For Engineers, By Engineers.

BLOW MOLDING DESIGN GUIDE

geminigroup.net/Regency

GEMINI GROUP

How to Use This Guide

If you've been tasked to design a hollow part for high-volume production (3,000+ parts per year), you've come to the right place. Blow molding allows for complex hollow parts with tight tolerances to be produced at a lower cost and with faster cycle times than other molding methods such as rotomolding. It's no wonder so many engineers are turning to blow molding to solve their high-volume production needs.

Design is the key driver of the cost and quality of blow molded parts. It has the power to give your company a competitive advantage or it can bring your project to a screeching halt. But where do you begin? You may have questions such as:

- Will blow molding work for my part?
- What does the blow molding process look like?
- What special design considerations do I need to make?

We were there once, too. That's why we created this guide. It provides the basic guidelines for designing blow molded plastic parts – from the perspective of an experienced molder. It is not exhaustive; the amount of information available is simply too great. However, it serves as a starting point to better understand what goes into the blow mold design process.

If you're new to blow molding, start with the **Understanding Blow Molding** section. It outlines the benefits of blow molding, provides an overview of its applications, and illustrates the blow molding process.

If you have an understanding of the blow molding process and just need to learn how to optimize your design, jump to **Blow Molding Design** to learn design principles for the most critical aspects of blow molding.

Questions?

We've found the most important factor for success in blow molding is partnering with our customers early in the design stage. So don't hesitate to reach out to us if you have questions regarding what you find on the following pages or how it relates to your specific application.

Our technical sales team is standing by at **plasticsales@geminigroup.net** and **248-435-7271**.

Enjoy!

Contents

UNDERSTANDING BLOW MOLDING

BLOW MOLDING DESIGN

SUMMARY

Understanding Blow Molding

BLOW MOLDING PROCESS

Blow molding is an umbrella term for forming hollow plastic parts by inflating a molten plastic tube, or parison, until it fills a mold and forms the desired shape. Think of it like inflating a balloon inside of a water bottle. There are three main types of blow molding:

Extrusion Blow Molding

Extrusion blow molding is the most common type of blow molding and is used to manufacture complex parts in large quantities.

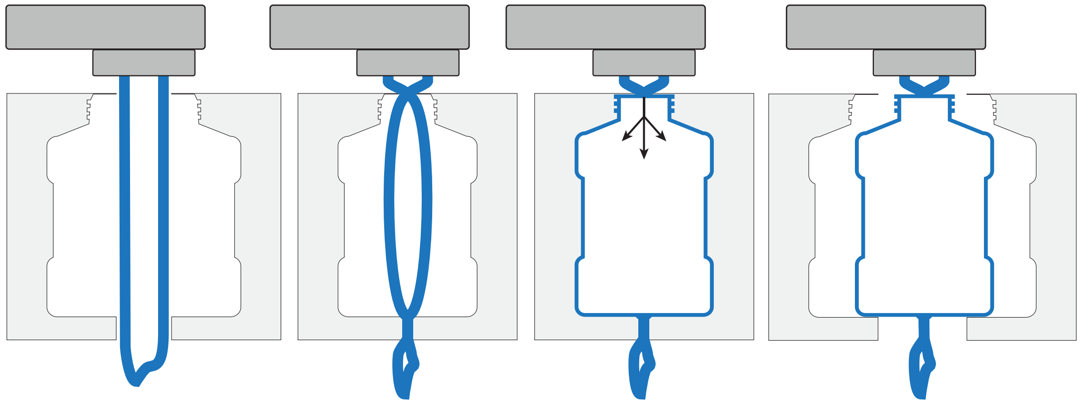
Stretch Blow Molding

Stretch blow molding is typically used to create plastic containers with simple geometries such as jars and bottles.

Injection Blow Molding

Injection blow molding is the least commonly used method and is used to manufacture small containers in small quantities.

