Autodesk® Moldflow® Insight 2012

AMI Recommended modeling details

Contents

Modeling a midplane fan gate 1

A fan gate is a wide edge gate with variable thickness which permits rapid and balanced filling of large parts through a large gate cross-section.

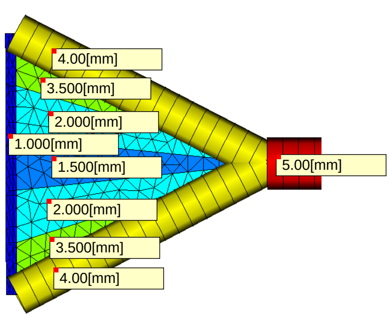
NOTE: For a Dual Domain or 3D model, it is strongly recommended that you model the gate directly onto the part in the CAD application and import both the part and the gate as one entity into Autodesk Moldflow Insight. If you already have a Dual Domain mesh, you can model the fan gate by itself in the CAD application, and add it to your existing part. If this is not possible, use the instructions to model a Midplane fan gate.

For a Midplane model, a fan gate can be modeled as an extension of the part. First create the outline of the fan gate by adding new nodes, and then create regions within the gate. The regions are assigned different thicknesses so that the gate has variable thicknesses.

Because of these thickness variations, the easiest way to model a fan gate is to use a 3D mesh. If a 3D mesh cannot be used, thickness changes are best detected when the following conditions exist:

- For a Dual Domain mesh, the thickness to width ratio of the cross section is greater than 4:1.
- For a Midplane mesh, the fan gate is modeled with triangular elements and beams.

  The following diagram shows a fan gate modeled with triangular elements and beams with thickness variations.

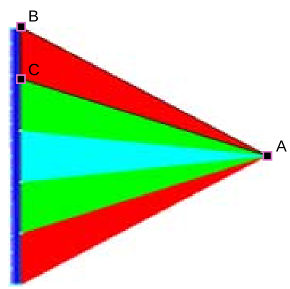


The gate land should be modeled with 3 rows of elements, and the center portion of the gate should be the thinnest section. The thicknesses required depend on the size and shape of the gate, the material being molded, and the injection time. When modeling the gate land, it is critical that all existing nodes of the part be used in the definition of the gate, so the gate and part are attached.

NOTE: A Midplane gate can be added to a Midplane or Dual Domain model, but you will not be able to run Cool or Warp analyses with the Dual Domain analysis because mixed mesh types are not supported on these solvers. If you have to run these analyses, please use the CAD program to model a Dual Domain fan gate.

To model a Midplane Fan Gate:

1. Use the **Nodes** tools (**Geometry tab > Create panel > Nodes**) to define the shape of the fan gate.
2. Click (**Geometry tab > Create panel > Regions > Region By Nodes**).
3. Click on the first node of a region, hold down the control key and select the rest of the nodes forming that region in sequence. In the following diagram, nodes A, B, and C have been selected to form a region.



4. Click (**Geometry tab > Properties panel > Assign**).
5. Click **New** and select **Cold gate surface (midplane)** from the drop-down list.
6. Enter an appropriate thickness for the fan gate region in the **Thickness** text box.
7. Repeat Steps 3-6 for each region in the fan gate, so that the fan gate is thickest at the injection point and thinnest where it joins the part.

   If additional nodes were created at the gate/part edge, use the stitch tool to connect the gate and the part.

   TIP: You can create curves on the fan gate sides to emphasize their thickness.

8. Click (**Mesh tab > Mesh panel > Density**) and enter an appropriate mesh density, such that each region in the gate will have at least three rows of triangles.
9. Click (**Mesh tab > Mesh panel > Generate Mesh**) and click **Mesh Now**. The option to remesh already meshed parts of the model should not be used to ensure the new regions have the correct density.

How plastic fills a mold

2

There are three distinct processing phases in injection molding.

