Rotational Molding Parts: Design Made Easy

by Michael Paloian, Integrated Design Systems, Inc.

Introduction

The photo shows a small .05" thick nylon part measuring less than 6 inches long

This large rotationally molded water filter section measures 6 feet across.

Designing parts that are rotationally molded (RM) is easy and fun. The reason it's fun is because of the process's versatility. You can design parts ranging in size from 6" long (Photo 1) to 6 ft in diameter (Photo 2). You can design parts that are simple, complex, soft or rigid. Parts can be molded with inserts, graphics, multiple colors, and in a variety of textures. Hollow parts can be filled with foams for added rigidity, cement for added weight or filled with gases and liquids. The possibilities are only limited to your imagination.

I prepared this article for *Rotation* magazine based on some of my personal observations and experiences with rotational molding. It was written to appeal to a general readership of designers with varying levels of experience in rotational molding. The information has been prepared and presented as an informal review of the process and materials based on part geometry and application versus the "do's and don'ts" of design. Since designers typically dislike rules, I have tried to minimize them in this article. Instead, I thought it would be more relevant to discuss part design based on its interrelationship with process, material and geometric features. These interdependencies will be explained with examples and illustrations.

Process Overview

Since virtually every reader of this magazine is familiar with the basic principles of the rotational molding process, a general overview of the process has been omitted. However, I thought it would be interesting to discuss the interrelationships of processing parameters and part design based on each phase of the three-step process. This overview should provide the reader with a better approach to creatively developing well-designed parts based on processing parameters and limitations.

Zone 1: Loading and Unloading

The first stage of the process involves preparation of the mold or molds for the next cycle. During a production cycle, the mold is opened, parts are removed and the mold is cleaned. After the mold has been prepared for the next shot, the following tasks may be completed.

Application of graphic decals

Transfer graphics are applied to the mold surface before the mold is filled with resin

Molded in graphics are frequently applied to rotationally molded parts for decorative, practical or safety reasons. Companies such as Mold In Graphic Systems® provide rub on stencils, which are applied to the walls of a mold as shown in the picture above (Photo 3). The only design limitations for incorporating molded in graphics is avoid the following:

- Avoid compound curved surfaces where the decal is applied. The decals come on a transfer sheet and are best applied over flat or cylindrical surfaces.
- Avoid small features such as embossed or recessed names where decals would be difficult to apply. Burnishing the plastic carrier sheet transfers the graphics onto the mold surface. It would be difficult to register and align the graphics to many small features.
- If the color is to be highly saturated and opaque, avoid designing graphics, with light colors over very dark colors. An example would be a graphic decal

with a large solid area of white molded over a black part. The black will have some bleed through.

Molded in inserts are placed in the mold before the resin is loaded.

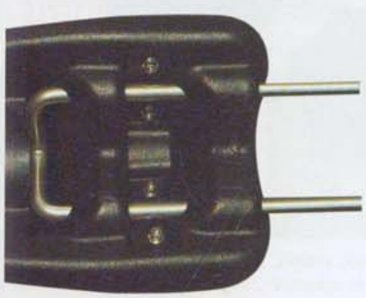
Installation of molded in threaded inserts

RM parts frequently require molded in threaded inserts to accept screws, nuts or other types of hardware. These inserts are usually available as stock hardware (photos 4 & 5), which is available in a variety of materials such as steel, aluminum, stainless steel and brass. You can check with your molder to help you to specify a particular type. I try to specify standard inserts whenever possible to minimize cost and avoid availability problem.

If the inserts are properly heated, resin will encapsulate them as shown.

However, there are occasions when standard inserts are not satisfactory and custom inserts must be designed. If you must design a custom insert, try to comply with the following guidelines:

- Design the insert for ease of machining to minimize cost.
- The insert should include features to secure it in all three axes within the plastic.
- There should be enough surface area on the face of the insert to absorb heat from the mold wall. This heat is required to fuse the plastic over the insert, locking it into place. This can be confirmed with your molder.
- The plastic material surrounding the insert should be sufficient to prevent premature failure. Avoid designing an insert where the surrounding plastic forms thin walls which may not fill or be too weak.
- Avoid designing inserts with through holes, unless you want them filled. Remember that the insert will be coated on all exposed surfaces, including holes.

