

# How TMG Uses Elements and Nodes

## Defining Boundary Conditions on Elements

You create a TMG thermal model in exactly the same way that you create any finite element model in I-DEAS with one important exception:

**All boundary conditions and modeling operations are performed on elements (or geometry that references elements), and never on nodes.**

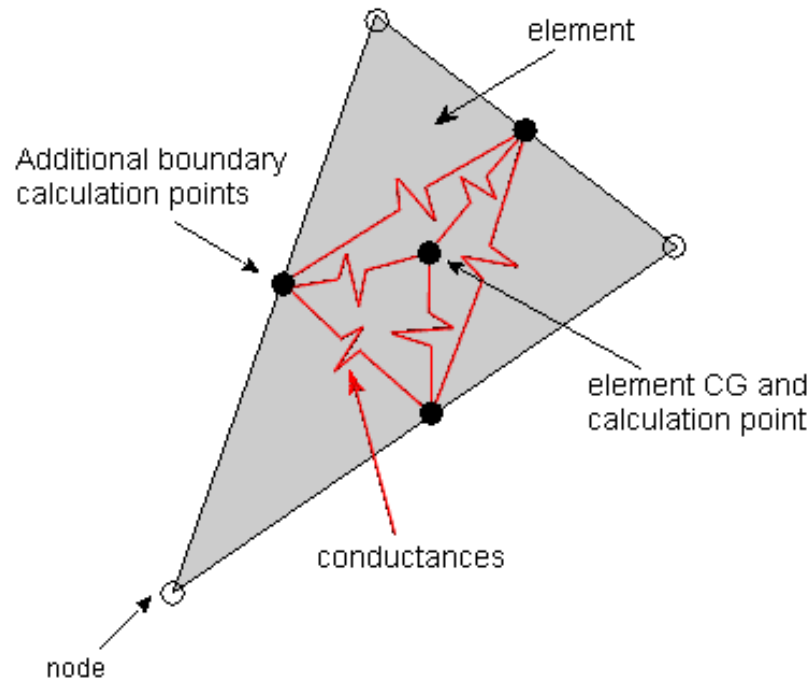
The elements (or geometry referencing elements) are typically grouped to aid selection, but it is the elements not the groups that the solver needs for the analysis.

## How the TMG Solver Uses Elements

Because TMG uses a control volume formulation for the solution, the element's nodes are used only to define the element's geometry. The nodes do not become calculation points in the numerical thermal model as they do with the finite element method.

A calculation point is established at each element's center of gravity (CG). In addition:

- For a 3-D element, an additional calculation point is added at the midpoint of each 2-D face of the element
- For a 2-D element, an additional calculation point is added at the midpoint of each 1-D edge of the element.



A conductance is established using an algorithm which constrains a piecewise-linear element temperature function to satisfy the governing partial differential equation (PDE) for conduction.

The centroidal node is used to compute distributed heat transfer (radiation, convection, and heat flux). Its temperature is computed assuming a piecewise-linear temperature distribution in the element.

The heat flow into the centroidal node is distributed to the boundary calculation points. TMG interpolates the temperature results from the calculation points to the element nodes for post processing and the CG temperature is kept as the element temperature.

## Applying Boundary Conditions to Edges, Surfaces and Solids

Because the nodes are not calculation points, you cannot apply boundary conditions to them. This has direct implications for thermal modeling.

A boundary condition in TMG is actually applied at a calculation point at the element's center of gravity (CG). For example, if you specify a fixed temperature at an element, you are actually fixing the temperature at the CG of the element, and not the temperature of the entire element.

Convection, radiation, thermal couplings and some thermal boundary conditions cannot be defined directly

on 3-D solid elements. In order to apply a boundary condition to one surface of a meshed volume, you must either apply the boundary condition to the free faces of the 3-D elements, or create additional shell elements on the surface of the volume. Similarly, if you want to apply boundary conditions to the edge of a meshed surface, you must either apply the boundary condition directly to the free edges of the 2-D elements, or create additional beam elements on the edge.

