Autodesk® Moldflow® Insight 2012

AMI The Mold

Autodesk®

Revision 1, 22 March 2012.

Contents

Mold

1

A mold is a series of machined steel plates containing a cavity or cavities into which molten plastic is injected at high pressure. When the plastic cools, it solidifies into the shape of the cavity defined by the steel plates.

Mold

The mold temperature can be defined using the mold surface temperature controller dialogs.

Editing the Mold surface temperature profile dialog

To access this dialog:

1. Click **Tools tab > Databases panel > New** to open the **New Database** dialog.
2. Select **Geometry/Mesh/BC** from the **Category** drop-down menu, and select **Cold runner** from the **Property type** menu. Click **OK**.
3. Click **New** in the **Properties** dialog to open the **Cold runner** dialog.
4. Select the **Mold Temperature Profile** tab and click **Select** to open the **Select Mold surface temperature profile** dialog.
5. Choose a profile and click **Select**.
6. Click the **Edit** button to open the **Mold surface temperature profile** dialog, and make appropriate edits.
7. Click **OK** to accept the edits and close the dialog.

Mold

The Mold Properties dialog is used to set properties of the mold.

Mold Block Surface dialog

This dialog is used to edit the properties of elements or regions of type **Mold block surface**.

To access this dialog to edit the properties of existing model entities, select at least one element or region of type **Mold block surface**, 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.

Mold material	Specifies the mold material in contact with the selected model feature.

Mold surface temperature profile dialog

Specify how the mold surface temperature varies over time.

To launch this dialog, select the area to be profiled on the surface of the model. Right-click the selected area and select **Properties** from the menu that appears. On the **Part Surface Properties** tab, select **Profile** from the drop-down menu in the **Mold surface temperature** pane and click **Select**. Click **Edit** from the dialog that appears.

Up to 50 pairs of time/temperature data can be used to specify the required profile.

Mold surface temperature profile	Specifies the mold temperature value to be assigned to the selected element(s). The specified mold temperature values will override the values set in the Process Settings Wizard and on the Temperature control tab of the Process controller dialog accessed via the Advanced Options.

NOTE: When defining a profile, the time component must be entered in ascending order.

NOTE: A profile can have temperatures that increase and decrease, but if the temperature drops below the transition temperature (Ttrans) of a thermoplastic material, it must remain below this value for the rest of the profile.

NOTE: The title entered in the **Name** text box will be used in the **Select mold temperature controller** dialog when allocating a profile to an area.

Select mold temperature controller dialog

Select or modify a predefined local mold surface temperature profile, or define a new profile.

To access this dialog, select the area on the surface of the model where the temperature profile is to be applied. Right-click the selected area and select **Properties**. On the **Part Surface Properties** tab, select **Profile** from the **Mold surface temperature** list, and click **Select**.

The **Mold temperature profile** list presents previously defined profiles. Select a profile from the list and, if required, edit and rename the profile to be applied to the selected area of the model.

- The **Edit** button allows you to modify the selected profile. Use this option to define a new profile by saving the modified profile to a new name.
- The **Select** button allows you to select a profile to be edited, exported, searched or displayed.

NOTE: Temperature profiles are not supported for Underfill Encapsulation analyses.

Cavity/Core Mold Temperature Settings dialog

This dialog is used to individually specify the mold temperatures for the cavity and core side of the mold . This allows the flow simulation to account for asymmetric cooling effects across the cavity thickness.

NOTE: This dialog is accessed by setting the Mold temperature control option to **Cavity differs from core** on the **Temperature Control** tab of the Process Controller dialog (via Fill+Pack analysis advanced options).

Preconditioning data dialog

This dialog is used to specify the data that will be taken into account for the calculation of the temperature of the molten plastic, or melt, as it starts to flow into the cavity.

To access this dialog, ensure that you have selected one of the Reactive Molding processes and an analysis sequence that includes Fill+Pack, click  (**Home tab > Molding Process Setup panel > Process Settings**), then click **Edit Data...** in the **Preconditioning analysis** section of the dialog.

Clamp force

2

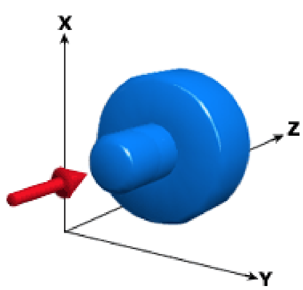
The clamp force is the maximum force required to keep the mold closed during filling. This force can be calculated.

The clamp force predictions for a part can then be compared with the clamp force limit of the injection molding machine to be used.

Clamp force calculation

The calculated maximum clamp force is a function of injection pressure and the projected area of the part. The projected area is the area of the model that is projected onto the XY plane. For the clamp force calculations to be correct, the model must be positioned so that the clamp force is applied along the Z axis direction as shown in the following diagram.

NOTE: Although clamp force planes can be changed from the default Z axis using the custom plots wizard, all solver calculations are still based on the Z direction.



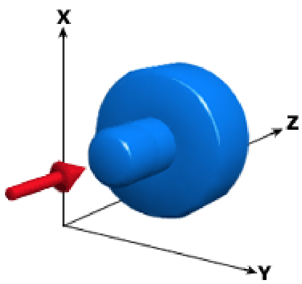
Clamp force is also calculated during the packing stage when the whole cavity might be pressurized to a higher value than it was during the filling stage.

Clamp force

The part design and orientationn can impact on the clamp force required for molding.

Orienting the model for correct clamp force prediction

You need to ensure your model is oriented correctly to correctly calculate clamping effects on the molding process for your part design. The clamp force direction is + Z direction in the coordinate axis. If your model is not oriented so that the clamp force is in the +Z direction, then you need to rotate your model so it is.



NOTE: The clamp open direction of your mold must be known so the model can be oriented accordingly in the X, Y, and Z planes.

---

1. In the **Layers** pane, ensure that nodes are visible.
2. Click (**Select**) and draw a rectangle around the model to select it.

   Ensure that you have selected the entire model.
3. Click **Geometry tab > Utilities panel >** **(Move tools) > Rotate**.
4. The **Select** box should contain a long list of node and element numbers, since you have already selected the entire model.
5. In the **Axis** drop-down list, select the axis to rotate the part around.
6. Enter the **Angle** to rotate the part around the selected axis.
7. Ensure that **Move** is selected.
8. Click **Apply**, and then **Close**.

Excluding undercuts from clamp force calculation

If a model contains undercuts or slides, the projected area calculation of the clamp force may be over-predicted.

The analysis log will show a double projection of surfaces to the parting plane for these regions. This over-prediction causes the clamp force calculation to be overestimated.

