Designers Guide to

Rotational Moulding

amber plastics ltd

Tel: +44 (0)1246 456525 www.amberplastics.co.uk

Amber Plastics Ltd. Broombank Road, Chesterfield Industrial Estate, Sheepbridge, Chesterfield, Derbyshire. S41 9QJ

Amber Plastics, here to help

AMBER is a specialist, with over 30 years of experience, in the **ROTATIONAL MOULDING PROCESS.** This guide has been compiled as a brief introduction to the factors affecting the design of rotational moulded products. It is intended to assist design engineers with development of initial concepts and designs.

The knowledge and experience of Amber Plastics is here to help ensure your product is a success. Our full service starts with design assistance so please ask for our input before finalising your designs.

Design Guide Contents

Benefits of Rotational Moulding

The process has many benefits over other plastic forming processes. These include:-

Control of Wall Thickness: It is possible to produce products that have a very uniform wall thickness {they do not have thin corners as in some processes}. Indeed, it is possible to "thicken- up" the wall thickness in areas which need to be stronger.

Low Stresses: Because no pressure is used in the process; parts have relatively low stresses in them. The lack of locked in stresses protects the items from cracking when they are subjected to physical stress or attacking chemicals.

Size: AMBER can produce parts up to 2m long

Waste: Just enough powder is placed into the mould to make the part with the desired wall thickness; therefore very little waste is created.

Shapes and Details: Hollow, totally enclosed items {such as tanks} can easily be made. Parts with intricate shapes and undercuts can be moulded. Metal inserts can also be moulded in. The process gives parts with excellent surface detail so special finished, textures, instructions etc. can be moulded in.

Lower Cost Tools: Because no pressure is involved in the process the moulds used can be made from sheet metal or cast / cnc machined aluminium. This means much lower moulds costs than other processes, allowing relatively low quantities to be economically produced.

Limitations of Rotational Moulding: The chief limitation of rotational moulding is that; the process uses a female tool only and is not able to provide a precisely detailed inside surface. Therefore accurately moulded internal bosses, ribs etc. are not practical.

This means thin sections, such as ribs and closely spaced walls, cannot be filled out; this needs to be allowed for in the design *(see later sections).*

Basics of the Rotational Moulding Process

Loading: The exact amount of powdered plastic, equal in weight to the moulded product; is placed in the bottom section of a split mould.

Heating: The tool halves are closed by clamping together. Heat is applied at a controlled rate whilst the mould is rotated slowly about two axes at right angles to each other. Because the tool is rotated slowly no centrifugal force {which would throw material outwards} is involved. The powered plastic material remains in the bottom portion of the tool.

As the tool rotates; all the surfaces of the cavity eventually pass under the bulk of powdered plastic material.

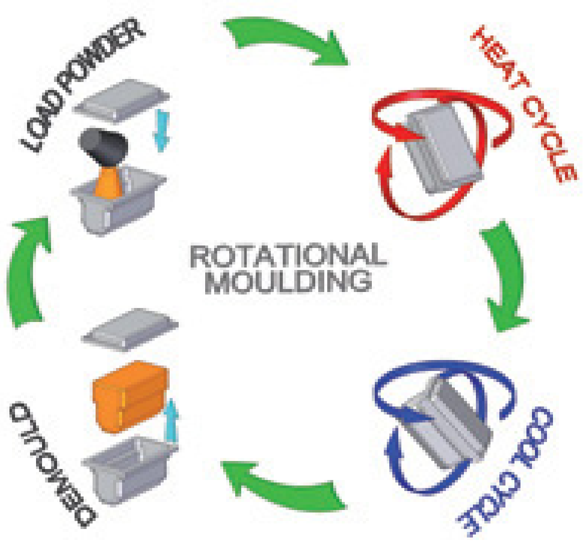
When the temperature on the inside surface of the tool becomes high enough, the plastic adheres to the inside walls as a thin layer.

As the tool continues to rotate, it will pass under the powdered plastic again & again, until all the plastic material has been layered onto the inside surface of the tool

Once all of the material has been layered, on the tool walls, the heating is continued to allow ; a} all the particles to melt and flow out to give a smooth inside surface to the part, and b} the air bubbles, initially trapped in the plastic wall, dissolve and give a completely solid void free moulding wall. When this has been achieved the tool is the cooled.

Cooling: The tool is transferred to the cooling station where is it subjected to air, water jets or water mist cooling. The tool cools down and the part solidifies retaining the shape.

