

Using Boundary Conditions Sets

Use *Sets* to select an existing *Boundary Condition Set* to be applied to the TMG thermal model. Pick ? to see a list of available boundary condition sets, then pick the one you want to use.

You can use boundary conditions sets to perform geometry-based analysis with an I-DEAS TMG thermal model. Loads and restraints applied to part geometry are automatically updated when the part is modified.

[Locate the icon.](#)

You create boundary condition sets in the *Boundary Conditions* task, as follows:

1. Set the *Analysis Type* to Heat Transfer
2. Create *Temperature Restraints* and *Heat Transfer Loads*, which comprise *Heat Fluxes*, *Convection*, *Radiation*, *Heat Sources*, and *Heat Generation*. You can collect many loads into a single load set.

Note that these restraints and loads may be applied to nodes, elements, and solid model geometry; you are not restricted to use only surface coat elements. You may also define them to have a temperature variation or a spatial variation using data surfaces.

3. Pick the *Boundary Conditions* icon to create a boundary condition set by selecting a Temperature Restraint and/or Load Set.

Precedence

TMG Temperature Boundary Conditions take precedence over temperature restraints defined on the same area of the model. Heat transfer loads are always added, and never superseded.

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