

# Creating Thermal Boundary Conditions

*Thermal Boundary Conditions* define known thermal or electrical conditions in the model. The Thermal Boundary Condition form sets up a variety of heat loads and fluxes, fixed temperatures, and joule heating. Separate forms are available to define initial temperatures and thermostats. (This document does not address *Boundary Condition Sets* and *Boundary Condition Mapping*.)

Depending on what type of boundary condition you wish to create, it can be applied to 2D thin shell elements, beam elements, 3D solid elements and lumped mass elements. Thermal Boundary Conditions can also be applied to geometry groups, which makes them associative.

Elements having fluid material properties do not support any Thermal Boundary Condition type.

[Locate the icon.](#)

## Thermal Boundary Condition Types

Use the *Thermal Boundary Condition - Create* form to specify most types of Boundary Conditions. The two exceptions are the *Thermostat* boundary condition, and the *Initial Temperature* boundary condition, which each have their own form.

- A heat load defines the heat flow (*energy/time*) or power into the selected geometry (or elements).
  - Select *Total Load* to define the total heat load for the selected elements. The software apportions the heat load to individual elements based on the volume of the 3D elements, or the surface area for other element types. You cannot mix 3D elements with other orders of element (2D, 1D or lumped mass element types) in your selection.
  - Select *Load/Element* to define the heat load for each element in the selection. Use when elements are the same order and same size.
- A *Flux* boundary condition defines the heat flux or heat load per unit area for shell, beam, or lumped mass elements (*energy / (time x area)*). For shell elements the area is the element area. For beam elements, area is equal to the element's beam section perimeter multiplied by its length. For Lumped mass elements, area is equal to the surface area of a sphere with the specified diameter. (3D elements are not supported.)
- A *Heat Generation* boundary condition defines the heat flux or heat load per unit volume (*energy / (time x volume)*). For shell elements, the volume is equal to the surface area multiplied by the element thickness. For beam elements, the volume is equal to the beam section area times the beam length. If the *Heat Generation* boundary condition is applied directly to 3D elements, volume refers to the volume of the selected elements.
- A *Temperature* boundary condition becomes a heat *source* or *sink* at a fixed temperature within the thermal model.
- A *Radiation* boundary condition defines a view factor and environmental temperature. Heat is radiated to/from the surface to the specified environment temperature as  $q_r = \sigma VF \varepsilon_s A_s (T_s^4 - T_e^4)$ , where  $\varepsilon_s$  and  $A_s$  are the emissivity and area of the surface

(respectively).

For shell elements the area is the element area. For beam elements, area is equal to the element's beam section perimeter multiplied by its length. For Lumped mass elements, area is equal to the surface area of a sphere with the specified diameter. For Radiation boundary conditions defined on geometry groups containing the edges of surfaces meshed with shell elements, the area is equal to the thickness of the shell multiplied by the length of the edge. For Lumped mass elements, area is equal to the surface area of a sphere with the specified diameter. Volumes and 3-D elements are not supported.

- A *Convective* boundary condition defines a heat transfer coefficient and environmental temperature. Heat is convected to/from the surface to the specified environment temperature as  $q_c = h A_s (T_s - T_e)$ . For shell elements the area is the element area. For beam elements, area is equal to the element's beam section perimeter multiplied by its length. For Lumped mass elements, area is equal to the surface area of a sphere with the specified diameter. For Radiation boundary conditions defined on geometry groups containing the edges of surfaces meshed with shell elements, the area is equal to the thickness of the shell multiplied by the length of the edge. For Lumped mass elements, area is equal to the surface area of a sphere with the specified diameter. Volumes and 3-D elements are not supported.
- *Voltage* and *Current* boundary conditions model Joule heating in an electrical network. In addition to specifying voltage or current values, you must also define the elements' electrical resistivity as a material property. All these values must be entered in SI (i.e. volts or amps on the Boundary Condition form, and ohm-meters on the Define Material form), regardless of any other units in the model.

