

Design In Plastics

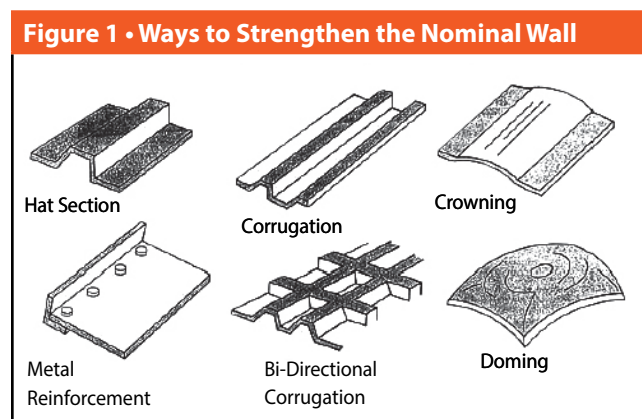
5-part series on designing and fabricating plastic parts.

Design Considerations For Injection Molded Parts – Part 1

Injection molding is an excellent way to bring out the best in plastics. It is universally used to make complex, finished parts, often in a single step, economically, precisely and with little waste. Designers face a huge number of options when they create injection-molded components. This article explores many of these options as they concern design approaches, resin selection and common features used to make optimum parts.

Design approaches

Design generally involves three overriding factors. First, build parts around functional needs. Combine them to reduce weight, eliminate fabrication and assembly steps, improve structural integrity and lower cost. Second, optimize cost and performance by keeping plastic use to a minimum while satisfying structural, functional, molding and economic demands. And the third, find the right resin.



Manufacturers often select a familiar grade of plastic from a similar application or rely on recommendations from molders, toolmakers or suppliers. Resins chosen this way may be adequate, but are rarely optimal. Plastic selection is a complex task that involves many considerations, such as:

- **Temperature** – look at thermal stress during normal and extreme end-use conditions, as well as during assembly, finishing and shipping.

- **Chemical resistance** – evaluate the effect on the part of every solid, liquid or gas that can contact it.
- **Other conditions** – consider all other items relevant to fabrication, assembly and end use. These include maximum loads, deflections and other mechanical stresses, relative motion between parts, electrical stresses, color and tolerances (to define the precision of various steps).
- **Agency approvals** – factor in governmental and private standards for properties such as heat resistance, flammability, and electrical and mechanical capabilities. Approval bodies include Underwriters Laboratories, NSF International (National Sanitation Foundation), Canadian Standards Association, U.S. Food and Drug Administration and U.S. Department of Agriculture.
- **Assembly** – ensure the proposed plastic works with all assembly steps, such as solvent bonding, mechanical fasteners or ultrasonic welding.
- **Finish** – also ensure the plastic can provide the desired gloss, smoothness and other appearance values as it comes from the mold or that it can be finished economically.
- **Cost** – use total finished-part cost to guide design. In addition to resin pricing, factor in manufacturing, maintenance, assembly and disassembly to reduce labor, tooling, finishing and other costs.
- **Availability** – make sure the resin is available in the amount needed for production.

Part elements

Injection-molded plastic parts tend to have similar elements. They all have walls of a certain thickness (the nominal wall) and usually have one or more of the following: ribs to add strength; bosses as points of attachment; radii where surfaces intersect; and open internal spaces.

Walls should be as thin as possible without sacrificing mechanical integrity. Depending on the plastic used, wall thickness in injection-molded plastic parts usually is 0.03 to 0.19 in. For instance, typical thicknesses with acetal fall between 0.03 and 0.12 in., while with polyester they are 0.025 to 0.125 in. (Table 1) If walls are too thin, parts can fail under load. If they are too thick, products can be unattractive, overweight or expensive, and take too long to mold.

Table 1 • Typical Nominal Thickness for Various Thermoplastics

Acrylonitrile-butadiene-styrene (ABS)	0.045-0.140
Acetal	0.030-0.120
Acrylic	0.025-0.150
Liquid crystal polymer	0.008-0.120
Long-fiber reinforced plastics	0.075-1.000
Nylon	0.010-0.115
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
Polyvinyl chloride	0.040-0.150

Wall thickness should not vary more than 10 to 25 percent. If greater variations are unavoidable, transitions should be gradual and the polymer should flow toward the thin sections, so thick areas fill well and stress is minimized.

Designers increase part strength and stiffness in several ways. One option is to switch to a different plastic or a

glass-reinforced version of the same resin. Another possibility is to thicken the nominal wall, but this can add cost and molding time. A third way is to beef up the part with ribs, gussets (small triangular ribs), corrugations, crowning and other features.

Ribs are most commonly used to add strength. They are varied in height and number according to load and available space. Multiple, evenly spaced ribs usually distribute loads better than one large rib. The thickness of a rib at its base is usually half that of the adjacent wall, but can equal that of the wall if structural integrity matters more than appearance or if the resin has little shrinkage. Rib height is typically 2.5 to three times wall thickness.

When attachment points and points of alignment are needed, bosses are added. They too should be no thicker than the nominal wall and can be strengthened by gussets to counter lateral forces. In bosses for self-tapping screws, configure inside diameter and wall thickness to avoid excessive hoop stress. When molding-in a tapered pipe thread, avoid a wedging action on the boss and, if possible, use the male rather than female thread.

Stress that accumulates in sharp corners as a plastic shrinks after molding can cause failure under high load or impact. To avoid this, use generous radii on corners where ribs, bosses, sidewalls and other features connect. Inside corner radii (fillets) vary with stress. In stress-free areas, radii can be as small as 0.005 in., while those where loads are high should have radii of over 0.02 in. The radius of an outside corner should equal that of the inside corner plus the wall thickness.

Cores add openings in parts to reduce weight or provide mounting points. They are created by pins that project into the mold cavity. The minimum core size depends on how well its pin can withstand the force of flowing plastic. Pins for blind holes are supported at one side of the mold. Their depth should be less than three times their diameter or minimum cross-section (or less than 2 times, if the minimal dimension is under 0.25 in.). Cores that extend to other parts of the mold can be twice as long as blind cores, because they are supported at both ends.