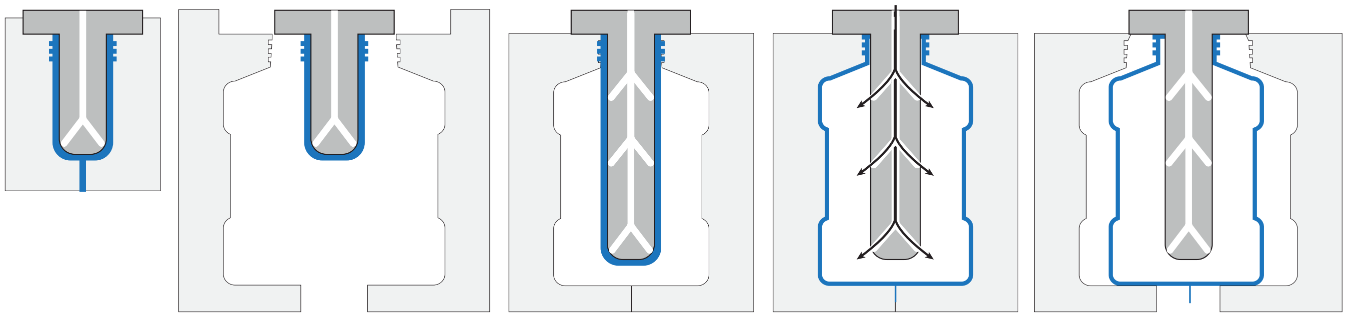
Extrusion Blow Molding



In extrusion blow molding, the parison is extruded vertically between the two open mold halves. The mold closes and the parison is inflated until it takes the shape of the mold cavity.

Understanding Blow Molding (continued)

Stretch Blow Molding



STEP 1
Parison is injection molded

STEP 2
Parison is placed into blow mold

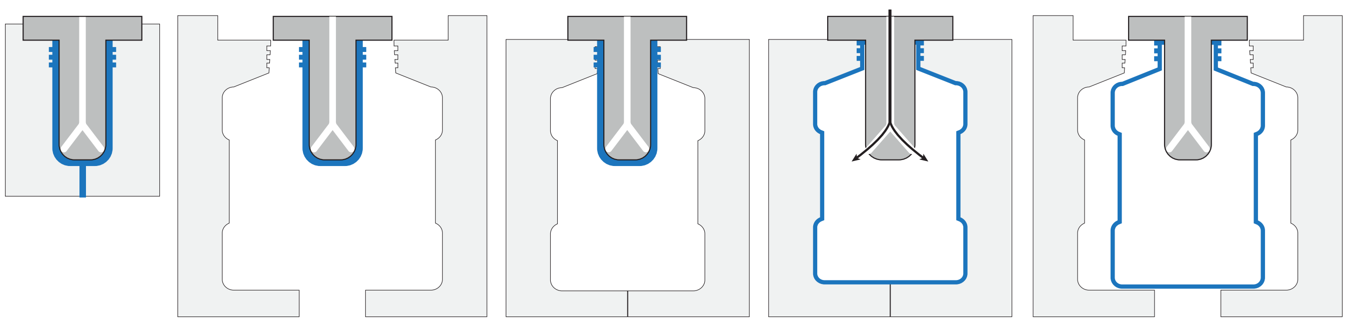
STEP 3
Mold seals around parison, parison is stretched downward

STEP 4
Parison is inflated with air

STEP 5
Parison cools and mold opens

In stretch blow molding, the parison is formed by injecting plastic around a core. When the injection mold opens, both the parison and core are transferred to the blow mold and securely clamped. The parison is then heated and stretched downward by the core. Finally, the air vents in the core open and inflate the parison into the desired shape.

Injection Blow Molding



STEP 1
Parison is injection molded

STEP 2
Parison is placed into blow mold

STEP 3
Blow mold seals around parison

STEP 4
Parison is inflated with air

STEP 5
Parison cools and mold opens

In injection blow molding, the parison is formed by injecting plastic around a core. When the injection mold opens, both the parison and core are transferred to the blow mold and securely clamped. Finally, the air vents in the core open and inflate the parison into the desired shape.

Understanding Blow Molding (continued)

Advantages of Blow Molding

With an increasing focus on lightweighting and sustainability, engineers across the globe in nearly every industry have turned an eye towards blow molding. Its many benefits include…

- **Low cost for high volume production**
  For mass-produced hollow parts, blow molding is more cost-efficient than other processes such as rotomolding and thermoforming.
- **Design flexibility**
  Complex part geometries can be achieved to adapt to unusual spaces between already existing components.
- **High strength-to-weight ratio**
  Parts can be reinforced with a lightweight foam core or internal ribs to increase structural strength and impact resistance.
- **Insulation and acoustical properties**
  Hollows can be vacuumed or filled with a material to achieve high insulation and noise reduction properties.