1. Select the undercut elements.
2. Right-click any of the selected undercut elements and click **Properties**.

   The **Part surface** dialog appears.
3. Select the **Exclude from clamp force calculation** check box.
4. Click **OK**.

Modeling for accurate clamp force prediction

Fill analysis calculates the maximum clamp force required to keep the mold closed during filling.

Clamp force calculation

The calculation for clamp force uses two values:

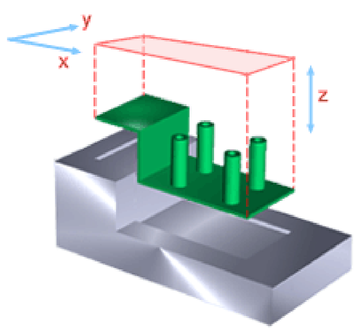
Area	The area of the model projected onto the XY plane.
Pressure	The injection pressure distribution.

The maximum force required to hold the mold closed is calculated by finding the integral pressure over the projected area of the cavity. That is, the area is divided into small segments, and the pressure is calculated for each of the small segments. The total clamp force is the sum of the products of each of these Pressure and Area components:

Clamp force = $\sum 1nPA$

where:

- **n** = the number of segments into which the total area is divided
- **A** = the area of each segment
- **P** = the average pressure at each segment



For the clamp force calculation to be correct, the model positioning must be such that the clamp force is applied along the Z axis direction.

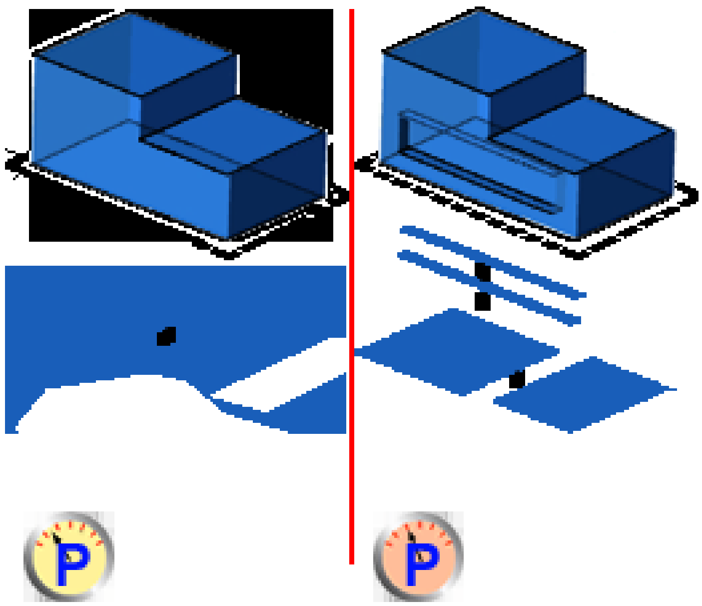
You can prepare your model at any convenient rotation, and when finished, correctly position it with the model manipulation tools.

Clamp force during filling and packing

Typically, both the pressure and the clamp force increase progressively during filling. Once into the packing phase, the cavity pressure becomes more uniform and, if the filling pressure is maintained, the clamp force will increase significantly. Under normal molding conditions, the packing pressure is lower than the filling pressure and so the clamp force required during packing may be higher than or lower than the clamp force required during filling.

Over-predicting clamp force

The projected area of the cavity is the sum of the area of all of the surfaces projected perpendicularly onto the parting plane. In some cases, some of these projected surfaces may overlap (see image below). The second and all subsequent overlapping surfaces do not contribute to the actual clamp force required in real life, but they are taken into account by the analysis during the clamp force calculation. As a result, the predicted clamp force can be greater than the amount actually required to keep the mold closed.

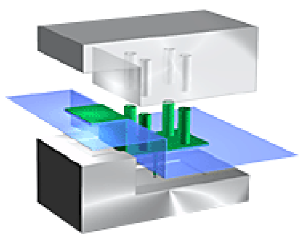


This is a very common problem when dealing with such undercut geometries normally requiring slide core molds to mold them, as well as with stack or multi-plate molds. It is difficult for the program to distinguish these special cases, so you may need to manually exclude undercuts from the clamp force calculation.

Parting plane

3

The parting plane is the contact surface of the stationary and moving halves of the mold that separate when the mold opens.



A newly imported model must be correctly oriented within the mold before an analysis can be run. When they are first imported, models are usually oriented so that the cavity side of the mold is oriented towards the sprue; however, the mold parting plane is in the X Y plane with the sprue entering the mold from the positive Z direction.

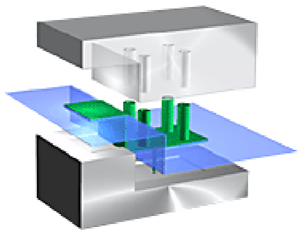
You can change the orientation of the model by using the rotate tool to rotate it about the center.

Parting plane

The parting plane is the plane where the two halves of the mold seperate. Gates are usually modeled on this plane.

Defining a parting plane

Since a mold must contain both a cavity and core, there may be a small gap between them. While this gap is too small for material to flow into, it still acts as a partial barrier to heat transfer. To simulate this effect, you need to model the surface between the cavity and core as a parting plane.



A parting plane represents the resistance of heat transfer across a surface inside a mold, usually where the two halves of the mold meet. By default, the solver assumes perfect conductivity between mating surfaces. Thermal interface conductance is the only attribute that can be specified as a parting plane property. If a parting plane has low contact pressure or a small gap, and it is located between the part and the cooling channels, it should be modeled.

To model a parting plane:

1 Create regions which represents the plane(s) where the mold cavity and core meet, using the **Geometry tab > Create panel > Regions** and **Geometry tab > Create panel > Regions > Hole** tools.

   The region(s) should completely surround the part and have a hole where the part is.

2 Select all the regions you just created.

3 Click **Geometry tab > Properties panel > Assign**.

4 Click **New > Parting surface**.

   The **Parting Surface** dialog appears.

5 Specify the interface conductance across the parting plane and click **OK**.

   The interface conductance depends on the mold material, the contact pressure, whether there is air or grease between the mold cavity and core, and the thickness of the gap (if any).

6 Mesh the new regions by clicking **Mesh tab > Mesh panel > Generate Mesh**.

7 If you have not done so already, model the external mold surface.

   Leave a small gap between the edge of the parting plane and the mold surface to avoid element intersection problems.

TIP: A parting plane can be placed at any location within the mold to view the thermal distribution across that plane.

Rotating the model to match the global coordinate system

The default parting plane for a mold is the XY plane of the global coordinate system. If your part is modeled with a different parting plane, you can rotate the model so the global coordinate system's XY plane coincides with the parting plane.

