Guide to Product Design

Issue #3: Injection Moulding Design Guidelines & Why 'DFM' is a Bad Term

04 February, 2021, Bryce Holliss

Design Knowledge Series (White Paper – Detailed Version)

'DFM' or 'Design for Manufacturing' is supposedly the process of designing products that are suitable for manufacturing. It's a skillset that you'll find listed on nearly every product or industrial designers resume, or a service offered on most design firms' websites.

The problem with using this term, and indeed its' overuse, or advertising it as a service, is that all real products should be designed to be suitable for manufacturing. 'DFM' isn't a service, it is an inherent requirement of a products' design. If your product hasn't been designed to be manufactured, it isn't a product design at all, simply an expensive and unusable CAD file.

Further, 'DFM' is generally spoken about as a stage of the production development process, that you can design a product first and then retrospectively optimise the design for manufacturing. This is an incorrect ideology; a good product designer will design your product with manufacturing in mind throughout the entire design development process – from initial concept assessment to CAD development to prototyping and to tooling and moulding.

Understanding How to Truly Design Manufacturable Products

From the very start of a product designs' life, you need to be thinking about how it is going to be made.

For the moment, let us focus on designing injection moulded parts. At the very start of a project, when a product idea has just been conceptualised, how the product will be physically assembled and form together needs to be thought out in precise and clear detail. This isn't just limited to the design itself, for example thought also needs to go in to:

- What material can be used? What are the performance properties?
- What is the environment of use? Does this affect the material that can be used?
- What size moulding machine will be required? Is specialised machinery required?
- Will all components be injection moulded or will there be other materials? How are these manufactured/sourced?
- Is the part going to be over-moulded?
- How will it be assembled? Are fasteners required? How/where can these be sourced?

As an experienced design, tooling, and injection moulding company, we want to provide not only the technical or theoretical guidelines for designing parts, but also give an insight into the further intuitive questions you need to be considering when designing injection moulded parts.

The following are important design considerations we believe you should make when making injection moulded products.

Wall-Section, & Structural Ribbing

Generally, for injection moulding, you want your products wall section to be as thin as possible without compromising the products functionality or performance. This is because a thinner wall section means less material and material cost, which in-turn means a smaller injection moulding machine with less tonnage and likely charged out at a lesser rate is required. Further, thinner parts cool quicker in the mould, allowing for shorter cycle times, meaning that more parts can be manufactured quicker.

Most injection moulded parts we design would range between 1.5mm to 4mm thick wall sections, depending on the material. Different injection moulding plastics offer different mould flow characteristics and subsequently have different maximum and minimum wall thicknesses – see the table below.

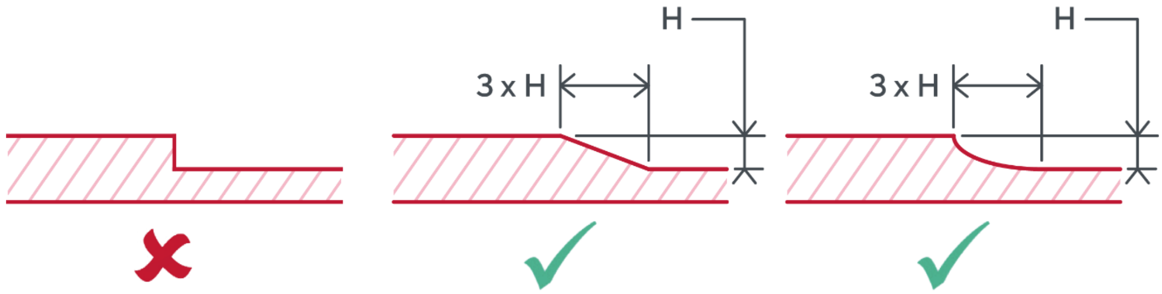
Materials	Minimum(mm)	Maximum(mm)
ABS	0.75	3.20
Acetal	0.50	3.20
Acrylic	1.00	6.00
Nylon	0.50	4.00
P.C	1.20	10.00
HDPE	1.20	5.50
LDPE	0.50	5.50
P.P	0.65	6.00
P.S	0.80	6.00
PVC	1.00	9.00

Uniform Wall Thickness

As the mould cavity is injected with plastic, you want the products wall thickness to be uniform so that the flow of molten plastic is not restricted or extremely variable. This is so the cavity fills completely and solidly with plastic.

