Begin Creating Plastic Injection Molds

In this chapter, you review how plastic parts are manufactured, learn the qualities of a good plastic part design, and review how to analyze draft on a part. You also learn the overall process for using Autodesk® Inventor® Tooling to create and communicate the components of a mold for manufacturing plastic injection parts.

Objectives

After completing this chapter, you will be able to:

■ Describe the basic concepts of mold design.
■ Describe the characteristics and process of creating a plastic part mold design and begin the process of creating a mold assembly.
■ Create a new mold design assembly, position a part or core and cavity, and set the part material.
■ Adjust the position and orientation of a placed part and pattern the part.
Lesson: Plastic Injected Part Design

This lesson describes the process of creating injection molded thermoplastic parts and the process of creating and communicating an injection mold design for manufacturing the parts.

The production of plastic parts and products has seen a steady and steep growth since the mid-70s. The parts being designed and manufactured are increasingly more complex and demand a higher level of quality. At the same time, there is increased global competition for the jobs with reduced lead times. Because of these factors, you must know how to design a mold that produces a part that meets the design requirements as efficiently and quickly as possible.

Objectives

After completing this lesson, you will be able to:

- Describe the process of designing injection molded plastic parts.
- Explain the overall process for creating and communicating a mold design.
- Describe the differences between single, family, and multi-cavity mold designs and the benefits of each type when it comes to manufacturing the part or parts.
About Designing Injection Molded Plastic Parts

As with any area of design, having an understanding of how injected molded plastic parts are designed makes it easier to design parts that meet the requirements of their uses and to efficiently manufacture them. By understanding what unique design requirements and limitations exist for plastic injection parts, you can quickly design plastic parts that produce a final product of high quality.

In the following illustration, a digital prototype of a plastic injected part is shown with the core and cavity halves which are required to manufacture the part.

Manufacturing an Injection Molded Plastic Part

Plastics are a polymer material. There are two types of plastics, thermoplastics and thermosets. Thermoplastics are the plastic material that can be melted and formed multiple times. When this training material refers to plastics and plastic parts, it is referring to thermoplastics and the injecting of thermoplastics into a mold.

The plastic injection manufacturing process enables you to create a complex part in one manufacturing operation. Because of the variety of available plastic materials, the properties of the final part easily meet their requirements for use. These types of plastic injected parts can resist chemicals, electrical current, and the transfer of heat. The parts can also be light weight, durable, strong, and have varying finishes and colors.

The injection molding process consists of filling, packing, cooling, and opening a mold to eject a part. During the filling step, the melted plastic is forced through a sprue, runners, gates, and into the cavity. When the cavity is completely filled, pressure is maintained to pack more material into the cavity or ensure no plastic flows out.
In the following illustration, a drawing of an industrial injection molding machine is shown to help you visualize the process of the plastic pellets being heated, injected into the mold, and the mold opening to eject the part.

To create a plastic injection molded part, you need at least two halves of a mold, a core and a cavity. The cavity is the half of the mold that contains the majority of recessed machined surfaces, while the core side of the mold has the majority of protruding machined surfaces which fit close to the surfaces in that half of the cavity mold. The area of the mold between the core and cavity halves is referred to as the mold cavity. It is within this void space between the core and cavity where the part is formed. The part is formed when thermoplastic resin is injected into the mold and fills the void.

The following illustration shows the resulting filled part, sprue, and runner with a cut away of the cavity half of the mold and the core half of the mold.

A mold base is an assembly of parts that holds and separates the core and cavity to allow for efficient mass production of the part. After the plastic is injected into the void and has cooled, the mold base separates the core and cavity enabling the part to be ejected. The cavity is on the static side of the mold base and the core is on the moving side. When the core and cavity are separated, the part automatically releases itself from the core as the mold base is withdrawn. The part sticks to the cavity until it is ejected by the mold base ejection system.
Characteristics of a Good Plastic Injected Part Design

The design of a part can greatly affect the quality of the manufactured part and the cost of the mold. You can create good plastic part designs when you have a thorough understanding of the fundamental techniques and requirements for plastic part design. The following are considerations for creating a mold design:

- The most critical factor is wall thickness. Try to keep wall thickness uniform throughout the part. Parts with walls less than 1.5 mm are considered very thin.
- Apply a proper draft angle to ease ejection from the mold. 1/2 to 2 degrees is normal. Use a higher value if the part surface is textured.
- Avoid undercuts requiring slider cores when possible. This increases the complexity and cost of the mold.
- Avoid sharp corners. Plastic resists flowing into them. A radius is recommended whenever possible.
- Avoid tall cores and ribs. Taller than 5:1 is more likely to warp or flow.