Set up an electrical network by defining at least one Voltage boundary condition, and at least one other Joule heating boundary condition (either Voltage or Current). Provide a pathway between the Joule heating boundary conditions made up of elements with a defined electrical resistivity. An electrical network is analogous to a thermal network, in that it must include at least three components:

- elements defined as a voltage source (analogous to a heat source)
- other elements defining a voltage differential or current (analogous to a heat sink)
- an electrical path of elements that share nodes and have electrical resistivity defined (analogous to a heat path with thermal conductivity defined)

TMG will solve the system to compute the complete current flow through the system. The dissipative heat loads generated by this current flow ( $p = vi$ ) are included in the thermal model. Since the electrical resistivity can be defined as a function of temperature, it is a fully coupled solution. *Interpolation Relationships* can also be used to define resistivity as a function of temperature, or voltage or current as a function of time. If you use this technique, you must enter values in SI (volts, amps and ohm-meters). See the article on *Defining an Interpolation Relationship*.

You can use the same element types (1-D, 2-D or 3D) for an electrical network as you do for a thermal network.

## Other Boundary Condition Types

## Thermostat

A *Thermostat* boundary condition controls the heat loads into elements based on the temperature of a single *Sensor Element*. Thermostats are only valid for transient analysis and will be ignored for steady state analysis. The heat load specified on this form is the only one controlled by the thermostat. For the different Boundary Condition types on the menu, see the explanation above. If you wish to control all heat loads defined using the Thermal Boundary Conditions form, select *All Applied Heat Loads* under *Heat Load Type*. The Cut-In and Cut-Out temperatures allow you to specify thermostat hysteresis.

## Initial Temperature

An *Initial Temperature* boundary condition defines the initial temperature of elements for a thermal simulation. The initial temperatures that you specify will not affect the final results, but can greatly influence solution time. Initial Temperature boundary conditions work in conjunction with the Initial Conditions settings defined for the analysis. They will only have an effect with the default *Auto Determine* option or the *Specify a Uniform Value* option. They will be ignored when used in conjunction with the *Start From Results in File* or *Start From Results in Directory* options.

## Mapped Temperatures

A Mapped Temperatures boundary condition allows you to use the results of a previous analysis to define fixed temperatures that vary across a surface, edge or volume. To create this type of temperature boundary condition:

1. Run an preliminary analysis to obtain the temperature results that you want to use as fixed temperatures.
2. Note the location of the TEMP file produced by the preliminary analysis.
3. While preparing the primary analysis, use the Model Manager to create a Generic Entity that references the temperature results in this TEMPF file, filling out the form as follows:
  - *Name*: (Give the Generic Entity a meaningful name.)
  - *Element Set 1*: (Select the elements whose temperature you want to fix. These should match elements in the TEMPF file containing the temperature results from the preliminary analysis)
  - *Type*: SINKFILE (enter this word exactly as shown)
  - *Text or Path*: (Enter or browse to the TEMPF file.)

### See Also

- [Using Boundary Conditions Sets](#)
- [Simulating a Peltier Cooler](#)
- [Mapping Convective Boundary Conditions](#)

## Applying Thermal Boundary Conditions to Edges of a Surface, or Surfaces of a Solid

All boundary conditions are performed on elements (or geometry that references elements), and never on nodes. This has implications for the way you define Thermal Boundary Conditions.

If you want to apply boundary conditions to the edge of a shell meshed surface, you can select

the Free Edges of the shell elements (or a geometry group containing the edge of the surface defining the Free Edges). Material properties for the Free Edges will be the same as for their shell elements. Similarly, if you want to apply boundary conditions to the surface of a solid meshed volume, you can select the Free Faces of the solid elements, or a geometry group containing the surface of the volume defining the Free Faces. Use the selection filter to select Free Edges or Free Faces.

However, if you want to distinguish the material properties or thickness of an edge from its parent surface, you can create additional beam elements on the edge, defining the beam elements with different characteristics. Similarly, you can distinguish the material properties of a surface layer on solid elements by creating additional shell elements on the surface.

### See Also

- [How TMG Uses Elements and Nodes](#)

## Time Varying Boundary Conditions

Select *Time Varying* if the boundary condition on the selected elements changes with time. The boundary condition at the elements at specific times during the transient solution will be interpolated from the tabular data you create. See *Working with Tables* (under *Working with TMG, Common Tools*) for details.

