Autodesk® Moldflow® Insight 2012

# AMI Modeling the feed system

Autodesk

Revision 1, 22 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

# Contents


Chapter 1 **Feed system** . . . . . 1

Feed system . . . . . 1

Deleting cavities, runners, gates or sprue . . . . . 1

Part/runner combinations . . . . . 2

Selecting the number of cavities . . . . . 3

Runner System Wizard . . . . . 4

Runner System Wizard . . . . . 4

Runner System Wizard . . . . . 5

Chapter 2 **Occurrence numbers** . . . . . 6

Occurrence numbers . . . . . 7

Editing occurrence number properties . . . . . 7

Occurrence numbers . . . . . 8

Occurrence number diagnostic dialog . . . . . 9

Using occurrence numbers to reduce model size . . . . . 9

Chapter 3 **Sprue** . . . . . 11

Sprue. . . . . . . . . . 11
Creating a sprue manually. . . . . . . . . . 11
Creating a sprue automatically. . . . . . . . . . 12
Changing sprue properties. . . . . . . . . . 12
Sprue. . . . . . . . . . 13
Edit Taper Section dialog. . . . . . . . . . 13
Cold sprue dialog—Sprue Properties tab. . . . . . . . . . 13
Cold sprue dialog—Mold Properties tab. . . . . . . . . . 13
Cold sprue dialog - Mold Temperature Profile. . . . . . . . . . 14
Cold sprue dialog—Overmolding Component tab. . . . . . . . . . 14
Cross-Sectional Dimensions dialog. . . . . . . . . . 14
Hot sprue. . . . . . . . . . 14
Hot sprue. . . . . . . . . . 14
Hot sprue. . . . . . . . . . 16

Chapter 4 Gates. . . . . . . . . . 18
Valve gates. . . . . . . . . . 18
Manually trimmed gates. . . . . . . . . . 20
Automatically trimmed gates. . . . . . . . . . 29
Hot gates. . . . . . . . . . 32
Modeling gates for 3D Fill+Pack analysis. . . . . . . . . . 33
Setting a gate contact diameter. . . . . . . . . . 34

Chapter 8 Runners. . . . . . . . . . 36
Runners. . . . . . . . . . 36
Creating a runner system manually. . . . . . . . . . 36
Converting runners from triangles to beams. . . . . . . . . . 37
Displaying the interior of an annular runner. . . . . . . . . . 38
Runners. . . . . . . . . . 38
Simplify Elements to beams dialog. . . . . . . . . . 38
Cold runner dialog—Runner Properties tab. . . . . . . . . . 39
Cold runner dialog—Mold Surface Temperature tab. . . . . . . . . . 39
Cold runner dialog—Mold Temperature Profile tab. . . . . . . . . . 39
Cold runner dialog—Mold Properties tab. . . . . . . . . . 39
Cold runner dialog—Overmolding Component tab. . . . . . . . . . 39
Cold runner (3D) dialog—Mold Properties tab. . . . . . . . . . 39

Cold runner (3D) dialog—Overmolding Component tab. . . . . . . . . . . . . 40
Cold runner (3D) dialog—Part Surface Properties tab. . . . . . . . . . . . . . . 40
Runner properties. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 40
Runner properties. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 41
Designing the runner layout. . . . . . . . . . . . . . . . . . . . . . . . . . . . . 42
Runner aspect ratios. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 43
Runner sizes. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 43
Runner sections. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 44
Runner cross-sections - Shape. . . . . . . . . . . . . . . . . . . . . . . . . . . . 44
Runner cross-sections - Diameter. . . . . . . . . . . . . . . . . . . . . . . . . . 45
Hot runners. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 46
Hot runners. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 46
Hot runners. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 47
Hot runners - heated runners. . . . . . . . . . . . . . . . . . . . . . . . . . . . . 48
Hot runners - insulated runners. . . . . . . . . . . . . . . . . . . . . . . . . . . 49
Runner systems for single cavity molds. . . . . . . . . . . . . . . . . . . . . . . . 50
Unbalanced flow in single cavity molds. . . . . . . . . . . . . . . . . . . . . . 50
Overpacking in a single cavity mold. . . . . . . . . . . . . . . . . . . . . . . . 51
Avoiding hesitation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 52
Runner systems for multi-cavity and family molds. . . . . . . . . . . . . . . . . . 53
Unbalanced flow in family molds. . . . . . . . . . . . . . . . . . . . . . . . . . 53
Unbalanced flow in multi-cavity molds. . . . . . . . . . . . . . . . . . . . . . . 54
Overpacking in multi-cavity and family molds. . . . . . . . . . . . . . . . . . . 55
Hesitation at the gates of multi-cavity and family molds. . . . . . . . . . . . . 56

# Feed system

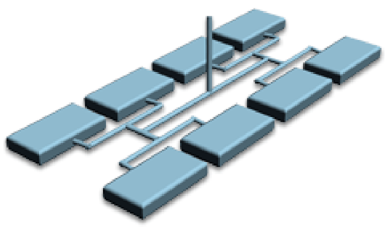
# 1

There are a number of factors to consider when designing the feed system, including the gate locations, the number of cavities, the shape of the runner system components, and flow balance.

The first step in designing the feed system is to determine the gate locations for each part in the mold. The rest of the components will fit into place depending upon each gate location. The objective when designing the feed system is to design it with balanced flow so that each part in the mold fills at the same rate. The creation of a well-balanced feed system requires careful consideration the following elements:

- Single-cavity, multi-cavity, or family mold
- Cavity layout
- Location of the sprue
- Runner system layout
- Shape of the sprue, runners, and gates

In general, make runners as short as possible, with the lowest possible shot weight. In the following diagram, the flow length for every part is the same. This is a naturally balanced runner system.



## Feed system

There are a number of factors to consider when designing the feed system, including the gate locations, the number of cavities, the shape of the runner system components, and flow balance.

### Deleting cavities, runners, gates or sprue

1 Click **Geometry tab > Selection panel > Select**.

2 Click on the cavity or on the section of the runner systems that you want to delete.

**TIP:** To delete more than one entity, hold down the **Ctrl** key as you select each item.

You can also use **Edit > Select** to select all runners; gates, cavities etc.

3 Click **Geometry > Utilities panel > Delete**.

The selected entities are deleted.

**NOTE:** Body faces or bodies of a CAD assembly cannot be deleted independantly.

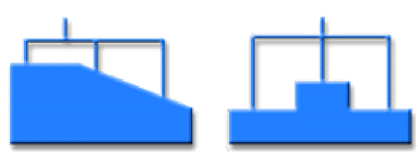
## Part/runner combinations

The following describe the most commonly used part/runner combinations in an injection molding machine.

#### Direct Injection

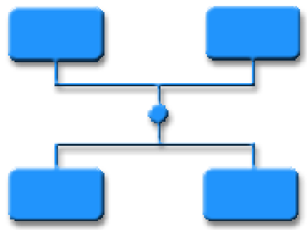
This figure represents a simple part-runner system combination, and must be modeled completely unless there are symmetries in the flow path for which occurrence numbers can be specified.

#### Injection into a multi-gated part



The left half of the picture illustrates a multi-gated cavity which is asymmetrical, thus the whole part and runner system must be modeled. However if the part was symmetrical as in the right hand picture then only one “repeating unit” or flow path of the part need be modeled. Occurrence numbers can be specified to identify how many times the flow paths are repeated.

#### Injection into a multi-cavity part



This picture illustrates a multi-cavity layout where, symmetry and occurrence numbers can be used to reduce the amount of modeling work required.

## Selecting the number of cavities

A number of factors should be considered when deciding how many cavities can be filled from a runner system.

The number of cavities which can be filled from any given runner system depends on the following:

- Machine size (available clamp force)
- Available shot volume
- Available production time
- Required product quantity
- Shape and size of the moldings
- Mold costs

The number of cavities filled from a particular runner system can affect the ease with which the program arrives at a runner balancing solution. The possible combinations of pressure drops in the runner system itself and in filling the cavity volumes can rapidly increase the complexity of the balancing problem.

Following are simple formulas for determining the number of cavities. Use the minimum value derived from these formulae.

#### Production schedule

If the dimensional tolerance of the part is not very critical and a large number of moldings are required, multi-cavity molds are preferred.

Number of cavities = LKKctm

where:

- L = number of parts in the lot
- K = the reject factor, expressed as K = 1/(1-reject rate)

- tc = cycle time to produce a single set of parts
- tm = available time to supply a lot of parts

#### Shot capacity

Number of cavities = SW

where,

- S = 80% of machine capacity
- W = part weight

#### Plasticizing capacity

Number of cavities = PX⊠W

where,

- P = machine plasticizing capacity
- X = estimated number of shots per minute
- W = part weight

# Runner System Wizard

The **Runner System Wizard** enables you define basic runners, sprues, and gates, to quickly produce a complete feed system.

Follow the prompts as the Runner System Wizard takes you through the steps. When you are familiar with the process, you can create your own runner system manually if you prefer.

## Runner System Wizard

You can produce a complete feed system quickly using the **Runner System Wizard**.

### Creating a runner system using the Runner System Wizard

1. Click **Geometry tab > Create panel > Runner System**.
2. Enter X and Y coordinates, or click **Center of Mold** to position the sprue.
3. Choose whether you want to use a hot runner system, and enter a value for the **Parting plane Z**. Click **Next** to take you to the next page of the Wizard.
4. Enter values for the diameter, length and included angle for the **Sprue**, the **Runners** and the **Drops**. Click **Next** to proceed to the next page.
5. Lastly, enter values for the start and end diameters, and the length of the **Gates**.
6. Click **Finish** to save and close the dialog.

## Runner System Wizard

The **Runner System Wizard** guides you through the process of creating a feed system.

Use this wizard to define the sprue, runners, drops, and gates to produce a complete feed system.

To access this wizard, click **Geometry tab > Create panel > Runner System**.

### Runner System Wizard—Layout

The **Layout - Page 1 of 3** page is used to specify the sprue position and the type of runner system, based on one or more injection locations set on the model.

---

**TIP:** To change the units of measure, click **> Options**, and select an alternative from the **General** tab.

---

### Runner System Wizard—Sprue/Runners/Drops

The **Sprue/Runners/Drops - Page 2 of 3** page is used to specify geometry information for the sprue, runners, and drops of the feed system.

---

**TIP:** To change the units of measure, click **> Options**, and select an alternative from the **General** tab.

---

### Runner System Wizard—Gates

The **Gates - Page 3 of 3** page is used to specify geometry information for the gates of the feed system.

---

**TIP:** To create a banana gate using the **Runner System Wizard**, specify a side gate by **Length**.

---

# Occurrence numbers

# 2

Occurrence numbers are typically used to simplify the amount of modeling required for a Fill+Pack analysis by specifying the number of times that a given flow path is repeated in a multiple cavity tool.

For example, for a naturally balanced 8–cavity tool, only one part and the feed system must be modeled.

**CAUTION:** If you intend to proceed to a Cool, Warp or Stress analysis, you cannot use occurrence numbers to represent a portion of a part.You must model the entire part.

Occurrence numbers are a useful shortcut that can be used to reduce the amount of modeling and analysis time required for models with symmetrical flow paths, especially in multi-cavity models, where only one cavity needs to be modeled and the other identical cavities can be referenced in the analysis by occurrence numbers. Identical flow paths are defined to have symmetrical physical geometries and identical volumes of plastic flowing through them.

The figure below shows a naturally balanced 8-cavity tool. Only the geometry shown in red must be modeled. The remaining cavities are accounted for by occurrence numbers.

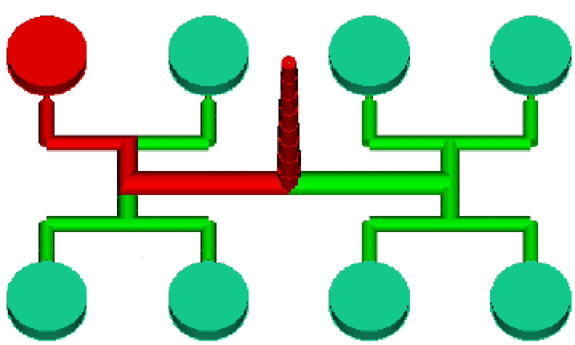


**Figure 1: Occurrence numbers in a naturally balanced 8-cavity tool**

Occurrence numbers in the figure above are defined as follows:

- The sprue is unique and therefore has an occurrence number of 1.
- The primary runner has an occurrence of 2 because the flow path splits in 2 at the base of the sprue.
- The secondary runners have an occurrence of 4, and the remaining part of the flow path has an occurrence of 8.

