Blow moulding design guide

I'm not robot

reCAPTCHA

Continue

The Custom-Pak blow casting design guide provides you with basic design tools for making engineered blow molded pieces. This guide focuses on the extrusion kick casting process. There are no two designs, so mold and process need to be adjusted to optimize each design. Software products can help predict casting characteristics and our engineers are here to help make your product great. Our design assistance is confidential and free. A. Blow-Molding Process B. Materials C. Capture Parison D. External Cavite-Mold Design E. Interior Core-Mold Design F. AirSpace G. Creating Structure H. Finishing A. Blow Molding Process The first step involves mixing, melting and pressing plastic (extrusion) to form it into a tube called parison, which will be used to make a part. Mold is used to make the part of the shape you want. The form has two halves that are closed around the molten parison. The air is blown into the inside of the parison to expand the molten plastic against the mold surface. The shape is cooled to set the plastic into a new shape of the mold. The moulded plastic part is removed from the mold, separated from the excess Parisian material called the flash, and finished. (Most final stages can be completed in form, but some are related to secondary operations.) B. Material selection is one of the most important aspects of design and should include a serious study: the plastic properties of the resin material cost the handling properties of your finished part of the purpose Although there are thousands of plastic materials, most of them will not meet the needs of your product. Experience with blow casting class materials is essential, and we have hands-on casting experience using every kick molded raw materials Some of the least expensive materials are also the easiest to handle. Polyethylene (PE) and polypropylene (PP) are the most popular resin casting kick. PE is currently cheaper, but PP tends to be tougher, which sometimes makes up for the difference in cost. These materials are resistant to most chemicals. One difference is the temperature performance with PE running better at -75 to 160 degrees Fahrenheit and PP works well from -0 to 170 degrees Fahrenheit. These materials usually form parts consistent with the principles discussed in this design guide. Engineering resins Many engineering pitches can be blow molded. Some of the cuts include PPO, PC, PETG, ABS, TPE - you get the idea. These pitches require special attention before casting. Most of them require drying before processing, specially designed extruder screws and specific processing conditions. The design criteria in this guide are not apply to parts cast from some engineering resins. Please get the correct design information for your particular project directly from our engineering staff. C. Capture Parison In order to develop a blow molded product, you need to understand, understand, between molten plastic parison and mold. If you blow the bubble out of chewing gum you can understand the blow of casting. The plastic material is stretched like an elastic band and if it gets too thin it is torn. Since parison is extruded as a tube, it is easy to make a tube or bottle shaped part, not much stretching occurs. Two halves of the mold are open, the parison is inserted, the mold halves are close and the part is blown up. The split between the halves of mold is known as the parting line. There is often a knife-like edge on the parting line around a part of the mold known as pinch-off. If the shape of the piece to be molded is changed from tube into flat part of the panel style, the parison tube should be flattened to make the panel. When this happens, the circumference of the parison becomes a surface that should cover the width of the panel. So we try to have a large enough diameter parison that, as it aligns, it can be captured around the perimeter of the panel on a pinch-off. If the parison doesn't extend to all areas of the pinch-off, it should stretch the rest of the way. Soft plastic can stretch only for a short distance before it begins thinning. Like chewing gum, the first thin spot is weak and it gets thinner faster until it appears. If a plastic parison appears it is called a blowout and results in no part forming at all. As the complexity of the part progresses to double-wall shapes with side walls and internal contours, the parison should not only be captured at all points along the parting line, but it must also meet the material thickness needs for a variety of form molding conditions specific to each area of the piece. Many of the design criteria used for a tray with molded internal shapes would be the same for designing a complex industrial part. D. External surface (mould cavity) The design of the inner and outer wall parts are formed simultaneously and holistically, but the interior and exterior design are essentially independent, so we review them separately. As the design evolves, the designer needs to start thinking about the interaction of plastic and mold that will produce the part. The visual appearance of many products is formed in one half of the shape called cavity. Below are some of the shape features the designer will want to consider. Cavity - Cavity Blow Ratio - W'gt;D bottle is a typical example of the impact molded part formed using 2 cavities of mold halves. The round bottle has a shock factor that consists of a diameter of width and depth (2:1). The result is an excellent distribution of the material in a round bottle. But not all parts will be round. As designers start to push the limits to draw down into the cavity, how far they Go? The answer depends on the elasticity of the lengthening material and how thin the wall you are willing to accept. But usually the material won't stretch much further down in the (Depth-D) than