Profile Plastics, Inc.

Founded in 1960, Profile Plastics is an award-winning heavy-gauge thermoforming company that specializes in providing design for manufacturing (DFM) assistance. Helping customers optimize part designs minimizes trial and error, leads to better manufacturing processes, and results in superior end products.

The following Thermoforming Design Guide provided by Profile Plastics has been prepared for designers, purchasing managers, and other professionals in the manufacturing industry who need to know more about the design and use of formed thermoplastic parts.

If you have questions or would like to discuss any of the following information further, please contact us. Profile Plastics is ready to respond to your inquiries and to help you understand more about the thermoforming design process.

65 Waukegan Road  P: 847.604.5100
Lake Bluff, IL 60044  F: 847.604.8030

The Guide is organized into the following sections. To go directly to a given section, click on the section name.

Table of Contents

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Processes Overview</td>
<td>3</td>
</tr>
<tr>
<td>Vacuum Forming</td>
<td>5</td>
</tr>
<tr>
<td>Pressure Forming</td>
<td>6</td>
</tr>
<tr>
<td>Twin-Sheet Pressure Forming</td>
<td>7</td>
</tr>
<tr>
<td>About Materials</td>
<td>8</td>
</tr>
<tr>
<td>Draw Ratio</td>
<td>9</td>
</tr>
<tr>
<td>Radii And Chamfers</td>
<td>11</td>
</tr>
<tr>
<td>Undercuts</td>
<td>13</td>
</tr>
<tr>
<td>Draft</td>
<td>15</td>
</tr>
<tr>
<td>Textures</td>
<td>16</td>
</tr>
<tr>
<td>Ribs and Louvers</td>
<td>17</td>
</tr>
<tr>
<td>Fastening/Inserts</td>
<td>18</td>
</tr>
<tr>
<td>Dimensional Tolerances</td>
<td>19</td>
</tr>
<tr>
<td>Next Steps</td>
<td>21</td>
</tr>
</tbody>
</table>
Thermoforming Processes Overview

The process of heavy-gauge thermoforming involves heating extruded thermoplastic sheet and then applying a force in the form of pressure, a vacuum, or, in some cases, both to form the sheet onto or over a custom 3D mold, often called the tooling. After the sheet is formed into a part, it needs to be trimmed using CNC equipment and a CNC trim program specifically developed to the part's requirements. Additional finishing steps may include painting, applying specialty coatings, and silk-screening.
Advantages of thermoforming:

- **Durability and industry compliance.** Thermoform material is extremely versatile and can meet the industry demands and performance requirements for everything from medical devices and fitness equipment to electrical enclosures and industrial safety equipment.

- **Volumes in the hundreds to thousands.** The economics of the thermoforming process make it ideal for moderate volumes.

- **Lightweight yet strong.** Thermoplastic materials can equal or exceed the strength of metal or fiberglass and yet are lighter.

Types of thermoforming processes:

The specific type of thermoforming used depends largely on the requirements of the part's specifications. There are three major types of thermoforming:

1. Vacuum forming
2. Pressure forming
3. Twin-sheet forming

Next, let's discuss each of these processes.
Vacuum forming is the process of evacuating air from the sealed space between the hot sheet and the mold, thus allowing atmospheric pressure (14.7 p.s.i.) to force the sheet to conform with the contour of the mold.

The most basic of the thermoforming processes, vacuum forming can be ideal when you need durable plastic parts that don’t require sharp non-tool side features.

**Applications:**
- Non-critical appearance covers, dunnage trays, internal covers
- Competes with sheet metal and fiberglass
- A low- to medium-volume process

**Vacuum forming can be an excellent solution for:**

- **Quick turnaround.** For the right application, vacuum forming is a great option for compressed lead times.

- **Lower cost tooling and prototyping.** Vacuum forming can be highly cost-effective for developing tools and prototyping parts. Also, the reasonable tooling costs allow for modifications without excessive expense.

- **High-gloss or clear plastic parts.** The vacuum forming process is ideal for high-gloss or clear plastic parts.

