

Autodesk® Moldflow® Insight 2012

AMI Geometry

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Contents

Chapter 1	Geometry	1
	Geometry.....	1
	Copying model entities.....	1
	Displaying/hiding model entities.....	2
	Selecting individual model entities.....	2
	Deselecting individual model entities.....	2
	Selecting model entities by properties.....	2
	Deselecting model entities by properties.....	3
	Selecting all model entities.....	3
	Deselecting all model entities.....	3
	Locating model entities by number.....	3
	Expanding the current model entities selection.....	4
	Selecting model entities by banding.....	4
	Saving a selection list.....	5
	Deleting a selection list.....	6
	Inverting selection of model entities.....	6
	Geometry.....	6
	Select Entity Types dialog.....	6

	Select by Property dialog.	6
	Query Entities tool.	7
	Expand Selection dialog.	7
	Select Saved Selection List tool.	7
	Save Selection List dialog.	7
	Delete Selection List dialog.	7
	Change Property Type dialog.	7
Chapter 2	Nodes.	8
	Nodes.	8
	Modeling nodes.	8
	Nodes.	9
	Node by Coordinate tool.	9
	Node Between Coordinates tool.	9
	Nodes By Dividing Curve tool.	10
	Node By Offset tool.	10
	Node By Intersect tool.	10
Chapter 3	Curves.	11
	Curves.	11
	Creating curves.	11
	Breaking curves.	12
	Modeling hot and cold sprue/gates/runners.	12
	Curves.	13
	Create Line tool.	13
	Arc by Points tool.	13
	Arc by Angle tool.	13
	Spline tool.	13
	Connect Curves tool.	14
	Break Curve tool.	14
Chapter 4	Regions.	15
	Regions.	15
	Creating regions.	15
	Creating mesh regions.	15

Regions.....	16
Region by Boundary tool.....	16
Region by Nodes tool.....	17
Region By Ruling tool.....	17
Region by Extrusion tool.....	17
Hole by Boundary tool.....	17
Hole by Nodes tool.....	18

Chapter 5	Manipulating model entities.....	19
	Rotating a part.....	19
	Mirroring a part.....	19
	Translating a part.....	19
	Scaling a part.....	19

Geometry

1

You can use a variety of tools to model a part, a runner system, or a cooling system.

Geometry

The following section describes different ways of selecting, deselecting, and saving model entities.



Copying model entities

You can add or copy existing model entities to your study to save modeling time. If you have existing model entities that you want to add to a model, such as cooling channels or runners modeled in a CAD program, you can add them to your study. You can also use this feature to quickly create a multi-cavity model.

NOTE: Bodies or assemblies that are added to an existing model are displayed in a new layer.

To copy entities from a different study, you must add the different study to the current study. It is not possible to select groups or individual entities and copy them to another study. After adding one study to another, you can manipulate the added study to achieve the required result.

TIP: Click [here](#) for a list of the supported CAD formats and entities.

- 1 Click  then click  **Open > Add**.
- 2 Navigate to the directory where the required model entity is located.
- 3 Select the correct file extension from the **Files of type** drop-down list.
- 4 Click on the file to select it.
- 5 Click **OK**.

The model section that you added is now displayed. You may need to use the modeling tools to correctly join the two model sections.

NOTE: When importing a runner system or cooling channels, model the entity as a simple line drawing in the CAD program and export the “Datum Curves and Points” as an (*.iges) file. The (*.iges) file should contain the center-line, requiring you to set its properties in Autodesk Moldflow Insight.


Displaying/hiding model entities

The **Layers** pane provides a list of all of the entities present on your model. You can display and hide entities, as well as create, rename, and delete entities.

- 1 In the **Layers** pane, select a check box to display that entity.
- 2 Perform any other tasks that are required, such as making a layer active or assigning entities to a layer.

Selecting individual model entities

You can use the Select tool to select one or more individual entities on your model and mesh.

- 1 Click  (**Select**).
- 2 Click on the entity in the display that you want to select.
- 3 To add another entity to the selection, hold down the **Ctrl** key and click on the next entity in the display that you want to select.


NOTE: To expand the selection, use the **Expand Selection** dialog.

Deselecting individual model entities

You can use the Select tool to deselect individual entities on your model and mesh.

- 1 Ensure that the **Select** tool is active.
- 2 Hold down the **Shift** key.
- 3 Click on the entity in the display that you want to deselect.