Unloading: After cooling, the tool is transferred to the unloading station where the two halves are opened up and the product is then removed. The tool can then be recharged with powder and the process will begin again.



amber plastics ltd

Materials for Rotational Moulding

AMBER uses materials such as Polyethylene, Polypropylene, Nylon 12, EBA.

POLYETHYLENE:

Polyethylene (or Polythene as it is also known) is, by far, the material most used for rotational moulding. It has ease of processing, good low temperature impact strength and excellent clinical resistance. It falls into various categories.

L.D.P.E. {Low Density Polyethylene} - This is flexible and tough, easy to process and has low strength, excellent chemical resistance.

L.L.D.P.E. {Linear Low Density Polyethylene} - Better mechanical properties than L.D.P.E. Higher stiffness, excellent low temperature impact strength and environmental stress crack resistance.

H.D.P.E. {High Density Polyethylene} - Better stiffness than the lower densities but more prone to warping during moulding and lower impact strength.

EBA Copolymer – Gives excellent low temperature flexibility. Has less strength than straight polyethylene.

POLYPROPYLENE:

AMBER is the leading processor of polypropylene in the UK. It gives excellent resistance to stress and high resistant to cracking (i.e. it has high tensile and compressive strength) and high operational temperatures with a melting point of 120°C.

It is highly resistant to most alkalis and acid, organic solvents, degreasing agents and electrolytic attack.

NYLON:

Nylon 12 – Low Moisture absorption and good chemical resistance but mechanical properties and heat resistance lower than, for example, Nylon 6. Nylon 12 is more easily processed but more expensive.

Nylon’s secondary finishing operations are more difficult than polyethylene’s.

Design of Rotational Moulded Products

Before a rotationally moulded plastic product can be designed and a material chosen it is necessary to determine exactly what is required of the part both in terms of its function and its environment. If required Amber Plastics can help with a Design Check List to help with material selection.

The design of a rotationally moulded plastic product will be influenced by various factors arising from the process and from the properties of plastic materials in general.

The Rotational moulding process has its own particular characteristics. In order to get the best possible rotationally moulded product at an optimum cost all characteristics need to be considered during the design of the product.

Various factors that need to be considered are as follows:

- Wall Thickness
- Corner Radii
- Taper and Draft Angles
- Flatness
- Ribs
- Difficult to Mould Geometry
- Undercuts
- Tolerances

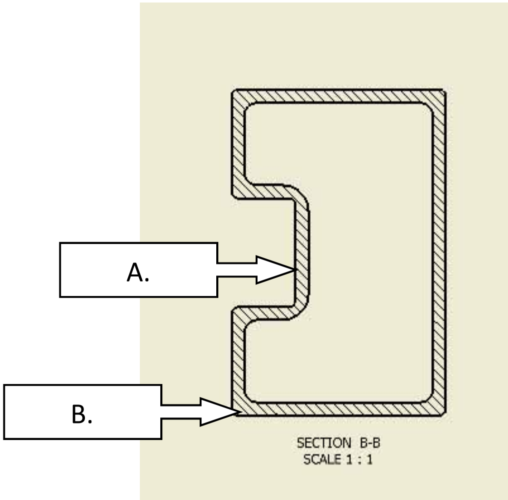
The following pages give more guidance on these factors.

Wall Thickness

The ideal way to specify the wall thickness on a rotationally moulded part is to specify the NOMINAL wall thickness and also the MINIMUM wall thickness that can be allowed anywhere on the part. Since moulding is done with a female tool only – with no matching male portion of the tool – accurate wall thickness is not attainable.

A tolerance of +-20% should be considered as a commercial tolerance whereas +-10% would be precision and be more expensive to maintain.

It should also be noted that because of material flow characteristics outside corners tend to be thicker that nominal wall {usually an advantage since outside corners are frequently heavily loaded} whereas projections into the moulding tend to thin out. These are generally not regarded within the general wall tolerance.



