Innovative Journal for Opportunities and Development in Science & Technology,
Volume 6, Issue 7, June. 2026
Paper Id: – IJ2025V6I7 -6 A-D
ISSN 2582-6026

FINITE ELEMENT BASED ELECTRO-THERMAL ANALYSIS OF BUSBAR UNDER DIFFERENT CURRENT AND AMBIENT CONDITIONS USING HYPERMESH

[1]Mukesh Sampatrao Autade,[2] Omkar Balasaheb Karande, [3]Dnyaneshwar Macchindra Khomane, [4]Karan Bhimrao Sagale, [5]Prof. Dr. Sandip Ghat, [6]Mr. Shantanu M. Jagatap

[1]UG Scholar Dept. Of Electrical Engg., Rajiv Gandhi Collage Engg, Karjule Harya Parner Ahilyanagar
[2]UG Scholar Dept. Of Electrical Engg., Rajiv Gandhi Collage Engg, Karjule Harya Parner Ahilyanagar
[3]UG Scholar Dept. Of Electrical Engg., Rajiv Gandhi Collage Engg, Karjule Harya Parner Ahilyanagar
[4]UG Scholar Dept. Of Electrical Engg., Rajiv Gandhi Collage Engg, Karjule Harya Parner Ahilyanagar
[5]HOD, Dept. Of Electrical Engg., Rajiv Gandhi Collage Engg, Karjule Harya Parner Ahilyanagar
[6]Scientist CIRT, Pune

karansagale20@gmail.com

ABSTRACT - A thorough investigation of the Finite Element Based Electro-Thermal Analysis of a Copper Busbar using Altair Hyper Mesh is presented in this paper report. This work's primary goal is to comprehend how ambient temperature and current flow affect a copper busbar's thermal performance in electrical distribution systems. There are two primary phases of CAD modelling and meshing, and Stage 2: Electro-thermal analysis. In the first stage, the 3D CAD model of the busbar was designed using Solid Works and then imported into Hyper Mesh for high-quality meshing Accurate findings were obtained in the subsequent analysis step by ensuring proper mesh refinement, aspect ratio, and element size. In complicated geometries, accurate thermal and structural predictions are made possible by the employment of tetrahedral solid constituents. In order to assess the temperature rise brought on by Joule heating, the second step, which will be carried out in the next phase, will combine electrical and thermal physics. The findings will aid in determining material compatibility, heat dissipation patterns, and safe current limitations. The groundwork for enhancing busbar design, boosting dependability, and lowering overheating failures in power systems is laid forth in this study. The FEM-based method offers engineers a simulation-driven, economical substitute for experimental testing.

I. INTRODUCTION

Busbars are essential to the safe and effective distribution of electric current among several circuits in contemporary electrical power systems. To guarantee low energy loss and maximum current-carrying capability, they are usually constructed from highly conductive materials like copper or aluminum. On the other hand, strong current flowing through a busbar causes Joule heating, which can raise the component's temperature considerably. The electrical system's longevity, dependability, and performance are all impacted by this temperature increase.Finite Element Analysis (FEA) offers a potent simulation-based method for precisely predicting and controlling this thermal behavior. Before doing physical testing, engineers can examine the combined impacts of electrical and thermal loads under different boundary conditions using FEA. This temperature rise affects the lifetime, reliability, and performance of the electrical system .A powerful simulation-based technique for accurately forecasting and managing this thermal behavior is provided by Finite Element Analysis (FEA). Engineers can use FEA to analyze the combined effects of thermal and electrical loads under various boundary conditions prior to doing physical testing.

1.1 Theoretical background

Fundamentals of Finite Element Method (FEM)

A popular numerical analytical method in engineering for resolving challenging physical issues including structural, thermal, electrical, and electromagnetic phenomena is the Finite Element Method (FEM). It makes it feasible to approximate governing differential equations over complicated geometries where obtaining analytical solutions is challenging or impossible. A physical domain is split up into several smaller, more straightforward areas known as finite elements in finite element modeling (FEM). Nodes are used to link these components and determine unknown field variables like temperature, voltage, and displacement. Interpolation functions are used to describe the behavior inside each element, and the individual element equations are assembled to give the global response of the complete system.FEM enables the simultaneous assessment of electrical current distribution and temperature fluctuation across the conductor for electro-thermal examination of a busbar. The mesh quality, element type, and boundary condition definition all affect how accurate the solution is.