Selecting model entities by properties


- 1 Click  **Geometry tab > Selection panel > Properties**.
- 2 Use the **Select by Property** dialog to choose the types of elements you want to select.
- 3 If you want to select the nodes associated with those elements, click **Select associated nodes also**.
- 4 If you want to add the selection to entities that you have already selected, click **Add to current selection**.

NOTE: If you do not select this check box, any entities already selected will be deselected.

TIP: Click  **Geometry tab > Selection panel > Layers** to select groups of model entities by layers.

Deselecting model entities by properties


To deselect groups of entities that you have previously selected using the **Select by Property** dialog:

- 1 Click  **Geometry tab > Selection panel > Properties**.
- 2 Use the **Select by Property** dialog to choose the types of elements you want to select.
- 3 If you want to select the nodes associated with those elements, click **Select associated nodes also**.
- 4 To deselect the entities that you have already selected, do not click the **Add to current selection** check box.
Any entities already selected will be deselected.

Alternatively, click  **Geometry tab > Selection panel > Deselect All**.


Selecting all model entities

To select all displayed entities:

- ☒ Click  **Geometry tab > Selection panel > Select All**.
All entities displayed will be selected.


Deselecting all model entities

To deselect all selected entities:

- ☒ Click  **Geometry tab > Selection panel > Deselect All**.
All selected entities will be deselected.


Locating model entities by number

If you know an entity identifier, such as T123 (triangle 123), and you want to locate it on the model:

- Click  **Geometry tab > Utilities panel > Query**.
You will need to add an entity identifier to the number.


Expanding the current model entities selection

Use the **Expand Selection** dialog to select one or more “layers” of entities surrounding the currently selected entities.


- 1 Click  **Geometry tab > Selection panel > Expand**.
The **Expand Selection** dialog appears.
- 2 In the text box provided, enter the number of “layers” of surrounding entities you want to add to the current selection.
- 3 Click **OK**.

Selecting model entities by banding



Use banding selection methods to select groups of entities on your model that do not share attributes. There are two banding selection tools that act as filters to control how many entities are selected. The banding selection tools may be used separately or together.

- 1 Click  **Geometry tab > Selection panel > Select**.
- 2 On your model, click and hold the left mouse button while you draw a rectangle around the entities you want to select.
Entities that are enclosed within and touched by the selection rectangle, through the thickness of the part, will be selected.

TIP: Rotate the model to see all of the selected entities.




- 3 Filter your selection using the **Select Enclosed entities** tool alone:
 - a Click  **Geometry tab > Selection panel > Select Enclosed entities**.
 - b On your model, click and hold the left mouse button while you draw a rectangle around the entities you want to select.
Only the entities that are fully enclosed within the selection rectangle, through the thickness of the part, will be selected.

TIP: Rotate the model to see all of the selected entities.

- c Click  again to deactivate this tool.
- 4 Filter your selection using the **Select Facing entities** tool alone:
 - a Orient the model so that the entities you want to select are facing you.
 - b Click  **Geometry tab > Selection panel > Select Facing entities**.
 - c On your model, click and hold the left mouse button while you draw a rectangle around the entities you want to select.

Entities on the face of the model that are enclosed within and touched by the rectangle will be selected. Entities through the rest of the thickness of the model are not selected.


NOTE: If the rectangle includes an edge of the model, some of the shared elements along the edge may be selected. This may result in some elements on the other, non-facing side of the edge being selected. You can manually deselect these elements by holding down the **Shift** key and clicking on the elements individually.

- d Click  again to deactivate this tool.
- 5 Filter your selection using the **Select Enclosed entities** and **Select Facing entities** tools together:
 - a Orient the model so that the entities you want to select are facing you.
 - b Click  **Geometry tab > Selection panel > Select Enclosed entities.**
 - c Click  **Geometry tab > Selection panel > Select Facing entities.**
 - d On your model, click and hold the left mouse button while you draw a rectangle around the entities you want to select.


Entities on the face and through the thickness of the model that are fully enclosed within the rectangle will be selected.

Saving a selection list


The Selection List allows you to save a list of selected entities so that you do not have to reselect the same individual entities each time. This can then be used for quick selection and for point of reference to complement model layers, and when modeling or meshing.

- 1 Open a model.
- 2 Click  **Geometry tab > Selection panel > Select.**
- 3 Highlight the entities of your choice on the model.

TIP: The **Save Selection List** drop-down list displays the entity number and the character used to indicate the selected entity. For example, N12 indicates node 12, T35 indicates triangle element 35, B400 indicates beam element 400 etc.


