Progress beyond

Amodel® PPA

Processing Guide

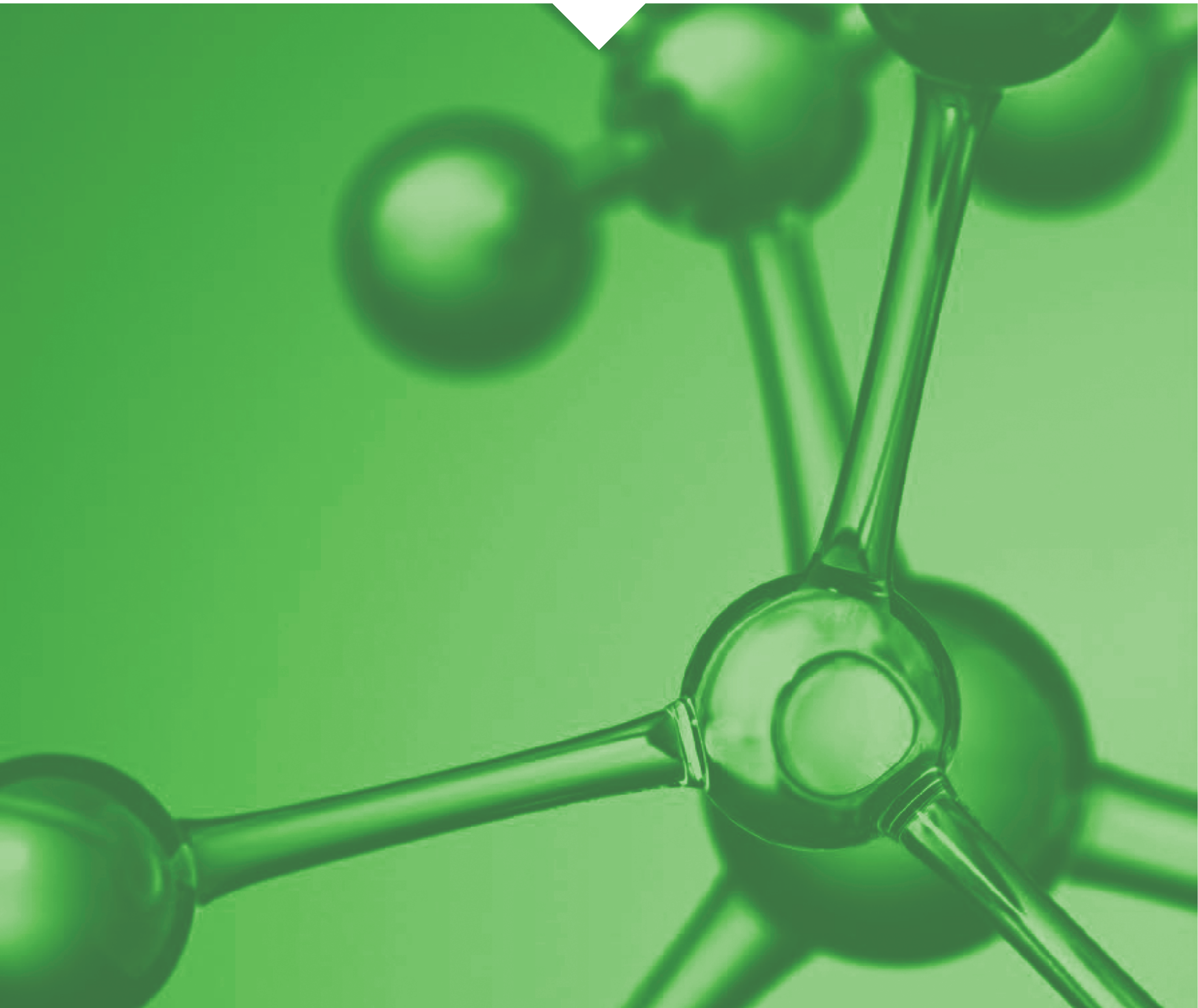


Table of Contents

Introduction

Amodel® Polyphthalamide (PPA)

Amodel® PPA is a family of semi-crystalline engineering polymers that bridges the cost-performance gap between traditional engineering thermoplastics – such as polycarbonate, polyamides, polyesters, and acetals – and higher-cost specialty polymers – such as liquid crystal polymers, polyphenylene sulfide, and polyetherimide.

Amodel® resin has excellent mechanical properties – strength, stiffness, fatigue and creep resistance – over a broad temperature range. Glass-reinforced grades provide higher stiffness, strength, and creep resistance at elevated temperatures for structural applications. Mineral-filled resins offer enhanced dimensional stability and flatness. Some of these grades can be plated and epoxy-coated. Impact-modified grades can provide significantly improved toughness – comparable to the super-tough nylons – but with much higher strength and stiffness across a broad humidity and temperature range.

Unreinforced grades of Amodel® resin are formulated for general-purpose injection molding and extrusion applications that require high surface gloss, lubricity, low warpage and toughness, along with a high level of mechanical performance at elevated temperature. General characteristics of the Amodel® resin families are shown in Table 1.1.

