Blow Molding Design Guideline

- Draft of Part
- Blow Ratios
- Radii / Corners
- Ribs / Tack-offs
- Shrinkage
- Insert Molding
- Texture
- Pinch Design
DRAFT OF PART

PARTING LINE | This is determined by the shape of the part and will optimize blow ratios

DIRECTION OF PULL | Determining this will eliminate undercuts (die-lock) or require inserts

DRAFT GUIDELINES | A minimum of 1° per side (2° per side is recommended)

- If a Texture is used, add 1° per .001" (0.0254 mm) of texture depth
- Increase draft on outside surfaces as blow ratios increase
- For Ribs & Protrusions, 10-30° may be necessary depending on size of feature

BLOW RATIOS

Determining Blow Ratios will help determine localized thinning (areas of the part that might get thin during the blow molding process).

CALCULATING THE BLOW RATIO

Blow Ratio = \( \frac{H}{W} = \frac{\text{max depth of cavity}}{\text{smallest dimension of cavity opening}} \)

Axisymmetric Female Mold Cavity: \( \frac{H}{W} \leq \frac{1}{3} \)
Long Female Mold Channel: \( \frac{H}{W} \leq 1 \)
Male Pin In Mold: \( \frac{H}{W} \leq 2 \)
Long Male Mold Projections: \( \frac{H}{W} \leq 2.5 \)
**Radii / Corners**

- Material will “freeze” when it comes into contact with the mold, causing it to stretch into corners.
- If parison is trapped (pinched off) at parting line, it must stretch into the cavity (pre-blowing the parison prior to mold closure helps minimize stretch).
- To maintain uniform wall thickness, corners and edges must be sufficiently rounded off to minimize stretch.
- Corner and edge radii should increase as blow ratio increases.
- In many cases, a *chamfer* is a better option than a large radius as it decreases stretch and provides a clean appearance.

**Ribs / Tack-Offs**

- Ribs and tack-offs are often used to provide structure to double-wall panels.
- Tack-offs are compression molded features, where both sides of the parison are welded together:
  - May be one-sided to maintain smooth finish on cosmetic side or double-sided to meet in center of cross-section.
  - Will tend to *blush* through, which can be reduced with an aggressive texture (*blushing* is a variation in the surface of a part’s appearance due to uneven cooling, moisture, or material flow).
  - Compression thickness should be designed at 1.5 - 2.0 times the parison thickness.
- Flanges are also compression molded features, oriented along the mold parting line:
  - Direction of pull must be considered.
  - Holes or slots can be molded into the flanges for mounting or alignment purposes.
**SHRINKAGE**

- *Each resin has its own inherent shrinkage rate* – refer to material data sheets for specific details
- Part shrinkage can be affected by:
  - Shape & Quantity of Local Features
  - Tack-offs
  - Pockets
  - Cores
  - Cooling Time
  - Wall Thickness
- Outside walls will shrink away from the mold cavity, requiring less draft
- Material will shrink and lock onto male mold cores, requiring more draft
- HDPE will continue to shrink for hours or days
- *Shrinkage* is generally treated uniformly, however longitudinal (parallel to parting line) shrinkage is typically greater than transverse (perpendicular to parting line) shrinkage
- Flat sides tend to warp during shrinkage due to build up of internal thermal stresses
- *Warpage* gets more exaggerated after an opening is cut

**Shrinkage / Warpage Diagram**

“Doming” or “crowning” of large flat surfaces can help control the shrinkage direction

**INSERT MOLDING**

- Insert molding is an effective way to incorporate functional features that are difficult or impossible to form in the blow molding process
- *Plastic components* (such as an injection molded bracket) and *metal components* (such as a threaded stud or nut) require a mechanical or retention feature to provide necessary pull-out and/or torque strength
• Retention strength is best when the insert molded component is positioned as far away from the parting line surface as possible - this allows the parison to blow over the insert and enables material to flow around it
• Insert must be located in area where sufficient parison thickness is available for encapsulation
  ○ Away from edges and corners
  ○ Away from deep pockets

TEXTURE
• Texture may be used to enhance cosmetic surface appearance
• Texture selection is different than for injection molding due to lower pressure process
  ○ Fine textures may not “print back”
  ○ Self-venting pattern desired to eliminate air entrapment
• Typical texture depth is in .006” - .010” (0.1524 mm - 0.254 mm) range
• Additional draft is required locally in areas to reduce scuffing of texture during part removal

PINCH DESIGN
• Particularly important for parts that are not contained, or can't be formed completely inside the mold
• In all parts, there will be some flash produced outside the mold cavity
• The flash is pinched off in the mold to allow separation from the molded part, resulting in a visible parting line
• A cosmetic pinch bead can be designed to hide the parting line
• A structural pinch bead can be used to strengthen the parting line, which is often a weak spot in the part
• In multi-layer fuel tanks, a structure pinch bead is used for both structural integrity, as well as fuel permeation reduction, as the gap in the barrier layer is reduced

For more information about how Gemini Group's innovative approach to the blow molding process can help optimize your part design, or to partner on a new product, please contact us at:

53 West Maple Road  (248) 435-7271
Clawson, MI  48017  plasticsales@geminigroup.net