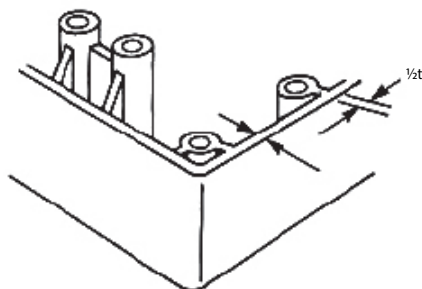
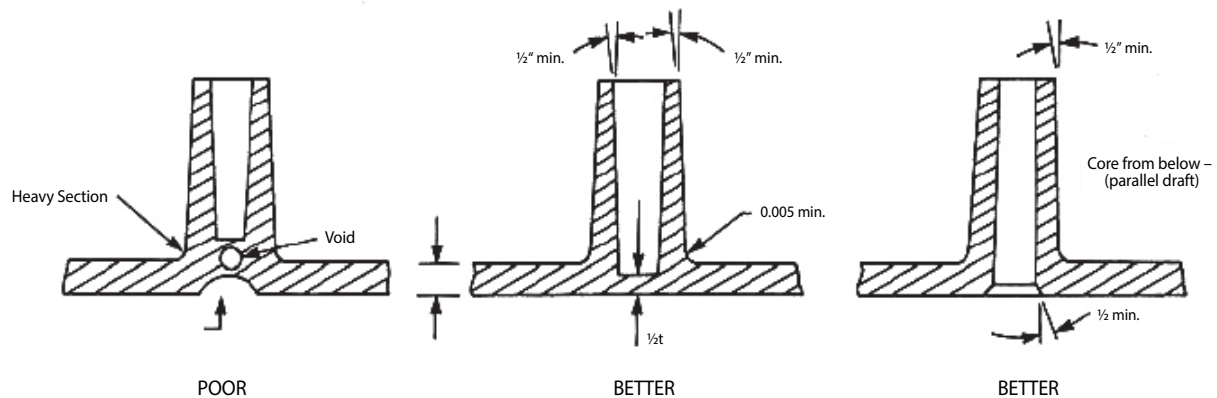
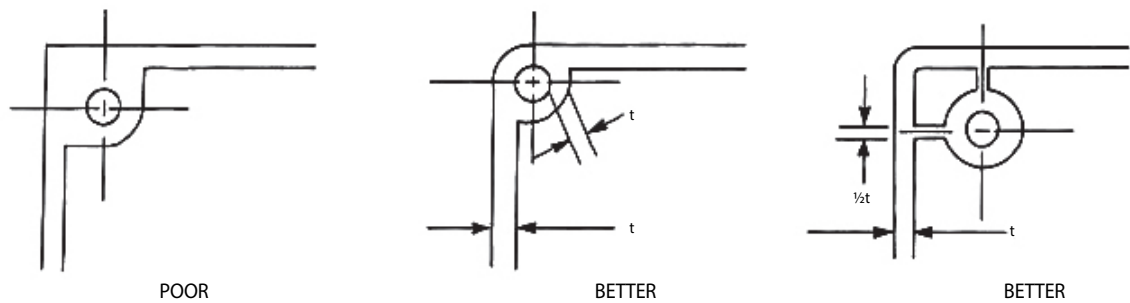
3 Mold release

Parts should be designed so mold tools open and eject the part easily. This is most often done by tapering ribs, bosses and other elements. Taper (or draft) should be at least 0.5 deg. per side, although 1.5 to 3 deg. per side is more common. Draft also depends on mold surface finish, and increases at least 1 deg. per side for each 0.001-in. depth of texture. A 1 deg. draft yields a 0.017-in. taper per inch of length.

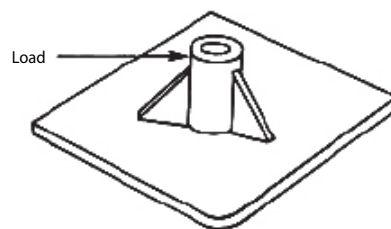
involved mechanical movements, such as side action, cams, pullers and collapsing cores, to enable release. This means considering where parting lines, undercuts and other features fall in the mold. In some cases, special mold surface treatments or a mold release spray may be needed.

In general, approach the design of plastic parts from a system standpoint. Choices made in the material selected, design of the overall part and its elements and the assembly methods chosen influence each other. As the design process progresses from concept to production, these choices should be reevaluated repeatedly.

Figure 2 • Design of Bosses



Connecting Bosses to Outside Walls with Ribs



Using Gussets Rather Than Very Thick Bosses When Resistance to Loading is Required

Physical And Mechanical Properties of Plastics – Part 2



Choosing a plastic for a specific use can be a daunting task. Designers face a seemingly endless variety of resins and a host of properties that define them. Each market usually needs a unique set of properties for the plastics used in it (Table 1). Electronic connectors, for example, are complex, precision components that need good flow in thin section and high dimensional stability. Packaging materials, by contrast, need stiffness, strength and good water vapor barrier properties. These requirements drive the process for selecting plastics. This article reviews plastic materials, and the key physical and mechanical measurements that define them. Plastics are either thermosets or thermoplastics. Thermosets flow before molding but undergo chemical change during processing, which cures or hardens them to create a complex, interconnected network. If too much heat is added after this, the polymer degrades rather than melts. Thermosetting plastics include phenolic, epoxy, alkyd polyester, polyurethane, urea-formaldehyde and unsaturated polyester resins. Natural and synthetic rubbers, such as latex, nitrile, millable polyurethane, silicone, butyl and neoprene, are also thermosets.

Thermoplastics, by contrast, soften when heated and harden when cooled. Molding does not change their chemical structure. They have a performance-based hierarchy from commodity to engineering grades. Commodity thermoplastics include low and high density polyethylenes and polypropylene. Engineering

Table 1 • Key Performance Characteristics by Market

E/E Interconnects ■ Good flow in thin walls ■ Dimensional precision ■ Heat resistance ■ Flame retardance Healthcare ■ Good flow in thin walls ■ Chemical resistance ■ Withstands sterilization ■ Stiffness, strength	Telecommunications ■ Good flow in thin walls ■ Dimensional precision ■ Stiffness, strength Automotive ■ Good flow in thin walls ■ Solvent resistance ■ Temperature resistance ■ Dimensional stability Business Machines ■ Good flow in thin walls ■ Dimensional precision ■ Chemical resistance	Packaging ■ Excellent barrier properties ■ Stiffness, strength Cryogenics ■ Excellent barrier properties ■ Good low temperature properties ■ Stiffness, strength Audio/Visual ■ Good flow in thin walls ■ Dimensional precision ■ Stiffness, strength ■ Temperature resistance
--	---	--

Table 2 • Comparison of Properties		
Amorphous polymers	Semi-crystalline polymers	Liquid crystal polymers
No sharp melting point	Relatively sharp melting point	Melt over a range of temperatures, low heat of fusion
Random chain orientation in both solid and melt phase	Ordered arrangement of chains of molecules and regular recurrence of crystalline structure only in solid phase	High chain continuity; extremely ordered molecular structure in both melt phase and solid phase
Don't flow as easily as semi-crystalline polymers in molding process	Flows easily above melting point	Flow extremely well under shear within melting range
Fiberglass and/or mineral reinforcement only slightly improves Deflection Temperature under Load (DTUL)	Reinforcement increases load bearing capabilities and DTUL considerably, particularly with highly crystalline polymers	Reinforcement reduces anisotropy and increases load bearing capability and DTUL
Can give a transparent part	Part is usually opaque due to the crystal structure of semi-crystalline resin	Part is always opaque due to the crystal structure of liquid crystal resin
Examples: cyclic olefinic copolymer (Topas COC), acrylonitrile-butadienestyrene (ABS), polystyrene (PS), polycarbonate (PC), polysulfone (PSu), polyetherimide (PEI)	Examples: polyester (Impet [®] PET, Celanex [®] PBT, Duranex [®] PBT), polyphenylene sulfide (Fortron [®] PPS), polyamide (Celanese nylon), polyacetal copolymer (Celcon [®] POM, Hostaform [®] POM, Duracon [®] POM)	Example: Vectra [®] LCP

thermoplastics include acetal, nylon, polycarbonate, polyphenylene sulfide and liquid crystal polymer. Those higher in the hierarchy generally carry greater loads and withstand impact, high temperature and chemical attack better.

