Investigation on the Thermal Performance of Bus Duct Conductor using Computational Fluid Dynamics

Mark Selvan, Mohd Sharizal Abdul Aziz & Yu Kok Hwa

School of Mechanical Engineering, Universiti Sains Malaysia, 14300 Nibong Tebal,
Seberang Perai Selatan, Penang, Malaysia

msharizal@usm.my

ABSTRACT

A conductor is the current carrying component in electric power distribution or bus duct. For material efficiency and cost competitiveness, global brand bus duct offers lower conductor size to carry the same rating current. As the conductor constitute close to 80% of the total cost of bus duct, local bus duct manufacturer must find a way to reduce the conductor size in order to carry the same current rating by improving heat dissipation, decrease contact resistance and improve the current carrying capacity. Carrying out experimental test can be increasingly costly and time consuming. The most cost-effective method is to develop a simulation model which can be used to investigate the thermal performance of the bus duct.

Key Words: Computational fluid dynamics

1. INTRODUCTION

Bus bar conductors have become essential to the power distribution system for factories and large buildings. Bus bar connects a variety of elements from generators, transmission lines to loads. It also offers a wide variety of interconnection methods and at the same time provides superior thermal performance. Thermal performance of the bus bar conductor is of utmost importance because the current carrying capacity is directly related to the temperature rise of the conductor. Several factors contribute to the thermal performance of the bus bar conductor such as the amperage, cross sectional dimensions, layout and thermal conductivity (Thirumurugaveerakumar et. al., 2014). Bus bar conductors typically comprise of aluminium housing whereas the conductors are of copper alloys. In order to improve the thermal performance of the bus bar conductor, optimizing the design of the heat sink of the housing is the aim of the research. IEC 60439-1 and IEC 60439-2 lays out a detail guideline to gauge the thermal performance of the bus duct housing. These includes many expensive laboratory test. In order to save cost on product development, a robust and adequate theoretical design would be needed. This would allow for the exploration of several different designs at a relatively low cost. In this research, Ansys Fluent was used as the computational solver for the present work. Ansys was used to model the thermal equations for two primary objectives; to determine the temperature distribution in the bus bar and to know the heat generated or absorbed by it (Delgado et.al., 2017).

Heat sinks are employed to dissipate heat from a hot object more effectively to the ambient through either force convection or natural convection. Alot of research and development have been made in many industries to lower the overall weight, space and cost of integrating heatsinks to devices. Geometry plays an integral role in the design and development of high performancing thermal heat sink. Some common geometrical modifications that are proven to improve the thermal performance of fins are interruptions, slots and perforations (Dhaiban and Hussein, 2020). Kobus and Oshio (2005) carried out a detailed study to investigate the thermal performance of a pin-fin heat sink and at the same time developed a robust theoretical model that was capable of predicting the thermal performance of various geometrical patterns of a heat sink. They concluded that as the diameter of the pin decreased, the convective heat transfer coefficient increased. It was also found that in near the natural convection domain, increasing the pitch size, decreased the fin bundle effect which in turn increases the convective heat transfer coefficient. Liu et. al (2018) carried theoretical and numerical studies for printed circuit heat exchangers (PCHE) and concluded that fin effectiveness is enhanced by several design parameters such as the material choice of the fin, reducing the ratio of fin thickness to channel radius and operating under low convective coefficient environment. Fin efficiency on the other hand was improved similarly by choosing materials of high thermal conductivity, thick fin and also low convective coefficient of the fluid. The grid independence test is of utmost importance in any simulation study. This is to ensure the results are independent of the mesh size. In the investigation of flow dynamics in the nasal cavity, Inthavong et. al (2018) discovered that mesh independence based on the convergence of flow parameters on a single line is an inadequate approach as the rest of the domain is ignored. This could result to certain regions of poorly converged mesh. However, mesh independence based on a single averaged value of global flow parameter misses locally poor mesh convergence. Thus, both local and global flow parameters should be examined in a mesh independence test.

