Autodesk® Moldflow® Insight 2012

AMI Injection Locations

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

Injection locations 1

The injection location represents the position where polymer is injected, enabling the software to simulate the flow pattern inside the mold cavity.

This topic describes the considerations affecting the selection of injection locations when running a Fill+Pack analysis.

To mold the best part possible, you must identify the optimum injection location. This will create a balanced filling pattern by allowing the extremities of the mold to fill at the same time and under the same pressure. You must also select the number of injection locations required and their positions so that the volumetric shrinkage at the end of the flow is close to the design value.

NOTE: There is no gate size associated with the injection cone; it simply represents the mathematical starting point of an analysis. A gate should be modeled to ensure accurate results.

If the initial analysis indicates that the fill pattern is unbalanced, you should alter the injection location, or add another one, to solve the problem.

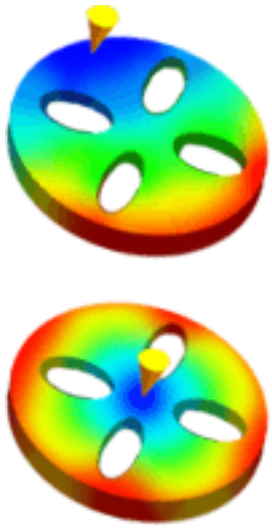


Figure 1: Unbalanced flow (above) Balanced flow (below)

It may be necessary to use several injection locations, or to even change the type of gate being used, for example, to an edge gate, to produce more uniform orientation effects in the product.

Gate Location Analysis

The Gate Location analysis is used to identify the most suitable injection location to create a balanced filling pattern.

Using the Advanced Gate Locator algorithm, the Gate Location analysis determines the best locations for the specified number of gates. Prohibited gate location regions are excluded from the analysis.

NOTE:

- You can iterate your use of the Gate Location analysis to find the next best injection locations as many times as required until a balanced flow path is determined by the proceeding Fill+Pack analysis.
- A new study is created at the end of the Gate Location analysis using the Advanced Gate Locator algorithm with the injection locations added to the model. The new study is ready for a Fill or Fill+Pack analysis.

Using the Gate Region Locator algorithm, if there are no injection locations specified, the Gate Location analysis will determine the best place for a single gate, given the selected material. If one or more injection locations already exist, the result suggests the best place for the next gate location given the selected material.

Gas-assisted Fill+Pack analysis

To use the Gas-assisted Fill+Pack analysis, you must identify the node at the gas injection location. Gas, like the molten polymer, always flows toward the area of lowest pressure. Therefore, select the gas injection location to ensure that the gas stays in the gas channel, and that the area of lowest pressure is near the end of the gas channel.

Injection locations

The injection location represents the position where polymer is injected, allowing the software to simulate the flow pattern inside the mold cavity.

Setting injection locations

An injection location defines the location where polymer is injected into the mold cavity, and is specified on a model node. An injection location on the part model is a compulsory input for a Fill analysis.

1 Ensure that you have done the following first:
   a Meshed and checked the part model.
   b Set the molding process.
   c Selected the analysis sequence.
   d Performed the required analysis setup tasks.
   e Selected a material.

2 Click (**Home tab > Molding Process Setup panel > Injection Location**), or double-click the injection location icon in the **Study Tasks** pane.

Cross-hairs appear, indicating that the injection location mode is activated.

3 Click the cross-hairs on the model where polymer will be injected.

A cone appears, indicating that the injection location is successfully set on the model.

NOTE: A Fill Preview of the part will appear.

4 Right-click the cross-hairs on the injection location cone and select **Finish Set Injection Locations**.

5 Click from the Quick Access toolbar to save the injection location on your model.

NOTE: If you are unsure of the best injection location, or the number of injection locations to set, first run the Gate Location analysis.

Moving an injection location

1 Click (**Home tab > Molding Process Setup panel > Injection Location**). Cross-hairs will appear.

2 Place the cross-hairs over the injection cone.

3 Click and drag the injection cone to the correct position.

4 Right-click the mouse button and click **Finish Set Injection Locations** to set the injection location and exit the tool.

