Available online at www.ijapie.org

International Journal of Advanced
Production and Industrial Engineering

IJAPIE-SI-IDCM 604 (2017) 16–19

Special Issue on

INNOVATIVE DESIGN
&
COMPUTATIONAL
MODELLING

Modelling and Simulation of Thermoelectric Peltier Effect Cooling Unit

Vineet Mathew*, Akhilesh Arora, B. B. Arora, Saurabh Dubey,
Rayhan Ali, Sushant Sahasrabuddhe
Delhi Technological University, Delhi-110042
Email: vineetdtu@gmail.com

Abstract : *Thermoelectric coolers being a Freon free technology with the added advantage of flexibility in mounting position and precise temperature control, are being extensively used in portable applications. The relatively lower efficiency compared to conventional cooling systems is neglected owing to its portability and pollution free nature. The aim of this study is to successfully model a Cooling Unit using a TEC and extended hot side and cold side heat transfer surfaces. The cooling medium is chosen as air. The heat transfer surfaces are Aluminium Extrusions. Modelling and simulation was done on ANSYS Icepak. ANSYS Icepak is a computational package for the thermal management of electronic components. ANSYS Icepak solves for fluid flow and includes all modes of heat transfer i.e. conduction, convection and radiation. In this study the effect of the radiation mode of heat transfer was neglected. A suitable turbulence model is selected that can properly describe flow in the system. After successfully meshing the assembly and setting up the prerequisite conditions, velocity and temperature contours have been plotted to show the nature of flow and temperature distribution across the surroundings as well the cooling elements. The analysis results can assist in the development of cooling strategies to further increase its effectiveness.*

Keywords: *TEC, Peltier Effect, Cooling Unit, ANSYS Icepak, Heat Transfer, Heat Sinks, CFD, Modelling, Simulation, Turbulence Model*

I. INTRODUCTION

Thermoelectric effect refers to the conversion of voltage difference into a temperature gradient or vice-versa. Conversely, a temperature difference will generate a voltage difference. Therefore, by connecting a TEC to a power source, a temperature difference can be generated on either side of the TEC. The thermoelectric effect can be used to generate electricity, measure temperature and also act as a temperature regulator as the hot and cold side of a TEC can be interchanged as it is dictated by the polarity of the applied voltage.

The Thermoelectric effect encompasses mainly two effects, The Seebeck effect and the Peltier Effect. The Peltier effect is named after French physicist Jean Charles Athanase who discovered it in 1834. It is the generation of a hot and cold side at an electrified junction of two different conducting materials. When a current is made to flow through a junction between two conductors A and B, heat may be generated (or removed) at the junction. The heat generated at the junction per unit time, $\dot{Q}$, is given by:

$$\dot{Q} = (\pi_A - \pi_B)I$$

Where Π_A ,Π_B are the Peltier coefficients of conductors A and B respectively, and I is the electric current (from A to B). Joule heating and thermal gradient effects may also effect heat generation. Peltier effect TECs allow cooling below ambient temperature, but unlike other cooling systems that allow this (vapour phase refrigeration), they are less expensive and more compact. Peltier elements are solid-state devices with no moving parts; they are extremely reliable and do not require any maintenance. These are extremely light weight components.

The temperature difference generated by a Peltier TEC is dependent on many variables. The TEC model used in the simulation is Melcor_CP1.0-127-06. The Heat Transfer is inversely proportional to the temperature difference between the Hot side and the Cold side. Hot Side temperature and Ambient Temperature also affect the performance of the Peltier Module.

Ozturk and Tari [1] did a Computational Fluid Dynamics Modelling on a Computer Chassis by mainly concentrating on the comparison of computational results and documented experimental results. The study showed reasonable results with the zero equation turbulence model citing higher computational costs of the k-epsilon model. Yu and Webb [2] used ANSYS Icepak to simulate a 80W CPU to find a probable heat management solution for AGP and PCI cards. The cooling solution was based on a vertical cooling duct. Chang [3] using Icepak was able to reduce the temperature of a 30W CPU socket by 10^{o}C using ducts when compared to a non-ducted setup. Alvin and Chu [5] studies the thermal management of LED lamps by getting a numerical solution using ANSYS Icepak. Additionally heat rejecting materials were also analysed in terms of their effectiveness.

The objective of this study is to successfully model a Portable Cooling Unit using ASYS Icepak by selecting an appropriate turbulent flow model for this application. The Models for Heat Sink and Cold Sink surfaces and TEC is selected from the comprehensive library in ANSYS Icepak.

