IOP Conf. Series: Journal of Physics: Conf. Series **877** (2017) 012011 doi:10.1088/1742-6596/877/1/012011

Modeling of various heat adapter plate 4 and 6 array for optimization of thermoelectric generator element using modified diffusion equation

Defrianto[1,a], W Tambunan[1] and Lazuardi[1]

[1]Department of Physics, FMIPA University of Riau
Kampus Bina Widya Simpang Baru, Pekanbaru 28293, Indonesia
[a] Corresponding author: defriaanto@gmail.com

Abstract. The use of waste heat from exhaust gas and converting it to electricity is now an alternative to harvest a cheap and clean energy. Thermoelectric generator (TEG) has the ability to directly recover such waste heat and generate electricity. The aim of this study is to simulate the heat transfer on the aluminum adapter plate for homogeneity temperature distribution coupled with hot side of TEG type 40-40-10/100 from Firma Eureka and adjust their high temperatures to the TEG operating temperature to avoid the element damage. Modelling was carried out using MATLAB modified diffusion equation with Dirichlet boundary conditions at defined temperature which has been set at the ends of the heat source at 463K and 373K ± 10% on the hot side of the TEG element. The use of nylon insulated material is modeled after Neumann boundary condition in which the temperature gradient is $\partial T/\partial n = 0$ out of boundary. Realization of the modelling is done by designing a heat conductive plate using software ACAD 2015 and converted into a binary file format of Mathlab to form a finite element mesh with geometry variations of solid model. The solid cubic model of aluminum adapter plate has a dimension of 40mm length, 40mm width and also 20mm, 30mm and 40mm thickness arranged in two arrays of 2x2 and 2x3 of TEG elements. Results showed a temperature decrease about 40.95% and 50.02% respectively from the initial source and appropriate with TEG temperature tolerance.

1. Introduction

Micro scale energy sources that produce an order of milliwatt electrical power from solar, vibration, thermal and biological resources has been a part of research on the current electronics design. A new device in micro energy generation systems nowadays is a thermoelectric generator element (TEG), known as Peltier element which in principle generates electric power based on the Seebeck effect utilizing the temperature difference. The power generated is relatively small and proportional to the magnitude of the temperature difference between the two sides of Peltier elements [1].

Some applications use the waste heat from an industrial process and automotive, like waste heat from a car muffler or the temperature difference between the two parts. In automotive vehicles, there is a large power dissipation resulted from fuel combustion in automobile exhaust chamber and as an energy source is still relatively rarely used [2]. A single auto muffler can produce high temperature from 600K up to 900K then it release to the free air. However, the waste heat of car muffler is not in accordance with the characteristics of a commercially available thermoelectric generators, so there needs to be an adjustment between a heat source in the exhaust plate with the surface temperature of a TEG element, so that an optimum power can be generated without damage to the device [3].

Heat transfer in the elements can be expressed mathematically in a diffusion equation. Transient heat conduction (or diffusion) equations with nonlinear source terms have been used as model equations for heat transfer problems in many different areas of mathematical physics, applied science, and engineering. In numerical methods, the boundary element method has become increasingly attractive to scientists and engineers since it has certain advantages over other methods such finite difference method (FDM). One of the advantages is its ability in reducing the dimensionality of a problem by one [4-5].

This study simulated the heat distribution in various form of the adapter plates using a modified diffusion equation. The TEG element used has rectangular shape and was purchased from Fa. Eureca Germany with technical data as given in Table 1.

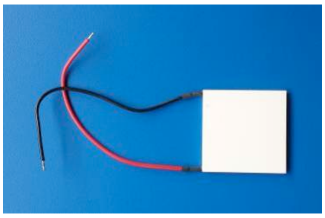


Figure 1. The thermoelectric generator element of Fa. Eureka

Table 1. Thermal, electrical and mechanical properties of TEG type 1-40-40-10/100 [6]

No	Parameters	Symbols	
1	thermal force	α	0.0823 V/K
2	resistance	ρ	1.67Ω
3	thermal conductivity	γ	1.56 W/K
4	maximum operating temperatures	Tmax	150°C
5	size of cold/ hot side	L x B x H	40.0x 40.0 x3.20 mm
6	weight	m	24g

Table 1 shows that the element TEG has maximum operating temperature around 150°C (under temperature of muffler). However, based on previous experiments, devices which operate near the operating temperature can damage TEG terminal served as a current wire. It is due to melting of soldering tin and inhomogeneity of heat distribution on the surface of the TEG resulted instability of the generated electrically output [7]. It is necessary to simulate the optimum homogeneous temperature on the surface of TEG and to be within the tolerance limits and to avoid permanently damage to the TEG elements. The arrangement of 2x2 array (4 elements) and 2x3 (6 elements) provided in this study aims that TEG element can be arranged in series and parallel to provide a large output results.