- 4 Click  **Geometry tab > Selection panel > Save selection.**

The **Save Selection List** dialog appears.
- 5 Enter a name and click **OK**.

- 6 Click the drop-down arrow in the  **Select Saved Selection** list to see the saved lists available for you to select.


Deleting a selection list

You can delete a previously saved selection list if you no longer need to refer to it.

- 1 Open a study in which you have saved one or more selection lists.
- 2 Click  **Geometry tab > Selection panel > Delete Selection**. The **Delete Selection List** dialog appears.
- 3 Highlight one of the selection lists and click **Delete**.

Inverting selection of model entities

To invert the selection of displayed model entities:

- 1 Click  **Geometry tab > Selection panel > Invert**.

Geometry


This group of tools is used to select and group specific entities to make modeling easier.

Select Entity Types dialog

This dialog appears when you have selected an action and your selection includes more than one type of model entity, for example, you want to delete a mixture of curves and regions.

NOTE: The entity type list on this dialog supports the standard multiple selection techniques, that is, Shift+left mouse button selects a range of entries in the list, Ctrl+left mouse button selects non-sequential entries in the list.

Select by Property dialog

This dialog is used to select groups of model entities according to their entity type and properties. To access this dialog, click  (**Geometry tab > Selection panel > Properties** or **Mesh tab > Selection panel > Properties**).


Query Entities tool

Use the **Query Entities** tool to locate an entity on the model, or to find the ID of an entity by clicking on it.

To access this tool, click  **Geometry tab > Utilities panel > Query**.

Expand Selection dialog

This dialog is used to expand the current selection by the number of layers of surrounding elements that you specify. For example, if you have selected a single node and want to also select the triangles and nodes immediately surrounding it, you can use this dialog to do so. To access this dialog, click

 (**Geometry tab > Selection panel > Expand** or **Mesh tab > Selection panel > Expand**).


Select Saved Selection List tool

The **Select Saved Selection List** drop-down list allows you to view those model entities selected and selection lists that you have previously saved.

Used in conjunction with the **Save Selection List** tool, if you click the drop-down arrow all saved names are available for quick access.


Save Selection List dialog

This dialog is used to save the list of entities displayed in the **Select saved selection list** drop-down list with a specified name.

To access this dialog, click  (**Geometry tab > Selection panel > Save Selection** or **Mesh tab > Selection panel > Save Selection**).

Delete Selection List dialog

This dialog is used to delete one or more previously saved selection lists.

To access this dialog, click  (**Geometry tab > Selection panel > Delete Selection** or **Mesh tab > Selection panel > Delete Selection**).

Change Property Type dialog

Use the **Change Property Type** dialog to change the basic type of an element.

This can be helpful when importing a model which does not correctly differentiate between part and insert elements, for example.

The list of choices depend on the model's analysis technology.


Nodes


2


Modeling nodes are a type of modeling entity that are used to define a coordinate position in space. For example, nodes are used to identify the end points of a curve, or to model runner systems.


The following commands are used to create nodes and can be found at **Mesh tab > Mesh panel > Create nodes**.

 **Node By Coordinate** command is used to create a node at the coordinate position on your model where you click.

 **Node Between Coordinates** command is used to create nodes on the imaginary straight line between two coordinates that you select.

 **Nodes by Dividing Curve** command is used to create the number of equally spaced nodes that you specify on the selected curve.

 **Node By Offset** command is used to create a new node at a specified distance and direction from an existing base coordinate .

 **Node By Intersect** command is used to create a node at the intersection of two curves.


Nodes

Nodes are a type of modeling entity, used to define a coordinate position in space.




Modeling nodes

A model node is a point which occupies a coordinate position in space, and can be used as a modeling aid, for example, to identify the end points of a curve.


- 1 Make sure that your modeling workspace is open, as follows:
 - If you plan to add to an existing model, make sure that model is open.
 - If you plan to create a new model, create a new study first.

- 2 Select whichever of the node-creation methods you plan to use by clicking  **Geometry tab > Create panel > Nodes**.
 - The relevant nodes creation dialog appears.
- 3 Use the nodes creation dialog to proceed.
 - Use the appropriate **Filter**.
 - Keep in mind that the new nodes will be created in the active local coordinate system. The values shown in dialog input boxes always reflect the active local coordinate system.
- 4 Click **Apply** to create nodes.
- 5 Click **Close** when you finish creating nodes.