The following illustrations show poor initial designs and improved versions. In these cases, the changes were made to maintain wall thicknesses in different areas of the parts.
Manufacturing a Plastic Injection Mold

There are five main manufacturing technologies that are used to machine the core, the cavity, and the different mold plates. The manufacturing methods are milling, drilling, grinding, wire EDM, and sink EDM. The manufacturing method used depends on the requirements of the designed part.

In the following illustration, an example of a mold plate A and plate B are shown with the core and cavity for that mold design. These components illustrate just some of the machining variation required in a mold design.
Examples of Injection Molded Plastic Parts

In the following illustration, each of the parts shown were designed so other parts match with them. Also considered was whether or not any internal parts might be movable parts and the locations of fasteners.
In the following illustration, an assembly file of a plastic injection mold for manufacturing plastic parts is shown.
Creating and Communicating a Mold Design

To manufacture a plastic part you must communicate the part design as well as the mold design to the vendors who manufacture the mold assembly or the different components of the mold. To create a digital prototype of a mold assembly and the documentation required to communicate the design, there are many tools and processes that you must learn. By understanding the overall process for creating and communicating a mold design, learning how to use the individual tools within the overall process can be quicker and easier.

The following illustration shows one of the many engineering drawings that can be generated to communicate the mold design. The parts list on the drawing includes the parts that need to be manufactured or purchased so the mold can be made.

The creation of a mold design starts by placing the digital prototype of the plastic part you want to manufacture. This plastic part becomes the basis for the creation of the core and cavity components. The different mold components, such as the mold base, ejector pins, and sliders, are all based on what is available from various suppliers in the industry. When the mold design is complete, you create a documentation package that can be distributed to the people who make the mold and its parts. The documentation package can be in the form of hardcopy drawings or digital files.
Process: Creating and Communicating a Mold Design

The following diagram gives an overview of the process of creating and communicating a mold design for the purpose of manufacturing an injection molded plastic part.

About Single, Family, and Multi-Cavity Molds

It is common for a mold to manufacture one or several parts at once. To accomplish this, the mold must have a core and cavity set for each part as well as several other common subsystems such as cooling channels, sliders, and ejector pins.

Definition of Single, Family, and Multi-Cavity Mold Designs

There are three types of injection mold designs that you can create, single cavity, family, and multi-cavity molds. Each type of mold has a unique result in the design and manufacturing process.

A single cavity mold is employed to manufacture one part per cycle. It is common for a larger or more complex part to be manufactured in a single cavity mold.

A family mold is used to create several nonidentical parts in one cycle. Several different core/cavity sets are incorporated into one mold design. The family of parts may constitute a kit. An example of a family mold would be a common toy model car or plane kit, or the mirrored halves of a product's exterior case.

A multi-cavity mold is employed to create several identical parts in one cycle. Several identical core/cavity sets are incorporated into one mold. It is common for these parts to be smaller or simpler in design, or to require high volume production.
Examples of Single, Family, and Multi-Cavity Molds

Single Cavity Mold
The illustration to the right shows the cavity portion of a single cavity mold for a large and complex part.

Family Mold
The illustration to the right shows the cavity portion of a family mold where the left and right half of a product shell is manufactured in one mold.
Multi-Cavity Mold
The illustration to the right shows a multi-cavity mold. In this case, four identical parts are molded per cycle. The number of core/cavity sets per mold can vary based on the size of the part, the material used, and the production volume requirements.
Exercise: Review a Completed Mold Design

In this exercise, you open an existing mold design and review the contents and the relationships of the components and files.

1. Open ..\Mold1\Mold 1.iam. This is the mold design top assembly.

2. Review the Mold Design browser.

3. In the browser under Representations, activate the design view No Mold Base. This design view turns off the visibility of all mold base components.

