Contents | Find

Mapping Convective Boundary Conditions

Use Boundary Condition Mapping to map externally generated boundary condition data (either convective or heat flux) onto your thermal model. You must prepare the data in a free format text file, consisting of the boundary condition values at locations specified in terms of global xyz coordinates. Convective data must be in the form of coefficients and fluid temperatures in model units. Heat flux data consists of values for Q (heat flux) in model units.

For each mapping run, elements on a single surface can be selected. Only thin shell elements (single & multi-layer), and axisymmetric shell elements are supported. Transient, and steady state solutions are supported. A convective or heat flux Thermal Boundary Condition is created for each selected element, as well as universal file datasets of the mapped data for visualization.

You can perform multiple Boundary Condition Mapping runs in the same Run Directory to create either convective or heat flux Thermal Boundary Conditions on different surfaces of the model. The mapping runs generate a single Include file *mapping.incl* in the Run Directory, which references the created boundary conditions. To include the results of the mapping in an analysis, select the *mapping.incl* file using the *Include File* icon on the Study Setup form.

The data points do not need to match the elements or the nodes. The mapping function computes values for each element by interpolating the data provided.

Set up the mapping as follows:

1. Create a text file containing data points for a single surface, defining the boundary condition in terms of global xyz coordinates, and either heat transfer coefficient and temperature values or heat flux values. This is called the data points file.

2. Define a group containing the elements on the same surface corresponding to the coordinates in the data point file. These elements will receive the thermal load of the mapped data.

3. Pick the Boundary Condition Mapping icon. A menu appears in the Graphics region.

4. Pick Create on the menu.

5. At the prompt, enter the path and file name of the data points file.

6. At the prompt, enter the name of the element group to which you wish to apply the convective or heat flux boundary conditions. Group name is case sensitive.

7. Enter a message level (brief or verbose) and enter *Yes* to perform the mapping. The convective or heat flux boundary condition is created automatically as file called incl1. dat, in the TMG input file (INPF) format. The mapping also creates an Include file named mapping.incl in the run directory.

8. You can repeat steps 1 through 7 as many times as you wish to map data to different surfaces, using the same run directory.

9. To include the results of the mapping run(s) in an analysis, select the mapping.incl file using the *Include File* icon on the Study Setup form.

Format for the Data Points File

The data file used by the mapping program contains the global coordinates of the data point (x, y, z) as well as the data to be mapped at that point: either convection coefficient (*h*) and temperature (*T*), or heat flux (*Q*). By using the appropriate file format, you can also define transient a convective boundary condition.

If you are mapping *h* and *T* (rather than *Q*), there is another option available. If the element group to which you are mapping the data is comprised of multi-layer shell elements or shell elements referenced by a Reverse Sides entity, you can define values for *h* and *T* on the top side, on the bottom side or on both top and bottom sides by using the appropriate file format.

Be sure that the units of the data file are consistent with the current model units. The maximum length for each line is 80 characters. Separator: use comma or space or both except as noted. Do not use tab.

Steady State Format

Convective: Format for data points file containing values for h and T. Each line represents a different point.

$$X_1, \ Y_1, \ Z_1, \ h_1, \ T_1$$
$$X_2, \ Y_2, \ Z_2, \ h_2, \ T_2$$
$$X_3, \ Y_3, \ Z_3, \ h_3, \ T_3$$
$$X_4, \ Y_4, \ Z_4, \ h_4, \ T_4$$
$$.$$
$$.$$
$$.$$

Heat Flux: Format for data points file containing values for Q. Each line represents a different point.

$$X_1, \ Y_1, \ Z_1, \ Q$$
$$X_2, \ Y_2, \ Z_2, \ Q$$
$$X_3, \ Y_3, \ Z_3, \ Q$$
$$X_4, \ Y_4, \ Z_4, \ Q$$
$$.$$
$$.$$
$$.$$

Transient Format

Data Points file format for each line of a transient data points file is similar to format for a steady state data points file. However, the arrangement of the lines is significant in this way: All the lines reflecting data at one analysis time are grouped together following the analysis time expressed in seconds on a separate line. In the examples below, t_n represents the analysis time for the lines that follow.