Table 1.1
Amodel® product families

Product Family	General Descriptions
Glass-reinforced	These resins cost-effectively maximize strength and stiffness properties and are typically used in applications requiring structural integrity at elevated temperatures.
Mineral/glass-filled	These materials offer stiffness and thermal performance approaching glass-reinforced products with better dimensional stability and lower cost.
Flame-retardant	These grades are specially formulated for flame retardance, UL 94 rated V-0 at 0.8 mm (0.032 in.), UL 94 rated 5V at 1.6 mm (0.062 in.), and are vapor phase and IR reflow solderable. Both glass-reinforced and mineral/glass-filled versions are available.
Unreinforced, impact-modified	These "super-tough" resins have notched Izod values greater than 800 J/m (15 ft-lb/in.) and combine superior impact performance with good strength and stiffness at both low and high temperatures.
Toughened glass-reinforced	These grades offer an excellent balance between stiffness, toughness or elongation, high thermal performance, and good dimensional stability. They also have excellent chemical resistance and good friction and wear properties.
Mineral filled	Mineral-filled grades provide an excellent surface finish for vacuum metallizing, painting, and chrome plating applications.

Nomenclature

The Amodel® PPA grade nomenclature system outlined in Table 1.2 is designed to communicate important compositional information. There are a few exceptions to this nomenclature, such as ET for extra tough grades and HFZ for high-flow grades.

To illustrate, consider Amodel® AFA-6133 V0 BK324 which is a flame-retardant product (AFA) that uses an A-600x base resin containing 33 % glass fiber by weight (6133). It has a UL94 rating of V0 at 0.8 mm (0.032 in.) and is pigmented black with colorant formula 324.

Table 1.2
Amodel® resin nomenclature system

Position	Characteristic	Meaning/Example
1st letter	Product family	A = Amodel®
		E = extra
Next letter(s)	Optional descriptor	E = electrical/electronic
		F = flame retardant
		P = paintable/plateable
		S = thick-wall parts (> 3 mm)
		T = toughened
-	Hyphen	
1st digit	Base resin	1 = A-100x base resin
		4 = A-400x base resin
		6 = A-600x base resin
		8 = A-800x base resin
		9 = A-900x base resin
		Bios R1 = R1-00x biosourced base resin
2nd digit	Filler or reinforcement type	0 = unfilled
		1 = glass
		2 = mineral A
		3 = mineral A + glass
		4 = mineral B
		5 = mineral B + glass
		6 = carbon or graphite fiber
		9 = glycol resistant
		Bios R1 = R1-00x biosourced base resin
3rd and 4th digits	Filler or reinforcement amount	33 = 33 % by weight
		45 = 45 % by weight, etc.
Space		
Next letter(s)	Suffix	HN = heat stabilized, not lubricated
		HS = heat stabilized
		HSL = heat stabilized and lubricated
		L = lubricated, not heat stabilized
		NL = neither lubricated nor heat stabilized
		V0 Z = UL 94 V0 at 0.8 mm (0.032 in.)
		HFFR = halogen-free flame retardant UL 94 V0 at 0.8mm (0.032in.)
Space		
Next 2 letters	Color code	NT = natural, unpigmented
		BK = black
		WH = white, etc.

Drying the Resin

Amodel® resins are hygroscopic. When exposed to humid air, they will absorb moisture. While the effects of moisture absorption on the properties and dimensions of molded parts are minimal, the effects on the molding process can be significant.

Amodel® resins are shipped in moisture-resistant packages, dried to moisture levels of 1,500 ppm (0.15 %) or less. While this level of moisture may be acceptable for some processes, it is generally recognized that additional drying to levels less than 1,000 ppm (0.10 %) is beneficial. It is important that the moisture level not only be low, but that it be relatively constant throughout the molding run to ensure process and part uniformity.

Drying Equipment

Typically, the resin is dried in hopper dryers mounted over the molding machine. The dryer system should be desiccated and capable of maintaining a dew point of – 30 °C (– 22 °F). Single-bed desiccant systems are usually adequate as long as the desiccant is replaced as necessary. Dual-bed systems are generally more reliable because they allow one bed to be regenerated while the other is drying. Even better are rotating-bed systems, because they continuously supply a source of fresh desiccant. Tray drying in air circulated ovens is practical only for short runs.

Drying Time and Temperature

The drying time and temperature will depend on the moisture content of the resin as received, the size of the hopper dryer used, and the throughput of the molding process. To determine the proper drying temperature, the capacity of the hopper dryer (in kilograms or pounds) should be divided by the resin consumption (in kilograms or pounds per hour). This will give the dryer residence time (in hours). Once the residence time has been established, use Table 2.1 to determine the drying temperature for Amodel® resins that are placed into the dryer directly out of the sealed shipping container.