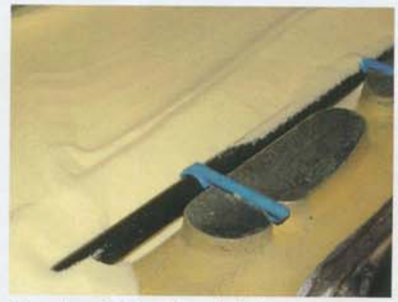


Large steel inserts can also be placed in the mold and molded in place for added rigidity.

Inserting special molded in hardware

Another benefit of rotationally molding polyethylene is the fact the large metal inserts can be molded within the part (photo 6). These inserts could be steel tubes, sheet metal plates, or complex machined parts. There are many advantages of including large structural metal inserts within RM such as improving rigidity, transporting fluids or providing special functionality. The same considerations previously cited for threaded inserts are applicable to the large inserts.

Weighing resin and filling mold



Granular resin has a low bulk density occupying more volume than molded resin.

More than 90% of RM parts are polyethylene (Photo 7). Unlike injection molding, where the material is pelletized, rotational molding grades of polyethylene are supplied as a finely granular powder. This powdered resin has more surface area than palletized resin, thus enhancing even heat distribution for a uniform melt. Another characteristic of this form of resin is its low bulk density. When the resin is weighed and poured into the mold, it requires more volume than the final part. In most cases this is not a problem since the volume of the mold is sufficiently large to accommodate the shot weight. However, there are special circumstances when the volume of the mold is inadequate to store the powdered resin. In these instances, you should be prepared to allow an extra chamber in the mold to be included to store the powdered resin. This may affect part design even though the extra bulb of material is eventually clipped off and discarded.

Molds are biaxially rotated in an open oven to melt the resin.

Zone 2: Heating

After the mold is loaded with resin and sealed, it is transferred to the second zone where it is heated (Photo 8). Heat is typically transferred to the mold by convection and to the resin by conduction through the mold wall. Uniform heating is one of the biggest challenges of rotational molding. It influences part design, mold design and processing. To gain a good appreciation for the complexity of this phase of the process, we should discuss polymers and viscosity.

Material Properties: Viscosity and Bridging

The distance between opposing parallel walls should be a minimum of 3 times the wall thickness.

One distinctive property common to all thermoplastic materials when they are heated to a molten state is high viscosity. This property is what makes plastic processing so challenging and unique when compared to cast metals.

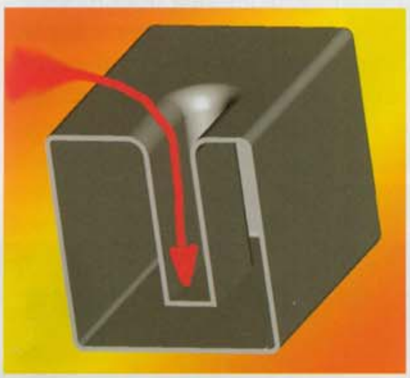
Viscosity is commonly defined in units of g/10min, which is called melt index. These units are based on an ASTM test, which measures the mass of material flow within a given time under specific conditions. Unlike metals, which simply melt and flow like water when heated, thermoplastics melt into viscose tar like mass. This phase change is essential for rotational molding, which requires the molten polymer, typically polyethylene, to melt and to stick to the walls of a hollow mold. You may question the significance of viscosity on part design. Part design is affected by the limitations viscosity imposes on the reproduction of certain features due to a phenomenon called bridging. As the name implies, bridging is when the resin forms a void in the part by "bridging" across narrow features instead of properly filling them. Examples of these features include molded in threads or sections where two opposing walls are too close together (Photo 9). To avoid this problem, directly opposing parallel walls should be separated no less than 3 to 5 times the nominal wall section as shown.