the width of the material available to fit into the cavity (Width-W). Thus, try not to design the cavity part to be deeper than the width. Cavity - Core Blow Ratio - W'gt;2D Many industrial parts are formed using a combination of cavity elements and the main shape, where the nucleus forms internal forms. The kernel changes the impact ratio settings. The diameter of the cylindrical parison, which forms the double part of the wall, should allow sufficient material to enter the form to adequately form each half of the part. Half of the cylindrical parison is used to form the outer half (cavity) part, and the other half of the cylindrical parison forms the inner half (core) of the part. Since there is no flow of material along the walls of mold (stretching only), it follows that the depth of the cavity (D) should be no more than half the length or width of the cavity (W). The design of a part that uses cavity depths that exceed this connection will be subject to serious thinning or blowing. Since the relationship between diameter and radius, parts of the nucleus cavity must have common impact ratios of W'gt;2D. With several or separated cavities, each cavity must meet this requirement of the W'gt;2D. The design of some complex parts will require changes in the location of the parting line to remain within this relationship. These steps in the steaming line should include a gap to re-open and close the halves of the mold and be positioned so that they do not pear parison during mold closing. The angle of the parting line of 10 project or more is usually designed in the form of parting line steps. When a 10 project is not possible, options such as fishing mold in the machine, so saying goodbye lines form a positive project in relation to each other or moving sections of mold can be used. Side walls and project plastic sticks and parison and starts to harden as soon as it gets into shape. The material then stretches to fill the cavity as the blow progresses. There is no flow of material along the mold walls. There are three aspects of thinning to consider. Thinning caused by stretching leads to weakness. Any subtle, weak location is subject to further thinning, because it has become thin and weak. Thinning is rapidly progressing in these places. The obvious stiffness (or strength) of any area in part proportionally depends on the square of the thickness of the wall. Changing the thickness of the wall can lead to deformation. Thinning along the sides and corners is the reason that the parts should have the outer corners of the project. The exterior of the project is not crucial to removing the part from the cavities, as the plastic is compressed from the outer walls of the mold as it cools. The design is recommended when the outer walls should be textured. Plastic contouring sections of parison according to these critical areas can improve the condition, but not eliminate it. Because of this, the design of the cavity should functions that contribute to thinning. Designs that use sharp corners 90 will result in parts with very thin, weak angles. There are various angular configurations that improve or soften. The most common approach is to fish the sidewall and put a radius or chamfer-corner on the corner. Removing the part can be a problem with the back-draft sections. The rear areas can block the part in the form. If possible, the part with the back of the project on one side should have an equal positive project on the opposite side. Thus, a part with 15 back project on one side and 15 positive project on the other can be removed as part of a non-project. Otherwise, the forms may need to move sections to remove the rear design function. The compression and compression of the deformation deformation varies depending on the material, the rate of temperature change and the thickness of the material. For PP and PE materials, the thickness of the material is the best predictor. Thin parts of the wall can be reduced by only 1%, and thick parts - more than 10%. The 0.060 thickest part will shrink by about 1.65% as it cools and the 0.125 thick part will shrink about 1.85%. Waiting for shrinkage should be taken into account when setting the size of the shape. Designs that allow the walls to thin in part can cause parts to deform. Thin areas will shrink less before cooling than thick areas. Changing the rate of shrinkage and distances can lead to deformation of the part. Some wall thickness and shrinkage changes occur in each molded product because the cooling speed of the plastic will change. The skin material against the mold metal will cool down and take the set before the material is not really touching the mold metal. The result is a tendency for external deformation walls inwards and is offset by the tendency of the inner wall to deform outwards. Using structural ribs, welded seams between walls, arcs or steps can create a structure that helps reduce deformation. Cooling To control the size, surface appearance and warpage, it is important to have as much control over the cooling part as possible. Flow speed is a major factor in removing heat and cycle time. By creating a turbulent flow, you can improve heat recovery time and cycle. To control the warpage in many designs, it is important that mold cooling be targeted to provide extra heat extraction in the heavier part wall part part. The overall flow pattern also affects the quality of the part. The water heats up as it flows through the mold. The shape should contain multiple inputs and sockets in the oscillating cooling model. A cold surface shape can also cause problems in the reproduction of surface details such as texture. can target water lines near each important section of the shape to ensure the necessary size control and appearance. Venting When the mold closes the parison is captured on a pinch-off. A certain number