1. Introduction

Piezoelectric materials have the capability to produce a voltage when deformed, and they deform when an electric voltage is applied. These are characterized as direct and indirect effects of piezoelectric materials respectively. The direct piezoelectric effect is utilized in energy harvesting mechanism since the system absorbs the vibration energy from the host structure and converts that to electrical energy.

As shown in Fig. 1, the piezoelectric energy harvester studied in this project consists of a vibrating host structure, a cantilever beam with an attached piezoelectric layer mounted to the host structure, and an energy harvesting circuit connected to the piezoelectric layer. When the host structure is vibrating, the vibration energy is absorbed by the piezoelectric material which deforms and induces a voltage output across their electrodes. This voltage can be stored in a battery or some storage device for potential use.

Deriving a dynamic model is crucial in order to design a piezoelectric harvester with optimized performance. In this preliminary study, we focused on developing a simulation model of piezoelectric harvester using the ANSYS workbench simulation software. In the next step of this project, we are going to extend the modelling of the circuit using ANSYS. Then the models of the cantilever beam and the circuit will be integrated together for the modelling of the whole energy harvesting system so that simulation can be performed for optimal design.

2. ANSYS Simulation Basis

The simulation started with gathering the engineering data for the piezo material. PZT-5A was chosen as the piezo material due to its high resistivity at elevated temperature, high sensitivity and high time stability. With high piezoelectric constants, the PZT5A material is able to generate higher voltages. Furthermore, the higher free dielectric constant of PZT5A also enable it to obtain a higher electrical energy stored under the same mechanical energy input. Then the geometry was determined for simulation purpose. In the setup section of the software, where a force was applied to the model, the harmonic force was applied which was around 2N. As the force is harmonic, it follows the
sinusoidal pattern to represent the real vibration in the system. Since ANSYS does not have fully developed simulation package for piezoelectric analysis, we need to rely on the third-party extension which helps to add anisotropic material properties of the material. The permittivity matrix values can also be entered using ACT extension, which is called Piezo and MEMS extension.

The simulation was completed by performing three analysis and combining them to obtain the output voltage from the piezo body. First Eqn. (1) was used to relate mechanical strain with electrical charge

\[
\begin{bmatrix}
K_{uu} & K_{uv} \\
K_{uv}^T & -K_{vv}
\end{bmatrix}
\begin{bmatrix}
U \\
V
\end{bmatrix}
+ \begin{bmatrix}
C_{uu} & 0 \\
0 & -C_{vv}
\end{bmatrix}
\begin{bmatrix}
U \\
V
\end{bmatrix}
+ \begin{bmatrix}
M_{uu} & 0 \\
0 & 0
\end{bmatrix}
\begin{bmatrix}
U \\
V
\end{bmatrix}
= \begin{bmatrix}
F \\
Q
\end{bmatrix}
\]  

(1)

with

Element matrices:

- \(K_{uu}\) - piezoelectric coupling
- \(F\) - force
- \(C_{uu}\) - structural damping
- \(V\) - negative electric charge
- \(K_{vv}\) - dielectric dissipation
- \(M_{uu}\) - mass
- \(Q\) - dielectric permittivity

Eqn. (2) was used in ANSYS solver,

\[
(T) = [c][S] - [e][E]
\]

(2)

with

- \(T\) - stress vector
- \(S\) - elastic strain vector
- \(E\) - electric field intensity
- \(c\) - dielectric permittivity
- \(e\) - piezoelectric matrix
- \(D\) - electric flux density

In Eqn. (2), \([e]\) is the important matrix which couples the mechanical and electrical field properties, \([T]\) is the stress vector describing the strain built up in the piezo patch. This strain will induce electric flux density in the piezo material which develops an alternating voltage as the stress tensor varies. \([c]\) is a symmetric matrix representing the anisotropic elasticity of the piezo, and \([E]\) is the dielectric permittivity matrix.

3. **ANSYS Simulation Procedures**

The following flowchart shows the steps followed for performing the simulation.

1. Pre-Processing
   a. Background reading for Piezoelectric Material
   b. Geometry Creation
   c. Getting ACT Extensions Installed in ANSYS

2. ANSYS Simulation process
a. Pre-processing – Modal Analysis
   i. Boundary condition
   ii. Natural frequency of the system obtained
b. Pre-processing – Harmonic response analysis
   i. Boundary condition
   ii. Solution
c. Pre-processing – Static Structural Analysis
   i. Defining various constrains on the process
   ii. Boundary conditions
   iii. Solution
d. Synchronizing model Static Structural and Harmonic response analysis

3.1. Pre-processing

The pre-processing step involves setting up of various piezo material properties to enter in the ANSYS. The properties are available in a standard format which is not compatible with the ANSYS format. Conversion of the properties was required so various matrix operations were performed to obtain the properties in required ANSYS format. This step also involves the model generation in ANSYS.