### Symmetrical Flow Paths

Before you consider modeling simple parts using occurrence numbers, it is helpful to understand the concept of “symmetrical flow paths”. Flow paths are said to be symmetrical when the physical geometry of the flow path is symmetrical and when the volume of plastic flowing through the paths are identical.

The figure below shows an artificially balanced 8-cavity tool. This runner system contains two distinct flow paths—respectively represented in red and blue—therefore the occurrence numbers differ from the first example.

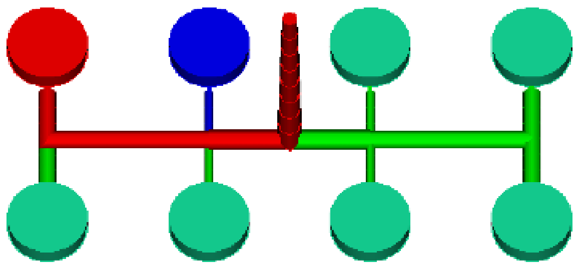


**Figure 2: Occurrence numbers in an artificially balanced 8-cavity tool**

Occurrence numbers for the figure above are defined as follows:

- The sprue—in red—is unique and therefore has an occurrence number of 1.
- The primary runner—in red—has an occurrence of 2 because the flow path splits in 2 at the base of the sprue.
- The secondary runners, gates and parts—represented in blue—have an occurrence of 4.

### Specifying occurrence numbers

By default, all surfaces and elements in the model are considered to have an occurrence number of 1. Using **Geometry tab > Properties panel > Edit**, you can specify an occurrence number other than 1, for example, for a model of a multi-cavity mold. All elements containing the same properties as the selected element will inherit the number, if the **Apply to all entities** option is selected.

# Occurrence numbers

When you need to model a multi-gated part or a multi-cavity mold where all flow paths are symmetrical and equivalent, you can use occurrence numbers to reduce the modeling time. Occurrence numbers are used to specify the number of times that a given flow path is repeated in a model.

## Editing occurrence number properties

The occurrence number property is supported for the following model entities:

- All runner system related entities apart from the sprue (runner, gate, overflow well).
- All part related entities (part, part surface, part beam.
- Part insert.
- Preform surface.
- Compression surface.

The occurrence number property is not supported for:

- All Cool analysis related entities (channel, bubbler, baffle, hose, part insert).
- Core (Stress analysis used for core shift prediction does not support occurrence numbers).

---

**NOTE:** Occurrence numbers are supported for Fill+Pack analyses only. If you intend to also run a Cool or Warp analysis, you will need to model all flow paths explicitly.

---

1 Click (**Select**).
2 Select the model sections that you want to specify an occurrence number for.
3 Right-click on the highlighted model section and select **Properties**.
   If more than one entity type was selected, the **Select Properties** dialog appears.
   a Select one or more entities of the same type and then click **OK**.
   b Follow this procedure to the end and then repeat from step **3** for all entity types.
4 Enter the required **Occurrence number** for the selected model entity or entities.

   ---

   **TIP:** Select the **Apply to all entities that share this property** check box to apply the occurrence number to all model entities with the same properties.

   ---

5 Click **OK**.
6 Click **Mesh tab > Mesh Diagnostics panel > Occurrence**, then **Show**, to check that the occurrence numbers have been correctly assigned.

## Occurrence numbers

The **Occurrence Number** diagnostic is used to check the occurrence numbers assigned to the model.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Occurrence**).

### Occurrence number diagnostic dialog

By default, the occurrence number of all elements in the mesh will be 1. If the modeled part is symmetrical or occurs several times in a multi-cavity or family mold, you can save modeling and analysis time by assigning higher occurrence numbers.

In the **Options** section of the dialog, decide how you would like to view the results. No other input parameters need to be set to run this diagnostic; simply press the **Show** button.

---

**CAUTION:** If you intend to proceed to a Cool, Shrink, Warp or Stress analysis, you cannot use occurrence numbers. You must model the part in full.

---

## Using occurrence numbers to reduce model size

Occurrence numbers can be used to simplify the amount of modeling required for an analysis by specifying the number of times that a given flow path is repeated.

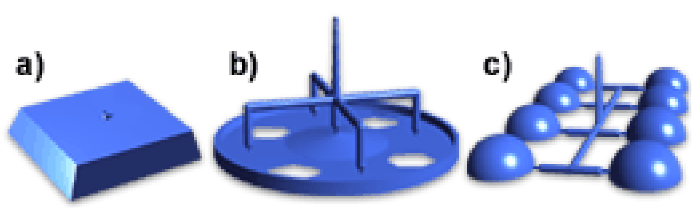
However, you must model the full part, instead of using occurrence numbers, when you want to run the following analyses:

- Cool analysis
- Warp analysis
- Stress analysis

Before we consider modeling simple parts using occurrence numbers it is helpful to understand the concept of symmetrical flow paths. Flow paths are said to be symmetrical when the physical geometry of the flow path is symmetrical and when the volume of plastic flowing through the paths are identical.

There are 3 main combinations of parts and runner systems:

- Direct injection into a cavity (a)
- Injection via a runner system into a multi-gated cavity (b)
- Injection via a runner system into multiple cavities (c)



Of course there are many other possible combinations apart from these three basic combinations, but for simplicity they will not be described here. By default, all surfaces and elements in the model are considered to have an occurrence number of 1.

Occurrence numbers can be specified using **Geometry tab > Properties panel > Edit** and then setting the Occurrence number on the **Part Surface Properties** tab.

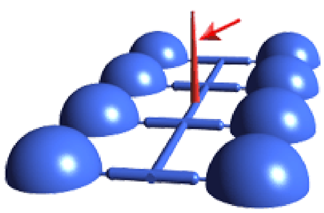
Occurrence number details are saved in the model file and are automatically read into the program with the model.

# Sprue

# 3

The sprue is the extension of the injection nozzle into the mold.

In a single cavity mold with a single injection location, the sprue can meet at the cavity wall. The sprue is usually connected to a runner system as shown in the following diagram.



The angle of the taper on a sprue should be large enough for it to be easily ejected, but not too large because the cooling time and the required amount of material increases along with the size of the sprue diameter.

## Sprue

You can create a sprue manually or automatically and edit the properties of an existing sprue.

### Creating a sprue manually

You can create a sprue manually by creating a line to the runner and meshing previously unmeshed parts.

Before creating the sprue, you must have already created the gate and runner.

1 Click **Geometry tab > Create panel > Curves > Create Line**.
2 Activate the **First** Coordinate box and click on the node in the center of the runner, or wherever you want the sprue to attach.
3 In the **Second** Coordinate area of the dialog, select **Relative** and enter the coordinates for the end of your sprue.
4 Click ... next to the **Create As** text box.
5 Click the **New** button and select **Cold sprue**.

6 From the **Shape is** drop-down box, select the shape you want, and click **Edit dimensions...**.

7 Enter a value for the **Start diameter**, and an angle in the **Tapered angle** box.

8 Click **OK** several times to return to the **Create Curves** dialog and then click **Apply**.

9 Click **Close**.

Now that you have created the geometry for the runner system, you need to mesh the new curves:

10 From the **Study Tasks** pane, right-click and select **Generate Mesh**.

Alternatively, click **Mesh tab > Mesh panel > Generate Mesh**. This will ensure that only the new curves are meshed.

11 Accept the defaults for the other options on the dialog and click **Mesh Now**.

## Creating a sprue automatically

The Runner System Wizard can be used to create a sprue automatically.

To create a sprue automatically, you must have an injection location identified.

1 Click **Geometry tab > Create panel > Runner System**.

The first page of the Wizard is used to specify the runner system layout, including the sprue position and the parting plane location, and whether you want to use a hot runner system.

2 The **Center of Mold** and **Center of Gates** buttons specify where you would like the sprue to be relative to either the gates or the mold configuration. Select the one you would like to use.

3 Different parting plane configurations will affect the Z coordinate for the runner system. Click the three parting plane specification buttons **Top**, **Bottom**, and **Gate Plane**, note the resulting Z coordinate and choose the one appropriate to your model.

4 Click **Next** to move on to the next Wizard page.
The second page of the Wizard is used to specify the sprue, runner, and drop sizes.

5 Enter the appropriate values on the second page of the Runner System Wizard for the **Sprue**, **Runner** and **Drops**.

6 Click **Finish** to create the runner system.

## Changing sprue properties

It is possible to change the shape and size of the sprue after it has been created.

This topic describes how you can change the properties of an existing sprue.

1 Click to change to selection mode.
2 Click on an element of the sprue.
The sprue color changes to red when selected.
3 Right click and select **Properties** from the list.
The **Edit Taper Section** dialog appears.
4 Choose whether you want to edit the entire sprue, or only the selected element. Click **OK**
The sprue properties dialog appears.
5 Use the **Sprue Properties** tab to change the sprue properties. The changes you make will only affect this sprue.
6 Click **OK** and view your changes.

# Sprue

The Create Sprue dialog is used to create the sprue and the Sprue Properties dialog is used to edit the sprue's properties.

To access this dialog to edit the properties of existing model entities, select at least one beam element or curve of type **Cold sprue**, then either select **Geometry tab > Properties panel > Edit**, or press **Alt-Enter**, or right-click and select **Properties**.

The collection of property values defined on the dialog are saved to a property set with the description shown in the Name box. In addition, you may be given the option to also apply the property values to related entities in the model.

## Edit Taper Section dialog

This dialog is used to choose whether you want to edit the entire tapered section, or only the elements that have been selected.

To access this dialog, select one or more elements on the tapered section. The selected elements will turn pink. Right-click the mouse then choose **Properties** from the drop-down menu.

## Cold sprue dialog—Sprue Properties tab

The **Sprue Properties** tab of the **Cold sprue** dialog is used to specify the geometric properties of the cold sprue.

## Cold sprue dialog—Mold Properties tab

The **Mold Properties** tab of the **Cold sprue** dialog is used to specify the properties of the mold block containing the cold sprue.

### Cold sprue dialog - Mold Temperature Profile

The **Mold Temperature Profile** tab of this dialog is used to select the temperature profile for the selected elements or regions.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

### Cold sprue dialog—Overmolding Component tab

The **Overmolding Component** tab of the **Cold sprue** dialog is used to specify which stage/component in the overmolding process the cold sprue relates to.

### Cross-Sectional Dimensions dialog

This dialog is used to specify the cross-sectional dimensions of the selected cold or hot runner/gate/sprue, part beam, cooling channel or part surface curves or beam elements.

Autodesk Moldflow Insight supports a number of different cross-sectional shapes for beam elements, as well as a generic definition of the cross-sectional shape based on a shape factor.

---

**NOTE:** If you select **Shape Is: Tapered (by angle)** for runner system related beams or elements, you are required to enter a **Tapered angle** value. The tapered angle is equivalent to half the **included angle**. For example, standard sprues have an included angle of .5[illegible]/12[illegible], or about 2.38°, which corresponds to a tapered angle of 1.19°.

---

## Hot sprue

A hot sprue is the initial part of a hot runner system.

A hot runner system maintains the material in the feed system of the mold at melt temperature. One advantage of a hot runner system is that the runner system is not ejected along with the part, and so there is less scrap produced.

### Hot sprue

You can create a hot sprue manually or automatically and edit the properties of an existing sprue.

#### Creating a hot sprue manually

Before creating the sprue, you must have already created the gate and runner.

1 Click **Geometry tab > Create panel > Curves > Create Line**.

2 Activate the **First** Coordinate box and click on the node in the center of the runner, or wherever you want the sprue to attach.

3 In the **Second** Coordinate area of the dialog, select **Relative** and enter the coordinates for the end of your sprue.

4 Click ... next to the **Create As** text box.

5 Click the **New** button and select **Hot sprue**.

6 From the **Shape is** drop-down box, select the shape you want, and click **Edit dimensions...**.

7 Enter a value for the **Start diameter**, and an angle in the **Tapered angle** box.

8 Click **OK** several times to return to the **Create Curves** dialog and then click **Apply**.

9 Click **Close**.

Now that you have created the geometry for the runner system, you need to mesh the new curves:

10 From the **Study Tasks** pane, right-click and select **Generate Mesh**.

Alternatively, click **Mesh tab > Mesh panel > Generate Mesh**. This will ensure that only the new curves are meshed.

