Engineering Design I: Methods and Skills
Topic Readings

Roberto Jaime and Steven H. Collins

December 2, 2015

Chapter 10

Design for Manufacture

As we learned in the previous chapter, there are many processes available to economically manufacture large numbers of components. A good manufacturing process can be selected using charts, for example based on quantity, material selection, based on simplified models of loading and geometry, and cost comparisons, using an approximation of component geometry as a basis for calculations. Once you have selected a process, you can refine the details of your design to make it better suited to the chosen means of manufacture. First, your design will have to conform to the constraints of the process. For example, injection molded parts cannot have features that that would prevent mold separation. Small changes to your design can also dramatically improve the cost and quality of the parts produced. For example, tapering cast features allows flow of molten metal during cooling to prevent voids. The process of optimizing a design in this way is called *design for manufacture*. In this reading we will discuss design for two common manufacturing methods, injection molding and investment casting.

10.1 Injection Molding

10.1.1 Applications

Injection molding is often a good choice for manufacturing large quantities of plastic parts. The initial costs of this operation are large but they allow a significantly lower variable costs once volume is above about 10,000 parts. The high initial costs are incurred due to tooling (mold production). In recent years the introduction of high-grade aluminum alloys has allowed for lower tooling costs. The introduction of these alloys allows smaller batches to be produced economically. Aluminum molds are easier to machine and therefore cost significantly less to produce, but also have a shorter life span restricting viable quantities.

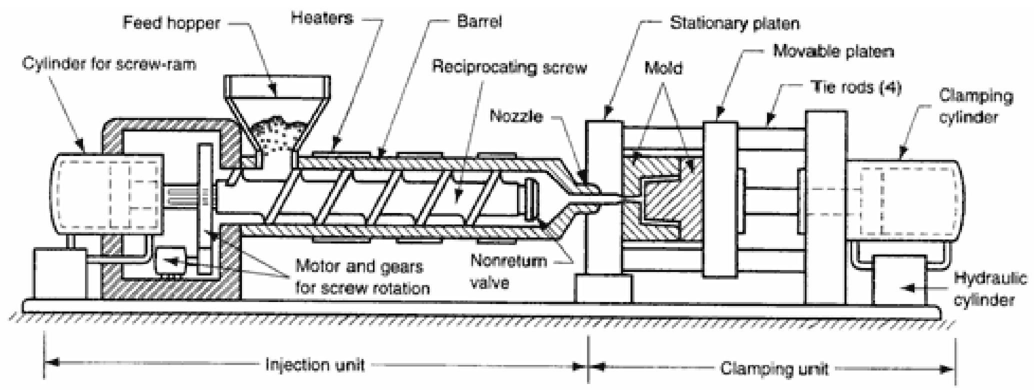

Figure 10.1: Injection Molding Setup (Xcentric Molds and Engineering).

10.1.2 Process

Injection Molding is a process by which molten plastic is injected into a mold cavity using machinery specializing in pressurizing molten plastics. The set up requires an injection press which consists of an injection unit and a clamping unit. These can be seen in Figure 10.1 with key components identified.

The injection process begins with the insertion of a granular plastic into a hopper. This media is then pressurized by an injection ram or a screw type plunger and heated by a set of bands located along the length of the plunger (or ram). As the plastic melts it is forced through a nozzle that further increases the pressure. This nozzle leads to the mold and molding cavity. The mold is kept at a constant temperature to ensure the plastic can travel far enough into the mold without solidifying, yet solidifies quickly once the cavity is filled. If the temperature of the cavity is too high the mold begins to see thermal stress and fatigue, lowering the lifespan of the mold. After the mold is filled, a constant pressure is maintained at the nozzle to compensate for shrinkage as the plastic solidifies. Once the plastic is set, the mold is opened and the part is ejected. The ejected part will have a sprue and runners (Figure 10.2) which must be removed.

10.1.3 Materials

Many polymers can be injection molded. An important property to consider is whether the material is amorphous or crystalline. Amorphous materials tend to contract less upon cooling, allowing tighter tolerances to be maintained, but have higher viscosity, requiring higher pressure during injection and making it difficult to fill long thin mold cavities (Figure 10.3).

