

ANSYS - Time Dependent Boundary Condition

ANSYS offers boundary condition features that allows users to apply time dependent properties for more advanced simulations. In the following example you will learn how to create such a boundary condition. Please download the spreadsheet and the workbench archive before you proceed.

[Download the spreadsheet here](#)
[Download the archive here](#)

Note: To restore the workbench file from the archive, click on **File > Restore Archive** and select your desired directory. The spreadsheet contains data of temperature in Celsius vs. time in seconds.

Procedure

Double click on  **Model** to launch ANSYS Mechanical. The model has been created and meshed for your convenience. The temperature boundary condition is set to a constant on the left face of the model. We will now demonstrate an easy way to create a time dependent temperature BC.

The data in the spreadsheet ends at **15** seconds. It is critical that you match the end time in the spreadsheet to the step end time in ANSYS Mechanical.

Click on  **Analysis Settings** under the  **Transient Thermal (A5)** tree in the outline window and enter **15** for the **Step End Time**.

Click on  **Temperature** and you should see a constant temperature at 22 degrees from 0 to 15 seconds. Copy the time data in spread sheet and **single click** on the first cell in the first column (Time) and copy. Do the same for the temperature data in the first cell in the second column. The tabular data should look like this:

