GENERAL

ansys steady state thermal analysis tutorial

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

ANSYS Steady State Thermal Analysis Tutorial: A Complete Guide

Thermal management is a critical concern for engineers across industries—from aerospace and automotive to electronics and energy. When designing components that generate or dissipate heat, predicting temperature distribution accurately is essential to ensure reliability, performance, and safety. ANSYS Mechanical offers a powerful suite of tools for conducting steady state thermal analyses, allowing designers to model heat transfer, evaluate thermal gradients, and optimize designs before they go into production.

This tutorial walks you through every step of a typical steady state thermal analysis in ANSYS Mechanical. Whether you're a seasoned CFD/FEA user or just starting out, you'll find clear explanations, practical tips, and best practice recommendations that will help you master the workflow and avoid common pitfalls.

What Is Steady State Thermal Analysis?

Steady state thermal analysis examines the temperature field in a system when it has reached equilibrium—meaning temperatures no longer change with time. In this mode, heat sources, sinks, and boundary conditions are constant, and the governing equation simplifies to the Laplace or Poisson equation:

" ·(k" T) + q0r Ò

where **k** is thermal conductivity, **T** is temperature, and **q0p** is volumetric heat generation. Because the time derivative drops out, the analysis is computationally faster than transient studies while still providing critical insight into maximum temperatures, heat fluxes, and temperature gradients.

Why Use ANSYS for Thermal Analysis?

- Comprehensive Physics Engine – ANSYS solves coupled multiphysics problems, allowing you to combine thermal, structural, and fluid dynamics analyses.
- Advanced Material Libraries – Access to a vast database of temperature dependent material properties.
- Robust Meshing Tools – Automatic mesh generation with refinement options tailored for thermal gradients.
- Intuitive Workflow – From geometry import to post processing, the interface is designed for engineers, not mathematicians.
- Extensive Documentation & Community – Tutorials, forums, and technical notes help you troubleshoot and learn.

Prerequisites for the Tutorial

1. ANSYS Mechanical (Student or Commercial license) installed on your workstation.
2. A basic 3 D CAD model (e.g., a heat sink, a PCB component, or a simple rectangular block).
3. Material data: thermal conductivity, specific heat, density, and, if applicable, temperature dependent properties.
4. Knowledge of basic finite element concepts (elements, nodes, boundary conditions).

Step by Step ANSYS Steady State Thermal Analysis Tutorial

1. Launching ANSYS Workbench

Open ANSYS Workbench. From the Project Schematic, drag the **Steady State Thermal** system onto the workspace. The system automatically includes the **Geometry**, **Model**, and **Solution** components.

2. Importing or Creating Geometry

You can either import a CAD file (STEP, IGES, Parasolid) or create a simple geometry within the Workbench. For this tutorial, we'll build a rectangular block (100 /mm /× /50 /mm /× /20 /mm) in the **Geometry** module.

- Open the DesignModeler by double click ing the Geometry cell.
- Create a Part !' Box with the specified dimensions.
- Apply a Fillet of 2 /mm to all edges to avoid stress singularities.
- Save and close DesignModeler; the geometry will appear in the Workbench tree.

3. Defining Materials

Assign material properties to the block. For a copper heat sink, you might use:

- Thermal conductivity: 401 /W/(m·K)
- Density: 8960 /kg/m³
- Specific heat: 385 /J/(kg·K)

In the **Model** cell, double click the **Engineering Data** icon. Create a new material, input the values, and link it to the geometry by selecting the part and applying the material.

4. Meshing the Model

A good mesh is crucial for accurate results. In the **Model** cell, click **Mesh** !**Generate Mesh**. For a steady state thermal problem:

- Use Hexahedral elements for regular geometries.
- Set Element Size to 5 /mm or less near expected temperature gradients.
- Apply Mesh Controls !' Size to refine the mesh near the heat source or boundary faces.
- Check the mesh quality metrics; aim for skewness

5. Applying Boundary Conditions

Boundary conditions dictate how heat enters or leaves the system.