A. Inward projections thin out. B. Outside corners thicker

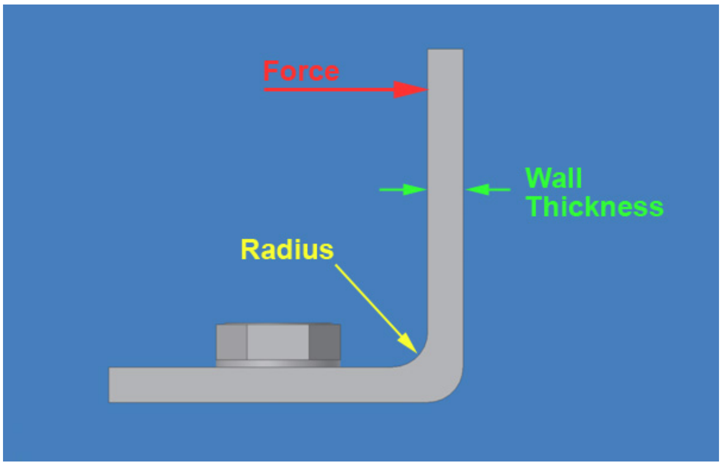
Although generally the process produces relatively even wall thickness it is possible to locally increase the wall thickness if necessary.

A typical wall thickness, at Amber, ranges from 3mm – 10mm.

Corner Radii

Radii on the corners of rotationally moulded parts fulfil two functions:

A. They distribute the corner stress of the part of a broader area which adds strength to the part.
B. They help the moulding of these corners by the process – too tight a radius can give an incomplete corner.



A plastic part, on loading, will be highly stressed when the radius R on the inside corner is less than 25% of the nominal wall thickness. The stress is reduced as the radii are increased up to 75% of the wall thickness. Increasing the radius has much less dramatic effect on the stress reduction above this.

As well as the effect on stresses, sharp corners are problem areas in moulding.
Sharp inside corners tend to:

A. Be the last portions of the mould to react to moulding temperatures.
B. The plastic has a tendency to flow quickly over these corners.

These two factors result in a general reduction in wall thickness in the moulded part where there are sharp inside corners.
Sharp outside corners cannot always be filled out completely. If the corner is too sharp the first layers of plastic picked up by the mould tend to bridge across the corner leaving air bubbles and incompletely formed radius

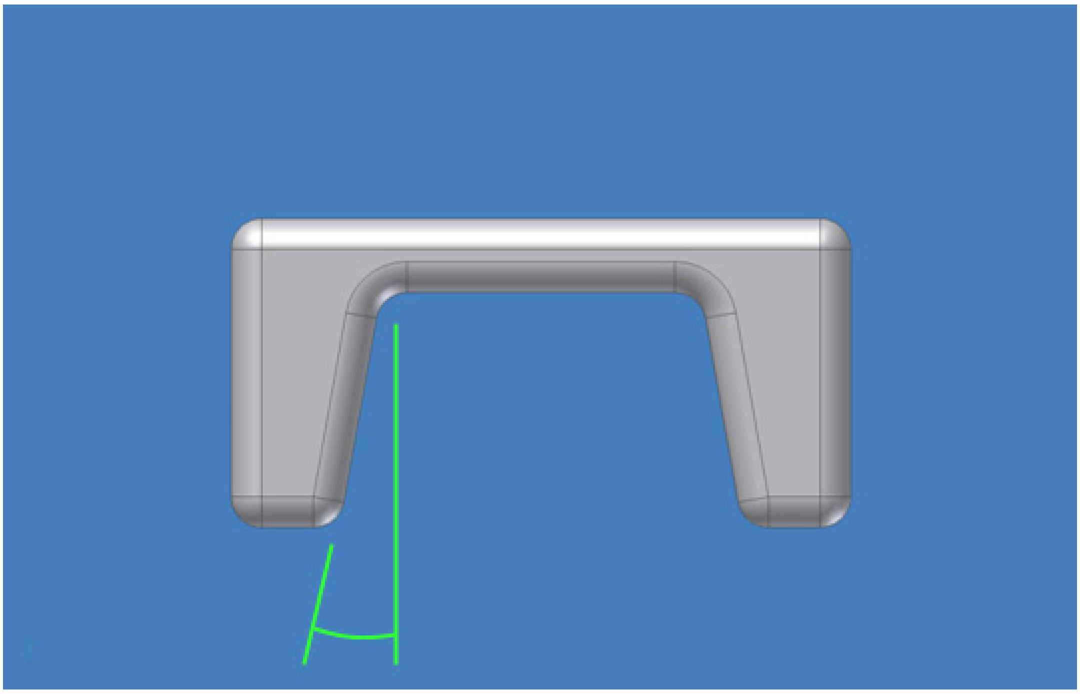
Therefore, in rotational moulding it is recommended that all corners have generous radii. The table below gives the recommended radius for Polyethylene and Nylon.