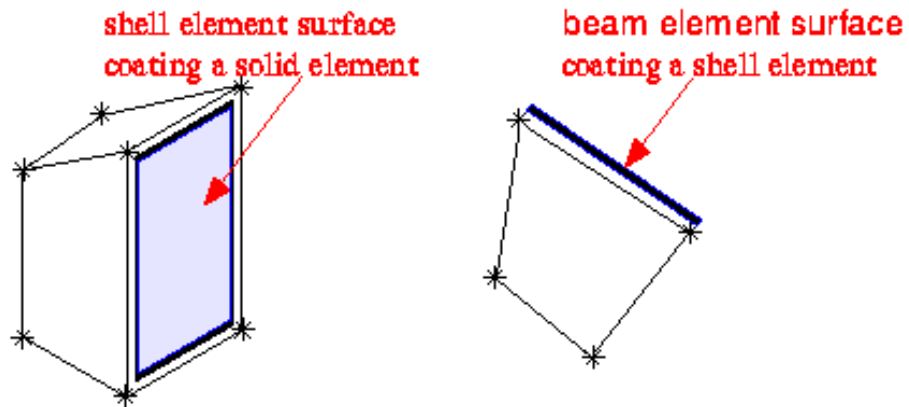
## Using Free Faces and Free Edges

If you choose to define the boundary condition on the free faces of the 3-D solid elements, or the free edges of 2-D shell elements, you must take into consideration the following:

- Not all entities support the selection of Free Faces or Free Edges. Check the online documentation for the specific entity. Generally, if the Free Face or Free edge is selectable when defining the entity, it is supported.
- Material properties of the higher order element (3-D solid or 2-D shell) should match requirements of boundary condition being defined on the lower order (free face or free edge respectively).
- thickness of shell element (some boundary conditions use the thickness to calculate area)
- cannot group free faces or free edges - use geometry groups containing geometry related to the free faces or free edges.
- to select the free faces themselves, use selection filter

## Creating Additional Shell or Beam Elements

If you choose to create additional elements of the appropriate (lower) order, use either the *Define Shell* (or *Beam*) *Mesh* icon or the *Surface Coating* command in the menus. In this way nodes will be shared between the higher and lower order meshes, automatically causing heat to be transferred via conduction. Description of both methods can be found in the *Meshing* task.



### Free or Mapped Shell Meshing

Create a lower dimension element on the boundary (free face or free edge) of the higher dimension element by free meshing the surface of a solid, or the edge of a shell. Apply the boundary condition on the lower dimension element.

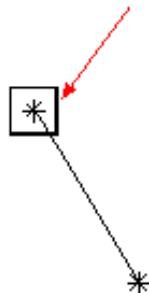
### Surface Coating Command

Create a lower dimension element on the boundary of the higher dimension element using *Surface Coating* in the Meshing task. Coat the surface of a solid, or the edge of a shell and apply the boundary condition on the lower dimension element.

### End-Coating Beam Elements

You can accomplish a similar effect on a beam element by creating a lumped mass on the end node of a beam. Create the lumped mass using the *Element* icon in the meshing task.

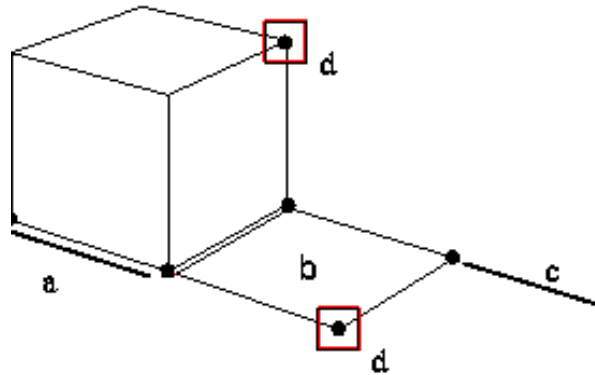
**lump mass element  
on the end of a beam**



## Improper Element Mesh

The following element meshes are improper and will not conduct heat between elements.

- a. Beam connected to the edge of a 3D solid element. Beams must be connected to an edge of a shell or to another beam.
- b. Shell connected along an edge to a 3D solid element. All nodes on a shell must be connected to all nodes on the face of a solid or connected along their edge to another shell.
- c. Beam connected at one end to a shell or solid element. Beams must be connected to an edge of a shell or to another beam.
- d. Lumped mass connected to the corner of a shell or solid element. Lumped mass elements must be placed on the end of beam elements.



## Properties for Coated Elements

As for all elements, these surface coated elements can be used to model physical characteristics. However, surface coated elements typically have no physical significance but rather are only created to apply thermal entities to the geometric model. In this case, you must ensure that they do not create an unintended heat path.

Create these elements using material properties with zero conductivity and zero specific heat. For shell elements use a physical property table with zero values for the shell thickness.

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