III. Using Finite Element Method’s for Analyzing Thermal Phenomena in ANSYS

3. Transient Thermal Analysis

Transient thermal analysis determines temperatures and other thermal quantities that vary over time. Engineers commonly use temperatures that a transient thermal analysis calculates as input to structural analyses for thermal stress evaluations. Many heat transfer applications - heat treatment problems, nozzles, engine blocks, piping systems, pressure vessels, etc. - involve transient thermal analyses.

A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time.

The ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional, and ANSYS FLOTRAN products support transient thermal analysis.

To specify time-dependent loads, it is possible to use the Function Tool to define an equation or function describing the curve and then to apply the function as a boundary condition, or to divide the load-versus-time curve into load steps.

For individual load steps, each "corner" on the load-time curve can be one load step, as shown in the following sketches.

![Figure 2. Examples of Load vs. Time Curves](image)

For each load step, it is need to specify both load values and time values, along with other load step options such as stepped or ramped loadsautomatic time stepping, etc. Then write each load step to a file and solve all load steps together.

3.1. Elements and Commands Used in Transient Thermal Analysis

Transient thermal analyses use the same elements as steady-state thermal analyses.

3.2. Tasks in a Transient Thermal Analysis

The procedure for doing a transient thermal analysis has three main tasks:

- Build the model.
• Apply loads and obtain the solution.
• Review the results.

3.3. Building the Model

To build the model, have to be used the ANSYS preprocessor (PREP7) for the next tasks:

1. Define the element types.
2. If necessary, define element real constants.
3. Define material properties.
4. Define the model geometry.
5. Mesh the model.

3.4. Applying Loads and Obtaining a Solution

In a transient analysis, the first steps in applying transient loads are to define the analysis type and then establish initial conditions for your analysis.

To establish the initial conditions, can be obtained a steady-state solution, or simply to be specified a uniform starting temperature at all nodes.

3.4.1. Specifying a Uniform Temperature

If we know that the model starts at ambient temperature, we can specify that temperature at all nodes.

The value, specified via the Uniform Temp dialog box or the TUNIF command defaults to the reference temperature, which in turn defaults to zero.

Note
Specifying a uniform starting temperature is not the same as applying a temperature DOF constraint.

The uniform starting temperature is the temperature in effect at the beginning of an analysis, while a temperature DOF constraint forces a node to have the specified temperature until it is deleted.

3.4.2. Specifying a Non-Uniform Starting Temperature

In a transient thermal analysis (but not in a steady-state thermal analysis), it is possible to specify one or more non-uniform starting temperatures at a node or a group of nodes.

We can also apply a non-uniform starting temperature to one or more nodes and at the same time have all other nodes use a uniform starting temperature. It has to be simply specified the uniform temperature before applying the non-uniform temperature to selected nodes.

If the initial temperature distribution is not uniform and is not known, it is need to do a steady-state thermal analysis to establish the initial conditions. To do so, perform these steps:
- Specify the appropriate steady-state loads (such as imposed temperatures, convection surfaces, etc.).
- Specify TIMINT, OFF, THERM (or Time-Time Integration menu) to turn off transient effects.
- Use the TIME command (or Time-Time Step menu) to define a value of time. Typically, the time value is extremely small (e.g., 1E-6 seconds).
- Specify ramped or stepped loading using the KBC command (or Time-Time Step menu). If ramped loading is defined, the effect of the resulting temperature gradients with respect to time should be considered.
- Write the load data to a load step file using the LSWRITE command (or Write LS File menu).

For the second load step, must be deleted any imposed temperatures unless it is known that those nodes will maintain the same temperatures throughout the transient analysis. Also, have to be issued TIMINT, ON, THERM in the second load step to turn on transient effects.

3.4.3. Specifying Load Step Options

For a thermal analysis, we can specify general options, nonlinear options, and output controls.

3.4.3.1. Defining Time-stepping Strategy

The transient problem can be managed either by defining multiple load steps (for stepped or ramped boundary conditions) or by using a single load step and tabular boundary conditions (for arbitrary time-varying conditions) with an array parameter to define time points. However, can be only applied the table method to heat transfer (only) elements, thermal electric elements, thermal surface effect elements, fluid elements, or some combination of these types.