2.0 METHODOLOGY

2.1. Governing equations

The governing equation for the heat conduction through a solid is given by:

$$k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+q=\rho c\frac{\partial T}{\partial t} \qquad (1)$$

$$k\nabla^2 T+q=\rho c\frac{\partial T}{\partial t} \qquad (2)$$

Where,

K = Thermal conductivity (W/K.m)
t = Time
T = Temperature (K)
q = Rate of heat flux/conveection/radiation
ρ = Density of material (kg/m3)
c = Specific heat capacity (J.kg/K)

$$\rho\left(\frac{\partial u}{\partial t}+u\cdot\nabla u\right)=-\nabla\rho+\nabla\cdot\left(\mu(\nabla u+(\nabla u)^T)-\frac{2}{3}\mu(\nabla\cdot u)I\right)+F \qquad (3)$$

Where,

U = Fluid Velocity
p= Fluid pressure
μ = Fluid Dynamic Viscosity

The first term of equation 2 describes the rate of heat conduction, the second term describes the heat flux, and the third term describes the rate of energy storage inside the volume. Equation 3 is the Navier-Stokes equations that govern the motion of fluids and can be seen as Newton's second law of motion for fluids. The different terms correspond to the inertial forces, pressure forces, viscous forces, and external forces applied to the fluid.

2.2 Objective

To develop a simulation model using Ansys 2021 to study the thermal performance of a bus duct housing. Bus duct housings have thermal performance requirements as laid out by IEC 60439-1 and IEC 60439-2. The temperature rise at any points of the bus duct housing shall not exceed 55°C above ambient temperature when operating at rated load current. Busduct system shall be able to operate at full rated current at a maximum ambient temperature of 40°C without derating.

2.3 Grid Independence Test

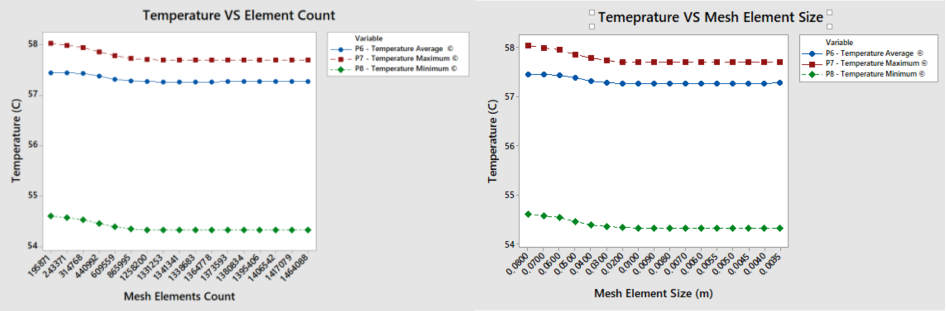


Figure 1: Graph of Temperature VS Mesh Element Count & Graph of Temperature VS Mesh Element Size

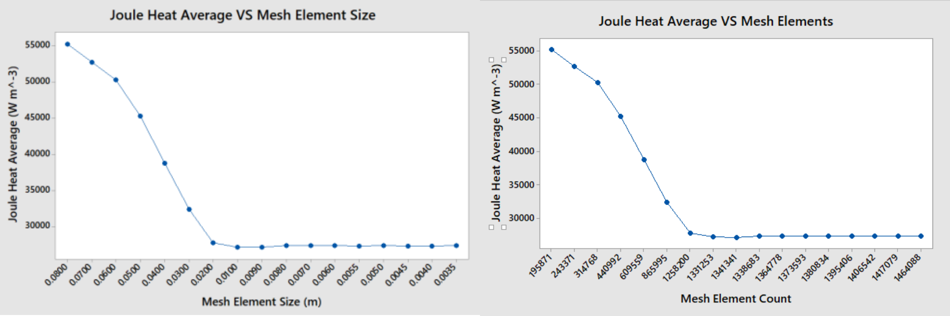


Figure 2: Graph of Joule Heat Average VS Mesh Element Size & Graph of Joule Heat Average VS Mesh Element Count