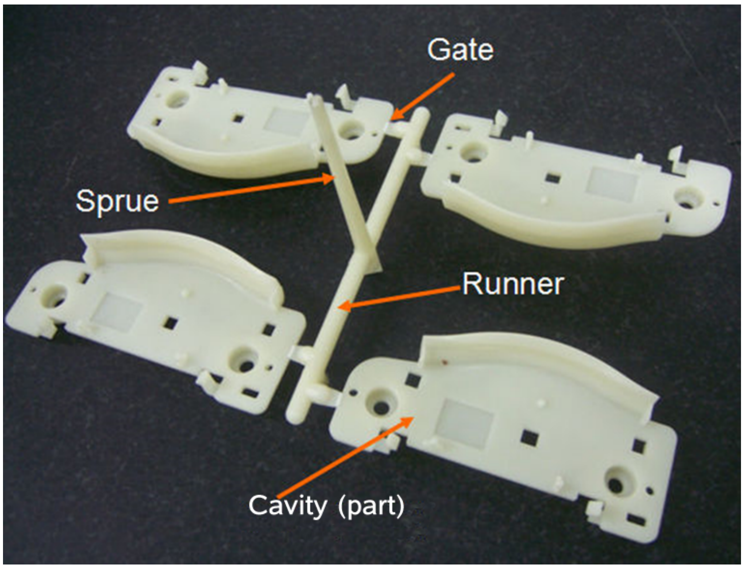

Figure 10.2: Injection molded parts just after ejection from the mold.

	Amorphous Materials Acrylic ABS Polystyrene PVC Polycarbonate	Crystalline Materials Nylon Polypropylene Acteal Polyester Polyethylene
Comparison Characteristics:		
Shrinkage	.004 - .012/in/in	.012 - .025/in/in
Ease of flow	relatively stiff flowing	easy above melting
Dimensional control of parts	easier to maintain close dimensional tolerances	temperature more difficult to maintain close dimensional control

Figure 10.3: Some materials used for injection molding and certain properties.

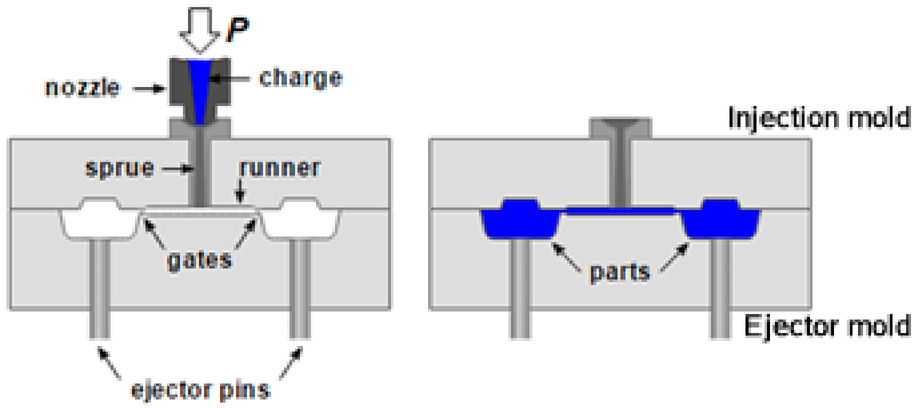

Figure 10.4: Injection and ejector molds and components.

10.1.4 Mold Function

Molds for injection molding primarily consist of two parts, the injection mold and the ejector mold (Figure 10.4). The injection nozzle connects to the injection mold though a channel called a sprue. The sprue then then connects to channels in the mold called runners that feed into the mold cavity. The runners lead to constrictions called gates, which lead into the cavity. Gates allow easier separation of the part from the runner following ejection. Air in the cavities is expelled through vents in the parting line of the mold, which are too thin for plastic to enter. Failure to vent air can result in defects such as burning or voids.

Molds are designed such that the injection-molded parts remain attached to the ejector mold after mold separation. This attachment allows the runners and sprue to be drawn out of the injection mold leaving it clean to begin the process again. Once the part is secured on the ejector mold, ejector pins push the part out of the mold and it is ready to start the process again.

The injection, cooling and ejection processes, and the mold hardware used to perform them, tend to make some component properties desirable. In the next few subsections, we will discuss several heuristics for improving part quality and cost.

