

# Design guide for injection moulded and SFM parts



This document describes the standard guidelines for developing parts suitable for plastic injection and structural foam moulding (SFM). Purpose of this document is to support engineers in construction parts with focus on:

- optimal visual appearance of the part;
- optimal dimensional stability and processing conditions;
- optimal mould quality and durability.

## Dimensions

- Design the parts nominal (in the middle of the tolerance field).
- Tolerances used by Pekago are conform:
  - DIN 16742 - TG5 (Standard Injection Moulding) and
  - DIN 16742 - TG7 (Structure Foam Moulding).
  - Finishing minimal +/-0.4 mm.
  - Depending on design, complexity and material. Crystalline materials tend to shrink and warp more, increasing the tolerance field.
- With "steel off", details where accurate fitting is required can be optimized. Define "steel off" before mould construction and design it too "small". Note: adding plastic means removing steel in the mould.
- With "steel off", one iteration can be made in the mould (after checking the FOT products). "Steel off" positions must be defined on the 2D drawings.

## Texture

The table below shows draft angles required when using spark eroding texture according to VDI 3400. Other angles may apply with other textures.

Draft angles needed for EDM (spark eroding) texture according to VDI 3400								
Material	0.5°	1.0°	1.5°	2.0°	2.5°	3.0°	4.0°	5.0°
HDPE, PP	VDI 21	VDI 24	VDI 27	VDI 30	VDI 33	VDI 36	VDI 39	VDI 42
ABS, PC, PMMA, PS		VDI 21	VDI 24	VDI 27	VDI 30	VDI 33	VDI 36	VDI 39
PA, POM	VDI 24	VDI 27	VDI 30	VDI 33	VDI 36	VDI 39	VDI 42	


Textured or polished faces must be defined on 2D drawings by line markings such as  or by using colours.



Image of an injection moulded part, indicating different textures

- **Brown** = Product faces with an EDM or etched texture.
- **Green** = Technical smooth product faces.
- **Grey** = Polished product faces.

### Draft

Cavity	Front side	Edges and sides	2-3°
	Front side	Holes and pockets	5°
Core	Rear side		0.5-2°
Sliders			5°
Pinch faces, parting lines	Depending on texture requirements and part design		7°
	Recommended on aluminium mould		10°
Radii			R0.3 or better

Above figures depend on design and material. Please consult if needed.

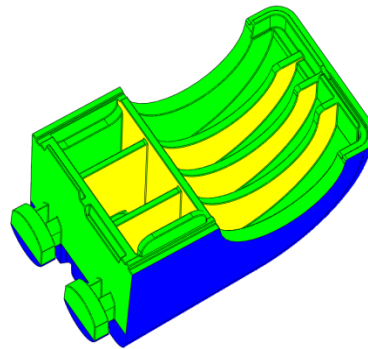
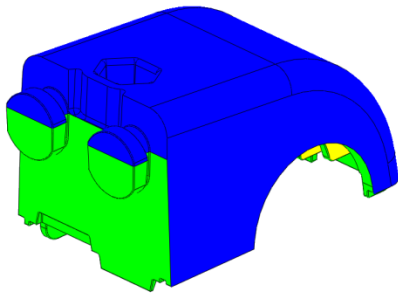


Image of a draft analysis

### Burr / parting faces

- Parting faces in the mould must be constructed solid and durable (minimal 2 mm width and 5 - 10° draft).
- On small cores or parting faces small markings, splitlines and a slight degree of burr can be expected. Approximately 0.25mm.
- A splitline which is straight or with smooth steps gives a better result than a splitline with sharp steps.
- No radii on a splitline.

## Cores

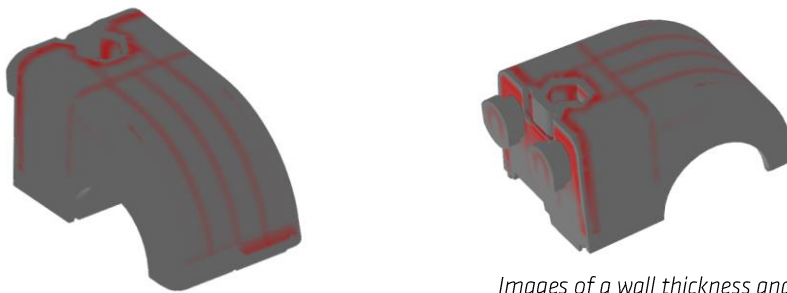
Thin and sharp mould parts must be avoided. These parts are weak and cooled badly. Gloss difference and tension may appear. A minimal width of 30% of the height is a general rule for these cores. Avoid sharp corners and thin steel in the mould, especial at pinch faces.