TIP: To quickly learn about different ways to create nodes:

- 1 Save your model.
 - 2 Click  then click  **New > Study**. This will clear your screen.
 - 3 Choose sub-menus from  **Geometry tab > Create panel > Nodes**
 - 4 Go to the **Filter** drop-down and select **Any Item**.
 - 5 Experiment by creating curves on the blank screen.
-

Nodes

You can access the nodes modeling tools from the **Nodes** drop-down menu:
 **Geometry tab > Create panel > Nodes**.

Node by Coordinate tool


The  **Node By Coordinate** tool allows you to create nodes when you are in the process of creating a new model or adding to an existing model.

The **Node By Coordinate** command is used to create a node at the coordinate position on your model where you click. To access this tool,

click  **Geometry tab > Create panel > Nodes > Node By Coordinate**.

Node Between Coordinates tool

The **Node Between Coordinates** tool allows you to create nodes when you are in the process of creating a new model or adding to an existing model.


To access this tool, click  **Geometry tab > Create panel > Nodes > Node Between Coordinates**.

The **Node Between Coordinates** command is used to create nodes on the (imaginary) straight line between two coordinates that you select. The two

coordinates need not be on an existing curve, and if there is a curve, it need not be a straight line.

You can specify the number of nodes to create between the two coordinates. They will be evenly spaced.

Nodes By Dividing Curve tool


The **Nodes By Dividing Curve** tool allows you to create nodes when you are in the process of creating a new model or adding to an existing model. To access this tool, click  **Geometry tab > Create panel > Nodes > Nodes By Dividing Curve**.


The **Nodes By Dividing Curve** command is used to create the number of nodes you specify, on the curve you select, at equal intervals.

You have the option to create nodes at the ends of the curve. These nodes will be included in the total number of nodes you specify.

Node By Offset tool

The **Node by Offset** tool allows you to create nodes when you are in the process of creating a new model or adding to an existing model. To access


this tool, click  **Geometry tab > Create panel > Nodes > Node By Offset**.


The  **Node by Offset** command is used to create a new node at a specified distance and direction from an existing base coordinate.

You can create a second new node at the same distance and direction from the first, a third new node at the same distance and direction from the second, and so on.

Node By Intersect tool

The **Node By Intersect** tool allows you to create nodes when you are in the process of creating a new model or adding to an existing model. To access

this tool, click  **Geometry tab > Create panel > Nodes > Node By Intersect**.

The  **Create Nodes by Intersect** command is used to create a node at each intersection of two curves.

Curves

3

A curve is a geometric line on your model. Curves can be straight lines between two points, or arcs comprising three or more points. Along with nodes and regions, curves are the building blocks of a model.

The three types of curve are as follows:

- **Line** - a straight line, defined by two end points or nodes
- **Arc** - a section of a circular curve, which is defined either by a center point, or an arc and angle, or three points on a plane.
- **Spline** - a cubic spline interpolation on a supplied set of points.


Curves

Use the following procedures to model curves.

Creating curves

A curve is a geometric line on your model. Curves can be straight lines between two points, or arcs made of three or more points. Curves, along with nodes and regions, are the building blocks of a model.




- 1 Make sure that your modeling workspace is open, as follows:
 - If you plan to add to an existing model, make sure that model is open.
 - If you plan to create a new model, create a new study first.
- 2 Click on whichever of the (**Geometry tab > Create panel > Curves**) sub-menus you want to use. The corresponding dialog opens.

For example,  **Connect Curves** creates a curve that connects two existing curves. It is commonly used to model cooling hoses.

- 3 Use the dialog to proceed.
 - Make sure to use the easiest method to select a coordinate, curve, or other entity.
 - Use the appropriate Filter.
 - New curves will be created in the active local coordinate system. The values shown in dialog input boxes always reflect the active local coordinate system.
- 4 Click **Apply** to create curves.


- 5 Click **Close** when you finish creating curves.

TIP: To quickly learn about different ways to create curves:

- 1 Save your model.
 - 2 Click  then click  **New > Study**. This will clear your screen.
 - 3 Choose sub-menus from  (**Geometry tab > Create panel > Curves**).
 - 4 Go to the **Filter** drop-down and select **Any Item**.
 - 5 Experiment by creating curves on the blank screen.
-



Breaking curves

Use the Break command to create new curves by breaking existing curves at their intersection.

- 1 Click  **Geometry tab > Create panel > Curves > Break Curve**.
- 2 Select the two curves and then click **Apply**.

Modeling hot and cold sprue/gates/runners

This topic describes how to specify a hot or cold sprue/gate/runner at the same time as creating the curve for the model entity.