Most thermoplastics are rigid, but some are highly elastic (thermoplastic elastomers, or TPEs), and can be stretched repeatedly to at least twice their original length at room temperature, then return to near their original length. Many injection-moldable TPEs are replacing traditional rubbers. They are often used to modify rigid thermoplastics to aid impact strength. Additionally, most thermoplastics are amorphous, but some have chains packed in an organized way and are considered partly or mostly crystalline. All crystalline plastics have amorphous regions between and connecting their crystalline regions. Liquid crystal polymer, a good example of a semicrystalline resin, has rodlike molecules set in parallel arrays that makes them stiffer, stronger and more resistant to creep, heat and chemicals (Table 2).

A plastic's properties depend on its chemistry, structure, chain length and the bonds between chains. Its properties can be altered by combining it mechanically and chemically with one or more polymers. Alloys, or mechanical blends of two or more polymers, improve performance and processability and lower material cost. Their properties usually fall between those of the starting polymers,

although some have better properties than either one alone. Copolymers chemically combine two or more repeating units. Copolymers and homopolymers (one repeating unit) in the same plastic family can have different properties.

A plastic's physical and mechanical properties can be modified with additives, fillers and reinforcements. In general, mechanical properties are best increased by adding reinforcing fibers made of glass, metal, carbon or other materials. Particulate fillers like talc or ground calcium carbonate generally increase modulus, while plasticizers decrease modulus and enhance flexibility. Other common additives include flame retardants, oxidation inhibitors, and thermal and UV stabilizers.

Physical properties

Properties can vary through a plastic and with the direction of measurement. Plastics are homogeneous if they have the same makeup throughout as in many unfilled thermoplastics, or they can be heterogeneous and vary from point to point. If properties are the same when measured in any direction, the plastic is isotropic. If not, then it is anisotropic. In molded crystalline and glass fiberreinforced resins, properties can differ in the cross-flow and in-flow directions.

Physical properties often evaluated for specifying a plastic include:

- **Density and specific gravity** – Density is mass per unit volume (lb/in³ or g/cm³), while specific gravity is the mass of a volume of material divided by the same volume of water (both at 23°C). As a dimensionless number, specific gravity is a good way to compare materials as regards part cost, weight and quality control.
- **Water absorption** – the percent increase in weight of a specimen (dried for 24 hrs.) before and after immersion in 23°C water for various times. This property affects dimensional stability and some mechanical and electrical properties.
- **Mold shrinkage** – how much a part's dimension changes as it cools and solidifies in a mold divided by the mold dimension. Molds are sized to allow for shrinkage, which varies with wall thickness, flow direction and other conditions. Reinforced and filled plastics shrink less than neat ones.
- **Elasticity** – the plastic's ability to return to its original size and shape after being deformed. This is high in rubber and TPEs. Contrast this with plasticity, the opposite of elasticity, in which a plastic holds the shape to which it is deformed. This occurs when a plastic is stressed beyond its yield point (where permanent deformation begins).
- **Ductility** – how well a plastic can be deformed without fracture or how well it can be stretched, pulled or rolled without destroying its integrity. It is measured as a percent elongation.
- **Opacity and transparency** – usually measured as haze and luminous transmittance. Haze is the percent of light transmitted through a specimen and scattered more than 2.5 deg from the incident beam. Luminous transmittance is the ratio of transmitted light to incident light.
- **Lubricity** – measures load bearing characteristics under relative motion. Good lubricity goes with low coefficient of friction and a tendency not to gall (to be worn away by friction).

Mechanical properties

Mechanical properties are crucial in nearly all plastic applications. Laboratory-generated, short-term data provide ideal values that are useful in selecting a resin. Lab tests usually subject samples to a single, steady force for a limited time. In the real world, many factors occur at once, so parts should be evaluated in actual use to gauge how varying force, temperature and other factors affect them. Note that short-term values do not work for such structural responses as creep, impact and fatigue.

The most significant mechanical properties include:

- **Toughness** – the ability to absorb mechanical energy without fracturing. High-impact, unfilled resins generally have excellent toughness. Brittle resins, which lack toughness, often have less impact strength and higher stiffness. Many glass-reinforced and mineral-filled materials are brittle.
- **Tensile strength** – the maximum amount of tensile load per unit area a material can withstand.
- **Tensile elongation** – how much length increases in response to a tensile load expressed as a percent of the original length. Elongation at break is the maximum elongation the plastic can undergo.
- **Flexural strength** – how much of a bending load a plastic can withstand before it ruptures.
- **Creep** – a plastic's deformation under load (tension, compression or flexure) over time. It does not include the initial change in dimension when the load is applied. The rate of creep depends on applied stress, temperature and time. It is usually measured at four or more stress levels and plotted as strain vs. log of time. Crystalline resins usually have lower creep rates than amorphous ones. Glass fiber reinforcement improves creep resistance. A plastic part will fail when too high a fixed strain is imposed for too long a time.
- **Impact loading** – evaluates how well a part absorbs energy from an impact and is dependent on the shape, size, thickness and type of material. Tests can be done on notched or unnotched samples. Notched tests measure how easily a crack propagates through a material. Impact tests are not analytical, but can be used to compare the relative impact resistance of materials.

The main impact tests are:

- **Izod impact strength (mainly the U.S.)** – a pendulum arm is swung from a height to impact a notched cantilevered specimen. The distance the pendulum travels after fracturing the material indicates the loss of energy, which is the Izod strength in ft-lb/in or J/m. This test also has unnotched or reversed notched (facing away from pendulum) versions.
- **Charpy impact (mainly in Europe)** – like the Izod measurement, except the beam is supported at both ends. The pendulum hits the beam at its midpoint.
- **Falling dart impact** – involves dropping a weight onto a flat disk. Height of release and dart weight are varied until half the specimens break.
- **Fatigue endurance** – evaluates the mechanical deterioration and failure of parts that are cyclically stressed. This test subjects a cantilevered beam to reverse flexural loading cycles at different stress levels.