The general steady-state heat conduction equation used in FEM is:

$$-\boldsymbol{\nabla} \cdot (\boldsymbol{k}\boldsymbol{\nabla}\boldsymbol{T}) = \boldsymbol{Q}$$

where:

k = Thermal conductivity of material (W/m·K)

T = Temperature (°C or K)

Q = Internal heat generation per unit volume (W/m^3)

By discretizing the busbar geometry into finite elements, FEM accurately predicts temperature gradients, current density distribution, and heat flow patterns under different operating conditions.

2.2 Joule Heating Theory

The main cause of temperature increases in electrical conductors is Joule heating, sometimes referred to as resistive heating. Electrical energy is transformed into thermal energy when electric current flows through a conductor with limited electrical resistance. Electrothermal analysis is based on this phenomena, which was initially reported by scientist James Prescott Joule.The heat generated due to current flow is proportional to the square of the current and the electrical resistance of the conductor. The Joule heating equation is:

$$\mathbf{P = I^2 R}$$

where:
P = Heat generated (W)
I = Current flowing through the conductor (A)
R = Electrical resistance (Ω)
The resistance of a conductor depends on its material properties and geometry and is given by:

$$\boldsymbol{R = \frac{\rho L}{A}}$$

where:
ρ = Electrical resistivity (Ω·m)
L = Length of conductor (m)
A = Cross-sectional area (m^2)
Joule heating may raise conductor temperature considerably in busbar systems with high currents. Reduced electrical conductivity, thermal expansion, insulation deterioration, and ultimately system failure might result from an excessive temperature increase. As a result, precise Joule heating prediction is necessary to provide dependable and safe busbar operation.

II. PROBLEM STATEMENT

2.1 Problem Definition

In electrical power distribution systems, busbars are frequently utilized to transport high current levels with little power loss. They are often used in industrial electrical systems, substations, power control centers, and switchgear panels. Electrical resistance in the conductor produces heat in the form of Joule losses (I^2R losses) as a result of the high current flowing through the busbar. This heat raises the temperature, which might have a negative impact on the electrical system's dependability, efficiency, and performance.A busbar's excessive temperature rise can cause a number of operational problems, including as decreased electrical conductivity, thermal expansion, oxidation of conductor surfaces, deterioration of insulating materials, and early electrical equipment failure. High current loading and high ambient temperature conditions exacerbate these effects even further.Thermal performance evaluation of busbars has traditionally been done through experimental testing, which is sometimes costly, time-consuming, and challenging to do under various operating situations. Thus, an efficient substitute for forecasting electrical and thermal behavior before to physical implementation is a simulation-based strategy utilizing the Finite Element Method (FEM).

"To perform a finite element based electro-thermal analysis of a copper busbar under different current loading and ambient temperature conditions using Altair Hyper Mesh and Opti Struct, and to evaluate the resulting temperature distribution, heat flux, and thermal performance."

The study's objectives are to locate temperature hotspots, comprehend heat dissipation properties, and evaluate the busbar design's adaptability for a range of operating scenarios.

2.2 Objectives

This project's main goal is to use finite element analysis (FEA) to examine the electro-thermal behavior of a copper busbar. The goal of the study is to comprehend how ambient temperature and electrical current affect the busbar's thermal performance.

The specific objectives are as follows: To develop an accurate three-dimensional CAD model of the copper busbar suitable for finite element simulation. To generate a high-quality finite element mesh using Altair Hyper Mesh for reliable electro-thermal analysis. To assign appropriate material properties such as electrical conductivity, thermal conductivity, density, and specific heat capacity of copper. To perform electrical analysis and determine the current density and voltage distribution within the busbar.

To evaluate Joule heat generation resulting from current flow through the conductor.

To conduct coupled electro-thermal analysis by integrating electrical and thermal fields within the simulation environment.
To study the effect of different current magnitudes on temperature rise and heat generation.
To investigate the influence of varying ambient temperatures on busbar thermal performance.
To determine temperature distribution, heat flux, and hotspot locations throughout the busbar geometry.
To validate simulation results using theoretical calculations and engineering principles.
To assess the safety and reliability of the busbar design under practical operating conditions.
To provide recommendations for thermal management and design optimization of busbar systems.

III. FINITE ELEMENT MODEL

3.1 Busbar Geometry Description

A realistic copper busbar that is often used in power control panels, switchgear assemblies, and electrical distribution systems is represented by the geometric model utilized for the electro-thermal study. The busbar maintains low electrical resistance and high current-carrying capacity while acting as a conductive conduit for the transfer of electrical power between different electrical components. SolidWorks was used to develop the geometry, which was based on typical industrial busbar layouts. A flat, rectangular copper conductor with mounting holes at both ends for mechanical and electrical connections makes up the model. The bolted couplings seen in electrical systems are replicated by these holes. The busbar's size were chosen to preserve computational efficiency during simulation while representing a genuine industrial use.