- **Precision forming on only one side.** In the vacuum forming process, only one side of the part touches the tool, making it ideal for distortion forming or clear parts.

- **Specific surface textures and printed features.** The nature of the vacuum forming process makes it effective for including printed features and creating a variety of surface textures.

- **Simple, large cosmetic parts.** Vacuum forming can achieve the sweet spot for cost-effective, non-intricate plastic parts that also need to be attractive.
Pressure forming is the process of applying compressed air (20–120 p.s.i.) to a hot sheet, thus forcing it to conform to the contour of the mold. Evacuation of the air between the sheet and the mold is required.

**Applications:**
- High-appearance covers, Shrouds, Enclosures, Medical equipment covers, Automotive interior panels, computer housings, microprocessor-based equipment, typically non-structural
- Part size is usually 12'' x 12'' and larger
- Competes with injection molding
- A low- to medium-volume process

**Pressure forming can be an excellent solution for:**

- **Quick turnaround.** For the right application, vacuum forming is a great option for compressed lead times.

- **Highly engineered cosmetic parts.** Pressure forming works exceptionally well for cosmetic parts that have molded in color and demand an extremely detailed surface.

- **Physically large parts.** For parts larger than 24'' x 24'', the economics of pressure forming become especially attractive compared to other plastic processes.

- **Rapid development and speed to market.** With pressure forming, a single part can be manufactured in as few as four to six weeks.

- **Color-matched parts with sharp, flawless detail.** Thermoplastic material with molded-in color is available in a spectrum of colors. And with the pressure forming process, parts have no gate marks, knit lines, sink marks, ejector marks, etc.

- **Multi-part programs with molded-in color.** Pressure forming can be ideal for two or more mating parts, where each one increases the degree of difficulty exponentially. In addition, using molded-in color material saves on painting time and costs.

- **Design flexibility.** Pressure forming works effectively for parts needing decorative materials or coatings. The technique also works great for parts with zero draft, complex undercuts, molded-in features, and trim features that need to be easily adjusted.
Twin-Sheet Pressure Forming

Twin-sheet forming is the simultaneous heating of two thermoplastic sheets and forming them between two pressure-forming tools. First, a vacuum force is applied independently to pre-form each sheet. Next, the tools are quickly brought together, and compressed air (20–120 p.s.i.) is injected into the space to force the material to fuse together.

Applications:
- Enclosures that require structural rigidity; appearance items that require high surface detail and yet are lightweight
- Competes with blow-molding and rotational-molding
- A low- to medium-volume process

Pressure forming can be an excellent solution for:

Quick turnaround. For the right application, vacuum forming is a great option for compressed lead times.

Hollow parts. The twin-sheet process creates a hollow part, which can allow for specific insulation, air flow, and/or rigidity needs.

Highly detailed cosmetic parts. Thanks to the advantages of pressure-forming used in the twin-sheet process, highly detailed—and lightweight—cosmetic parts are possible.

Lower cost tooling. Tooling costs for twin-sheet forming are lower than options like blow molding.

Quick turnaround for parts at a lower cost. Twin-sheet forming can produce parts at a lower cost and at a faster rate than rotational molding—and at higher volumes.

Difficult draw ratios. Twin-sheet forming makes it possible to obtain more difficult draw ratios than with other manufacturing options.

Flexibility with seam lines. To suit design preferences, welds and parting lines can vary in location around the perimeter of the finished part.

Structural integrity. The twin-sheet process enables structural integrity to be designed into the part resulting in lightweight, rigid parts.

Multi-colored parts. Parts can be produced from two different-colored sheets.
Thermoforming requires a previously extruded sheet. In the process the two operations—i.e., sheet extrusion and forming—are uncoupled. This simplifies the process but does add to the cost.

About Materials

Costs are increased because of the extra energy required to heat the polymer twice and because of the extruders, which are commonly custom processors that also need to generate profits to continue to exist.

On the other hand, gauge changes, color changes, and even material (i.e., polymer) changes can be done easily in thermoforming because, in essence, neither the oven nor the mold care what the sheet looks like.