2. Temperature distribution model

2.1 Model overview

The waste heat generated from auto muffler can be utilized as a heat source for the thermoelectric generator and be connected using a conductive plate as shown in Figure 2. For this purpose, a modified diffusion equation is used to model the heat distribution in steady state.

The heat from auto muffler at 190°C was feds through aluminum base and adapter cubic plate with different thickness of 20mm, 30mm and 40mm, so that the gradient temperature only in the direction of the z axis (heat deep penetration). Heat leakage between adapter plates is terminated using isolated materials such as pertinaks filled in gaps.

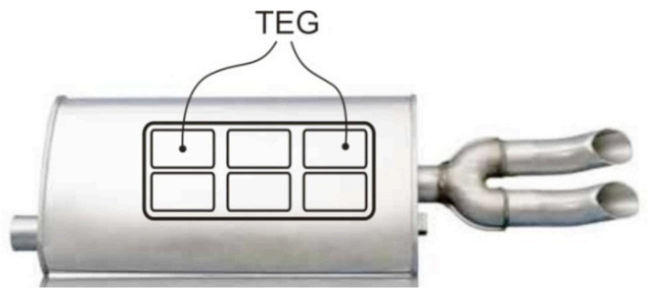


Figure 2. Array arrangement of TEG elements for simulating the muffler heat distribution

2.2. Modified diffusion equation

In this study, a thermoelectric generator consists of a base plate connected to the waste heat source from an auto muffler is modeled to simplify the simulation which is used to decrease the source temperature to 100 C°± 10% below the maximum operating temperature of TEG elements. This goes to the hot side of TEG surfaces. On the cold side, the heat will be removed in the form of convection and radiation, thus the temperature difference between both sides of the TEG (ΔT) becomes larger and the efficiency of TEG converted heat into electricity is higher. Figure 3 illustrates the schematic of the model and flow of heat distribution in three ways.

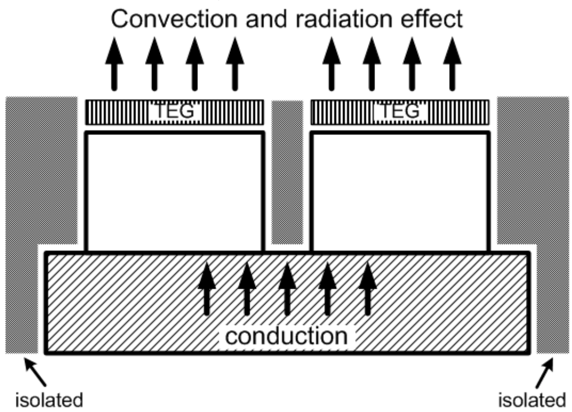


Figure 3. Components of heat transfer element TEG, conduction, convection and radiation

The TEG element as thermoelectric generator works using two different medium temperatures on its surfaces, as heat source and heat dissipation. Heat transfer is the exchange of thermal energy between physical systems in form of conduction, convection and radiation. This can be modeled using a heat conduction equation and in general is expressed as diffusion equation [8]:

$$k\nabla^2 T + q = \rho c \frac{\partial T}{\partial t} \tag{1}$$

Equation (1) is then modified by inserting a heat source q as wasted heat due to convection and radiation proces:

$$k\nabla^2 T - h(T - T_a) - \in \sigma\left(T^4 - T_a^4\right) = \rho c \frac{\partial T}{\partial t} \tag{2}$$

In steady state, the diffuse equation become Poisson equation by eliminating the time component:

$$k\nabla^2 T - h(T - T_a) - \in \sigma\left(T^4 - T_a^4\right) = 0 \tag{3}$$

Equation (3) is used to the calculate model at steady state condition, thus the function of time can be ignored. Using simulation, the heat energy reached to the hot side of the TEG device adjusted about 373K ± 10%, which is remaining under maximum operating temperature of the element, and can be tolerated, this does not cause permanent damage to the TEG devices.