11 Accept the defaults for the other options on the dialog and click **Mesh Now**.

### Creating a hot sprue automatically

The Runner System Wizard can be used to create a hot sprue automatically.

To create a sprue automatically, you must have an injection location identified.

1 Click **Geometry tab > Create panel > Runner System**.

The first page of the Wizard is used to specify the runner system layout, including the sprue position and the parting plane location, and whether you want to use a hot runner system.

2 The **Center of Mold** and **Center of Gates** buttons specify where you would like the sprue to be relative to either the gates or the mold configuration. Select the one you would like to use.

3 Select the **I would like to use a hot runner system** checkbox.

4 Different parting plane configurations will affect the Z coordinate for the runner system. Click the three parting plane specification buttons **Top**, **Bottom**, and **Gate Plane**, note the resulting Z coordinate and choose the one appropriate to your model.

5 Click **Next** to move on to the next Wizard page.

The second page of the Wizard is used to specify the sprue, runner, and drop sizes.

6 Enter the appropriate values on the second page of the Runner System Wizard for the **Sprue**, **Runner** and **Drops**.

7 Click **Finish** to create the runner system.

### Changing hot sprue properties

This topic describes how you can change the properties of an existing hot sprue.

1 Click  to change to selection mode.

2 Click on an element of the sprue.

The sprue color changes to red when selected.

3 Right click and select  **Properties** from the list.
The **Edit Taper Section** dialog appears.

4 Choose whether you want to edit the entire sprue, or only the selected element. Click **OK**
The **Hot sprue** dialog appears.

5 Use the **Sprue Properties** tab to change the sprue properties. The changes you make will only affect this sprue.

6 Click **OK** and view your changes.

## Hot sprue

These dialogs are used to change the properties of your hot sprue.

To access these dialogs, you must have created a hot runner system. Select an element of your hot sprue, right-click on it, select  Properties then follow the prompts.

### Hot sprue dialog—Sprue Properties tab

The **Sprue Properties** tab of the **Hot sprue** dialog is used to specify the geometrical and thermal properties of the hot sprue.

### Hot sprue dialog—Mold Properties tab

The **Mold Properties** tab of the **Hot sprue** dialog is used to specify the properties of the mold block containing the hot sprue.

### Hot sprue dialog—Overmolding Component tab

The **Overmolding Component** tab of the **Hot sprue** dialog is used to specify which stage/component in the overmolding process the hot sprue relates to.

## Cross-Sectional Dimensions dialog

This dialog is used to specify the cross-sectional dimensions of the selected cold or hot runner/gate/sprue, part beam, cooling channel or part surface curves or beam elements.

Autodesk Moldflow Insight supports a number of different cross-sectional shapes for beam elements, as well as a generic definition of the cross-sectional shape based on a shape factor.

---

**NOTE:** If you select **Shape Is: Tapered (by angle)** for runner system related beams or elements, you are required to enter a **Tapered angle** value. The tapered angle is equivalent to half the **included angle**. For example, standard sprues have an included angle of .5/12, or about 2.38°, which corresponds to a tapered angle of 1.19°.

---

# Gates

# 4

Gates connect the runner system to the cavity and are the orifices through which the melt enters the mold.

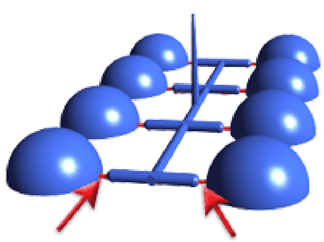
When you design gates, you should consider the following:

- The final appearance of the molded part
- Removal of the gate
- Complexity of the cavity
- The material used
- The volume of the material injected into the mold

Before designing the runner system, you should run a Gate Location analysis for each cavity to find out the best gate locations. For parts where appearance is important, the gates should be narrow to prevent large blemishes on the surface of the part. A smaller opening will also make gate removal easier.

Make gates short, to prevent large pressure drops and avoid sharp angles between gates and runners, which could contribute to a pressure drop in the system. Make corners rounded, so that the melt flow is not inhibited. The cross-sectional shape you choose for the gate depends on the shape of the runners.

The gates are highlighted in the following diagram.



## Valve gates

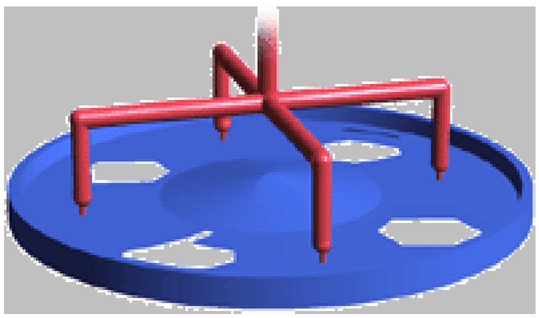
Polymer valve gates are used with a hot runner design in order to control how the cavity is filled, and to regulate the flow through each gate.

A polymer valve gate system opens and closes the gate with assistance of hydraulic or pneumatic cylinders. This topic details some issues to remember when setting polymer valve gates.

### Valve gate uses

A polymer valve gate system is used to control:

- How the cavity fills.
- Weld lines.
- Part weight.
- Cavity pressure.
- Overpacking.
- Flashing and drooling.



### Valve gate guidelines

- Valve gates can only be used in conjunction with hot runners (See picture above).
- There is no limit on the number of valve gates that can be assigned to a model.
- A valve gate must be assigned to a **single** hot gate element in each hot drop. For Gas-assisted Fill+Pack analysis, a valve gate can also be assigned to an overflow well (modeled using Midplane or beam elements for Midplane models, and tetrahedral elements for 3D models).
- A valve gate must not be assigned to the same element as an injection location. You should either model the feed system ahead of the valve gate, or ensure that the gate is meshed with more than one element along its length and then assign the valve gate to the gate element attached to the part.
- A valve gate is assigned to a hot gate or overflow well element by selecting a **valve gate controller** on the **Valve Control** tab of the properties dialog for the element. The properties that you assign to the valve gate controller specify the initial state and on/off timings of the valve gate.
- Each valve gate controller should have a unique name to identify it. The same valve gate controller can be assigned to more than one hot gate or overflow well element. To ensure you have set the correct controller at each gate, use a different property name for each valve gate controller and gate element.

- Valve gate controllers operate independently. One valve gate does not have the ability to close when the flow front reaches another valve gate.

---

**NOTE:** You cannot set a valve gate directly onto a runner. In order to set a valve gate, you must first select an existing hot gate or overflow well element.

---

## Manually trimmed gates

Gates can have many different configurations but they are broadly classified based on the method of gate removal into manually trimmed and automatically trimmed gates. Manually trimmed gates require an operator to separate the parts from runners during a secondary operation.

Manually trimmed gates are used for the following reasons:

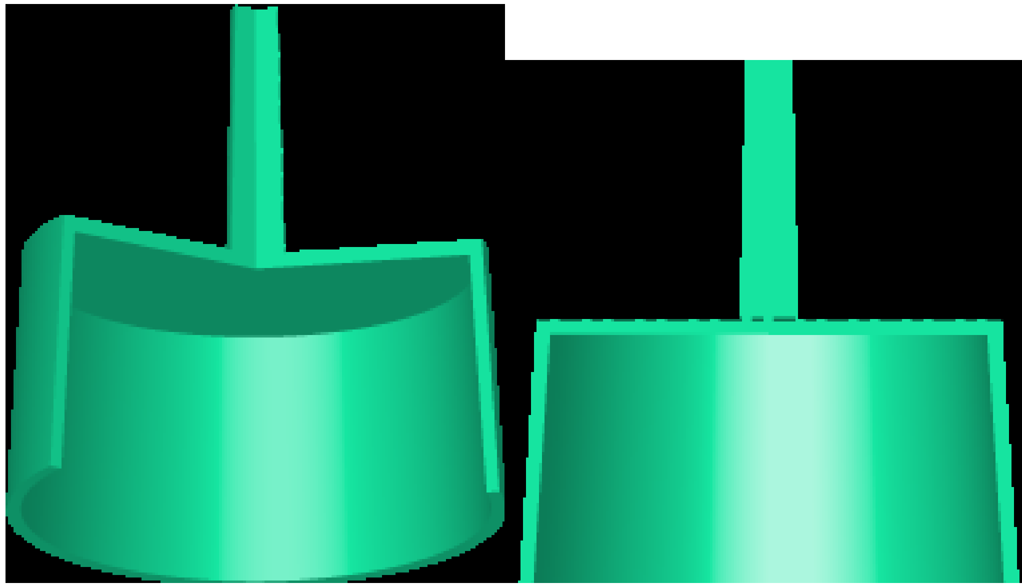
- The gate is too large to be sheared from the part as the tool is opened.
- Some shear-sensitive materials, such as PVC, should not be exposed to the high shear rates inherent to the design of automatically trimmed gates.
- Simultaneous flow distribution across a wide front to achieve specific orientation of fibers or molecules often precludes automatic gate trimming.

Manually trimmed gate types include:

- Direct or sprue gate
- Disc or diaphragm gate
- Edge or standard gate
- Fan gate
- Film or flash gate
- Overlap gate
- Ring gate
- Spoke or spider gate
- Tab gate

**Direct or sprue gate**

A direct gate is commonly used for single-cavity molds, where the sprue feeds mater directly and rapidly into the cavity with minimum pressure drop, as shown in the following diagram.



The disadvantage of using this type of gate is the gate mark left on the part surface after the sprue is trimmed off. Freeze-off is controlled by the part thickness rather th determined by the gate thickness. Typically, the part shrinkage near the sprue gate w be low; shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate.

**Dimensions:**

The starting sprue diameter is controlled by the machine nozzle. The sprue orifice diameter here must be about 1mm larger than the nozzle exit diameter. Standard spr can have tapers from 0.5 degrees to 1.5 degrees (1.0 degrees to 3 degrees included ang with a common size of about 1.2 degrees taper angle (1/2 inch per foot included ang Therefore, the sprue's orifice diameter and length will control the diameter of the sp where it meets the part. Typically, the sprue diameter will be well over double the w thickness of the part, controlling the molding cycle time.

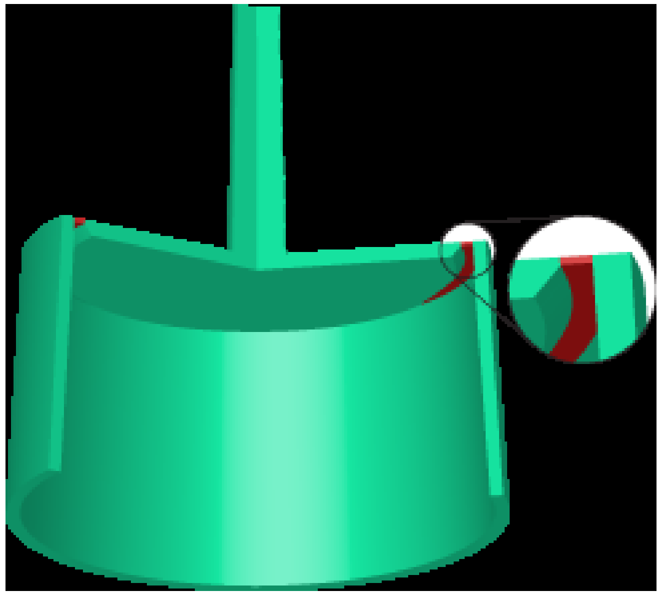
A smaller taper angle (a minimum of one degree) risks not releasing the sprue from t sprue bushing on ejection. A large taper wastes material and extends the cooling tir

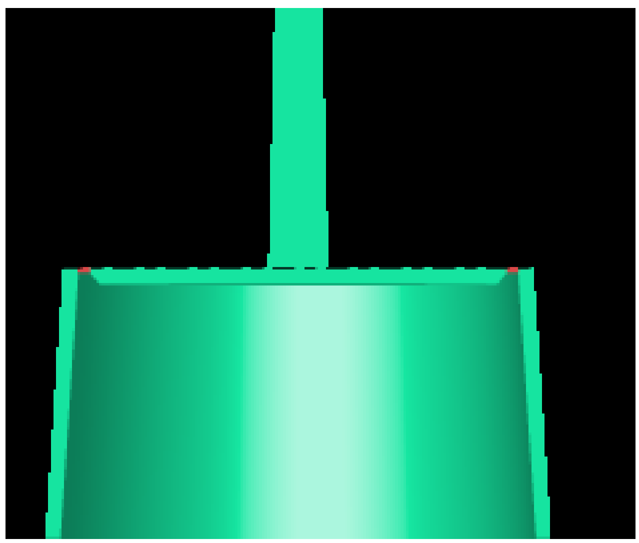
Non-standard sprue tapers will be more expensive to machine, with little gain.