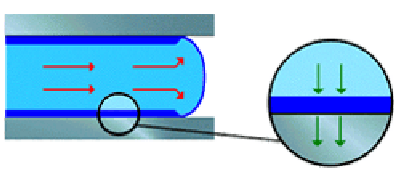
The three phases in the molding process are as follows:

- Filling phase
- Packing phase
- Cooling phase

1. Filling phase

During the filling phase, plastic is pushed into the cavity until the cavity is just filled. As plastic flows into the cavity, the plastic in contact with the mold wall quickly freezes. This creates a frozen layer of plastic between the mold and the molten plastic. At the interface between the static frozen layer and the flowing melt, the polymer molecules are stretched out in the direction of flow. This alignment and stretching is called orientation.

The following diagram shows how the flow front expands as material from behind is pushed forward. This outward flow is called fountain flow. The edges of the flowing layer freeze as they come into contact with the mold wall in a near-perpendicular direction. The molecules in the initial frozen layer are therefore not highly orientated, and when they are frozen, the orientation will not change.



The red arrows in the diagram show the flow direction of the molten plastic. The dark blue layers show the layers of frozen plastic against the mold walls. The green arrows indicate the direction of heat flow from the polymer melt into the mold walls.

The frozen layer gains heat as more molten plastic flows through the cavity, and loses heat to the mold. When the frozen layer reaches a certain thickness, equilibrium is reached. This usually occurs early in the injection molding process, after a few tenths of a second.

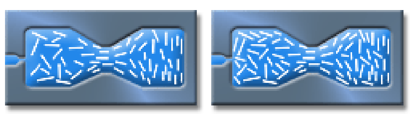
2. Packing phase

The packing phase begins after the cavity has just been filled. During this phase, further pressure is applied to the material in an attempt to pack more material into the cavity. This is intended to produce a reduced and more uniform shrinkage with reduced component warpage.

When the material has filled the mold cavity and the packing phase has begun, material flow is driven by the variation of density across the part. If one region of a part is less densely packed than an adjacent region, polymer will flow into the less dense region until equilibrium is reached. This flow will be affected by the compressibility and thermal expansion of the melt in a similar way to which the flow is affected by these factors in the filling phase.

The pvT (pressure, volume, temperature) characteristics of the material provide the necessary information to calculate parameters such as density variations with pressure and temperature, compressibility, and thermal expansion data. When combined with the material viscosity data, an accurate simulation of the material flow during the packing phase is possible.

The following diagram shows the difference between the end of the filling phase (left) and the end of the packing phase (right).



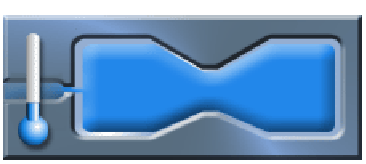
In practice, due to the limitations of pressure and available unfrozen flow channel, it is not possible to pack enough material into the mold to fully compensate for shrinkage. The uncompensated shrinkage must be allowed for by making the cavity bigger than the desired part size.

3. Cooling phase

Although the cooling of the plastic occurs from the commencement of the filling phase, the cooling phase is the time from the end of packing to the opening of the mold clamps. This phase is the extra time that is required to cool the part sufficiently for ejection. This does not mean that all sections of the part or runner system have to be completely frozen.

The material at the center of the part reaches its transition temperature and becomes solid during cooling time.

The rate and uniformity at which the part is cooled affects the finished molding quality and production costs. Mold cooling accounts for more than two-thirds of the total cycle time in the production of injection molded thermoplastic parts.



Cycle time

3

Cycle time is the total time required to complete all the stages of the injection molding cycle.

The cycle time is made up of the following stages:

Fill time. The time required to fill the mold with polymer. The injection molding machine controls the velocity (flow rate) of the molten polymer entering the mold during this stage of the cycle.

Packing time The stage of the injection molding cycle when pressure is applied to the polymer melt to compress the polymer and to force more material into the mold. This compensates for the shrinkage that occurs as the polymer cools from the melt temperature to ambient (room) temperature. From 5 to 25 percent more material can be added to the mold during the packing stage. The gate should freeze during the packing time to prevent material from exiting the mold.

The Packing time is also known as the Holding time.

Cooling time. The cooling time is the stage of the injection molding cycle when there is no more pressure being applied to the polymer. The mold is held shut and the polymer continues to cool until the part can be ejected. The cooling stage is normally the longest part of the molding cycle and can account for up to 80 percent of the total cycle time.