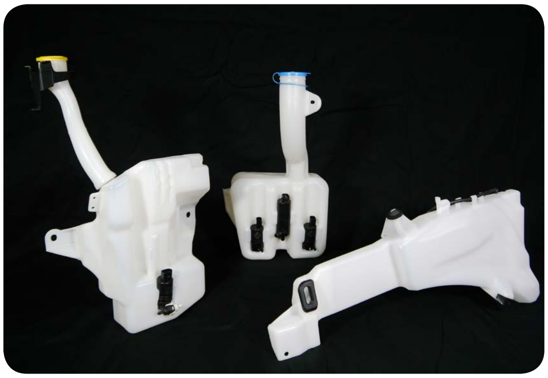
Technical Blow-Molded Products

Typically, when people think "blow molding," they think of water bottles, cosmetic containers, toys, and other consumer goods. It might come as a surprise to learn that blow molding is used to produce some highly technical products, too, including the following:

Under Hood Air Ducts

Intake ducts, resonators, tuners, HVAC ducts

Fluid Containers



DEF tanks, washer reservoirs, surge tanks

Structural Components

Seating, consoles, panels, foam-filled parts

Functional Components

Tubes, boots, sleeves, housings

Blow Molding Design

Blow Molding Design

Now that you've read a high-level overview of the blow molding process, it's time to start your design. It might seem a bit daunting at first, but you don't have to get it right the first time around. Just take your best shot, then present your design to your supplier. Companies such as Regency Plastics, a Gemini Group company, will work with you to ensure that your design is optimized for your application and for the blow molding process. Regency will even run your design through simulation software to eliminate problem areas before any tooling is built. To accelerate the process, just follow these general guidelines and your design should be in pretty good shape once it hits your supplier's hands.

Material Selection

When choosing a material for blow molding, it's important to remember the material will be stretched. If the elastomeric properties of the material are exceeded, a hole will tear through the material resulting in a defective part. Other relevant properties are largely based on your application and might include mechanical, physical, chemical resistance, heat, electrical, flammability or UV resistance.

Here are some questions you should consider:

- **Temperature exposure:** What are the minimum and maximum temperatures the part will be exposed to? Remember, the first shot substrate must be able to withstand temperatures above the second shot's melt point.
- **Bending stiffness:** Does the material need to be flexible or rigid? What should the bending strength or deflection be?
- **Impact resistance:** What level of hardness or surface toughness does each component require? Will the part need to withstand impact forces without shattering or breaking?
- **Tensile and compression strength:** Will either component need to withstand pull-apart or compression forces?
- **Interfacing components and assembly:** Does the part stand alone or is it part of an assembly? How will the part be joined to the interfacing parts (e.g., welding, adhesive, mechanically-joined, fasteners, etc.)?
- **Appearance:** Does either component need to have a particular color, grain, texture and/or gloss?
- **UV resistance:** Will the part be used in outdoor settings or otherwise exposed to UV rays?
- **Chemical exposure:** What chemicals –– e.g., gasoline, diesel fuel, antifreeze, etc. –– might this part be exposed to that could compromise function?
- **Aging:** What shelf life and product life is required of the part?

If a material meets most requirements but falls short somewhere, an additive may be able to change its properties. For example, nylon 6/6 flows well, is good for thin-walled parts, and has excellent impact resistance, but you might have rejected it because of its average strength and lack of resistance to heat. Adding glass-fiber filler to the resin not only makes the nylon much stronger, but far more heat resistant. Glass also reduces the chance of sink in thick sections, but may lead to warp in thin areas, depending on material flow during the molding process.