10.1.5 Draft Angle

Surfaces of an injection-molded component that are parallel to the direction of motion of the mold are typically angled slightly to aid in the release of the part. This angle is called the draft angle. The degree of the draft depends on the depth

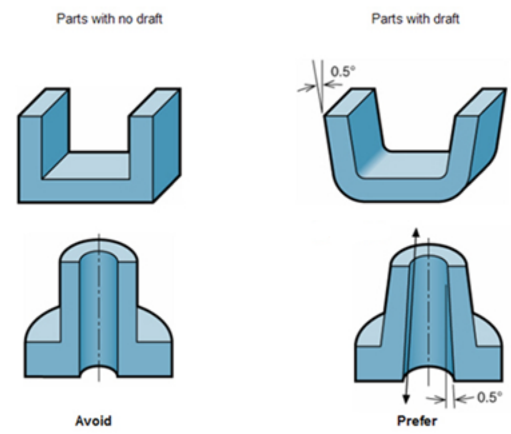

Figure 10.5: Draft angle examples (Not to scale; Mapeng).

of the cavity and the amount of shrinkage of the part. For most applications a draft angle of 1–2° is effective. Figure 10.5 presents some simple examples of draft angles. Designs that lack draft angles can be damaged upon release from the mold due to excessive friction. Parts lacking drafted features also have a higher probability of becoming stuck in the ejector mold, causing delays to clean and possibly repair the mold. Internal surfaces without a draft tend to be more of a problem, because the plastic tends to contract as it cools. Contraction tends to pull the part away from outer walls of the cavity and to bind it on inner surfaces. Additional mold features can aid with ejection of parts that require zero draft angles, but at increased cost.

10.1.6 Minimum Wall Thickness

Wall thickness should be minimized for injection molded parts, within structural and mold filling constraints. Reducing wall thickness reduces cooling time, which in turn reduces time for molding cycles and allows for a higher production rate and lower part cost. Thinner parts also require less plastic, further reducing variable costs. Wall thickness of an injection molded part can vary from about 0.020 inches to 1 inch, with the viable range depending on material (Figure 10.9).

10.1.7 Uniform Wall Thickness

Ideally, injection molded parts have uniform wall thickness. If this is impractical, transitions in thickness should be as gradual as possible. Maintaining uniform thickness avoids issues related to uneven cooling. Thicker sections take longer to cool, which can lead to defects such as sink marks or warping (Figure 10.7).

Resin	Inches
ABS	0.045 - 0.140
Acetal	0.030 - 0.120
Acrylic	0.025 - 0.500
Liquid crystal polymer	0.030 - 0.120
Long-fiber reinforced plastics	0.075 - 1.000
Nylon	0.030 - 0.115
Polycarbonate	0.040 - 0.150
Polyester	0.025 - 0.125
Polyethylene	0.030 - 0.200
Polyphenylene sulfide	0.020 - 0.180
Polypropylene	0.025 - 0.150
Polystyrene	0.035 - 0.150
Polyurethane	0.080 - 0.750

Figure 10.6: Recommended wall thickness ranges for different materials (Protomold). Thickness should be minimized for cost-optimal parts.

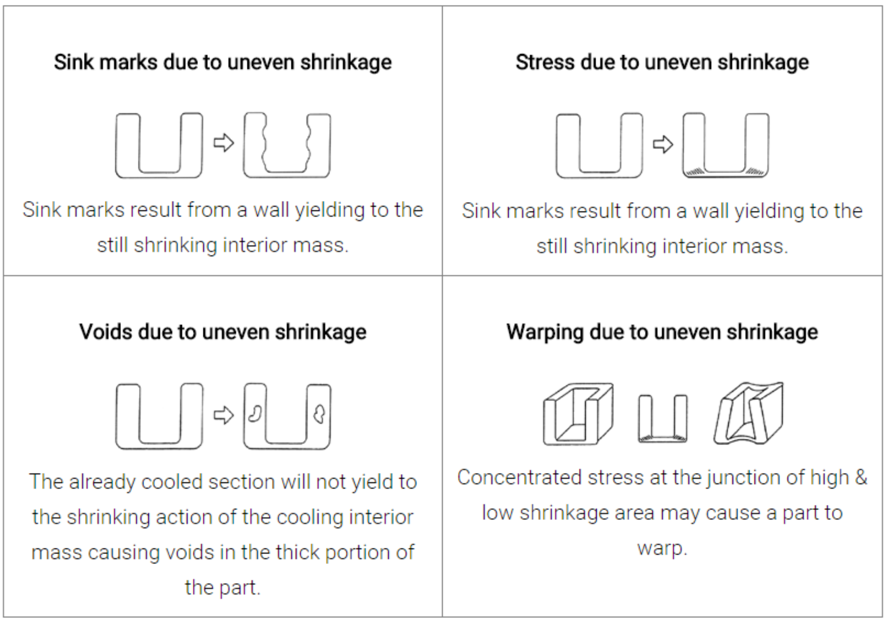

Figure 10.7: Uneven wall thickness leads to uneven shrinkage which can cause sinking, residual stress, voids or warping. Image from Plastics One.