is trapped between the outside of the parison and the shape of the cavity. When the air is blown up to expand the parison, the captured air is compressed by an expanding parison until the interior - the external equilibrium pressure is not reached. When this happens, the parison will not completely touch the wall of mold. The result is visible surface anomalies, loss of texture and engraving detail, the appearance of folds and drag lines, and longer cycles from poor mold cooling. The ventilation can be easily located on the edge of any insertion in the cavity. Slotted vents or porous metals can be purchased and fit almost anywhere. Some ventilation methods will produce visible markings on the finished part. Texture, insertion and other methods can be used to mask markings made in the ventilation area. E. Surface Interior (Mold Core) - The design of the inner surface of the two-wall blow of molded parts is usually formed by the mould of the nucleus. Since the core of the mold should fit inside the cavity, there should be no doubt it meets the same W'gt;2D overall size requirement as the cavity. Half of the parison that draping over the mold core is already beginning to establish, as air is injected into the parison. As in the cavity, the plastic begins to stretch to fit the shape. The flow is almost non-occurring. Unlike cavities, some different rules apply. Blow Ratio - W'gt;D As the mould halves close on the parison, the core presses on the parison and forces it into the cavity until the pinch-off is sealed around the perimeter of the piece. The highest point of the nucleus forms the deepest depression within the part. If the design of the two-wall part has a separation wall between the two compartments, this wall is formed by stretching the plastic into the groove in the nucleus shape. As the plastic starts stretching into the groove, it starts to thinn. If the groove is too deep, the plastic quickly reaches the point where it thins until the inner air blows through the wall on the outside of the part. No part will be formed. Because of this, there is one simple but absolute rule that regulates the design of ribs or divisions between compartments. The depth (D) of the groove between the main sections should not exceed the width (W) of the ribs of the W'gt;D. This rule also applies to other structural forms. For example, one tall, round pillar in the center of the tray should be 1 or more in diameter. If the design of the part requires a mold parting line that steps to different levels for the part to function properly, the core should have a positive project on these steps on the pinch-off to join the pinch-off on the tooling cavity element. Changing the pinch-off level can change the W-D ratio of nearby pockets or ribs. All levels in the unit must pass the W'gt;D requirement in each direction. Side and project When the mold closes, half of the parison is draped over the core form shape The interior part. As the plastic cools, it shrinks to the metal mass of the mold core. A positive project is needed on all sides of the kernel mold in order to remove the plastic part after it has shrunk. The more generous the project, the easier the part can be removed from the form. A part with a 5 positive project on all sides of the core can be removed using a pin pusher. Parts with a smaller project can also be removed using a pin pusher, but as the project on the core decreases, the risk of damage to the part during the release increases. If the main design requires a partition without a project or a reverse project, then, if possible, a positive project should be presented on the opposite side of the kernel. Snap-fits and small undercuts can be modified to allow for an outlier. With the main cavity form, the parison becomes fixed on two levels, at the top of the core and pinch-off. When the part is blown up, the fixed plastic walls are stretched (without flow) to meet the side of the core. A deep core with a small draught and sharp angle will produce a thin, weak part. The design, angular radii and chamfer-corners can help eliminate the thin walls of the compression and The Warpage Overall shrinking interior will match the shrinkage exterior. But the shrinkage of the internal shape is limited to the metal core used to form the shape. Minor shape size adjustments may be required to meet specific sizes. The design of the interior part should take into account the potential for warpage. The deformation will be caused by fluctuations in the thickness of the walls and the distribution of the material during cooling. Both W'gt;2D total size and W'gt;D localized draw ratios must be observed throughout the entire design part to prevent warpage cooling Often, the metal mass of the core is larger than the cavity and will require more cooling power. Focusing on water lines for optimal heat extraction can be crucial to the success of the part. Venting Any place where air can be captured between the parison and mold walls is a place for ventilation. Deep cores can trap large amounts of air and the speed of an explosion may require more ventilation power for trapped air to avoid. If in doubt, it is better to turn on the ventilation than to detect a problem at work. F. Air Space is a combination of closing the mold on the parison and expanding the air inside the parison that forms the part. The designer should leave enough space between the inside and outer of the surfaces to make adequate air blowing in each square inch of the piece. If the air passage inside the part is reduced or obstructed, the part is not formed. Distance between walls There are no clear rules regarding the amount of airspace needed between the inner and outer walls to form a part. Smaller and The sensor parts seem to require less airspace than larger and heavier parts. Sidewalls usually require a little more airspace than