GENERAL

ansys example transient thermal analysis of a pipe

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

ANSYS Example: Transient Thermal Analysis of a Pipe

When engineers design piping systems for power plants, chemical reactors, or HVAC installations, understanding how heat moves through the pipe walls over time is essential. A **transient thermal analysis**allows designers to predict temperature spikes, assess material integrity, and optimize insulation. In this guide, we walk through a detailed ANSYS example that demonstrates how to set up, run, and interpret a time dependent heat transfer simulation for a simple pipe geometry. Whether you are new to finite element analysis (FEA) or looking to refine your thermal modeling workflow, this article provides a step by step walkthrough, practical tips, and best practice insights.

Why Transient Thermal Analysis Matters for Pipes

Unlike steady state models that assume temperatures have reached equilibrium, transient analyses capture the dynamic evolution of heat. Pipes often experience rapid temperature changes due to:

- Sudden changes in fluid temperature (e.g., start up or shut down).
- Fluctuating environmental conditions (e.g., day night cycles).
- Thermal shocks from process interruptions or fault conditions.

By modeling these scenarios, engineers can:

1. Determine peak temperatures that may cause material fatigue.
2. Validate that temperature gradients stay within safe limits.
3. Design effective cooling or insulation strategies.

Setting the Stage: Problem Definition

Our example focuses on a cylindrical pipe segment that carries hot fluid. The pipe is made of carbon steel (AISI 1018) and is exposed to a cooler ambient environment on the outer surface. The simulation will capture the temperature distribution inside the pipe wall and on its outer surface over a 60 second interval.

Key parameters:

- Pipe inner diameter: 0.1 /m
- Pipe outer diameter: 0.12 /m
- Length: 1 /m (only a segment is modeled; symmetry is used)
- Fluid temperature: 350 /°C (constant during the simulation)
- Ambient temperature: 25 /°C
- Convection coefficient: 500 /W/(m²·K)
- Material properties (steel): density = 7850 /kg/m³, specific heat = 470 /J/(kg·K), conductivity = 50 /W/(m·K)

By keeping the geometry simple, we can focus on the workflow and interpretation of results without the distraction of complex meshing or boundary condition setups.

Step 1: Building the Geometry in ANSYS Workbench

Open ANSYS Workbench and create a new project. Drag the **Transient Structural** system onto the Project Schematic. In the **Geometry** module, we will create a cylindrical pipe segment.

1. Click New in the Geometry window. Choose Solid !’ Cylinder.
2. Set the Axis to Z, the Radius to 0.06 /m (outer radius), and the Height to 1 /m.
3. After creating the cylinder, use the Shell feature to subtract the inner cylinder (radius 0.05 /m). This creates the hollow pipe.
4. Apply a Mirror feature to generate a symmetric model. Mirror across the X plane to reduce the model to half the pipe, which saves computational time.

Once the geometry is ready, click **Save** and close the design module.

Step 2: Defining Material Properties

In the **Engineering Data** section, create a new material called “AISI 1018 Steel.” Enter the following values:

- Density: 7850 /kg/m³
- Specific Heat: 470 /J/(kg·K)
- Thermal Conductivity: 50 /W/(m·K)

Assign this material to the entire pipe geometry.

Step 3: Meshing Strategy

Mesh quality is critical for accurate transient results. Follow these guidelines:

1. Use Free Hexahedral elements for the pipe walls. They provide a good balance between accuracy and computational cost.
2. Set the Maximum Size to 0.01 /m and the Minimum Size to 0.002 /m near the inner and outer surfaces where temperature gradients are steep.
3. Apply Element Size Control along the Z axis to refine the mesh near the inlet and outlet if you plan to include fluid–solid coupling in future studies.
4. Check mesh quality metrics (Aspect Ratio

After meshing, review the **Mesh Statistics**to ensure you have a manageable number of elements ("H 20,000–30,000 for this example).

Step 4: Applying Boundary Conditions

Boundary conditions define how heat enters and leaves the system. For our pipe example:

1. Inner Surface (Fluid side): Apply a Temperature boundary condition of 350 /°C. In the Model tab, select the inner surface, choose Temperature, and set the value.
2. Outer Surface (Ambient side): Apply a Convection boundary condition. Set the convection coefficient h /= /500 /W/(m²·K) and ambient temperature T" /= /25 /°C.
3. Symmetry Plane: Apply a Symmetry condition to the plane created by the mirror operation. This ensures no heat flux across the symmetry plane.