Completing the Exercise
To complete the exercise, follow the steps in this book or in the online exercise. In the online list of chapters and exercises, click Chapter 1: Begin Creating Plastic Injection Molds. Click Exercise: Review a Completed Mold Design.
4. Review the visible subsystems. The visible subsystems include:
   ■ Core and Cavity workpieces
   ■ Locator Ring and Sprue Bushing
   ■ Cooling Channels
   ■ Slider Assemblies
   ■ Ejector systems

5. Open ...\Mold1\Mold 1_Mold Base\DME_E 1\Mold 1_MB.iam.

6. Review the components in this assembly. This assembly consists of the mold base components. The majority of these parts are placed automatically from the Mold library that is installed with Inventor Tooling.

7. Open ...\Mold1\Mold 1_Slider Components \HASCO_Single Locking 1\Mold 1_cartridge up coat_SA2.iam.

8. Review the components in this assembly. This is one of two slider subassemblies. A slider is used to assist in the creation of molded features that are not aligned with a part's pull direction.

9. Open ...\Mold1\Mold 1_CH.ipt.

10. Review the features in this part. This part represents the cooling channels. It is generated semi-automatically in the top assembly and is used as a boolean subtraction to create channels in the workpieces.

11. Open ...\Mold1\Mold 1_cartridge up coat_CR.ipt

12. Review this part. This is the core of the core/cavity set. It is associated directly with the molded part design file. When joined with the mating cavity, the thermoplastic material is injected into the void space to mold the part.

13. Close all files. Do not save changes.
Lesson: Create a Mold Assembly

This lesson describes the creation of a new mold design assembly file and the initial steps of creating a mold design. The initial steps include placing the digital prototype of the plastic part or an existing core and cavity as well as setting the material to be used to manufacture the part.

Using the process described in this lesson enables you to generate mold designs that are associated directly with the part design file. Changes made to the part design can easily be incorporated into the mold design.

The following illustration shows a partially exploded mold assembly. In this view, you are able to see a few of the many parts that are used to create a mold for the purpose of mass producing an injection molded part.

Objectives

After completing this lesson, you will be able to:

■ Create a new assembly file based on the Mold Design template.
■ Add one or more parts to the mold design or add a core and cavity.
■ Assign the material that will be used to manufacture the part.
Creating a Mold Design Assembly File

It is important to understand and follow the initial setup process for a mold design. The naming and placing of all mold design files are determined by the information provided at the start of the process.

The Mold Design Assembly file is the top assembly for the mold design. Special templates files must be used to create the Mold Design Assembly file. Once the assembly file is created, it cannot be renamed or converted to a standard assembly file. A standard assembly file cannot be converted to a mold design assembly file. Many processes such as file naming and file structure will be performed semi-automatically when the mold design environment procedures are followed.

When you create a new mold assembly by selecting a Mold Assembly template, the Create Mold Design dialog box displays. In this dialog box you enter the name of the new mold assembly file and specify the folder where the mold files will be created.

The Mold Design environment requires the Autodesk Mold Design add-in to be installed. If the add-in is installed, it will list in the Add-In Manager.
Process: Creating a Mold Design Assembly File

The following steps provide an overview of creating a mold design assembly file to begin the creation of a mold design.

1. Configure the project file to make the Mold Library active. The Mold library is added to the Content Center library during the Mold Design add-in installation.

2. Create a new mold assembly file using one of three template files:
   - Mold Design (in).iam
   - Mold Design (mm).iam
   - Mold Design (DIN).iam

3. Enter the Mold Design File Name. This will be the name of the mold top assembly (.iam).

4. Specify the Mold Design File Location. The file Mold Design File Name.iam will be located in it.
   - Example: <Mold Design File Location>/Mold Design File Name.iam>
Mold Design Files and Types

In the Mold Design environment, two categories of files exist; custom files generated by the Mold Design environment, and component files loaded by the Content Center. Mold Design file names are automatically generated at creation and cannot be altered. File names are generated using up to four elements in the order shown below.
1. The Mold Design File Name you enter in the Create Mold Design dialog box.
2. The part file name you select using the Plastic Part tool on the Mold Layout tab.
3. Two letter file type abbreviations identifying subcategories.
4. A unique identifier appended to the file name.