or upper surfaces. Rule thumb minimums for airspace in parts up to 1 sq m. will be 3/8 on the sidewalls and 3/16 on the upper or lower surfaces. Parts of more than 1 sq m. should form adequately with 5/8 airspace between the sides and 5/16 on the upper and lower surfaces. The more airspace you can afford, the better the part will be formed. It is also interesting to note that thin panels can become stronger by increasing the distance between walls. Webbing There is a molding phenomenon called webbing that can occur when mold closes. As parts of the core and cavity mold are close at the parison, the parison is quickly transformed from the shape of a tube or bag into a functional configuration. As the core pushes the parison into the cavity possible for the opposite sides of the parison to touch before the air is injected to form a part. When this happens, the plastic welds together inside the parison and when the parison is inflated, the welding resists separation. The result is either a part with a very thin, weak section around the welding net or if the nearby material tears when the parison is inflated, it blows out and no part is formed. Webbing is more pronounced in deep cavities with appropriate deep cores. However, some configurations can make a parison collapse or fold back on themselves to cause eardrums when the mold closes. It is a good idea to ask for a simulation test on the deep parts that can produce webbing. Squeezing many functional designs are greatly enhanced by incorporating compression of molded tabs, locks or mounting surfaces. Compression molded tabs can be added at any time along the mold parting line on the same plane as the pinch-off. To change the angle of the tab relative to the main parting plane, you need to create a line of parting the shape at the desired angle. This can be done with angular steps of the parting line or inserts around the perimeter of the part or moving inserts inside the part. When the impact of molded parts must be combined with other parts by mounting screws, bolts or rivets, an exceptionally strong mounting surface can be secured by squeezing the inner and outer walls together. The two walls can be compressed together at almost any angle or location as long as there is enough room around the compression for good airflow and as long as the mold halves can close without interference. By squeezing the casting of the inner and outer walls together, some of the stiffness and directness can be greatly improved. It's also a great way to provide styling strength when dealing with heavy G loads. Create a structure in the plastic part. The properly designed two-wall part will be significantly stronger than the ribbed single-wall part of equal weight and can easily surpass the metals in the Applications. There are several ways to add strength to blow up molded part designs. Welding cones and Tack-Offs By designing the mold to close in specific places at a distance that is smaller than the total thickness of the internal and external walls welding is formed. The amount of compression establishes the strength of the weld. By adjusting the distance between the halves of the mold to between 60% and 80% of the combined thickness of the internal and outer walls, welding can resist both the forces of grip and separation. The location of the welds in the part will determine the rigidity and ability to maintain loads. Designers need to remember to pay attention to the impact of the relationship between welds. The ribs and curves are very effective at adding strength to the pieces. The ribs can be designed to support the expected strength in almost any direction. Side walls can be ribbed to add styling strength. The panels can be ribbed to improve stiffness and sagging control. For best results, the ribs must weld the inner and outer walls together at controlled intervals. The location of the rib and the length should also be such that the action of hinging does not develop when applying loads. Alternating the direction of the ribs in non-homogeneous patterns, the ribs create excellent stiffness and the ability to maintain loads. By combining curvature with ribs and welded cones, lightweight plastic parts can become stronger than steel. Circles and arches create some of the strongest structures on earth. Custom-Pak engineers can help with design evaluation and testing so you can be sure you'll achieve the results you're looking for. H. Finishing There are a limitless number of secondary operations that can be performed on the blow molded part to meet the needs of the finished product. Drilling, sawing, milling, cnc routing, die cutting, punching, riveting, screwing, sound, spin or thermal welding, and surface processing are common operations. Almost any secondary step can be done economically if the necessary equipment is available. A surprising number of operations can be done in the form. Custom-Pak provides a huge variety of secondary operating equipment for free. Decorating moulded parts requires planning at the design stage. To transfer heat or hot brand decorations, part of the design should provide the means to support the tonnage stamping process. For in the form of labels, magazines hold labels and surface locators form must be prepared. For embossed plaques, the attachment method should be included in the part design. The texture is usually applied to mold surfaces. The blow casting texture is usually etched .008 to .012 deep in the surface of the mold (much deeper than casting injections). The project can be to make the texture form and continue to let out of the mold. There are many ways to get the look that you want in your kick molded parts. We can help you make sure that your design will look very long after the consumer has made his purchase. Purchase. Purchase. blow moulding design guide pdf. blow moulding design guidelines