A Numerical Investigation of Variably Heated Enclosure to Enhance its Heat Transfer Rate using CFD

P. Ragupathi¹, S. Aravind¹, D. Sathish Kumar¹, G. Vignesh¹

¹Department of Mechanical Engineering, Karpagam Academy of Higher Education, Coimbatore-641021, Tamil Nadu, India

ponragupathi@gmail.com

Abstract. The flow simulation with the help of computational tools helps in solving many engineering problems related to fluid dynamics. This in turn reduces the time and cost incurred in doing real time experimental investigations. In this present research work, computational investigation was carried out in a variably heated enclosure to enhance the life span of the electronic components inside the enclosure. The investigation was performed over a twilight enclosure system, the purpose of this system is switch on or off the street lights with support of sensors automatically. This particular enclosure is subjected to heating as it is continuously exposed to sunlight through the day. This enclosure consists of delicate electronic components which are sensitive to heat. The heat transfer can be enhanced in this system by the prevailing natural flowing air in the surrounding by free convection process. To enhance and optimize this heat transfer process, numerical simulation using computational tools was carried out by changing the physical layout of the enclosure. To perform this numerical simulation, flow simulation module available in the SolidWorks was used. Also, to model this system, the same modeling software was used. From the outcome of this numerical flow simulation, it showed a drop of 36.3% in the heat transfer rate when compared to the existing model to the proposed model used for the simulation. By using the modified and proposed model, the life span of the entire system will be improved.

Key words: Computational tool, Enclosure, Free convection, Heat Transfer, Simulation.

1. Introduction

In the modern world, automotive revolution is going on in a fast pace. The automotive sectors are trying in recent days, the use of sensor based electronic components enclosed in an enclosure is widely used in many places and also, they are exposed to the surrounding conditions. The surrounding conditions here it indicates the heat transfer effect. All electronic components work properly upto the indicated temperature range. If heat accumulates owing to radiative heat transfer, then the temperature goes beyond the operating temperature limit suggested by the component manufacturer and leads to malfunction of such electronic components. One such example selected for this study is the twilight switch box which is generally used in street lights on or off automatically based on sensor at an average height of 6 to 6.5 m from ground level. This switch box is subjected to radiative heat transfer as it is exposed to sunlight for long duration. Such radiative heat transfer is accompanied with the natural convection, which tries to transfer the heat from the switch box to the surrounding thereby enhancing the life of switch box. The rate of convection depends on many parameters one such parameter can be considered to be the design of the switch box. Markatos et al. [1] in their study performed numerical analysis on a square cavity subjected to natural convection under laminar and turbulent cases. They also
The inclination angle. It showed that a basic power.

such numerical study is conducted to compare the heat transfer by natural convection in SolidWorks simulation module. The test results revealed that the modified switch box has effective heat transfer.

2. Problem Description
The twilight switch box selected for the study is made of ABS plastic and glass material which is placed in a lamp post for sensing the light intensity and automate the ON/OFF of the street light. Exposing of such switch box with sensing element in sunlight for the entire day results in high radiation phenomenon, as the major part is made of glass. This results in the faulty working of the sensor due to prolonged...
exposure to such radiation heat transfer. In this present work, it is tried to reduce such radiation heat transfer by two types of modified switch boxes. The model is created using SolidWorks and analysed using SolidWorks flow simulation. And the results are compared between the prevailing model (figure.1) and the proposed model (figure.2).

3. Methodology

The created model is exported to the SolidWorks flow simulation module. Any CFD packages have these three stages and they are the pre-processing, Solution and Post processing. The pre-processing stage includes the selection of geometry, type of analysis to be carried out, materials selection, boundary conditions and mesh settings [11, 17]. The existing model and the proposed models are corrected in order to avoid the meshing error and to have a reduced time while it undergoes analysis. Then the type of analysis is selected to be external flow heat transfer along with radiation. The materials are set to be ABS plastic for the switch box cover and the front panel as glass material, the fluid medium is selected as air.

The boundary condition for air is set to be with flow velocity of 8, 5 and 3 m/s along x, y and z directions. The air is set to standard atmospheric pressure of 1.01325 bar and temperature around 27°C. The boundary condition for radiation is set with solar intensity of 1000 W/m² along the y and z directions based on the sensor box position. And the corresponding surfaces subjected to radiative heat transfer is selected. The initial mesh level is set to be at third level with narrow channel refinement. The mesh produced is shown in figure. 3. With these initial steps involved in pre-processing stage, the models are subjected to analysis. That is the solver is made to run to solve the governing equations related to the above set of conditions.
4. Results & Discussion

The SolidWorks flow simulation solver takes some time to analyse the computational domain and ends up with the computational results in the post processing stage. The following figures 4 & 5 shows the air flow vector along x, y & z directions for the existing and proposed model.

Figure 4. Air flow vector in (a) X axis (b) Y axis (c) Z axis around the existing assembly

Figure 5. Air flow vectors in (a) X axis (b) Y axis (c) Z axis around the proposed assembly
The following contours in the figures 6 & 7 shows the surface temperature of the switch box for the existing model and the proposed model. The surface temperature contours show that the surface temperature is more for the prevailing model when related to the proposed model. Around 25.4% more than the proposed model when related to the prevailing model [15].

![Figure 6 Surface temperature of (a) Existing model (b) the proposed model](image)

The maximum total heat flux for the existing model is about 1060 W/m², where as it is around 1036 W/m² for the proposed model. But the minimum heat flux is 42.2% higher for proposed model when compared to the existing model. The total heat flux contours for the existing & proposed model is shown in the figure 6.

