GENERAL

ansys transient thermal analysis tutorial

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

ANSYS Transient Thermal Analysis Tutorial: A Complete Guide

When engineers design anything that generates or dissipates heat—whether it’s an electric motor, a microprocessor, or a structural component exposed to high temperatures—understanding how temperature changes over time is crucial. **ANSYS**offers a powerful suite of tools for simulating these time dependent thermal phenomena, allowing designers to predict performance, optimize cooling strategies, and avoid catastrophic failures.

This tutorial walks you through every step of setting up, solving, and interpreting a transient thermal analysis in ANSYS. Whether you’re a seasoned CFD user or a newcomer to finite element analysis, the guide is structured to provide clarity and depth without overwhelming you with jargon.

What Is Transient Thermal Analysis?

Transient thermal analysis, also known as time dependent heat transfer analysis, studies how temperature fields evolve over a specified period. Unlike steady state analysis—which assumes temperatures have reached equilibrium—transient studies capture the dynamic response of a system to changing heat sources, boundary conditions, or material properties.

Common applications include:

- Evaluating the thermal response of a laptop during heavy usage.
- Simulating the cooling curve of an engine block after a shutdown.
- Predicting temperature spikes in a spacecraft during re entry.
- Assessing the heat distribution in a battery pack during charging cycles.

Why Use ANSYS for Transient Thermal Simulations?

ANSYS provides a comprehensive, industry standard platform that integrates geometry creation, meshing, physics definition, solving, and post processing in a single environment. Key advantages include:

- Robust solvers: Advanced algorithms handle stiff equations and large time steps efficiently.
- Material libraries: Built in temperature dependent properties reduce modeling time.
- Coupled multiphysics: Seamless interaction with fluid, structural, and electromagnetic solvers.
- Extensive documentation: Step by step guides and tutorials aid learning.

Prerequisites for This Tutorial

Before diving into the workflow, make sure you have:

1. ANSYS Workbench installed (any recent version will work).
2. A basic understanding of heat transfer fundamentals.
3. Familiarity with the ANSYS Workbench interface.
4. Sample geometry or a CAD file ready for import.

If you’re new to ANSYS, consider reviewing the *ANSYS Basics*section in the help files to get comfortable with the workspace.

Step 1: Setting Up the Project

Open ANSYS Workbench and create a new project. Drag the **Transient Thermal** system from the toolbox onto the Project Schematic. The system will automatically create three components:

- Geometry – where you import or create your CAD model.
- Model – the physics setup and solver configuration.
- Results – post processing and visualization.

Double click the **Geometry** cell to launch the DesignModeler or SpaceClaim environment, depending on your preference.

Importing Geometry

For this tutorial, we'll use a simple rectangular heat sink geometry. Import the CAD file (e.g., *heat_sink.cae*) by selecting **File !' Import !' CAD File**. Once loaded, verify the model for any gaps or inconsistencies that could affect meshing.

Defining the Simulation Domain

After confirming the geometry, close the editor and return to the **Model** component. Click on the **Model** cell to open the Mechanical workspace.

Step 2: Material Assignment

Right click **Materials** in the tree view and choose **New Material**. Give it a descriptive name (e.g., "Aluminum 6061"). Populate the following properties:

- Density (< ' b #s   ¶röÛ2à
- Specific Heat Capacity (Cp) – 900 J/(kg·K).
- Thermal Conductivity (k) – 167 W/(m·K).
- Temperature Dependent Properties: If your model requires temperature dependent data, click Add Property and input a function or table.

Assign this material to the entire geometry by dragging it onto the model or using the **Apply to All** option.

Step 3: Defining Boundary Conditions

Transient analysis relies heavily on accurate boundary definitions. The following are typical boundary condition types you might encounter:

- Heat Flux (q) – applies a specified rate of heat transfer per unit area.
- Temperature (T) – fixes the absolute temperature on a surface.
- Convection (h) – models heat transfer to a surrounding fluid.
- Radiation (;RÂ T") – accounts for radiative heat exchange.

Applying a Time Dependent Heat Flux

Right click the geometry surface that receives the heat source and choose **Heat Flux**. In the details pane:

1. Set Heat Flux Value to 500 W/m² initially.
2. Click the Expression button next to the value field.
3. Enter a time dependent function, e.g., 500*exp(-t/5), to simulate a decaying heat source over 5 seconds.