Mold open time. The time for which the mold is open before the next molding cycle begins. This time includes the following:

- Opening the mold
- Ejecting the part
- Preparing for the next cycle, such as loading inserts (not always part of the cycle)
- Closing the mold

4 How thickness affects flow

Molten plastic will preferentially flow through thicker sections of a mold, but subtle changes to the part geometry can help to balance the flow.

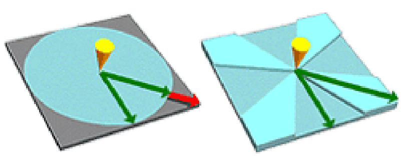
Flow leaders and deflectors

A **flow leader** is an increase in thickness along a flow path to increase the rate of flow along that path.

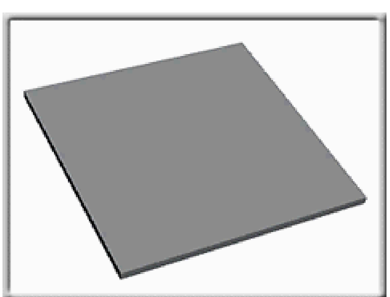
A **flow deflector** is a decrease in thickness along a flow path to decrease the rate of flow along that path.

Flow leaders and deflectors can be used to ensure all flow paths within the cavity fill at the same time to achieve balanced flow paths. Often the most suitable polymer injection location will not define equal flow paths, and the use of multiple polymer injection locations creates extra unwanted weld lines. Altering thicknesses within the design specifications is then the most appropriate way to balance the flow paths.

The following diagram of a square plate of uniform thickness with a polymer injection location in the center will demonstrate flow. The part on the left shows a radial flow pattern with unbalanced filling causing areas that fill early to be overpacked, which results in distortion problems. In the part on the right, the thickness of the plate has been increased from the center to the corners of the part, decreasing the resistance to flow in these directions



The following animation shows how changing the thicknesses in some sections of a part can create a more balanced flow pattern. The flow leaders and deflectors will create a fill pattern that is closer to a balanced flow, and additional refinements can be made to balance the flow further.



NOTE:

- Where possible, use flow deflectors instead of flow leaders to minimize the weight of the part.
- Shrinkage is a function of thickness so always consider the effects that changes in thickness might have on warpage.

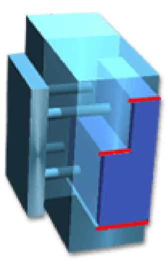
Tapering walls

5

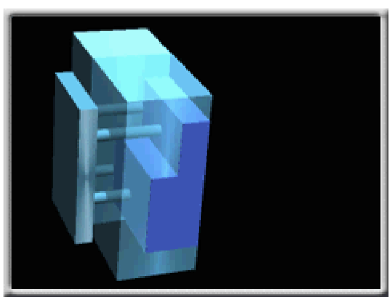
Some mold surfaces must be tapered so that the plastic part can be ejected from the mold when it has cooled sufficiently.

Untapered mold walls

The walls of the mold in the following diagram, which are marked with red, are not tapered. When the ejector pins push the finished part out of the mold, the force applied must overcome the friction between the mold wall and the plastic part.

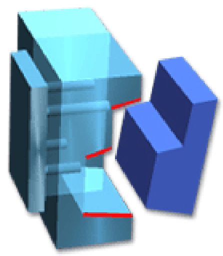


As shown in the following animation, when the mold walls are not tapered, the frictional resistance continues throughout the ejection process. Plastic parts that do not have tapered walls can be impossible to eject from the mold. Even if the part is ejected, the surface can be scuffed in the process, making the part visually unacceptable.

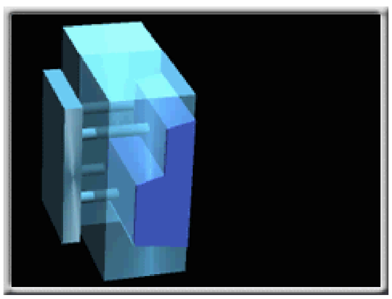


Tapered mold walls

The walls of the mold in the following diagram, which are marked in red, are tapered.



As shown in the following animation, when the ejector pins push the finished part out of the mold, there is an initial resistance due to friction which then is reduced to zero when the part is moving. Depending on the surface finish of the part, a draft angle of 1.5º for highly polished surfaces up to 6º-8º for a leather like surface will enable the part to be easily ejected from the mold.