The model must be rotated before you run the analysis.

1. Click (**Geometry tab > Utilities panel > Move**) and select **3 Points Rotate** from the drop down list.
2. Select all the elements in the model, either with the mouse or (**Geometry tab > Selection panel > Select All**).
3. Select three points on the model, all of which are on the parting plane. The points must not be in a straight line.
4. Click **Apply**.

The part is now rotated as required.

Specifying a local coordinate system for the parting plane

The default parting plane for a mold is the XY plane of the global coordinate system. If your part is modeled with a different parting plane, you can specify a local coordinate system for the parting plane.

1. Click (**Geometry tab > Local Coordinate System panel > Create LCS**).
2. Select three points on the model, all of which are on the parting plane. The points must not be in a straight line.
3. Click **Apply**.
4. Click (**Results tab > Plots panel > New Plot > Custom**).
5. Select **Clamp Force On Parting Plane**.
6. Select the local coordinate system you created, and set the parting plane normal to the Z axis.
7. Click **OK**.

   The new plot appears in the results list as a user-defined plot.

Creating a parting plane

The parting plane is the plane where the gates will be modeled, and is parallel to the X-Y plane.

1. Click **New Layer** to create a new layer for the parting plane. Name this new layer with an appropriate name, and make sure that all other layers are deactivated.

2 Click **Geometry tab > Local Coordinate System panel** and enter coordinates in the **First** box. Click **Apply** then **Close**.

3 Select the local coordinate system that you have just created, right-click and select **Activate as Modeling Plane** from the drop down menu.

4 Using the modeling plane as a reference, create regions which represent the plane(s) where the mold cavity and core meet, using **Geometry tab > Create panel > Regions**.

The region(s) should completely surround the part and have a hole where the part is.

5 Select all the regions you just created.

6 Click **Geometry tab > Properties panel > Assign**.

7 Click **New > Parting surface**.

The **Parting Surface** dialog appears.

8 Specify the interface conductance across the parting plane and click **OK**.

The interface conductance depends on the mold material, the contact pressure, whether there is air or grease between the mold cavity and core, and the thickness of the gap (if any).

9 Mesh the new regions by clicking **Mesh tab > Mesh panel > Generate Mesh**.

10 If you have not done so already, model the external mold surface.

Leave a small gap between the edge of the parting plane and the mold surface to avoid element intersection problems.

Parting plane

This dialog is used to edit the properties of elements or regions of type **Parting surface** which are used to represent mold component interfaces. These interfaces provide a significant resistance to heat transfer and therefore need to be taken into consideration, particularly in a Cool analysis.

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

Parting Surface dialog

In the **Interface conductance** section of this dialog, enter a value representative of your mold. The greater the interface conductance, the greater the ability for heat to be extracted.

In the **Name** section of this dialog, enter a unique name.

Cavity duplication

4

Cavity duplication can be facilitated by the **Cavity Duplication Wizard**. Alternatively, you can perform the process manually.

The Cavity Duplication Wizard allows you to create a standard multi-cavity layout with a user-defined layout. Following cavity duplication, you can run the Runner System Wizard to connect all the cavities.

Alternatively, you can manually duplicate the cavities, which provides you with more flexibility over the final layout.

Cavity duplication

A multi-cavity mold has two or more cavities. The cavities may be identical, to produce multiple copies of a part in a single injection cycle, or they may be different (a family mold), to produce related parts in a single injection cycle.

Manual cavity duplication

You can duplicate your model manually, or using the Cavity Duplication Wizard. This section describes how to perform this task manually.

Before duplicating the cavity, you should model your runner and gate, so all these components can be duplicated together.

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

2 Click **Geometry tab > Utilities panel >** **(Move tools) > Reflect**.

NOTE: Using the **Reflect** tool, you will obtain a mirror image of the part.

The **Rotate** tool ( ) will give you a copy of that part.

---

3 Select **YZ** plane from the **Mirror** drop-down list.

4 Activate the **Reference point** box and then click on the node furthest from the part, which you created when you made the runner.
The coordinates of the node you created, when you made the runner, should now appear in the **Reference point** box.

5 Select **Copy**.

6 Select **Attempt connection to existing model**.

When this option is on, Autodesk Moldflow Insight will automatically make any necessary adjustments to the model, for example, merging coincident or very close nodes to ensure that the copied entities are correctly connected to the original entities.

7 Click **Apply** and then **Close**.

8 Click (**Zoom All**) to rescale the model.
There should now be two connected cavities.

NOTE: You should now model the sprue.

Cavity Duplication Wizard

The **Cavity Duplication Wizard** leads you through a simple way to create a multi-cavity layout.

The **Cavity Duplication Wizard** needs to know the injection point required.

1 From the **Study Tasks** pane, double-click on **Set Injection Locations**.

2 Click on the node you want to use for the injection location.

3 Right-click in the **Model** pane and select **Finish Set Injection Locations**.

4 Click **Geometry tab > Modify panel > Cavity Duplication**

5 Enter values into the various text boxes, to reflect the number of cavities you want. You may need to switch from **Columns** to **Rows** to ensure that the injection locations are placed appropriately for a balanced runner system.
Ensure **Offset cavities to align gates** is ticked. This option is useful when the gate is not positioned exactly midway along the side of the part.

6 Click **Preview** to view the layout.

7 Click **Finish**.

8 Use the **ViewCube** to switch to **Front** view, and click (**Zoom All**) to view the multi-cavity layout that has been created.

Cavity duplication

The **Cavity Duplication Wizard** dialog allows you to create a standard multi-cavity layout by duplicating a selected part model in a specified layout.

To access this wizard, click **Geometry tab > Modify panel > Cavity Duplication**.

Cavity Duplication Wizard dialog

With this dialog, you can choose to use the default values which will simply duplicate the part and supply a default spacing between the two parts.

Alternatively, in the **Number of cavities** box, enter the number of cavities you want to create. Decide how many rows and columns you would like, and enter the spacing between them. Click **Preview** to view the layout before you finish.

You can choose to connect all the cavities with runners, or use the **Runner System Wizard** to connect all the cavities, or model the runner system manually.

NOTE: The **Cavity Duplication Wizard** is not directly accessible for CAD entities. If you want to duplicate a CAD body, you need to mesh it first, and then run the **Cavity Duplication Wizard**.

Inserts

5

There are two types of inserts, mold inserts and part inserts.

Both of these inserts are modeled similarly.

Mold inserts

Mold inserts are made of metal and are attached to the mold. They are used to create a cavity in the part.

