

# Design Guide - Injection Molding

**Machine Size:** 80 to 3000 ton

## Advantages

Low cost per part  
Repeatable  
Large selection of materials  
Good surface finish

## Drawbacks

Requires investment in tooling  
Longer lead time to first parts

## Tips & Tricks

Remove undercuts if possible  
Specify a lower grade surface finish  
Mold in standard plastics – pp, abs etc  
Reduce weight of part

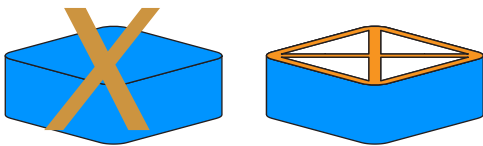
## Surface Finishes

Textured in tool  
Coloured in tool  
Painted or Plated  
Silk/pad printed

## Materials

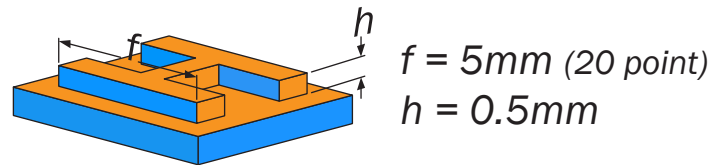
Too many to list. Talk to PoMo to find out material options for your project.

**Rapid Injection Molding** - is a low to medium volume production process. First step, make a mold tool. Then install in injection mold press and inject molten plastic into the mold tool under pressure. Cool, eject part, and repeat.

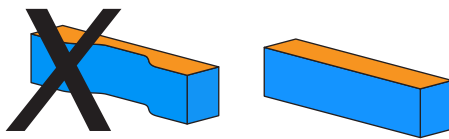


**Hollow Out** - any thick sections on the part should be hollowed out, ribs can be used for strength.

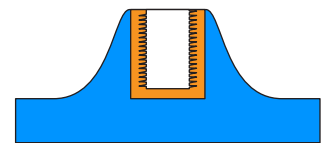
**Text & Logos** - embossed text on the part is preferable as this can be machined into the mold tool.



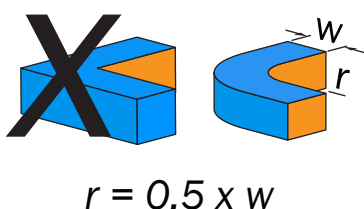
**Wall Thickness** - use uniform wall thicknesses. Walls 1-3mm are recommended for most materials.



**Threads** - threaded inserts can be over molded into the part.



**Corners** - round all corners if possible. A radius of 0.5x wall thickness is advised.



**Draft** - draft angles should be added to all parts in the direction of draw. Draft prevents surface scratches and damage when demolding & prolongs life of tool. Two degrees' draft + is recommended.

