

## Vacuum Thermoforming Process & Design Guidelines

### 1) Materials:

Generally speaking amorphous materials like polystyrene, ABS, polycarbonate, PVC, and PVC/Acrylic blends are easier to vacuum form.

PTI will default to ABS unless specified differently.

### 2) Process Sequence:

- 1) Sheet is heated to thermoforming temperature.
- 2) A male or female mold and the hot sheet are moved into contact and a vacuum is then used to evacuate the air trapped between the sheet and the mold.
- 3) Part cools below its Heat Distortion Temperature (HDT) and is removed from tooling.
- 4) Part is trimmed and finished as required.

### 3) Uniform Material Distribution:

All thermoforming techniques are stretching processes. The stretching of a flat sheet of plastic material into a larger shape results in a larger surface area and a corresponding reduction in the sheet's original thickness. The thinning down of the sheet's original thickness is not necessarily uniform. The cardinal rule for the design of all formed plastic parts is to try and maintain uniform material distribution.

Mold depth and part geometry will vary wall thickness from part design. Processing techniques play a big part in material distribution.

### 4) 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.

## 5) Dimensional Tolerances:

Prototype thermoforming general tolerances are based off of our FDM 3D Printer.

- FDM General Accuracy tend to be .003" to .015" greater than aluminum production grade tools. All tolerances are geometry dependent.

Aluminum Tooling: For molded in dimensions of parts from a machined aluminum male mold use +/- .010" for the first inch adding an additional +/- .001" for each subsequent inch. For female tools use +/- .015 + .0015 due to process control loss of the material shrinking away from the tool surface during cool.

Higher tolerances can be quoted with CNC trimming.

Please use this tolerance planning when designing your components:

- Dimensioning scheme:  
Typically it is not customary to specify dimensions to the non-tool side of the part. This is due to the need to increase tolerancing significantly when including wall thickness variation. In such cases when tight tolerance is needed on nontool side of parts, second operation machining may be considered.
- Formed features:  
Male mold feature +/- .010" + .001 per inch (.3mm + .001mm per 1mm)  
Female mold feature +/- .015" + .0015 per inch (.4mm + .0015mm per 1mm)
- Trimmed holes:  
Hole Dia. 1" and less +/- .010 (.3mm)  
Hole Dia. 1" to 5" +/- .015 (.4mm)  
Hole Dia. 5" + +/- .020 (.6mm)
- Trimmed hole to hole:  
1" and less +/- .010 (.3mm)  
1" to 5" +/- .015 (.4mm)  
5" + +/- .020 (.6mm)
- Trim feature to molded feature:  
1" to 5" +/- .020 (.6mm)  
5" + +/- .030 (.8mm)
- Trim feature to trim feature:  
1" to 5" +/- .015 (.4mm)

## **6) Drawing Dimensioning:**

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.

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.

## **7) Radii & Chamfers:**

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" - ?

## **8) Draft:**

Draft is the degree of taper of a vertical sidewall to facilitate removal of the part from the tool. Always provide the maximum draft possible for any given vertical feature, as this can help minimize de-molding problems and may help reduce part cost. 1.5-2° of draft is a minimum on a female feature, and a deep texture will require more. For male features, minimums are in the 4-6° range. The deeper the feature or rougher the surface texture, the more draft is required.

## **9) Under Cuts:**

Undercuts are features protruding from or into the tool surface, which would prevent removal of the part from the tool. Such features are possible with the use of movable "cores-pulls" in the tool, which are often automatic. These cores are used to form the feature, then are retracted to allow the part to be removed. Under cuts are generally not to exceed 5/8" in depth.

## **10) Draw Ratio:**

The draw ratio expresses the relationship between the beginning surface area of the unformed sheet which covers the opening of a feature, and the ending surface area of the interior of the feature once formed. A 3 to 1 ratio is generally a maximum draw ratio. Avoid multiple tall features too close to each other. Generally, the distance between two features, like ribs or cooling vents, needs to be no less than 2x the material thickness. A caution about corners: Avoid intersections of walls that are  $>90^\circ$ , as it can be very difficult to get material to flow into that feature.

## **11) Reference Points:**

Reference points should be designed into a part, which allows a measurement from a controlled (molded) surface or point to a critical feature of the part, such as drilled hole-centers, cutouts, or other features.

## **12) Ribs, Bosses & Hardware:**

Ribs may be formed into the part according to the parameters above. Ribs are used to support a flat surface. Ribs or other reinforcement may also be machined independently and glued in at added cost. Internal bosses may also be independently machined and glued in.

All manner of hardware can be attached in secondary operations. And oftentimes hardware can be "insert-molded" into the part. Other parts, to include bosses can be heat staked or adhered to a thermo formed body by bonding.

## **13) Texture:**

Texture may be incorporated into a given part either by using a textured raw material or by texturing the face of the tool.

Texturing is of course an added expense and requires increased draft. Parts can also be painted to add texture and color.

## **14) Joining Lines:**

Two parts may be joined together such as a front and back case. The preferred joint is a lap joint, which would form an under cut in one of the parts adding expense to the tool. The joint should also require a witness line of about .060" to accommodate tolerance variation of the joint.