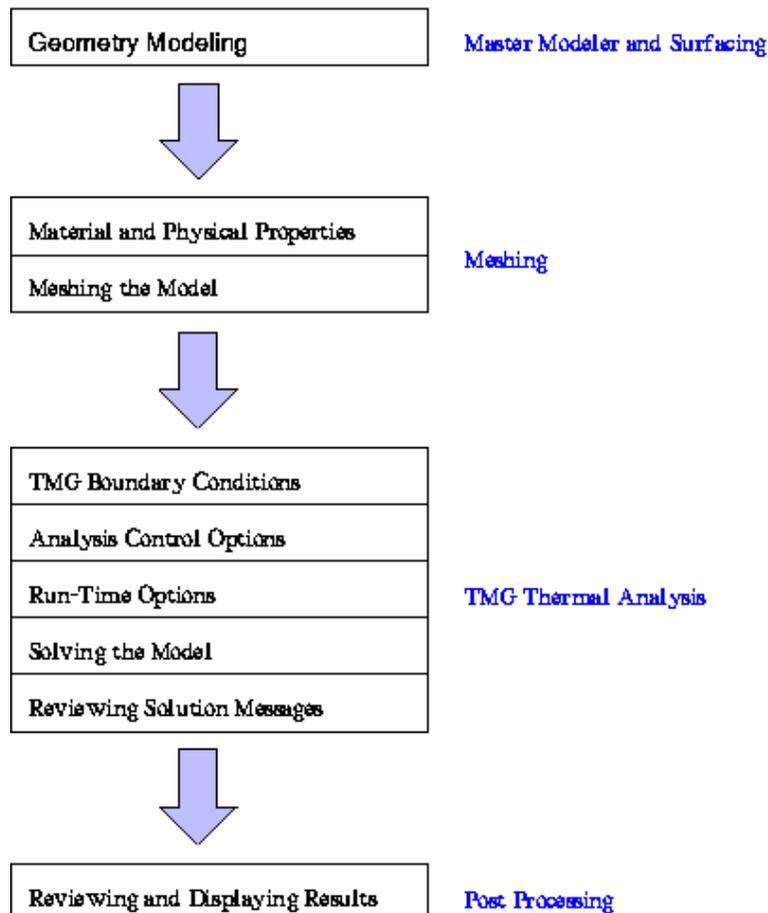# Overview of TMG Modeling Process

Once you have read this article, and perhaps a few other articles relevant to your specific application, try out TMG Thermal Analysis for yourself. Several **online tutorials** will guide you step-by-step through the entire modeling process. You can access the same page from the I-DEAS icon panel by picking *Help*, then *Tutorials* and looking under the heading *Simulation* to find the link to TMG Projects.

The basic procedure for modeling heat transfer in any TMG model, independent of its complexity, is the same. The most important rule to remember with all analysis procedures is to start with as simple a model as possible and add detail if required. In practice this means that you should initially:

- Omit less significant parts or features in an assembly

- Simplify geometric representations

- Use a coarse element mesh

- Check solution results using basic heat transfer calculations

Section meshing provides a powerful tool set for idealizing and simplifying the geometry of your model. See **Defining TMG Entities** for more info.

Once you are confident that the initial thermal model is reasonable then add detail and use a finer mesh if necessary. The basic I-DEAS TMG modeling process is as follows:

| Geometry Modeling | Master Modeler and Surfacing |

| Material and Physical Properties / Meshing the Model | Meshing |

| TMG Boundary Conditions / Analysis Control Options / Run-Time Options / Solving the Model / Reviewing Solution Messages | TMG Thermal Analysis |

| Reviewing and Displaying Results | Post Processing |

As with all simulation in I-DEAS the process begins with modeling the geometry of the components and assembly. The Modeling and Surfacing simulation tasks provide excellent tools for modeling any part or assembly. The *Meshing* simulation task creates the thermal model's finite element mesh and defines associated material and physical properties. The *TMG Thermal Analysis* simulation task defines additional heat paths, thermal heat loads, fixed temperatures, radiative sources, orbits and fluid flow networks. In addition, *TMG Thermal Analysis* is used to set thermal simulation options and to launch the solution process. The *Post-Processing* simulation task graphically displays results and creates graphs or tabular data to communicate your results to the design team.

Data Structure of the Model

The model file is the basis for all modeling.

- Within the model file, you create, adapt or import the model's solid geometry, which is comprised of a heirarchy of vertices, edges, surfaces and volumes.

- Within the model file, you create one or more Finite Element Models (FEM) to contain all simulation data.

- Within the FEM, you create (or import) a mesh of 1-D, 2-D or 3-D elements, with associated material and physical properties.

- Also within the FEM, you define the thermal model using TMG's thermal modeling tools, solve the thermal model and post process the results.

