

# Setting Transient Analysis Parameters

Use *Transient Analysis Parameters* to complete the transient analysis setup and to set the solver controls. For any transient simulation, you must define the time span for the solution and the results output interval. In addition, you should also specify the *Time Step*, which is the time "mesh" for the solution. For most models you will not have to reset any of the other control parameters: the default settings will provide good results.

Define the time interval for the simulation by entering a *Start Time* and an *End Time*. The thermal model will be solved for transient responses over the specified time span. There are two ways to specify End Time:

- Use *At time* for most analyses, specifying the analysis time in seconds.
- For orbital analysis *At the end of* allows you to specify the number of orbits in the analysis as an integer.

## Integration Controls

### Solution Method

Select the method that TMG will use to solve the thermal model:

#### Backward

The *Backward* solution method is an implicit differencing scheme, where the element heat balance equations are evaluated at the end of the integration time step. Because the method is implicit, all element temperatures must be solved iteratively at every time step; this makes it less efficient than the *Forward* method for a given time step. However, the Backward technique is unconditionally stable (will not diverge) for any value of integration time step. It also conserves system energy.

This method is recommended for most applications: it is the most reliable and easy to use option. In particular, it is most effective for models where the minimum element time constant ( $\tau_{\min}$ ) is small compared to the solution interval. It is more accurate than the Forward-Backward method under conditions of rapid temperature change.

#### Forward

The *Forward* differencing technique is a fully explicit integration scheme. It is accurate, computationally efficient, and conserves the system energy. Elements with zero mass (capacitance) are solved directly by iteration at every time step.

The drawback of the *Forward* method is that the numerical solution can become unstable (divergent) if the integration time step is larger than the minimum element time constant ( $\tau_{\min}$ ). Therefore this method is recommended only for models where  $\tau_{\min}$  is reasonably large: generally it should be no less than .0001 times the solution interval. See *Time Step* below for help on determining  $\tau_{\min}$ . Also, it is possible to use *Model Simplification* to increase a model's  $\tau_{\min}$ ; see *Simplifying the Model*.

## Backward-Forward

The *Backward-Forward* or Crank-Nicholson method is an implicit differencing scheme, where the element heat balance equations are evaluated at the midpoint of an integration time step. It is generally quite accurate, but can yield errors and temperature oscillations when the temperature change is very non-linear.

Since the method is implicit, all element temperatures must be solved iteratively at every time step; this makes it less efficient than the Forward method for a given time step. However, the Forward-Backward technique is unconditionally stable (will not diverge) for any value of integration time step. It also conserves system energy. It is the recommended option for conditions of linear temperature change.

## Exponential Forward

The *Exponential Forward* method is an explicit technique where the element temperature profile over an integration time step is assumed to be an exponential decay rather than a linear change. It is as efficient as the *Forward* method, and is unconditionally stable. Its disadvantage is that it does not conserve system energy over time, and can therefore produce inaccurate results with time steps which are larger than  $\tau_{\min}$ . The *Exponential Forward* method can be very effective for solving nonlinear models with rapidly varying boundary conditions where the system energy balance is dominated by external source terms.

## Time Step

The *Time Step* is the time "mesh" for the transient solution. Smaller values (a finer mesh) will always give more accurate results at the price of increased computation time. Model solve time is inversely proportional to the size of the *Time Step*.

To define a fixed time step, either specify a *Number of Time Steps* to be evenly distributed over the length of the analysis or enter the *Constant* value of the time step.

Select *Time varying* to specify a variable time step; use *Create Table...* to set up the variation of the time step with solution time. This is a very effective way to refine the time mesh for only those portions of the solution interval where the temperatures are changing rapidly.

The element time constant,  $\tau$  is equal to the RC product, each element's Resistance times its capacitance. This value determines how quickly or slowly an element responds to temperature changes in the region.

You can set the *Time Step* to be related to the minimum element time constant  $\tau_{\min}$ . Select *Time Constant Multiplier* and enter a value. During the solve, TMG will determine the current value of  $\tau_{\min}$  and use it to compute the integration time step.

## Auto-calculate Time Step

The Auto-calculate Time Step option allows the integration time step for transient runs to be automatically increased until a maximum temperature change criterion is reached. You must specify a maximum temperature permissible change over the time step in model units.

Auto-calculate Time Step may reduce calculation time in models where a small time

step is required only for certain times in the transient analysis. During the solve, TMG estimates the temperature change at each integration time step, and if the estimated temperature change for all elements is less than the specified value, the time step is appropriately increased. Similarly, as the temperature change increases, the time step decreases. The Minimum Time Step option allows you to set a limit on how small the time step can get.

