White paper

How to buy the right mould

If you require parts that are made using injection moulding or thermoplastic Structural Foam Moulding (SFM), then you will need moulds. How do you choose the right mould to buy? This is (often) a difficult question to answer. With this white paper we would like to give you a helping hand.

MOULD
A mould is essentially nothing more than a hollow container into which molten or hot liquid material is injected to create an object as it cools and hardens.

The mould contains a hollow space (cavity) which is filled with molten plastic under (high) pressure. The product then hardens, after which it can be 'cast' out of the mould. This process can be repeated to make products with the same desired shape and dimensions.

If you need a plastic injection moulding or SFM product, you will need a custom-made mould. That mould - and therefore that investment – is unavoidable. The first and most important question is: what requirements must the injection mould fulfil?

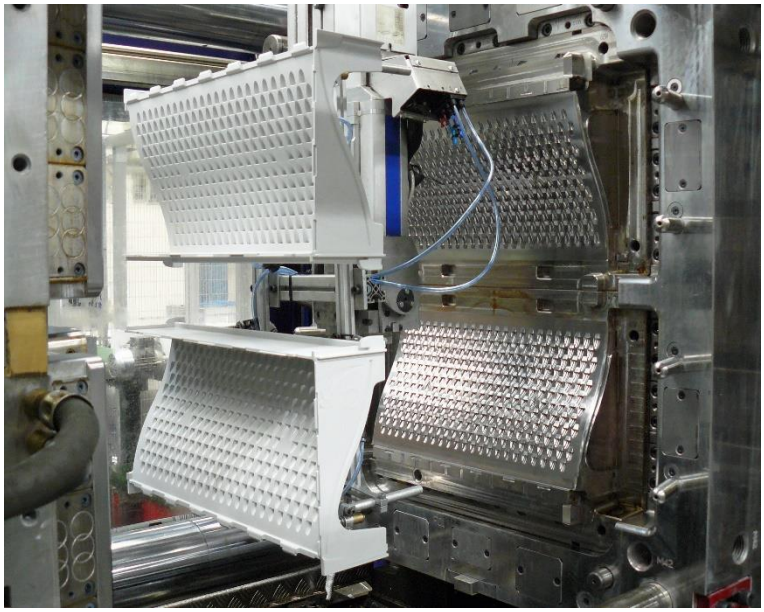
An engineer will have laid down the requirements and specifications of the product to be made in a document and/or 2D drawing (if all goes well). In addition, a 3D file of the product should be available. As a point of departure, this is an important first input for the mould buyer. The second and equally important input includes: numbers, annual volumes and the terms and conditions of purchase of the product to be made.

Requesting exactly the right injection mould from suppliers is a challenge. The prices quoted for the mould generally vary greatly, even though the suppliers will have based their offers on the same request.

How can you correctly assess the differences in the offers made? We will come back to this later in this white paper.

A single or multiple-cavity mould or a family mould

A *single cavity mould* has one cavity, which means that only one product can be made each time. A *multiple-cavity mould* contains several cavities. This allows one to make more of the same or different products at the same time. More cavities make the mould more complex to 'make' and so the mould costs increase. However, the price per product is lowered, because multiple products can be made at the same time.



A multiple-cavity mould on an injection-moulding machine.

Two – in this case almost identical – products can be injection moulded and then ejected at the same time.

If a mould can be used to make several different products, it is called a *family mould*. In theory, a family mould is cheaper than several separate moulds for individual parts.

Although family moulds are attractive in terms of costs, they are often more challenging from a technical point of view. If the various injection moulded parts that are to be made in the same mould differ greatly from each other (e.g. material type, weight, size and wall thickness), expert knowledge is required to make all the parts in the right quality. After all, it is important that the (liquid) plastic is injected into the cavity in precisely such a way that it is filled optimally at the right pressure. This may mean a different pressure or a different amount of plastic for different cavities. Another disadvantage of a family mould is that it produces the same amount of all the products, and perhaps a lot less of one of the

products that is actually needed. Although it is possible in theory to close one or more cavities off before injection moulding, this changes the pressure distribution in the mould, which can have a major adverse effect on the quality of the parts that are injection moulded. It also puts a greater load on the mould than was originally accounted for.