- Mold inserts can be created with a Midplane, Dual Domain or 3D mesh.
- Modeling the mold insert as a volume improves the accuracy of the heat transfer calculations. When you assign a mold material to elements instead of modeling the insert, a default small thickness is used for the insert material, which may be less accurate. This approach may be used when investigating mold design, but for more precise results you should model the insert as a volume.

Part inserts

Part inserts are components that are placed into the mold before the injection phase and ejected with the part. They can be made from metal or a thermoplastic material. It is assumed there is a new part insert for each shot.

- A part insert can be modeled with a Midplane mesh if the part insert is not required to be a solid model.
- The part insert is modeled separately from the plastic part when using a 3D mesh or Dual Domain mesh.

Consideration of inserts in Midplane, Dual Domain, and 3D analyses

The following table list the types of inserts that are considered by Midplane, Dual Domain, and 3D analyses where applicable.

Inserts considered in Midplane analyses

	Fill+Pack	Cool BEM	Warp
Part insert	✓	✓	✓
Mold insert	Not applicable	✓	Not applicable
In-Mold label	Not available	✓	Not applicable *List item.* on page 17

	Fill+Pack	Cool BEM	Warp
Core	✓	Not applicable List item. on page 17	Not applicable

NOTE:

1 In-Mold labels do not have a significant effect on warpage.

2 In Midplane analyses, a core is considered as a mold element and included in the mold cool analysis.

Inserts considered in Dual Domain analyses

	Fill+Pack	Cool BEM	Warp
Part insert	✓	Not considered	✓
Mold insert	Not applicable	✓	Not applicable
In-Mold label	Not available	✓	Not applicable List item. on page 17
Core	✓	Not applicable List item. on page 17	Not applicable

NOTE:

1 In-Mold labels do not have a significant effect on warpage.

2 In Dual Domain analyses, a core is considered as a mold element and included in the mold cool analysis.

Inserts considered in 3D analyses

	Fill+Pack	Cool BEM	Cool FEM	Warp
Part insert	✓	✓	✓	✓
Mold insert	✓	✓	✓	Not applicable
In-Mold label	✓	✓	Not applicable List item. on page 18	Not applicable List item. on page 18

	Fill+Pack	Cool BEM	Cool FEM	Warp
Core	✓	✓ *List item.* on page 18	✓	Not applicable

NOTE:

1. Although In-Mold labels are ignored in a Cool (FEM) analysis, it is possible to use the HTC conditions to simulate the label's insulating effect.
2. In-Mold labels do not have a significant effect on warpage.
3. The core temperature is calculated when the **Calculate internal mold temperature** option is enabled in the Cool 3D solver parameters. This calculation uses default mold steel material properties unless another material was defined for the core.

Part inserts

Part inserts are components that are placed into the mold before the injection phase, and are ejected with the part.

A part insert can be modeled with a Midplane mesh if the part insert is not required to be a solid model. The part insert is modeled separately from the plastic part when using a 3D mesh or Dual Domain mesh.

Examples of part inserts include In-mold labels, which are very thin part inserts (less than 1mm thick). Labels are applied to the mold before each injection. As the label has different material properties, which greatly affect the flow and cooling behaviour, they need to be modeled separately.

Part inserts

The meshing technology used to model a part will affect the way an insert is modeled. The type of insert should also be considered.

Modeling a part insert using a Dual Domain mesh

Ensure the part mesh is on the same plane and has the same boundary as the part insert.

1. Create a model of the insert separately and import the part insert.
2. Mesh the part insert as a Dual Domain mesh with the same density as the part.
3. Ensure the mesh does not have any errors.
4. Use a Dual Domain mesh to model the part insert if you do not want to run an overmolding analysis. Assign the **Part Insert** property to the part insert elements.

5 Assign values to the **Part Insert** properties:

- material (metal or polymer)
- local mold surface temperature control (where the insert does not touch plastic, the default assumes it is touching the mold)
- occurrence number (this should be the same value as the part)
- mold properties (the properties of the mold elements which the part insert contacts)
- initial temperature if the part insert is heated.

6 Add the part insert to the model of the part by clicking then click **Open > Add**.

7 Ensure the part insert and the part are correctly oriented by clicking **Mesh tab > Mesh Diagnostics > Orientation**.

The elements at the boundary between the part and part insert should completely overlap. If there are elements that do not overlap, convergence problems may occur.

Modeling part inserts using a Midplane mesh

There are two types of inserts, both of which are modeled similarly.

- Mold inserts are made of metal and are a portion of the mold. They are often used to create a core like feature.
- Part inserts are components that are placed into the mold before the injection phase and ejected with the part. It is assumed there is a new part insert for each shot.

NOTE: A Midplane mesh can only be used to model simple part insert geometries, such as a brass threaded insert because the thickness of the insert is nearly uniform. For complex geometries, a 3D mesh should be used.

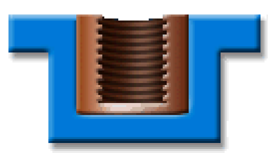


Figure 1: Example of a threaded insert

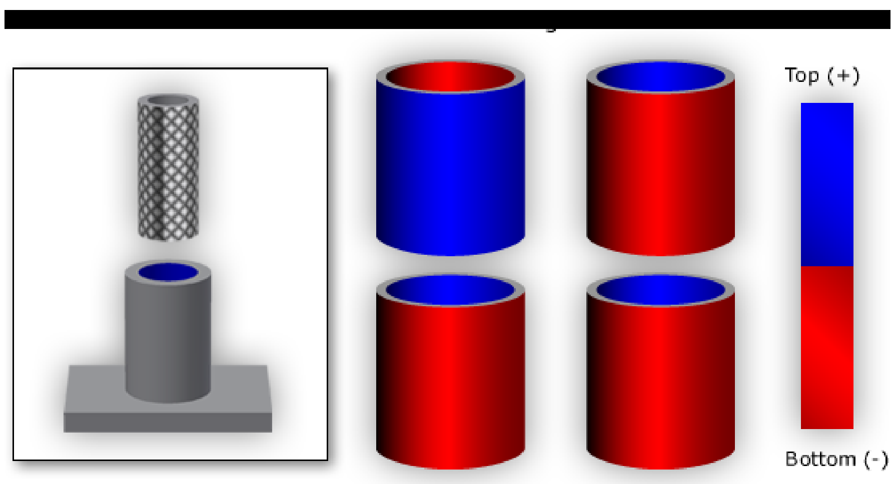
To create a threaded part insert using a Midplane mesh, apply the following steps.