Repeat the process for any other surfaces, such as the base exposed to a coolant. For convection, use the

Convection boundary condition and set the convection coefficient (h) and ambient temperature (T").

Setting Initial Conditions

Right click the model and selec**Initial Conditions !’ Temperature**Set the initial temperature to, say, *25°C*(298 K). This establishes the starting point for the transient solution.

Step 4: Mesh Generation

Mesh quality directly impacts solution accuracy and computational cost. For transient thermal problems, a fine mesh near heat source surfaces is essential.

Choosing Element Types

Use **SOLID186** or **SOLID187** elements for 3 D solids. If your geometry is thin, conside**PLANE186**elements for 2 D approximations.

Refining the Mesh

Apply a **Size Function** to the heat sink fins:

1. Select the fin surfaces.
2. Right click and choose Size !’ Size Function !’ Mesh Size.
3. Set Maximum Mesh Size to 0.5 mm for fine resolution.
4. Apply a Mesh !’ Generate Mesh command.

For the base, a coarser mesh is acceptable; set the maximum mesh size to *2 mm*.

Step 5: Solver Settings

Click on the **Transient Thermal** system in the tree view. Under **Solution**, configure the following:

- Time Step: Choose a fixed time step of 0.1 s for stability, or use an adaptive time step for efficiency.
- Time Range: Set the total simulation time to 10 s.
- Non Linear Solver: Enable Newton if temperature dependent material properties are used.
- Under Convergence Criteria, set residual tolerance to 1e 6.

For large models, enable **Parallel Solver** to distribute the workload across multiple cores.

Step 6: Running the Simulation

Click the **Run**button. The solver will progress through each time step, updating temperature fields according to the defined boundary conditions and material properties.

Monitor the solver log for any convergence issues. If residuals stagnate, consider reducing the time step or refining the mesh.

Step 7: Post Processing the Results

Once the solution completes, navigate to the **Results**component. Use the following tools to analyze and visualize the data:

Temperature Contour Plots

Select a plane cut through the heat sink and apply a **Temperature**contour. Adjust the color map to highlight hotspots.

Time Dependent Temperature Curves

Right click the geometry surface and choose **Probe !' Temperature** In the details pane, set the probe location and generate a plot of temperature versus time. This is invaluable for verifying transient behavior.

Exporting Data

Right click the plot and select **Export !' CSV** to save the data for external analysis or reporting. For high quality images, use **Export !' Image** and choose PNG or JPEG format.

Best Practices for Accurate Transient Thermal Analysis

- Validate Mesh Independence: Run a mesh convergence study by refining the mesh until temperature differences fall below a chosen tolerance.
- Check Time Step Adequacy: Ensure that the time step captures rapid temperature changes; too large a step can smooth out peaks.
- Use Symmetry: If the geometry is symmetric, exploit symmetry planes to reduce model size.
- Include All Relevant Heat Transfer Modes: Conduction, convection, and radiation should all be considered if they significantly influence the system.
- Monitor Residuals: Persistent residuals indicate solver instability; adjust solver settings or refine the mesh.

Case Study: Cooling a High Power LED Array

To illustrate the workflow, let's examine a real world scenario: an LED array operating at 200 W. The goal is to design a heat sink that keeps the LED junction temperature below 80 /°C during operation.

1. Geometry: A rectangular heat sink with fins, dimensions 100 /mm × 50 /mm × 20 /mm.
2. Material: Aluminum 6061.
3. Boundary Conditions:

   Heat flux of 200 /W applied uniformly across the LED surface.
4. Convective cooling on the fin surfaces with h /= /50 /W/m²·K and ambient temperature 25 /°C.
5. Radiative losses from all surfaces with emissivity 0.8.

By analyzing the temperature profile, we can confirm that the heat sink design meets the thermal requirement. If not, we might increase fin density or add a liquid cooling channel.

Common Pitfalls and How to Avoid Them

- Ignoring Material Non Linearity: Temperature dependent properties can significantly alter results; always verify that the material database includes these dependencies.
- Using Inappropriate Boundary Conditions: A fixed temperature boundary can mask the true transient behavior; use convection or radiation where realistic.

Baca Juga (Artikel Terkait)

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

Tags: #ansys #transient #thermal #analysis #tutorial