1. Heat Source – In the Model cell, under Environment !' Heat Flux, define a uniform heat flux of 10 /kW/m² on the top surface.
2. Convection – Apply natural convection on the side surfaces: Convection !' Convection Coefficient = 10 /W/(m²·K), Ambient Temperature = 25 /°C.
3. Fixed Temperature – Set the bottom face to a fixed temperature of 25 /°C to simulate a heat sink attached to a cooling plate.

Use **Selection** tools to pick faces accurately; double click the face in the graphics window to confirm.

6. Setting Solver Parameters

Open the **Solution** cell. Right click **Solution** !**Insert** !**Temperature**. Then:

- Choose Temperature as the primary result.
- Insert Heat Flux to view the magnitude of heat flow across surfaces.
- Insert Temperature Gradient to identify hot spots and potential thermal stresses.

Under **Solver Settings**, set the solver to **Explicit** or **Implicit** based on problem complexity. For most steady state problems, the default **Implicit** solver works well.

7. Running the Simulation

Click **Compute** in the Solution cell. Monitor the convergence history: the residuals should drop below 1e-6 within 10–15 iterations. If convergence stalls:

- Check for mesh quality issues.
- Verify that all boundary conditions are correctly applied.
- Increase the Solver Tolerance or switch to a more robust solver.

8. Post Processing Results

Once the solve completes, explore the results.

- Temperature Distribution – Use the Temperature result to create color maps. Hover over the graphics to see temperature values.
- Heat Flux Vectors – Display Heat Flux vectors to visualize heat flow direction and magnitude.
- Contours – Overlay temperature contours on the geometry for a quick visual assessment.
- Export data: Right click the result !' Export Data !' CSV to analyze temperature at specific points.

Use the **Report** feature to generate a PDF summary of key results, including maximum temperature, average temperature, and heat flux statistics.

9. Validating and Verifying the Model

Validation ensures that the model reflects reality.

1. Analytical Comparison – For simple geometries, compare with analytical solutions (e.g., heat conduction in a slab).
2. Experimental Data – If available, compare simulation results with temperature measurements from thermocouples or IR imaging.
3. Mesh Sensitivity Study – Refine the mesh further and check if temperatures change by less than 1 /°C.

10. Advanced Topics and Extensions

Temperature gradients can induce thermal strains. Add a **Structural** analysis and couple it to the thermal solution using the **Thermal Structural** coupling feature. This allows you to evaluate stress, deflection, and potential failure modes.

For high temperature components, include radiation by adding **Radiation** boundary conditions. Specify emissivity values (e.g., 0.8 for oxidized metal) and view radiative heat fluxes.

Use the **Design Exploration** module to vary parameters such as heat flux magnitude, convection coefficient, or material conductivity. This helps optimize design for worst case scenarios.

Integrate fluid flow (via ANSYS Fluent) to capture convective heat transfer more accurately, especially in forced convection scenarios. Export fluid domain results to the thermal solver using the **Coupled Solver** interface.

Common Pitfalls and How to Avoid Them

- Incorrect Boundary Conditions – Double check that each face has a single, well defined condition. Over constrained surfaces can lead to non physical results.
- Mesh Coarseness – Coarse meshes may miss steep gradients. Use local refinement around heat sources.
- Material Property Errors – Ensure temperature dependent properties are defined correctly. A typo in conductivity can skew results drastically.

- Solver Divergence – If the solver does not converge, start with a linear solver, then switch to a nonlinear solver if necessary.
- Ignoring Convergence Criteria – Rely on residuals, not just the number of iterations. Verify that residuals are below a strict tolerance.

Tips for Optimizing Your Analysis Workflow

- Use Project Templates – Save a custom template with your preferred mesh settings and boundary conditions to speed up future projects.
- Automate with Python – Write scripts to batch run analyses or post process results.
- Leverage GPU Acceleration – If your workstation has a compatible GPU, enable GPU acceleration for faster solves.
- Document Assumptions – Keep a log of all assumptions (e.g., convection coefficient, emissivity) for future reference and audit trails.

Conclusion

Steady state thermal analysis is a foundational tool for engineers designing heat critical components. By following this ANSYS steady state thermal analysis tutorial, you've learned how to build a model, define materials, apply realistic boundary conditions, mesh effectively, run the solver, and interpret the results. With these skills, you can confidently predict temperature distributions, optimize designs, and ensure compliance with safety