**Disc or diaphragm gate**

A disc gate is often used for gating cylindrical or round parts that have an open ins diameter. This gate is useful when concentricity is an important dimensional requirement, and the presence of a weld line is objectionable. These gates are typica difficult and expensive to trim from the part.

As shown in the following diagram, the disc gate has a thin land around the inside edge of the part, which facilitates the removal of the gate. Since the disc is fed fron concentric sprue or hot drop, uniform flow to all parts of the gate is easy to mainta



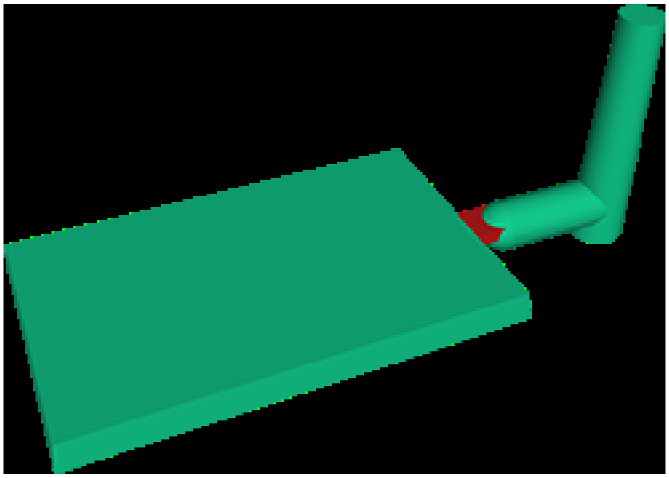


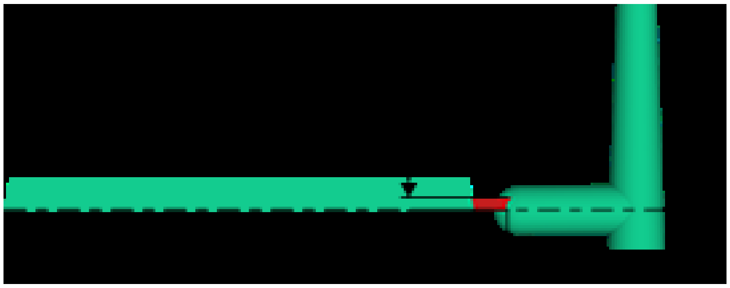
**Dimensions:**

The typical gate thickness (H) is 0.2 to 1.3 mm.

**Edge or standard gate**

An edge gate is located on the parting line of the mold, as shown in the following diagram. The gate cross section is rectangular and can be tapered in width and/or thickness between the part and runner.





**Dimensions:**

The typical gate thickness (H) is 25 to 75 percent of the part thickness, and the wic
is typically two to ten times the thickness. The gate land should be short, typically
to 1.0 mm in length. Larger parts can have longer land lengths.

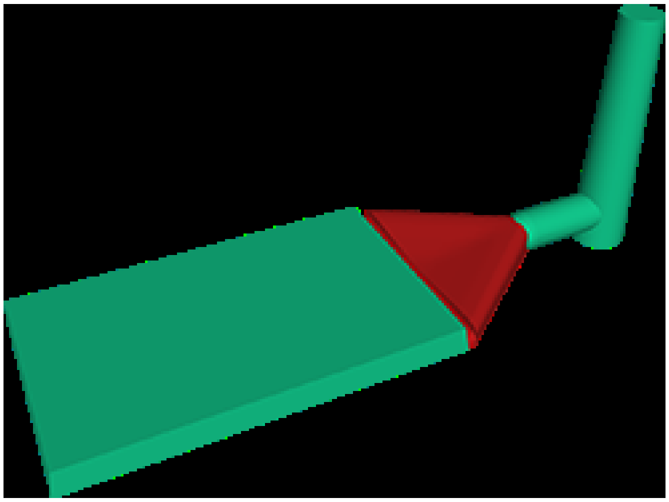
## Fan gate

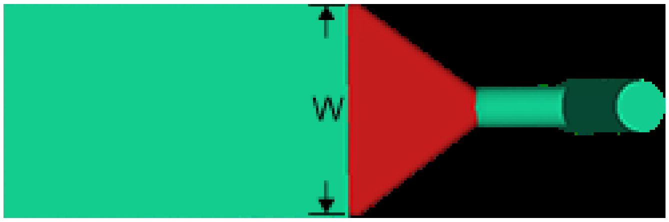
A fan gate is a wide edge gate with variable thickness, which permits rapid filling o
large parts or fragile mold sections through a large entry area. Fan gates are used to
create a uniform flow front into wide parts where warpage and dimensional stabili
are main concerns.

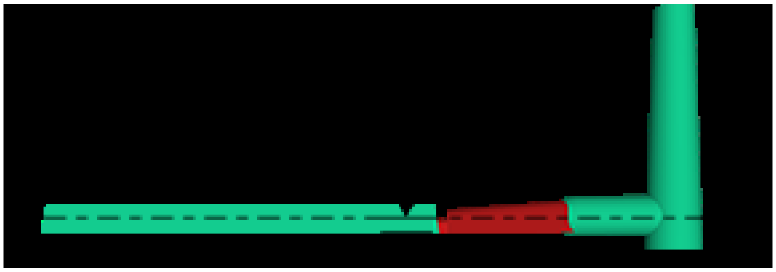
As shown in the following diagrams, the fan gate tapers in both width and thickne
to ensure the following:

- The flow front velocity will be constant across the entire width

- The entire width is being used for the flow
- The pressure is the same across the entire width





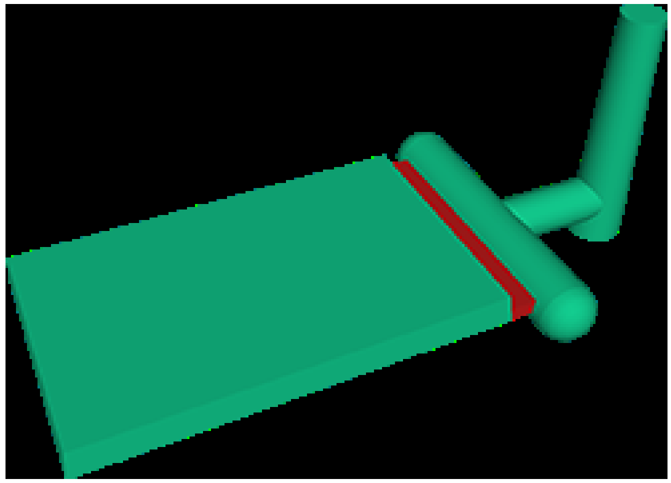
**Dimensions:**

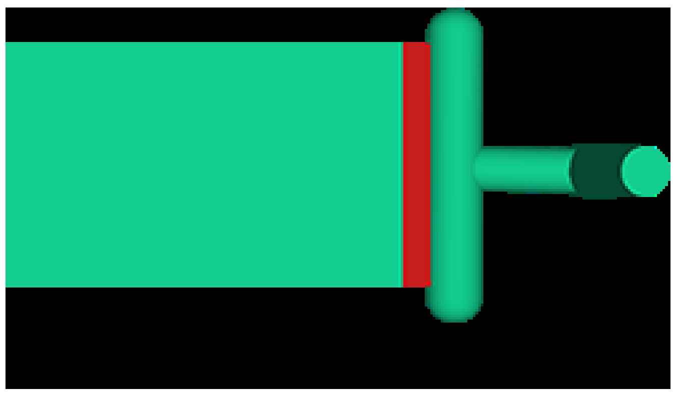
Well-designed fan gates have a narrow land, typically 2.0 mm or less. This land will very thin, typically under 1 mm. The gate width is typically 25 mm to as wide as tl part.

The main body of the gate will be thin in the center and thick on the edges to prom flow to the outer edges.

**Film or flash gate**

A film gate consists of a straight runner and a gate land across the entire width of t cavity or a portion of the cavity, as shown in the following diagrams.





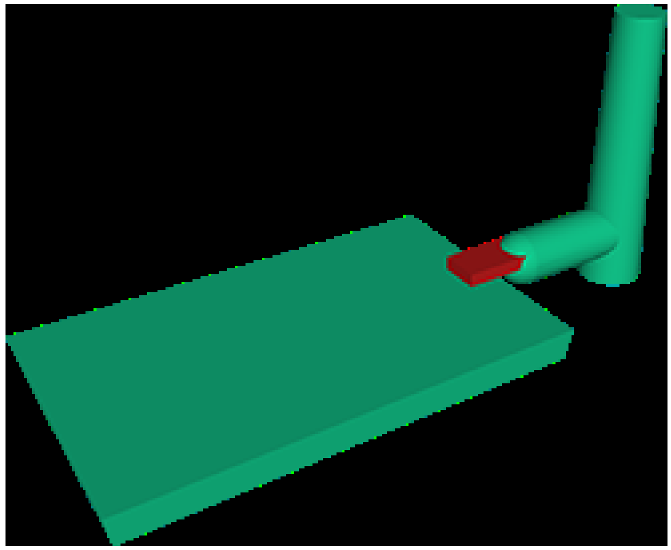
It has the same objectives as a fan gate but it is more difficult to achieve. The thin la
of the gate has areas that cause hesitation, and it is very sensitive to the thickness o
the land, diameter of the runner and flow rate.

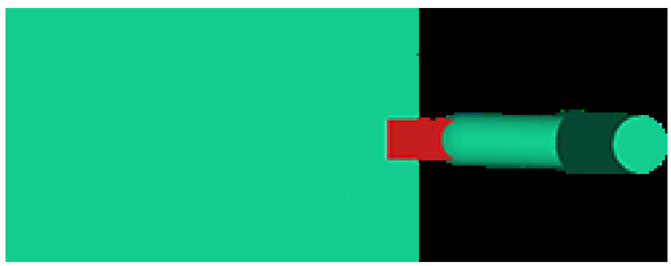
**Dimensions:**

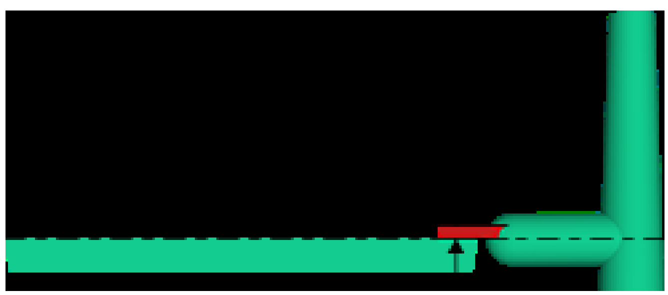
The typical gate size is small, approximately 0.2 to 1.0 mm thick (H). The land area
(gate length (L)) must also be kept small, typically under 1 mm.

**Overlap gate**

An overlap gate is similar to an edge gate but a portion of the gate overlaps the part
shown in the following diagrams.