Ensure that all surfaces are properly labeled and that the boundary conditions are applied to the correct faces.

Step 5: Selecting the Solver and Time Parameters

ANSYS Workbench’s **Transient Thermal** solver uses implicit time integration, which is stable for the short time horizon of 60 /seconds. Configure the solver as follows:

- Time Step: Start with a fixed time step of 1 /second. This yields 60 steps for the 60 second simulation.
- Output Time: Request results at every 5 /seconds to monitor temperature evolution.
- Enable Time Dependent Convergence Control to automatically adjust the step size if the solution struggles to converge.

Run a quick **Convergence Test** by reducing the time step to 0.5 seconds for the first 10 seconds. If the solution converges smoothly, you can revert to the larger step size to save time.

Step 6: Running the Simulation

Click **Run** and watch the solver progress. During the simulation, monitor:

- Residuals for temperature (RMS and Max) to ensure they drop below 1e 5.
- Solver time per step to gauge performance.
- Memory usage; adjust the mesh if the simulation consumes too much memory.

Once the simulation completes, move to the post processing stage.

Step 7: Visualizing Temperature Distribution

Open the **Results** section and create a **Temperature** plot:

1. Choose Surface as the plot type.
2. Select the outer surface to view the temperature gradient from the fluid side to the environment.
3. Apply a color map (e.g., Rainbow) to highlight hot spots.

Generate a **Contour Plot** for a cross section at the pipe's midpoint (Z = 0.5 m). This reveals how temperature varies radially across the pipe wall.

Step 8: Interpreting Results

Key observations from the transient simulation:

- The inner surface quickly rises to near 350 /°C within the first few seconds.
- The outer surface temperature increases slowly, reaching about 120 /°C after 60 /seconds.
- Temperature gradients across the wall are steep during the early phase and gradually flatten as the system approaches a quasi steady state.

These results help answer engineering questions such as:

1. Does the pipe wall experience temperatures that exceed the material's allowable limits (e.g., 450 /°C for AISI 1018)?
2. Is the insulation thickness sufficient to keep the outer surface below a critical threshold (e.g., 150 /°C for surrounding equipment)?
3. How long does it take for the pipe to reach a steady state, and what is the impact on process scheduling?

Step 9: Sensitivity Analysis and Design Optimization

Once you have a baseline simulation, you can explore design changes:

- Adjust Convection Coefficient: Simulate different cooling scenarios by varying h from 200 to 800 /W/(m²·K).
- Change Pipe Material: Replace steel with stainless steel or aluminum to study the effect on thermal response.
- Modify Pipe Dimensions: Increase wall thickness to reduce temperature gradients.

Use ANSYS's **DesignXplorer** or **Parametric Study** tools to automate these variations and extract performance metrics such as peak temperature, time to reach 90 /% of final temperature, and temperature rise rate.

Step 10: Documenting and Reporting Findings

For stakeholder communication, create a concise report that includes:

- A clear statement of objectives and assumptions.
- Geometry and material details.

- Mesh quality metrics.
- Boundary condition definitions.
- Key results: temperature plots, time history curves, and summary tables.
- Recommendations for design changes.

Export figures as high resolution PNG or PDF files and embed them in the report. Use captions that reference the ANSYS example and highlight the transient nature of the analysis.

Common Pitfalls and How to Avoid Them

Baca Juga (Artikel Terkait)

- **advanced engineering electromagnetics solution manual edition hardcover by balanis constantine a published by wiley**
  URL: http://www.denniswildberger.com/advanced-engineering-electromagnetics-solution-manual-edition-hardcover-by-balanis-constantine-a-published-by-wiley.pdf
- **advanced engineering electromagnetics solutions manual**
  URL: http://www.denniswildberger.com/advanced-engineering-electromagnetics-solutions-manual.pdf
- **advanced engineering electromagnetics wiley 1989 grading**
  URL: http://www.denniswildberger.com/advanced-engineering-electromagnetics-wiley-1989-grading.pdf
- **advanced engineering fluid mechanics by biswas**
  URL: http://www.denniswildberger.com/advanced-engineering-fluid-mechanics-by-biswas.pdf

Tags: #ansys #example #transient