Load step method procedure:

1. Specify the time at the end of the load step using one of these methods.
2. Specify whether your loads are stepped or ramped.
3. Specify the load values at the end of the load step.
4. Write information to a load step file using one of these methods.
5. Repeat steps 1 through 4 for the next load step, then the next, and so on until you have finished writing all load step data to the file.

   To delete any loads (except temperature constraints) we have to set them to zero over a small time interval instead of deleting them.

To use table parameters, we have to follow the next procedure:

1. Define loading profile (i.e., load vs. time) using TABLE type array parameters.
2. Specify automatic time stepping on (AUTOTS, ON). Specify either time step size (DELTIM) or number of substeps (NSUBST).
3. Specify the time step reset option. You can choose to not reset the time stepping during the solution, to reset the time based on an already-defined array of time values (keytimes), or to reset the time based on a new array of keytimes.
4. If in case of selection new while working interactively, the system will ask to fill in the nx1 array of keytimes at this time. For working in batch mode, have to be defined
the array before issuing **TSRES**, which resets the time step to the initial value as specified on **DELTIM** or **NSUBST**.

5. In case of using an array of time values \((FREQ = \%\text{array}\% \text{ on the OUTRES command})\) in conjunction with a time step reset array (TSRES command), have to ensure that any **FREQ** array time values exceed the nearest **TSRES** array value by the initial time step increment specified with **DELTIM, DTIME** or **NSUBST, NSBSTP**. For example, if we have a **FREQ** array with the values 1.5, 2, 10, 14.1, and 15, and a **TSRES** array with the values 1, 2, 10, 14, and 16 (where the time stepping would restart at those values), and we specify an initial time step increment of \(DTIME = .2\), the program will stop. In this example, the requested **FREQ** array value of 14.1 does not exist, because the **TSRES** value specified that the time step be reset at 14 and increment at an interval of .2; therefore, the first available time for the **FREQ** array would be 14.2.

**Note**

**TSRES** is used only with **AUTOTS, ON**. If constant time stepping is used (**AUTOTS, OFF**), **TSRES** is ignored.

6. To create a keytime array, the time values in the array must be in ascending order and must not exceed the time at the end of the load step as defined on the **TIME** command.

7. During solution, the time step size will be reset at the keytimes identified in the array. Time step sizes are reset based on initial time step size \([\text{DELTIM, DTIME}]\) or number of substep \([\text{NSUBST, NSBSTP}]\) settings.

8. Specify when the information is to be written to the results file using an nx1 array type parameter, just as it was done with the keytime array. We can use the same keytime array that we used to reset time stepping, or we can use a different array. If working interactively, we can create the array at this time or use an existing array. If working in batch mode, we must define the array before issuing **OUTRES**.

**Note**

We can use the **TSRES** command and time stepping strategy *only* if using the following heat transfer (only) elements, thermal electric elements, thermal surface effect elements, fluid element FLUID116, or some combination of these types.

### 3.4.3.2. General Options

General options include the following:

- **Solution control option**

  This option turns solution control heuristic ON/OFF for thermal analysis. With this option turned ON, we normally specify the number of substeps (**NSUBST**) or the time step size (**DELTIM**), and the time at the end of the load step (**TIME**). The remainder of the solution control commands then default to their optimal values for the particular thermal problem.

  To turn solution control ON or OFF, use either of the following:

- **The time option**
This option specifies time at the end of the load step.

The default time value is 1.0 for the first load step. For subsequent load steps, the default is 1.0 plus the time specified for the previous load step.

- **Number of substeps per load step, or the time step size**

A nonlinear analysis requires multiple substeps within each load step. By default, the program uses one substep per load step.