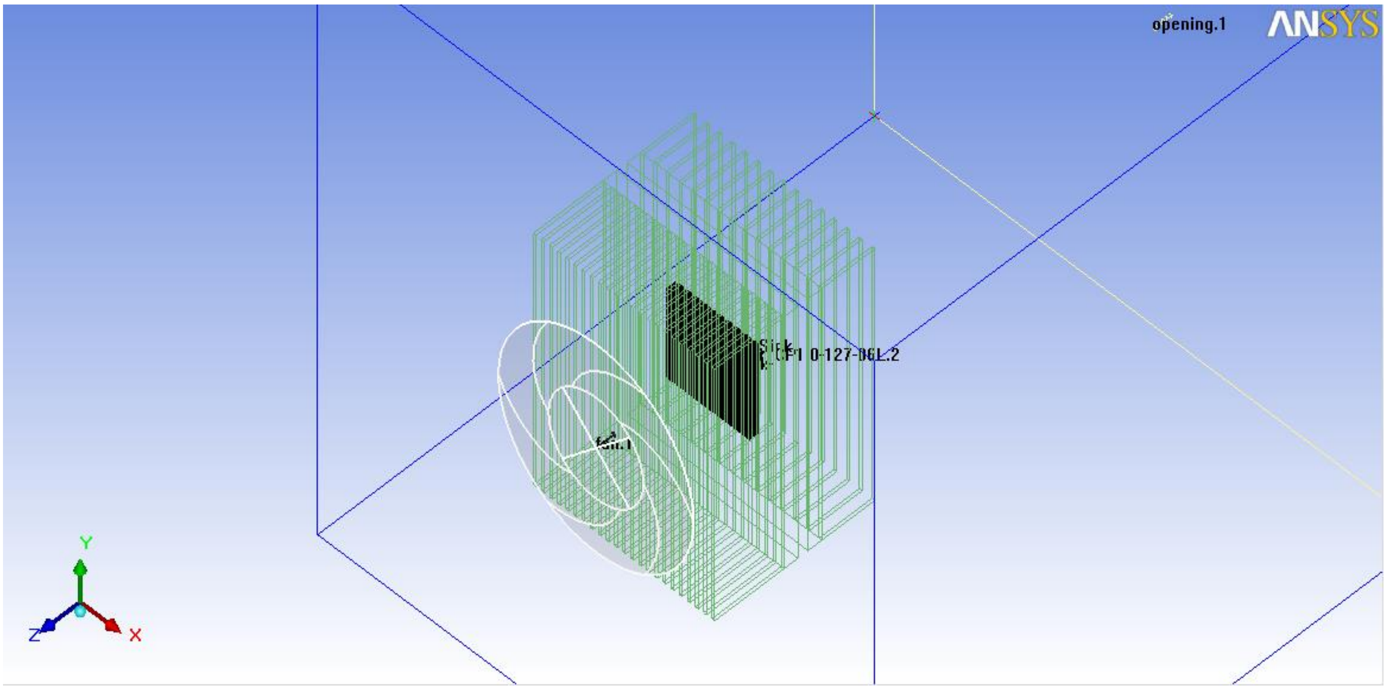

Fig. 1 Isometric view of the assembly

ANSYS Icepak is specifically suited for CFD and Thermal Analysis for electronic components [4]. ANSYS Icepak uses FLUENT as its solver.

II. GEOMETRY

The model consists of a Peltier model Melcor_CP1.0-127-06L.2 from the TECs library in ANSYS Icepak. The dimensions for the heat transfer surfaces are specified in the table below. Cabinet Size is 0.2 X 0.1 X 0.1 (m).

The peltier module is sandwiched between two extended heat transfer surfaces. The interface is assumed to ideal such that no heat is lost at the contact plane. An intake fan is placed at a distance 10mm from the hot side extended heat transfer surface. The flow rate is taken as fixed in terms of volumetric flow rate. This is a reasonable assumption since we are targeting a steady state solution. The fins for both the Heat Sink and the Cold sink are long the Y-Axis.

