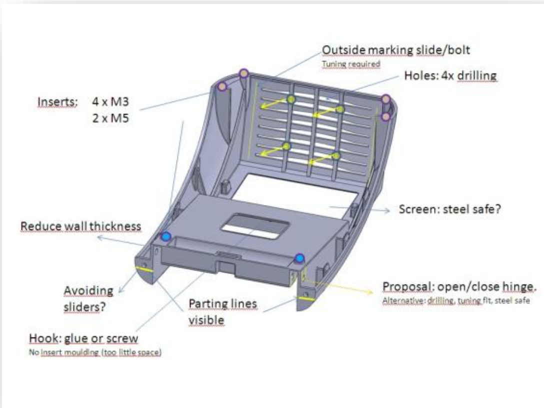
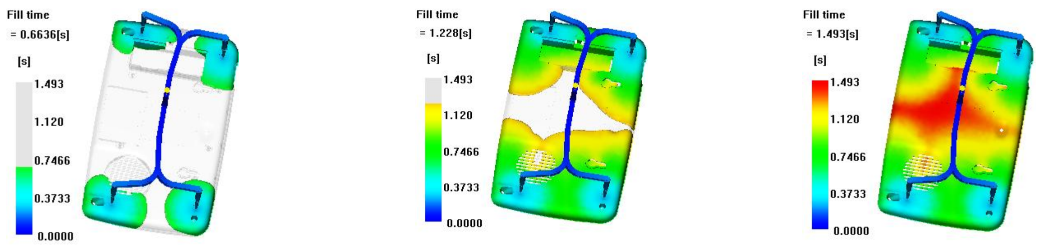
White Paper

Manufacturability: underestimated aspect of product design

A functional, beautiful and reproducible plastic product: that should be the outcome of a good collaboration between customer and supplier. In many instances, the customer – eventually in collaboration with the design studio - determines the function and design.

Although the manufacturability of the product would normally already have been taken into consideration, it mostly not has been given full attention yet.





Moldflow analysis. A tool used to determine, among other things, the position of the sprues.

Function and design are the first points of consideration when developing plastic injection moulded parts. This is logical, because those are the elements that determine the customer's user experience. And obviously the product is elaborated with a specific production technique in mind.

That notwithstanding, the sting is always in the tail. After all, if a hinge does not work optimally or the design fails to deliver the high-quality look intended due to flow lines or glossy spots, disappointment soon takes over. This invariably unleashes a series of rescue operations intended to achieve an acceptable end result. Such operations invariably incur unexpected costs, longer time to market and lower-quality products than previously intended.

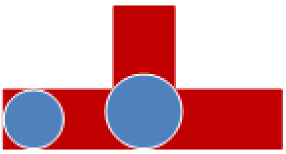
Function, design and manufacturability: three elements that can only yield an optimal product when applied together.

Tips for a better injection moulded product

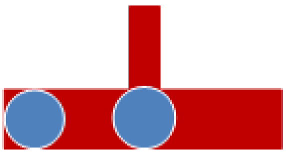
From makeable to touchable
What are the most important factors that determine a superior appearance in plastic injection moulded products? How does one prevent a component from merely being functional? How does one make sure the component will have a high-quality look?

Tolerances
Designers and developers are always tempted to work 'on the edge'. In other words, whenever the tolerance table indicates a tolerance level of + or - 1 mm, that tolerance will invariably be used. If possible, the tolerance levels will be reduced even more to create an even smarter product. This could force the supplier to create the product with a great deal of effort or excessive cost, especially in the case of relatively low annual volumes. The designer is therefore advised to create sufficient room to manoeuvre in the development phase: Will the design actually accommodate those tolerances? Is it possible to adjust it afterwards? Is it possible to create elongated slots instead of so many fits?

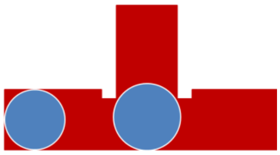
Wall thickness ratios
Although sink marks are a familiar phenomenon, it is not always taken into account in the development phase. Mostly this is caused by time pressure and unfamiliarity with effective solutions. In most cases, the cause is an erroneous choice of wall thickness ratios. The solution is invariably the golden 60% rule: the rib thickness should be a maximum of 60% of the wall (in the case of e.g. PP, it is only 50%). The trick of creating little ducts also helps to make the wall thinner.



Rib as thick as wall. The circles inside show the difference in the amount of material used.



Rib thickness approx. 50% of the wall. The material distribution is far better and yields vastly improved shrinkage distribution.



Rib as thick as wall, but now with reductions. A possible solution if a thinner wall is not possible.

Design guide

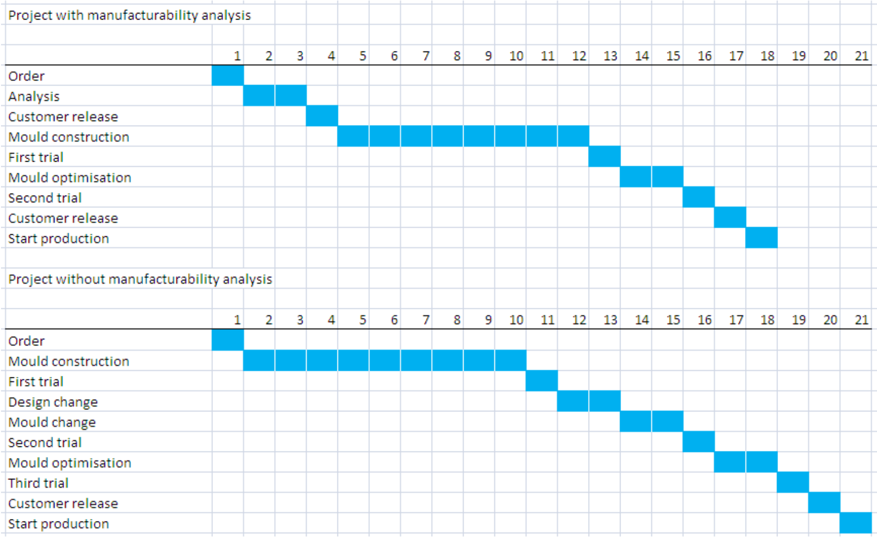
We have outlined the most important attention points concerning the manufacturability of plastic injection mould parts in our design guide. It also covers SFM, sprue choices and the norms that were used.