In regions of severe thermal gradients during a transient (e.g., surfaces of quenched bodies), there is a relationship between the largest element size in the direction of the heat flow and the smallest time step size that will yield good results. *Using more elements for the same time step size will normally give better results, but using more substeps for the same mesh will often give worse results.* When using automatic time stepping and elements with midside nodes (quadratic elements), ANSYS recommends to control the maximum time step size by the description of the loading input and define the minimum time step size (or maximum element size) based on the following relationship:

\[
\text{ITS} = \frac{\Delta^2}{4\alpha}
\]

The \(\Delta\) value is the conducting length of an element (along the direction of heat flow) in the expected highest temperature gradient. The \(\alpha\) value is the thermal diffusivity, given by \(k/\rho C\). The \(k\) value is the thermal conductivity, \(\rho\) is the mass density, and \(C\) is the specific heat.

If the above relationship (\(\text{ITS} = \frac{\Delta^2}{4\alpha}\)) is violated when using elements with midside nodes, ANSYS typically computes unwanted oscillations and temperatures outside of the physically possible range. When using elements without midside nodes, the unwanted oscillations are unlikely to occur, and the above recommendation for the minimum time step can be considered somewhat conservative.

**Caution**

*Avoid using extremely small time steps, especially when establishing initial conditions.* *Very small numbers can cause calculation errors in ANSYS. For instance, on a problem time scale of unity, time steps smaller than 1E-10 can cause numerical errors.*

In case of applying stepped loads, the load value remains constant for the entire load step. For ramp loads (the default), the load values increment linearly at each substep (time step) of the load step.

- **Monitor Results in Real Time**

The **NLHIST** command allows to monitor results of interest in real time during a solution. Before starting the solution, we can request nodal data such as temperatures or heat flows and can also request element nodal data such as thermal gradients and fluxes at specific elements to be graphed. The result data are written to a file named Jobname.nlh. Nodal results and contact results are monitored at every converged
substep while element nodal data are written as specified via the OUTRES setting. You can also track results during batch runs.

3.4.4. Nonlinear Options

For single-field nonlinear thermal analysis, ANSYS allows a choice of three solution options. The Full option corresponds to the default full Newton-Raphson algorithm. The Quasi option corresponds to only selective reforming of the thermal matrix during solution of the nonlinear thermal problem. The matrix is only reformed if the nonlinear material properties changed by a significant amount (user-controlled). This option performs no equilibrium iterations between time steps. Material properties are evaluated at the temperatures at the beginning of the load step. The Linear option forms only one thermal matrix at the first time step of a load step. This option should only be used to obtain a quick approximate solution.

These options in ANSYS can be selected by the THOPT command. The Quasi and Linear solution options perform direct assembly of the thermal matrix and only the ICCG and JCG solvers support solutions under this option. We can choose either of these solvers using the EQSLV command.

For the Quasi solution option, have to be also specified the material property change tolerance use for matrix reformation. The reform tolerance defaults to .05, corresponding to a 5% change in material properties. The Quasi option sets up a single fast material table, with equal temperature points between a maximum and a minimum temperature for evaluation of temperature-dependent material properties. Using this option have to be also specified the number of points (defaults to 64) and the minimum and maximum temperature (defaults to the minimum and maximum temperature defined by the MPTEMP command) for the fast material table. All other nonlinear load options are valid with the THOPT command.

Specify nonlinear load step options only if nonlinearities are present. Nonlinear options include the following:

- **Number of equilibrium iterations**

  This option specifies the maximum allowable number of equilibrium iterations per substep. With SOLCONTROL,ON, this command defaults to between 15 and 26 iterations, depending upon the physics of the problem.

- **Automatic Time Stepping**

  Also called time step optimization in a transient analysis, automatic time stepping allows ANSYS to determine the size of load increments between substeps. It also increases or decreases the time step size during solution, depending on how the model responds. In a transient thermal analysis, the response checked is the thermal eigenvalue. For the THOPT,Quasi option, the time step size is also adjusted based on property change during solution. If the eigenvalue is small, a larger time step is used and vice versa. Other things considered in determining the next time step are the number of equilibrium iterations used for the previous time step, and changes in the status of nonlinear elements.
For most problems, we can turn on automatic time stepping and set upper and lower limits for the integration time step. The limits, set via the NSUBST command or DELTIM command, or the menu path shown below, help to control how much the time step varies.

