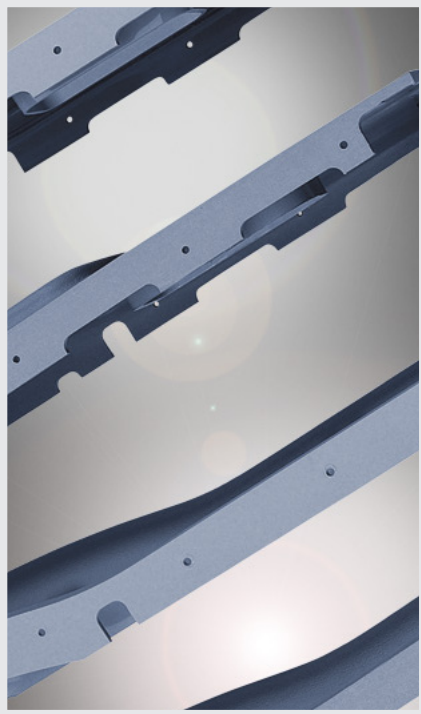




FORMED SOLUTIONS, INC.

DESIGN GUIDE



OVERVIEW

DRAW RATIO

RADII AND CHAMFERS

UNDERCUTS

DRAFT

TEXTURES

RIBS & LOUVERS

FASTENING

DIMENSIONAL TOLERANCES

DIMENSIONING

THERMOFORMING PROCESSES

VACUUM FORMING:

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.

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

PRESSURE FORMING

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. vacuation of the air between the sheet and the mold is required.

Applications: High appearance covers, computer housing, 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.

TWIN-SHEET PRESSURE FORMING

The process of injecting compressed air (20-120 p.s.i.) between two hot sheets, thus forcing it to conform to the contour of each of two molds mounted opposed to each other. Evacuation of the air between the sheet and the mold is required.

Applications: Those enclosures that require structural rigidity. Appearance items that require high surface detail and yet light weight. Competes with blow molding and rotational molding. A low to medium volume process.

ABOUT THE SHEET

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. Costs are increased because of the extra energy required to heat the polymer twice, and the fact the extruders are commonly custom processors that also need to generate profits to continue to exist. On the other hand, gauge changes, color changes, even material (i.e., polymer) changes can be done very easily in thermoforming because 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, temperature of processing, crystallinity of the resin, or, in general, the degree of difficulty of extruding the sheet. The most common resins (and therefore the most cost effective ones to specify) are FRABS, ABS, DKE or Kydex, HIPS, HDPE or HMWHDPE.

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:

Draw Ratio = the Surface Area of the part / Footprint of the part

Example #1:

Assume a part is 10"x 12"x 2" deep.

Therefore the Draw Ratio will be:

$$\begin{aligned}\text{Surface Area} &= 2(10'' \times 2'') + 2(12'' \times 2'') + 10'' \times 12'' \\ &= 40'' + 48'' + 120'' \\ &= 208''\end{aligned}$$

$$\begin{aligned}\text{Footprint} &= 10'' \times 12'' \\ &= 120''\end{aligned}$$

$$\begin{aligned}\text{Draw Ratio} &= 208'' / 120'' \\ &= 1.7\end{aligned}$$

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:

Draw Ratio x Desired Finished Gauge = Minimum Starting Gauge

$$1.7 \times 0.100'' = .170'' \text{ Assuming perfect material distribution.}$$

Example #2:

Assume a part size of 10" x 11" x 5" deep.

$$\begin{aligned}\text{Surface Area} &= 2(10'' \times 5'') + 2(11'' \times 5'') + \\ &(10'' \times 11'') = 300''\end{aligned}$$

$$\text{Footprint} = 10'' \times 11'' = 110''$$

$$\text{Draw Ratio} = 2.73$$

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 (i.e., 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 ball park"

when calculating the necessary starting gauge. Many part quotations will have two gauges specified (a high and 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.

RADII AND CHAMFERS

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 proper size radius or chamfer. These features are typically needed to allow for part strength, retention of material thickness, and/or esthetics.

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 over 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 check to see 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, causing either a heavier starting gauge to be used or an adhesive filler applied to the inside of the part backing up the corner and adding 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 a 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:

Depth of Part Radius

0" - 3"	.015" - .125"
3" - 6"	.125" - .250"
6" - 12"	.250" - ?

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 questions.

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 not nearly the amount it would be if it were an injection molding or structural foam molding tool.

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 on 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. This is not normally a problem on an undercut, which is an inward facing flange, since the parting line can be hidden at the point at which the part turns in. 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.

DRAFT

GUIDELINES:

For parts that are formed into a female mold with a texture allow one degree of draft for every thousandth of an inch of texture depth. With parts formed over a male mold allow three degrees of draft as a minimum.

OVERVIEW:

The need for draft 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. 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 high probability that the part will not release from the mold or will have severe scuffing from any texture that is on the mold. By designing in draft 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.

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 draft. In thermoforming the advantage of a one sided molding process becomes apparent with 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 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 back to the discussion of draw ratio. 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 the sheet 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.

FASTENING

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:

Molded in inserts allow for one of the most cost effective ways of installing fasteners because the molding process does all the work. The only added cost is for the insert; the part will take just as long to form whether there is an insert in it or not. The sheet is the “adhesive” that secures the fastener. There is no need for glue or sonic welding of the insert.

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, the labor to machine the block to the correct height and add

a hole for the fastener or insert, all are significant. Add to this the greater probability of a lower yield rate due to the number of additional operations. The simplicity of a molded in insert has real advantages.

There are a number of 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.

DIMENSIONAL TOLERANCES

GUIDELINES:

For molded in dimensions of parts from a machined aluminum mold use $\pm .015$ " for the first inch adding an additional $\pm .001$ " for each subsequent inch. For molded in dimensions of parts from a cast aluminum mold use $\pm .030$ " for the first 16" adding an additional $\pm .001$ " for each subsequent inch. Trimmed dimensions, regardless of the mold but using CNC trimming equipment, should have a general tolerance of $\pm .015$ ". For hole diameters use $\pm .005$ ".

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, method of trimming and quality of trimming fixtures.

SPECIFICS:

Most thermoforming resins have a coefficient of thermal expansion in the range of $.000060$ " to $.000120$ " per °F per inch [$6.0 - 12.0 \times 10^{-5}$ 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 summer to winter.

Extrusion quality will affect the part many ways. The control of the extruder from run to run is very important. Changes in extrusion speed, direction, temperature, and 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. 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 $\pm .030$ " for the first 16", adding an additional $\pm .001$ " for each subsequent inch.

The forming process must be very consistent from run to run to insure 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. CNC trimming allows for tighter tolerances and consistent parts. A typical CNC trimming machine controls the movements to $\pm .0002$ "; the trimming machine itself is designed for $\pm .005$ ". For cost effectiveness we use holding fixtures that allow a tolerance of $\pm .015$ ". Hole sizes and hole patterns can be held to a tolerance of $\pm .005$ ". To try and hold tighter tolerances would require fixturing approaching that of the original mold cost.

DIMENSIONING

GUIDELINES:

All dimensions on the print should be generated from the mold side of the part. When possible the starting gauge of the sheet should be specified.

OVERVIEW:

One of the most common mistakes in thermoforming is that the dimensions are generated from the non-mold surface of the part. Once the mold surface has been determined there should be no confusion on dimensioning. All of the dimensions, formed in and trimmed in, must be referenced to the mold side of the part.