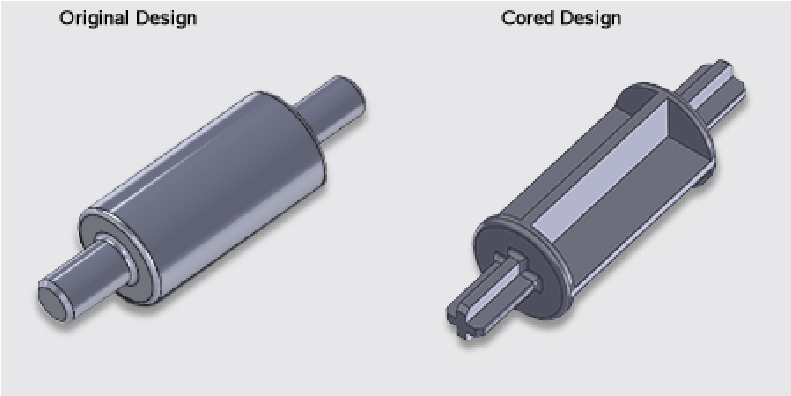

Figure 10.8: Example of a component designed for lathe production and its cored counterpart suitable for injection molding (ProtoLabs).

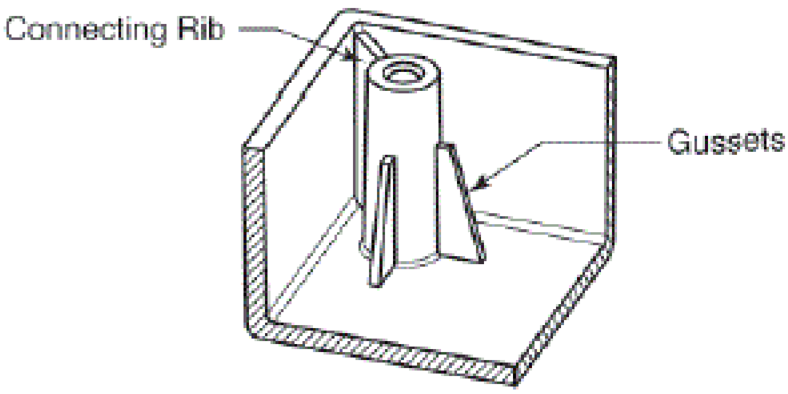

Figure 10.9: Examples of boss strengthening (Stratasys Direct Manufacturing).

10.1.8 Coring

One way to maintain uniform wall thickness is to 'core' thick sections. An example is shown in Figure 10.8. Proper coring tends to decrease strength only slightly. While complexity is increased, part cost and quality are usually improved overall.

10.1.9 Bosses

Bosses are often used in injection molding to facilitate mating parts, attach fasteners or accept inserts. If a boss is not visible, its wall thickness can be kept at the nominal thickness. If it is visible, it is recommended that 60% of the nominal thickness be used to reduce surface defects. In cases where a boss is isolated and requires extra strength gussets or ribs may be used.

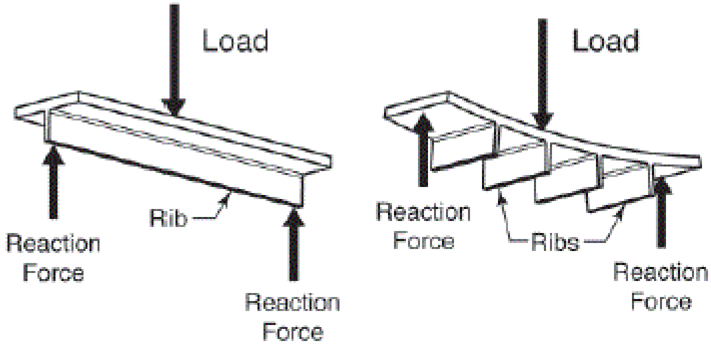

Figure 10.10: Rib orientation should be chosen carefully to maximize strength for the expected loading conditions (Stratasys Direct Manufacturing).