![Figure 7 Total Heat flux of (a) Existing model (b) the proposed model](image)

The figure 8 depicts the resulted surface heat flux for the described conditions which shows that the surface heat flux has reduced from a maximum of 1611.5 W/m² for existing model to 1247 W/m² for the proposed model. Similarly, the minimum surface heat flux for the proposed model has reduced to about 27.8% when compared to the existing model.

![Figure 8 Surface Heat flux of (a) Existing model (b) the proposed model](image)
The surface temperature of the sensor inside the enclosure has reduced from a maximum value of 64.3°C for existing model to 52.2°C for the proposed model, this in turn definitely improves the efficiency of the sensor in rendering its need. This also increases the life time of the sensor [14-17]. The figure 9 shows the surface temperature contour of the sensor in existing & proposed model.

![Figure 9](image1.png)

Figure 9 Surface temperature on the sensor of (a) Existing model (b) the proposed model

From the graph shown in the figure. 10 shows the substantial decrease in the emissive power for the proposed model from the existing model.

![Figure 10](image2.png)

Figure 10 Emissive power of the existing & proposed model

![Figure 11](image3.png)

Figure 11 Heat transfer rate for increasing Rayleigh number from $10^4$ to $10^9$ for existing model
The graphs shown in the figure 11 & 12 shows the heat transfer rate for increasing Rayleigh number in the range of $10^4$ to $10^9$. The average Nusselt number varied from range of 6 to 105 for different Rayleigh number. The average heat transfer coefficient varied from 247.8 to 4406.0 W/m²K for varying Nusselt number. These properties were calculated based on the co-relations available in the standard heat transfer data tables.

From the graphs, it is found that the heat transfer rate to the switch box has a drop of 36.3% from the existing switch box model to the proposed switch box model [15-17]. The heat transfer rate calculated analytically based on the existing co-relations are in par with the heat flux values that are found numerically using the SolidWorks flow simulation module.

5. Conclusion

The present study performed on the twilight switch box with the objective of reducing the rate of radiative heat transfer to avoid malfunction of the sensor for automated ON/OFF of the street lights has resulted in a considerable alteration of changing the existing switch box model to the proposed model. As the proposed design has better heat transfer characteristics when related to the existing design. Various heat transfer parameter like the surface temperature, total heat flux, surface heat flux, emissive power and the heat transfer rate for the proposed design is better than the prevailing design. Hence it can be concluded that by replacing the existing model with the proposed model will certainly result in increased sensitivity as well as increased life of the sensor.

The proposed model has been manufactured and the numerical study performed has helped to increase the quality of the twilight switch box.

References
[1] Markatos, N.C. and Pericleous, K.A., 1984. Laminar and turbulent natural convection in an enclosed cavity. International Journal of Heat and Mass Transfer, 27(5), pp.755-772.
[2] Ostrach, S., 1988. Natural convection in enclosures, 1175-1190.
[3] Dehghan, A.A. and Behnia, M., 1996. Combined natural convection–conduction and radiation heat transfer in a discretely heated open cavity, 56-64.
[4] Manz, H., 2003. Numerical simulation of heat transfer by natural convection in cavities of facade elements. Energy and Buildings, 35(3), pp.305-311.
[5] Chan, Y.L. and Tien, C.L., 1985. A numerical study of two-dimensional laminar natural convection in shallow open cavities. International Journal of Heat and Mass Transfer, 28(3), pp.603-612.
[6] Polat, O. and Bilgen, E., 2002. Laminar natural convection in inclined open shallow cavities. International Journal of Thermal Sciences, 41(4), pp.360-368.
[7] Bilgen, E. and Oztop, H., 2005. Natural convection heat transfer in partially open inclined square cavities. International Journal of Heat and Mass Transfer, 48(8), pp.1470-1479.

[8] Huelsz, G. and Rechtman, R., 2013. Heat transfer due to natural convection in an inclined square cavity using the lattice Boltzmann equation method. International journal of thermal sciences, 65, pp.111-119.

[9] Sahu, K.B. and Singh, R.K., 2014. Analysis of heat transfer and flow due to natural convection in air around heated triangular cylinders of different sizes inside a square enclosure. Procedia Engineering, 90, pp.550-556.

[10] Pinto, R.J., Guimarães, P.M. and Menon, G.J., 2016. Numerical Study of Natural Convection in Square Cavity with Inner Bodies Using Finite Element Method. Open Journal of Fluid Dynamics, 6(02), p.75.

[11] Ji, Y., 2014. CFD modelling of natural convection in air cavities. CFD letters: An International Journal, 6(1), pp.15-31.

[12] Mondal, B. and Mishra, S.C., 2008. Simulation of natural convection in the presence of volumetric radiation using the lattice Boltzmann method. Numerical Heat Transfer, Part A: Applications, 55(1), pp.18-41.

[13] Yu, E. and Joshi, Y.K., 2000. Natural convection air cooling of electronic components in partially open compact horizontal enclosures. IEEE Transactions on Components and Packaging Technologies, 23(1), pp.14-22.

[14] Arik, M., Becker, C.A., Weaver, S.E. and Petroski, J., 2004, January. Thermal management of LEDs: package to system. In Third international conference on solid state lighting (Vol. 5187, pp. 64-75). International Society for Optics and Photonics.

[15] Anton, R., Jonsson, H. and Moshfegh, B., 2009. Detailed CFD modelling of EMC screens for radio base stations: a parametric study. IEEE Transactions on Components and Packaging Technologies, 32(1), pp.145-155.

[16] Galloway, J.S., 1998, May. Thermal analysis of portable electronic products. In ITherm'98. Sixth Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (Cat. No. 98CH36208) (pp. 1-7). IEEE.

[17] Lee, T.Y., Chambers, B. and Mahalingam, M., 1995. Application of CFD technology to electronic thermal management. IEEE Transactions on Components, Packaging, and Manufacturing Technology: Part B, 18(3), pp.511-520.