- **Time integration effects**

  These load step options determine whether the analysis includes transient effects such as structural inertia and thermal capacitance.

**Note**

The ANSYS program assumes time integration effects to be on in a transient analysis (unless they were turned off to establish initial conditions). If time integration effects are turned off, ANSYS calculates a steady-state solution.

- **Transient integration parameters**

  These parameters control the nature of your time integration scheme and specify the criteria for automatic time stepping.

  To minimize inaccuracies in a solution, can be set the transient integration parameter (the THETA value) to 1.0.

- **Convergence tolerances**

  The ANSYS program considers a nonlinear solution to be converged whenever specified convergence criteria are met. Convergence checking may be based on temperatures, heat flow rates, or both. We specify a typical value for the desired item (VALUE field on the CNVTOL command) and a tolerance about the typical value (TOLER field). The convergence criterion is then given by $VALUE \times TOLER$. For instance, if we specify 500 as the typical value of temperature and 0.001 as the tolerance, the convergence criterion for temperature is 0.5 degrees.

  For temperatures, ANSYS compares the change in nodal temperatures between successive equilibrium iterations ($\Delta T = T_i - T_{i-1}$) to the convergence criterion. Using the above example, the solution is converged when the temperature difference at every node from one iteration to the next is less than 0.5 degrees.

  For heat flow rates, ANSYS compares the out-of-balance load vector to the convergence criterion. The out-of-balance load vector represents the difference between the applied heat flows and the internal (calculated) heat flows.

  As nonlinear thermal analysis proceeds, ANSYS computes convergence norms with corresponding convergence criteria each equilibrium iteration. Available in both batch and interactive sessions, the Graphical Solution Tracking (GST) feature displays the computed convergence norms and criteria while the solution is in process. By default, GST is ON for interactive sessions and OFF for batch runs.

- **Termination settings for unconverged solutions**
If the ANSYS program cannot converge the solution within the specified number of equilibrium iterations, ANSYS either stops the solution or moves on to the next load step, depending on what you specify as the stopping criteria.

- **Line search**

  The line search option allows ANSYS to perform a line search with the Newton-Raphson method.

- **Predictor-corrector option**

  This option activates the predictor-corrector option for the degree of freedom solution at the first equilibrium iteration of each substep.

### 3.5. Phase Change

One of the ANSYS program's most powerful features for thermal analysis is its ability to analyze phase change problems, such as a melting or solidifying process. Some of the applications for phase change analysis include:

- The casting of metals, to determine such characteristics as the temperature distribution at different points during the phase change, length of time for the phase change to occur, thermal efficiency of the mold, etc.
- Production of alloys, where chemical differences instead of physical differences cause the phase change.
- Heat treatment problems.

To analyze a phase change problem, have to be performed a nonlinear transient thermal analysis. The only differences between linear and nonlinear transient analyses are that, in nonlinear analyses:

- We need to account for the latent heat; that is, heat energy that the system stores or releases during a phase change. To account for latent heat, have to be defined the enthalpy of the material as a function of temperature.

![Sample Enthalpy vs. Temperature Curve](image)

*Figure 3. Sample Enthalpy vs. Temperature Curve*
Enthalpy, which has units of heat/volume, is the integral of density times specific heat with respect to temperature:

\[ H = \int \rho c(T) \, dT \]

In nonlinear analysis, must be specified a small enough integration time step for the solution. Also, have to be turned on automatic time stepping so that the program can adjust the time step before, during, and after the phase change.

- Use lower-order thermal elements, such as PLANE55 or SOLID70. If have to use higher-order elements, choose the diagonalized specific heat matrix option using the appropriate element KEYOPT. (This is the default for most lower-order elements.)
- When specifying transient integration parameters, set \( \text{THETA} \) to 1, so that the Euler backward difference scheme is used for the transient time integration. \( \text{THETA} \) defaults to 0.5.
- It can be find the line search option helpful in phase change analyses.