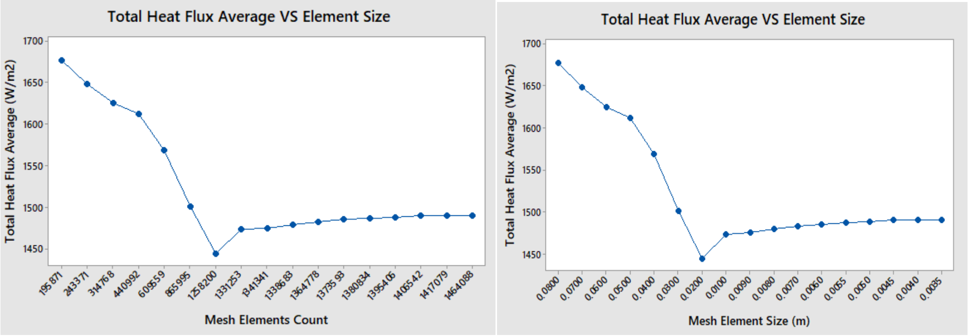


Figure 3: Graph of Total Heat Flux Average VS Mesh Element Count & Graph of Total Heat Flux Average VS Mesh Element Size

Figure 1 shows the average temperature of the two bodies in interest (conductor and heatsink) as the element size is reduced. No significant fluctuation in temperature was observed. The temperature range was steadily maintained between 57.5C to 58C. However, it was observed for both Average Joule Heat and Average Total Heat Flux, both readings decreased significantly until element size of 0.01 and continued to have a steady reading as the element size was reduced. Thus, in order to find the optimal balance between computational resources and result accuracy, element size of 0.01 which resulted to an element count of approximately 1.3 million, was the choice to go with. With an element size of 0.01, the maximum skewness obtained was 0.988943, whereas the minimum orthogonal quality was 0.012672.

2.4 System Coupling

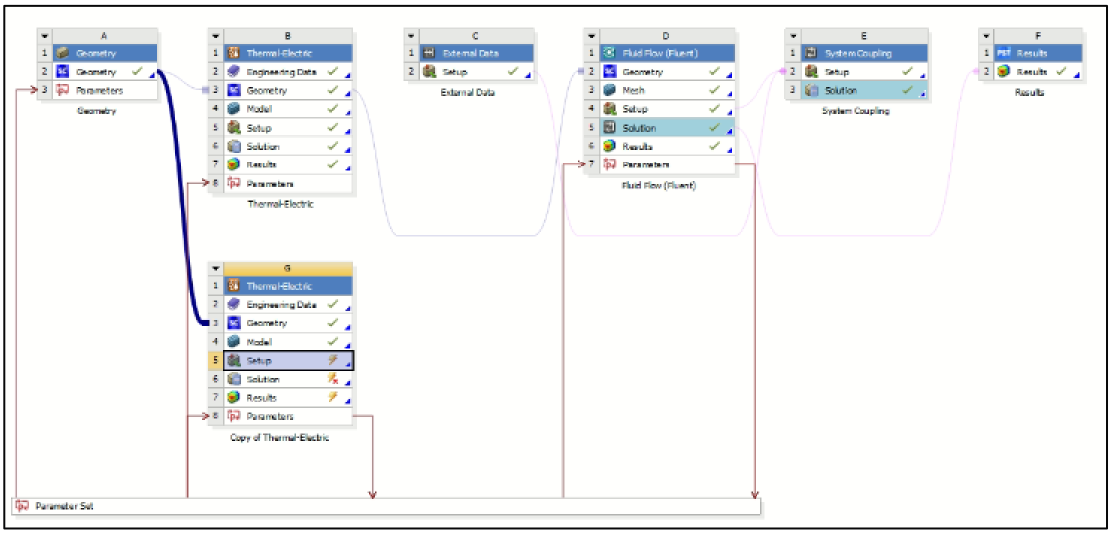
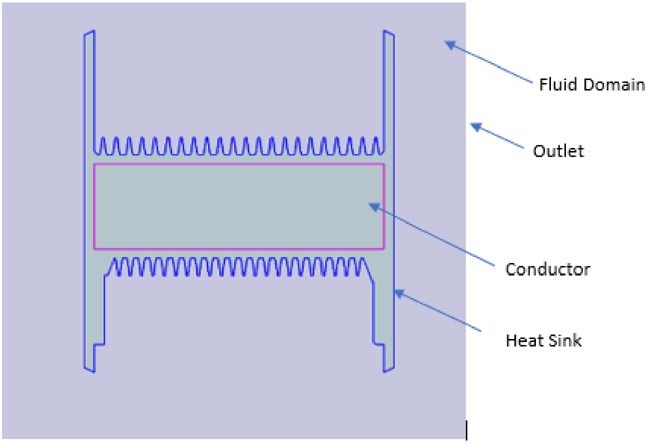


