Thermal analysis of a Thermoelectric Generator (TEG) using FEM technique

Prajwal K T¹,³ and Pritam Bhat²

¹Department of Electrical Engineering, M S Ramaiah University of Applied Sciences, Bangalore, India
²Department of Mechanical and Manufacturing Engineering, M S Ramaiah University of Applied Sciences, Bangalore, India
³E-mail: ktprajwal@gmail.com

Abstract. Finite Element Method (FEM) is one of the numerical technique based on minimization of energy, developed to solve complex engineering problems by converting governing differential equations of the physical phenomenon into algebraic equations that can be easily solved by a computer. Finite Element Analysis (FEA) is a computer application that uses FEM technique to analyze the physics of given problem based on the user inputs provided such as material data, geometry, and other pre requisites to study the behavior and structural strength in case of structural mechanics. A device working on principle of Seebeck-Thomson effect is Thermoelectric Generator (TEG) that converts the difference of heat energy into electrical energy. Thermal analysis of a TEG is a necessary step in design engineering as the electricity generated by the device is dependent on the difference in heat flux (temperature difference) between the source and sink (two ends) of TEG. In this paper, Finite Element Analysis of Thermoelectric Generator is carried out. The model is constructed, and thermal analysis is carried out in this paper.

1. Introduction

Amid growing environmental concerns regarding the clean power generation, TEG has set its footprint as a promising alternative technology to generate electricity from available thermal energy. TEG is a device that works using thermoelectric materials that have a property by virtue of which, heat flux (temperature difference) is converted into electrical energy due to Seebeck-Thomson effect [1]. Due to the advancement in the availability of thermoelectric (TE) materials, TEG finds wide applications in waste heat recovery units/plants as they can readily convert low grade heat into electricity. [2] The other advantages of TEG are, there are no mechanical parts thus free of wear, tear, noise and vibrations without leaving any carbon footprint on the environment. Some specific domain such as wearable read application in other energy fields. [3]

A thermoelectric material subjected to differential heat flux at their ends, generate electricity while connected in a circuit known as thermoelectric module. A thermoelectric effect is produced due to presence of two dissimilar thermoelectric materials with a p-type (positively charged) and an n-type (negatively charged) semiconductor joined at their ends. Whenever there is a temperature difference between the two materials, it so happens that charges (electric current) flow in the circuit. By placing a resistive load in the circuit, voltage is generated. Generally, magnitude of electric current generated is directly related to the applied differential heat flux (temperature difference). [4]

Thermoelectric modules are used in power generating plants to recover waste heat rejected into the atmosphere. Due to very high temperatures that exist in exhaust of chimneys, thermoelectric devices are subjected to enormous stress and strains arising due to thermal load on the system. Thus,
a FEA analysis of TEG can help in predicting the induced thermal stresses and performance leading
to optimized design suitable to the given application. The geometry of the TEG and the selection of
right thermoelectric materials has a great effect on its efficiency as it has to be designed to work
under harsh conditions that exist in power plants [3].

In this study a numerical model of a typical TEG is modelled in commercial FEA package
ANSYS APDL to study the thermal characteristics and the behavior of thermoelectric materials
under a temperature difference of around 3000 C. The thermal stresses and other parameters are
evaluated, and the obtained results can serve a purpose in design and development of an efficient
TEG specifically for waste heat recovery from an IC engine exhaust.

2. Methods and course of the study
A typical TEG is shown in Figure 1. It consists of heat source, heat sink, P and N- type junctions that
are connected to resistive load. T_h and T_c are the heat source and heat sink temperatures respectively.
As the heat load is absorbed at the hot end and resistive load R_l is applied, current I will flow through
the circuit due to Seebeck effect. The Thomson effect is also introduced due to joule heating at the
two ends [3].

![Figure 1. Schematic of the thermoelectric generator considered for FEA analysis](image)

The TEG can be modelled using two approaches namely numerical model and thermal resistance
model [2]. Numerical model can include the thermoelectric-fluid coupling effects, temperature
dependent material properties. Thermal resistance model accounts for temperature dependent
material properties as well as thermal contact resistance between different materials and can heavily
safe the computing time as required in numerical models.

In this work, we have adopted a numerical modelling technique with few assumptions to perform
the thermal analysis of TEG. The assumptions made are realistic and do not affect the solution of the
problem. Several energy balance equations are required to numerically model the TEG. Equations
used in FEM solver are detailed as follows [5-6].