POLYETHYLENE	INSIDE RADIUS	OUTSIDE RADIUS
Ideal	**12**	**6**
Minimum	**4**	**3**
NYLON		
Ideal	**20**	**12**
Minimum	**5**	**5**

Taper and Draft Angles

An advantage of rotational moulding, over other processes, is that products can be moulded with no draft angle at all, since the tool is of female form with no internal core for the moulding to shrink into. As the product cools it shrinks away from the walls of the mould making it easier to remove. It should be noted, however, that textured surfaces, moulded in logos etc. may necessitate taper even on a female shape.

The draft angles below are recommended to be incorporated unless it interferes with the functional requirements of the part.



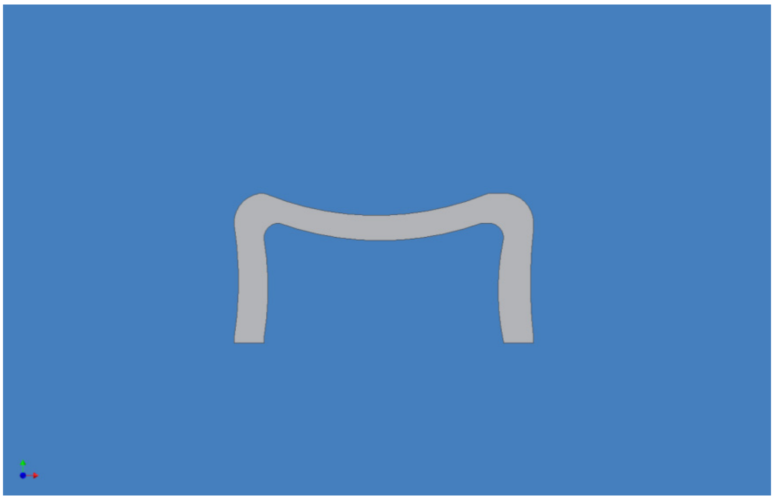
	INWARD PROJECTIONS
Minimum	**1**
Recommended	**2**

Flatness of Rotational Moulded Surfaces

The flatness of product surface is subject to the design and cooling process.

Typical flatness tolerances for polyethylene would be 5% ideal with 2% as a commercial tolerance and 1% as a precision tolerance.

If at all possible, the design of parts to be rotationally moulded should avoid large flat areas. If absolutely necessary they should be broken up by reinforcing ribs *(see next section)* or possibly have a gentle curvature on them. Moulded in detail, such as lettering, logo's etc. can also break a flat surface up visually so that lack of flatness is disguised.



Corners and their radius can affect the flatness of adjacent surfaces as differential cooling rates can cause the corner angle to distort. This effect is minimised by careful design of the corner, the tooling and cooling process.

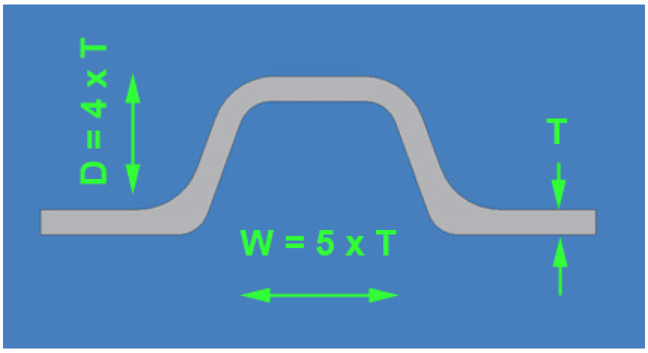
amber plastics ltd

Ribs in Rotational Moulded Products

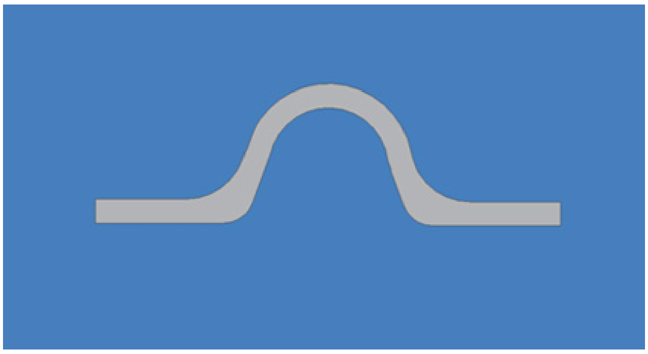
Some thermoplastics used in rotational moulding are not inherently stiff materials e.g. PE, EBA. In order to increase product stiffness, ribs are used extensively without great increase in part weight. The correct use of stiffening ribs results in stiff lightweight parts that can be produced more economically because the wall section uses less material and is quicker to mould.