There are three boundary conditions for the model in Figure 3. First condition is the temperature maintained at T = Ts for the base plate coupled to the heat source, which is stated by the Dirichlet boundary conditions. The temperature at Dirichlet boundary condition is applied choose to any heat source. Second condition is for insulated walls which is reject the heat propagation in the x-axis and y-axis and apply $\partial T/\partial n = 0$ the Neumann boundary condition, and last condition is on the cold side of TEG element. The heat is released in the appearance of convection and radiation using equation $-k\frac{\partial T}{\partial n} = h(T - T_a) + \in \sigma(T^4 - T_a^4)$ which is Neumann boundary conditions and Ts = 463K from sources temperature.

The previous TEG model discussed in Figure 3 is computationally solved using Finite Difference Method (FDM). To determine the temperature distribution of T (x, y, z) in a computational domain using the FDM, the Poisson equation (3) must be discrete towards the x, y and z plane and can be written in equation (4), (5) and (6).

$$\frac{\partial^2 T}{\partial x^2} = \frac{T(x+\Delta x, y, z) - 2T(x, y, z) + T(x-\Delta x, y, z)}{\Delta x^2} \tag{4}$$

$$\frac{\partial^2 T}{\partial y^2} = \frac{T(x, y+\Delta y, z) - 2T(x, y, z) + T(x, y-\Delta y, z)}{\Delta y^2} \tag{5}$$

$$\frac{\partial^2 T}{\partial z^2} = \frac{T(x, y, z+\Delta z) - 2T(x, y, z) + T(x, y, z-\Delta z)}{\Delta z^2} \tag{6}$$

Taking the value of Δx = Δy = Δz = h, then the steady state temperature distribution become discrete near the points of (i, j, k) as written in equation (7).

$$T_{i+1,j,k} + T_{i-1,j,k} + T_{i,j+1,k} + T_{i,j-1,k} + T_{i,j,k+1} + T_{i,j,k-1} - 6T_{i,j,k} = \left(\frac{h}{k}\right) q_{i,j,k} \tag{7}$$

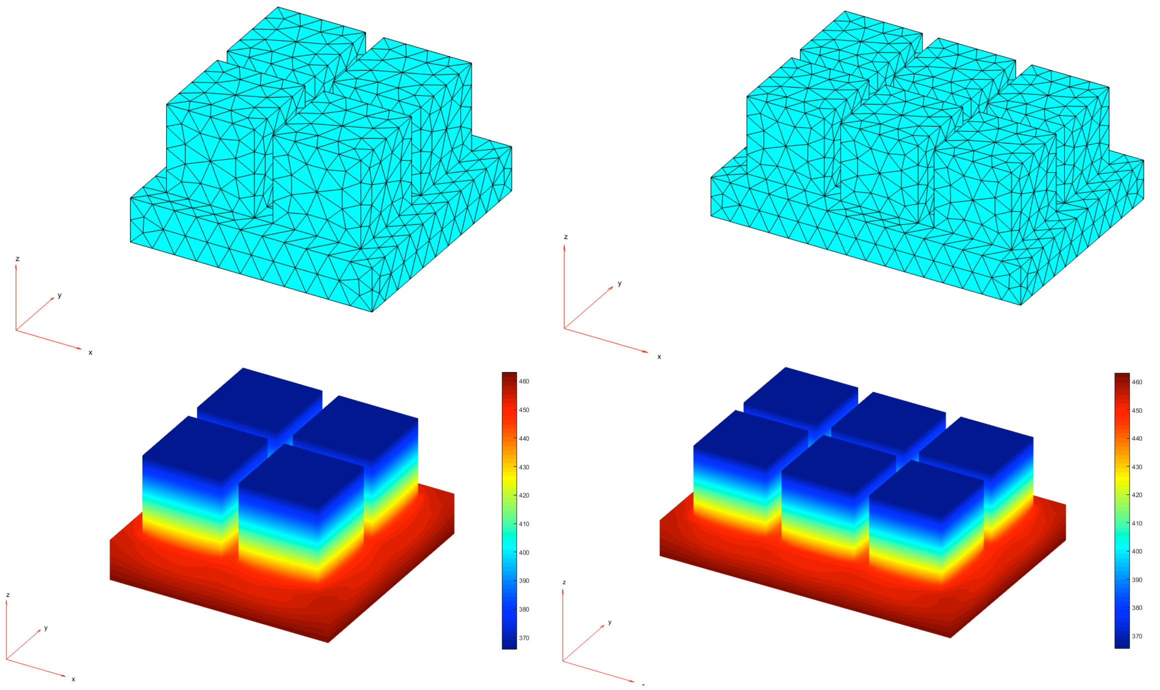
By inserting the all values of i, j and k, the matrix equation A.T = Q will be obtained.