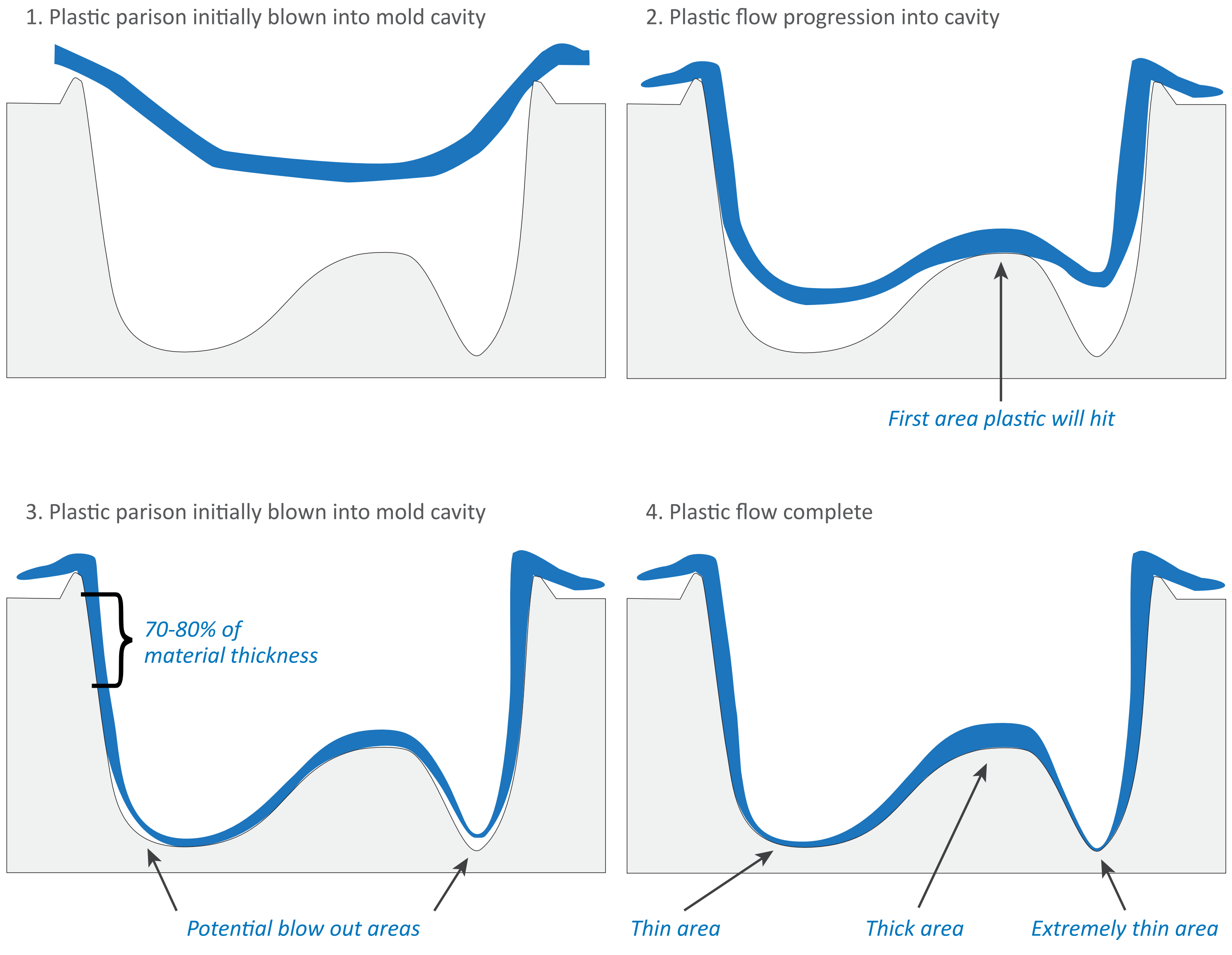
There are hundreds of materials and thousands of ways to adjust, blend or fine-tune them to produce the desired results. Resin manufacturers, compounders and independent resin search engines have material data online. It is highly recommended that you consult with your supplier when choosing a material. They can either advise you directly or put you in touch with an expert at one of their material suppliers.

Blow Molding Design (continued)

BLOW RATIO

After the material has been selected, the part design needs to be examined to determine if the blow ratio, or the ratio of depth-to-width in the mold cavities, is acceptable. A poor blow ratio can cause the inability to maintain minimum wall thickness, thin spots in the corners or deep draw areas, unnecessary weight increase, part shrink, and warpage. ***This is why blow ratio is one of the most important design constraints in blow molding.***

Blow Ratio Progression



Blow Molding Design (continued)