1 Select the elements on the model that touch the insert—typically the side walls of a hollow boss.

2 Create a new layer and assign the selected elements to it. This ensures that all the elements are properly selected.

3 Click **Geometry tab > Utilities panel > Move > Translate**.
4 Select the elements on the layer that you created in step 2.
5 Enter a vector greater than the length of the insert, and remember this vector.
6 Click **Copy**
7 Click **Apply** to copy the selected elements, then click **Close**
8 Select the new elements, then change the property type to Part insert.
9 Assign the necessary properties such as material (metal or polymer), interface conductivity and initial temperature if the insert is to be heated.
10 Assign the thickness.
11 Ensure the mold insert and the part are correctly oriented by clicking **Mesh tab > Mesh Diagnostics > Orientation**.

NOTE: The orientation of the mesh defines what side of the part the insert is on. If the top side of the insert is touching the top side of the part, the insert is touching the top of the part. If the top side of the insert is touching the bottom side of the elements the insert is on the bottom side of the part elements.



Modeling a part insert using a 3D mesh

TIP:

- You can use MDL 6.0 and later to mesh assemblies. This ensures a proper mesh between the part and the part insert.
- If you used **Direct Import** to import a CAD assembly, you can directly assign a property to each body before meshing the assembly. For example, you can assign the Part insert (3D) property to one of the bodies of an assembly.

NOTE: The part and part insert must be modeled with the same coordinate system.

1 Create a model of the part insert in a separate CAD file and import it into a new study file.

2 Mesh the part insert as a Dual Domain mesh with the same density as the part.

3 Before creating a 3D mesh, ensure that the elements of the part insert are perfectly aligned with the elements of the plastic part, so that the faces of the part insert elements are in perfect contact with the plastic part elements.

   To check that the part insert and plastic part are perfectly aligned, follow the steps below:

   a Click then click **Open > Add** to add the part study into the part insert study.

   b Click **View tab > Appearance panel > Entities**.

   c On the **Default Display** tab, set all mesh elements to **Transparent + Element Edges**, then click **Ok**.

   d Check if the mesh of the plastic part and part insert perfectly align.

   TIP: Use cutting planes to assist in viewing the mesh.

   e If the mesh is not aligned, delete a portion of the part insert that does not align with the plastic part, and use the imported plastic part elements to re-construct the part insert.

   NOTE: Ensure there are no mesh defects such as free edges, holes, overlaps or intersections.

   f Delete any remaining portion of the plastic part model.

4 Change the part insert mesh type to 3D, and then create a 3D mesh.

5 Use the Mesh Repair Wizard to check the quality of the 3D mesh

6 Assign the Part insert (3D) property to all the elements of the part insert.

7 Assign values to the **Part Insert** properties:

   - material (metal or polymer)
   - local mold surface temperature control (where the insert does not touch plastic, the default assumes it is touching the mold)
   - occurrence number (this should be the same value as the part)
   - mold properties (the properties of the mold elements which the part insert contacts)
   - initial temperature if the part insert is heated.

8 Add the part insert to the model of the part by clicking then **Open > Add**.

Modeling in-mold labels

In-mold labels are very thin part inserts (less than 1mm thick). Labels are applied to the mold before each injection. As the label has different material properties which greatly affect the flow and cooling behaviour, they need to be modeled separately.

NOTE: You must run a Cool analysis to include in-mold labels in calculations.

There are two different methods for creating in-mold labels. You can model them as part inserts if they are more than 1mm thick, or you can follow the directions below.

NOTE: Ensure the mesh on the part is on the same plane and has the same boundary as the label.

---

1 Select the elements you want to apply the label to.

2 You can create a new layer and assign the label elements to it, or follow steps 3-5 below.

3 Click  **Geometry tab > Utilities panel > Move > Translate**.

4 In the **Vector (x,y,z)** box, in the Translate tool pane, enter an arbitrary vector to temporarily offset the label so that properties can be assigned to the label (i.e. 0 0 100). Make sure you remember the offset amount.

5 Select the option for **Copy** and click **Apply**. Resize the screen to view the newly created elements.

6 Highlight the elements for the label and click  **Geometry tab > Properties panel > Assign** then click **New > In-mold label**.

7 Assign the necessary properties such as material, interface conductivity and thickness.

NOTE: A label must have a thickness and material assigned to it. If a label is more than 1mm thick, you will need to model it as a part insert.

---

8 Ensure the label and the part are correctly oriented.

- For Midplane models, if the blue side of the label faces the blue side of the part, then the label is located on the blue side of the part. If the red side of the label faces the blue side of the part, the label is located on the red side of the part.
- For Dual Domain and 3D models, ensure that the blue surface of the label faces the blue surface of the part.

9 If you created a translated copy of the elements, you need to move the copy back onto the part. In the **Vector (x,y,z)** box, in the Translate tool

pane, enter the opposite of the amount you offset the label elements in step 3. Select the **Move** option, and click **Apply**.

NOTE: If the solution doesn't converge when you run the model, model the in-mold label as a part insert.

Modeling heated part inserts

Part inserts are often heated to:

- improve bonding between materials
- alter warpage effects
- slow the cooling of the material around the insert, which results in less residual stress

Modeling the insert as a volume rather than assigning material properties to the elements improves the accuracy of the heat transfer calculations.

You can assign an initial temperature to an insert if you are creating a 3D meshed model. To simulate a heated insert with Midplane and Dual Domain meshed models, you can model cooling channels through an insert, and use oil heated to an appropriate temperature as the coolant.

To set the initial insert temperature for 3D meshed models, apply the following steps.

1 Select the part insert elements.
2 Click **Geometry tab > Properties panel > Properties**
3 Enter a value for the **Initial temperature**, and click **OK**.

Part inserts

Part inserts need to be defined so that their properties can be incorporated into filling and cooling analyses.

To access this dialog to edit the properties of existing model entities, select at least one element of type **Part insert** or **Part insert (3D)**, then either select **Geometry tab > Properties panel > Edit**, or press **Alt-Enter**, or right-click and select **Properties**.

Part Insert (Midplane) / Part Insert (Dual Domain) dialog

This dialog is used to edit the properties of elements or regions of type **Part insert**.

To access this dialog to edit the properties of existing model entities, select at least one element or region of type **Part insert**, 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.

Part Insert (3D) dialog—Part Insert Properties

This dialog is used to edit the properties of tetrahedral elements of type **Part insert (3D)**.

In the **Material from which this feature is made** section of the dialog, choose a type of material and then the specific material of that type.

In the **Local heat transfer coefficients** section of the dialog, select the heat transfer effectiveness (HTE) criteria to be applied to the elements.

Specify the initial temperature of the part insert when it is placed in the mold and assign a unique name to the insert.

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.

Part Insert (3D) dialog—Mold Properties

The **Mold Properties** tab of this dialog is used to specify the properties of the mold block in contact with the part insert.