A. *Extended Heat Transfer Surface Specifications:*

	Heat Sink	**Cold Sink**
Fin Material	Aluminium Extrusion	Aluminium Extrusion
Base Material	Aluminium Extrusion	Aluminium Extrusion
Thermal Conductivity (W/m-K)	204	204
Density (kg/m3)	2800	2800
Dimensions (mm)	85 X 65 X 30	85 X 65 X 24
Base Height (mm)	5	5
Number of Fins	23	15

B. *Cooling Fan Specifications:*

Inner Diameter (mm)	17
Outer Diameter (mm)	35
Flow Rate (cfm)	44

C. *Peltier Module Specifications:*

Hot Side Temperature (^{o}C)	25	50
Qmax (Watts)	25.7	29.1
Delta Tmax (^{o}C)	67	77

The specifications for the Heat Sink, Cold Sink as well as the cooling fan have been taken from a CPU cooling unit available in the market.

III. SOLUTION SETUP

ANSYS Icepak is capable of automatic meshing. Non-Conformal Mesh was created using the automatic unstructured Mesher-HD (Hex dominant). The advantage of generating a non-conformal mesh is that the number of elements is relatively lower and the degree of accuracy remains largely unaffected.

- An initial coarse mesh was generated. Mesh was examined for skewness using cut plains in the geometry

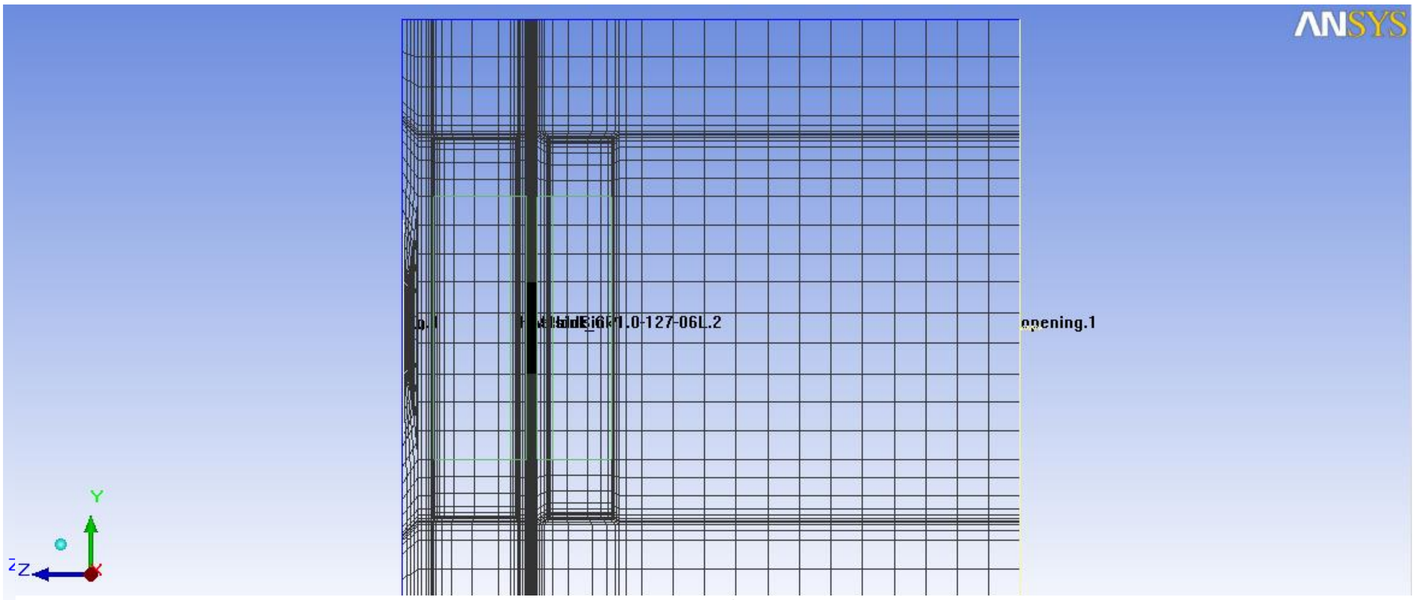

Fig. 2 Y-Z plane view of the generated mesh

- To appropriately capture regions of reversed flow and the formation of wakes, virtual air boxes were placed at such regions and were meshed separately. This is done so that the total number of elements does not exceed the maximum limit and details in the temperature and flow distribution can be accurately visualized.

- Skewness in a few elements is present found near the fan. This is because the unstructured Mesher-HD automatically creates a skewed mesh to better reciprocate the air flow from the fan.

A. *Mesh properties*

Total elements in the mesh are 220330 with the number of nodes 230530. Quality check of the mesh produced a face alignment quality of the range 0.0819063 ->1. Since this is the acceptable range, the mesh is appropriate.