You don't make an injection mould from wood

Moulds can be made of different materials. The choice of material is - if properly calculated - the *sum* of the requirements stipulated for the injection moulding to be made and the number of products required. For example, a test mould for just a few samples can be made from silicone. Such a mould can be made quickly and (therefore) relatively cheaply, as it does not have to last long. An aluminium mould can be used for simple plastic parts, for which the total requirement is perhaps a thousand pieces. To make large quantities, for example hundreds of thousands of pieces of complex components, a hardened steel mould is usually required, because the mould would otherwise wear out before all products are made.

A practical example: an aluminium mould from which the product could not be removed easily.

This damaged the mould. During the injection moulding, the plastic not only flowed into the cavity, but also into the damage caused to the mould, which left a skin (residue of plastic) on the product.

It is up to the mould maker to calculate the right type of material in the right (surface) hardness. If the right mould is not chosen in a calculated manner at the outset, this may ultimately disrupt production and cause quality problems in the injection mouldings. Do not underestimate this! 'Buy cheaply, pay dearly' still occurs a lot.

The plastic must enter the mould cavity somewhere

SPRUE
'Sprues' are passages through which the molten plastic flows from the injection-moulding machine into the mould cavities.

One of the most difficult issues in mould construction is the choice of injection system. Basically, two types of systems can be applied: a hot runner system and a cold runner system. There are a number of factors to consider when choosing an

injection system. For example, the visual and mechanical requirements of the product, the lifespan of the mould and the costs of production.

The table below shows the most common injection systems:

	Injection system	Injection mark / position on product	Process (injection moulding / SFM)	Mould investment	Remark
	Cold runner				
1	Cone on the visible side of the product	Exterior Centre of product		+	Requires finishing / Removal of the sprue/flash (higher product price)
2	Cone not on the visible side of the product	Inside Centre of product		++ (Higher than 1)	More complex mould
3	Sub-gate on the visible side of the product	Side or in hole if present. Not in the centre of the product	Mould flow analysis optional	++	Product requires no finishing
4	Sub-gate not on the visible side of the product	Not in the centre of the product	Mould flow analysis optional	++	For example, on a rib or pin that can remain/stay on product.
5	Film sprue on the visible side of the product	Side or in hole if present. Not in the centre of the product	Mould flow analysis optional	++	Requires finishing / Removal of the sprue/flash (higher product price). Often necessary for the characteristics of the product
	Hot runner				
6	One opening on the visible side of the product	Exterior (often in the centre of the product)	Mould flow analysis optional	+	No finishing (lower product price). Injection point
7	Several openings on the visible side of the product	Exterior	Mould flow analysis needed	++	No finishing (lower product price). Multiple injection points
8	One opening not on the visible side of the product	Inside (often in the centre of the product)	Mould flow analysis optional	++	Injection and ejection pins on the same side.
9	Several openings not on the visible side of the product	Inside	Mould flow analysis needed	+++	Injection and ejection pins on the same side.

MOULD FLOW ANALYSIS
A mould flow analysis is a field in itself. It predicts how the hot, liquid plastic will flow into the mould and solidify there.

A white paper on mould flow analysis is also available on our website (in Dutch/German).

The position on the product where the injection will be made will depend on the product's aesthetic requirements, geometry, mechanical requirements and the material to be used. There are many possibilities. We recommend discussing the system of injection (type and position) at an early stage, so that a well-considered choice can be made before requesting an offer for a mould.

Note that a hot runner system is usually a lot more expensive than a cold injection system. On the other hand, a hot runner system often improves product quality and avoids a need for finishing (manual removal of the sprue/flash on the product). In the long run, a hot runner system may therefore prove more economical. It is clear that the choice of an injection system must be based on the requirements placed on the product. After all, you would not build a house from paper because it is cheap.

SFM or TFC?
SFM (Structural Foam Moulding) and TFC (Thermoplastic Foam Casting) are different names for one and the same production technique.

HOT or COLD?
The cold runner injection system always leaves a small 'pin' of residual plastic on the product.

If one must inject at a visible point on the product for technical reasons, this can be concealed after the injection moulding process by, for example, milling and/or painting over it. Of course, injecting at a point on the product that is not visible is better. In that case, the 'pin' can be mechanically or manually broken off immediately after the injection moulding process. Or you can opt for a hot runner system!