The **Mold material** section of this dialog select the type of material you would like to use for the mold.

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.

In-Mold Label dialog

This dialog is used to edit the properties of elements or regions of type **In-mold label**.

To access this dialog to edit the properties of existing model entities, select at least one element or region of type **In-mold label**, 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.

Material from which this feature is made	Select whether you want the part insert to be constructed with either resin or metal.
Interface conductance	Specifies the ability of heat to be dissipated from the selected region of the mold.
Thickness	Specifies the required thickness of the model area represented by the selected element(s).

Select Insert/Label/Core Material dialog

This dialog is used to specify the material from which the selected insert/label/core is made.

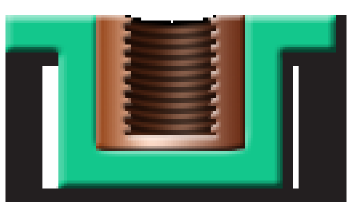
Mold material	This dialog is used to specify the material from which the selected insert/label/core is made.

Mold inserts

Inserts are parts of the mold that are created separately from the mold cavity block. Inserts are inserted in the block to achieve a desired cavity shape or cooling effect.

The effect of inserts on heat transfer

An insert will only assist heat transfer if a cooling channel is located in or near the insert. Inserts can be used to modify the rate of cooling in specific areas of the mold. A common example is a part with ribs that are thinner than the main surface. There is a natural tendency for the part to deflect away from the thinner ribs as the main surface has higher area shrinkage. By running the rib area hotter (conductivity insert), the part can be deflected back to the required shape. In some cases, an insert of lower conductivity or containing a separate cooling circuit, can be used to form the ribs. This gives better control of the rib temperature.



Modeling inserts

Inserts are modeled with regions representing each of the faces of the insert. In its simplest form, the complete insert is effectively a closed volume defined by the six surfaces of a cube. Inserts can have complex cross-sections to match features on the cavity model, in which case they consist of more than the basic six surfaces. Inserts can be added to inserts to create more complex shapes. When small inserts abut onto larger inserts, the surface of the larger insert must have an internal boundary for the smaller insert to connect with.

When the edge of a plastic surface runs across an insert surface, the insert surface should have an internal boundary added to ensure that the mesh on the plastic surface and the mesh on the insert surface are compatible. Inserts must not contact the mold outer surface.

NOTE: Each insert must be meshed and correctly oriented.

Mold inserts

Mold insert elements that touch the plastic part must be on the same plane as the part elements. There must be no gaps between the insert elements and the part elements. The default color for mold insert elements is gray.

Modeling mold inserts

1. Select the plastic elements that touch the insert.
2. Create a new layer and assign the selected elements to it.

   TIP: Use the new layer to ensure all the elements that touch the insert are selected.

   ---

3. Click  **Geometry tab > Create panel > Inserts** .
4. Select all the elements on the new layer defined in step 2.
5. Select the mold insert direction. The default direction **Part Normal** will not work when there are elements in multiple planes. The most common direction in which to create an insert is the Z direction (the direction in which the mold opens).

   NOTE: You can select to extend the insert using the **Projection distance** by a specified distance. Enter the **Specified distance** as the total length the insert will be after creation. A positive value indicates the extrusion direction with be in the positive direction along the selected axis.

   ---

6. Select **Apply** to create the insert. A new layer is created. On this layer there is a copy of the selected part elements with a mold insert property,

and any additional elements required to create the insert to part gap defined by the Specified distance.

7 Assign values to the properties:

- material properties
- interface conductivity

8 Ensure the mold insert and the part are correctly oriented by clicking **Mesh tab > Mesh Diagnostics > Orientation**.

The orientation of an insert is the same as that of a Dual Domain model: the blue side points outward.

9 Delete the elements where the cooling channel passes through the mold insert.

Failure to do this may lead to analysis convergence problems.

10 Update the mold material properties for the cooling channel elements that pass through the mold insert.

Mold inserts

The properties of a mold insert need to be defined.

Inserts tool

This tool allows you to create a mold insert in your design. A mold insert can be of a different material from the rest of the mold base, for example, a special steel used to produce a core or a copper alloy for efficient heat removal. To access this tool, click **Geometry tab > Create panel > Inserts**.

In addition to modeling a different mold material, a mold insert can represent a thermal interface between the insert and the rest of the mold.

NOTE: An insert is modeled as a closed volume. Optionally, it can have water lines passing through it, as in a baffle.

Mold Insert Surface dialog

This dialog is used to edit the properties of elements or regions of type **Mold insert surface**.

To access this dialog to edit the properties of existing model entities, select at least one element or region of type **Mold insert surface**, 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.

Modeling mold inserts

Inserts are modeled with flat shell surfaces representing each of the faces on the insert. The complete insert is effectively a closed volume defined by the top bottom and side surfaces. Inserts can have complex cross sections to match features on the cavity model. Inserts can also be joined to one another to create more complex shapes.

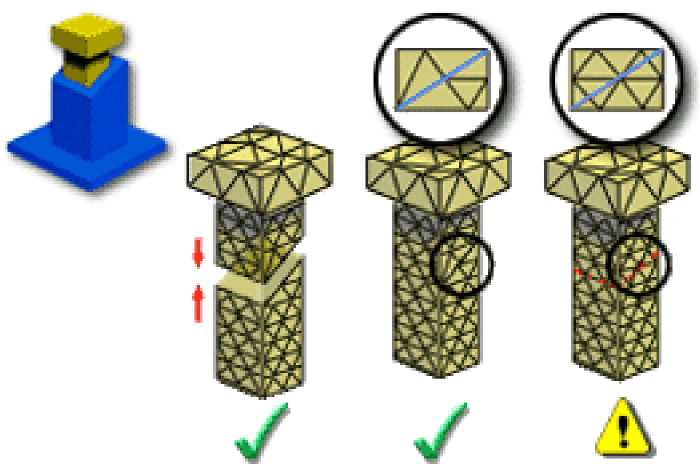
Modeling requirements

Do not apply a thickness values to the insert surface(s).

Inserts must not contact the mold outer surfaces. When a small insert butts onto a larger insert, the surface of the larger insert must have an internal boundary where the smaller insert contacts it.

When the edge of a plastic surface runs across an insert surface, the insert surface should have an internal boundary added to ensure that the mesh on the plastic surface and the mesh on the insert surface are compatible.

Where an insert projects from a feature such as a boss on the plastic part, the insert surfaces should either have internal boundaries coincident with the open edge of the boss or the insert should be modeled as two inserts stacked end to end, with the junction occurring at the height of the end of the boss. This will ensure that the mesh on the insert surface is compatible with that on the coincident plastic surface. This modeling requirement is illustrated in the following figure.

