air are so complicated that it is not a simple problem to predict the temperature rise.

Joule's heat due to current in the main conductor and heat due to induced eddy currents in the tank is calculated via Magnetic Analysis. The heat transfer to the tank and atmosphere is done by means of radiation and natural convection. Heat transfer coefficients are not constant and vary on many parameters such as model geometry and material constants, making them difficult to calculate and apply to the boundaries. By considering the factors stated above, Nusselt number is used to analytically compute the temperature dependent heat transfer coefficients on the boundaries. The proposed research work presents a coupled magnetic and thermal field analysis supported by finite element analysis of extra-high voltage GIS bus bar with the help of ANSYS Maxwell and ANSYS Fluent. The results of the coupled finite element approach accord well with the values derived analytically.

Key words: Coupled magnetothermal analysis, Heat transfer, Temperature rise, ANSYS

1. INTRODUCTION

Gas Insulated Switchgear (GIS) is an important part of urban power systems that was created to meet a vital need for more compact and dependable substations. The gas insulated bus bar is a crucial component of the power equipment in the substations with voltages over 100kV [1].

Any new equipment's design and development has ramifications that must be evaluated as a whole in order for the engineering content of the job to match the power system's technical requirements [2]. The electrical requirements are used to determine the diameters of the conductors and enclosures. The maximum working temperature of the bus bar, which must be forecasted according to the specifications of the IEC 62271 standard, limits the current-carrying capacity of the bus bar in general [3]. The magnitude of the rated peak dynamic current under short circuit and the rated short time current, along with the resulting temperature rises, will influence the dimensions of equipment in GIS. Dielectric losses in the insulating spacers used in a GIS produce heat. Because of the thermal resistance of the insulator, these losses produce a temperature rise inside the bulk of the insulator. Due to the rising temperature, dielectric losses increase, and the temperature rises even more. If this cumulative process is not stabilized, thermal breakdown will occur. As a result, in the design of GIS, forecasting the rise in temperature in a bus bar is critical.

The proposed work focuses on the development of an approach for predicting the rise in temperature in the bus bar of a gas insulated substation. From the literature survey performed, a lot of techniques have been identified to predict the temperature rise. The fundamental concept used in each of these techniques is based on Coupled Magnetothermal analysis [3]-[4]. The objective of the proposed work is to perform coupled Multifield 2-dimensional finite element analysis for predicting the temperature rise of EHV GIS bus bar and compare the results from both analytical method and the 2- dimensional finite element method to predict accurate temperature rise. 2-D models based on the FEM are used to study the distribution of temperature in the bus duct systems.

2.COUPLED MULTIFIELD ANALYSIS

2.1 Magnetic Analysis

Magnetic field analysis is used to compute Joule's heat due to current in the main conductor and heat due to induced eddy currents in the tank.

The power losses in the main conductor and the tank are found by magnetic analysis. The main conductor current and eddy currents in the tank generate power losses in the bus bar of GIS. The computation of power losses is influenced by a number of factors. Materials characteristics, conductor size, and current- carrying mechanism are just a few of the variables [5]. These power losses must be estimated before being used as an input into the thermal analysis that will determine the temperature rise. For an accurate power loss estimate, variation of electrical resistivity with temperature is taken into consideration and also to get the power losses in terms of the inner conductor temperature. Figure 1 shows the main current and eddy currents profile in a GIS bus bar.

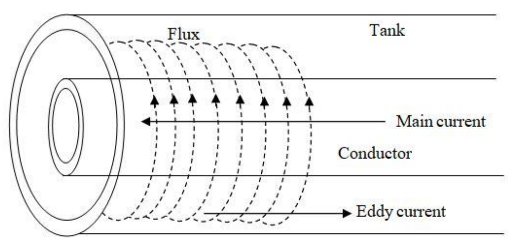

Figure 1: Current profile in a GIS bus bar

$$P = I^2 R \tag{1}$$

2.2 Thermal Analysis

Heat is developed due to the current flowing in the main conductor of the GIS. This heat is transferred to the tank and then to the atmosphere by means of radiation and natural convection. In the thermal analysis, the heat transfer coefficients are found from the principle of energy balance that equates the total heat transferred to sum of heats transferred by natural convection and by radiation [6]-[8].