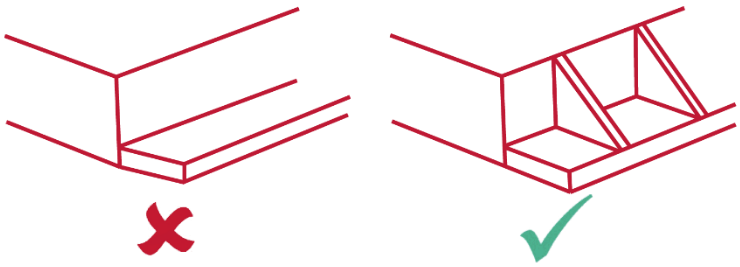
Additionally, the thicker the plastic, the slower it cools. Because of this, thinner wall sections will solidify before thicker ones; this is one of the many reasons that causes cosmetic part defects like warping, sinking, or deformation.

As plastic cools, internal stresses are created. You want these stresses to be predictable and uniform so that the part remains consistent. Where non-uniform wall sections are unavoidable, the transition between these sections should be as gradual as possible. A good rule of thumb is to make transitions between different thickness 3 times the length of the change of thickness.



Another option to keep wall thickness uniform, is to ‘core’ the part. ‘Coring’ involves removing material from the opposing side of where the thickness transition occurs. This leaves the part with a ‘step’ as opposed to a thickness transition.

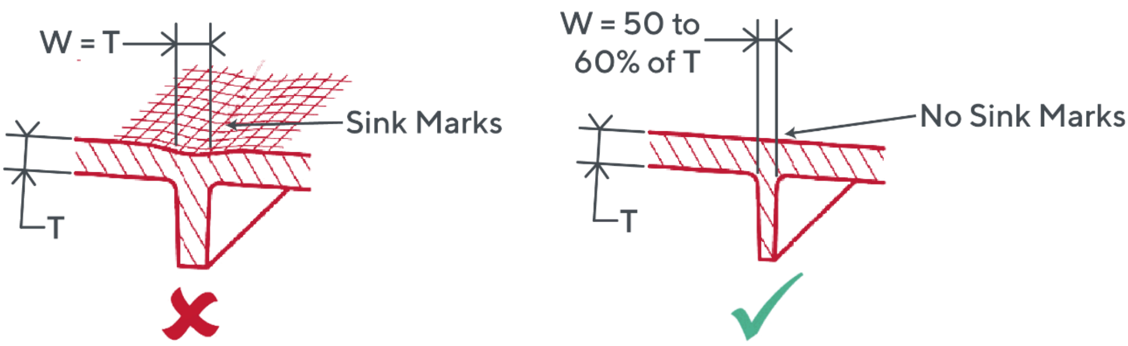
Where a change in thickness cannot be gradually transitioned or cored, to minimise warping from the different thicknesses cooling at different rates, structural gussets can be added along the join between thick and thin sections. Gussets are a type of angled / triangular ribbing, these should be 50-60% the thickness of the nominal walls’ thickness so that the gussets don’t add to the warping problem or creating sinking.



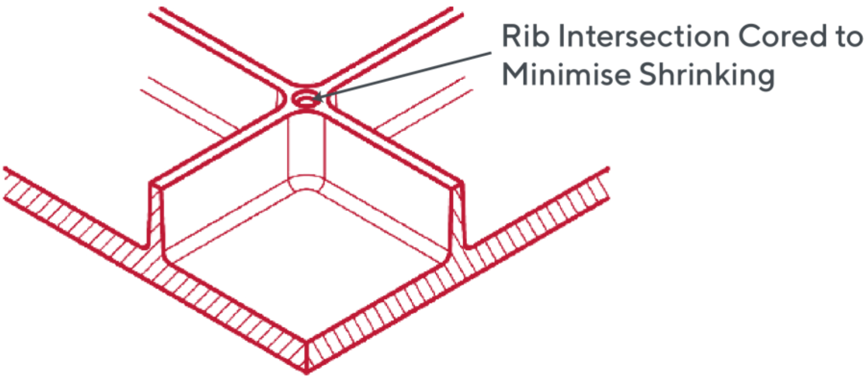
Structural Ribbing

Ribs are a great way to increase a product’s bending stiffness without increasing the products’ wall thickness. Where possible, adding ribs should be done before considering increasing wall thickness.