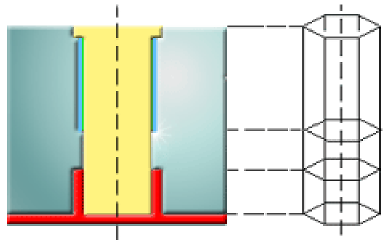


Modeling mold inserts—examples

This topic provides examples of an insert that is in poor contact with the surrounding mold, an insert that is in good contact with the mold, and a stepped insert.

Inserts in poor contact with the mold

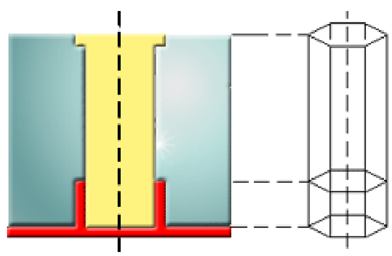
This modeling assumes there is poor contact between the insert and the rest of mold material due to clearance for ease of mold assembly.



This picture illustrates how inserts in poor contact with the mold material are modeled. The surface in contact with the air gap is assigned a very low interface conductance value. The insert is made from three surfaces, each with eight sides, creating three independent closed volumes.

Inserts in good contact with the mold

This modeling assumes there is a good contact between insert and rest of the mold material.

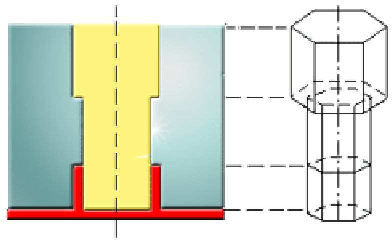


NOTE: To satisfy the requirement for a discrete boundary on the insert surfaces, where they are crossed by the edge of the surfaces forming the plastic boss, you can create one insert which is the length of the boss and then another, as shown in the image above: from the end of the plastic boss into the mold to represents the total insert depth. These inserts can be most easily created in Autodesk Moldflow Insight.

Alternatively, create a single insert of the required length with internal points or boundaries.

Stepped inserts

Model stepped inserts by creating one insert to represent the small end and a second to represent the larger end. The end face of the larger insert must have an internal boundary the same size as the smaller insert to ensure connectivity between the inserts.



Optional settings

6

There are a few concepts that may be worth considering, but which are not required for basic modeling.

Grid

The modeling grid enables you to create or relocation of parting planes, runner and cooling modeling planes, and runner and cooling channel sections.

You can choose to display the grid when you are modeling the cooling channels and runners or sizing the mold. You can also choose whether to snap to grid which helps define exact dimensions.

When defining the grid, you can choose the model grid origin, and the grid size and color.

Gravity direction

The direction of gravity is an optional analysis input that you can specify for a 3D Fill+Pack analysis. This allows you to investigate the effects of gravity on the injection molding process for your part.

NOTE: Gravity direction is available only if you are using the Navier Stokes numerical solver. The Navier Stokes solver takes flow inertia effects into account.

Gravity direction considerations

When molding thin walled parts, viscosity is usually low and flow of polymer is fast into the mold cavity. Therefore, the effect of gravity on the molding process is not a significant factor in the molding process.

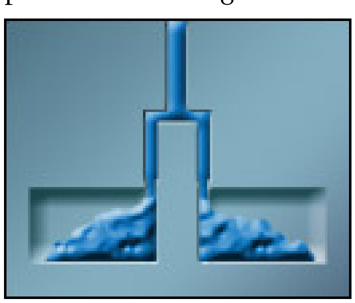


Figure 2: Gravity effect in thick-walled parts

For thicker walled parts where flow is more viscous, defects such as hesitation or jetting can occur due to the effect of gravity. Therefore, in thick-walled 3D models, you can set the direction of gravity to determine whether gravity has an effect on the injection molding process.

Gravity direction

You can set the gravity direction to any direction for a 3D Fill analysis. Setting the direction of gravity allows you to investigate the effects of gravity on the molding process for your part design.

Specifying the gravity direction

Gravity can affect the quality of thick-walled parts, causing problems such as hesitation or jetting. The default down gravity direction is -Y in the coordinate axis system.

1. Click **Home tab > Molding Process Setup panel > Process Settings**, or double-click the Process Settings icon in the **Study Tasks** pane.
2. Click **Advanced options**;
3. Click the **Edit** associated with **Solver parameters**.
4. Click the **Solver parameters** button.
   A dialog will appear, allowing you to modify the solver parameters.
5. Click the **Simulate gravity effect** check box, and click **Edit gravity direction**.
6. Enter the X, Y, and Z components of the gravity vector.

   For example, an orientation set of **0 1 0** will orient the part so that the gravity pushes in the +Y direction (i.e., the -Y direction is “up”).

   NOTE: You cannot change the strength of gravity. The vector you enter will be normalized to length 1.0 when passed to the solver.

Cavity/core side mold temperatures

If you have a molding scenario where certain areas of the mold are known to have higher or lower temperatures, you can simulate this using the **Ideal cavity-side mold temperature** and **Ideal core-side mold temperature**. For a Fill+Pack analysis, use this property to simulate a mold that does not have a constant temperature.

TIP: To specify **Ideal cavity-side mold temperature** and **Ideal core-side mold temperature** values for the entire model, click **Analysis Process Settings Wizard**, if necessary click **Next** until the **Fill+Pack Settings** page appears, click **Advanced Options**, click **Edit** in the **Process Controller** group, click the **Temperature Control** tab, set the **Mold temperature control** option to **Cavity differs from core**, click **Edit mold temperatures**, and enter the desired temperature values.

Cavity-side vs Core-side

Creating an accurate model of not only the part, but the mold used to make the part, is critical to a successful flow simulation. Often, the mold used to make a plastic part consists not only of a cavity hollowed from the mold plates, but also a core (the moving plate), which extends into the overall plane of a part to make a cylinder or depression. Inherently, the use of a core in a mold creates problems with the cooling system. A core area is more difficult to penetrate with cooling channels, bubblers, baffles, and other cooling devices. Because the core is deep within the mold base, it is not only less accessible to the cooling system, but it also has more difficulty in diffusing heat away from the part.

Unbalanced cooling, which is the result of temperature differences between the mold walls, can result in asymmetrical thermal-induced residual stress. This kind of unbalanced cooling results in an asymmetric tension-compression pattern across the part, and consequently a bending moment that tends to cause warpage of the part.