In order to guarantee accurate modeling of current flow routes and heat transfer properties, special emphasis was paid to geometric precision during model building. Because these areas may encounter localized current concentration and temperature rise during operation, features including mounting holes, connecting regions, and sharp edges were kept. Before being exported in STEP format for additional processing in Altair Hyper Mesh, the finished CAD model was verified for geometric correctness. The accuracy of the finite element analysis is increased and meshing errors are reduced with proper geometry preparation..

Key Features of Busbar Geometry

Material: Copper

- Rectangular flat conductor profile
- Mounting holes for terminal connections

- Uniform cross-sectional area
- Industrial busbar configuration
- Suitable for electro-thermal simulation studies

3.2 Meshed Model from Stage-I

The electro-thermal study carried out in Stage II is based on the meshed model created at Stage I. A number of geometry cleanup procedures were performed after importing the CAD geometry into Altair Hyper Mesh in order to remove flaws that can have an impact on mesh production and solver performance.The continuous physical domain was converted into a numerical model appropriate for FEM analysis by discretizing the busbar geometry into finite elements. A three-dimensional solid mesh was created to precisely depict current flow and temperature distribution across the busbar volume since the investigation involves linked electrical and thermal processes.Because of its capacity to accurately depict complicated geometries and curved surfaces, a tetrahedral solid mesh was used. In areas where greater temperature gradients and concentrations of current density were anticipated, such as bolt holes, terminal positions, and edge transitions, mesh refinement was used.An appropriate balance between computational precision and solution time is provided by the produced mesh.

IV. ELECTRO-THERMAL ANALYSIS METHODOLOGY

4.1 Model Preparation

The construction of the finite element model created during Stage I is the first step in the electro-thermal analysis. After being loaded into Altair HyperMesh, the CAD geometry produced in SolidWorks is examined for geometric integrity.To eliminate redundant surfaces, free edges, and other features that might degrade mesh quality, the model is subjected to geometry cleansing procedures. Before moving on to the analysis phase, the created finite element mesh is checked for connectivity, continuity, and element quality. Coupled electrical and thermal simulations are based on the finished model..

4.2 Material Assignment

Following model preparation, the finite element mesh is given the copper material characteristics. Density, specific heat capacity, thermal conductivity, and electrical conductivity are among the given material attributes.

These properties enable the solver to calculate:

Electrical current flow ,Current density distribution ,Joule heat generation, Temperature distribution ,Heat flux variation

The material is assumed to be homogeneous and isotropic throughout the busbar volume for the present steady-state analysis.

4.3 Electrical Boundary Conditions

Electrical boundary conditions are applied to simulate actual operating conditions of the busbar.One end of the busbar is assigned an electrical potential or current input, while the opposite end is specified as the reference ground terminal. This creates a potential difference that drives current through the conductor.

The electrical boundary conditions applied include:

Input current loading ,Voltage potential assignment ,Ground reference condition ,Current flow path definition The electrical solver computes voltage distribution and current density throughout the busbar geometry based on these conditions.

4.4 Thermal Boundary Conditions

Thermal boundary conditions are applied to represent the interaction between the busbar and the surrounding environment.The ambient air temperature is specified on all exposed surfaces, and natural convection is considered as the primary heat dissipation mechanism.

The thermal boundary conditions include:

Ambient temperature specification, Convective heat transfer coefficient, Surface heat dissipation Steady-state thermal environment, These conditions allow the solver to determine the balance between internally generated heat and externally dissipated heat.

4.5 Coupled Field Setup

Electrical and thermal physics interact together in electro-thermal analysis. Thermal analysis uses the Joule heating produced by the electrical solution as an input heat source.

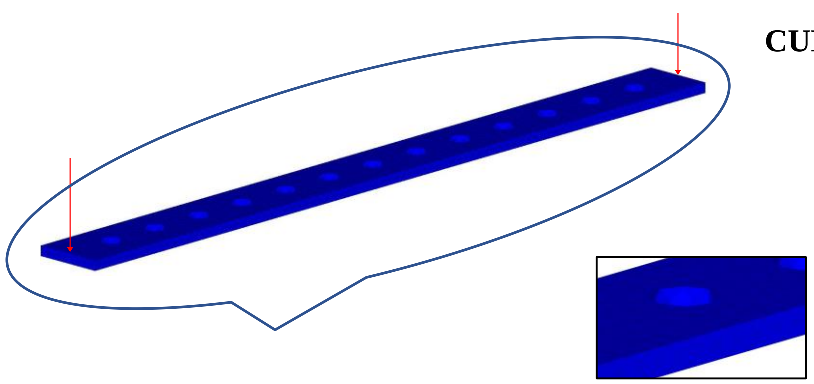
Innovative Journal for Opportunities and Development in Science & Technology,
Volume 6, Issue 7, June. 2026
Paper Id: – IJ2025V6I7 -6 A-D
ISSN 2582-6026