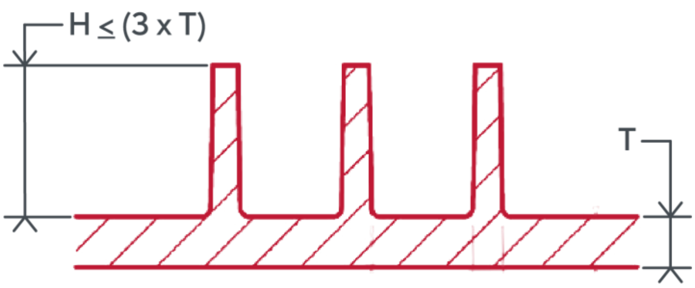
Rib thickness should always be less than the nominal wall thickness. Ideally, you want it to be 50-60% the nominal wall thickness. If it isn’t this thin, the outer wall of the product will show sink marks.



Where ribs intersect, the intersection should be cored out so that a thick point is not created. Ideally, these cores should have a diameter of 5mm or greater.



Further, rib height should be limited to 3 times the nominal wall thickness of the product. Ribs should be spaced a minimum 2 times the nominal wall thickness apart.



It is important to remember that with injection moulding, where there isn't plastic, there is steel, and this steel must be able to cool and it can't be too fragile – you can't have a really long, thin piece of tooling steel as it simply won't last the repetition, heat, and forces of the injection moulding process.

Shrinkage, Warpage, & Sinking

Wall thickness also has an impact on the way parts shrink and warp after they have been moulded. After being moulded, plastic continues to cool and shrink for hours. This is why parts often need to be laid flat or positioned so they do not deform as they shrink. Generally speaking, to be safe, parts should be left for 24hrs after moulding.

Intersections between different wall thicknesses can cause troublesome shrinkage problems. This often occurs if ribs, bosses, and other plastic features are not designed or positioned correctly within the outer walls of a product. For example, if a boss is positioned on a wall, thick sections are created, this plastic takes longer to cool, and as it cools will pull on the material surrounding it, creating sink marks.

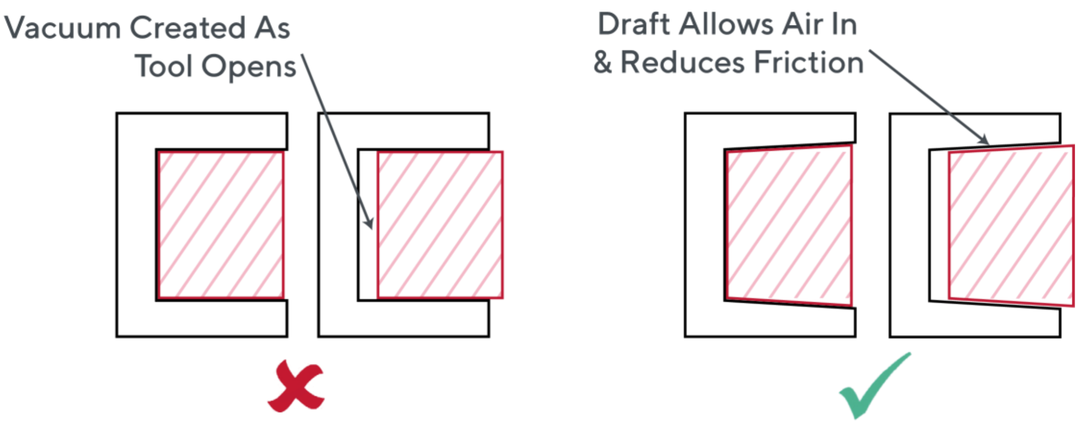
Different materials also have different shrink rates. When designing parts for injection moulding, you usually draw the part to your desired dimensions, however then cut the mould cavity in the tool slightly larger to account for the shrinkage. This is why you need to know the material you want to mould your product in prior to constructing tooling as it is difficult to change material and keep the desired original dimensions.