Because the final part almost always weighs less than the starting sheet, the trim must be captured and returned to the sheet extruder for reprocessing. This results in a “scrap credit,” which is included in the price of every part.

Material Selection

Almost any thermoplastic can be thermoformed. As with any manufacturing process, there are wide variances in the cost of different materials due to the polymer cost, the temperature of processing, the crystallinity of the resin, or, more generally, the degree of difficulty of extruding the sheet.

The most common resins—and, therefore, the most cost-effective ones are:

- FR ABS
- ABS
- Kydex
- Polypropylene
- HIPS
- TPO
- HDPE
- HMWHDPE
- PMMA
- Polycarbonate
**Draw Ratio**

**GUIDELINES**
The deeper or taller the part, the heavier the starting gauge of sheet required. Allowing the part or any feature of the part to be narrower than it is tall will thin the sheet at a much quicker rate.

**OVERVIEW**
The draw ratio is the key to understanding thermoforming processes. The part has a finite amount of surface area that needs to be covered by a flat two-dimensional sheet. When the sheet is heated and forced over or into a mold, it must stretch to conform to that shape. As the sheet stretches it thins out. Local design features on the part may cause the sheet to thin at a greater rate than in adjacent areas.

**Specifics**
The draw ratio can be described numerically if the surface area can be calculated. The formula for expressing the draw ratio is as follows:

\[
\text{Draw Ratio} = \frac{\text{Surface Area of the part}}{\text{Footprint of the part}}
\]

**Example #1**
Assume a part is 10” x 12” x 2” deep. Therefore the Draw Ratio will be:

- Surface Area = 2(10” x 2”) + 2(12”x2”) + 10” x 12” = 40” + 48” + 120” = 208”
- Footprint = 10” x 12” = 120”
- Draw Ratio = 208”/120” = 1.7

If the desired ending wall thickness of the part is 0.100”, use the draw ratio as follows to estimate the starting gauge of the sheet:

\[
\text{Draw Ratio} \times \text{Desired Finished Gauge} = \text{Minimum Starting Gauge}
\]

(1.7 x 0.100” = .170” Assuming perfect material distribution.)
Example #2

Assume a part size of 10” x 11” x 5” deep.
Surface Area = 2(10” x 5”) + 2(11” x 5”) + (10” x 11”) = 300”
Footprint = 10” x 11” = 110” Draw Ratio = 2.73

If the desired ending wall thickness is 0.100”
use the draw ratio value as follows:

\[ 2.7 \times 0.100’’ = .273’’ \text{ starting gauge. Assuming perfect material distribution.} \]

The above examples ignore the effect that a specific feature on the part (e.g., a localized severe draw like a sharp corner) may have on the thinning of the sheet. The draw ratio is designed to get the part designer in the ballpark when calculating the necessary starting gauge.

Many part quotations will have two gauges specified (a high and a low) because of the difficulty in predicting the proper starting gauge. Because the starting gauge is so critical to the cost of the part, it is important to get proper feedback from the thermoformer when reviewing draw ratio.

There are many thermoforming techniques and mold designs used to help the sheet stretch as uniformly as possible. A competent thermoformer will be able to implement them in a project with a difficult draw ratio.
GUIDELINES
Avoid a sharp three-sided corner by using a radius or chamfer. The radius at the bottom of the draw is most critical. The deeper the part, the larger the radius or chamfer required.

OVERVIEW
The key to good part design in thermoforming is understanding the need for a properly sized radius or chamfer. These features are typically needed to allow for part strength, retention of material thickness, and/or aesthetics.

Specifics
One of the most difficult features in thermoforming is the three-sided sharp corner in a female mold. This feature accentuates the draw ratio because it forces the material to cover the three walls as it is pushed into the corner.

The material appears to stretch and thin out at a geometric rate, usually causing the material to either thin to an unacceptable ending gauge or actually tear and create a hole in the part.

A quick way to check if this condition is occurring is to hold a part up to a light source and inspect the corners to see if the gauge is so thin that light can be transmitted through the part.