Creating the Thermal Model

TMG Thermal Analysis uses two types of tools to define a thermal model: entity creation tools and parameter definition tools.

- Entity creation tools allow you to specify boundary conditions and other entities that characterize a specific portion of the model. TMG entities are always applied to elements, either directly by selecting the elements, or indirectly by selecting a group. The group may contain elements or geometry that references elements.

- Parameter definition tools allow you to set global values and specify options that govern the entire model. They are always applied to the FEM as a whole, never to specific elements or geometry.

Modeling Conduction

TMG Thermal Analysis uses a finite volume formulation for conduction.

Conduction is always modeled between elements that share nodes, providing the following conditions are met:

- Thermal conductivity (material property) for the elements must be defined as non-zero.

- Shell elements must have non-zero thickness (physical property); beam elements must have a beam section defined.

Modeling Convection and Fluid Flow

TMG provides a tool to create forced convection couplings between a fluid network and a solid model. It automatically calculates the heat transfer coefficient for each fluid element using a correlation you select or define.

Similarly, TMG can automatically calculate the heat transfer coefficient for natural convection from the solid model to a fluid, using a correlation you select or define. You may use either of two methods to simulate free convection:

- The *Convect to Ambient* option simulates heat transfer between specified elements and an ambient fluid you define.

- Convection to the fluid elements in a duct flow network simulates heat transfer within the model. This is the preferred method for low velocity "chimney effects".

Duct flow modeling in TMG uses a one-dimensional flow network represented by curves, beam sections and beam elements. In the duct network, curves meshed with beam elements are used to define the center of the duct path. The cross section and hydraulic diameter of the duct are defined by applying beam sections to these elements. Additional duct characteristics, fans/pumps and inlets/outlets are defined as boundary conditions on beam elements.

Modeling Radiation

TMG's radiation simulation capability is based on view factors (also called shape factors or form factors).

TMG calculates black body view factors between all radiation elements. It combines these factors with radiative surface properties and calculates a radiative conductance.

Only surface elements (shells, beams with non-zero perimeters, lumped masses with non-zero areas) can radiate in TMG.

Shadowing checks generally dominate view factor computational time. For every element, all others must be checked for shadowing. To minimize shadowing checks, request that view factors only be calculated between elements which you know might have a view of each other (ie: an enclosure).

Radiative sources can also be defined in TMG which are assumed to emit diffuse solar spectrum flux. View factors from the source to illuminated elements are then calculated. (The sun is considered a distant point source.) Gray body view factors are also computed (Gebhart's method) using the element's solar spectrum properties (absorptivity, reflectivity and transmissivity). Ray-tracing is automatically executed to model the specular component of reflection.

TMG uses ray-tracing to model radiative heat transfer involving specularly reflecting, refracting and transmitting surfaces. When TMG computes black body view factors, it launches rays from elements and traces them through subsequent reflections until the ray is extinguished due to:

- the ray impinging a fully diffuse element.

- the ray's magnitude having been reduced to less than 0.1% of its original value.

- the ray having been traced through 100 reflections.

An orbit definition tool allows you to calculate solar, planetary and albedo heating based on spacecraft orientation, solar heat loads, as well as spacecraft, planet and orbit geometry. Orbits can be displayed.

Electronic System Cooling and TMG Thermal Analysis

Most items created in TMG are fully compatible with I-DEAS Electronic System Cooling (ESC), a powerful Computational Fluid Dynamics (CFD) tool.

A combined ESC/TMG simulation allows:

- Element-based finite volume unstructured solving methods.

- 3D fluid velocity, temperature and pressure prediction.

- Turbulence modeling.

Getting Results

TMG allows you to select three separate solution modes: Steady-State, Transient or No Temperature Calculations. Options for these three modes are found on the *Analysis Control* form.

Transient thermal models are solved by integrating over discrete time intervals. These time steps are simply the mesh of the time domain. Large or rapid temperature changes require a finer time step. Solution time is related to time step. Numerical stability is often related to the minimum element time constant, RC_{min}. (RC_{min} is analogous to an electrical RC circuit time constant.)

After every solution, take the time to verify the convergence of the model. Review the message file "MSGF.dat" for global heat balance, mass balance for fluid networks, any undesirable warnings and check the view factor sums.

Modifying Your Model

Changes in geometry are easily accomplished by using the history tree in the Modeling task. A part update applies the change and regenerates the mesh. Entities defined with geometry groups are preserved and updated as well. Finite element-based boundary conditions must be re-defined on the newly generated elements.

The entity manager provides a quick and convenient method for accessing and modifying all thermal entities. Select the entity type from the icon panel and then use the entity manager to modify, delete, copy, select new elements, graphically display elements and list the entity's elements and properties. If you assign descriptive names to the entities when you create them it is much easier to identify the entities in the manage list.