Figure 4. Project Schematic in Ansys Workbench

Figure 4 shows the project schematic the simulation model setup. 2 system analysis were employed, the Thermal-electric system and the Fluent system. The thermal-electric system was employed to model the joule heating, conduction and convection of heat through the bus duct. An approximate value for the convection coefficient was used initially used in system B. Once the solution was completed, the data was transferred to the External Data component for system coupling together with the Fluent system. The analysis setting for the system coupling was set to 20 seconds with 0.01 time step. Upon completion of the simulation iterations in system coupling, both the Heat Transfer Coefficient and the Wall Heat Flux Temperature data was exported to Copy of Thermal Electric system (G). Similar boundary conditions were used for the Copy of Thermal Electric System.

2.5 Fluent Setup

<table>
<tr><th>Items</th><th>Description</th></tr>
<tr><td>Gravity</td><td>Enabled (9.81 ms2 in +z direction)</td></tr>
<tr><td>Model</td><td>k-epsilon full realizable</td></tr>
<tr><td>Material</td><td>Heatsink(Al), Conductor(Copper), Insulator(EM6), Fluid(Air), all material are from fluent database except EM6 material properties provided by Furutec</td></tr>
<tr><td>Fluid operating temperature</td><td>35°C</td></tr>
<tr><td>Cell Zone</td><td>Heatsink (Solid), Conductor(Solid), Fluid Domain(Fluid)</td></tr>
<tr><td>Heat source</td><td>Conductor (35 000Wm-3)</td></tr>
<tr><td>Boundary Condition</td><td>


Boundary condition for outlet set to zero gauge pressure to simulate natural convection</td></tr>
<tr><td>Method</td><td>SIMPLE</td></tr>
<tr><td>Initialization</td><td>Standard, all default settings except temperature set to 35C</td></tr>
<tr><td>Calculation</td><td>Number of time step(1500), Time Step Size(0.01s), Max Iteration/Time Step (1)</td></tr>
</table>

2.5 Thermal-Electric Setup

Items	**Description**
Current	Conductor face, 1500A
Voltage	Opposite conductor face, 0V
Symmetry region	2 conductor faces (current and voltage faces)
Radiation	0.2 emissivity, ambient temperature at 35C