Mold Design Folder Structure

The mold design file structure is contained within one top mold design folder. The top folder name is defined when you enter the Mold Design File Name in the Create Mold Design dialog box. Components are organized in subfolders identified by type within this top folder. These subfolder names are categories.
Example: Cover Mold Top Folder Contents
In this example we see the file structure of a developed mold design. This structure is built within the specified Mold Design File Location folder by the mold design environment during the design process.

- Cover Mold is the Mold Design File Name.
- Subfolder names are generated using the Mold Design File Name and category names.
- Top level file names are generated using the Mold Design File Name and two letter abbreviations for file types.

Example: Cover Mold Expanded Folders
In this example, we see the expanded folder structure under the same Mold Design File Location folder. Within each subfolder are content center components and mold subsystem assemblies.
## Mold Design File Types

The following table defines the two letter abbreviations that are appended to files. These files are automatically created and named during the mold assembly development process. This table corresponds to a similar table in the Help system.

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MA</td>
<td>Mold Design Assembly - Top level assembly containing all standard and custom components.</td>
</tr>
<tr>
<td>LY</td>
<td>Layout Assembly - Contains design files for a single plastic part. Several layout assemblies can be located under the Mold Design top assembly. The layout assembly enables you to construct a family mold for multiple plastic parts.</td>
</tr>
<tr>
<td>PZ</td>
<td>Part Zone Assembly - Contains design files for a cavity in a plastic part. Several duplicated part zone assemblies can be located under a layout assembly. The part zone assembly enables you to create the mold design for a plastic part with multiple cavities.</td>
</tr>
<tr>
<td>MP</td>
<td>A moldable part that is derived from a BP (basic part) and is scaled with shrinkage value. The mold design is based on the MP, including the core and cavity design and the design of standard components.</td>
</tr>
<tr>
<td>WP</td>
<td>Workpiece</td>
</tr>
<tr>
<td>CR</td>
<td>Core</td>
</tr>
<tr>
<td>CV</td>
<td>Cavity</td>
</tr>
<tr>
<td>IS</td>
<td>Insert Sketch</td>
</tr>
<tr>
<td>IN</td>
<td>Insert</td>
</tr>
<tr>
<td>CH</td>
<td>Cooling Channel</td>
</tr>
<tr>
<td>RG</td>
<td>Runner</td>
</tr>
<tr>
<td>MB</td>
<td>Mold Base</td>
</tr>
<tr>
<td>SA</td>
<td>Slider Assembly</td>
</tr>
<tr>
<td>LA</td>
<td>Lifter Assembly</td>
</tr>
<tr>
<td>LS</td>
<td>Lock Sets</td>
</tr>
</tbody>
</table>

Reference the Inventor Tooling help system for more information on file naming. In the Help system, click Mold Design Help > Inventor Mold Design Environment > File Naming.
Placing the Plastic Part or Core and Cavity

When you create a mold design, you start creating the mold base based on one or more parts or one or more existing cores and cavities. To do this, you need to know how to place a part or a core/cavity set into the mold design.

The following illustration shows a core and cavity for the front of a cell phone. The core and cavity were generated directly from the digital prototype of the final part.

Place Plastic Part

The first part you place is positioned automatically. The initial orientation of the placed part is based on the part's coordinate system relative to the assembly's coordinate system. It may need to be reoriented so that the part's pull direction matches the opening direction of the mold. Subsequent parts that are placed must be located manually with respect to the first part.

The following illustrations show the mold design browser, the mold model browser and the mold design file screen after placing a core and cavity into a new mold design assembly file.
Place Core and Cavity

You can place an existing core and cavity into a new mold design. It is not necessary to place a plastic part in this case. The core and cavity files can be imported from a different software package.

Access

Ribbon: Mold Layout tab > Mold Layout panel

Toolbar: Mold Design

Keyboard: PP
Access

Place Core and Cavity

Ribbon: Mold Layout tab > Mold Layout panel

Toolbar: Mold Design
Keyboard: CC

Process: Placing a Plastic Part

The following steps give an overview of placing a plastic part in a mold design.
1. Start the Place Part tool.
2. Select the part file (IPT) of the part to be molded.