Thermal, Electrical And Chemical Properties Of Plastics – Part 3

Plastics often must meet multiple demands in any one application. Beyond the need for specific physical and mechanical properties such as strength and stiffness (discussed in the previous article in this series), plastics often encounter temperature, electrical and environmental stresses. In selecting the right plastic for an end use, designers must understand how key properties in these areas are measured. This article offers a compendium of the most significant thermal, electrical and chemical parameters as a guide.

Thermal properties

Higher temperatures make plastics more sensitive to mechanical stresses and vulnerable to chemical attack, while lower ones generally make them less ductile. In designing a part, it is important to understand what thermal conditions it will meet during processing, assembly, finishing, shipping, and end use to ensure it retains its integrity. The main thermal properties of plastics include:

- **Melting point** – a property important for molding and assembly. Crystalline thermoplastics have sharp, welldefined melting points, while amorphous ones soften and grow more fluid over a wider range.
- **Melt index** – measures how much plastic is extruded through a heated apparatus in 10 minutes (reported as g/10 min.). Greater flow rates indicate greater viscosity. The results guide the selection of molding conditions.
- **Vicat softening point** – the temperature at which a small, circular, heated and lightly loaded probe penetrates a set distance into a specimen. It indicates a material's ability to withstand short-term contact with a heated surface. It is useful for crystalline plastics, but of limited value for other types that can creep during the test.
- **Coefficient of linear thermal expansion (CLTE)** – measures the change in one dimension against the original dimension per unit change in temperature (in./in./°F·10⁻⁵ or cm/cm/°C·10⁻⁵) (Table 1). This is important in gauging stresses when assembling dissimilar materials. CLTE can vary with temperature, and anisotropic materials have different CLTEs in different directions.

- **Deflection temperature under load (formerly called heat deflection temperature)** – the temperature a test bar under a given load deflects a set amount. It is a relative measure of how well a plastic performs under load at elevated temperatures.

Table 1 • Coefficient of Linear Thermal Expansion for Thermoplastics and Other Materials*

Material	in./in./°F 10 ⁻⁵	cm/cm/°C 10 ⁻⁵
Liquid Crystal (GR)	0.3	0.6
Glass	0.4	0.7
Steel	0.6	1.1
Concrete	0.8	1.4
Copper	0.9	1.6
Bronze	1.0	1.8
Brass	1.0	1.8
Aluminum	1.2	2.2
Polycarbonate (GR)	1.2	2.2
Nylon (GR)	1.3	2.3
TP Polyester (GR)	1.4	2.5
Magnesium	1.4	2.5
Zinc	1.7	3.1
ABS (GR)	1.7	3.1
Polypropylene (GR)	1.8	3.2
Epoxy (GR)	2.0	3.6
Polyphenylene Sulfide	2.0	3.6
Acetal (GR)	2.2	4.0
Epoxy	3.0	5.4
Polycarbonate	3.6	6.5
Acrylic	3.8	6.8
ABS	4.0	7.2
Nylon	4.5	8.1
Acetal	4.8	8.5
Polypropylene	4.8	8.6
TP Polyester	6.9	12.4
Polyethylene	7.2	13.0

* Plastics are unfilled unless marked as GR or glass filled. Table shows typical average values and does not account for differences in grades, molding conditions, wall thickness, flow direction, etc.

- **Thermal conductivity** – the rate heat energy travels along or through a plastic. This is important in heatgenerating applications and where heat dissipation is needed.
- **Aging at elevated temperatures** – looks at how a plastic's physical, mechanical, thermal or electrical properties change over time when samples are stored at high temperature. Elongation increases at higher temperatures and the effects of strain manifest more quickly. At lower temperatures, plastics tend to lose impact strength.
- **Flammability tests** – looks at a material's resistance to combustion. UL 94 is the most widely used flammability test for electrical devices. Ratings, which are V-0, V-1, V-2, 5V and HB, define how well a material combusts after a flame is removed.
- **Limiting oxygen index** – measures how much oxygen is needed to sustain combustion. A plastic needing less than 21% oxygen will burn in air.

Electrical properties

As good insulators, plastics provide essential dielectric properties. This quality is affected by temperature, moisture, time in service and other factors. Common electrical properties of plastics include:

- **Dielectric strength** – measures the highest current that can be applied to a plastic before it allows a current to pass. It is expressed as the voltage just before this happens divided by the thickness of the sample (in volts/mil). It is affected by temperature, thickness, how the sample was conditioned, rate of voltage increase, test duration and contamination.
- **Dielectric constant (or permittivity)** – looks at how easily a plastic can be polarized relative to a vacuum (Table 2). The test imposes an electrical field across an insulator, reverses the field and measures how long it takes to equilibrate. This dimensionless number, which is important in high-frequency applications, varies with temperature, moisture, frequency and thickness.
- **Dissipation factor** – measures the energy dissipated during rapid polarization reversals, as with an alternating current (Table 2). It can be seen as the ratio of energy lost as heat to current transmitted. It is usually measured at 1 MHz. This factor should be low when plastics are used as insulators in high-frequency applications such as radar and microwave equipment.

- **Arc resistance** – measures time (in seconds) for an electrical arc imposed on a plastic to create a conductive path. It was developed for thermosets in which a path forms from decomposition products due to localized heating. A higher quantity is better where arcing is a possibility, as in switches, circuit breakers and auto ignitions.
- **Comparative tracking index (CTI)** – an Underwriters' Laboratories test that measures how much voltage is needed to create a conductive path between electrodes on a surface after an electrolyte (0.1% ammonium chloride solution) is placed on that surface. CTI relates to the arc resistance of real-world equipment with surface contamination.
- **Volume resistivity** – a standard measure of conductivity when a direct current potential is applied across a material (measured as ohms x area of the smaller electrode/specimen thickness). Materials measuring above 108 ohm-cm are insulators.
- **Surface resistivity** – expresses how well a current flows over the surface of a material between electrodes placed on the same side of a specimen. While volume resistivity is a property of the material, surface resistivity measures how susceptible a plastic is to surface contamination, especially moisture. It is useful when surface leakage may be a problem, but, since it is not exact, it should be used with wide margin of safety.

Table 2 • Dielectric Constant and Dissipation Factor for Various Thermoplastics at Room Temperature

Material Description	Dielectric Constant	Dissipation Factor
Acetal	3.7-3.9	0.001-0.007
Acrylic	2.1-3.9	0.001-0.060
ABS	2.9-3.4	0.006-0.021
Nylon 6/6	3.1-8.3	0.006-0.190
Polycarbonate	2.9-3.8	0.0006-0.0260
TP Polyester	3.0-4.5	0.0012-0.0220
Polypropylene	2.3-2.9	0.003-0.014
Polysulfone	2.7-3.8	0.0008-0.0090
Modified PPE (PPO)	2.4-3.1	0.0002-0.0050
Polyphenylene Sulfide	2.9-4.5	0.001-0.002
Polyarylate	2.6-3.1	0.001-0.022
Liquid Crystal Polymer	3.7-10	0.010-0.060

Environmental stresses

Chemicals (acids, bases, solvents and fuels), ultraviolet radiation, humidity and other environmental stresses can cause plastics to craze, crack, discolor, lose properties, soften or dissolve. Data from suppliers on how their products react to such stresses generally include exposure time, concentration and temperature. While this is helpful, the real world usually combines stresses that affect resins in different ways than one stress acting alone.