When analyzing temperature rise, the convection heat flux is given by,

$$q_{conv} = h\ (T_s - T_\infty) \tag{2}$$

The radiation heat flux is given by,

$$q_{rad} = h_r\ (T_s - T_\infty) \tag{3}$$

Total heat flux which is transferred from GIS bus bar to the atmosphere is given by,

$$q = q_{conv} + q_{rad} \tag{4}$$

As the heat flux is the flow of energy per unit area, it can be expressed in terms of energy and as power is the rate of change of energy, power can be inferred as the product of heat flux and area.

3.ENERGY BALANCING EQUATION

Energy balancing equation can be applied as the quantity of heat generated is equal to the amount of heat transferred to the atmosphere. For simplicity in the calculations, we consider the power balancing equation for both heat generated and heat transferred to the atmosphere.

$$I^2 R(T_s) = hA\ (T_s - T_\infty) + \varepsilon\sigma A\ (T_s^4 - T_\infty^4) \tag{5}$$

4.ANALYTICAL CALCULATIONS

Considering all the laws and conditions stated above, we have the flowchart in the figure 2 for the analytical calculations below which comprises of two parts, one for calculating the electrical losses and the other part for calculating the heat transferred to the atmosphere. To find out the temperature rise, we equate the corresponding power losses on both the sides taking care of the units. The heat transfer occurs in two modes namely natural convection and radiation from inner main conductor to the tank and the tank to the atmosphere. Table 1 lists the parameters that were utilized in the computations.

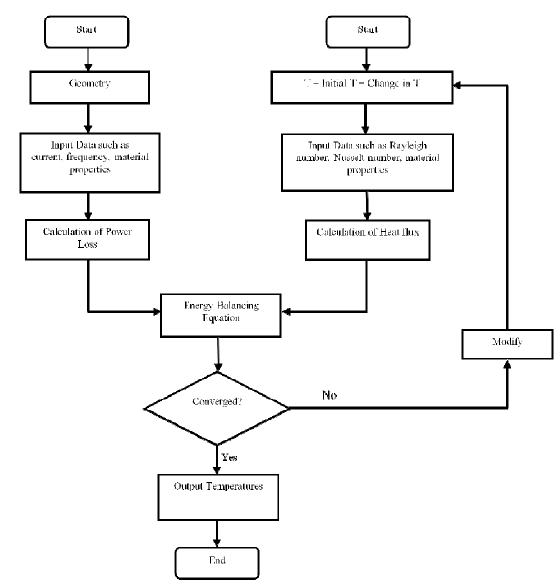

Figure 2: Flowchart for analytical calculations showing both magnetic and thermal analysis

5.DESIGN AND MODELLING OF GIS BUSBAR

This work has been implemented using ANSYS Maxwell 16.0 and ANSYS Fluent present in ANSYS Workbench 14.5.

The model is first built using the existing mechanical forms present in the Maxwell interface. The region of Electromagnetic field can be selected according to the accuracy level required and the type of solver.

Once the desired model is built, the materials are given as an input to the model. Cloning operation is performed to avoid intersecting meshes. Then boundaries and symmetry conditions are specified. The excitation is input to the main conductor as the main current flowing in the conductor.