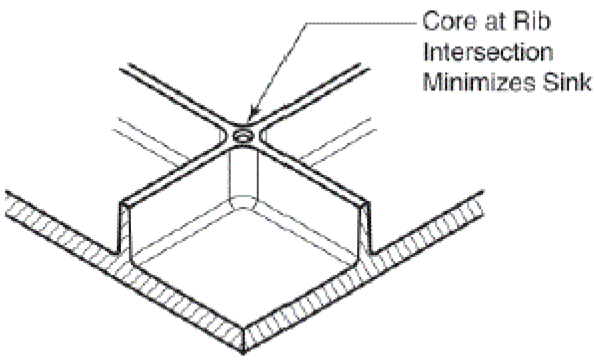

Figure 10.11: Coring rib intersections reduces sink (Stratasys Direct Manu.).

10.1.10 Ribs

One of the best ways to increase strength or stiffness in injection molded parts is to add structural ribs. Ribs can cause defects, however, and the following guidelines are intended to minimize negative effects. Rib thickness should generally be about 60-80% of nominal wall thickness, and rib height should be limited to about three times the thickness. Separation between ribs should be at least two times nominal wall thickness. This allows for better temperature distributions during cooling. Of course, the location and orientation of ribs should be chosen so as to maximize strength against expected loads (Figure 10.10)

If ribs intersect, it recommended that a core hole be inserted at the intersection (Figure 10.11). This keeps the effective wall thickness more uniform, reducing the likelihood of sinks.

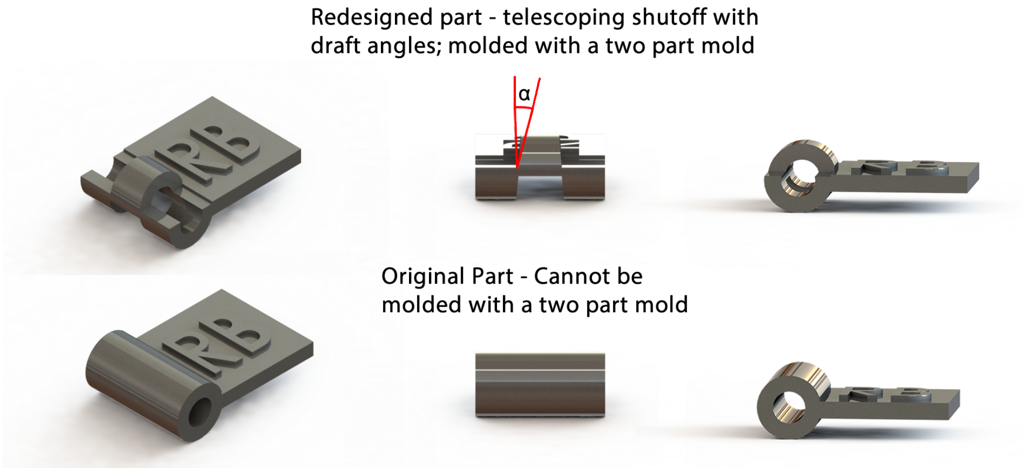

Figure 10.12: Side action part and example of a functional alternative.

10.1.11 Side-Action Molds

Parts with geometry that cannot be formed by two separable mold plates require an additional molding plate and actuator, which greatly increases mold cost. The third mold section is inserted into one of the main mold plates and moves in and out from the side. After the cooling stage, but before the main molds separate, the side action mold is removed to allow ejection. The addition of side action increases both variable costs, because the duration of the operation is increased, and tooling costs, because of the mold and actuator complexity is increased.

Side action is usually used to create undercuts or un-drafted sidewalls. For example, the part at the bottom of Figure 10.12 would require a side-action mold. Most likely, a straight pull mold would be used to create the overall shape and letter bosses, while a side pull would be used to create the hole for the hinge. Sometimes, side action can be avoided. An example of an alternative approach is shown at the top of Figure 10.12. In this case, similar functionality is provided using telescoping shutoffs. In this shutoff approach, both sides of the main pull mold come into contact at the through hole. When using shutoffs it is recommended that a larger draft angle be used. Of course, this alters the appearance of the part, as well as its strength, and therefore may not be an effective workaround in some situations.