Calculating the machine intensification ratio

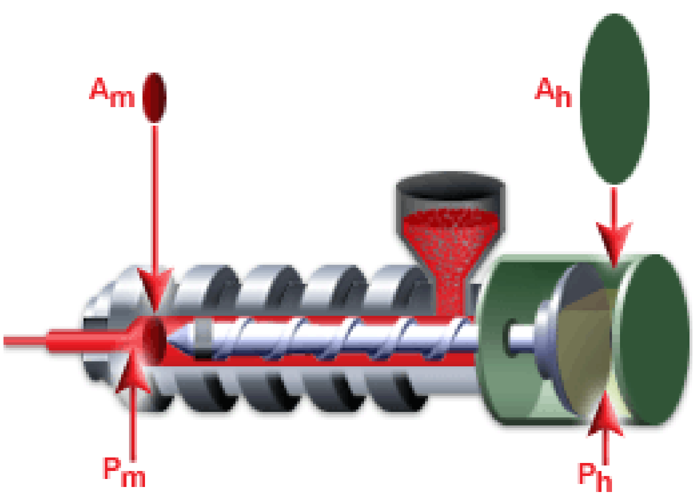
6

The machine intensification ratio, or the material/hydraulic pressure ratio is the direct relationship between the injection pressure and the hydraulic pressure. This is the ratio of the resin pressure in front of the screw, compared to the oil pressure in the piston of the injection molding machine.

The injection pressure and hydraulic pressure differ significantly in an injection molding machine. The injection pressure is the pressure applied directly to the plastic by the ram, which causes the material to flow. The injection pressure can be measured directly by locating a transducer in the nozzle. The hydraulic pressure is the pressure in the main supply line from the pump, which moves the ram; it is typically measured by means of a gauge in the hydraulic line.

Refer to the machine manual to find the machine intensification ratio. A typical ratio is 10, and the typical range of the ratio is between 7 and 15.

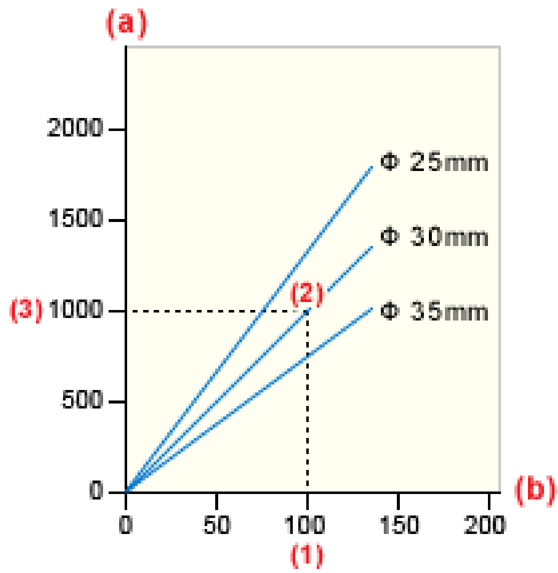
To calculate the machine intensification ratio, divide the piston area (A_h) by the screw area (A_m), as shown in the following diagram.



where:

- A_h is the piston area in the hydraulic oil
- A_m is the screw area in the melt
- P_h is the pressure in the hydraulic oil
- P_m is the pressure in the melt
- $P_m = P_h (A_h / A_m)$

The following graph is an example of a machine intensification ratio graph in a machine manual. In this example, several screws are used on one machine: 25mm, 30mm, and 35 mm. The ratio depends on the screw area; therefore, to calculate the ratio, choose a round value, for example 100 bar, on the pressure axis (1), project to the curve of the screw used on the machine (2), and then project to the injection pressure axis (3), divide the injection pressure (3) by the hydraulic pressure (1) to find the machine intensification ratio.



NOTE: Autodesk Moldflow Insight results show the injection pressure, not the hydraulic pressure. Multiply the hydraulic pressure by the machine intensification ratio to obtain the injection pressure, neglecting losses. Autodesk Moldflow Insight allows you to specify the maximum machine hydraulic pressure in addition to the machine intensification ratio. Multiplying these two values gives the maximum injection pressure for the machine in the simulation.
