GENERAL

ansys thermal electric analysis tutorial

Published: September 19, 2026 | Index ID: #-

ANSYS Thermal Electric Analysis Tutorial: A Complete Guide to Simulating Thermoelectric Devices

Thermoelectric technology has emerged as a promising solution for waste heat recovery, solid state cooling, and power generation in remote or harsh environments. Whether you are an engineer designing a new thermoelectric generator (TEG), a researcher studying the Seebeck effect, or a student learning about coupled thermal electric phenomena, mastering simulation tools is essential. ANSYS Workbench offers a powerful, user friendly environment for performing coupled thermal electric analyses, enabling you to predict device performance, optimize material selection, and reduce development time.

This tutorial walks you through every step of an ANSYS thermal electric analysis, from model setup to post processing. By the end, you will have a solid understanding of the workflow, the physics involved, and how to leverage ANSYS to accelerate your thermoelectric projects.

Table of Contents

1. Overview of ANSYS Thermal Electric Analysis

ANSYS Workbench integrates several physics modules—such as **Thermal**, **Electrical**, and **Coupled Fields**—to create a seamless environment for thermoelectric analysis. The core of the simulation relies on the finite element method (FEM), which discretizes the device geometry into small elements, solving the governing equations for temperature, electric potential, and heat flux simultaneously.

Thermoelectric analysis in ANSYS typically involves:

- Defining the geometry of the thermoelectric module (legs, junctions, contacts).
- Assigning temperature dependent material properties, including electrical conductivity, thermal conductivity, and the Seebeck coefficient.
- Applying boundary conditions that mimic real operating environments (heat flux, temperature, voltage).

- Coupling the thermal and electric domains through the Seebeck effect and Peltier heat.
- Running the solver to obtain temperature, voltage, current, and power output.
- Visualizing results to identify hotspots, voltage drops, and performance metrics.

2. Why Use ANSYS for Thermoelectric Simulations?

ANSYS is widely respected for its accuracy, robustness, and extensive material database. Here are some reasons why it is the go to tool for thermoelectric analysis:

- Coupled Physics Engine: Simultaneously solves heat transfer, electromagnetics, and mechanical stresses.
- Material Library: Access to temperature dependent properties for common thermoelectric materials such as Bi ,Te ƒ, PbTe, and skutterudites.
- Advanced Meshing: Adaptive mesh refinement ensures accurate capture of steep temperature gradients.
- Post Processing Capabilities: Built in tools for extracting efficiency, figure of merit (ZT), and power density.
- Automation: Workbench allows scripting via Python or COM, enabling batch processing of multiple design variations.

3. Key Concepts in Thermoelectric Modeling

Before diving into the workflow, it is essential to understand the fundamental equations governing thermoelectric phenomena.

3.1 The Seebeck Effect

When a temperature gradient exists across a thermoelectric material, an electric potential is generated. The Seebeck coefficient (S) quantifies this effect:

V = -S · 9E@

In simulations, the Seebeck coefficient is typically temperature dependent and must be provided as a material property.

3.2 The Peltier Effect

Conversely, when an electric current flows through a junction of two different thermoelectric materials, heat is absorbed or released. The Peltier coefficient (: ' —2 &VÆ FVB Fò F†R 6VV&V0k coefficient via the Kelvin relation: : Ò 2 r T.

3.3 Figure of Merit (ZT)

ZT is a dimensionless measure of thermoelectric performance:

ZT = (S² · <2 r T) / ;

where <0 is electrical conductivity, ; is thermal conductivity, and T is absolute temperature. High ZT values ("e1) indicate efficient thermoelectric materials.

3.4 Coupled Governing Equations

ANSYS solves the following coupled equations:

- Heat Equation: -" ·(k" T) + Q = 0, where Q includes Joule heating and Peltier heat.
- Electrical Current Equation: " ·(<2 ub' ² {rƒÃS" T) = 0.