Something is moving in the mould

After the plastic has solidified in the mould, the injection moulding part must be removed from the mould. We often call this 'ejection': parts built into the mould push the product out of the cavity. Sometimes the product can simply fall onto a conveyor belt. Sometimes a robot or employee picks up the product and puts it away or packs it immediately.

In many cases, the product is moved in one direction, directly from the mould. The simplest mould construction is therefore an open-close construction, in which the product is removed from the cavity in one direction.

Injection moulded products that require holes or protrusions need a more complex mould. During the 'deformation' (removal from the mould) of this type of product, 'sliders' are first moved to release the difficult-to-remove parts of the product. Only then will the whole mould open. Sliders can be moved in an injection mould in two ways:

1. By mechanical displacement,
2. By using cylinder(s).

Both systems have advantages and disadvantages. However, if sliders or cylinders can be avoided, then avoid them! The more moving parts there are in an injection mould, the more there is to break. So if a slider is not positioned correctly during production when the mould opens, this can result in a very unpleasant situation – it's like locking your bike while cycling! To prevent this from happening, mould protection is certainly not a superfluous luxury.

To avoid sliders, we recommend intelligently changing the shape of the injection mould product (if possible). Or you may opt for finishing: drilling a hole afterwards. This will reduce your mould costs (although this does place higher demand on labour).

What do I get for my money?

As mentioned before, offers for a mould may vary enormously from each other. In order to be able to compare the offers made properly, the following points should be considered:

Support before, during and after mould construction

Moulds are complex and often expensive tools that one must be able to reliably use to produce injection moulded parts over a long period of time. They must satisfy a wide range of specifications. Support is valuable throughout the entire process, from product design, the resulting mould design, mould construction, the test phase and actually taking delivery of the mould from the mould maker. The following always applies: trust is good, control is better! What kind of support does the mould maker offer you in his offer?

Does it work?

And all at once that block of steel is sitting right in front of you. It doesn't tell you anything, and nor is it immediately clear whether the mould meets your stipulated requirements. Have sample release products also been included to demonstrate good product quality? These are all questions that should be answered beforehand. Here, too, the following often applies: cheap is...

Perfect job on the first try. Or not.

That would be nice: that the mould immediately makes beautiful products. However, it seldom happens that everything immediately turns out perfect. Nevertheless, mistakes can be avoided and an almost-perfect mould, as well as an almost-perfect product, can be realised by preparing the mould well and by being very critical of the product design.

However, changes and optimisations are generally unavoidable. Who bears what responsibility? Mould optimisation may be necessary if the products made do not meet the requirements, because the mould does not meet the requirements. This is the combined responsibility of the injection moulder and the mould manufacturer. Think of a part that does not release properly, film on the product or injection geometry that needs to be adjusted.

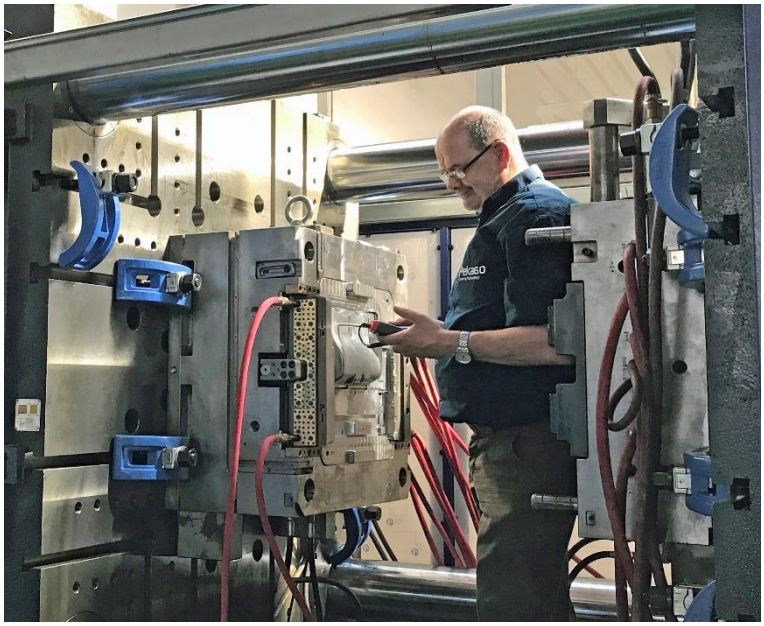
Discuss the product design with the injection moulder and mould manufacturer at an early stage. A warned person counts for two: if the design has not been optimised for injection moulding, then the mould will not work well either. You can find more information about the design of plastic injection moulded parts in our Design Guide.