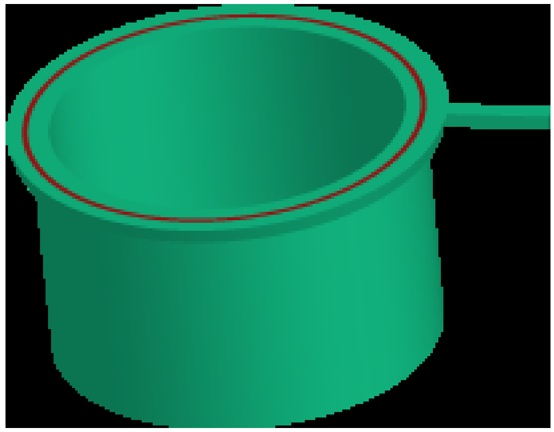


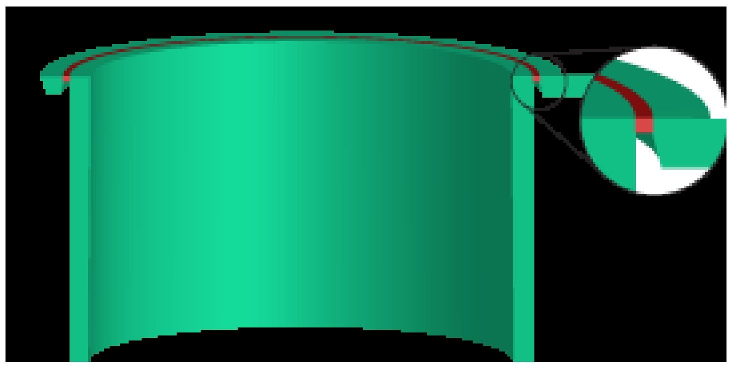
**Dimensions:**

The typical gate thickness(H) is 25 to 75 percent of the part thickness, and the wid
is typically 2 to 10 times the thickness. The gate land should be short, typically 0.5
1.0 mm in length. Larger parts can have longer land lengths.

**Ring gate**

With a ring gate, the material flows freely around the core before it moves down as
uniform tube-like extrusion to fill the mold, as shown in the following diagrams.





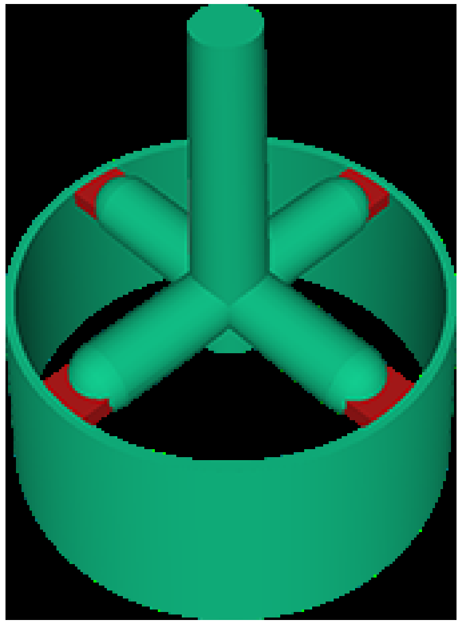
**NOTE:** In practice this is difficult to achieve because the gate relies on hesitation in t
thin gate land to achieve a balanced fill. Uniform fill is sensitive to the gate land, runn
around the part and the injection time.

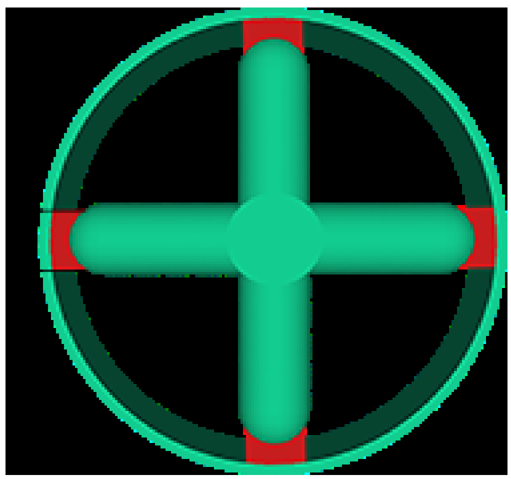
**Dimensions:**

The typical gate thickness (H) is 0.2 to 1.5 mm.

### Spoke or spider gate

The spoke gate, which is also called a four-point gate or cross gate, is shown in the following diagrams. This gate is used for tube-shaped parts and offers easy gate remo and material savings. Disadvantages are the possibility of weld lines and the fact th perfect roundness is unlikely.



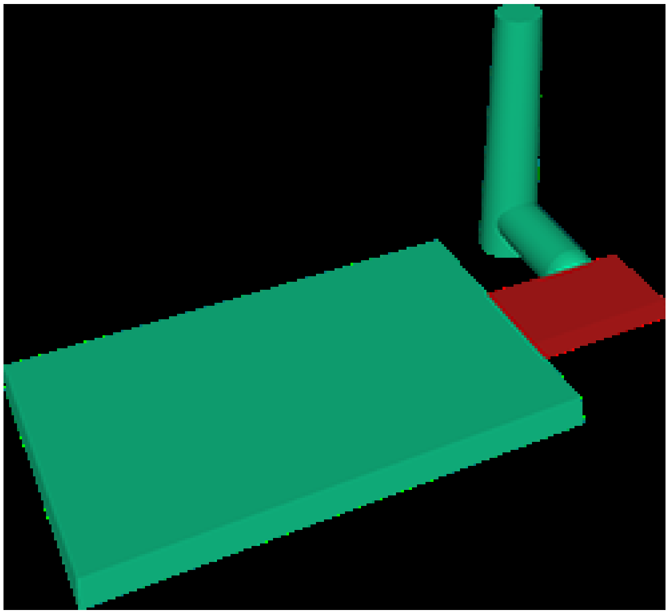


### Dimensions

The gate cross section can be rectangular like an edge gate and will have similar nomi dimensions, or it can have a circular cross section and be configured like a circular tapered gate.

### Tab gate

A tab gate is typically employed for parts that require low shear stresses, lsuch as opti parts. The high shear stress generated around the gate is confined to the auxiliary t which is trimmed off after molding. A tab gate, which is shown in the following diagra is used extensively for molding PC, acrylic, SAN, and ABS material types.



**Dimensions:**

The typical minimum tab width (W) is 5 mm. The typical minimum tab thickness (
is 75 percent of the depth of the cavity.

**NOTE:** Ranges of typical dimensions are given for different gate types. Actual gate dimensions will vary depending on the selected material, part geometry, and number of gates.

## Automatically trimmed gates

Gates can have many different configurations but they are broadly classified according to their method of degating into manually trimmed and automatically trimmed gates. Special features are incorporated into automatically trimmed gates so that the gates are trimmed or sheared when the mold opens and the parts are ejected.

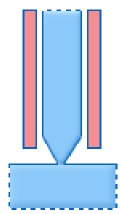
Automatically trimmed gates are used to avoid gate removal as a secondary operation, and to minimize gate scars

Automatically trimmed gate types include:

- Hot-runner or hot-probe gate
- Pin gate
- Submarine, tunnel, or chisel gate
- Valve gate

**Hot-runner or hot-probe gate**

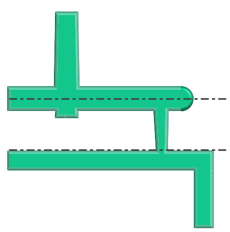
A hot-runner gate, which is shown in the following diagram, is generally used to deliver hot material through heated runners directly into the cavity to produce runnerless moldings.



The gate or gate tip can have many different configurations from full round to annular. The geometry and size of the gate tip will determine how the gate freezes and the gate scar formed.

**Pin gate**

The pin gate is used in a three-plate mold design, where the runner system is on a secondary mold parting line and the part cavity is in the primary parting line. Reverse taper runners drop through the middle plate, parallel to the direction of the mold opening as shown in the following diagram.



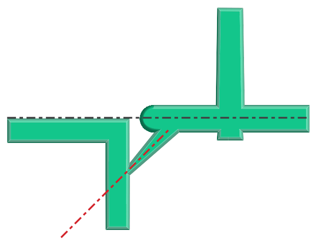
As the mold cavity parting line is opened, the small-diameter pin gate is torn from the part. A secondary opening of the runner parting line ejects the runners. Alternatively, the runner parting line opens first. An auxiliary, top-half ejector system extracts the runners from the reverse taper drops, tearing the runners from the parts.

**Dimensions:**

Typical gate sizes are 0.2 to 1.5 mm in diameter. The design is particularly useful when multiple gates per part are needed to assure symmetric filling, or where long flow paths must be reduced to assure packing to all areas of the part.

**Submarine, tunnel, or chisel gate**

A submarine gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line, as shown in the following diagram.



As the parts and runners are ejected, the gate is sheared at the part.

If a large diameter pin is added to a non-functional area of the part, the submarine gate can be built into the pin, avoiding the need of a vertical surface for the gate. If the pin is on a surface that is hidden, it does not have to be removed.

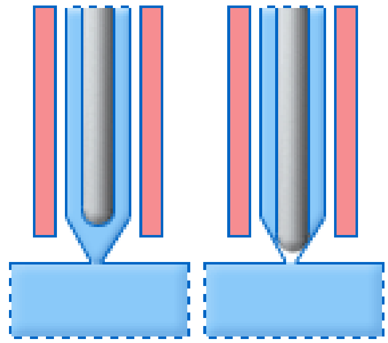
Multiple submarine gates into the interior walls of cylindrical parts can replace a diaphragm gate and allow automatic degating. The out-of-round characteristics are not as good as those from a diaphragm gate, but are often acceptable.

**Dimensions:**

The typical orifice diameter of the gate is 30 to 75 percent of the part wall thickness. The gate is tapered at a minimum of 10 degrees per side to ensure proper ejection. It is common to have the gate taper to the diameter of the runner.

**Valve gate**

The valve gate adds a valve pin to the hot runner gate. Valve gates have a larger gate diameter and they can be opened and closed as needed as shown in the following diagram. This smooths over the gate scar.



Since the packing cycle is controlled by the valve pin, better control of the packing cycle is maintained with more consistent quality.

**NOTE:** Ranges of typical dimensions are given for different gate types. Actual gate dimensions will vary depending on the selected material, part geometry, and number of gates.

## Hot gates

The Dynamic Feed system is a technology invented at Stanford University in 1993, and developed since 1997 by Synventive Molding Solutions™, a leader in advanced hot runner technology.
Dynamic Feed is a melt delivery system which provides independent, real-time, closed-loop process control at each gate in the mold.
To simulate this technology within Autodesk Moldflow Insight, hot gate controllers are used.

#### What is a hot runner system?

Hot runner systems are melt delivery systems that maintain a constant supply of melted plastic in between the injection molding machine nozzle and the part cavities of the mold. Hot runners eliminate the scrap associated with cold runner systems because cold runners cool and solidify, and are ejected from the mold with the part. The cold runners then have to be ground up, re-melted, and reused, or disposed of as scrap. With hot runners, generally only the parts themselves are ejected from the mold.

#### How does the Dynamic Feed system work?

This feature uses pressure controllers at each gate in a mold, and Autodesk Moldflow Insight can simulate up to 32 injection locations. Signals come from the pressure sensors of the hot runner system(s) to control multiple hot runner valve gates dynamically, in order to achieve a pre-determined pressure profile for each gate.

The general machine controller, or the central controller, is set to use a high (usually maximum) pressure, to ensure the flow-rate requirement for injecting the melt through the multiple valve gates controlled by the individual pressure controllers. There is no velocity/pressure switch-over, as there is no velocity control stage.

The application is typically used for multi-cavity and family molds, and for multi-gated large part injection moldings, in particular.

The current manufacturing set-up procedure is to use trial-and-error for an individual cavity to get a desired pressure profile as a "finger-print", and then apply it to individual valve gate controllers for the multi-cavity case. For a multi-gated large part, it is difficult to set up, which makes the software simulation very important beforehand.

## Modeling gates for 3D Fill+Pack analysis

When analyzing a 3D model, the implied gate size of the model is of critical significance when predicting injection pressure for a part.

### Modeling gates for 3D analyses

The 3D flow solver has two ways to achieve accurate injection pressure results:

- Values derived from a modeled gate and a specific gate injection node.
- If no gate is modeled (direct injection to qualitatively test the effect of gate size on the results), a virtual gate diameter (gate contact diameter) is automatically assigned based on the part geometry, or in the case of small parts the average facet size of the tetrahedral elements on the surface around the injection location. The gate contact diameter can also be manually specified from the **Solver Parameters** dialog.

The gate contact diameter is used by the solver to set the appropriate viscosity of the tetrahedron nodes at/near the injection point. All nodes inside the gate diameter are treated as injection nodes and will have the same temperature, pressure, and shrinkage.

If you want to compare 3D and Dual Domain pressure results, a runner system is required.

### Recommended gate modeling technique

In order to achieve accurate injection pressures for 3D Fill+Pack results from a modeled gate, model the gate as a tab connected to the cavity. Then you can define the whole face of the tab as the injection location, and regardless of the mesh size, you will have the same area for the injection gate. As an alternative, you can pick more of the surrounding nodes in the fine mesh, and so make a gate that has approximately the same size as the gate in the coarse model.