Recent developments in a new molding compound developed by Mold In Graphic Systems® have eliminated this problem. They have introduced a clay like substance of polyethylene, trade named RMC³ (Rotational Molding Compound), that can be placed in these troublesome areas. During the heating cycle the material simply changes form a its initial state to a molten state without flowing. All details are thus accurately preserved as shown in the pictures below (Photo 10).

Heat Transfer

Mold-In Graphics has recently introduced their new RMC³ compound that can reproduce features that were once impossible to rotationally mold.

Most of us designers do not bother to think of things like heat transfer when we design parts, but in rotational molding it is one of those things in which we should be aware. Unlike injection molding where a slug of molten resin is injected into a mold under a pressure of 20,000 psi, rotational molding relies on gravity and uniform heat to melt the resin. Uniform heat distribution in a mold is dependent upon equipment, mold design and part design. As the mold is biaxially rotated in the oven, powdered resin forms a pile at the bottom of the mold coating the interior walls. When the interior mold surface reaches approximately 450° F the finely powdered resin begins to melt onto the interior surface forming a thin coating. It's analogous to freezing rain hitting pavement and forming a thin layer of ice. Wall thickness builds as new layers of resin fuse to the melted undercoating. Areas that are not heated sufficiently will be thin or barren, creating voids. Molders take advantage of the phenomenon by intentionally insulating areas of the mold to create openings. The insulated surfaces are usually PTFE, polytetraflouroethylene, or PEEK, polyetheretherkeytone.



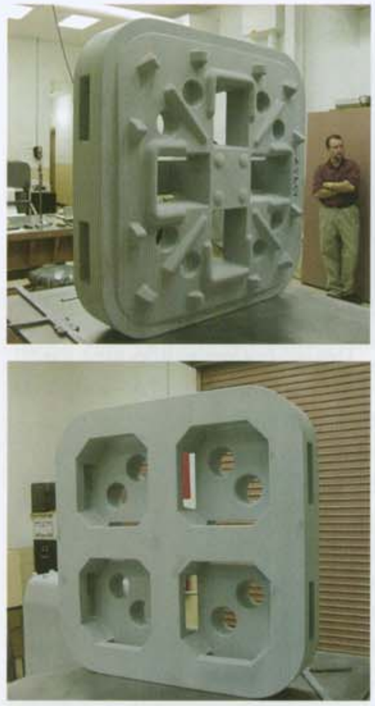
Mold-In Graphics has recently introduced their new RMC[3] compound that can reproduce features that were once impossible to rotationally mold.

Large broad surfaces will heat uniformly, provided the mold wall section is constant. However, even heating becomes more challenging with complex parts where there are deep pockets or areas where the mold wall is thick (Photo 11). These conditions should be discussed with a molder or mold maker before you finalize the design. Molders will sometimes concentrate heat in specific areas by using air amplifiers, black paint, or heat fins and pins. Incorporation of these techniques permits control and variation of wall thickness in desired areas of the part.

Heat Transfer/ Flatness

Heat distribution also affects warpage, flatness, bowing, and twist.

Slightly domed surfaces prevent unpredictable warpage that could occur on a flat surface.



Features such as these add a great deal of rigidity to a part.

These problems can be controlled with part design. For example, let's say we want to design this part with a large flat surface. Let's say its 36" square. It looks great on the computer screen but one thing is for sure, it won't be flat when it's molded. Most likely it will be twisted, bulged, warped or concave, or any combination of these. So what do you do? Avoid large flat surfaces by placing an intentional convex or a concave crown in the surface (Photo 12). This will be accurately and predictably reproduced during production. Other solutions include kiss-offs on the one side or through holes for added strength (Photo 13). Another solution is to add reinforcement. Sometimes parts can be filled with polyurethane foam for added rigidity as well improved flatness.