Set the Part Material

Material selection is an important decision in the design and manufacturing of plastic parts. The service and function of the part is a major factor in determining the material requirements. The primary benefit of selecting a material during the design of the mold is the part and mold can be analyzed for material fill and quality.

Access

Select Material

Ribbon: Mold Layout tab > Mold Layout panel

Toolbar: Mold Design
Keyboard: SM
Select Material

After a part has been placed into the mold assembly, you can select the material to use to manufacture that part. The material you want to use can be selected from a list of commonly used materials or from a list of materials after having selected the manufacturer. To help you locate the manufacturer and trade name of a material that matches your use criteria, you can conduct a search of the more than 7,700 materials listed in the Mold Design database. Prior to accepting the material for the part, you can select to review the property and manufacturing information for the selected material or review a report of the material information.

A list of the most commonly used materials. You may manually add and remove entries in this list.

Lists all materials in the database sorted by manufacturer then trade name.

Lists selectable trade names available from the selected manufacturer.

Opens an advanced search criteria dialog box.

Lists the details of the selected material.

Generates an on-screen material data report.

Identifies the polymer family that the selected material belongs to.

A relative value calculated from predicted injection pressures and cooling times for different geometries.

When checked, inserts the selected material into the commonly used materials list.
Process: Select Part Material

The following steps provide an overview of assigning which plastic material is used to manufacture the part.

1. Start the Select Material tool.
2. In the Select Material dialog box, select the material to use in the manufacturing of the part.
Exercise: Create a Mold Assembly

In this exercise, you create a new mold design, place the part to be manufactured into the mold design and then select the plastic material that is used to manufacture the part.

1. Start a new assembly file using the template named Mold Design (mm).iam.

2. In the Create Mold Design dialog box
   - For the Mold Design File Name, accept the default name of Mold Design1.iam
   - Accept the Mold Design File Location, which in this case is the project workspace.
   - Click OK

Note: It may take a few moments for the mold design environment to load.

4. In the project workspace, select MouseBase.ipt.
5. The part will display in this orientation.

6. Close all files. Do not save changes.
Lesson: Adjust and Pattern a Placed Part

This lesson describes reorienting and repositioning a placed part for optimal manufacturing. This lesson also describes the patterning of a part for the purpose of creating a multi-cavity design of the same part.

It is important to know how to pattern a part in a mold because it offers many advantages over placing the part multiple times. All part instances in a pattern maintain associativity with the original part so that all instances of the part in the pattern update when the original part is modified.

The following illustration shows a pattern created from one placed part.

Objectives

After completing this lesson, you will be able to:

- Adjust the orientation of a placed part.
- Adjust the position of a placed part.
- Pattern a part to create a multi-cavity design.
Adjusting Orientation

When a new mold design is created, the opening direction is aligned to the positive Z axis by default and cannot be changed. When placing a part into a mold, it is common that the pull direction of the part file does not align to the mold opening direction. It is often necessary to reorient the placed part to align its pull direction to the mold opening direction. You need to know how to use the Adjust Orientation tool to reorient placed parts in a mold assembly.

In the following illustration, the initial orientation of the part is shown on the left and an adjusted orientation is shown on the right. This part had its orientation adjusted to make it easier and more cost effective to create the mold.

Access

Ribbon: Mold Layout tab > Mold Layout panel

Toolbar: Mold Design

Keyboard: AO
Adjust Orientation

You use the Adjust Orientation tool to review the opening direction of the mold and realign the part in 3D space. You adjust the orientation of a part by rotating it a specific number of degrees or aligning part geometry with an axis. The axes you revolve or align to are the X, Y, and Z axis. Rotation is accomplished when you select the Rotate Around Axis method option to adjust orientation. Alignment of part geometry to an axis occurs when you use the Align With Axis method option.

When you use the Rotate Around Axis method option, you can rotate the part about a selected axis by any amount. To have the angle you enter effect the part orientation, you do have to click Apply. When you use the Align With Axis option, the part automatically adjusts its orientation after you select the geometry on the part.