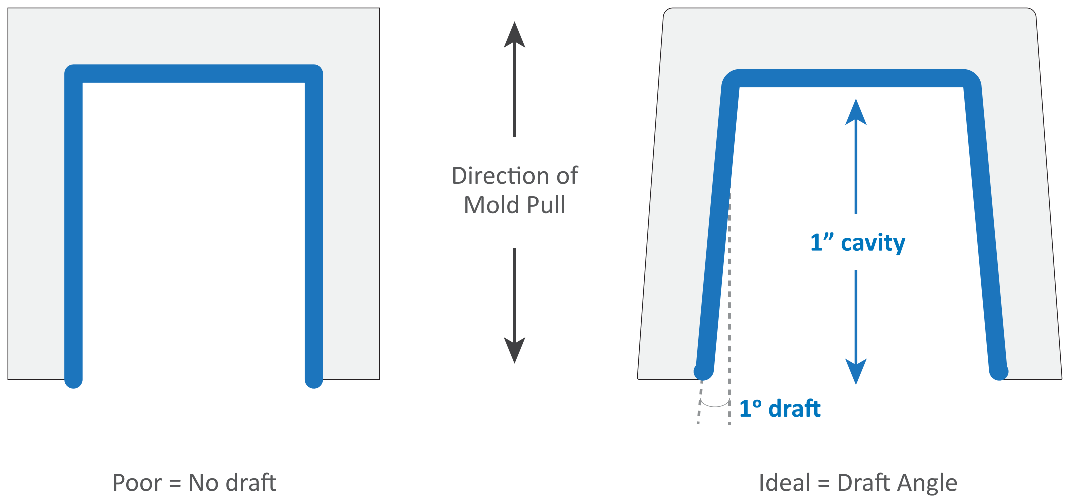
BLOW RATIO (CONTINUED)

Determining the blow ratio will help determine localized thinning (areas of the part that might get thin during the blow molding process). Keep in mind, when a part design has multiple blow ratio conditions, each individual draw area can affect the other. Once you've determined the blow ratio, adjustments can be made to the thickness of the parison as it is being extruded, enabling it to have different thicknesses at specific points along the tube to compensate.

Calculating the blow ratio on your own is a daunting task. There are dozens of formulas online, each suggesting a different way of making the calculation for a different scenario. Even if you find a formula that looks applicable, variables such as material and part requirements may render your findings not applicable. This is why we highly encourage you to involve your supplier from the start. Some suppliers, such as Regency Plastics, have **software that simulates the blow molding process** so problem areas can be tackled before any tooling is built.

DRAFT OF PART

Utilizing draft, or taper, is critical to the moldability of your part. Without it, parts can have problems ejecting from the mold, damaging both the part and the mold. No single draft angle can be applied to all part designs, but a good rule of thumb is to allow for as much draft as possible, with at least one degree of draft per inch of cavity depth.



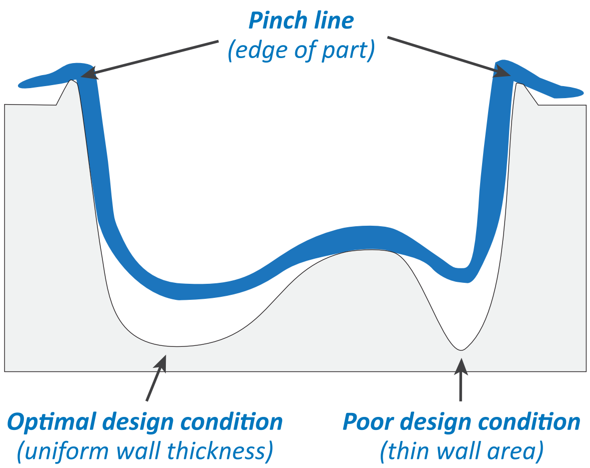
Draft guidelines - a minimum of 1° per side (2° per side is recommended)

- 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 and protrusions, 10° to 30° may be necessary depending on size of feature

Blow Molding Design (continued)

RADII / CORNERS

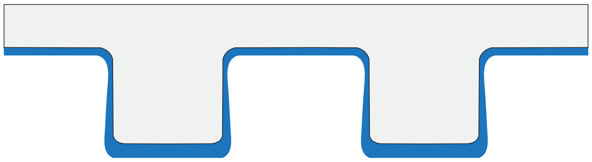
The parison begins to cool and harden as soon as it hits the mold wall. The farther it stretches to fill the cavity, the thinner it becomes. To prevent weak spots caused by such exponential thinning, corners and edges must be sufficiently rounded off to minimize stretch.



- Corner and edge radii should increase as blow ratio increases
- In many cases, a chamfer (a symmetrical sloping edge) is a better option than a large radius as it decreases stretch and provides a clean appearance

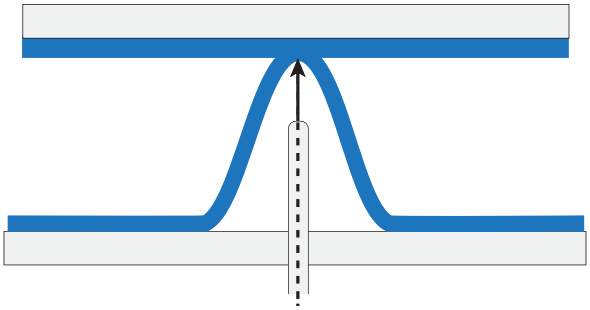
RIBS / TACK-OFFS

Ribs and tack-offs are often used to provide structure and strength to load-bearing parts and double-wall panels. Flanges are most commonly used to facilitate attachment to another object.