3.0 RESULT & DISCUSSION

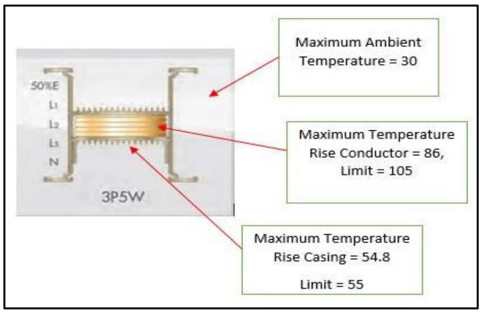


Figure 5. Experimental data from company

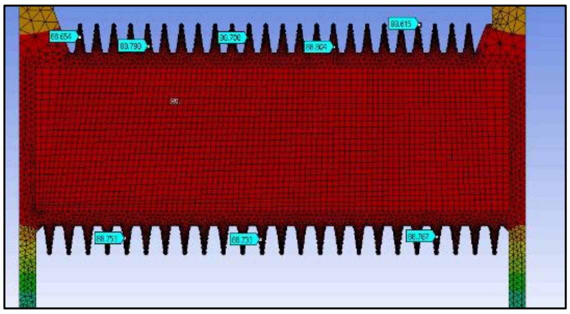


Figure 6. Temperature results from simulation model.

Figure 5 shows the temperature of the bus duct housing when the experiment was conducted. The maximum temperature rise of the conductor was found to be 86°C, thus the absolute temperature was at 116°C, whereas the maximum temperature rise of the casing was 54.8 resulting to an absolute temperature of 84.8°C. Figure 6 shows the temperature distribution of the cross section of the bus duct. The temperature of the fins at 8 different fin instances shows a strong correlation to the experimental results. The temperature readings were all within 5% error from the experimental data.

Position	Experimental Data	Simulation Data	Error (%)
1	84.8	88.65	4.54
2	84.6	88.81	4.97
3	84.9	88.71	4.49
4	84.8	88.80	4.71
5	84.7	88.62	4.62
6	85.1	88.75	4.29
7	85.0	88.73	4.39
8	84.9	88.77	4.56

The simulation model developed shows a strong correlation to the experimental data. The error was within 5% of the experimental data. A number of things were identified to be possible reasons for the deviation from the experimental data. The thin insulator layer with thickness of 0.000125 was not taken into consideration. The thermal conductivity of the material is 0.14 W/(cm·K). This may result to a drop in temperature on the thermal fin thus reducing the error.

4.0 CONCLUSION

The simulation model developed was validated with experimental data and was proven to have a deviation of approximately 5%. Thus, it can be used for the investigation of different geometrical designs for future research.

REFERENCES

C. J. Kobus and T. Oshio, “Development of a theoretical model for predicting the thermal performance characteristics of a vertical pin-fin array heat sink under combined forced and natural convection with impinging flow,” *Int. J. Heat Mass Transf.*, vol. 48, no. 6, pp. 1053–1063, 2005, doi: 10.1016/j.ijheatmasstransfer.2004.09.042.

F. Delgado, C. J. Renedo, A. Ortiz, I. Fernández, and A. Santisteban, “3D thermal model and experimental validation of a low voltage three-phase busduct,” *Appl. Therm. Eng.*, vol. 110, pp. 1643–1652, 2017, doi: 10.1016/j.applthermaleng.2016.09.002.

H. T. Dhaiban and M. A. Hussein, “The optimal design of heat sinks: A review,” *J. Appl. Comput. Mech.*, vol. 6, no. 4, pp. 1030–1043, 2020, doi: 10.22055/jacm.2019.14852.

K. Inthavong, A. Chetty, Y. Shang, and J. Tu, “Examining mesh independence for flow dynamics in the human nasal cavity,” *Comput. Biol. Med.*, vol. 102, no. June, pp. 40–50, 2018, doi: 10.1016/j.compbiomed.2018.09.010.

S. Liu, Y. Huang, and J. Wang, “International Journal of Thermal Sciences Theoretical and numerical investigation on the fin effectiveness and the fin efficiency of printed circuit heat exchanger with straight channels,” *Int. J. Therm. Sci.*, vol. 132, no. June, pp. 558–566, 2018, doi: 10.1016/j.ijthermalsci.2018.06.029.

S. Thirumurugaveerakumar, M. Sakthivel, and S. Valarmathi, “Experimental and analytical study on the bus duct system for the prediction of temperature variations due to the fluctuation of load,” *J. Electr. Eng. Technol.*, vol. 9, no. 6, pp. 2036–2041, 2014, doi: 10.5370/JEET.2014.9.6.2036.