Many times the tool is constructed ignoring this problem, necessitating either a heavier starting gauge to be used or an adhesive filler applied to the inside of the part to back up the corner and add strength. These "cures" will add significant cost to the part and should be avoided if at all possible.
A common design technique is to use radii and/or chamfers on the part, preventing the material from having to continue deeper into the corner, thus arresting the thinning that would normally occur.

The other advantage of radii and chamfers is that they distribute stress over a larger area than a sharp 90-degree corner. A chamfer does not distribute the stress as well as a radius, but it gives the designer the option of sharp corners at the transition points of the chamfer. Where a three-sided corner does occur, one large radius with a chamfer or smaller radius on the other edges is often sufficient to solve the thinning and strength problems that occur.

As the draw ratio gets larger, the radii will almost always have to be increased. Use this chart as a very rough rule of thumb to help determine the approximate radius you may need:

<table>
<thead>
<tr>
<th>Depth of Part</th>
<th>Radius</th>
</tr>
</thead>
<tbody>
<tr>
<td>0&quot; - 3&quot;</td>
<td>.015&quot; - .125&quot;</td>
</tr>
<tr>
<td>3&quot; - 6&quot;</td>
<td>.125&quot; - .250&quot;</td>
</tr>
<tr>
<td>6&quot; - 12&quot;</td>
<td>.250&quot; - ?</td>
</tr>
</tbody>
</table>

Sometimes it may be necessary to prototype a particular corner or feature of the part prior to the start of mold construction. This is usually a quick method of answering any questions regarding material thickness.
Undercuts

GUIDELINES
Keep the distance that the undercut projects into the part to a minimum. Typical undercut sizes are .375", with some localized tabs of up to 1". Because the undercut will require even more stretching of the sheet stock, it is important to keep the draw ratio in mind when designing for undercuts.

OVERVIEW
Undercuts are a feature that can be added to thermoformed parts very cost-effectively. Undercuts offer increased part strength, a locating edge, a fastening point, and/or the ability to hide a trimmed edge. Tooling costs will be increased, but will still be less than using injection molding or structural foam molding tooling.

Specifics
Typically most undercuts are an inward facing flange. However, other types of undercuts might include a reverse drafted wall, a molded-in countersink, or a design line that is not parallel to the direction of pull out of the mold.

Because these features increase the surface area of the part, they increase the draw ratio of the part. One of the most common requests is to carry an undercut flange into one or all four corners of a part.

The problem this presents is that it causes the material to stretch even more in an area that is typically the thinnest on the part if it is a female tool. By stepping the undercut back in the corners, you allow for better material distribution.
Some undercuts do not require the mold to be collapsible or removable. The undercut may be small enough or the material flexible enough to allow the part to strip out of the mold. This is more likely to occur in a female mold because the part will shrink away from the sidewalls of the tool, as opposed to a male mold which finds the material shrinking tighter around it.

If the undercut requires a moving section in the mold, you must allow for a parting, or witness, line on the part. Since the parting line can be hidden at the point at which the part turns in, this is not normally a problem on an undercut, which is an inward facing flange.

But on other undercuts, there needs to be an allowance for the parting line. Many times the parting line is used as a point of demarcation between a textured and untextured surface.

The safest way to incorporate an undercut in the mold is with an articulating section. In the past, removable loose pieces have been used and placed back in the mold once they were freed from the part prior to the next shot.

Because of the risk of tool damage due to improper alignment or the marring of a textured surface, the best long-term approach is to incorporate the articulation of the undercut section with the controls of the forming machine operating it automatically. A quality thermoformer will have this capability.
GUIDELINES
For parts that are formed into a female mold with a texture, allow one degree of the draft for every thousandth of an inch of texture depth. With parts formed over a male mold, allow three degrees of a draft angle as a minimum.

OVERVIEW
The need for a draft angle is driven by the coefficient of thermal expansion of the plastic. As the part is held in the mold, it is cooled below the set temperature. This temperature change can be anywhere from 100 to 300 degrees, depending on the resin.

This change in temperature coupled with the coefficient of thermal expansion will cause the part to shrink. The draft also allows for better material distribution by opening up a corner area to allow clearance for a plug-assist to push material.