Materials	Shrink Values
ABS	0.004-0.008
Acetal	0.020-0.035
Acrylic	0.002-0.010
Nylon	0.005-0.010
P.C	0.005-0.007
HDPE	0.015-0.030
LDPE	0.015-0.035
P.P	0.010-0.030
P.S	0.002-0.008
PVC	0.002-0.030

Draft, Texture, & Embossing/Debossing

Generally, draft needs to be applied to product surfaces that are perpendicular to the open/close direction of the injection moulding tool. This Is because if the product surface is parallel, as the tool opens a vacuum is created as the tool retracts along the product edge. Further, the part may also shrink over the tool and become stuck on.

So that the injection moulding tool can open, and that parts can eject a minimum 1.5^{o} to 3^{o} draft is required; however, the more that is possible the better. As the tool opens, this drafted surface separates with the tool cavity and allows air to rush into the tool, meaning that no vacuum is created.



Draft is further required when applying textures or adding embossed or debossed lettering or logos to a plastic component. Draft for textures varies depending on the texture, however a good rule of thumb is to add an additional 1.5^{o} of draft for every 0.025mm depth of texture.

The required depth and height of embossed or debossed text is often overestimated. Ideally, text should be 0.2mm to 0.6mm deep or high. Beyond this depth/height, text may have issues in moulding, creating thick sections and cosmetic defect problems.

Plastic Threads

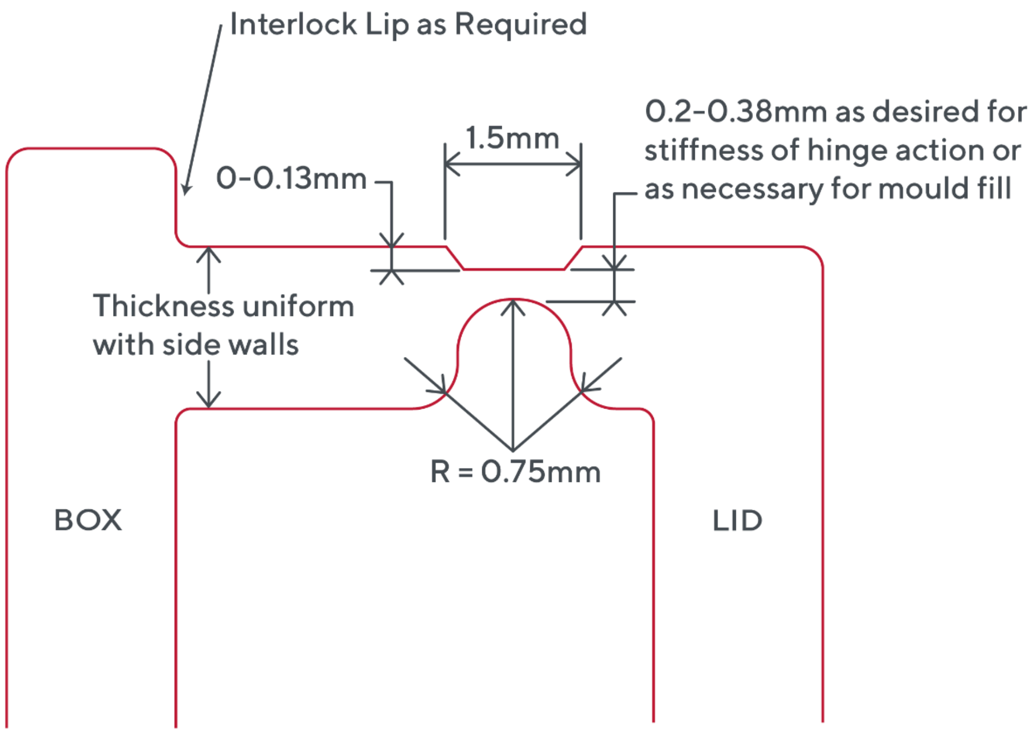
To plastic injection mould a complete internal thread, an unscrewing tool is required. This method of tooling is more difficult to construct and generally requires a longer cycle time to injection mould. Thus, a way to produce internal threads cheaper, is to manufacture them with an open/ shut tool. To do this, you can redesign the part to have a stagged and disjointed thread.