**NOTE:** The dependence on mesh size for determining a modeled gate size is different for 3D analyses. This is inherent in the nature of 3D analyses. In shell models, the thickness of the gate is assigned as an attribute. In 3D, the thickness is a concept that only comes from the surrounding geometry.

#### Comparing injection pressures

Using the ability of Autodesk Moldflow Insight to use beam runner feed systems with a 3D cavity model, there is a second option available that allows you to compare pressures in 3D Fill+Pack against those in Dual Domain Fill+Pack. If you model a short beam runner system (even just the gate will do) for both 3D and Dual Domain, and then pick the end node as the injection node, then the gate size in the two analyses will match.

## Setting a gate contact diameter

A gate contact diameter is used by the Coupled 3D Flow solver in a Fill or Fill+Pack analysis.

To access this dialog, choose an analysis sequence which includes Fill or Fill+Pack. Click **Analysis > Process Settings Wizard** . Click **Next**, if necessary, until the Fill or Fill+Pack page appears. Click **Advanced Options**, then click **Edit** in the Solver Parameters group. On the **Thermoplastics injection molding solver parameters (3D)** dialog, select the **Fill+Pack Analysis** tab. In the Solver setup group, select **Use coupled solver**, then click **Solver Parameters**.

The **Gate contact diameter** option assigns a virtual gate diameter to each injection node when a modeled gate or runner system is not provided (direct injection). This improves the flow front prediction and provides more accurate pressure results and better jetting representation around the gate. However, where possible, it is advisable to include a modeled gate or runner system.

**NOTE:** The gate contact diameter has no effect when injecting into a beam element or at a gas injection location. However, polymer injection during a Gas-assisted Injection Molding analysis is affected.

**Automatic** Assigns a gate contact diameter based on the facet size around the injection node. The default diameter assigned is based on the molding process, but is checked and potentially adjusted based on the mesh size.

- Thermoplastic—default gate size 2mm
- Thermoset—default gate size 1mm
- Underfill—default gate size 0.1mm

To ensure that the applied gate diameter is reasonable, mesh-dependent upper and lower limits are applied. The effective surface area (A)of the injection node is calculated as one third of the sum of the tetrahedral surface facets surrounding the node. From (A), an effective diameter (D) is calculated. The upper and lower limits are then set as (5D) and (D), respectively.

For example, if the effective diameter (D) is 2mm, then the upper and lower limits are set as 10mm (5D) and 2mm (D), respectively.

- If (default gate size value) < D, then the lower limit D is used.
- If (default gate size value) > 5D, the the upper limit 5D is used.
- Otherwise the default values are used.

As a result of these calculations, if more than one injection location is specified, the individual gate diameters may vary.

**Specified** Allows you to enter the actual diameter of the gate(s).

If more that one injection location is specified, all gates will have the same size, temperature, pressure and shrinkage irrespective of the part geometry or facet size.

# Runners

# 8

The runners are the feed channels that connect the sprue to the gates. The design of the runners is important to ensure even filling of the cavities.

The design of the runners is dependent on whether the mold is single cavity, multi-cavity, or part only. This topic explains how to lay out the runners and avoid uneven filling, hesitation, and overpacking.

## Runners

Runners are modeled along planes. They can be adjusted to meet the molding requirements.

### Creating a runner system manually

1 Click (**Geometry tab > Create panel > Nodes > Node by Coordinate**) to create the nodes that will define the feed system.

---

**NOTE:** You can use any node creation tool.

---

**TIP:** You do not have to create nodes first to create the gate and runners. You can create curves only, if you want.

---

2 Click (**Geometry tab > Create panel > Curves > Create Line**).

3 Enter the start coordinate of the curve or select the node that defines the start of the curve.

---

**TIP:** Set the Filter to Node or Nearest node to ensure you select a node rather than a point in space close to an existing node.

---

4 Select **Absolute** or **Relative** depending on how you want to define the end coordinate.

5 Enter the end coordinate of the curve or select the node that defines the end of the curve.

6 Select the appropriate property from the **Create as** drop-down list.

If the required feed system property is not in the list, click **Change...** and select the required property.

7 Click **Apply**.

8 Repeat steps **3-7** for each curve of the feed system you are creating.

---

**TIP:** The second coordinate of the last curve created becomes the first coordinate of the next curve you are creating.

---

9 Duplicate the cavity and runners as necessary to reduce the entities being created manually.

10 Mesh the runners.

---

**TIP:** If you want to place the curves and beams for the feed system on the same layer, make sure to select the **Place mesh in active layer** option.

---

11 Click (**Mesh tab > Mesh Diagnostics panel > Connectivity** ) to check the connectivity of the runners with the cavities.

12 Click on the element at the top of the sprue, then click **Show**.

13 If there are disconnected mesh elements, click (**Mesh tab > Mesh Repair panel > Global Merge**).

---

**NOTE:** In order to avoid merging unnecessary nodes, make sure the **Merge nodes along an element edge only** option is not selected. After applying the tool, check to ensure only the necessary nodes were merged. if unnecessary nodes were merged, click (**Undo**), then reduce the tolerance and try the merge again, or fix the problem manually.

---

14 Click (**Home tab > Molding Process Setup panel > Injection Locations**) and set an injection location on the top of the sprue.

## Converting runners from triangles to beams

Runner systems must not be modeled with triangular elements. If a model is imported and meshed with runners, the runners must be converted to beam elements. Manually converting the runners to beams will take a significant amount of time. A tool can be used to identify and convert runner sections quickly.

---

**NOTE:** Before performing a conversion process, it is important to ensure that the mesh is free of sharp edges, overlaps and intersections.

---

1 Click **Geometry tab > Modify panel > Elements to Beams**.

The **Simplify Elements to beams** panel appears in the **Tools panel**.

2 Identify the **simple sections** of the runner system. Simple sections are straight or gently curved and have no junctions or sharp corners.

3 Repeat the following for all simple sections:
   a Select any node on the section.
   b Click **Apply**. The simple section containing the node is converted into a beam.
4 When all simple sections have been converted, click the **Start stage 2** checkbox.
5 Repeat the following for all remaining parts of the runner system (corners and junctions):
   a Select any node on the section.
   b Click **Apply**. The section containing the node is converted into a beam section.
6 Click **Close** to dismiss the dialog.

---

**NOTE:** The gates will have to be modeled separately. Deleting the existing mesh prior to modeling the gate will cause holes in the mesh that will also need to be repaired.

---

**NOTE:** The beam element properties may need to be redefined for a more accurate runner system (Select the element/s to be redefined, right-click and select **Properties**, or click **Geometry tab > Properties panel > Edit** to open a dialog where the properties can be altered).

---

### Displaying the interior of an annular runner

Displaying the interior of an annular runner allows you to view both the inner and outer section of the annular runner.

1 Click **View tab > Appearance panel > Entities**.
2 On the **Default Display** tab, set **Beam Element** to **Transparent + Element Edges**.

You are now able to view both the inner and outer section of the annular runner.

## Runners

These dialogs are used to create runners and edit their properties.

To access the cold runner dialog, click **Geometry tab > Properties panel > Assign**, then in the **Assign Property** dialog, click **New** and select **Cold runner**.

### Simplify Elements to beams dialog

This panel steps you through the conversion of a runner into beam elements.

**NOTE:** The gates will have to be modeled manually. Deleting the existing mesh prior to modeling the gate will cause holes in the mesh that will also need to be repaired.

To use this tool, click (**Geometry tab > Modify panel > Elements to Beams**).

**NOTE:** The beam element properties may need to be redefined for a more accurate runner system. Select the element/s to be redefined and select (**Mesh tab > Properties panel > Edit** ) to open a dialog where the properties can be altered.

## Cold runner dialog—Runner Properties tab

The **Runner Properties** tab of the **Cold runner** dialog is used to specify the geometrical properties of the cold runner.

## Cold runner dialog—Mold Surface Temperature tab

The **Mold Surface Temperature** tab of the **Cold runner** dialog is used to specify a fixed temperature for the mold surface in contact with the cold runner (Reactive Molding, Multiple-Barrel Reactive Molding and Microcellular Injection Molding processes only).

## Cold runner dialog—Mold Temperature Profile tab

The **Mold Temperature Profile** tab of this dialog is used to select the temperature profile for the selected elements or regions.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

## Cold runner dialog—Mold Properties tab

The **Mold Properties** tab of the **Cold runner** dialog is used to specify the properties of the mold block containing the cold runner.

## Cold runner dialog—Overmolding Component tab

The **Overmolding Component** tab of the **Cold runner** dialog is used to specify which stage/component in the overmolding process the cold runner relates to.

## Cold runner (3D) dialog—Mold Properties tab

The **Mold Properties** tab of the **Cold runner (3D)** dialog is used to specify the properties of the mold block containing the cold runner.

## Cold runner (3D) dialog—Overmolding Component tab

The **Overmolding Component** tab of the **Cold gate surface (Dual Domain)** dialog is used to specify which stage/component in the overmolding process the cold gate relates to.

## Cold runner (3D) dialog—Part Surface Properties tab

The **Part Surface Properties** tab of the **Cold runner (3D)** dialog is used to specify the geometrical properties of the cold runner.

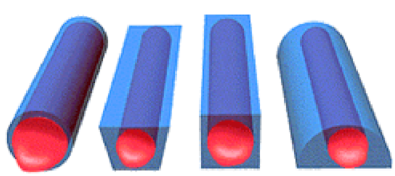
# Runner properties

The shape and diameter of the runners are important factors in successful mold design.

The shape of the runner affects the volume of material that remains molten, and the diameter affects the temperature of the melt in the runners and, thereby, the quality of the product and material waste.

### Effects of shape

The cross-sectional shape of the runners affects the flow of the polymer through the runner system. When the hot melt hits the cold metal of the runner a layer freezes and forms a skin on the surface of the runner. The center of the runner remains molten while the polymer is being injected into the mold. The following diagram shows the molten core for different shaped runners.

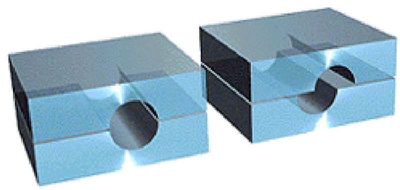


A circular cross section provides the greatest proportion of polymer in a molten state. Runners with a curved or angular cross section require less force to remove from the mold than rectangular or square runners.

Although circular runners are the best choice for material flow and ejection, they are also the most expensive. This is partly because the runner needs to be cut into both plates and it is difficult to cut both halves of the runner so that they meet exactly.

A trapezoidal cross section can be used as a compromise. Trapezoidal runners often provide acceptable flow and ejection characteristics, and are cheaper to produce than round runners.

If you do decide to use a circular runner, extra care is needed to align the two halves of the circular runner to avoid an increase in injection pressure due to the reduced effective flow cross-section. In the following diagram, the runner on the left is correctly aligned, but the runner on the right will have a smaller molten center that will restrict flow.



#### Effects of diameter

A small runner diameter causes shear heating in the runners so the plastic temperature is higher in the runners than in the barrel. Higher melt temperatures reduce residual stress levels and the tendency of parts to warp, but high barrel temperatures can cause degradation of the material.

To minimize material waste and decrease the barrel temperature required, design the runners with a small cross-sectional area.

---

**NOTE:** Changes made to the diameters of runners should be gradual . Avoid creating a large difference in size between the runner diameter and the gate diameter, or the gate diameter and the part surface thickness. Sizeable changes in thickness at these intersections may cause the following molding problems:

- Rapid changes in flow resistance
- Flow instabilities
- Increased injection pressure

---

## Runner properties

Default and individual runner properties can defined.

### Setting default runner properties

There are two ways to affect the runner type, shape and size. This topic describes how you can set defaults for these properties before you create the runner.

You can also change the properties of existing runner sections.

1 Click  **Geometry tab > Properties panel > Defaults > Runner Properties**.

   The **Runner properties** dialog appears.

2 In the **Runner type** drop-down list, select the desired runner type.

3 In the **Runner shape** drop-down list, select the desired runner shape. Available shapes differ depending on the selected runner type.

---

**NOTE:** The **Annular** runner shape is only available for the **Hot** runner type.

---

4 Change or enter a value for each runner attribute. The values you are requested to enter depend on the selected runner shape.