- 1 Click  **Geometry tab > Create panel > Curves > Create Line**.
The **Create Line** dialog appears.
- 2 Click in the **Model** pane to select the start point, or type the coordinates of the start point in the **First** text box.
- 3 Click **Absolute** or **Relative** depending on how you want to specify the end point coordinates.
- 4 Click in the **Model** pane to select the end point, or type the coordinates of the end point in the **Second** text box.
- 5 Ensure the **Automatically create nodes at endpoints of curve** option is selected.
- 6 Click the **Create as** drop-down menu to display the properties available for selection. Cold runner is available by default. If you want to assign a different property, click  **Browse**.
The **Assign Property** dialog appears.
- 7 Click **Select** and choose a curve property, such as Hot gate. If you do not see the required property, click **New**, select one from the list, and click **OK**.
- 8 Click **Edit** and change the properties as required.
- 9 Click **OK** to accept your changes and close the **Assign Property** dialog.


- 10 Ensure the required property is selected from the **Create as** menu and click **Apply**.
- 11 (Re)mesh the model.

Curves


The Create Curves dialogs are used to create curves in a variety of different ways. To access these tools, click **Geometry tab > Create panel > Curves**.


Create Line tool

The **Create Line** tool allows you to create curves when you are in the process of creating a new model or adding to an existing model. To access this tool, click  **Geometry tab > Create panel > Curves > Create Line**.


The  **Create Line** command is used to create a line between two coordinates that you specify. Simply click on the coordinates on the model for easy selection.


Arc by Points tool

The **Arc by Points** tool allows you to create curves when you are in the process of creating a new model or adding to an existing model. 
(**Geometry tab > Create panel > Curves > Arc By Points**)


The  **Arc by Points** command is used to create an arc or circle, using three coordinates that you specify. Radio buttons on the panel give you the option to create an arc or a circle.


Arc by Angle tool

The **Arc by Angle** tool allows you to create curves when you are in the process of creating a new model or adding to an existing model. To access this panel, click  **Geometry tab > Create panel > Curves > Arc By Angle**.


The  **Arc by Angle** command is used to create an arc or circle, using the center point, radius, starting angle, and ending angle that you specify.


Spline tool

The **Spline** tool allows you to create curves when you are in the process of creating a new model or adding to an existing model. To access this tool, click  **Geometry tab > Create panel > Curves > Spline**.


The  **Spline** command is used to create a spline through all the points that you specify.


Connect Curves tool

The **Connect Curves** tool allows you to create curves when you are in the process of creating a new model or adding to an existing model. To access this tool, click  **Geometry tab > Create panel > Curves > Connect Curves**.

The  **Connect Curves** command is used to create a curve that connects two existing curves. This is commonly used to model cooling hoses.

Break Curve tool

The **Break Curve** tool allows you to create curves when you are in the process of creating a new model or adding to an existing model. To access this panel, click  **Geometry tab > Create panel > Curves > Break Curve**.

The  **Break Curve** command is used to create new curves by breaking existing curves at their intersection.

Regions

4

A region is an area or group of adjacent entities. Regions may include both planar and non-planar surfaces.

The boundary lines of a region may be curved or straight, but they must be fully connected and must not intersect. Along with nodes and curves, regions are the building blocks of a model.



Regions


These procedures show you the different ways you can create regions.

Creating regions

Regions are areas or selections of connected entities that may include both planar and non-planar surfaces. Regions can represent surfaces of the part, inserts, labels, or the mold. It is often convenient to model surfaces as regions so that properties can be assigned to all elements in one action.

Regions can be created from an STL model, from a Midplane or Dual Domain mesh, or manually by specifying coordinates.

- 1 Choose one of the menu items under  **Geometry tab > Create panel > Regions** (by Boundary, by Nodes, by Ruling, or by Extrusion).
- 2 Select the nodes and/or curves that form the boundary of the region.
- 3 Click  **Browse** to assign a property to the region(s).
- 4 Click **Apply** to create the region.
- 5 Click **Close** to finish creating regions.

NOTE: To create a single region using all surfaces of the part, use  **Geometry tab > Create panel > Regions > Region from Mesh/STL**.

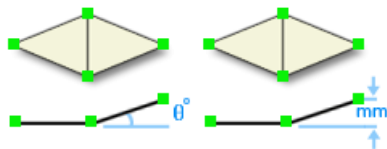
Creating mesh regions

The Create Regions dialog allows you to define areas on the mesh and then assign default properties to these regions. It is often easier to work with regions



rather than a mesh, because when using a region it is possible to group mesh elements together logically and ensure that individual mesh elements are not omitted from receiving a property.