NOTE: To delete an injection location, select the injection cone and click (**Geometry tab > Utilities panel > Delete**).

Deleting injection locations

An injection location specifies the location where polymer is injected into the mold cavity. If you have set the injection location(s) in the wrong position, or want to remove an injection location, use one of instructions below to delete it.

1 Ensure that the (**Select**) tool is activated.

2 Do one of the following:

- Click **Edit** (**Action History**), click **Injection Locations**, and then click **Undo**.

- Select the injection cone, and then right-click and select  (**Delete**).
- Select the injection cone, and click  (**Delete**) in the **Standard** toolbar.

Injection locations

This dialog is used to set the injection location coordinates exactly.

Injection Location Properties dialog

Use the **Injection Location Properties** dialog to set the coordinates for your injection point(s).

To access this dialog, select an injection cone, and then right-click and select  (**Properties**) from the context menu.

Enter a value in the X, Y and/or Z text box to reposition the injection point.

Design rules of polymer injection locations

2

The positioning of injection locations greatly affects material orientation and part warpage.

In some cases, changing the gate position is the only way of controlling the orientation effects and producing a satisfactory design.

Each polymer injection location injects the material under the same pressure. Unless an end-of-fill spike occurs, the pressure increase during the injection period is generally linear. The aim of proper gating is to avoid problems associated with overpacking, such as variation in shrinkage and product sticking in the cavity.

Design Rules

Gate centrally to provide equal flow length

Centralized gates provide equal flow lengths to all extremities of the part. This results in more even packing in all directions and a lower shrinkage difference, which leads to a higher quality part and lower rejection rate.

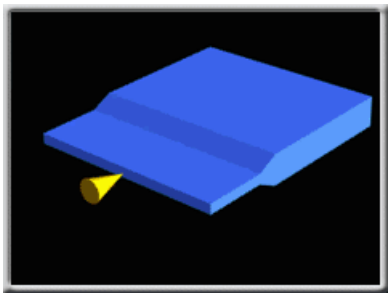
Gate symmetrically to avoid warpage

Symmetrical parts should be gated symmetrically to achieve balanced flow and avoid differential shrinkage and subsequent warpage of the part.

Gate into thicker sections for better filling and packing

Place polymer injection locations in thicker regions of the part, preferably at a spot where the function and appearance of the part are not impaired. This causes the material to flow from the thickest areas to the thinnest areas, and helps maintain the flow and packing paths. Gating into thinner sections can result in hesitation or sink marks and voids.

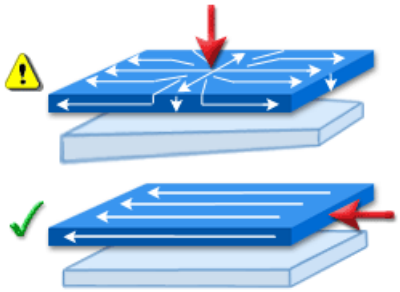
The following animation shows how restrict material flow through a thin section can lead to material flow problems.



Gate long, narrow parts from an end for uniform flow

When a long narrow part is centrally gated, packing near the gate and variable molecular or fiber orientation throughout the part will cause differential shrinkage, which will warp the part. Gating a long part at one end will provide a uniform molecular and fiber orientation in the length direction. Although the end with the gate will be packed more than the opposite end, the resulting difference in shrinkage will not cause warpage.

The following figure shows the prefered injection location.



Position the gate away from load-bearing areas

The high melt pressure and high velocity of flowing material at a gate causes that area to be highly stressed. For this reason, you should locate the gate away from load-bearing areas.

Hide the gate scar

The removal of the gate will leave a mark on the part, which could be visually unacceptable. Place the gate so that the gate scar can be hidden or easily disguised.

Vent properly to prevent air traps

The gate location should prevent air traps by enabling the air in the cavity to escape during injection. Failure to vent the air will result in a short shot, a burn mark on the molding, or high filling and packing pressure near the gates.

Gate for proper weld-line location and strong weld lines

The gate location should cause weld and meld lines, if any, to form at appropriate positions that are not objectionable to the function, external load, or appearance of the part. Place the gate so that weld lines are formed early in the filling phase or at a high pressure area to ensure proper melding of the weld lines.