These equations are solved simultaneously to capture the interdependence of temperature and electric fields.

4. Setting Up the Model

We’ll walk through a typical workflow using ANSYS Workbench. The example involves a simple two leg thermoelectric module.

4.1 Create a New Project

1. Open ANSYS Workbench.
2. Drag the Thermal Electric system from the toolbox into the Project Schematic.
3. Double click the Geometry cell to launch DesignModeler or SpaceClaim.

4.2 Build the Geometry

- Define the dimensions of the n type and p type legs (e.g., 5 /mm /× /5 /mm /× /20 /mm).
- Create a central junction where the two legs meet.
- Optionally, add contact pads for electrical connections.

4.3 Define the Study

1. Double click the Model cell.
2. Under Study Type, choose Steady State for a static analysis.
3. Set the solver type to Explicit or Implicit based on convergence requirements.

5. Material Properties and Seebeck Coefficient

Accurate material data is crucial. ANSYS allows you to import temperature dependent properties.

5.1 Assigning Materials

1. In the Model window, right click Materials and choose New Material.
2. Name the material (e.g., Bi2Te3_n).
3. Set properties:
4. Electrical Conductivity (<2“ e.g., 2 /× /10 u /S/m.
5. Thermal Conductivity (k): e.g., 1.5 /W/m·K.
6. Seebeck Coefficient (S): 200 /µV/K, temperature dependent.
7. Define the temperature range (e.g., 300 /K /– /500 /K).

5.2 Seebeck Coefficient Input

In the **Material** editor, under **Transport Properties**, specify the Seebeck coefficient as a function of temperature. ANSYS accepts **table** or **equation** inputs. For a linear approximation:

S(T) = S € + a·(T - T €)

where *S* i€s the baseline coefficient at reference temperature *T*,€ and *a* is the slope.

6. Meshing Strategies

Thermoelectric devices often exhibit steep temperature gradients near junctions. A fine mesh in these regions ensures accuracy.

6.1 Mesh Settings

1. Right click Mesh and select Mesh Settings.
2. Choose Hexahedral or Hybrid elements.
3. Set Element Size to Fine near the junction; use Coarse elsewhere.
4. Activate Mesh Refinement or Size Function based on temperature gradient.

6.2 Mesh Quality Checks

After generating the mesh, review the **Mesh Quality** report. Aim for element skewness

7. Physics Setup: Coupling Thermal and Electric Fields

ANSYS allows you to define coupled physics in a single analysis branch.

7.1 Thermal Boundary Conditions

- Apply a Heat Flux on the hot side (e.g., 100 /W/m²).
- Set the cold side to a fixed temperature (e.g., 300 /K).
- Add convection or radiation if necessary.

7.2 Electrical Boundary Conditions

- Assign a Voltage on one side (e.g., 0 /V).
- Set the other side to a Ground or a specified voltage to simulate load.
- Define a Current Density if modeling current injection.

7.3 Coupling Parameters

In the **Physics Setup** tree, enable the **Seebeck Coupling**option. Ensure that the solver recognizes the Seebeck coefficient and Peltier heat contributions. ANSYS automatically adds the Peltier term (: t¢’ Fò F†R †V B W V F–öâà

8. Running the Solver

With all settings in place, it’s time to solve.

8.1 Solver Settings

Baca Juga (Artikel Terkait)

- [applied mechanics for engineering technology keith m walker](http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-keith-m-walker.pdf)
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-keith-m-walker.pdf
- [applied mechanics for engineering technology answers](http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-answers.pdf)
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-answers.pdf
- [applied mechanics for engineering technology 8th edition solution manual](http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-8th-edition-solution-manual.pdf)
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-8th-edition-solution-manual.pdf
- [applied mechanics for engineering technology 8th edition](http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-8th-edition.pdf)
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-8th-edition.pdf

Tags: #ansys #thermal #electric #analysis #tutorial