emould@work

Injection mould tool design checklist

Product design decisions with the biggest impact on injection mould tool cost and lead time.

As designers of injection moulded products you are probably well aware of design best practice. What you may not appreciate is the important contribution a well-designed mould tool can make to the new product development process.

The wrong decisions taken at the critical early stage of a project will add cost and time to mould tool design, manufacture and maintenance, put pressure on product launch dates, and, of course, adversely affect product manufacture and in-use performance.

Material selection

Have you specified a material *and* grade *before* the mould tool is designed?

✔ **YES.** Well done! Go to wall thickness.

✘ **NO.** You are in trouble already!

Wall thickness

Walls cool at different rates depending on thickness; have you taken this into account when designing your part, particularly at the junctions between thick and thin sections?

✔ **YES.** Congratulations! Go to draft angles.

✘ **NO.** Your part will suffer from sink and warpage unless you have designed-in ribs.

Draft angles

Have you added one degree draft on all *non-textured* walls, ribs, bosses and any other feature created in the line of tool opening?

✔ **YES.** Good job! Follow the link to the next question about draft angles.

✘ **NO.** The tool will need special parts, adding cost.

Have you added at least three degrees to all shut out faces (places where tool surfaces touch), for example, part openings?

✔ **YES.** Keep up the good work! Go to the final question about draft angles.

✘ **NO.** Tool wear will increase significantly, driving up maintenance costs and reducing tool life.

If your part is textured, have you increased draft angles accordingly?

✔ **YES.** Give yourself a pat on the back! Go to ribs.

✘ **NO.** It will be difficult to remove the part from the mould.

Ribs

Are the ribs that you have specified for strength - to avoid warpage and/or to add functionality - no more than 50% of the thickness of the walls with which they mate?

✔ **YES.** How good is that! One more question on ribs.

✘ **NO.** Your part will suffer from sink and warpage.

Are the ribs you have designed very thin or deep?

✔ **NO.** Smashing! Go to bosses.

✘ **YES.** This will add time and cost to tool production.

Injection mould tool design checklist

emould@work

Bosses

If you need bosses for screw fixings, have you specified a one degree draft angle and is the section no more than 50% of the thickness of the wall with which it mates?

✔ **YES.** Way to go! Go to clips.

✘ **NO.** The tool will need special parts, adding cost, and your part will suffer from sink.

Clips

If clips are more appropriate than screw fixings, have you considered the cost implications of moving cores and have you specified a draft angle of at least three degrees on each side of the clip?

✔ **YES.** Super! Go to internal undercuts.

✘ **NO.** Tool maintenance will increase and tool life decrease.

Internal undercuts

If internal undercuts are required for fixing and location features, or because part geometry is the same in both sides of the mould tool, have you confirmed that the cores will move away from the undercut as the part is ejected?

✔ **YES.** Magic! Go to external undercuts.

✘ **NO.** The tool will not operate.

External undercuts

To produce the external undercuts required for holes and openings, or dictated by geometry of the total part, have you confirmed that the cores will operate as the tool opens?

✔ **YES.** Cool! Go to split/parting lines.

✘ **NO.** The part will get trapped.

Split/parting lines

Have you considered where the tool should be split so that it:

1. Follows a smooth path?

2. Can vent gases created during moulding?

3. Avoids cosmetic and functional faces?

✔ **YES.** Have three pats on the back! Go to part ejection.

✘ **NO.**

1. Tool life will be reduced significantly and additional, less efficient cutting operations will add time and cost
2. You risk burning the part and trapping gas
3. Appearance will be compromised and functional faces will have witness lines

Injection mould tool design checklist

emould@work

Part ejection

The selection of an ejector type depends on part geometry and features such as bosses; have you considered how your part will be ejected from the mould?

✔ **YES.** Brilliant! Go to feeding.

✘ **NO.** Insufficient ejection risks the part distorting or suffering from stress marking.

Feeding

To ensure optimum filling, have you specified the location and type of feed gate *before* the mould tool is designed?

✔ **YES.** One more to go! Go to cooling.

✘ **NO.** Parts of the mould will not be sufficiently filled, causing sink and warpage.

Cooling

To ensure optimum processing of the material, mould temperature must be correctly controlled; have you considered how the mould can be heated and cooled?

✔ **YES.** You've done it!

✘ **NO.** 1. The mould will not fill correctly
2. Material properties will be affected
3. Part dimensions will not be correct
4. The part will be distorted

Sink

Sink marks are caused when a wall section is either too thick or there is a sudden change in section, typically where ribs and bosses are placed internally or where the extremity of the part is the same section or larger than the area where the feed is occurring. Cosmetic appearance is affected and dimensional accuracy will be lost. In extreme cases there will be a void inside the part potentially leading to mechanical failure.

Some materials are more susceptible to sink than others with polypropylene and polyamide the worst culprits.

Warpage

Warpage like sink is caused when the section is either too thick or there is a sudden change in section and there is insufficient strength in the part to hold the required shape. Appearance and dimensional accuracy are affected.

All materials will warp but polypropylene and polyamide perform particularly badly.