Does it appear that the product design is not realisable after all, or is a decision taken to change the product design? If the geometry of the product is changed (and this usually

means a new 3D file), then this is a responsibility of the customer. In that case, the injection moulder or mould manufacturer will draft a new offer for the changes to be made to the mould; possibly including the costs for a new test injection, validation, measurement report, etc.

Test production included or excluded

A test injection must be performed to assess a mould and the product that comes out of that mould. The functioning of the mould is tested during a test injection and the machine and process settings are determined if the result is good. The mould can be finished after this test and the settings can be used when the mould goes into serial production. That the test production (and the conditions that are attached to it) forms a part of an order would be useful. Otherwise, unexpected costs relating to the test production may be incurred.



A test production can take place at the mould builder. Here Wil Mathijssen, Process Engineer, is testing a new mould at one of our partners.

At Pekago, a new test injection is performed on the machine on which the mould will run.

Are drawings included?

To make a mould, a construction drawing (in 3D) is generally drafted. If the mould is accidently damaged, exactly the same mould can be made again on the basis of the construction drawing. A mould construction drawing is also required when transferring from one injection moulder to another. The drawing also ensures that repairs and changes can be made much more easily and quickly.

Am I the owner?

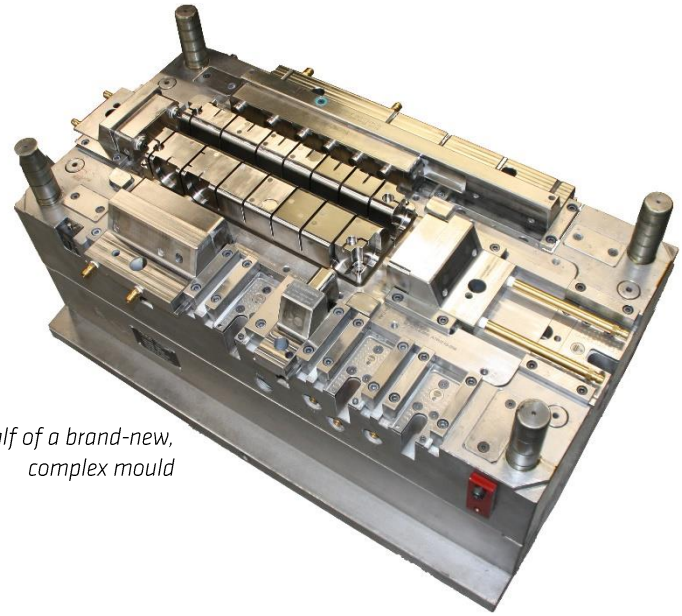
Yes, of course. At least, one would hope so. Is the mould entirely your property after paying for it? Or are you 'just' paying part of the investment and will the mould remain the property of the manufacturer or injection moulder who commissioned the construction? Get this put down in writing! The mould is a tool over which you should preferably have full ownership. Unless, of course, you have made other agreements, such as including the mould costs in the production costs. In any case, it is important to know what has been agreed concerning the ownership of the mould.

I want to go to another mould injector

How does that work if you decide to transfer your products to another injection moulding company, for example because you are not satisfied with your old supplier or because your old supplier is no longer able to deliver? If you are the full owner of the mould, you can transfer the mould to another injection moulding company. However, there remains the question as to whether the mould will be suitable for use by another supplier. It may be that the mould was not 'universally' built and that adjustments need to be made.

Valuable tool

A mould is a valuable tool. If this 'block of steel' is well prepared, well constructed and suitable for the job, then you will be happy with the result for years to come. And vice versa... so get your feet wet.



Half of a brand-new, complex mould

Pekago Covering Technology is an injection moulding company that makes plastic housings and technical components for its customers using injection moulding and SFM. We also offer painting, assembly and printing services. We develop our own injection moulds and advise our customers on the manufacturability of their plastic products. Our markets include medical devices, aircraft interiors, analytical devices and imaging systems.

More information about Pekago can be found on our website: www.pekago.com