Convective Example

$$t_1$$
$$X_1, \ Y_1, \ Z_1, \ h_1, \ T_1$$
$$X_2, \ Y_2, \ Z_2, \ h_2, \ T_2$$
$$X_3, \ Y_3, \ Z_3, \ h_3, \ T_3$$

```
X_4, Y_4, Z_4, h_4, T_4
.
.
.
t_2
X_1, Y_1, Z_1, h_1, T_1
X_2, Y_2, Z_2, h_2, T_2
X_3, Y_3, Z_3, h_3, T_3
X_4, Y_4, Z_4, h_4, T_4
.
.
.
```

If the set of points changes from one analysis time value to the next, you must include the text NEW POINTS following the time value. For instance (continuing from the preceding example) if the third data point is removed at analysis time t_3, then replaced at analysis time t_4, the format must be as follows:

```
t_3 NEW POINTS
X_1, Y_1, Z_1, h_1, T_1
X_2, Y_2, Z_2, h_2, T_2
X_4, Y_4, Z_4, h_4, T_4
.
.
.
t_4 NEW POINTS
X_1, Y_1, Z_1, h_1, T_1
X_2, Y_2, Z_2, h_2, T_2
X_3, Y_3, Z_3, h_3, T_3
X_4, Y_4, Z_4, h_4, T_4
.
.
.
```

Heat Flux Examples

```
t_1
X_1, Y_1, Z_1, Q_1
X_2, Y_2, Z_2, Q_2
X_3, Y_3, Z_3, Q_3
X_4, Y_4, Z_4, Q_4
.
.
.
t_2
X_1, Y_1, Z_1, Q_1
X_2, Y_2, Z_2, Q_2
X_3, Y_3, Z_3, Q_3
X_4, Y_4, Z_4, Q_4
```

.

.

.

If the set of points changes from one analysis time value to the next, you must include the text NEW POINTS following the time value. For instance (continuing from the preceding example) if the third data point is removed at analysis time t_3, then replaced at analysis time t_4, the format must be as follows:

```
t₃ NEW POINTS
X₁, Y₁, Z₁, Q₁
X₂, Y₂, Z₂, Q₂
X₄, Y₄, Z₄, Q₄
   .
   .
   .
t₄ NEW POINTS
X₁, Y₁, Z₁, Q₁
X₂, Y₂, Z₂, Q₂
X₃, Y₃, Z₃, Q₃
X₄, Y₄, Z₄, Q₄
   .
   .
   .
```

Multi-layer Shell Elements and Reverse Sides (for H and T Only)

For convective boundary conditions only (not heat flux), you can also format the data points file to include data to be mapped to different layers of multi-layer shell elements, or to the front and/or reverse sides of radiating shell elements. The surface elements selected must be defined as either multi-layer elements (see Multi-layer Elements) or have a related Reverse Sides entity (see Working with Front and Reverse Sides).

In the explanation that follows applies to Front and Reverse Sides sides of radiating shell elements as well as top and bottom layers of multi-layer shell elements. For clarity, only multi-layer elements are addressed. If you substitute *front side* for "top layer", and *Reverse Sides* for "bottom layer", the format requirements are identical.

Based on the line's format, data is mapped to the top layer of a multi-layer element, the bottom layer of a multi-layer element, or both the top and bottom layers. (Separator: use comma or space or both except as noted. Do not use tab.)

Format for data to be mapped to the top layer only:

```
X, Y, Z, h_top, T_top
```

Format for data to be mapped to the bottom layer only (comma is required, space optional):

```
X, Y, Z, , , h_bot, T_bot
```

Format for data to be mapped to both top and bottom layer:

```
X, Y, Z, h_top, T_top, h_bot, T_bot
```

Format for multi-layer shell elements, and reverse sides transient solution follows a pattern similar to that required for standard (single-layer) shell elements.

Axisymmetric Elements

For convective boundary conditions or heat flux, you can also format the data points file to include data to be mapped to different axisymmetric elements. Based on the line's format, the data is mapped to the axisymmetric element (Separator: use comma or space or both except as noted. Do not use tab.) Convective or Heat Flux analysis of axisymmetric models is supported for both steady state, and transient solutions. The format of the data points file for axisymmetric models is similar to the other formats, in that each new line represents a new set of data.

Format for steady state axisymmetric convective data to be mapped (h & T):

$$X_0, \ Y_0, \ h_0, \ T_0$$

Format for steady state axisymmetric heat flux data to be mapped (Q):

$$X_0, \ Y_0, \ Q_0$$

Format for axisymmetric transient solution follows a pattern similar to that required for standard (single-layer) shell elements.

Mapping Output

When Boundary Condition Mapping is executed, the following files are created in the run directory.