Stiffening ribs in rotationally moulded parts cannot be designed as solid sections (as with injection moulded or compression moulded parts) - the relevant sections would not fill out. Instead the stiffening ribs are designed as hollow sections similar to corrugated sheet. They can be raised or embossed.

The diagram below shows good average proportions for ribs where the depth D is at least four times the wall thickness T and the width W is at least five times the nominal wall. Increasing the depth D increases the stiffness but it also increases the difficulty in moulding and part removal. Decreasing the width W makes moulding more difficult increasing the chance of material bridging off between the two walls and not fully filling the rib.



Round ribs shown below are often specified as they are much easier to mould however they do not provide as much of an increases in stiffness as there is less depth D lying perpendicular to the wall.

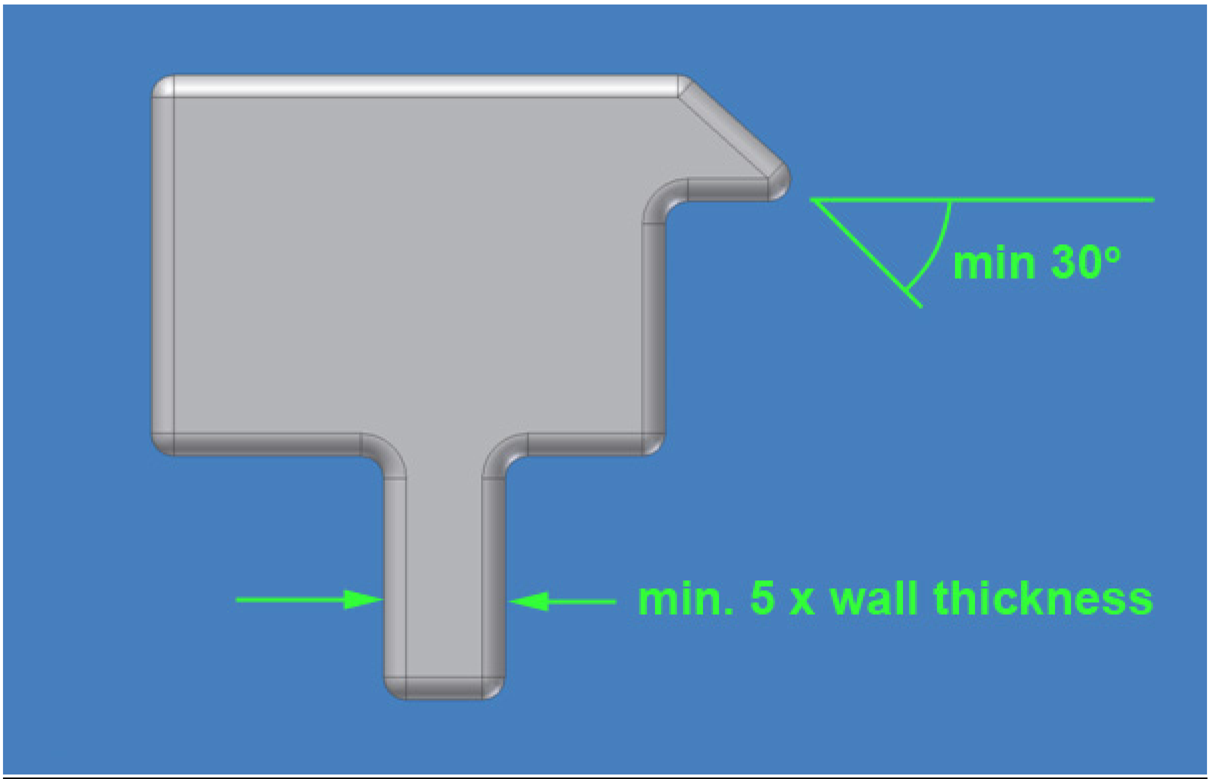


Difficult to Mould Geometry

If the design, of the product, is not done with the rotational moulding process in mind certain areas can prove troublesome to mould. These include small corner angles, walls very close together and parts with undercuts.