You can download this document [here](#).

Time to market

Fast start but still too late

The project must be realised in the shortest possible time… However, it is rare indeed for time not to become a discussion point and for the supplier to be given all the time in the world to deliver the required moulds. If the delivery time is eight to ten weeks, then the deadline will invariably be eight weeks, and preferably seven weeks, if possible. The customer expects the moulds to be produced immediately after placement of the order and delivery of the final files.

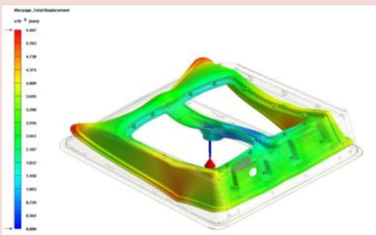


Practice has shown that this is hardly ever the quickest way to get things done. A thorough analysis of the files for manufacturability and expected problems invariably results in a faster delivery time in the long run. The diagrams above clearly show the difference.

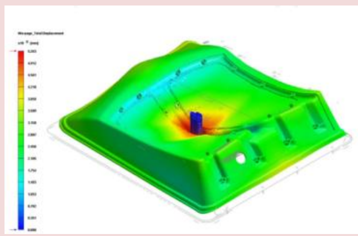
Of course, this example leaves plenty of room for discussion. It is very hard to prove in retrospect that manufacturability analyses always yield shorter delivery times. That notwithstanding, the first test injections very often yield much better products for most customers where detailed manufacturability analyses are conducted. In most cases, the time spent to produce the end product is subsequently significantly shorter.

Conducting a manufacturability analysis regularly helps to avoid crucial mistakes. It is quite common to discover deformation in this phase. This invariably leads to situations where one is forced to subsequently

Product amendments for an endoscope cleaner
In this specific instance, the CAD file of the lid of a cleaner was subjected to a warpage analysis. A series of iterations led to the conclusion that leaving out the holes and adding glass bead filler yielded the desired results. The holes were subsequently drilled in (which yielded the correct fit).



Initial design with holes and material without glass bead filler.



End result after various test iterations.

The exaggeration in the images is intended.

repair the product by hook or by crook. It also makes it possible to avoid mould-related problems, such as steel sections that are not properly aligned.

Deformation
It is possible to do elaborate analyses on CAD files to ensure the early discovery of deformation in a product and to assuage all the related concerns. The analyses could cover the influence of aspects such as chosen materials, sprue, wall thickness and mould cooling. It is then possible, following the analysis, to amend crucial details or material choices to ensure higher quality levels for initial products coming from the mould. An average moldflow or warpage analysis costs between € 1,500 and € 3,000. Although this might seem expensive, this kind of analysis usually helps to shorten the production time and also to lessen the likelihood of unexpected costs arising.

Moulding risks
Previously, the reproducibility was referred to. It often happens that we underestimate the consequences of choosing complex shapes on the required investment. The more complex the injection mould, the greater the likelihood of production problems arising. A few examples:


Detail of a mould with long, thin ejectors.

- Long thin cores (limited amount of steel, high breakage risk)
- Steel sections with limited incline (possible erosion)
- Moving parts, such as sliders
- Wall thickness of the product (the thinner, the greater the pressure build-up)
- Flow-off distances: the longer the distance between the sprue to the end product, the higher the pressure build-up
- The positions of the venting valves (in many cases it is possible to determine these through prior analysis)
- Temperature management in the mould (e.g. the expansion coefficient of the steel must be taken into account, problems could arise in areas with inadequate cooling)
- Different components in a single mould, which may reduce the investment, but can only be combined with great difficulty, e.g. due to major differences in weight, wall thickness and desired surface quality.

It is therefore advisable to consider the risks inherent in the moulds in the manufacturability analysis, especially as this helps to reduce the likelihood of damage, production interruptions and quality issues.

Conclusion

The function of the product, the design and the manufacturability are all inextricably bound together and all three aspects deserve our full attention. If we give due attention to manufacturability - in the correct phase of the project – it is possible to avoid quality issues, unexpected added expenses and unnecessary delays. Attention to manufacturability must focus on the details of product design, the injection moulding process and the design of the tools, such as the moulds.

Find more information on f.i. injection moulding techniques, designing plastic parts and buying the right mould, on the download page of our website.

About Pekago

Since 1983, Pekago Covering Technology has been active as a specialist providing development, mould construction, production, coating and assembly services for plastic housing elements and technical components used in industrial devices.

Over the past 40 years, Pekago has accumulated a great deal of experience in advising customers on manufacturability through our own engineering work. An example can be found in this business case on plastic Fresnel lenses for horticultural greenhouses.

More detailed information about Pekago is available on our website.