## Sink marks

In general, for a plastic part: the thickness of a rib must not be more than 40% (crystalline plastic) or 60% (amorphous plastic) of the nominal wall thickness. Avoid thick wall sections:

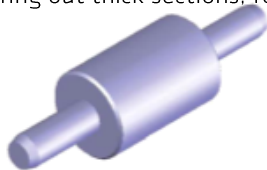


Through wall thickness analyses, Pekago can indicate where sink marks can be expected:



Images of a wall thickness analysis. The red lines indicate thick sections which could cause sink marks.

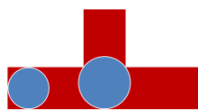
Wall thickness ratio can be improved by coring out thick sections, for example:



Original geometry



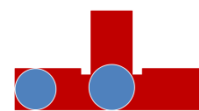
Cored geometry



Compact injection moulding: a rib of the same thickness as the main wall may cause sink marks.



Reducing the rib thickness reduces the amount of material below the rib, causing less absolute shrinkage.



Designing specific details can also help to reduce shrinkage when a certain rib thickness is required.

Recommended wall thickness ratios:

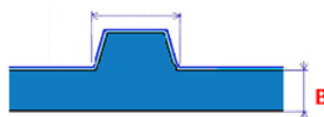
Compact: max. 60% of A

Max 60% of A



SFM up to 120% of B

TSG up to 120% of B



Wall thickness (mm)	Rib thickness (mm)
Compact	
2	1.2
3	1.8
4	2.4
SFM	
5	3
6	4
7	5
8	8
10	12

## Ejector pins / joggle pins

Pins give markings of approximate 0.15 mm in or on product.



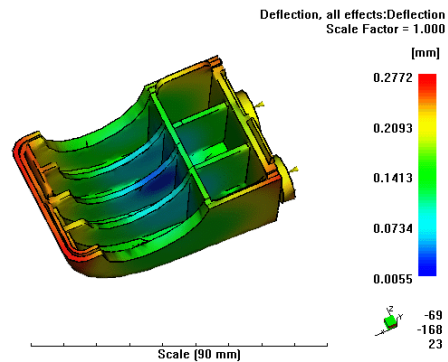
*Picture of an ejector pin mark*



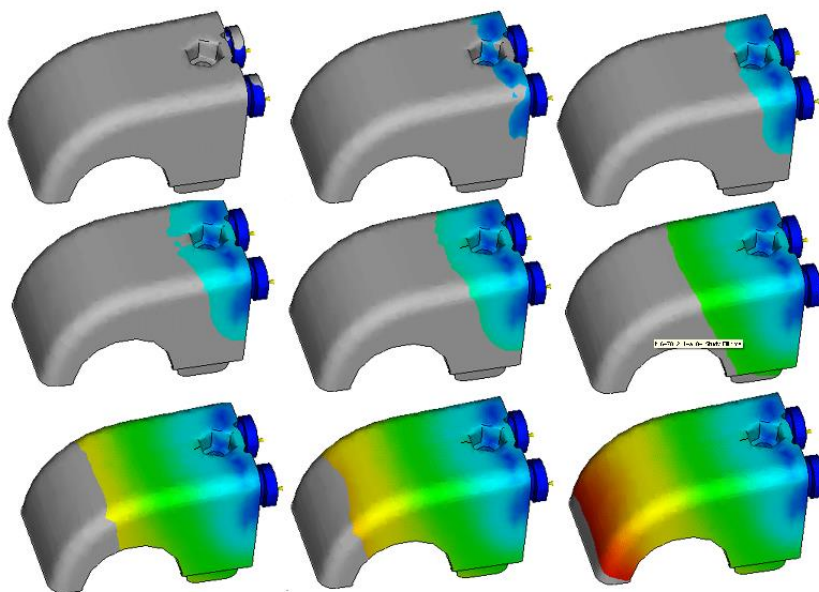
*Picture of a sleeve-ejector mark*

## Filling

- Avoid thin walls in a plastic product (thinner than 0.8mm).
- Standalone features: where possible connect with the wall or construct them in such a way that they can be made with a separate mould part (e.g. a pin-bush ejector). This is necessary for proper venting and ejection of the product.
- At the start of mould construction (in China) a brief filling analysis and warpage indication of the product will be made. Optionally, more comprehensive analysis can be preformed.