In the study, Flow as well as Temperature was solved for. The effects of radiation were neglected with the scope restricted to conduction and convection. Ambient Temperature selected was 35^{o}C. Examination of the model reveals a clear possibility of flow separation as well reversed flow; therefore the turbulent regime model selected was the Enhanced Realized K-epsilon model [6]. The analysis was conducted to produce a steady state solution.

B. *The following are the convergence criteria:*

Maximum number of iterations	2000
Flow	1e-3
Energy	1e-7
Turbulent Kinetic Energy	1e-3
Turbulent Dissipation Rate	1e-3
Joule heating	1e-7

IV. ASSUMPTIONS

The contact between the TEC, The Heat Sink, and The Cold Sink is assumed to be perfect. The presence of a convective film is neglected. The Cabinet is assumed to be fully insulated from the surrounding. The effects of radiation are neglected. The flow type of the fan is fixed and the intake air is of ambient temperature. Since preliminary investigation reveals the possibility of reversed and separated flow, zero equation turbulence model has not been used since the solution of the simulation may be incomplete or misleading [7].

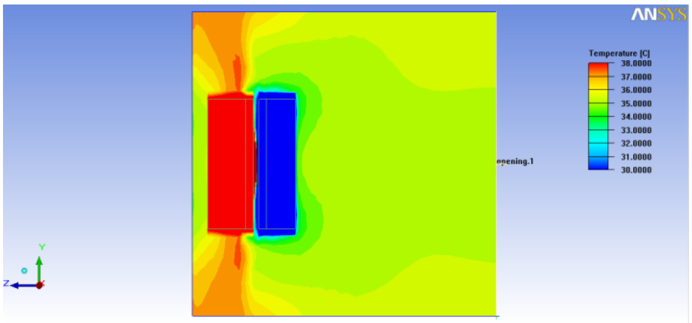

Fig. 2 Temperature Distribution between 30-38 **^{o}C**

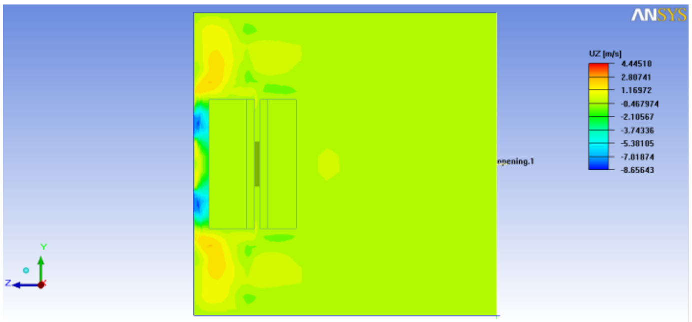

Fig. 4 Y-Z plane z-Velocity contour

Fig. 3 Y-Z plane Temperature Contour

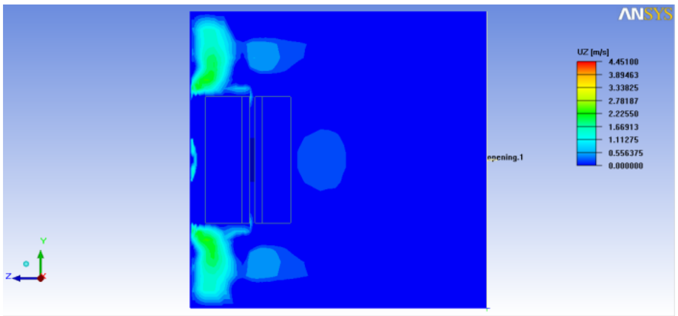

Fig. 5 Y-Z plane reverse flow

The analysis reveals a maximum temperature of 53°C and minimum temperature of 14°C as shown in Fig 2. This is the temperature of the hot side and cold side of the Peltier Module. On further examination in the temperature range of 30°C to 38°C (Fig. 3) the penetration of the cold side temperature of the Peltier Module can be visualized. Velocity Contours in the Z direction (along the model) in Fig 5 show regions of reverse flow (positive values). Further resolution in the reverse flow range can be seen in Fig 6. Reverse flow is generated at the top and bottom of the heat sink which is expected because of the air flow due to the intake fan. A region of reverse flow is also generated right after the cold sink.