a. Piezoelectric Material Properties conversion in standard Ansys format

The piezo material properties are available in a standard format which is governed by ANSI/IEEE Std 176-1987. Since ANSYS does not take that format directly, it has to be converted into a format which is acceptable. Eqn. (3) illustrates the stiffness matrix conversion from IEEE to ANSYS compatible format, where \( c_{ij} \) is the elastic stiffness constant in \( N/m^2 \).

\[
\begin{bmatrix}
  c_{11} & c_{21} & c_{31} & c_{41} & c_{51} & c_{61} \\
  c_{21} & c_{22} & c_{23} & c_{24} & c_{25} & c_{26} \\
  c_{31} & c_{32} & c_{33} & c_{34} & c_{35} & c_{36} \\
  c_{41} & c_{42} & c_{43} & c_{44} & c_{45} & c_{46} \\
  c_{51} & c_{52} & c_{53} & c_{54} & c_{55} & c_{56} \\
  c_{61} & c_{62} & c_{63} & c_{64} & c_{65} & c_{66}
\end{bmatrix} \rightarrow 
\begin{bmatrix}
  c_{11} & c_{21} & c_{31} & c_{41} & c_{51} & c_{61} \\
  c_{21} & c_{22} & c_{23} & c_{24} & c_{25} & c_{26} \\
  c_{31} & c_{32} & c_{33} & c_{34} & c_{35} & c_{36} \\
  c_{41} & c_{42} & c_{43} & c_{44} & c_{45} & c_{46} \\
  c_{51} & c_{52} & c_{53} & c_{54} & c_{55} & c_{56} \\
  c_{61} & c_{62} & c_{63} & c_{64} & c_{65} & c_{66}
\end{bmatrix}
\]

Eqn. (4) illustrates the electric flux density matrix conversion from IEEE to ANSYS compatible format, where \( d_{ij} \) is the piezoelectric constant and its unit is \( C/N \).

\[
\begin{bmatrix}
d_{11} & d_{21} & d_{31} \\
d_{21} & d_{22} & d_{23} \\
d_{31} & d_{32} & d_{33}
\end{bmatrix} \rightarrow 
\begin{bmatrix}
d_{11} & d_{21} & d_{31} \\
d_{21} & d_{22} & d_{23} \\
d_{31} & d_{32} & d_{33}
\end{bmatrix}
\]

In both above equations, the interchange of \( i \) and \( j \) indicates that the axis of force and the direction of charge generation.
Table 1 shows the anisotropic elasticity properties entered in the engineering material tab. As the elasticity matrix is symmetric diagonally, we only need to enter the values of material property for only half of the matrix.

Table 1. Anisotropic Elasticity

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(D[^{1,1}]) (Pa)</td>
<td>(D[^{1,2}]) (Pa)</td>
<td>(D[^{1,3}]) (Pa)</td>
<td>(D[^{1,4}]) (Pa)</td>
<td>(D[^{1,5}]) (Pa)</td>
<td>(D[^{1,6}]) (Pa)</td>
</tr>
<tr>
<td>2</td>
<td>1.32E+11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>7.3E+10</td>
<td>1.15E+11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>7.1E+10</td>
<td>7.3E+10</td>
<td>1.32E+11</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>2.6E+10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0</td>
<td>0</td>
<td>2.6E+10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0</td>
<td>2.6E+10</td>
<td></td>
<td></td>
<td></td>
<td>3E+10</td>
</tr>
</tbody>
</table>

b. Geometry Creation

The design of geometry was created in the Design Modler application. Fig. 2 shows the piezo patch and the beam geometry used for simulation. The upper part is the piezo patch placed firmly on the top of the beam. It will induce strain in the piezo patch as the beam element vibrates. This vibration will be oscillatory which will induce an alternating voltage in the piezo body.

![Figure 2 Geometry for Simulation](image)

Figure 2 Geometry for Simulation

c. ACT Extension

As ANSYS workbench cannot perform the simulation for the piezoelectric material itself we need to install another extension as shown in Fig. 3 which can help to add a material property in matrix form for the piezo material. ACT Piezo & MEMS extension is a customization made with the ACT to integrate ANSYS piezoelectric & MEMS capabilities in Mechanical. The extension consists of one XML file (Configures the UI content) and one python script (Implements the extension functionality). The PiezoAndMEMS extension adds lots of features into the mechanical application, which increases the capability of mechanical to perform simulation which involves multi-physics problems. Also, it adds result posting ability into the application which helps a lot
to view the solution in many different ways. It replaces command snippets, which are required to enter all the properties of the material, into interactive objects. It also creates customized loads and boundary condition to apply on various model objects.