After modeling heat distribution connected to hot side surface of Peltier element which is simulated using a diffusion equation, the distribution of color patterns will be resulted which describes the rate of heat transfer from the source and depict the penetration of heat transfer from source through an aluminum heat plate adapter with specific geometries. Optimization is obtained by changing the parameters of the equation (7) which are the boundary conditions and the geometries of heat source.

3. Results and Disscusion

The termoelectric plate model was designed using Autocad software which consisted of two parts, namely a base plate and an adapter plate [9]. The base plate has dimensions of 110mm length, 110mm width for 4 element array of TEG and 160mm width for 6 elements array, each having a thickness of 20mm. The adapter plate laid on the base plate formed a solid rectangular cube and connected to the hot side of the TEG Eureca with dimension of 40mm length and 40mm width. Waste heat from the auto muffler at 463K will spread to the base and the adapter plate and reached a final temperature around 373K ± 10%, lower than the maximum operating temperature of TEG. The slit between the adapter plates was isolated to cancel the heat transfer between them, so that final temperature could be reduced by adjustingthe thickness of the adapter plate.

A computer aided design program, ACAD was used to create the geometric design plate in the format of stereo lithography and then imported into MATLAB program for meshing process using a triangular pattern connected elements on the plate [10]. Figure 4a presented the meshing adapter plate for 4 arrays and 6 arrays and the results of the modeling. Figure 4b showed the heat distribution on the hot side of the TEG elements originating from the auto muffler waste heat. Each of the 4 and 6 elements illustrate a homogeneous blue color on the top of adapter cubic plate at the same temperature around 373K or 100°C.

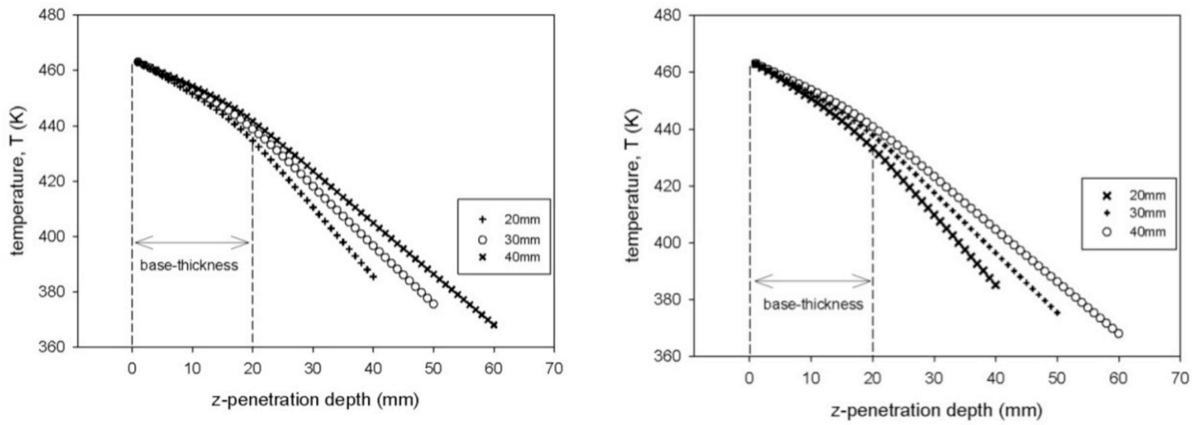


(a)

(b)

2 x 2 Array TEG 2 x 3 Array TEG

Figure 4. (a) Design and meshing adapter plate and (b) heat distribution simulation using MATLAB



(a) (b)

Figure 5. The reduction of temperature from heat sources at various plate thicknesses for (a) 2x2 array and (b) 2x3 array of TEG element. The thickness of the base plate was 20mm each.