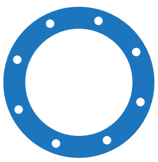
Ribs

- Be conscious of the orientation of the rib to the bending load or there will be no increase in stiffness
- Follow general draft guidelines when incorporating ribs



Tack-Offs

- 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

- Direction of pull must be considered
- Holes or slots can be molded into the flanges for mounting or alignment purposes

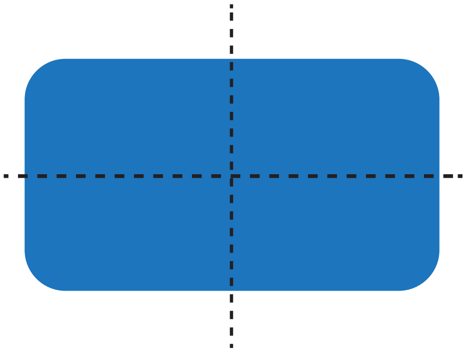
Blow Molding Design (continued)

SHRINKAGE

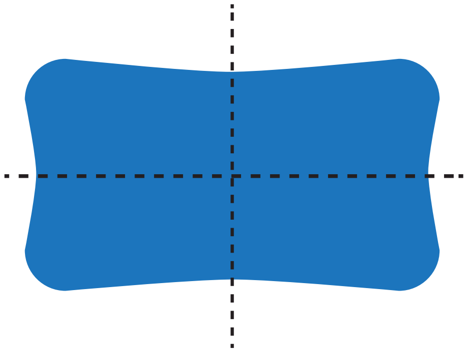
Every plastic resin has a shrink rate. This means your part and part features will inevitably shrink as they cool. Understanding how design impacts the degree of shrinkage will help you avoid unacceptable shrinkage and other deformations. Part shrinkage can be affected by:

- Shape & quantity of local features
- Cores
- Tack-offs
- Cooling time
- Pockets
- Wall thickness

Controlling Shrinkage

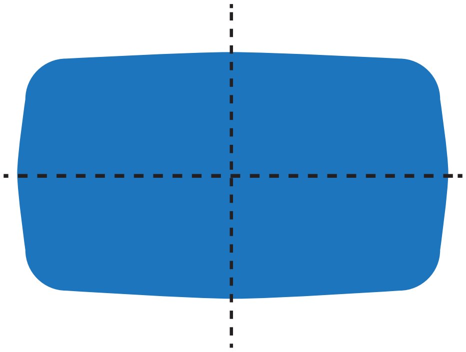


Desired shape



Resulting shape

"Doming" or "crowning" of large flat surfaces can help control the shrinkage direction:



Slight bow in cavity

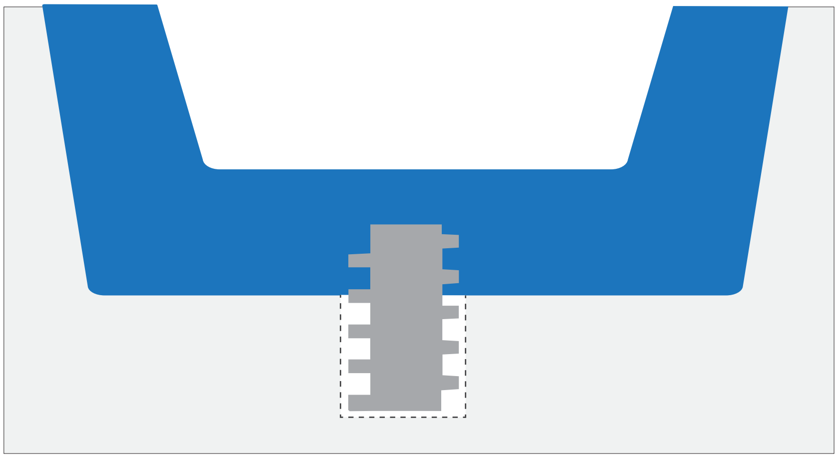
Things to keep in mind:

- 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 buildup of internal thermal stresses
- Warpage gets more exaggerated after an opening is cut

Blow Molding Design (continued)

INSERT MOLDING

The use of insert molding is used to create functional features that would be difficult or impossible to form in the blow molding process. Incorporating inserts directly into the mold eliminates the need for secondary joining operations and achieves a stronger bond between the insert and blow molded components.