	Hardened Steel	Aluminum	Beryllium-Copper
Material Cost	2	1	3
Machinability	3	1	2
Thermal Conductivity	3	1	2
Hardness	3	1	2

Table 10.1: Mold material property ranking, where 1 is lowest and 3 is highest.

10.1.12 Mold Material Cost vs Lifetime

Typical material choices for molds include hardened steel, aluminum and beryllium-copper alloys. The use of hardened steel allows for a large volume of production with minimal wear to the mold, but material and tooling cost are higher. Aluminum molds are typically used for smaller batch sizes since they are not as resistant to wear but have a much smaller cost in tooling and significantly better heat dissipation (Table 10.1).

10.2 Investment Casting

10.2.1 Applications

Investment casting, also known as lost wax casting, has been used for thousands of years, and technological advances in the past hundred years have allowed it to flourish. A wide variety of materials and shapes can be cast with this method, including metal forms unobtainable by die-casting, with a high degree of repeatability and robustness. In general, investment casting is more expensive per unit but has lower equipment costs when compared to sand casting or die-casting. Its main advantage over other processes is the ability to create complex shapes within tight tolerances.

10.2.2 Investment Casting Process

Wax injection

The investment begins with a sacrificial wax pattern that can be made using different molding processes. When creating this mold, shrinkage of wax and ceramic slurry must be taken into account and can sometimes require multiple trials to get final parts within tolerances. This is part of the reason these molds are expensive.

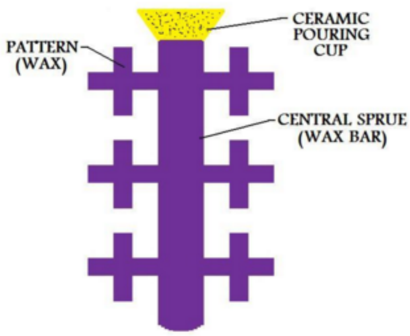

Figure 10.13: Wax pattern tree for investment casting.

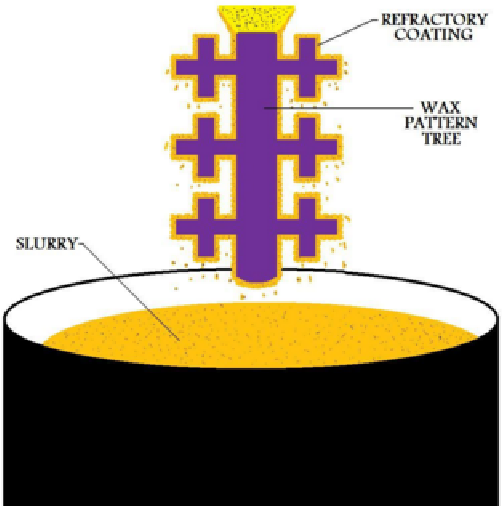

Figure 10.14: Ceramic coating on the wax pattern.

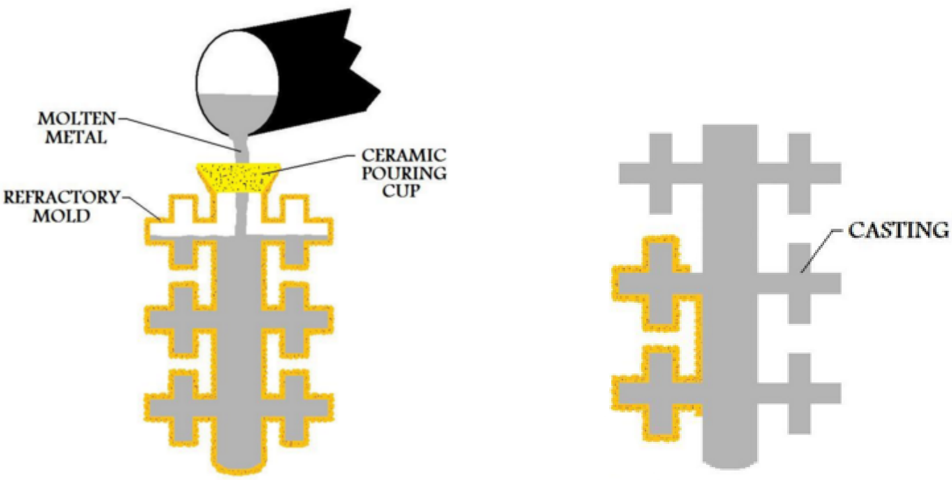

Figure 10.15: *Left:* Pouring metal into the ceramic mold. *Right:* Knocking off the ceramic leaves the castings and the sprue, which must then be cut off. Images are from the Library of Manufacturing.