When evaluating a resin, it is best to expose it to the environments they will meet during assembly and end use. A resin's compatibility to an environment is often determined by time of exposure; concentration of adverse substances; whether these substances are solids, liquids or gases; radiation level and intensity; and the use of coatings and/or other protective barriers.

A chemical may interact in different ways to different plastics and even to different grades in the same family. It may react with the polymer to reduce molecular weight and alter short-term mechanical properties. It may dissolve the plastic, showing up as a shift in weight, dimension (swelling) and property loss. Or it can be absorbed into the polymer and cause plasticization, reducing strength, stiffness, and creep and impact resistance. A chemical may not look like it's affecting a plastic when there is no stress applied, but may lead to failure under stress. And plastics are more susceptible to environmental stresses when they are under load or when temperature increases.

Chemical compatibility is measured by placing test bars of a plastic in a chemical for a set time at a set temperature and evaluating them for mechanical, appearance and other properties. End use testing is essential if chemical exposure is an issue.

All plastics degrade (fade, chalk or grow brittle) when exposed to the UV found in sunlight. UV testing exposes samples for set times either directly to the sun, or to highintensity xenon or carbon-arc lamps in special test cabinets that accelerate testing.

A final word about properties

Resin screening should consider all factors a plastic will meet during handling, assembly, finishing and end use. In using published test data, keep in mind that many standard tests have a narrow focus and offer comparison data rather than absolute values. Since standard tests are done under controlled laboratory conditions, they usually do not include all factors that influence a part in actual use.

Some characteristics, like adhesion or wear cannot be predicted accurately by standard laboratory tests. Other properties, such as dimensional stability, strength, and rigidity, combine the effects of several factors and are hard to measure in the lab. For example, dimensional stability might combine coefficient of thermal expansion, moisture absorption, post-mold shrinkage, and relaxation of mold-in stresses. In nearly all cases, parts should be tested under the conditions they will encounter during their life cycle.

Injection Molding And Design With Plastic – Part 4

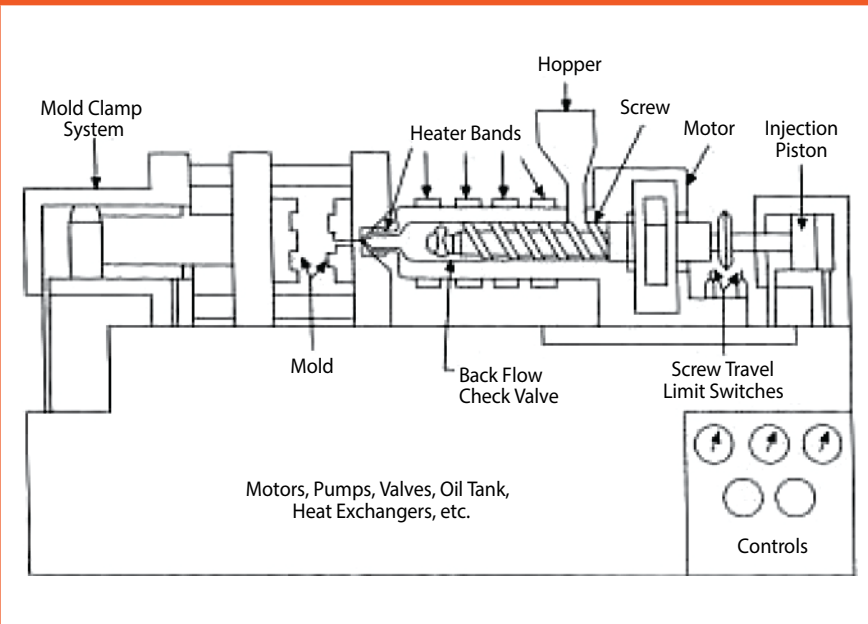
Injection molding is the most prevalent way to make plastic components because it is economical, efficient and accurate. In working with this method, product designers must account for both the mold and the part itself. The overall design task should take into account every aspect of process from molding machine to processing conditions to resin.

Injection molding is simple in concept – plastic is melted, injected into a cavity and released when it has sufficiently hardened. Molding machines have an injection section that plasticizes and pushes the resin into the mold, and a clamping mechanism that opens and closes the mold. The process is usually worked with thermoplastics rather than thermosets, so only thermoplastics are considered here.

Molding overview

Most molding machines feed small, chopped plastic pellets from a hopper to a reciprocating screw that moves them through a heated barrel (the interaction between the rotating screw and the plastic also generates heat). The plastic melts and collects in front of the screw, which retracts until enough molten plastic is present to fill the mold. Shear generated by the flights of the screw mixes the melt so it is uniform.

Figure 1 • Schematic of Reciprocating Screw Injection Molding Machine

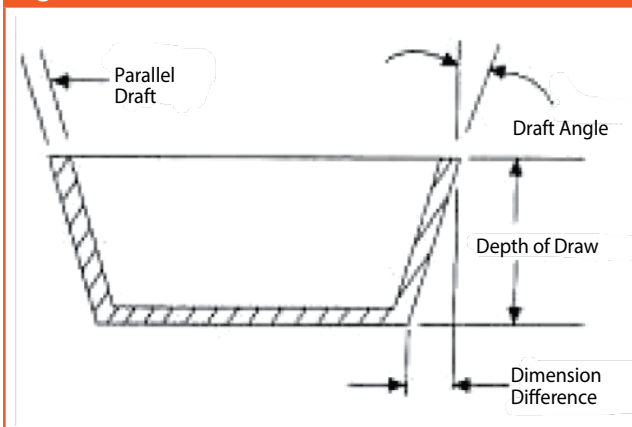


The screw is then pushed forward hydraulically (a “shot”), forcing the melt through a nozzle into the mold where it is distributed by runners and passes through gates into part cavities. This occurs at pressures as high as to 30,000 psi. Water, or another fluid circulating through channels cut in the mold, removes heat so that the plastic cools and solidifies from the walls of the cavity inward. When parts are rigid enough, the mold is opened and they are removed. The next shot is melted in the barrel as parts cool in the mold. Cooling time generally controls mold cycle time and thus production rate.

Many variations exist on the basic injection molding process, for instance:

- **Double shot molding** – a two-step process that injects one color or material and, after the first shot hardens, injects a second color or material into or around the initial shape.
- **Push-pull molding** – reduces anisotropy in fiber reinforced or self-reinforcing plastics (such as liquid crystal polymers) by alternately filling the mold from two directions. The multiple layers have different orientations so the part has more uniform properties than if it were molded from just one direction.