Multiple gates shorten flow lengths

Add gates so that flow paths are within the flow limits for the thickness, distance, and process conditions. Each gate should have equal flow rates and volumes.

Locate gates on either side of a weak core or insert

To reduce core shift (core deflection), the gate location should enable balanced flow and pressure distribution on all sides of the core or insert.

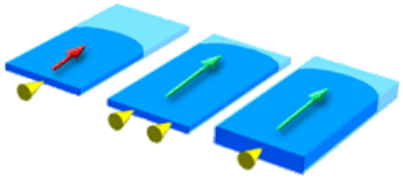
Number of polymer injection locations

3

Because each part is different, there are no specific rules for determining the number of polymer injection locations; however, there are some general factors that must be considered.

Flow length

Flow length refers to how far the polymer must flow from the polymer injection location. Generally, parts with thicker walls can have longer flow paths than thin-walled parts because the material will flow more easily in the thicker regions, as shown in the following diagram.



The flow characteristics of the plastic material affect how far a material will flow for a given thickness. The shorter the flow length, the more gates required to fill the part. Each material has its own flow length. The materials datasheets from material suppliers contain information about flow lengths that can be achieved for each specific material at a range of thicknesses.

Very large parts, thinner walled parts and higher viscosity materials will typically require more gates to fill a part.

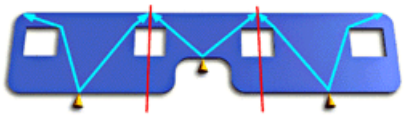
Part volume

Generally, parts with larger volumes require more polymer injection locations to fill properly.

Selecting the correct number of injection locations

First, try a single gate in the flow centroid of the mold, and check that all flow paths fill at nearly the same instant in time.

If this criteria cannot be met with a single gate, then try using multiple gates. Mentally, divide the part mold into sub-moldings, with a gate positioned at the flow centroid of each sub-molding or in the middle of one side. The runner system dimensions should be set up so that each sub-molding is filled at nearly the same instant in time, as shown in the following diagram.



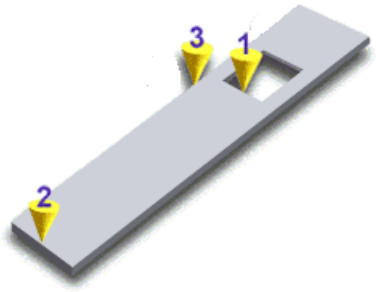
The positions of the gates should achieve both uniform and acceptable values of shrinkage across the part. Where thick and thin sections are present, it is good practice to position the gate nearer to thicker sections. This avoids insufficient packing, which is caused by premature freezing of the material between the gate and the thicker regions.

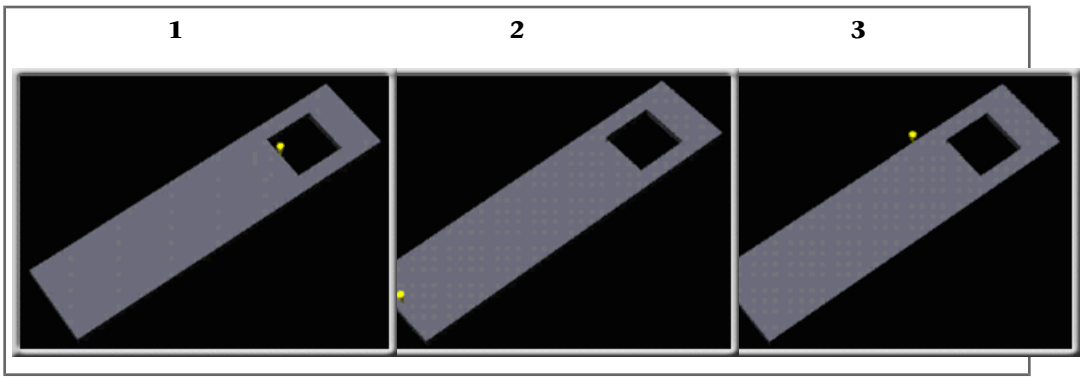
Selecting polymer injection locations

4

Because injection locations greatly influence the way in which the plastic flows into the mold cavity, their postioning directly affects part quality.