Assembly

Once a wax pattern is made, it is connected to a central wax component (sprue) along with other similar patterns to compose a casting cluster or tree (Figure 10.13). This wax structure will be lost (hence the name lost wax). A new wax positive must be molded for every metal casting.

Investment casting shell

Having the casting cluster completed, the wax is dipped in a ceramic slurry multiple times until there is a ceramic coat around the entire tree (Figure 10.14). Once the coating is thick enough it is left out in air to allow it to dry and harden. Once the ceramic coating is dry, the assembly is placed in an oven. The wax is first melted out, leaving the cavity into which the metal will be poured. Once the wax has been removed, the oven is brought up to higher temperatures to strengthen the ceramic and remove any leftover water or wax.

Metal cast pouring

The mold is heated to a high temperature, to avoid thermal stress, and the molten metal alloy is poured into the ceramic and allowed to set.

Knockout and cutoff

Once the metal has cooled the ceramic is chipped off. The final casting is cut from the sprue and additional operations, such as machining or grinding, are performed.

10.2.3 Draft Angle

Draft angle for investment casting is not a requirement. Zero draft or negative draft can be achieved with the initial wax shape, and therefore the final metal part, because of the way the wax shrinks as it cools.

10.2.4 Shape

Investment casting offers flexibility in the shape of the final part because wax positives can be easily molded and machined, and can be built up to complex shapes in multiple steps. However, some technique is required in order to retain tolerances and avoid defects, as explained in the following subsections.

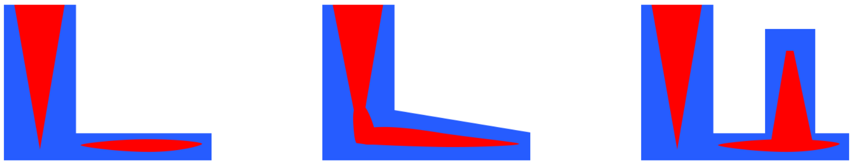

Figure 10.16: *Left:* Molten material with no access to sprue will probably lead to a shrinkage void. *Center:* Tapering the feature protects the flow line at the corner, allowing flow from the sprue throughout solidification. *Right:* A sacrificial riser provides liquid metal to feed the thinner part solidifying below, and takes the void. The riser is removed in post processing. Images from ESP International.

10.2.5 Material Flow

Molten metal flows in from the gate at the sprue and works its way to all the extremities of the part. As the metal cools and becomes solid within the mold, it shrinks. If any pockets of molten metal become isolated from the gate, metal cannot flow into that pocket as it cools and shrinks. This leads to defects such as voids, surface sinking and high residual stresses. It is therefore desirable to shape the part such that a single, continuous section of molten metal, connected to the gate and sprue, will be maintained throughout the freezing process. Material freezes at the thinnest sections of the component first, and so good shapes for investment casting have their thickest features at the gate and their thinnest sections farthest away from the gate. If a thin section separates two thick sections, it can freeze first, leading to defects in the isolated thick feature. If such isolation cannot be avoided, creating gradual transitions helps to mitigate the flaws.

Flow considerations explain why the sprue tends to be large; the metal pool in the sprue should be the last to harden as the casting cools, so that it can provide a reservoir of liquid metal to feed the part as it shrinks. The sprue is recovered and melted down for future castings to avoid waste.

10.2.6 Controlling Flow for Shrinkage

Voids and residual stresses occur when there is not enough molten metal to feed the space created by shrinkage. In most cases this happens when an internal area is molten and the path to the sprue has been cut off by solidified material (Figure 10.16). One way to reduce these defects is to add material to taper the feature, maintaining a continuous pool throughout freezing. Another method is to add a riser. Risers are thicker than the feature, holding molten metal longer and causing shrinkage effects to occur off of the part. Risers must be cut off after casting.