5 Click **Runner Balance Constraints...** to set the **Runner Balance** optimization level.

6 Click **OK** to save and close the dialog.

### Changing runner properties

There are two ways to affect the runner type, shape and size. This topic describes how you can change the properties of an existing runner section.

You can also set default runner properties before you create a runner section.

1 Click  to change to selection mode.

2 Double-click the runner to display the runner properties in the **Tools** pane.

The runner color changes to red when selected.

---

**TIP:** You can also hold-down the **CTRL** key, multi-select the runners, and then right-click and select **Properties**.

---

3 Use the **Runner Properties** dialog to change the runner properties. The changes you make will only affect the selected runner section.

---

**TIP:** For multiple runners with the same properties, hold-down the **CTRL** key, multi-select the runners in the **Select entity to edit** list, and then change the properties.

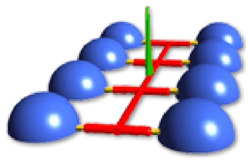
---

4 Click **Apply** and view your changes.

## Designing the runner layout

The combination of sprue, runners, and gates is used to transport the melt from the injection nozzle to the injection location for each part.

The design of the runner layout affects the amount of material used and the quality of the parts produced. If the flow within each cavity is unbalanced, overpacking and hesitation can lead to poor part quality. Long or poorly designed runners can cause large pressure drops and require a larger injection pressure to fill the part. The following diagram shows a typical runner system for a multi-cavity two-plate mold.



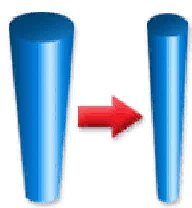
When designing the layout of the runner system, you must:

- Determine the number of cavities required
- Determine the material type
- Determine the processing conditions
- Adjust any flow leaders or deflectors as necessary
- Flow balance the cavities

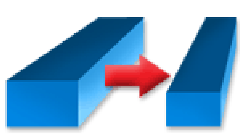
## Runner aspect ratios

The runner balance calculation is set to maintain the same geometry aspect ratio of runner dimensions when balancing runners.

For example, if the ratio of end diameters for a tapered runner is 2:1, the program will try to maintain this ratio for the balanced runner, as shown in the following diagram.



Similarly, the program will try to maintain the ratio of the sides for rectangular runners, as shown in the following diagram. Exceptions to this rule occur when you have specified standard runner sizes.



## Runner sizes

Generally, the minimum dimension of a runner cross-section should be 1.5mm greater than the thickness of the part in millimeters. This enables the cavity to pack evenly and produce an even volumetric shrinkage.

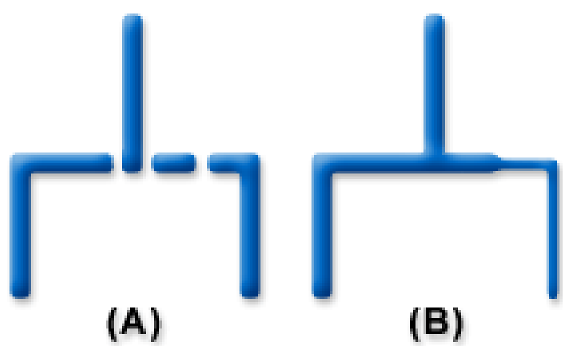
The selection of runner dimensions may be limited by the type of material and the design of the part. For example, a 1mm diameter runner of styrene may snap when the part is being ejected from the mold, but a 1mm diameter runner of nylon may flex. The results also depend on the type of ejection system that is used.

## Runner sections

Before modeling runner surfaces, it is important to understand how runner sections are used by the runner balance calculation.

Runner balancing ensures that all the cavities finish filling simultaneously. The aim of a runner balance analysis is to achieve the balanced filling of all cavities in the mold by using different runner diameters.

In the following diagram, (A) shows a runner system where the runner on the left has a single runner section, while the runner on the right has two sections. Diagram (B) shows how the Runner Balance tool might balance the system by assigning a single size to the left runner section, and assigning different sizes to the two sections in the right runner. This will result in a balanced runner system that has varying thicknesses in the secondary runners which may or may not be desirable depending on the mold requirements.



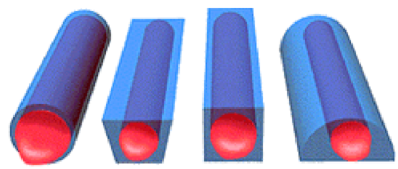

You can control the size increase or decrease of the runner dimensions by carefully considering the size of the runner surfaces.

### Runner cross-sections - Shape

The shape of the runner affects the volume of material that remains molten.

#### Effects of shape

The cross-sectional shape of the runners affects the flow of the polymer through the runner system. When the hot melt hits the cold metal of the runner a layer freezes and forms a skin on the surface of the runner. The center of the runner remains molten while the polymer is being injected into the mold. The following diagram shows the molten center of different runner shapes.

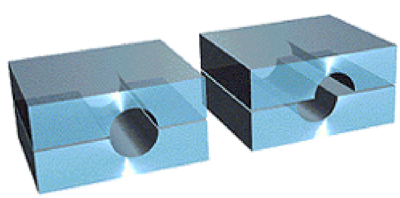


A circular cross section provides the greatest proportion of polymer in a molten state. Runners with a curved or angular cross section require less force to remove from the mold than rectangular or square runners.

Although circular runners are the best choice for material flow and ejection, they are also the most expensive. This is partly because the runner needs to be cut into both plates and it is difficult to cut both halves of the runner so that they meet exactly.

A trapezoidal cross section can be used as a compromise. Trapezoidal runners often provide acceptable flow and ejection characteristics, and are cheaper to produce than round runners.

If you do use a circular runner, extra care is needed to align the two halves of the circular runner to avoid an increase in injection pressure due to the reduced effective flow cross-section. In the following diagram, the runner on the left is correctly aligned, but the runner on the right will have a smaller molten center that will restrict flow.



## Runner cross-sections - Diameter

The diameter of the runners influences the temperature of the melt in the runners and thereby the quality of the product and the amount of material waste.

### Effects of diameter

A small runner diameter causes shear heating in the runners so the plastic temperature is higher in the runners than in the barrel. Higher melt temperatures reduce residual stress levels and the tendency of parts to warp, but high barrel temperatures can cause degradation of the material.

To minimize material waste and decrease the barrel temperature required, design the runners with a small cross-sectional area.

**NOTE:** Change made in the diameters of runners should be gradual. Avoid creating a large difference in size between the runner diameter and the gate diameter, or the gate diameter and the part surface thickness. Sizeable changes in thickness at these intersections can cause the following molding problems.

- Rapid changes in flow resistance
- Flow instabilities
- Increased injection pressure

---

# Hot runners

Hot runners have heated runners. The material in the runners remains in the molten state and are not ejected with the molded part. Hot runner systems, which are also referred to as hot-manifold systems or runnerless molding, minimize flash and gate stubs and reduce material waste.

## Hot runners

Hot runners are created the same way as cold runners, but they are assigned the property **Hot runner**.

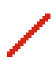
### Creating a runner system

1. Click  (**Geometry tab > Create panel > Nodes > Node by Coordinate**) to create the nodes that will define the feed system.

   ---
   **NOTE:** You can use any node creation tool.

   ---
   **TIP:** You do not have to create nodes first to create the gate and runners. You can create curves only, if you want.

   ---

2. Click  (**Geometry tab > Create panel > Curves > Create Line**).
3. Enter the start coordinate of the curve or select the node that defines the start of the curve.

   ---
   **TIP:** Set the Filter to Node or Nearest node to ensure you select a node rather than a point in space close to an existing node.

   ---

4. Select **Absolute** or **Relative** depending on how you want to define the end coordinate.
5. Enter the end coordinate of the curve or select the node that defines the end of the curve.
6. Select the appropriate property from the **Create as** drop-down list.

   If the required feed system property is not in the list, click **Change...** and select the required property.

7 Click **Apply**.

8 Repeat steps **3-7** for each curve of the feed system you are creating.

---

**TIP:** The second coordinate of the last curve created becomes the first coordinate of the next curve you are creating.

---

9 Duplicate the cavity and runners as necessary to reduce the entities being created manually.

10 Mesh the runners.

---

**TIP:** If you want to place the curves and beams for the feed system on the same layer, make sure to select the **Place mesh in active layer** option.

---

11 Click (**Mesh tab > Mesh Diagnostics panel > Connectivity** ) to check the connectivity of the runners with the cavities.

12 Click on the element at the top of the sprue, then click **Show**.

13 If there are disconnected mesh elements, click (**Mesh tab > Mesh Repair panel > Global Merge**).

---

**NOTE:** In order to avoid merging unnecessary nodes, make sure the **Merge nodes along an element edge only** option is not selected. After applying the tool, check to ensure only the necessary nodes were merged. if unnecessary nodes were merged, click (**Undo**), then reduce the tolerance and try the merge again, or fix the problem manually.

---

14 Click (**Home tab > Molding Process Setup panel > Injection Locations**) and set an injection location on the top of the sprue.

## Hot runners

These dialogs are used to set the properties of the hot runner, the mold block containing the hot runner and, depending upon your analysis, which stage of the overmolding process the hot runner relates to.

To access these dialogs, click **Geometry tab > Properties panel > Assign > Hot Runner**.

### Hot runner dialog—Runner Properties tab

The **Runner Properties** tab of the **Hot runner** dialog is used to specify the geometric and thermal properties of the hot runner.

### Hot runner dialog—Mold Properties tab

The **Mold Properties** tab of the **Hot runner** dialog is used to specify the properties of the mold block containing the hot runner.

### Hot runner dialog—Overmolding Component tab

The **Overmolding Component** tab of the **Hot runner** dialog is used to specify which stage/component in the overmolding process the hot runner relates to.

### Hot runner (3D) dialog—Runner Surface Properties tab

The **Runner Surface Properties** tab of the **Hot runner (3D)** dialog is used to specify the thermal properties of the hot runner.

### Hot runner (3D) dialog—Mold Properties tab

The **Mold Properties** tab of the **Hot runner (3D)** dialog is used to specify the properties of the mold block containing the hot runner.

### Hot runner (3D) dialog-Overmolding Component tab

The **Overmolding Component** tab of the **Hot runner (3D)** dialog is used to specify which hot runner elements belong to the first or second shot in an Overmolding analysis.

### Hot runner for Overmolding Second Component (3D) dialog

This dialog is used to specify the properties of the selected tetrahedral elements or region(s) of type **Hot runner for Overmolding Second Component (3D)**.

Use this property rather than the Hot runner (3D) property if the hot runner is used for the second component in the overmolding process.

To access this dialog to edit the properties of existing model entities, select at least one tetrahedral element or region of type **Hot runner for overmolding second component (3D)**, then either select **Edit > Properties**, or press **Alt-Enter**, or right-click and select **Properties**.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

### Insulating properties dialog

This dialog is used to specify the insulating properties that will be used to calculate the rate of heat transfer between the hot runner and the mold.

The **insulation layer conductance** value quantifies the ability of heat to pass between the hot runner and the surrounding mold material.

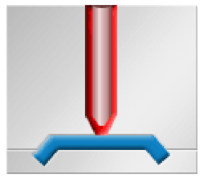
The **Air gap** value represents the space, in millimeters, between the insulated hot runner and the mold.

## Hot runners - heated runners

Heated runners are a type of hot runner in which heat is supplied to maintain the material in a molten state. Hot runner systems can be internally heated or externally heated.

**Internally heated runners**

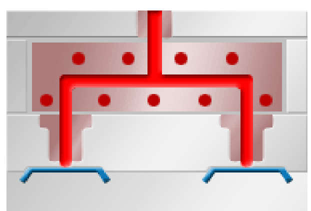
Internally heated runner systems have annular flow passages, with the heat being furnished by a probe and a torpedo located in the passages, as shown in the following diagram. This system takes advantage of the insulating effect of the plastic melt to reduce heat transfer loss to the rest of the mold.



The advantages of an internally heated runner system are improved distribution of heat and better temperature control. The disadvantages of this type of heated runner system are higher costs and a more complicated design.

**Externally heated runners**

Externally heated runner systems consist of a cartridge-heated manifold with interior flow passages, as shown in the following diagram. The manifold is designed with various insulating features to separate it from the rest of the mold, thus reducing heat loss.