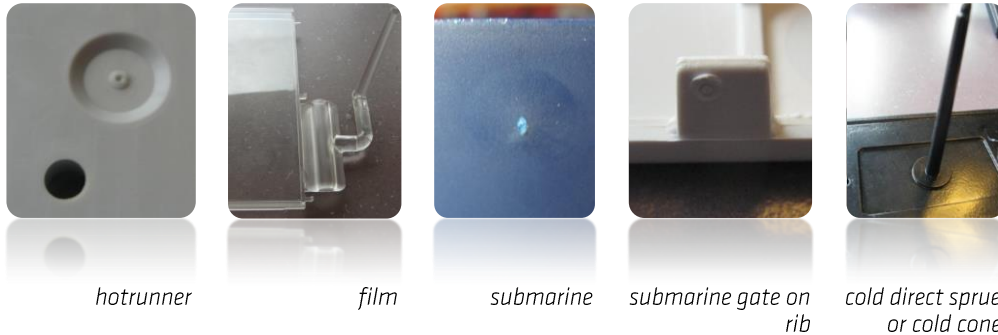
*Image of a warpage analysis*



*Image of a filling analysis*

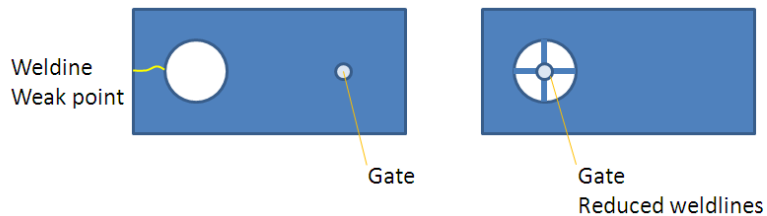
## Sprue (injection type and position)

- For each product Pekago can point out the injection position.
- Depending on the injection type, each injection point gives a mark on the product of approximately 0.4 mm.
- Injector types:



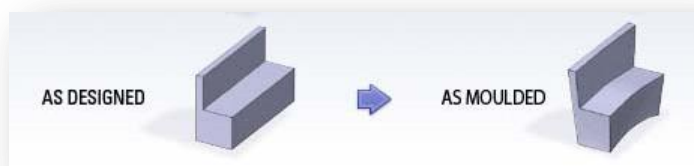
## Weldlines

Weldlines appear around holes and gaps. They follow the filling direction, starting from the injection position.



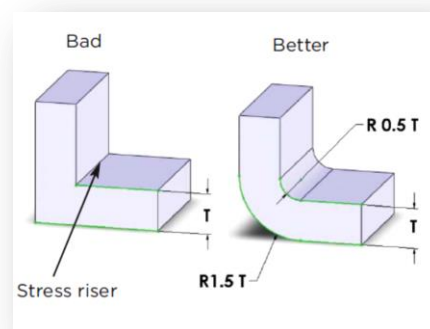
## Warpage

Depending on material tensions, wall thickness variations, fibre orientation, injection location and process cooling, parts may deform after production. This can be simulated by moldflow and warpage analysis.



## Fillets

In order to make the part stronger, avoid stress cracking, make the mould easier to produce (milling) and ease the plastic filling, please round the sharp inside and outside corners, wall thickness transitions and ribs:



## Information required on 2D drawings

- Dimensions which need to be checked on (FAI) measure report
- Tolerances - steel off
- Texture or polish positions
- Paint or coating positions
- Injection positions
- Position date clocks, article number, material indication (e.g. >ABS<) and other texts

## Appendixes (on request)

- Kerb Konus Soniclock inserts
- EJOT screws
- Design guide DSM – SABIC – BASF
- Tolerance tables
- Material datasheets

## About Pekago



Since 1983, Pekago Covering Technology is a specialist in mould construction and the development, production, coating and assembly of plastic housings and technical components for the manufacture of industrial equipment.

Pekago puts a lot of effort in optimizing the product design before mould making commences. An engineering department with sufficient capacity in project management, CAD (SolidWorks), a tool shop and long time experience in injection moulding and structural foam moulding is present.



Vital information, such as on visual appearance, tolerances, shot guarantees and risk management is decided in close cooperation with the customer prior to mould construction. This ensures that the product meets customer expectations, avoids disappointments and reduces the chance of unexpected costs.

More information about Pekago can be found on our website: [www.pekago.com](http://www.pekago.com)