Specifics
Parts can be molded with little or no draft. However, there is a high probability that the part will not release from the mold or will have severe scuffing from any texture that is in the mold.

By designing in the draft angle, the part is able to release from the mold much sooner in the release cycle. The greater the draft, the quicker the release and the lower the risk of part hang-up or texture scuffing.

The draft also "opens" up a corner (two- or three-sided) and allows for a better draw ratio. The drafted wall also allows for an assist plug to move material down into the mold with less risk of the plug hitting the sidewall.

Almost every plastic molding process requires a draft. In thermoforming, the advantage of a one-sided molding process becomes apparent with a draft. In a female mold, the material wants to shrink away from the sidewall of the mold. There is no mold "core" to prevent it from doing so.

In this respect, the shrink of the part actually helps keep the draft requirement to a minimum. On a male mold, the part actually shrinks tighter on the mold making the draft requirement greater.

A texture on the mold actually represents a series of undercuts in the mold. The deeper the texture, the greater the undercut and the greater the draft angle that will be required. Because on female molds the part shrinks away from the mold, there is less chance the part will scuff during release.

A final note on draft: Be sure that there are no male features on the female mold that will prevent the part from shrinking away from the sidewall. A male section at the bottom of a female mold will be a big problem unless allowance for the adequate draft has been made.
Textures

GUIDELINES
The finer textures (.002" or less) are difficult to form. Coarser textures allow for better replication, and they also cover many mold or sheet imperfections.

OVERVIEW
Texture has a direct effect on the ability to evacuate the air between the sheet and the mold during the forming cycle. Some textures actually trap air causing voids in the texture of the part. There are many texture patterns available which are thermoformable.

Specifics
When the air is evacuated between the mold and sheet, there is a need for a clear path for the air to move away. While the sheet is suspended above the mold, this is not a problem. When the sheet touches the mold, the air is trapped and needs a way to move toward an evacuation hole.

In the instance of a texture that is designed to imitate a splatter paint finish, the mold will actually have on its surface a series of small recesses that, once the material covers them, will not allow the air to escape. Because of the trapped air, the sheet will not conform to the textured recess and the part will have a flat or ill-defined surface.

The ideal texture will be a continuous pattern of interconnecting ridges or recesses. This pattern allows for the air to move along a path to a nearby evacuation hole. Cast aluminum molds have a rougher surface finish and, usually, a coarser pattern is required to cover up those imperfections.

The size of the evacuation holes will also determine the type of texture required. A .060" hole with a .003" deep texture will stand out. Typical evacuation holes are .015" to .035". Very fine textures with depths of .002" or less will highlight holes of this size.
Ribs and Louvers

GUIDELINES
The distance between each rib or louver should be greater than or equal to its depth.

OVERVIEW
The ability of the material to form into the recesses created by the mold will determine the dimensioning of the ribs or louvers. Typically the louvers are molded in and trimmed off from the back of the part.

Specifics
The ability to form in features relates directly to the discussion of draw ratio (see above). The material can be pushed into a recess, but it might thin out to an unacceptable gauge. In other instances, the sheet is unable to form into a recess because it is thicker than the recess and the hot strength of the material will not allow it to form into the recess.

Louvers can also be trimmed in with a CNC router. This, however, does not allow for a molded-in return, which would increase the strength of the louver and improve the appearance.
GUIDELINES
Thermoforming allows for molded-in inserts or bosses. Many different types of fasteners are available to be added to the part after molding and trimming. Avoid the bonding of blocks; this adds cost to the part.

OVERVIEW
Fastening is one of the most difficult issues to resolve in the design of thermoformed parts. This is because it is not possible to mold in a boss or insert without it being visible on both sides of the part.

Specifics
Using bonded PVC blocks with a brass insert can also be done, but there is a cost tradeoff. The cost of the PVC, the labor to bond it in place, the cost of the adhesive, and the labor to machine the block to the correct height and add a hole for the fastener or insert—these all are significant. Add to this the greater probability of a lower yield rate due to the number of additional operations.

For reasons like the above, the simplicity of a molded-in insert has real advantages.

