



Modeling Duct Flow in Previous Models

Should I Use the Old Method or the New Method?

The Master Series 6 release of I-DEAS TMG provided a completely new and greatly improved set of tools for modeling duct flow. If you are creating a new model, see *Creating A Duct Network* for complete instructions. However, if you only want to maintain or modify a duct flow model created with a release of TMG *prior to Master Series 6* you must use the older set of tools documented in this article.

Note that the two types of duct modeling are not compatible; entities created with the new method, Duct Networks, cannot be mixed with entities created with the older method, Duct Fluid Flow, nor vice versa. As an aide in distinguishing the entities, the TMG icon panel contains only Duct Network icons; Duct Fluid Flow commands are in the TMG menus under Duct Fluid Flow. Setting ambient conditions for the older method is the same as for the new method.

How the Old Method Works

You can use I-DEAS TMG to simulate fluid flow and heat transfer in a network of one dimensional flow channels. You first construct a geometric model of the flow channels and assign fluid properties, then create convection couplings from surface elements to the fluid elements. TMG formulates a finite difference hydraulic model of the flow network, and solves it in parallel with the thermal model. The flow resistances, advective heat transfer, and convective conductances are calculated automatically and dynamically from the model data.

Modeling Duct Flow in Pre-MS6 Models

1. Define the fluid material properties (use TMG Gas or TMG Liquid; see *Materials*).
2. Generate a mesh of *Pipe* elements to model the flow channels.
3. Create *Damper* elements to represent fans or pumps in the system, and use *Duct Flow Boundary Conditions* to define their characteristics.
4. Create *Gap* elements to model any additional flow resistances (e.g. inlet or exhaust); use *Duct Flow Resistance Modifier* to define their value.
5. Create *Rigid Link* elements to characterize the shape of the flow channels.
6. Specify *Ambient Conditions* if you need to change the default settings (see *Defining Ambient Conditions*).
7. Use *Forced Convection* to create convective couplings from shell elements to the fluid elements.

Meshing a Flow Channel in Pre-MS6 Models

You can use *Beam Meshing* to generate a mesh of fluid elements to model a flow

channel. The procedure is as follows:

1. Create curves which describe the flow paths. These should lie roughly along the centerline of the flow channels.
2. Attach the curves to the part.
3. Define a Beam Mesh on the curves; select the desired fluid element type (normally Pipe elements), and specify a mesh density.
4. On the *Beam options* form, pick *Beam Direction* to display arrows indicating flow direction for each curve. Clicking on an arrow reverses its direction.
5. Generate the Beam Mesh.

You can also model flow channels using direction creation of nodes and elements.

Fluid Elements in Pre-MS6 Models

Three types of elements model a straight flow channel or duct:

- *Pipe* elements represent the default duct.
- A *Damper* element is used to designate a fan or pump, and must have a flow boundary condition specified for it.
- A *Gap* element is used to model a specific (length-independent) flow resistance, and must have a defined resistance value.

A fluid element represents the fluid in the flow channels, and not the walls of the channel. Each fluid element has three physical attributes:

- a shape
- a direction
- material properties

Creating Flow Sections in Pre-MS6 Models

The shape of a fluid element is determined by its length and its Flow Sections. A fluid element must have flow sections defined at both ends. These can be different, in which case the element is a nozzle or a diffuser. The fluid element's shape is used in determining its flow resistances, flow velocity, and thermal capacitance.

In order to ensure flow section consistency at the junction of two fluid elements, flow sections are not assigned directly to an element. Rather, a *Flow Section* element is used to define the flow section at a node in the network: *Rigid Link* elements are used for this purpose (see below). The procedure for creating these elements is as follows:

1. Select *Element* from the icon panel in the Meshing task.
2. Select the *Element* to be *Other*.

3. Select the *Element Family* to be *Rigid*, and the *Element Type* to be the three-branch entity.
4. Assign the *Material* to be that of the fluid in the flow channel (this must be a *Fluid/Thermal - Liquid* or *Fluid/Thermal - Gas* material).
5. Select *OK*
6. Pick the node in the flow network where the flow section is to be defined.
7. Hold down the Shift key, then select a series of up to 32 nodes which define or circumscribe the flow section

The *Duct Flow Section Override* entity overrides the flow section shape defined by the Rigid Link element.