The first two of these result from material bridging over the space between the two walls of the moulding when they are too close together. If bridging occurs the bottom part of the moulding will not fill out

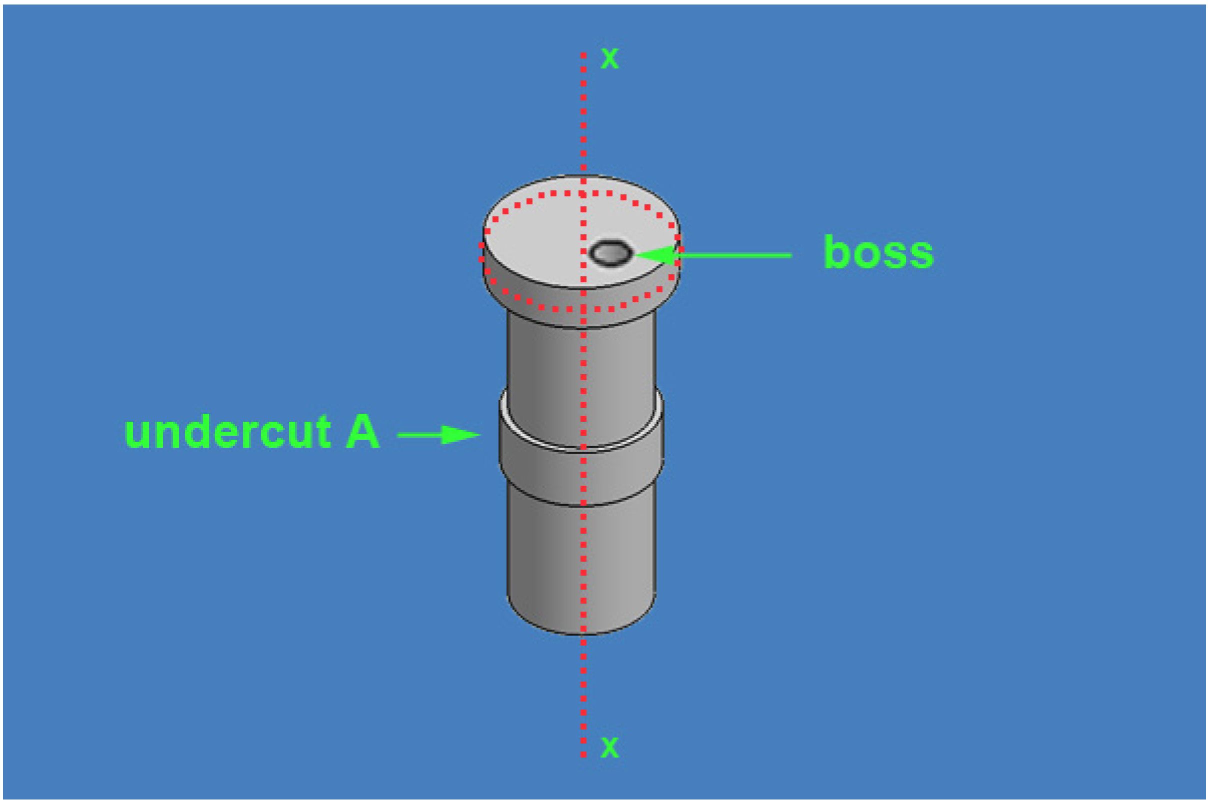
To be sure of proper fill the width should be a minimum of five times the wall thickness.



If an angle is too acute we have a similar effect. If possible the angle should be kept to a minimum of 30 degrees and all radii should be kept as large as possible.

Undercuts on Rotational Mouldings

An undercut is any wall projecting inwards or outwards, parallel to the parting line of the tool, which makes the removal of the moulding difficult if not impossible.



Since the rotational moulding process uses hollow tools, with no male core, it is sometimes possible to use the shrinkage during moulding to enable small undercuts to be removed.

Hence in the diagram if the undercut at A is small enough *(and even better if there is a generous radius at A)* then the part shrinkage will enable the part to be removed. If the undercut is too big then, the split line must be moved to XX and the inward projecting boss has then to be moulded using a removable loose piece. Thus the part has to be designed with product removal very much in mind.

Tolerance of Rotational Moulding

As previously mentioned; it is difficult to hold rotationally moulded parts to tight tolerances. The outside dimensions of a rotationally moulded plastic part are free to draw away from the inside surfaces of the tool as the plastic cools and shrinks.

It should be noted however that where the tool form represents a male form and the moulding shrinks on to the mould it is possible to hold tolerances much more closely.

It should also be noted that since the inside surfaces of the part are formed only by flow of the plastic and not against a males core it is very difficult to control these to any repetitive degree of accuracy.

Although each product design is a special case which must be given individual consideration; in general, a tolerance of 2-3% would be considered a commercial tolerance and 1% to be precision. As with all tolerances the best is the broadest tolerance that will satisfy the end use requirement of the part.