Contents | Find

Creating a Duct Network

Steps to Create a Duct Network

1. In the *Master Modeler* task, create curves defining the center lines of the ducts.

2. *Attach* these curves to your part.

3. Create one or more flow sections.

4. In the *Meshing* task, define and generate a fluid beam mesh on the curves. Under *Beam Options* select a flow section you created.

5. Create *Fan/Pump* and *Inlet/Outlet* entities if appropriate to your model.

Locate the icon.

6. Use *Duct Properties* to override the duct characteristics at a given point.

Locate the icon.

7. Set up *Ambient Conditions*.

Locate the icon.

8. Use *Forced Convection* or *Free Convection* to create convective couplings from shell elements to the fluid elements.

Locate the icon.

9. For hard to converge model you may need to adjust the *Flow Options* of the *Solver Control*.

Locate the icon.

Defining Fluid Elements for a Duct Network

Use *Beam Meshing* to generate a mesh of fluid elements to model a duct network. Parabolic and linear beams are supported, as are curved beams. A fluid element represents the fluid in the duct, and not the walls of the duct. Each fluid element has four characteristics:

- position

- shape

- material properties

- a duct friction value (by default smooth)

Curved Beam elements are also supported. Head losses and changes in heat transfer coefficients as the fluid passes around bends are automatically calculated based on the radius of curvature and other geometrical information. See the article *Friction Factors and Heat Coefficients in Rough and Curved Ducts* for details.

Position

Beam elements are normally defined on curves or edges. The orientation and location of this geometry defines the centerline of the duct segment. The position of the fluid elements in releation to model geometry and coordinate systems is used to calculate convection and buoyancy.

Shape (Flow Section)

The shape of a fluid element is determined by its length, its curvature and its Flow Section. TMG uses the fluid element's shape to determine its flow resistances, flow velocity, and thermal capacitance.

Ducts can have any cross section. The cross section must first be created as a beam section, then applied to the element. In the *Beam Sections* task, use *Standard Shape*, *Key-In* or *General Sections* to define one or more beam sections. TMG uses only the hydraulic diameter and cross-sectional area of the flow section in its calculations.

The *Flow Section* **represents the fluid in the duct, not the duct itself**. Therefore, specify the beam section as a **solid** section with outside dimensions equal to the desired inside dimensions for the duct. **Do not** use *Pipe* beams, *Box* beams, or any section with a hole in it.

For example, to model a rectangular duct with inside dimensions of 20 x 60 cm, create a *solid* rectangular beam with these values as outside dimensions.

When using *Key-In Section* to create a flow section, only *Area* and *Perimeter* are required (or supported).

See Also

- **Creating a Beam Cross Section Using Standard Shapes**

- **Creating a Beam Cross Section Using General Shapes**

To apply a beam section to fluid beam elements, use the *Define Beam Mesh* icon. Under *Beam Options* select a beam section (*Flow Section*) you created.

You can define different *Flow Sections* for different elements. During the solve, the software models flow effects for abrupt changes in *Flow Section* where two beams with differing sections meet. With curved and parabolic beam elements, TMG automatically applies an appropriate head loss. You can also specify a tapering beam of any length.

A *Duct Flow Section Override* entity overrides the flow section shape defined by the element's Beam Section. Taper Across Branch overrides all.

Material

The material for the fluid elements can be any *Fluid/Thermal - Liquid* or *Fluid/Thermal - Gas* material (see *Materials*).

You can apply different fluids to different branches of your duct network. To model mixing (two or more branches with different fluids connecting to a single duct), you have to define the property of each fluid as well as the mixture.

Friction

By default, smooth pipe values for duct friction are automatically applied to the duct walls during solve. You can override this value by using *Duct Properties* to define friction.

Viewing the Duct Network

After you have created the beam elements for a duct network, it is a good practice to inspect the network. First select the TMG Fluid elements (using Filter, Elements, Attributes, Material, [fluid material], Pick Only), then pick *Display Selected*. The duct network should appear in the graphics region. If any element has the wrong material, it will not appear.

Ambient Conditions

Use *Ambient Conditions* to specify the default characteristics of the free ends of the duct network. In SI, default values for ambient conditions applicable to duct flow are:

- Static Pressure: 101.351 mN / mm^2

- Ambient Temperature: 20°C

- Gravitational Acceleration: 9.81 m / sec^2

- Gravity vector: 0,0,1000 (X,Y,Z)

To model a flow system which does not use *Ambient Conditions* at its free end(s), you must create an Inlet/Outlet entity to specify temperature and pressure for each free end of the duct network. You should still make sure that the *Ambient Conditions*, especially the Gravity Vector, apply to your model.

To model a **closed loop** system, simply define pressure of one of the elements or nodes in the loop.

Boundary Layer Modeling

In calculating *Forced Convection* heat transfer coefficients, TMG automatically accounts for the buildup of the boundary layer in the flow channel. By default, TMG models these at the free end of a duct network, at a fan and at an abrupt change in flow section. You can add additional start points with the *Boundary Layer Start* option on Duct Properties.

Recovering Duct Flow Results

You can recover three types of flow model results for post-processing in I-DEAS (see *Setting Results Options*):

- Velocity

- Reynold's Number

- Pressure

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 *Element Reports* for details).

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