Plastic Injection Molded Part Design Guidelines

Plastic design principles easily fill entire books. I have attempted to reduce these principles to these most basic guidelines. As guidelines, there is room for them to be stretched or even broken; indeed, I have broken every single one of them in actual production parts. Holding to the guidelines, however, can minimize costly surprises.

1. **Plastic wall thickness should be 1-3mm nominal.** Thinner walls can have filling problems, and thicker walls can have cooling problems that lead to long cycle times and part warpage.

2. **Rib thickness should be 66% or less of the nominal wall thickness. 50% or less is even better.** When a rib is as thick as or thicker than the wall it attaches to, there will likely be a surface imperfection called "sink" on the outside. It can also cause the part to warp.

3. **Avoid undercuts if possible.** Undercuts are common and can be accommodated by an additional action in the tool with a "slide" or a "lifter". However, the tool will be less expensive if you avoid undercuts.

4. **All vertical features should have draft to allow the part to slip out of the mold.**

5. **Draft angle should be 3° or greater if possible.** Cosmetic surfaces are often textured. Textured surfaces require greater draft than glossy surfaces.

6. **Draft on screw bosses, internal ribs, or glossy surfaces may be .5° to 1°.**

7. **Bosses should have support ribs.** Ribs help the material flow during the injection process and also strengthen the boss.

8. **There should be no solid sections of plastic that are thicker than the nominal wall.** Again, this would cause sink and warpage.

9. **No sections should be less than .75 mm in thickness.** Thin sections run the risk of not filling properly and are often manifested as undesirable sharp edges.