As can seen from Figure 5 and Figure 6, the initial temperature for 2x2 arrays of TEG element was 463K and decreased about $\Delta T = 77{,}81$K which is proportional to 40.95% heat losses intended for a plate thickness of 20mm and 50.02% heat losses for 40mm. For 2x3 arrays, a minimum temperature was achieved for 40mm thickness as well as 49.98% heat losses. Temperature was decreasing gradually toward the z-axis, while on the direction of the x-axis and y-axis, the temperature was remains constant. Based on the computer simulation, the received heat temperature at the hot side of the TEG elements could be adjusted within operating range by means of plate thickness.

The cold side of TEG element would undergo two heat transfer process; convection and radiation, while the surface temperature kept constant at T_{cold} = 300K. From the described results, the temperature difference between the hot side and cold side, can be calculated and resulted $T_{hot} - T_{cold}$ = 68K. The greater temperature difference arised between the two sides of the elements; more electrical energy will be produced by the TEG element.

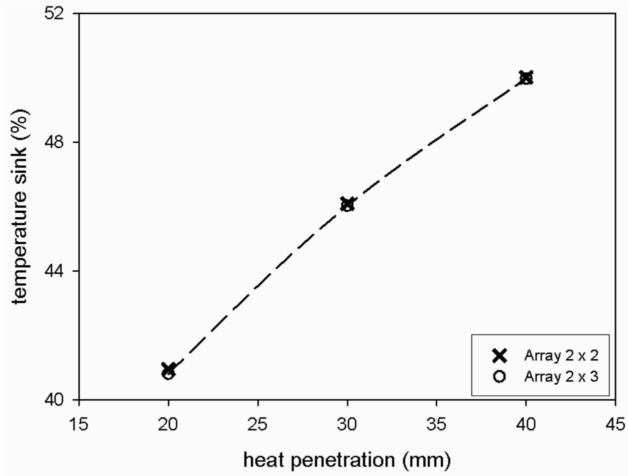


Figure 6. Temperature decreased in the z-axis of the initial temperature

Theoretically, there is no heat transfer between plate adapters since it has been isolated and the heat will only flow through the z-axis. Due to the thermal conductivity γ factor of the element, a small quantity of heat from hot side as amount of 1.56W goes through cold side, thus decreasing the temperature difference. Based on these results, the simulated temperature is complied with the TEG temperature by adjusting the adapter thickness. This can be used to achieve the efficiency and optimization of the electrical energy generated without deleterious effects on the device.

4. Conclusion

Heat distribution on an adapter plate for TEG element has been successfully simulated. Based on the results, the geometry and shape of adapter plate affects the temperature decrease in at the hot side of TEG element. Plate made of conductive material such as aluminum with high conductivity and high emissivity is able to reduce the muffler heat temperature with difference of 77,81K for 2x2 array and 94,97K for 2x3 array configuration. The temperature difference between two surfaces TEG, hot side and cold side, can be calculated and result as $T_{hot} - T_{cold} = 68K$. Greater the temperature difference arises between the two sides of the elements; more electrical energy will be produced by the TEG element. The simulated temperature is complied with the TEG temperature for optimum output and avoids damage from overheating.

Acknowledgement

This work was supported by Project of Hibah Bersaing 2016, DRPM Indonesia Ministry of Research, Technology and Higher Education.

References

[1] Date A, Date A, Dixon C and Akbarzadeh A 2014 *Renewable and Sustainable Energy Reviews* **33** 371
[2] Zhang X, Chau K T and Chan C C 2008 *Journal of Asian Electric Vehicle*, **6**(2) 1119
[3] Salerno D 2010 *Journal of Analog Innovation*, **20**(3)
[4] Chang C Y and Ma C C 2001 *International Journal of Heat and Mass Transfer* **44**(13) 2423
[5] Hu K Q and Chen Z T 2013 *International Journal of Heat and Mass Transfer* **62** 445
[6] Eureka 2010 EURECA Messtechnik GmbH, http://www.eureca.de
[7] Edwards M and Brinkfeldt K 2013 *Proc. 14th International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Microelectronics and Microsystems, EuroSimE 2013*
[8] Kontogeorgis G M and Folas G K 2010 *Thermodynamic Models for Industrial Applications: From Classical and Advanced Mixing Rules to Association Theories* (Wiltshire: Wiley) p710

[9] Autocad version 2015 http://www.autodesk.com
[10] Matlab version R2015a https://www.mathworks.com