## Notes on Selecting a *Time Step*

- You should choose the *Time Step* in the same manner that you select a mesh density in generating elements. If element temperatures can change rapidly, a smaller time step (higher time mesh density) is required for accurate representation.
- If you have selected the *Forward* differencing solution method, you must select *Time Constant Multiplier* and use a multiplier of 0.5 or less. Be aware that often  $\tau_{\min}$  can be so small that the resulting solution time will be impractically large. Use another solution method for such cases.
- You can determine  $\tau_{\min}$  by selecting the Element Reporting option *Element Time Constants*. The solver must be activated to produce this output; simply run a steady-state case with an *Iteration Limit* equal to 1. You then select the appropriate *Time Step* and restart the model to solve for transient response.
- To generate a post processing display showing Element Time Constants (RC Products), create a card 9 text entity that includes the following text:

```
PRINT 0 0 ELRCPROD
```

See the article *Defining TMG Entities* for more information on text entities. After the analysis, click the *Get Results* icon, and select the *All Results* option. The Element Time Constant (RC product) results appear in the Results Selection form list as RC PRODUCT.

## Convergence Control

To solve a transient thermal model, TMG uses iterative techniques at every time step. You can control this process with the following parameters.

### Iteration Limit

Enter a limit on the number of iterations that TMG will perform for a single time step. This limit does not affect solution accuracy; it simply sets a "hard stop" for the iterative solver. If you find that convergence has not been achieved for some time steps, increase this limit. You may also consider some *Advanced Solver Options* to improve convergence.

### Relaxation Factor

Use the *Relaxation Factor* to increase the stability of the iterative solution. Smaller relaxation values can reduce numerical instabilities in the iterative solve at the expense of slower convergence. For highly non-linear models with a coarse time mesh, you may need to specify a relaxation value of 0.1 or less; you should also increase the *Maximum Number of Iterations*. For linear models with a fine time mesh, a *Relaxation Factor* of

1 minimizes solve time.

## Matrix Updating

TMG uses an iterative scheme to solve the model across a given time step when an implicit integration scheme is selected. At each iteration, the conductance matrix is re-linearized using the current temperatures, and the Conjugate Gradient algorithm is used to solve the resulting linear system of equations.

*Update Radiative Conductances when  $dT >$*  allows you to control when certain conductances are updated during the analysis. If, between one iteration and the next, the temperature for at least one element has changed by more than the specified value, radiative and other nonlinear conductances will be updated. If the maximum element temperature change is less than the specified value, the time step is converged. Enter the value for determining convergence; typically a value of  $1^\circ$  will give good results. Smaller values will increase both solution accuracy and solve time. (Note that the temperature change is not the final indicator of a converged solution. You should always check the global heat balance for the solution.)

*Convergence when Temperature Difference  $<$*  allows you to set a maximum permissible temperature difference between successive iterations at a given time step. TMG will determine that the problem has converged because the differential temperature is less than the value that you have set. Enter the value for determining convergence; typically a value of  $0.001^\circ$  will give good results. Smaller values will increase both solution accuracy and solve time. TMG will update the conductance matrix at every iteration. This option is preferred when radiative conductances vary greatly from one time step to the next. (Note that the temperature change is not the final indicator of a converged solution. You should always check the global heat balance for the solution.)

## Periodic Convergence

With *Periodic convergence*, TMG uses the temperature difference criterion to compare the temperature of each element at the beginning and end of one cycle. If the maximum temperature difference of all elements is less than the temperature difference criteria, the solution is reached.

When using *Periodic convergence*, the *End Time* specified for the transient analysis controls the maximum number of periods for the analysis.

You can obtain results for the whole analysis or for the last period only.

You can also use the Periodic Convergence option to run a transient analysis until it reaches a steady state. Specify a Convergence Period and a  $\Delta T <$  that imply thermal equilibrium. Deselect the option for Keep Last Period Results Only.

## Results Output

Thermal model results (Analysis Datasets) will be produced at the specified time interval. Results are always produced at the beginning and end times. See *Setting Solution Data* for a description of the different types of thermal model data that are available.

You can also define results creation as a specified number of times for each orbit.

With *At Specified Times*, you can generate results at any time during the transient analysis. Pick the *Specify...* button to open the Results Output Times form. The form shows a table with the start time and the end time, at which times output is always generated. The Results Output Times form provides tools for specifying output times:

- You can enter time values in seconds in the text entry field at the lower left side of the form, then click the dot icon to enter the time in the table.
- You can enter an integer in the *intervals* field, then click *Fill Table with* to generate the requested number of equally spaced intervals between output times.
- You can click *Import from file...* to import a text file that contains a list of time values (one value per line). The imported results output data is read into the model file and retained.
- You can export a table of results output times by clicking on the *Export to file ...* button. The exported file can later be imported into another model by selecting Import from File.

## Additional Options and Flow Options

Use *Additional Options* to adjust the default convergence parameters, or to select an alternate conduction algorithm. An option to output detailed convergence data is also available. See *Additional Solver Options*.

Use *Flow Options* to set the duct flow solver controls. See *Setting Flow Options*.

---

*Copyright (c) 2007 UGS Corp. All Rights Reserved.*