V. RESULTS AND DISCUSSION

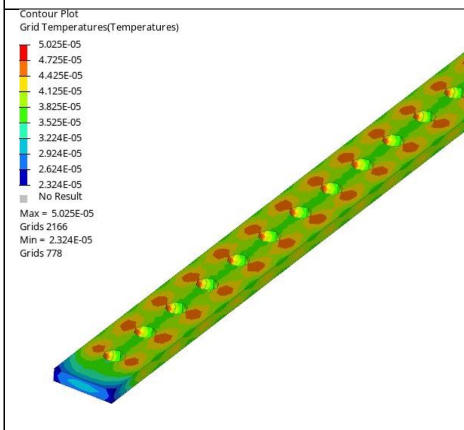
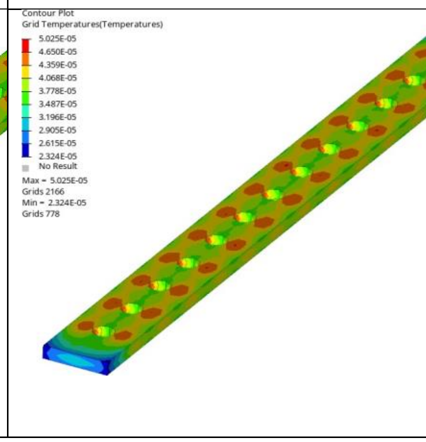
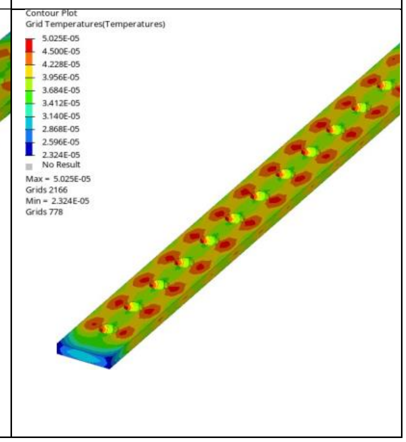
The electrical and thermal behavior of the copper busbar under various operating situations was accurately anticipated by the electro-thermal analysis. Because current and Joule heating have a quadratic connection, the simulation results show that current magnitude is the primary factor influencing temperature growth.Current density contours revealed crucial areas close to terminal connections, and the electrical potential distribution verified appropriate current flow via the conductor. Increased Joule heat generation in certain areas resulted in localized temperature increases and the creation of hotspots.Temperature contours showed that the great thermal conductivity of copper efficiently disperses produced heat across the busbar. Nonetheless, the places of current entry and departure were consistently shown to have the greatest temperatures. The direction and amount of thermal energy transmission were further verified by heat flux and thermal gradient investigations.Comparison studies showed that increasing current levels significantly increase operating temperature, whereas elevated ambient temperatures reduce cooling effectiveness and further increase thermal stress on the busbar. Overall, the findings show that, throughout the examined working range, the busbar design maintains thermal stability. Prior to physical testing, the linked electro-thermal simulation technique was successful in finding hotspots, forecasting temperature distribution, and assessing design dependability.

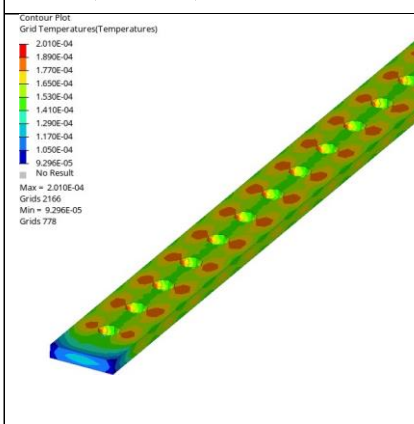
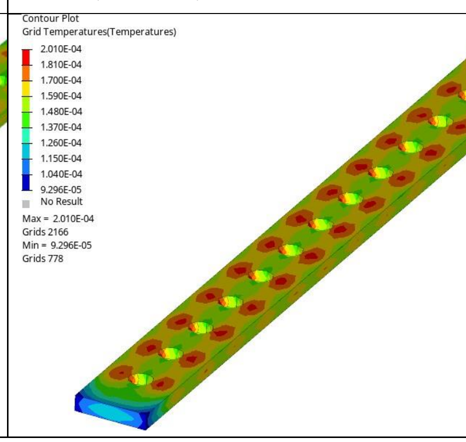
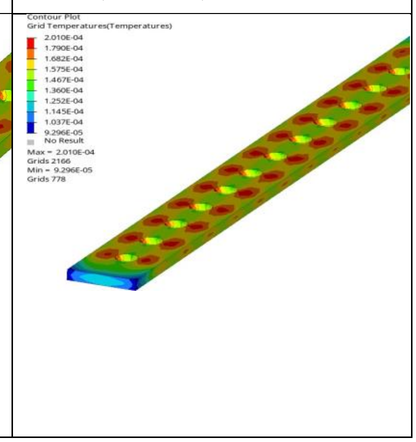
CURRENT IN