Table 1: Parameters of Air and SF_6

S.No	Property	Air	SF_6
1	Kinematic Viscosity (m^2/s)	19.6 x 10^{-6}	1.31 x 10^{-6}
2	Thermal Conductivity (W/mK)	0.024	0.0136
3	Density (kg/m^3)	1.026	13.5
4	Specific Heat (J/mK)	29.19	96.6

In this simulation shown in the figure 3, eddy current solver has been employed to implement the Maxwell's EM equations to obtain the power losses i.e. ohmic losses in the main conductor along with eddy current losses in the tank. The eddy solution is a full wave solution that includes electromagnetic wave radiation effects. Solution setup is added. The model is validated for all types of checks and then analyzed to determine the ohmic losses. The ohmic losses are then plotted.

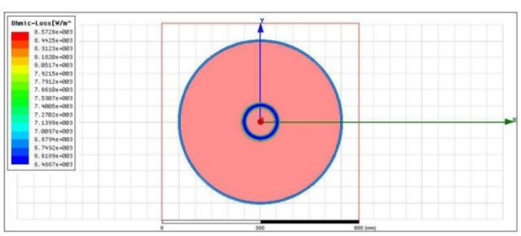

Figure 3: 2D design of GIS Bus Bar for magnetic analysis to obtain ohmic losses along with eddy current effects

The workbench setup is used as a platform for integrating both Maxwell and Fluent interfaces. The Fluent uses computational fluid dynamics to perform heat transfer calculations for the given geometry. The geometry developed in Maxwell 2D is imported into the geometry of Fluent. The Fluent uses computational fluid dynamics to perform heat transfer calculations for the given geometry. The geometry developed in Maxwell 2D is imported into the geometry of Fluent.

The fluid domain is established with Sulphur Hexafluoride as the medium of heat transfer between the conductor and tank and air as the medium between the tank and the atmosphere. The solution is initialized and run to get the desired temperature profile over the main conductor and tank domain.

6.RESULTS AND DISCUSSIONS

This section deals with the results of simulation work and their discussions classified into various cases in order to have a better knowledge of the temperature rise profile from conductor to the tank when different materials are utilized inside the GIS.

The figure 4 shows the temperature rise inside the GIS bus bar i.e. conductor and tank. The inner conductor temperature is found to be 313.8 K and the outer tank temperature is 295 K. The analytical calculations performed indicate that the inner conductor temperature obtained is 315.75 K and the outer tank temperature is 297.3 K. The calculated values accord well with the obtained values from 2D model of the GIS after performing Coupled Magnetothermal Finite Element Analysis.

Table 2 compares the results of analytical calculations to the findings of coupled magnetothermal study performed with ANSYS finite element software.

Table 2: Comparison of results from Analytical Calculations and coupled FEA Using ANSYS

S.No	Temperature	Analytical Calculations	Coupled FEA Using ANSYS
1	Conductor	315.75 K	313.8 K
2	Tank	297.3 K	295 K

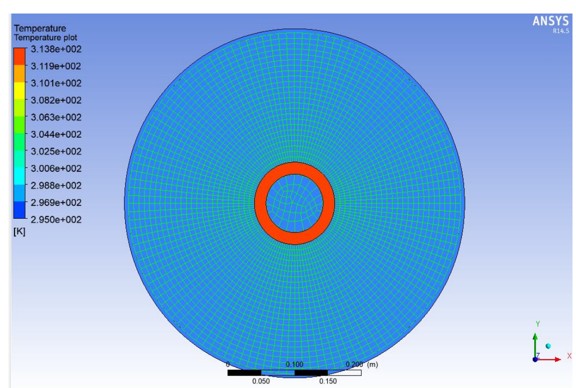

Figure 4: Temperature rise profile within GIS bus bar after performing coupled FEA using ANSYS

The figure 5 shows the comparison of results from both the methods.