![ACT Extension with Added Features](image)

**Figure 3** ACT Extension with Added Features

### 3.2. ANSYS Simulation process

The process of simulation involves synchronizing three analysis: 1) Model Analysis, 2) Harmonic Analysis, and 3) Structural Analysis. In this energy harvester device, power generation is mainly dependent on system natural frequency and operating frequency. When these two frequencies are equal, then resonance occurs. Large amount of power is generated at resonance. Therefore it is necessary to analyze the mode of piezoelectric cantilever beam. Harmonic analysis is necessary to determine the steady-state response of a linear structure to load that vary sinusoidally. Finally, structural analysis is conducted to calculate the actual voltage. Following sections describe all the analysis in detail.

![Support Condition to Zero in All Directions](image)

**Figure 4.** Support Condition to Zero in All Directions

a. **Modal Analysis**
The modal analysis was used to obtain the natural frequencies of the combined system, i.e., the piezo patch and beam element. The applied boundary condition at the end of the beam is given in Fig. 4. This boundary condition ensures that there will be no displacement in the x, y, and z-
direction, but it can rotate about the x-axis. That is how the vibration is shaking the beam and the piezo element. Fig. 5 shows that the natural frequencies at the first three modes obtained for the system were 35.78 Hz, 339 Hz and 413 Hz respectively.

![Figure 5. First, Second and Third Mode Shapes](image)

b. **Harmonic Analysis**

The harmonic analysis examines system performance when a sinusoidal load is applied to the structure. The beam is subjected to a vibration input, it in turn will induce strain in the piezoelectric layer which will lead to voltage generation. Fig. 6 shows the boundary condition, a sinusoidal force, applied in the harmonic analysis.

c. **Structural Analysis**

The structural analysis helps to get output voltage results from the piezo patch. Table 2 shows the
piezoelectric material properties in matrix format. The term PIEZ e31 represent the value of the permittivity matrix when the polarization axis is y.

\[
\begin{bmatrix}
K_{33} & 0 & 0 \\
K_{11} & 0 & 0 \\
K_{11} & 0 & 0
\end{bmatrix} \rightarrow
\begin{bmatrix}
K_{11} & 0 & 0 \\
K_{33} & 0 & 0 \\
K_{11} & 0 & 0
\end{bmatrix} \rightarrow
\begin{bmatrix}
K_{11} & 0 & 0 \\
K_{11} & 0 & 0 \\
K_{33} & 0 & 0
\end{bmatrix}
\]

(5)

Eqn. (5) shows permittivity constants for different polarization axes. Eqn. (6) shows the piezoelectric constants for different polarization axes, and Eqn. (7) shows the matrix conversion.

Figure 6. Sinusoidal Force on The Edge

Table 2. Piezoelectric Body Element Matrix
d. **Synchronization of Three Analyses**

The synchronization involves working with three analyses together. The model analysis generates output in form of a number of modes of natural frequencies for a physical model. These natural frequency values were used as an input frequency for the physical model in harmonic analysis. The harmonic analysis applies sinusoidal force on the edge of the beam, which in turn induce strain in the piezo patch and that will lead to an alternating voltage generation. For calculating generated voltage in the piezo patch, the structural analysis was performed. The structural analysis helps in visualizing the voltage built up in the piezo patch as the beam vibrates. Fig. 7 shows the synchronized project schematic page. The blue lines show the connection between various parts of the analysis.

4. **Simulation Result**

Figs 8 to 10 show the voltage generated in the piezo body. An alternating voltage developed using real physical model was around in the range of 1.8 – 2.6V and the result obtained through ANSYS simulation shows the result is within the range which is 2.05V, which resembles the voltage occurred in the physical model. Fig 8 shows the voltage distribution in the piezo patch after the initial phase and Fig 9 is the voltage in the second phase. Fig. 10 shows the final result.

5. **Conclusion**

The mathematical modeling of the energy harvester involves numbers of parameter and variables, which are governed by constitution equations, requires perfect manipulation to get a general
equation which defines the full model. In other words, the mathematical work to combine the stress and strain equation with the electric flux density equation becomes cumbersome as the equation develops. So, it becomes very important to do programming for mathematical calculation work as with the increase in the length of the equation the factor of human error also increases.

To avoid such a time-consuming work, one can take advantage of software specially designed and programmed to do calculation task related to combined field equations. One of the well know software is ANSYS, which is used for solving coupled/combined equation of various physics domain can be employed for solving the constitutive equation of piezoelectric material.

The project describes how the ANSYS software uses the coupled equations for getting a solution. It also describes the full process for simulation of an energy harvester device which in this case was a piezo patch and beam element.

Figure 7. Synchronized Analysis Project Schematic
Figure 8. Voltage Distribution in The First Phase

Figure 9. Voltage Distribution in The Second Phase

Figure 10. Final Voltage Distribution in Piezo Patch

References