There are some fasteners that work well with thermoformed parts. A qualified thermoformer should be willing to assist you in working out the fastening requirements of your parts.
GUIDELINES
For molded-in dimensions of parts from a machined aluminum mold, use +/-0.015" for the first inch, adding +/-0.001" for each subsequent inch.

For molded-in dimensions of parts from a cast aluminum mold, use +/-0.030" for the first 16", adding +/-0.001" for each subsequent inch.

Trimmed dimensions, regardless of the mold but using CNC trimming equipment, should have a general tolerance of +/-0.015". For hole diameters use +/-0.005".

OVERVIEW
Thermoformed part tolerances are determined by the coefficient of thermal expansion of the resin, extrusion conditions, type and temperature of the mold, consistency of the forming process, the method of trimming, and quality of trimming fixtures.

Specifics
Material Shrink
Most thermoforming resins have a coefficient of thermal expansion in the range of 0.000060" to 0.000120" per °F per inch [6.0–12.0 x 10^-5 in./°F/in.]. This will be a more significant factor when the part is large and the “in use” temperature of the part varies.

On parts over 48", it is good practice to add a note to the drawing specifying a temperature at which the dimensions should be measured. We have found 40–50°F temperature variances in our plant from summer to winter.

Material Sheet Extrusion
Extrusion quality will affect the part in many ways. The control of the extruder from run to run is very important. Changes in extrusion speed, direction, temperature, and the gauge will modify the amount of stress that the sheet has when it is delivered to the thermoformer.

Differences in that stress will change the rate at which the part molds, thus changing the dimensions. Molds must be temperature controlled with internal cooling channels to allow for consistent mold temperature.
Tooling Materials
Aluminum is the material of choice because its very high coefficient of thermal conductivity allows for very consistent cooling cycle times through the entire production run of parts.

Wood, epoxy, or plaster do not allow for this. Because of the amount of shrink that takes place at the aluminum foundry when a mold is cast, the typical tolerances (from the print to the mold) begin at +/- .030” for the first 16”, adding +/- .001” for each subsequent inch.

Forming Process
The forming process must be very consistent from run to run to ensure dimensional consistency. The use of digital controls on the forming machine allows for a high degree of accuracy in the cycle times.

A change in the amount of time the part spends in the mold has a direct effect on the amount of part shrinkage that occurs. The part must be held in the mold until the set temperature of the resin has been reached.

A quality thermoformer will have fully digital controls on their forming equipment, along with the capability to continually monitor the sheet temperature during the heating cycle.

Trimming Process
CNC trimming allows for tighter tolerances and consistent parts. A typical CNC trimming machine controls the movements to +/- .0002”; the trimming machine itself is designed for +/- .005”.

For cost effectiveness, we use holding fixtures that allow a tolerance of +/- .015”. Hole sizes and hole patterns can be held to a tolerance of +/- .005”. To try and hold tighter tolerances would require fixturing approaching that of the original mold cost.
YOUR NEXT MOVE: seek early support for best results

Thermoplastic material and the thermoforming process can produce exceptional parts for a wide range of applications. The information above is intended to help you in the design and ultimate use of heavy-gauge thermoplastic parts.

For the best results, be sure to reach out to a qualified heavy-gauge thermoformer as early as possible in your design process. Early supplier involvement, especially in the form of design for manufacturing assistance, can help you optimize the ultimate performance of your part and improve the efficiency and cost-effectiveness of your overall project.

Profile Plastics has been providing thermoforming solutions to a wide range of customers since 1960. When involved from concept to production, we can save customers money on the total cost of their project and enable them to have a higher quality end product.

Contact us today if you have design questions or would like to discuss your next thermoplastic part project.

65 Waukegan Road  P: 847.604.5100
Lake Bluff, IL 60044  F: 847.604.8030
We Deliver Parts... Under Pressure

Profile Plastics, Inc.
65 Waukegan Road
Lake Bluff, IL 60044

P: 847.604.5100
F: 847.604.8030

Profile Plastics, Inc. Quality Management System is registered and conforms to ISO 9001:2015