When you start the Adjust Orientation tool, along with the Adjust Orientation dialog box, a bounding box with black arrows is displayed in the graphics window around the part. The black arrows indicate the pull or opening direction of the mold. The red, green, or blue arrow indicates which axis is currently selected in the Adjust Orientation dialog box.
Process: Adjust Orientation

The following diagram gives an overview of adjusting the orientation of a placed part in a mold design.

Start the Adjust Orientation tool.

Select the adjustment method.

Select the axis you want to rotate around or align to.

Rotate Around Axis Align with Axis

Enter the number of degrees to rotate.

Select the face, plane, or edge that needs to align with the selected axis.

Click Apply to adjust the part based on the entered angle.

Adjusting the Position

When you place multiple parts into a family mold, the first placed part is positioned at the origin of the assembly. Subsequent placed parts must be positioned manually. Adjusting the position of parts in a mold is necessary for designing runners and gates in the next steps of the mold design process. To optimize this aspect of the mold design it is important for you to know how to use the many options available to adjust the part positions in a family mold.

The illustrations below show the family of parts before and after adjusting a part position.

Part family before adjusting the part position.  Part family after adjusting the part position.
Access

Adjust Position

Ribbon: Mold Layout tab > Mold Layout panel

Toolbar: Mold Design
Keyboard: AP

Adjusting the Position

You use the Adjust Position tool when you need to move the position of one part in a mold design with multiple placed parts. The Adjust Position tool has several options for adjusting the position of a part. The Adjust Position tool enables you to:

- Align a feature of one part to a feature of another.
- Align the XY plane with a reference point.
- Align the center of the selected part with a reference point.
- Move a selected part in the X, Y or Z axis by a specified amount.

When you change the position of a part, you select the part to move and a reference part. The select option titled Plastic Part is for specifying the part to be moved. The select option titled Reference is for specifying to which part the moving part should relatively move. Only one Adjust Position can be applied to a part.

![Adjust Position dialog box]

You cannot adjust the position of parts in a single cavity mold or patterned part.
Process: Adjusting the Position of a Part

The following steps give an overview of adjusting the position of a part in a mold design when more than one part has been placed.

1. Start the Adjust Position tool.
2. Select the adjustment method.
3. Select the part to adjust. This is the part to be moved.
4. Select a part as the reference part. This part will remain stationary and the other part will move with respect to it.
5. Enter the required offset values.

Patterning a Part

When creating a multicavity mold you must create a pattern from a placed part. There are several pattern options available to you in the Pattern dialog box. It is important to know how to use the Pattern tool to obtain the best mold design options.

This image shows four core/cavity sets in a pattern for a multi-cavity mold.

Pattern Parts

You use the Pattern tool to duplicate the same part or core and cavity in the design multiple times in a rectangular, circular, or variable pattern. Patterned parts maintain their associativity between the original part model and multiple cavities in a mold. When the part design changes, all of the patterned core/cavity sets update accordingly.
Lesson: Adjust and Pattern a Placed Part

Access

Pattern

Ribbon: Mold Layout tab > Mold Layout panel

Toolbar: Mold Design
Keyboard: PT

Pattern

When you choose to create a rectangular pattern, the orientation of the duplicated parts in the pattern is based on the selected Pattern Type option. The duplicated parts either have the same orientation, have an orientation mirrored about the X axis, or they have an orientation mirrored about the Y axis. The number of duplicated parts and the distance between them is based on the quantity and distance values you enter in the X Direction and Y Direction areas of the dialog box.
The circular pattern tool creates multiple instances of the selected part, called elements, to create a circular pattern of parts in the mold. The patterned parts maintain their associativity with the original part and update accordingly when the original part design changes. The pattern type option in the circular pattern tab affects the orientation of the parts within the pattern. Additionally, you can specify the part count for the pattern, the radius for the pattern, and the angle between the elements in the pattern.