- 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

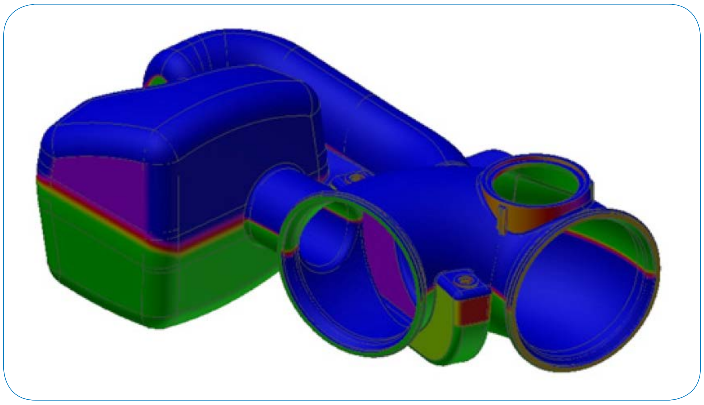
Adding textures, lettering, and other markings in-mold can be costly. It's best to consult with your supplier to determine the most efficient way to incorporate these elements. If you're sure that you need in-mold textures or lettering, you can get started by following these basic guidelines:

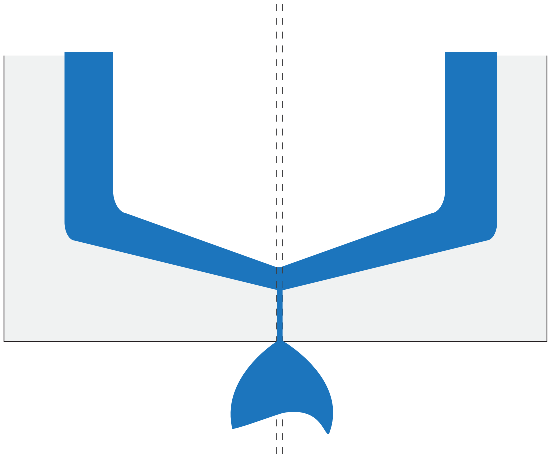
- Use a san-serif font such as Arial or Calibri, at least 1.5mm or larger
- Texture may be used to enhance cosmetic surface appearance
- Texture selection is different than for injection molding due to lower pressure process:
    - Fine textures may not "print back"
    - Self-venting pattern desired to eliminate air entrapment
- Typical texture depth is in .006" to .010" (0.1524 mm to 0.254 mm) range
- Additional draft is required locally in areas to reduce scuffing of texture during part removal

Blow Molding Design (continued)

PINCH DESIGN

In all blow molded parts, there will be some flash produced outside of the mold cavity. The flash is "pinched off" in the mold to allow separation from the molded part. The pinch design can be enhanced in quite a few ways to accommodate the functional and aesthetic requirements of a part:

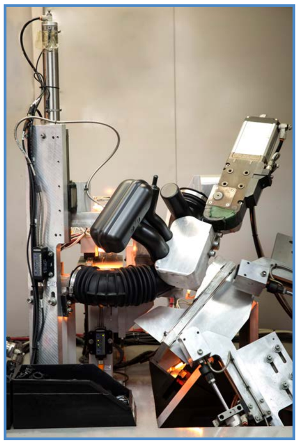




- A cosmetic pinch bead can be designed to hide the parting line
- A structural pinch bead can be used to strengthen the parting line, which is often a weak spot in the part
- In multi-layer fuel tanks, a structure pinch bead is used for both structural integrity, as well as fuel permeation reduction, as the gap in the barrier layer is reduced
- If protruding product features such as tabs or hose connectors are needed, it's best if they are positioned along the pinch seam
- Special design considerations must be made for parts that aren't fully contained or can't be completely formed inside the mold

SECONDARY OPERATIONS

Every blow molded product will require some degree of additional processing after it leaves the blow molding machine. These secondary processes can be used to remove unwanted materials, add features, and improve functionality. Whether routine flash removal or something more advanced such as welding and assembly, designers should consider which secondary processes will be critical in finishing their blow molded product.