Autodesk Moldflow Insight allows you to set your own cavity-side and core-side temperatures so that the flow simulation can provide you with predictions of warpage. Using the simulation results, you can tailor your process to minimize the temperature difference, and therefore a major cause of warpage, as much as possible.

Generally speaking, only a small portion of a mold is designed to operate under different cavity-side and core-side mold wall temperatures. Autodesk Moldflow Insight provides this feature so you can determine the optimal core-side and cavity-side temperatures for minimum warpage. For all elements in which the cavity-side temperature equals the core-side temperature, you do not need to assign the temperatures separately.

NOTE: As a rule of thumb, the temperature difference for the cavity and core sides should not exceed 20°C (36°F).

Cavity/ core side mold temperatures

If you do not have the Cool module, you can manually assign separate cavity-side and core-side mold temperatures to assess the effect these variations may have on the filling of the part.

Different mold temperatures for cavity/core

In the injection molding process, there are areas of the mold that may have a higher or lower temperatures due to part/mold design. A Cool analysis will predict and take into account such local variations in the temperature of the mold.

If you do not have the Cool module, however, you can manually assign separate cavity-side and core-side mold temperatures to assess the effect these variations may have on the filling of the part. Autodesk Moldflow Insight provides three levels of mold temperature assignment:

- The mold surface temperature value on the Fill/Fill+Pack Settings page of the Process Settings Wizard.

  This is the default global mold temperature value that will be assigned to all elements unless overridden by one of the following two options.

- Separate cavity-side and core-side mold temperature values on the Process Controller dialog of the Advanced Options.

  These separate mold temperature values are assigned to all elements of a surface. Which side of each element is assigned the cavity-side value, and which side is assigned the core-side value is determined by the **Cavity/core side assignment** setting in the element properties together with the orientation (top/bottom) of the element.

- Separate top and bottom mold temperature values in the properties of an element.

  These mold temperature values are assigned to specific elements. The top and bottom side of the element is determined by the orientation of the element.

Specifying different mold temperatures for cavity/core by surface

In the injection molding process, there are areas of the mold that may have a higher or lower temperatures due to part/mold design. A Cool analysis will predict and take into account such local variations in the temperature of the mold.

If you do not have the Cool module, however, you can manually assign separate cavity-side and core-side mold temperatures to assess the effect these variations.

To set separate cavity-side and core-side mold temperature values for a surface, complete the following steps.

1. Click **Home tab > Molding Process Setup panel > Process Settings**.
2. Click **Advanced Options...**, then click the **Edit** button next to the **Process controller** option.
3. Click the **Temperature Control** tab, select **Cavity differs from core** in the **Mold temperature control** drop-down, then click **Edit mold temperatures**.
4. Enter the required cavity-side and core-side mold temperatures values, then click **OK** to accept the changes.
5. Click **OK** twice to return to the Process Settings Wizard, navigate to the final page of the wizard, then click **Finish**.
6. Click **Mesh tab > Mesh Diagnostics panel > Orientation**, then **Show** to display the orientation of all elements.
7. To check the cavity/core assignment in various areas of the model, select a group of elements in the model pane, right-click and select **Properties**, select the **Mold Properties** tab.

NOTE: For Dual Domain models the cavity/core side assignment must be set to specify which mesh surface is in contact with the cavity side of the mold and which mesh surface is in contact with the core side of the mold. If this is not set, both surfaces will use the cavity temperature setting.

Specifying different mold temperatures for cavity/core by element

In the injection molding process, there are areas of the mold that may have a higher or lower temperatures due to part/mold design. A Cool analysis will predict and take into account such local variations in the temperature of the mold.

If you do not have the Cool module, however, you can manually assign separate cavity-side and core-side mold temperatures to assess the effect these variations and bottom side of the element is determined by the orientation of the element.

To set separate cavity-side and core-side mold temperature values for specific elements of a surface, complete the following steps.

1. Click  **Mesh tab > Mesh Diagnostics panel > Orientation**, then **Show** to display the orientation of all elements.
2. Select the required elements in the model pane, right-click and select **Properties**.
3. Select the **Mold Properties** tab, then select **Local mold surface temperature** in the **Cavity/core side assignment**.
4. For Dual Domain elements, enter the mold surface temperature value in the edit box provided. For Midplane elements, click **Edit temperatures** to display the **Local Mold Surface Temperature** dialog, enter the required top and bottom mold temperature values, then click **OK**.

Cavity/core side mold temperatures

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.

Core (3D) dialog—Part Insert Properties tab

This dialog is used to edit the properties of tetrahedral elements of type Core (3D).

The **Part Insert Properties** tab of the **Core (3D)** dialog is used to specify the properties of the selected elements or regions of type **Core (3D)**.

To access this dialog to edit the properties of existing model entities, select at least one tetrahedral element of type Core (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.

Dialog Properties	Comment
Material from which this feature is made	Use the drop-down menu to select whether you want the feature to be constructed from resin or metal. Click **Select** to edit the properties of the selection you made.
Local heat transfer coefficients	Specify the heat transfer effectiveness (HTE) criteria to be applied to the selected tetrahedral elements. **Use global setting in advanced options**- use this option if you want to use the thermal properties specified in the **Advanced options** of the **Process Setting Wizard**. **Perfect contact** - When selected there is no heat resistance between the part and the mold surface. When this option is selected the wall nodes are a fixed thermal boundary for analysis purposes. **Local heat transfer coefficients** - Allows you to specify the heat transfer coefficients for the selected tetrahedral elements. Click **Set** to adjust or view the Heat Transfer Coefficients for the selected tetrahedra.
Mold surface temperature	Specify how the mold surface temperature of a selected area on the surface of the model is defined. **Use mold surface temperature in process settings**- the value is obtained from the **Process Settings Wizard (Home tab > Molding Process Setup panel > Process Settings)**, and used throughout the molding cycle. This option is selected by default. **Constant** - this option allows you to specify a constant temperature for the selected area, which is different from the temperature value that is assigned in the **Process Settings Wizard**. **Profile** - this option allows you to define a varying mold surface temperature for the selected area throughout the molding cycle.

Dialog Properties	Comment
Initial temperature for production start-up	This property is only used for Production Start-up analyses. Enter the temperature of the mold before the analysis starts.
Name	The collection of property values defined in this dialog are saved to a property set with the description shown in the Name box. You can leave the default or type in a different name.

Core (3D) dialog—Mold Properties tab

This dialog is used to edit the properties of tetrahedral elements of type Core (3D).

The **Mold Properties** tab on the **Core (3D)** dialog is where you can set special mold properties to be in contact with this component, or use the global mold properties specified in the process settings.

To access this dialog to edit the properties of existing model entities, select at least one tetrahedral element of type Core (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.