Zone 3 – Cooling

Heat Transfer/Shrinkage and Draft

The last phase before a part is removed from the mold is cooling. During this phase of the process the viscoelastic melt will cool and shrink to a solid state. Polyethylene shrinks a great deal. As a matter of fact, it shrinks.015 to .030 in per inch as it solidifies. This means that a mold for a 24" long part should be almost 3/4" bigger than the part itself. So what do you think happens to parts as they shrink? That's right; as the plastic shrinks it will lock around any protruding features in the mold and pull away from mold walls. If these features are straight the part won't come out very easily. So it's desirable to add a slight taper or draft to these walls. A taper as small as 1 degree, will promote easy part release. Depending upon where the parting line is, draft may not be required on some exterior surfaces since the plastic will naturally pull away from these walls.

Tolerances should be generous as possible and should be discussed with a molder.

Shrinkage/Tolerances

Shrinkage contributes to another processing condition causing dimensional variation because no two parts ever shrink exactly the same. One part may shrink a little more or less than another. Designers must account for this part variation by designing parts with adequate clearances and features to account for tolerance variation (Photo 14). Anyone can design parts to fit line to line but good designers must account for molding tolerances so parts can easily be manufactured within spec. If designers follow the published SPE acceptable tolerance ranges for rotational molding it would be impractical since every part would require excessive clearance to fit to one another. I personally don't follow the guidelines. I've found that tolerances can be held as tight as +/- .03" to +/-.1" for parts ranging in size between 4" and 25". What I frequently do is provide as much clearance between parts as possible and check with the molder about tighter tolerances. I also discuss what design modifications or fixtures could be made if there is a tolerance problem after parts are molded. Of all the processing parameters I've discussed so far, tolerance predictability is one of the greatest challenges for designers and molders alike. There are many factors, which make tolerance prediction in rotational molding so difficult. They can be attributed to part geometry, mold design, molding conditions, resin grade and lot, as well as post processing operations.

Secondary Operations and Part Design

Trimming

RM parts are often trimmed after they have been removed from the mold. Trimming operations are usually done manually to remove flash, drill holes or mill openings using a router. On higher production runs, a CNC router performs trimming. Trimming operations are performed to add features and openings not possible by rotational molding. Sometimes two or more parts are molded in the same tool as one part to minimize tooling and part cost. They are cut into individual parts and trimming in secondary operations to their final shape.

Foam filling

Polyurethane foams add rigidity and insulation to parts.

When parts must be very rigid, polyurethane foam is injected into the hollow section as a secondary operation (Photo 15). Foam density can be varied to produce various levels of strength and rigidity. Polyurethane foams also improve thermal insulation and are often used to produce insulated chambers. It should be mentioned that polyethylene foams have also been rotationally molded for added rigidity. These foams are added at the beginning of process.

Spin welded fittings

Spin welded inserts require adequate space of the chuck.

Injection molded polyethylene fittings are typically spun welded into RM parts when screw threads are required. These parts are standard hardware, which is fused to the body of a RM part by spin welding. The resulting joint is water tight and as strong as the polyethylene. When you design a part with molded in threads I recommend specifying this type of fitting. You should provide enough room around the fitting to permit the spin welding chuck to fit to the insert (Photo 16). I should also mention that the recently introduced polyethylene clay referred to as RMC by Mold In Graphics Systems can also be used in place of these spin welded fittings. The RMC would be added to the mold before the cycle as opposed to being added in a secondary operation. RMC is a proprietary material exclusively manufactured by Mold In Graphic Systems®.

Sheet metal molds are a very cost effective alternative for rotational molds.

Molds and Part Design

Well-designed RM parts require some basic understanding of mold design and construction. Rotational molds are typically fabricated from sheet metal, cast aluminum or machined aluminum. Large simple parts are typically molded using sheet metal molds (Photo 17), which are the least expensive. Complex parts are molded using cast aluminum (Photo 18), which is the most popular based on cost and versatility. Parts requiring tight tolerances and excellent detail are molded in machined molds (Photo 19), typically the most expensive. Since cast molds are the most commonly used, I will focus on this type of mold.

Cast aluminum molds are the most popular method of making rotational molds,

Machined rotational molds are the most accurate and expensive.

Uniform wall thickness is a key consideration in rotational molds, which provides even heat transfer to the resin. Mold thickness typically ranges from 1/4" to 3/8". A surrounding welded tubular steel frame supports the irregular shell, providing protection as well a flat surface for mounting to the arm. Compression springs are often placed between the cast mold and the frame to take up tolerance variations between them. Interior mold walls are coated with Teflon or PTFE to enhance easy release of the part.