Figure 2 • Illustration of Draft



- **Fusible cores** – placed in molds to produce complex, hollow structures. Heating the part after molding melts the core, which flows out of the part.
- **Gas injection molding** – injection of inert gas (often nitrogen) into the melt as it enters the mold. The gas forms channels in thicker sections, packing the plastic into the mold to reduce cycle time and minimize stress, sinks and warping. Each area of a molding machine involves important variables that affect a part. For instance, barrel temperature helps determine resin temperature and thus resin viscosity, while the pressure exerted on the melt controls injection pressure and speed. Other important machine conditions include screw speed, shot size, cavity pressure, clamp tonnage, gate size, mold temperature, venting and much more.

The role of the plastic

The properties of the plastic used, such as its flowability, heat transfer ability and cooling shrinkage, affect molding efficiency. Many variations exist on the basic injection molding process, for instance:

Flowability is a relative measure of how far a plastic will travel from a gate. It is often measured as spiral flow curves, which show the distance a plastic flows at a set pressure and temperature after being injected into the center of a spiral runner in a mold. Flowability depends on a resin's melt viscosity, shear resistivity and thermal conductivity and affects gate type, size and placement, the temperature and pressure used, and other factors.

Once a plastic fills a mold, the plastic should have enough heat transfer so parts do not warp because of differential cooling in the mold. A relatively uniform mold temperature also helps optimum part characteristics to develop as

crystalline resins crystallize or amorphous ones anneal.

Mold cavities are sized to account for shrinkage as a thermoplastic solidifies from a shot, so finished part dimensions fall within tolerances. Shrinkage depends on the filler used and its orientation (fibrous fillers may cause more shrinkage in one direction than another). It also depends on part thickness and geometry (thin areas shrink less), gate and runner size, flow distance in the mold, and tool cooling and heating.

Mold design

A well-designed mold allows for the broadest possible processing window so short-term changes in resin viscosity, hydraulic pressure and barrel temperature, and longer-term variations like screw and barrel wear, can occur over time with little loss in part quality.

Mold tool form and cost is often driven by the nature of the part. Complex parts with deep recesses and screw threads, for instance, may need expensive release mechanisms such as collapsing cores, side pulls, slides, multiple plates, automatic unscrewing devices or intricate parting lines. In addition to part complexity, mold designers must consider many other factors including:

- **Gate type, size and placement** – affects pressure distribution in the cavity, how resin molecules and filler elements align, mold-filling time and shrinkage. Plastic should flow smoothly from a gate to the limits of the mold. Problems related to mold filling (e.g., flow lines and other surface defects or irregular density) often can be corrected at the gate. For instance, gates are often placed so weld lines (where two or more melt streams meet after flowing around a core) occur in noncritical areas of a part.

Table 1 • Dimensional Difference for Various Drafts

Draw Depth (inches)		1/8	1/4	1/2	1	2	3	4	5
	1	0.002	0.004	0.009	0.017	0.035	0.052	0.070	0.087
	2	0.004	0.009	0.017	0.035	0.070	0.105	0.140	0.175
	3	0.007	0.013	0.026	0.053	0.105	0.157	0.210	0.263
	4	0.009	0.018	0.035	0.070	0.140	0.210	0.280	0.350
	5	0.011	0.022	0.044	0.087	0.175	0.262	0.350	0.437
	6	0.013	0.026	0.052	0.105	0.209	0.314	0.420	0.525
	7	0.015	0.031	0.061	0.123	0.244	0.367	0.490	0.612
	8	0.018	0.035	0.070	0.140	0.279	0.419	0.559	0.700
	9	0.020	0.040	0.078	0.158	0.314	0.472	0.629	0.787
	10	0.022	0.044	0.087	0.175	0.379	0.524	0.669	0.875

- **Number of cavities per tool** – varies with part size and production volume, i.e., higher production volume and smaller parts usually mean more cavities per tool (sometimes hundreds per mold).
- **Draft** – enables the part to release by creating a clearance as soon as the mold opens. It involves adding a slight taper in the mold, usually a minimum of 0.5 deg. per side and more usually 1.5- to 3-deg. per side (Table 1 and Fig. 1). The degree of draft is affected by surface finish, i.e., the higher the polish the less the taper.
- **Large surface areas on parts** – may stress the clamping system if a machine is not large enough. If the clamping force is too low, mold may open slightly and allow a fringe of plastic or flash to form. Flash must be removed after molding.
- **Runners** – the nature of the system affects scrap generation. Conventional cold runners produce a lot of scrap, while hot sprue bushings reduce scrap by extending the nozzle into the mold. Hot runner systems yield no scrap, but they cost more and are harder to operate than cold runners. Scrap often can be reground and reused in the process.

Design aids

The injection molding design task can be highly complex. Just the need to meet set tolerances ties in to all aspects of the molding process, including part size and shape, resin chemical structure, the fillers used, mold cavity layout, gating, mold cooling and the release mechanisms used.

The machine itself is also important in how well it controls temperature, pressure and clamping forces, as well as screw and barrel wear.

Given this complexity, designers often use computer design tools, such as finite element analysis (FEA) and mold filling analysis (MFA), to reduce development time and cost. FEA determines strain, stress and deflection in a part by dividing the structure into small elements where these parameters can be well defined. While this method assumes a linear elastic material and is most often applied to structures under constant load, it can model nonlinear behaviors such as creep. Use of an anisotropic material makes the calculations more complex.

MFA evaluates gate position and size to optimize resin flow. It also defines placement of weld lines, areas of excessive stress, and how wall and rib thickness affect flow. Other finite element design tools include mold cooling analysis for temperature distribution, and cycle time and shrinkage analysis for dimensional control and prediction of molded-in stresses and warpage.

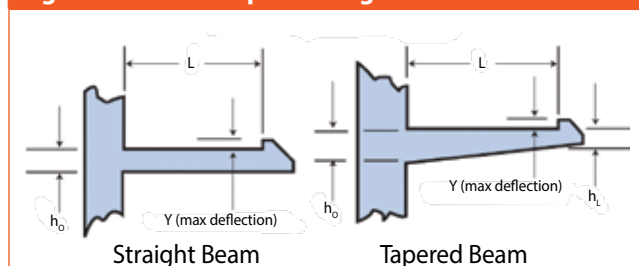
In considering the overall design task for injection-molded parts, the high mold costs means that parts should have as many functions as possible. The ability to do this depends to a great degree on the plastic used. Designers should explore several candidate materials to find the best balance to meet the needed performance in the operating environment, cost constraints and other special requirements. And finally, designers should strive to use the least amount of plastic while satisfying structural, functional, appearance and moldability requirements.