Regions are always planar. Since mesh elements may deviate slightly from being coplanar, you can specify a tolerance within which adjacent mesh elements will be considered part of the same plane. By doing this, you can ensure that they become part of the same region.

There are two types of tolerance to choose from: angular or planar. The angular tolerance enables you to specify the maximum allowable angle between mesh elements in degrees as shown in the image below left. The Planar tolerance enables you to specify the maximum allowable distance from the same plane between mesh elements in mm, as shown in the image below, right. Angular tolerances are often used for parts with large curved surfaces. Planar tolerances are used when you want to preserve the shape of your part.



Create Regions from Mesh/STL produces one region that includes all the mesh elements.


- 1 Click  **Geometry tab > Create panel > Regions > Region From Mesh/STL.**
- 2 Click either **Planar** or **Angular** in the **Tolerance** area and enter a value.
- 3 Click **STL** or **Mesh** in the **Create from** area, depending on the type of model used.
- 4 **Optional:** Click  **Browse** in the **Selection option** group and select a different property type for the region to be created.
- 5 Click **Apply**.
Regions are created on the model as specified in step 4.


Regions

The Create Regions tools are used to create regions in a number of different ways. To access these tools click **Geometry tab > Create panel > Regions**

Region by Boundary tool


The **Region by Boundary** tool allows you to create regions when you are in the process of creating a new model or adding to an existing model. To


access this tool, click  **Geometry tab > Create panel > Regions > Region By Boundary.**

The  **Region by Boundary** command is used to define the shape of a part surface.

Region by Nodes tool


The **Regions by Nodes** tool allows you to create regions when you are in the process of creating a new model or adding to an existing model. To


access this tool, click  **Geometry tab > Create panel > Regions > Region By Nodes.**

The  **Region by Nodes** command is used to define the shape of a part surface by the nodes that you specify.

Region By Ruling tool


The **Region by Ruling** tool allows you to create regions when you are in the process of creating a new model or adding to an existing model. To


access this tool, click  **Geometry tab > Create panel > Regions > Region By Ruling.**

The  **Region by Ruling** command is used to define a region by the two separate sets of curves that you specify. To create a region by ruling, the curves must be coplanar.

Region by Extrusion tool


The **Region by Extrusion** tool allows you to create regions when you are in the process of creating a new model or adding to an existing model. To


access this tool, click  **Geometry tab > Create panel > Regions > Region By Extrusion.**

The  **Region by Extrusion** command is used to create a region along the specified vector from the curve that you specify.



Hole by Boundary tool

The **Hole by Boundary** tool allows you to create holes in regions when you are in the process of creating a new model or adding to an existing model.

To access this tool, click  **Geometry tab > Create panel > Regions > Hole By Boundary.**

The  **Hole by Boundary** command is used to define an area to remove from a part surface by the curves that you select.

Hole by Nodes tool

The  **Hole by Nodes** command is used to define an area to remove from a part surface by the nodes that you select. To access this tool, click  **Geometry tab > Create panel > Regions > Hole By Nodes**.

Manipulating model entities

5

The **Move/Copy** tools help you manipulate selected entities, or the entire model, around a specific coordinate system. These tools are useful for creating multi-cavity or family parts.

Rotating a part

The Rotate tool enables you to change the orientation of entities to align with the coordinate system. Model orientation is important for creating an effective runner system and for calculating the clamp force.

When you create a runner system, the sprue will be created along the Z-axis, which should be perpendicular to the machine plate with the injection side originating from the fixed plate. This ensures the sprue is modeled correctly. The clamp force is calculated relative to the machine plate.

NOTE: If you change the part orientation after creating a runner system, the runner system will become disconnected.

Mirroring a part

The Mirror tool enables you to create a mirror version of a cavity that has already been modeled.

NOTE: Mirroring a part usually generates a left or right hand version of the original part. Use the Rotate tool if you want to create an identical copy of the original part.

Translating a part


The Translate tool enables you to adjust individual parts in a multi-cavity layout.

Scaling a part

Scaling a model to a different system of measurement changes the size of the model.

If you use the wrong system of measurement, or units, when you import or construct a model, you can scale the model to the correct units. For example,

you can correct the size of a 10 mm wide part that is shown to be 10 inches wide.

Scaling is not the same as changing measurements to metric or English units, which can be done in the **General** tab of the **Options** dialog:  **> Options**.

NOTE: Thickness cannot be scaled because it is a property.