Compression springs are used between the frame and casting to absorb tolerance variations.

Complex parts with undercuts require multiple piece molds to produce all desired features. Parting lines are formed along seams where the mold sections are split (Photo 21). The size and

Complex features forming undercuts require extra removable parts in the mold.

appearance of these parting lines can be a very pronounced in low cost molds and older molds. They may appear as wide irregular lines (sometimes as wide as 1/4") dissecting the part along every section where the mold was separated (Photo 22). If cosmetics are important, these parting lines should be located in areas that are not visible or they should be disguised. Molding an intentional recess along the parting line or placing them at the bottom edge of the part can disguise parting lines.


Parting lines can sometimes produce thick witness lines as shown in this photo.

Holes and openings can be molded into parts by informing the molder to insulate the open area with PTFE or PEEK. These high temperature plastics will prevent polyethylene from melting in the specified area, creating an opening. This is technique is especially effective if you are designing a part with many holes thus saving extensive secondary trimming costs (Photo 23).

Secondary machining can be minimized by molding parts with PTFE inserts which produce molded in holes as shown in this golf club POP cart.

When large inserts specified as a molded in feature, you should consider how the insert would be supported in the mold. Features to support the insert must be provided in the part geometry.

Structural properties can be enhanced by designing RM parts with double walls interrupted with molded in kiss-offs or openings. These features should be designed to provide adequate air circulation and heat transfer. If the holes are too narrow relative to the depth, heat transfer will be limited resulting in thin walls or voids. Details for these parameters can be discussed with a molder. The importance of gaining input from molders throughout the design process can't be over emphasized. I try to make this a habit in all the projects we are involved with. I find that discussions with the molder early in the development process results in more creative design solutions and a better product.

Tolerances/Absolute and Relative

Tolerances can be grouped into two classifications. The first is absolute deviation and the second is relative deviation. Absolute deviation is a variation from a print dimension or CAD file. It is based on a cumulative buildup of errors resulting from the pattern, mold and molded part. Tolerances in the pattern are usually very minor. Today most patterns are usually within +/- .005" to .010" everywhere, since they are typically cut on a CNC miller directly from our design CAD file. However, these patterns are oversized from the print with two shrinkage factors, one for the resin and the other for the aluminum casting. This can be as much as 4 to 5%. This estimation is where our second tolerance stack up occurs. In order to minimize this deviation, mold makers try to maintain a uniform 1/4" nominal wall in the casting for predictable and consistent shrinkage. After the mold is cast, it is cleaned and prepared for production. From this point on, all parts molded in this tool will be identical except for variations caused by production. I call these relative tolerances, which is the third level of tolerance stack ups.

Relative deviations are caused by factors such as the resin's melt index, cycle time, temperature and a host of other variables. I try to design parts based on relative tolerances whenever possible because it's easier for the molder to predict based on his/her experience. Maintaining relative tolerances gives the molder a bit more leeway, which keeps him happy, and your part costs lower. Therefore, if you're designing rotational molded parts that must fit to one another and overall dimensions are not critical, this should be communicated to the molder. After the first run, you and he/she could discuss what must be maintained during production for proper form, fit, and function.

However, in certain situations tight dimensions must be maintained. For example, if you are designing a part that must mate with a purchased component such as a motor, then the tolerances between holes must be maintained in order for the two parts to be properly assembled. This is an example of absolute tolerances. During these situations you should discuss your requirements with the molder early in the design process, as you're developing your design. Your molder can get in touch with his/her toolmaker and propose methods for achieving these tolerances. On method is to include the addition of removable inserts that can be replaced or adjusted after the first production run.

Conclusion

This article is meant to be both informative and interesting. As a designer, I hope you will be able to design your next rotationally molded part with a better understanding of the causes and affects between design, process and materials. Applying this insight with imaginative design solutions will provide you and your company with innovative designs in the present and future.