The variable pattern enables you to create an irregular pattern. Using the variable pattern, you use an instance of a part, called an element, as a starting point. You add additional instances of the part to establish a variable pattern by right-clicking in the table and then clicking Add on the shortcut menu. You enter the offset values to establish the distance between the selected element and the new element.
Process: Patterning a Part

The following steps give an overview of patterning a part.
1. Start the Pattern tool.
2. If more than one part is placed in the design, select the part you want to pattern.
3. Specify the pattern type and values.
Exercise: Adjust and Pattern a Placed Part

In this exercise, you adjust the orientation of a placed part and pattern that part so you can ultimately create a multicavity mold design.

3. Review the current orientation of this part relative to the black pull direction arrows. The orientation of this part will need to be changed.

The completed exercise

Completing the Exercise

To complete the exercise, follow the steps in this book or in the online exercise. In the online list of chapters and exercises, click Chapter 1: Begin Creating Plastic Injection Molds. Click Exercise: Adjust and Pattern a Placed Part.

1. Open ...\MD-Pattern Part\MD-Pattern Part.iam.

2. On the ribbon, click the Mold Layout tab > Mold Layout panel > Adjust Orientation.

4. In the Adjust Orientation Dialog box:
   - For the Method, select Rotate Around Axis.
   - Select X as the Axis of Rotation.
   - Enter 90.00 deg as the rotation value.
   - Click the green check box to reorient the part.

5. The orientation of the part now aligns with the opening direction of the mold.
6. In the Adjust Orientation dialog box:
   - Select Rotate Around Axis as the method.
   - Select Z as the Axis of Rotation.
   - Enter -90.00 deg as the rotation value.
   - Click the green arrow to reorient the part.

7. With the part now oriented as shown, click Done to close the Adjust Orientation dialog box.

8. On the ribbon, click the Mold Layout tab > Mold Layout panel > Pattern.
9. Since there is only one part in the assembly, it is selected by default.

10. In the Pattern dialog box, Rectangular tab:
    - Select X Balance as the Pattern Type
    - For the X Direction, enter 3 for the pattern count and 90 mm for the pattern distance.
    - For the Y Direction, enter 2 for the pattern count and 140 mm for the pattern distance.
    - Click OK to generate the pattern.

11. Verify that the pattern appears as shown.

12. In the assembly browser:
    - Expand MouseBase and Pattern
    - Right-click Pattern
    - Select Edit Feature from the shortcut menu.
13. In the Pattern dialog box:
   ■ Edit the X Direction distance to 80 mm.
   ■ Edit the Y Direction distance to 120 mm.
   ■ Click OK to close the Pattern dialog box.

14. Verify the pattern appears as shown.

15. Close all files and do not save.
Challenge Exercise

In this exercise, you use what you learned to create a new mold assembly, place and orient the part, and assign the material that will be used to manufacture it.

The completed exercise

Completing the Exercise
To complete the exercise, follow the steps in this book or in the online exercise. In the online list of chapters and exercises, click Chapter 1: Begin Creating Plastic Injection Molds. Click Challenge Exercise: Begin Creating a Plastic Injection Mold.

1. To begin creating a new assembly for the mold design:
   - Create a new assembly using the Mold Design.iam template.
   - For the mold design file name, enter MD-Kappe1.iam.
   - For the file location, enter MD-Kappe1 for the Workspace subfolder.

2. Use the Plastic Part tool to insert kappe-1.ipt.

3. Use the Adjust Orientation tool to align the identified edge with the Y axis.

4. Use the Adjust Orientation tool to align the identified face with the Z axis. Flip the results so the selected face is on the bottom.
5. Rotate the orientation as needed to achieve the final results as shown relative to the X, Y, and Z axes.

6. For the material the part will be manufactured in, specify Alpha-Gary's 2215/3X-70.
Chapter Summary

In this chapter, you reviewed how plastic parts are manufactured, learned the qualities of a good plastic part design, and reviewed how to analyze draft on a part. You also learned the overall process for using Autodesk Inventor Tooling to create and communicate the components of a mold for manufacturing plastic injection parts.

Having completed this chapter, you can:

■ Describe the characteristics and process of creating a plastic part mold design and begin the process of creating a mold assembly.
■ Create a new mold design assembly, position a part or core and cavity, and set the part material.
■ Adjust the position and orientation of a placed part and pattern the part.
Chapter 1: Begin Creating Plastic Injection Molds