*Table Multiplier* multiplies the dependant variable Y (boundary condition value) by the specified value. This tool allows you to define different (boundary condition versus time) relationships using a common table.

## Applying Boundary Conditions to Different Orders of Geometry/Elements

TBC Type	Applied to	Select	Solver uses
Total Load <sup>1</sup> , Total Current <sup>1</sup> , Load/Element <sup>2</sup> , Current/Element <sup>2</sup>	Volume	Solid elements or geometry group containing volume meshed with solid elements	Actual volume
	Area	Shell elements or geometry group containing surface meshed with shell elements	Actual surface area
		Free faces of solid elements or geometry group containing surface of volume meshed with solid elements	
	Implied Area	Beam elements or geometry group containing edge/curve meshed with beam elements	Area = Perimeter of Beam Section × length of edge

		Free edges of shell elements or geometry group containing edge of surface meshed with shell elements	Area = Thickness of shell elements × length of edge
		Lumped mass	Area = Surface of a sphere of the specified diameter (physical property)
Temperature, Voltage	Volume	Solid elements or geometry group containing volume meshed with solid elements	Specified value at each element's main calculation point
	Surface	Shell elements or geometry group containing surface meshed with shell elements	Specified value at each element's (or free face's) main calculation point
		Free faces of solid elements or geometry group containing surface of volume meshed with solid elements	
	Edge	Beam elements or geometry group containing edge meshed with beam elements	Specified value at each element's (or free edge's) main calculation point
		Free edges of shell elements or geometry group containing edge of surface meshed with shell elements	
Point	Lumped mass	Specified value at point	
Heat Flux <sup>4</sup> , Radiative, Convective	Area	Shell elements or geometry group containing surface meshed with shell elements	Actual surface area
		Free faces of solid elements or geometry group containing surface of volume meshed with solid elements	
		Beam elements or geometry group containing edge meshed with beam elements	Area = Perimeter of Beam Section × length of edge

	Implied Area	Free edges of shell elements or geometry group containing edge of surface meshed with shell elements <sup>4</sup>	Area = Thickness of shell elements × length of edge
		Lumped mass	Area = Surface of a sphere of the specified diameter (physical property)
Heat Generation <sup>3</sup>	Volume	Solid elements or geometry group containing volume meshed with solid elements	Actual volume
	Implied Volume	Shell elements or geometry group containing surface meshed with shell elements	Volume = Surface area × thickness of shells
		Beam elements or geometry group containing edge meshed with beam elements	Volume = Area of Beam Cross Section × length of edge
		Lumped mass	Volume = Volume of a sphere of the specified diameter (physical property)

**Notes:**

1. Total value is apportioned to individual elements based on element volume for solid elements; based on element surface area for surfaces/shells; based on implied element surface area for edges/beams and lumped mass. If the selection for a Boundary Condition includes solid elements, then no surfaces, shells, beams, edges, or lumped mass element may be included.
2. Load or Current per element should only be applied to elements of the same order and size (or geometry groups related only to elements of the same order and size). The specified value (Load or Current) is applied to each element.
3. Total value is apportioned to individual elements based on element volume, or implied element volume.
4. Heat Flux cannot be applied to Free Edges of shell elements nor to the edge of surface meshed with shell elements. Mesh the edge with beams instead.
5. Do not mix different orders of geometry in one geometry group intended to define a boundary condition.

**Axisymmetric Element Selection**

TBC Type	Applied to	Select	Solver uses
Total Load, Total Current, Load/Element,	Volume	Axisymmetric solids	Rotated volume
	Area	Axisymmetric shells	Rotated area

Current/Element		Free edges of axisymmetric solids	Rotated area
Temperature, Voltage	Volume	Axisymmetric solids	Specified value at each calculation point
	Area	Axisymmetric shells	
		Free edges of axisymmetric solids	
Heat Flux, Radiative, Convective	Area	Axisymmetric shells	Rotated area
		Free edges of axisymmetric solids	
Heat Generation	Volume	Axisymmetric solid	Rotated volume

---

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