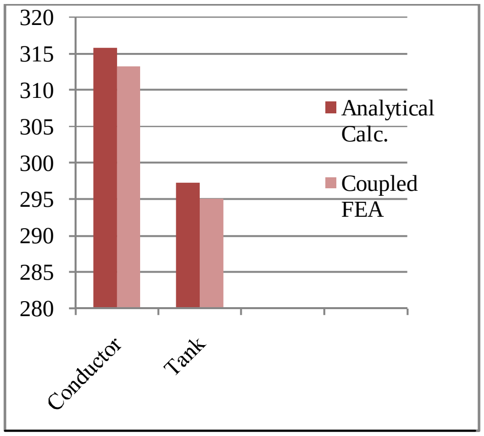

Figure 5: Graph showing the comparison of results

The simulation work has been carried using different conductor and tank to have a better grasp on the temperature rise profile from conductor to the tank when different materials are used inside GIS. The first case deals with the change in conductor material to copper and the tank material to stainless steel. This would decrease the ohmic losses by a significant amount which will lead to a decrease in the temperature rise inside the GIS.

The coupled FEA using ANSYS results in an inner conductor temperature of 305K and the outer tank temperature of 295K. But copper is not much used as a conductor due to its high cost even though the temperature rise is low.

7.CONCLUSION

The temperature rise in the bus bar of a GIS is one of the key parameters which effects the current carrying capacity of GIS bus bar at extra high voltages. This work focused on predicting the temperature rise in an EHV GIS Bus Bar by Coupled Multifield Finite Element Analysis.

First the magnetic analysis is performed to obtain power losses in GIS bus bar which include ohmic losses due to the main current flowing in conductor and eddy current losses induced in the tank. Variation of electrical resistivity with temperature is also considered while performing the magnetic analysis. These losses are then given as an input to the thermal analysis where these power losses are converted into heat. Temperature rise in the tank and the conductor is then calculated by means of Energy Balancing Equation. To validate the correctness of the preceding method, finite element analysis was performed using ANSYS.

The results of the coupled finite element approach accord well with the values derived analytically. With this it can be concluded that, the coupled magnetothermal analysis provides a precise and dependable approach for calculating the temperature rise in GIS bus bar. Since the current carrying capacity of the conductor is constrained by the maximum operating temperature, this can be extended into the design of GIS power apparatus.

8.FUTURE SCOPE

This work can be utilized to design the power apparatus in gas insulated substations such as gas insulated bus bar. The novelty of this method lies in the application of FEM to analyse coupled electric and thermal field problems by considering the variations of electric and thermal conductivities with temperature. The proposed method is efficient and accurate enough to be applied to some practical problems. The computational error can be further diminished by designing a three dimensional model of GIS bus bar. This work can be further optimized by considering the external factors such as solar radiation and resistance between the conductor and insulating spacers.

REFERENCES

1. M.S. Naidu, V.Kamaraju, **High Voltage Engineering**, 4th edition, 2009, TheMcGraw-Hill companies.
2. M.S. Naidu, **Gas Insulated Substations**, 2008 I.K. International Publishing House Pvt. Ltd
3. S. W. Kim, H. H. Kim, and S. C. Hahn, "**Coupled finite-element analytic technique for prediction of temperature rise in power apparatus**," IEEE , vol. 38, no. 2, pp. 921–924, Mar. 2002.
4. Haoyong Song, Guobin Hou, Wei Wang, Xiaofeng Deng, Qingdan Huang, Wenxiong Mo, Makoto Hasegawa, Xingwen Li "**Application of Computational Fluid Dynamics to Predict the Temperature-Rise of Gas Insulated Switchgears"**, 2017 4th International Conference on Electric Power Equipment.
5. W. Z. Black, B. A. Bush, and R. T. Coneybeer, "**Steady-state and transient ampacity of busbar**," IEEE Trans. Power Delivery, vol. 9, no. 4, pp. 1822–1829, Oct. 1994.
6. E. C. Guyer and D. L. Brownell, **Handbook of Applied Thermal Design**. New York: McGraw-Hill , 1988.
7. F. P. Incropera and D. P. De Witt, **Fundamentals of Heat and Mass Transfer**. New York: Wiley, 1996.
8. M. N. Ozisik**, Heat Transfer a Basic Approach**. New York: McGrawHill, 1990