- Flash removal
- Plastic welding
- Riveting and staking
- Foam pad assembly
- Acoustic pad assembly
- Blow mold assembly

Blow Molding Terminology

BLOW MOLDING TERMINOLOGY

- **Blow pin:** part of the head tooling used to form the parison, usually a hollow tube through which air is blown into the mold
- **Blow ratio:** the ratio of the mold cavity diameter to the diameter of the parison or hollow form to be blown up
- **Blowout:** when the blowing air ruptures the parison wall
- **Deflashing:** a finishing operation for removing flash on a plastic molding such as tails, handle plugs and material around thread areas
- **Draft:** a slight taper built into the mold walls to facilitate removal of the part from the mold
- **Flash:** any excess plastic discarded from the perimeter of the product after it is removed from the mold
- **Insert:** an integral part of a plastic molding consisting of plastic, metal or other materials that has been preformed and inserted into the blowing mold so it becomes an integral part of the finished item
- **No blow:** when the parison does not fully expand to fill the mold cavity
- **Parison:** a tube-like piece of plastic that is inflated to the desired part form
- **Pinch-off:** the sharp part of the perimeter of the mold that defines the part shape when it closes on the plastic parison and separates the part from the excess flash
- **Pre-blow:** an initial expansion of a parison before it is fully expanded to a product’s final size and shape
- **Rib:** a thin wall-like feature that increases the bending stiffness of wall sections and bosses
- **Shrinkage:** the contraction of a plastic molded part as it cools
- **Swell:** expansion of the parison as its exits the parison die
- **Vent:** a shallow channel or opening cut in the cavity to allow air or gases to escape from the mold at the parison is expanded to fill the cavity

Case Study

CASE STUDY: CONVERTING A ROTOMOLDED DEF TANK TO BLOW MOLDING




The Market

The customer is a Tier 1 supplier of custom fluid management solutions to heavy-duty vehicles, machines, and equipment for the construction, agriculture, transportation, and aerospace industries.

The Challenge

The customer was overburdened by sourcing six different rotomolding suppliers to meet their annual DEF tank production volumes. Aware of the inefficiencies and risks associated with this sourcing model, the customer sought to rationalize their supply chain to two or fewer suppliers. The problem: all six rotomolders were at capacity. The process they'd been using from the beginning couldn't scale to their high-volume production needs.

The customer had looked into blow molding as an alternative manufacturing method for high volumes but couldn't justify the higher tooling costs. That is, until they met with the technical blow molding experts at Regency Plastics, a Gemini Group company.

The Solution

Drawing from over three decades of blow molding design, engineering, and manufacturing experience, the team at Regency Plastics worked with the customer to enhance the DEF tank design to be compatible with the blow molding manufacturing process. Coupled with Regency's ability to design and build all tooling in-house, the customer now had a design in hand that would see tooling payback in one year or less – that is, the tooling would literally pay for itself in one year's time.

The Result

The customer now has a single **scalable** source for DEF tanks that meets all production and performance requirements at a **37% reduction in piece price.**

Cost Drivers	Rotomolding	Blow Molding
Tooling	$150,000 / yr	$200,000 / lifetime
Production Rate	2 parts / hr	70 parts / hr
Piece Price	$35 / part	$22 / part

A Word of Advice

A WORD OF ADVICE

Blow molding is easy to understand, but difficult to master – especially for highly technical products. We strongly recommend that you involve your supplier from the start. Tell them what you intend to accomplish. Let them help you succeed!

As a Tier 1 and 2 automotive supplier, Gemini Group is comfortable being responsible with the entire design project or assisting you as a design partner. We also offer what other suppliers cannot -- countless process alternatives.

Our plastic processing expertise includes:

- Technical blow molding
- Two-shot injection molding
- Profile extrusion
- Compression molding

Using the expertise of all our divisions along with rapid prototyping, in-house tooling, and material expertise enables us to evaluate multiple processes and material selection for your unique application.

When a major automotive Tier 1 supplier had process and supply chain inefficiencies, we helped them convert a key product from rotomolding to blow molding. This gave them a ***scalable*** solution moving forward and resulted in a ***37% decrease in piece price.***

With over three decades of technical blow molding experience and a lineup of machine sizes ranging from ½ lb to 35 lbs, Regency Plastics, a Gemini Group company, is the top tier choice for complex parts with precise dimensions and ambitious functional requirements. We are fully automated for quality and cost control, have our own internal CMM operations, and design and manufacture our tooling in house.

Let Gemini Group help you succeed in your next technical molding project. Give us a call today!

Gemini Group, Inc. Customer Technical Center
3250 University Drive, Ste. 110
Auburn Hills, MI 48326

plasticsales@geminigroup.net
248-435-7271