It is also worth noting, that you can't have a perfectly smooth external thread with an open and shut tool. This is because undercuts would be created along the centred parting line. You'll notice that external threads that are moulded with an open/close tool will always have a slight flat section at the parting line.

Living Hinges

When designing living hinges, material selection is as important as the physical design of the hinge. In order to be viable as a hinge, the material needs to be able to elastically deform repeatedly without failure. Polypropylene and polyethylene are the two best materials to use when injection moulding living hinges because of their high elasticity. At Dienamics, we have successfully mould nylon living hinges in the past, however this is not very common in industry.

Further, for a living hinge to work, it also needs to be designed correctly. Too thin and a hinge won't last and may tear. Too thick and the plastic may plastically deform and visibly 'whiten' or stress lines may appear. The diagram below illustrates how to successfully design polypropylene and polyethylene living hinges.

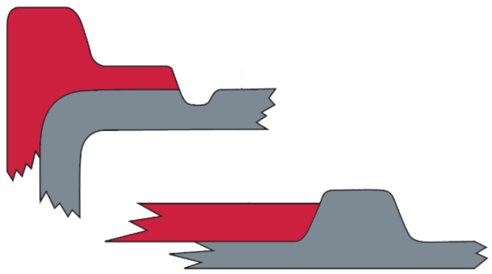


Overmoulding

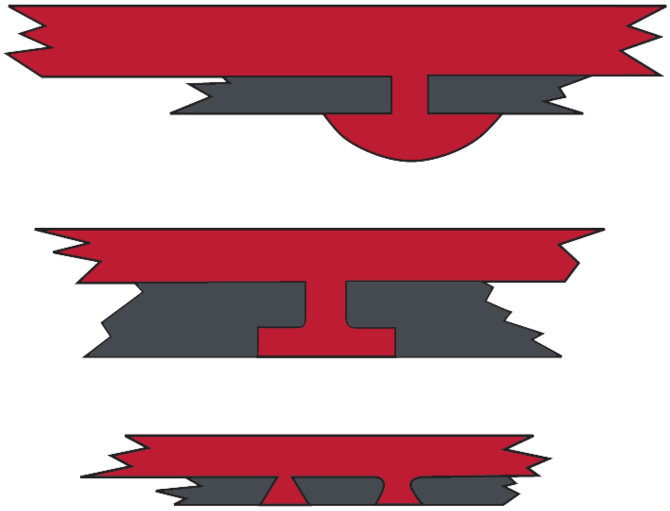
When over-moulding, material selection is vital. Before deciding to utilise over-moulds, you must ensure that the material that you wish to over-mould with, and the substrate material are compatible. Some plastics will not chemically bond together, and whilst it is sometimes possible to mechanically bond materials, often this creates gas venting and material flow issues when injection moulding.

Some basic design guidelines when over-moulding include:

- Keeping the substrate wall thickness uniform and over-moulding with a uniform thickness
- 1.5mm to 3mm thick over-moulding is generally ideal.
- The over-mould thickness should be less than or equal to the thickness of the substrate.
- Transitions between over-mould material and substrate material should be ‘trenched’ (as previously mentioned in Issue #2: Making Features from Faults), so that the over-mould tool can easily shut off against the substrate component.



To ensure a strong bond, on top of chemically bonding, mechanical interlocks can be used around the over-mould. The below image illustrates what some different types of mechanical interlocks look like.



As an experienced design, tooling, and injection moulding company, we understand that good product design isn't solely about reading the guidelines and following the handbook. It is about having the experience to intuitively make design decisions that can ensure a products' manufacturing. You don't want to run the risk of using a design firm that lacks manufacturing expertise, only to find out they've developing an unusable CAD model and that you have to start from scratch.

At Dienamics, we have been in the design and manufacturing industry for over 20 years, rest assured we have the experience and the intuition to bring your product dream into reality.