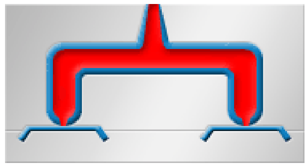


The advantages of an externally heated runner system is its sophisticated heat control. The disadvantages of this type of heated runner system are higher costs, a more complicated design, and the thermal expansion of various mold components that has to be taken into account.

## Hot runners - insulated runners

Insulated runners are oversized passages formed in the mold plate.

As shown in the following diagram, the oversized passages allow an open, molten flow path to be maintained because of the insulating effect of the plastic that freezes on the runner wall, combined with the heat applied to the system with each shot.



The advantages of an insulated hot runner system include simple design and low mold cost. The disadvantages of this type of hot runner system include the following:

- Unwanted freeze at the gate
- Fast cycle time needed to maintain melt state
- Long start-up periods needed to stabilize melt temperature
- Problems in uniform mold filling

# Runner systems for single cavity molds

When the design of the runner system and the position and number of injection locations is optimal, an evenly filled part will be produced. When the number and positioning of injection locations is not ideal, or the runner system is unbalanced, problems like overpacking and hesitation will occur.

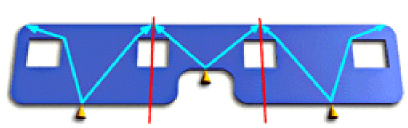
## Unbalanced flow in single cavity molds

When a part has complex geometry, it may require more than one injection location. You must ensure that the flow of the material is balanced to avoid molding problems.

There are two stages in balancing flow. The first stage is determining how many injection locations are needed and where they should be. The second stage is designing the runner system so that the part fills evenly.

### Injection locations

It is helpful when you are balancing flow paths to split the part into imaginary sections that will fill simultaneously. The following diagram shows a part that requires three injection locations to fill effectively. The three sections separated by the red lines show the sections will all fill at the same time. The pale blue arrows in the diagram indicate the flow paths of material, and the yellow cones indicate the injection locations.



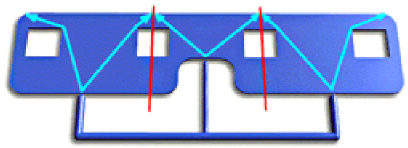
---

**NOTE:** When you design the sections in a mold, avoid putting section boundaries where weld marks would be undesirable.

---

### Runner system

The next stage is to design the runner system so that each section fills simultaneously. For a part with two equal sections, the runners should be the same diameter, length and distance from the sprue. For parts with unequal sections, the pressure in each runner and section should be the same and all the flow fronts should meet at the same time. The following diagram shows a runner system designed to finish filling all three sections at the end of fill.



## Overpacking in a single cavity mold

Overpacking occurs in complex parts that have more than one injection location when flow fronts meet before the part has filled.

In the following animation, the material from the central gate merges with the flow from the other two gates before the part has filled, causing underflow and overpacking. To overcome this type of overpacking, divide the part into imaginary sections in which the flow fronts meet at the end of fill. Design the runners to ensure that the pressure in each section is equal.

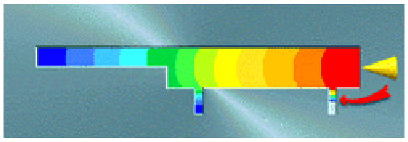
In the following animation, the gates have been moved so that each section of the mold fills simultaneously.

Runners can be designed to achieve balanced flow and prevent underflow by equalizing the pressure where the flow fronts meet. You can use the pressure results to observe the pressure in the cavity.

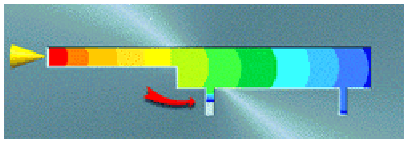
## Avoiding hesitation

Hesitation occurs in parts of various thicknesses when the melt moves preferentially into thicker areas and melt in the adjacent thin area lies stagnant. The stagnant melt looses heat while the thicker area continues to fill.

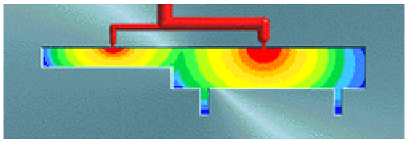
Hesitation can usually be avoided by using multiple injection locations with a balalnced runner system. The following diagram shows a part that requires multi-gating because of the two thin ribs in the design. If the gate was located as shown, hesitation would occur in the thin rib near the gate. The plastic in the rib would freeze of while the thick area is being filled. The hesitation, which is indicated by the red arrow, is caused by restricted flow.



The gating shown in the following diagram would only be marginally better because the polymer still flows more readily in the thicker section than the thinner section, causing hesitation in the thin section indicated by the red arrow.



The solution for this problem is using two gates with an artificially balanced runner system, as shown in the following diagram. The gates are positioned so that the thin ribs are at the end of flow paths, which prevents hesitation.



# Runner systems for multi-cavity and family molds

The runner system should be designed so that all the parts finish filling simultaneously.

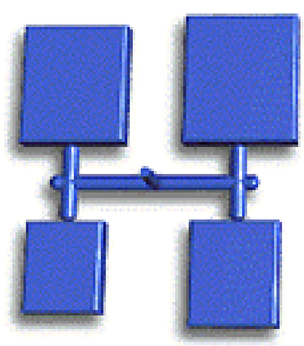
Unbalanced runners can result in hesitation, underflow, or overpacking. You might have to artificially balance the runners, or rearrange the cavities to create a naturally balanced system.

## Unbalanced flow in family molds

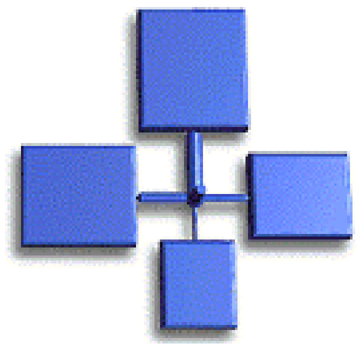
The runner system should be designed so that all of the parts finish filling at the same time.

Each part should be analyzed before you design the runners for a family mold. When you have confirmed that each cavity will fill, you can design the runner system to create balanced fill paths in each cavity. Unbalanced runners can result in molding problems such as hesitation, underflow and overpacking.

The following diagram shows an unbalanced family mold. The runners are all the same length ad diameter, but because the cavities are of different sizes, the flow will be unbalanced. The smaller part (bottom left) will fill first, resulting in overpacking, and the largest part will fill last.



The following diagram shows a balanced family mold. The smallest cavity has the thinnest runner, restricting plastic flow into it. This means that the four cavities will all fill at the same time, reducing the possibility of molding problems. You can use a Runner Balance Analysis to balance the runners.

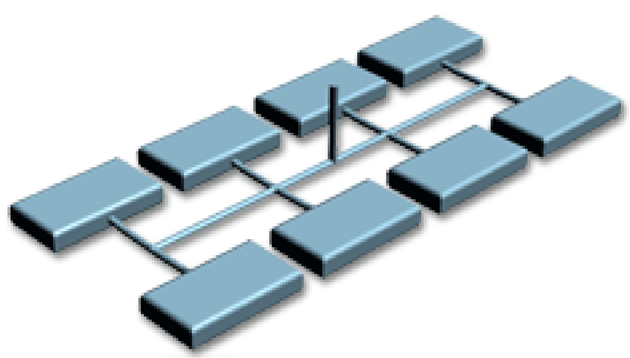


## Unbalanced flow in multi-cavity molds

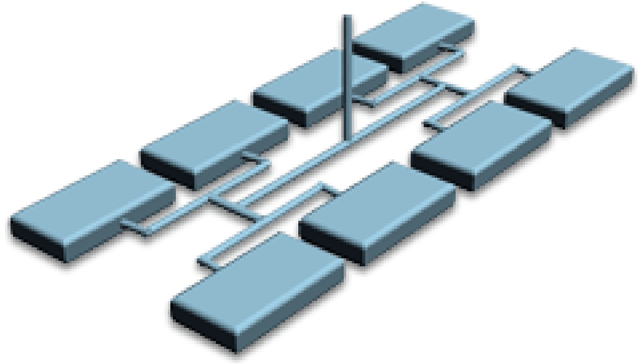
It is important for the feed system in multi-cavity molds to be balanced so that the plastic melt fills each seperate mold cavity simultaneously.

Non-uniform fills can result in some cavities producing good parts and other cavities producing inferior parts due to short shots, overpacking or flash.

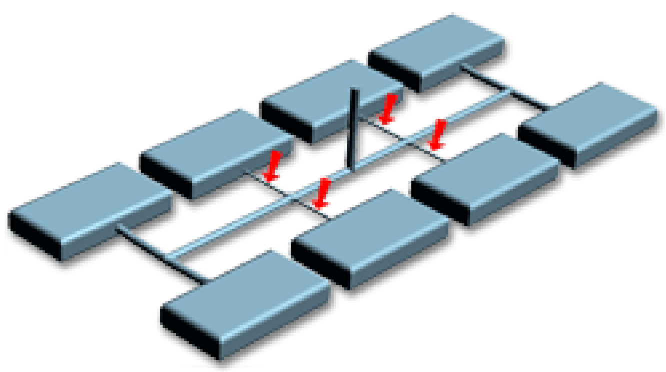
Before designing a multi-cavity mold, you should analyze each cavity without the runner system. When you know how each cavity will fill, you can design the runner system to create balanced fill paths. The following diagram shows a conventional way of filling a multi-cavity part, which may cause molding problems because the flow path for the outer parts is much longer than the flow path for the inner parts. In this example, the runners must be balanced.



In the following diagram, the runners in this naturally balanced runner system all have the same flow length which means that the polymer will reach the gate of each part simultaneously.



Another method of balancing flow paths is to use artificial runner balancing, where the runners have different diameters to promote flow to the more distant cavities. If you have a conventional runner layout, you can artificially balance the runners using this method.



## Overpacking in multi-cavity and family molds

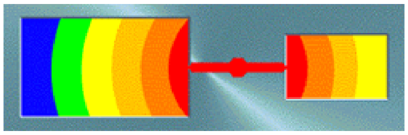
Overpacking in multi-cavity or family molds occurs when one part fills before another.

Overpacking might occur because:

- One part is significantly larger than the other
- The runners are unbalanced, so the flow lengths vary in parts with a similar size.
- The pressures at the injection location of each part vary.
- The runner system is poorly designed.

Overpacking can be avoided in multi-cavity or family molds by ensure that each part fills correctly as a single part. Once you have found the best injection location for each individual cavity, design the runner system so that all parts finish filling simultaneously. The Fill Time result will help you do this.

In the following diagram, the smaller part will fill first and will overpack because pressure must be maintained to fill the larger part.

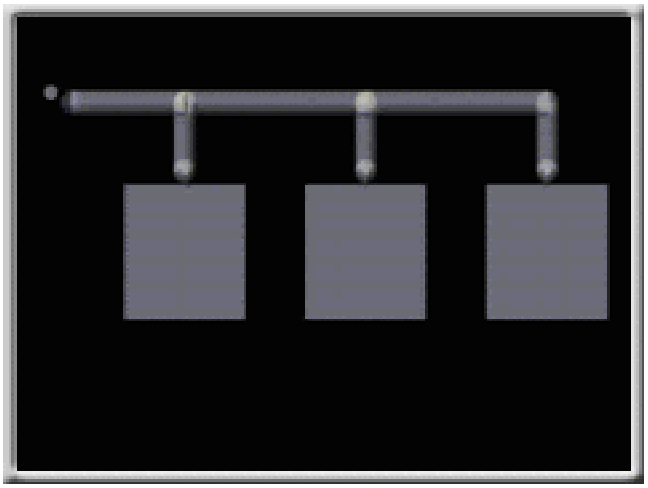


## Hesitation at the gates of multi-cavity and family molds

When designing a mold for a multi-cavity part, ensure the flow of the material is balanced.

With unbalanced runners, hesitation can occur in the gates of the parts closest to the sprue. Hesitation causes underpacking and short shots.

In the following animation, the material reaches the first gate before the runner system has filled. The higher pressure at the gate will cause the polymer to flow more readily through the runners. Polymer flow will slow at the first gate and then freezes, as indicated by the red arrow.



To avoid hesitation at the gates, the runners should be balanced so that the flow reaches the gates simultaneously. You can use the Fill time result to ensure that the flow paths are balanced. Hesitation at the gates might lead to underpacking or a short shot.