The equation of heat flow \( q \) is
\[
\rho C \frac{\partial T}{\partial t} + \nabla \cdot q = \dot{q}
\] (1)

Where \( \rho \) is the density, \( C \) is specific heat, \( t \) is the time, \( \dot{q} \) is the heat generation rate per unit
volume.

Electric charge equation is given by
\[
\nabla \cdot \left( J + \frac{\partial B}{\partial t} \right) = 0
\] (2)

Where \( J \) is the electric current density, \( D \) is the electric flux density vector

Thermoelectric constitutive equations are as follows
\[
q = \pi \cdot J - \kappa \cdot \nabla T
\] (3)
\[
J = \sigma \cdot (E - S \cdot \nabla T)
\] (4)
And the dielectric medium constitutive equation is

\[ D = \varepsilon \cdot E \]  \hspace{1cm} (5)

Where \( \pi \) is Peltier coefficient, \( \varepsilon \) is the dielectric permittivity, \( \kappa \) is the thermal conductivity.

The scalar electric potential \( \phi \) is introduced to obtain the electric field for irrotational electric field and time constant magnetic field.

\[ \mathbf{E} = -\nabla \cdot \phi \]  \hspace{1cm} (6)

Combining all the above equations, thermoelectricity equations are obtained as follows.

\[ \rho C \frac{\partial T}{\partial t} + \nabla \cdot (\pi \cdot \mathbf{J}) - \nabla \cdot (\kappa \cdot \nabla T) = \dot{q} \]  \hspace{1cm} (7)

\[ \nabla \left( \varepsilon \cdot \nabla \frac{\partial \phi}{\partial t} \right) + \nabla \cdot (\sigma \cdot S \nabla T) + \nabla \cdot (\sigma \cdot \nabla \phi) = 0 \]  \hspace{1cm} (8)

The above coupled thermoelectric equations are solved in FEM solver to obtain the heat absorbed and the electric current generated by the TEG. The thermal and electrical effects due to Seebeck and Peltier along with Thomson effects (Joule heating at source/sink) are considered in these equations. The output power and TEG efficiency is respectively given by

\[ P = I^2 R_l \]  \hspace{1cm} (9)

\[ \eta_{TEG} = \frac{P}{Q_{in}} \]  \hspace{1cm} (10)

Following are the assumptions made in the FEM solution to save the computing time [7].

- Radiation and convection heat losses are neglected
- Internal thermal contact resistance between the materials are ignored

2.1. Designing of TEG using fem technique

I. Construction

There are 3 layers in the top and 2 layers in the bottom, between the P, N junction layers.

Layer 1: Material 1: The topmost layer is the aluminum layer.
Layer 2: Material 2: Aluminum Ceramic
Layer 3: Material 3: Copper
Layer 4: Material 4: N-Type
Layer 5: Material 5: P Type
Layer 6: Copper
Layer 7: Aluminum Ceramic

The steps followed in the ANSYS APDL is as follows:

1. Select the preferences as thermal and electric.
2. Preprocessor - choose element type - add/edit/delete - add solid 279, circuit124
3. Material props - material models - add thermal conductivity, and electrical resistivity for all 5 materials and Seebeck coefficient for material 4 and 5.
Table 1. Geometric and Material property values used in FEA Model

| Material and geometry | Definitions | Al | Al₂O₃ | Cu | N-type BiTe | P-type BiTe |
|-----------------------|-------------|----|-------|----|-------------|-------------|
| Electrical Resistivity (ohm m) |             | 0.25 | 1*10e18 | 0.18 | 9.8 | 13.8 |
| Thermal Conductivity (W/mk) |             | 200 | 37.2 | 398 | 1.78 | 1.22 |
| Seebeck Coefficient |             | -- | -- | -- | 16.8 | 16.8 |
| Thickness |             | 2.5 | 2.5 | 2.5 | 2.5 | 2.5 |
| Length |             | 10 | 1 | 0.5 | 20 | 20 |
| Width |             | 50 | 40 | 40 | 25 | 15 |

*All dimensions are in mm.

4. Model the TEG model as per the dimensions in Table 1[8].
Modelling: create - using areas the given model, extrude them to a depth of 2.5
Booleans - glue- volumes.
Thus, the modelling of the TEG is complete. The model of the TEG is obtained as shown in Figure 2.

![Figure 2](image1.png)

**Figure 2.** The geometric model of thermoelectric generator in ANSYS APDL

5. Meshing - mesh tool -
Set element attributes at volumes. Assign the material properties to different materials. Mesh the given model. Figure 3. Shows the tetrahedral elements used in meshing of TEG. The grid independence test was conducted in order to minimize error in results due to mesh size.