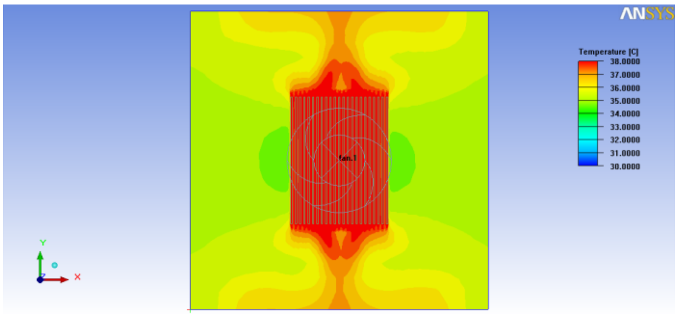

Fig. 6 X-Y plane Temperature Contour at 15% distance from the fan

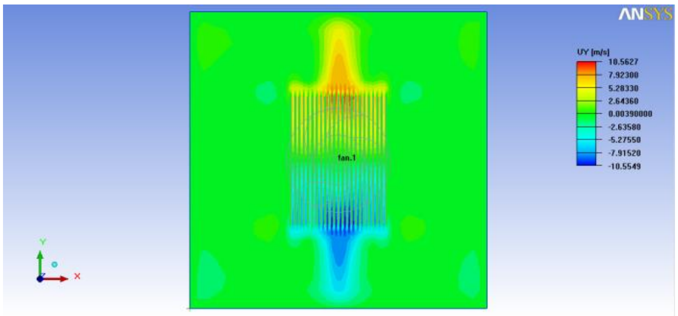

Fig. 7 X-Y plane y-Velocity Contour at 15% distance from the fan

Temperature distribution in the Heat Sink and the Cold Sink shows a concentration at the centre. The temperature bands follow a concave contour with respect to the base of the heat or cold sink. The Heat dissipated from the hot side of the Peltier Module spreads along the length of the fins due the presence of an intake fan. This is also a region of reversed flow. Fig. 7 shows the temperature contour in the X-Y plane at 15% the length of the cabinet. The Y component contour for the velocity closely resembles the temperature contours at the same location (Fig.8). This shows the influence of the intake fan.

V. CONCLUSION

In this study a TEC cooling unit with Hot Cold extended transfer surfaces was modelled and simulated. The following are the insights that could be drawn from this study.

- The zero equation turbulence model may give suitable results when it comes to maximum and minimum temperatures but for an accurate visualization of the distribution of temperature and velocity across the system the k-epsilon model is more suitable.

- To capture details regarding reverse flow, non-conformal meshing using assemblies should be carried out at regions of interest rather than increasing the elements size throughout the system.

- The effect of radiation can be ignored due to the overbearing influence of forced convection.

The analysis was conducted for the cooling unit in air as the surround medium. The cold sink can be inserted into a container of different medium. The methodology used in the setting up this solution can be extended to a water cooling apparatus.

REFERENCES

[1] Emre Ozturk & Ilker Tari *CFD Modeling of Forced Cooling of Computer Chassis*, Engineering Applications of Computational Fluid Mechanics, 2007.

[2] C.-W. Yu, and R. L. Webb, "*Thermal Design of a Desktop Computer System Using CFD Analysis*," Seventeenth IEEE-SEMI-THERM Symposium. 2001.

[3] J. Y. Chang, C. W. Yu, and R.L.Webb, "*Identification of Minimum Air Flow Design for a Desktop Computer Using CFD Modelling*", 7th Intersociety Conference on Thermal and Thermomechanical Phenomena in Elec-tronic Systems, Vol.1, pp. 330-338,2000.

[4] Zahn, B., *"Evaluating Thermal Characterization Accuracy Using CFD Codes - A Package Level Benchmark Study of IcePak™ and Flotherm®,"* Proceedings of 6th Inter-society Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (I-THERM), pp. 322-329, 1998.

[5] Christian Alvin, Webber Chu, Ching-Hung Cheng, Jyh-tong Teng, "Thermal analysis of extruded aluminum fin heat sink for LED cooling application", *Microsystems Packaging Assembly and Circuits Technology Conference (IMPACT) 2011 6th International*, pp. 397-400, 2011.

[6] C.C. Liu, Y.M. Ferng, C.K. Shih,"*CFD evaluation of turbulence models for flow simulation of the fuel rod bundle with a spacer assembly*", Applied Thermal Engineering, Volume 40, July 2012, Pages 389-396

[7] http://jullio.pe.kr/fluent6.1/help/html/ug/node402.htm