One of the goals when selecting polymer injection locations is to ensure that all flow paths in the cavity fill at the same time. This prevents overpacking along the flow paths which might otherwise fill first. The three possible polymer injection locations in the following diagram of a model show how the polymer injection location can be used to help balance flow.





The polymer injection location can also be used to change the position of weld lines and air traps, and reduce hesitation and other molding problems. The above animation shows how polymer injection locations 1 and 2 cause a weld line to form on the right of the model, and polymer injection location 3 causes a weld line to form on the bottom right of the model. In some cases it is necessary to have more than one polymer injection location. Other methods can be used to help balance flow paths, such as including flow leaders or flow deflectors.

Injection location temperature during packing and cooling

5

The injection location temperature moves toward the mold temperature during the packing and cooling phases because there is negligible flow from the barrel during these phases.

The injection location temperature affects the flow of the melt into the part during packing, usually by decreasing the flow. To achieve realistic warpage values, it is important to take the injection location temperature into account during the packing and cooling phases.

NOTE: The injection location temperature cools when using thermoplastic materials and heats when using thermoset materials.

Two different calculations are used depending on the type of model that you are analyzing.

Midplane and Dual Domain The initial melt temperature constraint at the injection location is ignored when filling is complete. The injection location temperature change is calculated by using the mold temperature as well as convection and shear heating information.

3D The mold temperature and the cylindrical diameter of the injection location are used to calculate the injection location temperature change during packing and cooling. The injection temperature at the end of fill is used for the initial temperature in the calculation, and only heat conduction is considered.

When injecting into a beam element, the calculation uses the diameter of the beam element. When injecting directly into the cavity, the contact diameter of the gate is used. Smaller injection location or beam diameters result in faster temperature changes.

NOTE: You can use an automatic gate contact diameter or set a specific gate contact diameter when injecting directly into the cavity.

Injection locations on 3D mesh 6

The injection location represents the position where polymer is injected, allowing the software to simulate the flow pattern inside the mold cavity.

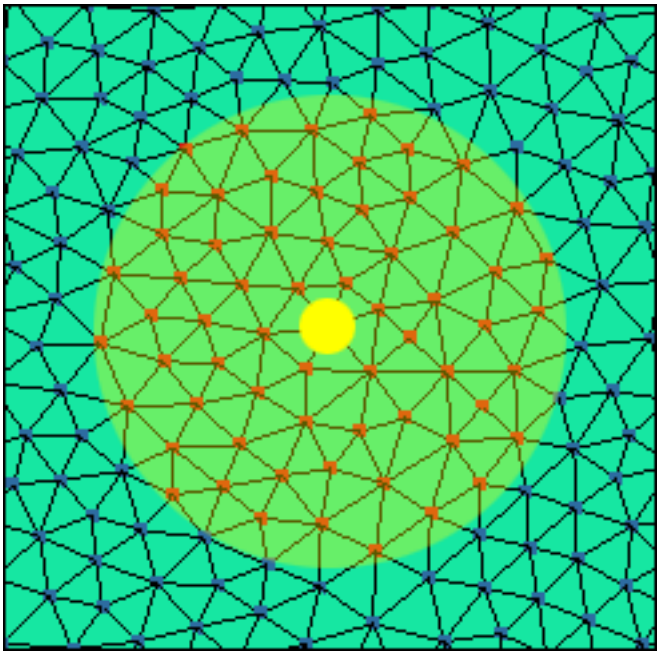
This help topic describes some injection location considerations when running a Fill+Pack analysis on a 3D model in Autodesk Moldflow Insight.