![Figure 3](image2.png)

**Figure 3.** Meshed model of thermoelectric generator in ANSYS
6. Solution - loads - define loads - apply - thermal - temperature - on areas
The analysis of the thermal distribution is carried out by assigning the temperature of 327°C to the top face of aluminum plate and 27°C to the other side of the cold ceramic plate made up of AL2O3.

7. Solution - solve - current ls - ok

8. General postproc - plot results - nodal solution - nodal temperature. The results are obtained.

2.2. Selection of proper element type and boundary conditions in ANSYS

SOLID 279, CIRCU124 are the two element types used in the TEG model simulation in ANSYS. SOLID279 an element type that is used in ANSYS for analysis purposes. It is quadratic in nature and can be used for thermal analysis purposes. The element is subjected to spatial orientation in nature and can be represented by many nodes. Each node represents a temperature nodal equation.

The element is anisotropic in nature. The input to these elements can be heat. The heat can be represented by the convection and the radiation techniques. The temperature load can be represented by nodal or surface loads.

The solution production related by the element are in binary methods:
- Nodal temperatures comprised in the complete nodal solution
- Use SOLID279 Covered Thermal Solid to model heat conduction in layered thick shells or solids.

CIRCU124:
This element is used for electrical analysis. Using this method, the electrical output from the system can be analyzed. Using this there is an option of getting upto 6 nodes in direction as the element contains 6 nodes in different direction.

The three degrees of freedom that can be associated with the element is:
- VOLT (voltage)
- CURR (current)
- EMF (potential drop)

3. Research Results
The result analysis is carried out using the ANSYS APDL solution module. The TEG module is evaluated for the temperature distribution of the hot and cold side of the model. The solution can be obtained by first meshing the module. The model can be meshed using many types of meshes. In this triangular meshes is used.

Figure 3 illustrates the meshed model of the TEG. The meshing of the model is carried out to divide the model into several smaller pieces of finite elements to solve the equations and obtain the solution.
The meshing can be done in many methods. Simple and efficient method of meshing is by using fine mesh.

Figure 4 shows temperature distribution of the generator. From the figure it can be seen that the hot side of the generator is experiencing highest temperature while cold side of the generator settles to lowest temperature. The hot side temperature ranges up to 327°C and cold side of the temperature is about 27°C.

This approves by the temperature loads allocated previously while modelling.

4. Conclusions
Using FEM, we have modeled thermoelectric generator and studied the thermal analysis of the same. FEM technique has proved to be the best way to analyze engineering structures. The FEA technique can be effectively leveraged to predict the performance of thermoelectric generators that helps to design and improve the deployment and potential of TEG in real life applications.

References
[1] Van Herwaarden, A. W., and P. M. Sarro. "Thermal sensors based on the Seebeck effect." Sensors and Actuators 10.3-4 (1986): pp. 321-346.
[2] Luo, Ding, Ruochen Wang, and Wei Yu. "Comparison and parametric study of two theoretical modeling approaches based on an air-to-water thermoelectric generator system." Journal of Power Sources 439 (2019): 227069.
[3] Shittu, Samson, et al. "Advancements in thermoelectric generators for enhanced hybrid photovoltaic system performance." Renewable and Sustainable Energy Reviews 109 (2019): pp. 24-54.
[4] Ramadass, Yogesh K., and Anantha P. Chandrakasan. "A battery-less thermoelectric energy harvesting interface circuit with 35 mV startup voltage." IEEE Journal of Solid-State Circuits 46.1 (2010): 333-341.
[5] Antonova, Elena E., and David C. Looman. "Finite elements for thermoelectric device analysis in ANSYS." ICT 2005. 24th International Conference on Thermoelectrics, 2005. IEEE, 2005.
[6] Zhang, Mengjun, et al. "Influence of Thomson effect on the thermoelectric generator." International Journal of Heat and Mass Transfer 137 (2019): pp. 1183-1190.
[7] Fan, Shifa, and Yuanwen Gao. "Numerical analysis on the segmented annular thermoelectric generator for waste heat recovery." Energy 183 (2019): pp. 35-47.
[8] Zulkifli, Muhammad Nubli, et al. "Finite element analysis on the thermoelectric generator for the waste heat recovery of solar application." AIP Conference Proceedings. Vol. 1877. No. 1. AIP Publishing LLC, 2017.