Plastics Assembly Methods – Part 5

The best way to deal with parts assembly is to avoid it entirely, instead molding multiple parts as one unit. The next best option is to minimize handling through proper design and by choosing the best joining method. This article reviews the most common ways for assembling plastic parts, which range from molded-in features like snap-fits to chemical bonding, thermal welding and mechanical fasteners.

Molded-in assembly features are fast, inexpensive and minimize errors. Many popular CAD programs make these elements easy to create during the design process.

Figure 1 • Basic Snap-Fit Designs

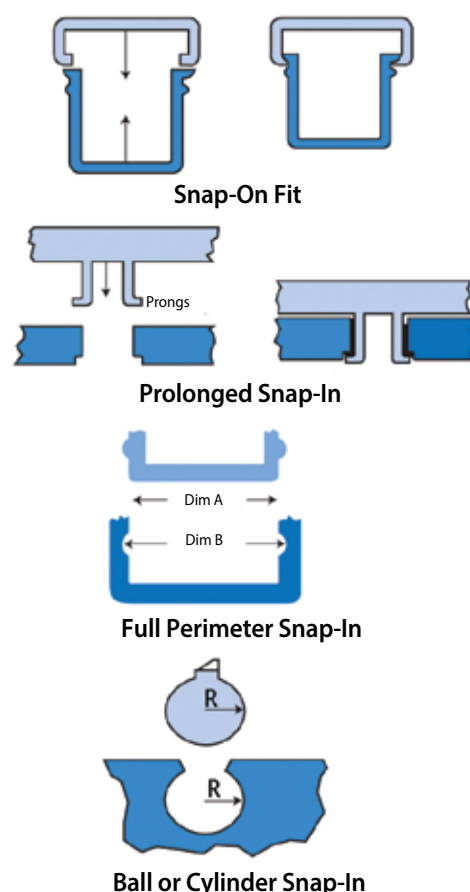


The best examples are snap-fits, which contain an element that flexes past an interference and rapidly returns to its unflexed position (Fig. 1). Snap-fits are useful for high-volume production for either one-time assembly or for release and reassembly over hundreds of cycles, such as battery compartment covers in calculators. In designing snap-fits, avoid sharp corners or structural discontinuities that increase stress. A tapered finger provides more uniform stress distribution. Snap-fits may require complex tooling, such as molds with side action that generally need undercuts. Snap-ons or snap-ins are variations of this design for round part applications. They involve small deflections of large portions of a part or even the entire part (Fig. 2).

Another type of molded-in feature that allows rapid assembly contains male and female threads for containers, caps and other parts. Internal threads need more complex tooling (e.g., unscrewing or collapsing mechanisms) than external threads, which can often be molded by splitting the mold cavity across the parting line. It usually is not practical to mold threads finer than 28 pitch.

Press-fits involve inserting a pin into a hole in a boss and usually need relatively simple tooling. They should be designed to prevent excessive hoop stress in the boss that can cause creep, stress relaxation, fractures or crazing.

Figure 2 • Typical Snap-On and Snap-In Configurations



Chemical bonding uses solvents and adhesives to join similar and dissimilar materials. It is fast, economical, requires little or no part preparation or special equipment, creates excellent liquid and gas seals, and causes no assembly stresses.

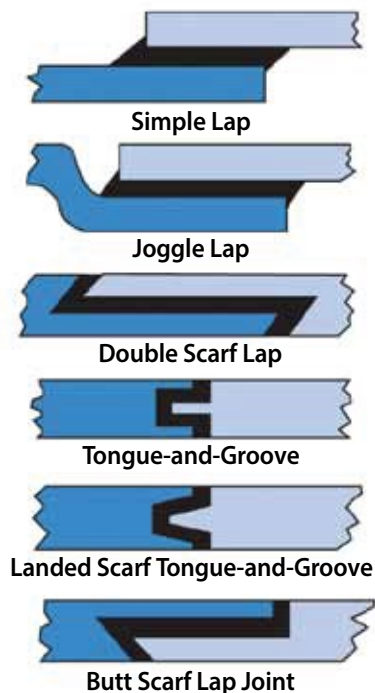
In solvent bonding, a liquid applied just before assembly dissolves the joint surfaces. This is enough for a weld to remain after the solvent evaporates. This method is limited to compatible materials that dissolve in the same solvent or solvents. The chemical resistance of many plastics, especially crystalline resins, limits this method.

In adhesive bonding, a third substance bonds a plastic to another plastic or to metal, rubber, ceramic, glass, wood, etc. Adhesives frequently used with thermoplastics include epoxy, acrylic, polyurethane, phenolic, rubber, polyester and vinyl. Cyanoacrylate adhesives are popular because they work rapidly. Many adhesives contain solvents that partially dissolve the plastic surface, which improves adhesion. Surface preparation is also critical for successful adhesion.

Many materials must be roughened or etched to eliminate overly smooth surfaces. They also may need thorough cleaning because grease, mold release compound, and other contaminants can spoil a bond.

Joints for this type of bonding can take many forms (Fig. 3) and should be tested in the actual end use. Testing should also make sure the chemicals used do not deteriorate the plastics. This effect is generally slow, often requiring long clamp times and sometimes special ovens or curing conditions. In addition, the chemicals used may be toxic, so worker protection, ventilation and solvent recovery can be issues.

Figure 3 • Joint Designs for Solvent and Adhesive Bonding



Thermal welding involves melting the bond line between two parts to form a weld. This method is a fast, economical, and safe way to weld compatible plastics having similar melt temperatures.

Ultrasonic welding is the most common thermal method for joining small and medium-sized parts of amorphous and crystalline plastic. The process normally lasts less than 2 seconds and forms a continuous, leak-proof joint that often is as strong as the base material, using equipment that applies high-frequency energy (20 to 40 KHz) directly to the interface between parts. Hygroscopic resins should be welded as soon as possible after molding, since moisture can weaken the bond.

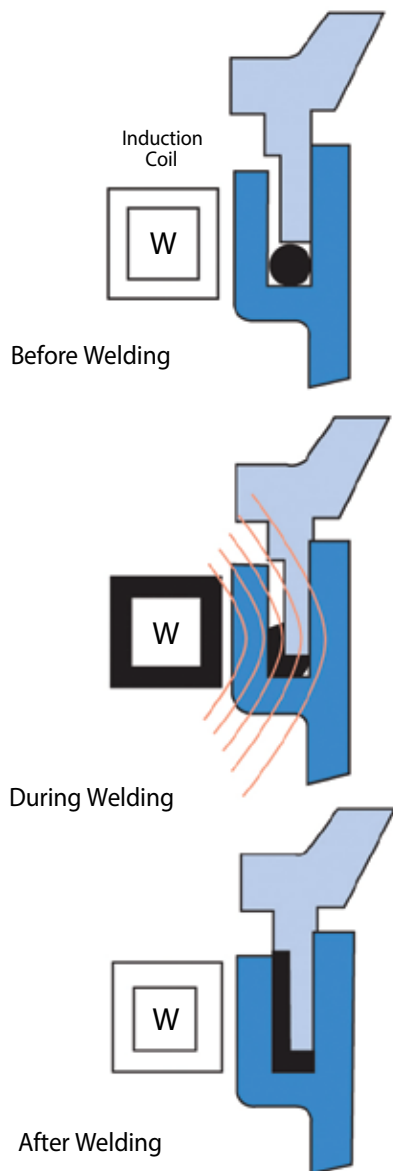
In *vibration welding*, parts are rubbed together to create frictional heat. Rubbing usually involves amplitudes of 0.1- to 0.2-in. and frequencies of 120 and 240 Hz. It creates strong joints and works best with large parts that have irregular joint interfaces.