Using Directional Propagation in Pre-MS6 Models

In order to simplify modeling of constant section flow channels, TMG provides *Directional Propagation* of flow sections. The "direction" of a fluid element is defined by its node connectivity; that is, it points from the first node to the second node. You can display a fluid element's direction by toggling the *Element First Edge Switch* under *Display Options*. Flow sections defined using a Rigid Link element are propagated along the fluid elements in this direction to all succeeding nodes in the network, until a new Rigid Link flow section is encountered. Therefore, for a constant section flow channel, you need only create a single Rigid Link flow section element at the first node in the chain, ensuring that all of the fluid elements point away from that node.

Because of this, TMG requires that all of the fluid elements in a single branch of the flow network have the same direction. If this is not the case, the model will not be solved.

Assigning Fluid Properties in Pre-MS6 Models

TMG requires that all of the elements in a single flow network have the same material properties. Therefore, material properties are defined at the inlet node(s) of the system and assigned to all connected elements by directional propagation. TMG always assigns *Ambient Conditions* to all free ends of a flow network; that is, at any node which is connected to only one fluid element, and which does not have a Rigid Link flow section attached to it. In such cases, the fluid properties for all elements in the flow network are automatically those specified for the ambient fluid (see *Defining Ambient Conditions*).

To model a flow system which does not use the ambient fluid, you must:

1. Create Rigid Link flow section elements at all free ends of the network.
2. Define the Material for the Rigid Links to be the desired fluid; use *Fluid/Thermal - Liquid* or *Fluid/Thermal - Gas* material types.
3. Fix the total pressure at the Rigid Link elements using *Duct Pressure Boundary Conditions*
4. Fix the temperature at the Rigid Link elements using *Temperature Boundary Conditions*

To model a closed loop system, simply define the appropriate Material for one of the Rigid Link flow section elements, and fix its pressure.

Boundary Layer Modeling in Pre-MS6 Models

In calculating *Forced Convection* heat transfer coefficients, TMG accounts for the buildup of the boundary layer in the flow channel. By default, a Rigid Link flow section element triggers the start of a new boundary layer. You can prevent this by selecting the Rigid Link element in a *Boundary Layer Continuation* entity.

For example, to model a continuously divergent flow channel, you must create Rigid Link flow section elements at every node in the chain. By default, TMG will start a new boundary layer at each one of these. To correctly model this situation, you must identify all Rigid Link elements other than the one at the inlet as *Boundary Layer Continuation* elements.

Buoyancy Modeling in Pre-MS6 Models

In solving the flow model, TMG computes density changes in the fluid using the ideal gas law. The resulting buoyancy forces in the system are also modeled by calculating a pressure rise across an element equal to

$$F_b = g \times (\rho - \rho_{amb}) \times \Delta height$$

This enables the modeling of natural convection using flow networks. It is most effective for "chimney" type situations, where the heated gas passes through a well-defined flow channel.

You can deactivate buoyancy modeling for selected elements using the *Element Buoyancy Switch*.

Solving the Flow Modeling Pre-MS6 Models

When solving a model that includes flow simulation, you should

1. Under *Flow Options*, set the *Relaxation factor* to 0.3 or less.
2. Under *Flow Options*, specify a *Pressure Convergence Criterion*; generally a value of 0.01 gives good results (see *Setting Flow Options*).

Recovering Duct Flow Results in Pre-MS6 Models

You can recover the following types of flow model results for post-processing in I-DEAS (see *Setting Solution Data*):

- Velocity
- Reynold's Number
- Pressure

Use *Results Options* to activate these.

You can also get a comprehensive Report File output of the flow model data. Use the *Element Heat Balance* output on the Element Reporting form, and select the desired fluid elements (see the article *Element Reports* for details).

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