CURRENT OUT

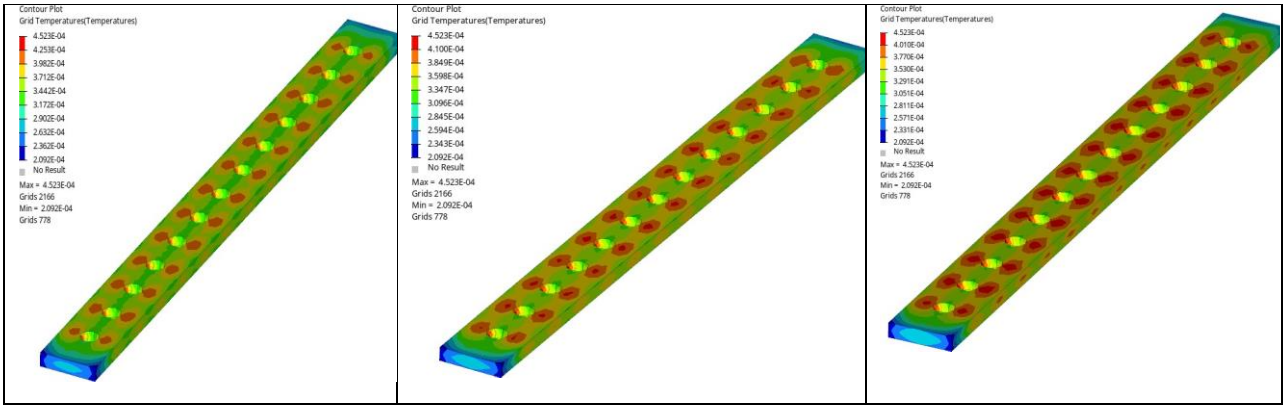


Innovative Journal for Opportunities and Development in Science & Technology,
Volume 6, Issue 7, June. 2026
Paper Id: – IJ2025V6I7 -6 A-D
ISSN 2582-6026

CURRENT = 100A AMB. TEMP. = 25°C	CURRENT = 100A AMB. TEMP. = 40°C	CURRENT = 100A AMB. TEMP. = 50°C
		

CURRENT = 200A AMB. TEMP. = 25°C	CURRENT = 200A AMB. TEMP. = 40°C	CURRENT = 200A AMB. TEMP. = 50°C
		

CURRENT = 300A AMB. TEMP. = 25°C	CURRENT = 300A AMB. TEMP. = 40°C	CURRENT = 300A AMB. TEMP. = 50°C



VI. CONCLUSION

This study effectively used the Finite Element Method (FEM) in Altair Hyper Mesh and Opti Struct to examine the electro-thermal behavior of a copper busbar under various current and ambient temperature conditions. A combined electrical and thermal study was performed on a three-dimensional finite element model of the busbar. The distribution of electrical potential, current density, Joule heat production, temperature distribution, heat flow, thermal gradients, and hotspot development were all assessed. The simulation findings showed that the flow of electrical current causes Joule heating inside the conductor, which raises the temperature. Heat dissipation conditions, ambient temperature, and current magnitude all had a significant impact on the temperature distribution. The finite element method made it possible to identify crucial areas inside the busbar and accurately visualize thermal behavior. The simulation methodology's validity was verified by comparison with theoretical calculations. The total research proved that the busbar design can be successfully assessed using electro-thermal finite element methods and performs properly within the examined operating range.

REFERENCES

[1] Rao, S. S., "The Finite Element Method in Engineering", 6th Edition, Butterworth-Heinemann, 2017.

[2] Altair Engineering Inc., Hyper Mesh User Guide, Altair 2022 Documentation.

[3] Altair Engineering Inc., Opti Struct Solver Reference Guide, Altair 2022 Documentation.

[4] Altair Engineering Inc., HyperView Post Processing Guide, Altair 2022 Documentation.

[5] Copper Development Association (CDA), Copper Busbar Design and Applications Handbook.