Spin welding joins parts with circular joint surfaces using relatively simple equipment, sometimes just a drill press. It involves holding one part firmly and pressing a rotating part against it at a steady pressure. The weld usually forms in less than 3 seconds.

Radio frequency (RF) welding, or heat sealing, is widely applied to flexible films and sheets of such materials as vinyl and polyurethane. It is also used to join films and injection-molded parts. Welding occurs as heat from a strong RF field is generated by a metal die in the shape of the joint. The die also applies clamping pressure to complete the weld.

In *electromagnetic or induction welding*, a RF magnetic field excites fine, magnetically sensitive metallic or ceramic particles in a preform (Fig. 4). The heat induced in the preform, which is placed at the joint, fuses the plastic parts as they are pressed together. The preform becomes part of the weld.

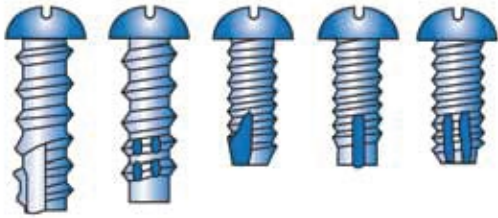
Figure 4 • The Electromagnetic Welding Process



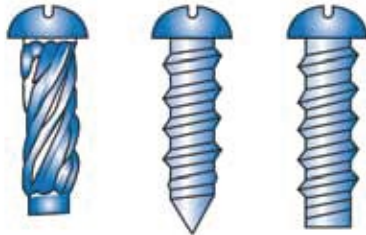
Mechanical fasteners are readily available, easy to install and usually allow for nondestructive disassembly. They often have high assembly costs and require that extra parts be stocked. Most fasteners used with metals also work with plastics, e.g., screws, bolts, nuts and lock washers. When selecting metal fasteners, be aware that these components can overstress plastic parts. This can be prevented through proper design, the appropriate fastener and torque-limiting drivers.

Threaded metal inserts permanently installed in molded bosses eliminate the need for a nut, so assembly is simplified. These can include female threads, threaded male studs, locating pins and bushings. Ultrasonically installed inserts are especially popular, because the surrounding plastic melts around the insert, making it strong and relatively free of stress.

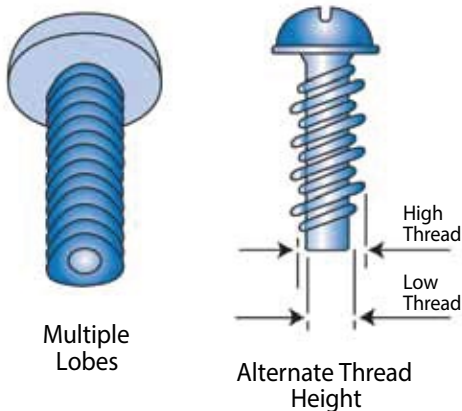
Self-tapping screws are also popular (Fig. 5). Cutting style screws are best because they act like thread-cutting taps and remove material, without generating high hoop stresses as do forming-style screws. Screws with multiple lobes and those with alternating thread heights offer excellent holding power and reduced stress levels. When using self-tapping screws, size the hole properly to avoid excessive hoop stress in the boss. Also, make the hole deep enough so the screw does not bottom out, and the wall of the boss thick enough to resist the stress created by the screw.

Figure 5 • Screws Often Used with Plastics

Thread-cutting Screws



Thread-forming Screws



Metal rivets give fast, permanent assemblies. They should have large heads to spread the load. Form the rivet against the metal part of an assembly, or against a metal washer if both parts are plastic.

Stamped sheet metal fasteners provide light-duty threads or push-on assemblies. Push-nuts, for example, are simply pressed onto plain, molded plastic studs or bosses in permanent assemblies. They are easy to install, inexpensive and vibration-proof. Another such fastener, boss caps (cup-shaped parts pushed onto a plastic boss), add partial metal threads for self-tapping or sheet metal screws, and reinforce the boss against the expansion forces of the screw.

Molded plastic screws, rivets and other similar fasteners can be used for light-duty plastic assemblies, especially where appearance is important, such as to attach trim and faceplates.

The role of the plastic

Manufacturers clearly have many choices when it comes to joining plastic parts. Selecting among them can be an involved decision that balances design, material usage, assembly considerations, and other factors to gain the greatest efficiency at the lowest cost. The choice can be even more complex, because many plastic units need more than one assembly method, for example, ultrasonic insertion of a threaded insert for a machine screw, adhesives that create an airtight seal in a bolted assembly, and solvents that strengthen a snap-fit.

ENGINEERED MATERIALS
celanese.com/engineered-materials

Engineered Materials

- Celanex® thermoplastic polyester (PBT)
- Hostaform® and Celcon® acetal copolymer (POM)
- Celstran®, Compel® and Factor® long fiber reinforced thermoplastic (LFRT)
- Celstran® continuous fiber reinforced thermoplastic (CFR-TP)
- Fortron® polyphenylene sulfide (PPS)
- GUR® ultra-high molecular weight polyethylene (UHMW-PE)
- Impet® thermoplastic polyester (PET)
- Riteflex® thermoplastic polyester elastomer (TPC-ET)
- Thermx® polycyclohexylene-dimethylene terephthalate (PCT)
- Vandar® thermoplastic polyester alloy (PBT)
- Vectra® and Zenite® liquid crystal polymer (LCP)

Contact Information

Americas

8040 Dixie Highway, Florence, KY 41042 USA

Product Information Service

t: +1-800-833-4882 t: +1-859-372-3244

Customer Service

t: +1-800-526-4960 t: +1-859-372-3214
e: info-engineeredmaterials-am@celanese.com

Europe

Am Unisys-Park 1, 65843 Sulzbach, Germany

Product Information Service

t: +(00)-800-86427-531 t: +49-(0)-69-45009-1011
e: info-engineeredmaterials-eu@celanese.com

Asia

4560 Jinke Road, Zhang Jiang Hi Tech Park
Shanghai 201203 PRC

Customer Service

t: +86 21 3861 9266 f: +86 21 3861 9599
e: info-engineeredmaterials-asia@celanese.com