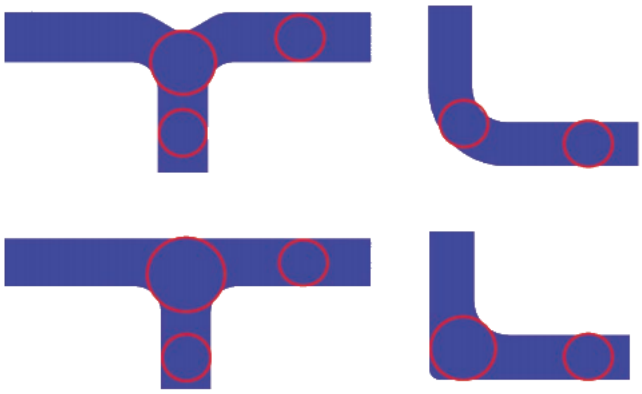

Figure 10.17: Wall junction design (ESP International). Junctions on the top row have relatively even wall thickness, which is preferable, while the junctions on the bottom row are uneven and more likely to lead to defects.

10.2.7 Fillets and Junctions

Sharp corners can cause high residual stresses when cooling. Internal sharp corners can also cause ceramic slurry to overheat, creating a rough finish. External corners can also cause fast cooling regions, leading to hardening similar to that which occurs during heat treatment. Fillets mitigate these issues and help with material flow around corners. At wall junctions, cutouts and rounding allow more uniform wall thickness to be maintained (Figure 10.17).

10.2.8 Wall Thickness

Walls can be as thin as 0.030 inches, but the thinnest walls commonly produced are 0.060–0.090 inches thick, depending on feature surface area and alloy being cast. Considerations for wall design are similar to other casting methods, in that it is desirable to maintain a similar wall thickness throughout the component. Ribs are often used to strengthen parts.

10.2.9 Aspect Ratio

If a section of a component is long and thin, uneven cooling of the walls can lead to small pockets of liquid metal becoming trapped during freezing. In long thin features this can lead to warping, in addition to other defects. To avoid these issues, a maximum aspect ratio of about 4:1 is typically recommended.

DIMENSION	NORMAL	PREMIUMS
UP TO 1/2"	±.005	±.003
UP TO 1"	±.005	±.004
UP TO 2"	±.010	±.005
UP TO 3"	±.015	±.008
UP TO 4"	±.015	±.010
UP TO 5"	±.020	±.010
UP TO 6"	±.020	±.010
UP TO 7"	±.030	±.015
UP TO 8"	±.040	±.015
UP TO 9"	±.040	±.015
UP TO 10"	±.050	±.020

Figure 10.18: Standard linear tolerances for investment casting.

Hole Type	Size Range	Dia.to Length Ratio
blind	3/16 and up	1 : 1-1/2
thru	1/8 to 3/16	1 : 1
	3/16 to 1/4	1 : 1-1/2
	1/4 to 1/2	1 : 2
	1/2 to 1	1 : 3
	1 and up	1 : 4

Figure 10.19: Typical guidelines for maximum hole depth in investment cast parts.

10.2.10 Tolerance limits

Linear tolerances for cast features are provided in Table 10.18. Guidelines for other requirements, such as concentricity or straightness, can be found in reference texts or online.

10.2.11 Holes

Small holes present a difficulty in investment casting because they may not be completely filled with ceramic slurry during the coating process. For example, air bubbles can form within the hole in the wax positive and prevent penetration by slurry. The incomplete cylinder formed in the ceramic mold will allow metal to partially fill the part hole during casting. Similarly, it can become difficult to remove the ceramic from long, thin holes after casting. Maximum hole depth therefore depends on the hole diameter, whether the hole is through (easier) or blind (harder), the ceramic and the coating process involved. A typical set of guidelines for maximum hole depth is provided in Table 10.19. Longer hole depths can be achieved using processes such as 'ceramic coring', in which a ceramic positive is made separately and inserted into the ceramic negative, but at increased cost. A counterbore or countersink can be incorporated easily.

10.3 Other Details and Processes

Similar guidelines for other features and for a wide variety of other manufacturing processes can be found in manufacturing textbooks, on manufacturer web pages and among other online resources. Many molding processes use similar procedures and tools to the ones described here, which can help in identifying the 'known unknowns'. For example, die casting utilizes similar molds as injection molding, with similar considerations of draft angle, coring, ribbing and pulls. Die casting also uses molten metals as with investment casting, with similar considerations for flow of material to freezing regions. When in doubt, search for guidelines that address your particular design situation.

10.4 Acknowledgments

Thanks to Roberto Jaime for help in drafting this chapter.