File Name	Description
mapping.incl	A TMG Include File referencing the .dat file created for each surface. Select this file when performing the analysis.
incl*.dat	Naming convention for each data file containing the convective Thermal Boundary Condition for one surface in INPF format. Each successive mapping run in the same Run Directory is numbered incrementally as incl1.dat, incl2.dat, incl3.dat etc.
elemt*.unv	Naming convention for each Element Dataset containing the mapped fluid temperatures for one surface. Useful for verification and troubleshooting. Each successive mapping run in the same Run Directory setting is numbered incrementally as elemt1.unv, elemt2.unv, elemt3.unv etc.
elemh*.unv	Naming convention for each Element Dataset containing the mapped heat transfer coefficients for one surface. Useful for verification and troubleshooting. Each successive mapping run in the same Run Directory setting generates a file numbered incrementally as elemh1.unv, elemh2.unv, elemh3.unv etc.
elemq*.unv	Naming convention for each Element Dataset containing the mapped heat flux for one surface. Useful for verification and troubleshooting. Each susuccessiveapping run in the same Run Directory generates a file numbered incrementally as elemq1.unv, elemq2.unv, elemq3.unv etc.

report*.lst	Naming convention for each report file listing details and results of the mapping for one surface.

Pick *File, Import* to import the universal files produced by the mapping (elemh*.unv elemt*. unv and elemq*.unv) for visualization using the Post Processing tools.

To include the results of the mapping in an analysis, select the mapping.incl file using the *Include File* icon on the Study Setup form. This file is automatically identified as an include file. If other include files are needed for a thermal analysis, they must be added manually to the *mapping.incl* file.

Understanding Boundary Condition Mapping

Harmonic Mean Method: Boundary Condition Mapping is based on a weighted sum equation by default. This method is faster, more accurate and more robust than the projection method used in previous versions. First, the algorithm looks for the *N* data points that are closer to the center of gravity of the selected elements. The temperatures and the h coefficients are interpolated from the temperatures and the h coefficients of the *N* data points. The interpolation is based on a weighted sum equation:

$$T_{CG} = \sum_{i=1}^{N} a_i T_i \quad (1)$$

where

$$a_i = \frac{1}{d_i \sum_{k=1}^{N} 1/d_k} \quad (2)$$

and where d_i is the distance between each point and the element center of gravity. In this equation T_{CG} is the temperature at the element center of gravity, and T_i are the temperatures at the *N* closest data points. The algorithm uses 4 points ($N = 4$) by default, and this is generally sufficient for all applications. To change this setting, the Boundary Condition Mapping must be run at least once to create the file input.dat file in the current directory. Add the following 2 lines to that file to specify the method and the order:

Method: harmonic

Order: 6

You must rerun the Boundary Condition Mapping *manually* in a DOS shell to make the change effective (TMG/if/mapping.exe without argument). Running Boundary Condition Mapping from the user interface will overwrite your new settings in the file input.dat.

The projection method: it is still possible to use the projection method instead of the weighted sum equation. To do so, the Boundary Condition Mapping must be run once to create the file input.dat file. Add the following line to that file to specify the method:

Method: project

You must rerun the Boundary Condition Mapping *manually* in a DOS shell to make the change effective (TMG/if/mapping.exe without argument).

With this method, the data points are first projected onto the surface(s) defined by the selected 2-D elements. Projection of the Boundary Condition data onto the surface proceeds as follows:

The software automatically calculates a tolerance for data point mapping based on model geometry and data point locations. Data points outside the tolerance are ignored and listed in *report1.lst*. Data points inside the tolerance are projected as follows:

1. If a data point is normal to the surface of an element that data point is projected onto the surface of the element.

1. Data points that are not normal to an element due to curvature of the surface but lie within the tolerance will be projected onto the edge of the element. This is only true if the element edge is not a free edge.

1. Data points not associated with either an element or an edge but lie within the tolerance distance to a node are projected onto that node. This is true if the node does not lie along a free edge.

1. Data points which are within the tolerance of a free edge are projected onto the extended plane of the nearest element. These data points are not projected onto the element itself.

The convection coefficient and fluid temperature for each element is then determined using triangular interpolation. For each element, the 10 data points closest to the element centroid are identified. A set of three data points is then selected from these based on the following criteria:

- the three points must not be collinear when projected onto the plane of the element

- the area of the triangle formed by these three points must not be close to zero

- the length of any side of the triangle must be greater than zero

- the center of gravity of the element must fall within the triangle formed by the three data points

If the three closest points fail any of the above conditions, a different trio is tested and so the loop continues until all requirements are met. Should no combination of the ten points meet the four tests, the last requirement (center of gravity within the triangle) is dropped and the loop is restarted. The subroutine linearly interpolates (or extrapolates) from the selected three data points to determine the convection coefficient and temperature at the element's center of gravity.

Troubleshooting

Here are some things to check if the mapping does not complete successfully:

1. Units of data points and the structural model must be the same.

2. The Run Directory option of TMG is ignored for the data points file; use the full directory path.

3. The data points file should not include tab characters. If you create this file from Excel, be sure to export as a .prn file.

4. The case of group names should be the same in the input.dat file and in your model (scxtmg.dat file). Otherwise, bcmapping will not find any element in this group.