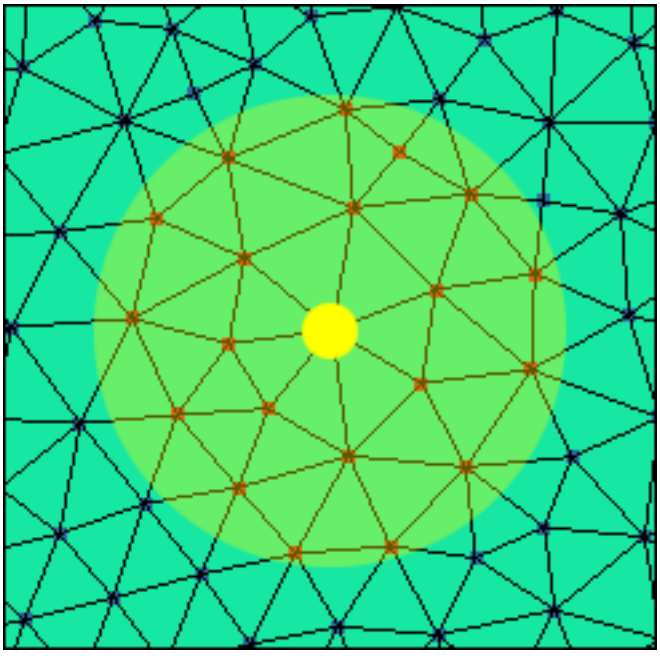
3D Fill+Pack analysis

When you select an injection location on a tetrahedral mesh, the location is associated with a node on the mesh. When a Fill+Pack analysis is performed with no gate modeled, the gate size 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.

TIP: The gate diameter can also be manually specified from the **Solver Parameters** dialog.

The actual area used for material injection is defined by the number of nodes which fit inside the virtual gate diameter. If a fine mesh is used, the actual injection area will be closer to the virtual gate size than for a coarse mesh. For example, in the first image below the fine mesh size fits well into the virtual gate diameter, but the second image shows that a coarse mesh could produce a smaller than expected gate size.





NOTE:

- If you are using a runner system that is modeled using 1-D beam elements, you need to specify only one injection point at the beginning of the sprue or the gate.

- If you are using a runner system that is imported from a CAD model and meshed as tetrahedral elements, you need to specify each of the internal nodes on the polymer entrance face of the sprue (or the gate, if only the gate is modeled). This way, the plastic will be injected through the entire face of the sprue (or gate).

  NOTE: Do not specify the nodes along the edge of the sprue tip or gate tip, because the elements on the side will also be included in the gate region.

Injection time 7

The injection time is the time it takes for the mold to fill completely.

When you set up a Fill+Pack analysis sequence, the software can be set to either calculate a machine injection time automatically, or on the basis of a user-specified value. By default, the injection time is calculated automatically.

NOTE: To specify how an injection time is calculated, go to **Home tab > Molding Process Setup panel > Process Settings** . From the **Filling Control** drop-down list, either select **Automatic** to let the software automatically calculate the injection time, or select **Injection Time** and enter a numeric value to specify the injection time.

Automatic injection time

If you set the injection time to Automatic, the analysis finds the injection time which gives the lowest injection pressure. The following graph shows the results from nine analyses on the same part. The blue points represent the analyses where the injection time was set to a particular value. The red point represents the analysis where the **Automatic injection time** check box was selected, which shows the lowest possible injection pressure for the part.

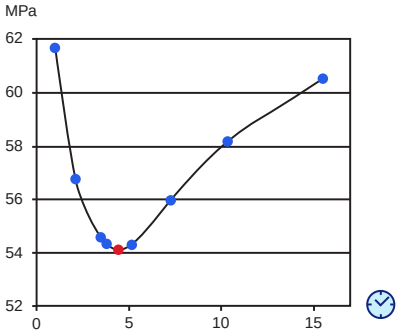


Figure 2: Injection pressure as a function of time

The variation of injection pressure against injection time has two influences. Firstly, as the injection time increases from zero, the pressure required to force the molten plastic through the part decreases. Secondly, as the injection time increases, the polymer temperature decreases due to heat transfer to the mold, which causes the viscosity and frozen layer thickness to increases, which in turn increases the injection pressure.

Specified injection time

If a specified injection time is entered, the Summary lists an actual injection time that is slightly higher than the value entered. The extra time is to account for material compressibility.