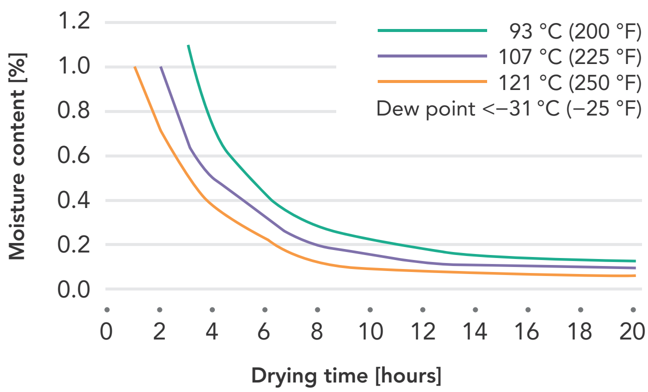
Table 2.1
Residence time vs. drying temperature

Dryer Residence Time [Hours]	Drying Temperature [°C (°F)]
8 or more	79 (175)
6	93 (200)
4	107 (225)
2	121 (250)

Amodel® resin that has been exposed to the atmosphere in an opened container for more than 24 hours will absorb enough moisture to require drying for at least 8 hours at 121 °C (250 °F).

Figure 2.1 shows the time required to dry "wet" Amodel® resin at various temperatures. The graph shows that 93 °C (200 °F) is not very effective and 107 °C (225 °F) or 121 °C (250 °F) is preferred. Drying at temperatures greater than 121 °C (250 °F) can cause natural colored resins to darken.

Figure 2.1
Drying times for wet resin



Moisture levels can be measured with conventional gravimetric analysis equipment. Test conditions should be 170 °C (338 °F) for 10 minutes.

Maintain Resin Dryness

When loading resin into a hopper dryer from sealed bags, the bags should be opened individually and immediately loaded. Keep bags sealed until they can be loaded into the hopper. Do not open several bags and allow an air conveyor system to transfer the resin to the hopper as needed. This will allow the resin to pick up moisture, and may result in processing problems.

When loading resin from a 500 kg (1,100 lb) box, a small cut should be made in the foil liner through which the air conveyor "wand" is placed. Reseal the foil around the top of the wand. If only part of the container is to be used, reseal the foil liner tightly.

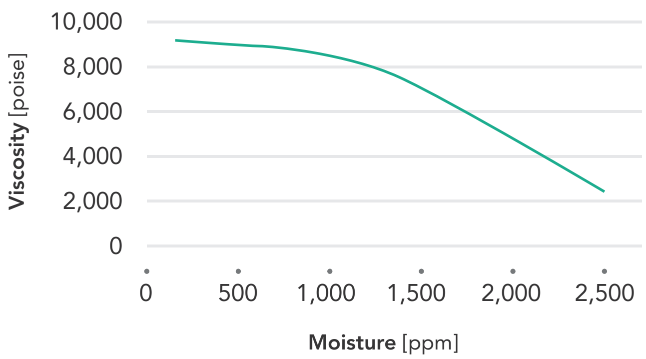
Dried air should be used to convey resin. The use of "shop air" can introduce moisture into the resin.

Consequences of Inadequate Moisture Content Control

Molding Amodel® resins containing excessive moisture will affect the molding process and may affect molded part properties. The severity of these effects will depend on the moisture level, the resin grade, and the particular tool and part.

The amount of moisture in the resin during processing determines the melt viscosity. As shown in Figure 2.2, higher moisture content results in lower melt viscosity because the polymer is losing molecular weight.

Figure 2.2
Moisture content affects melt viscosity



At the same time, additional volatiles are being generated, thereby increasing the load on the vents. In extreme cases, the molded parts will exhibit burn marks and have poor mechanical properties. Even relatively small variations in the moisture content can affect the viscosity enough to cause dimensional and aesthetic problems.

If moisture content is not maintained under rigid control, it will be difficult to establish a stable molding process.
If the moisture content varies and parts are molded using a constant hold pressure and hold time, the part density will vary. The change in part density will be manifested as changes in part dimensions and part weight. In extreme cases, either flash or short shots may occur.

Molding machine indicators can be used to determine if the viscosity (moisture level) is changing, often before the change can be seen as an unacceptable molded part. These indicators are:

- **Hydraulic pressure at transfer position**

  If this pressure decreases, it is an indication that the resin is becoming lower in viscosity. Less pressure is required to achieve the set injection velocity. This indicates increasing moisture levels.

- **Final cushion position**

  When the cushion decreases at a constant hold pressure, the resin is becoming lower in viscosity, most likely as a result of increasing moisture content.

Many molding machines have the capability to monitor one or both of these parameters. Often it is possible to set limits and have an alarm signal that the parameter is deviating from the set point.

Processing Equipment

Amodel® resins can be processed on conventional injection molding equipment. Because Amodel® resins are hygroscopic, the press should be equipped with a hopper dryer. Specific recommendations regarding drying equipment can be found in “Drying Equipment” on page 5.

Injection Molding Machine Considerations

The injection molding machine should be capable of controlling the injection portion of the process by velocity and screw position. Most machines made after 1980 have this capability. Older machines can be retrofitted with linear transducers and electronic controllers. It is desirable if the press controls include the capability to monitor and allow alarms to be set for the following process variables:

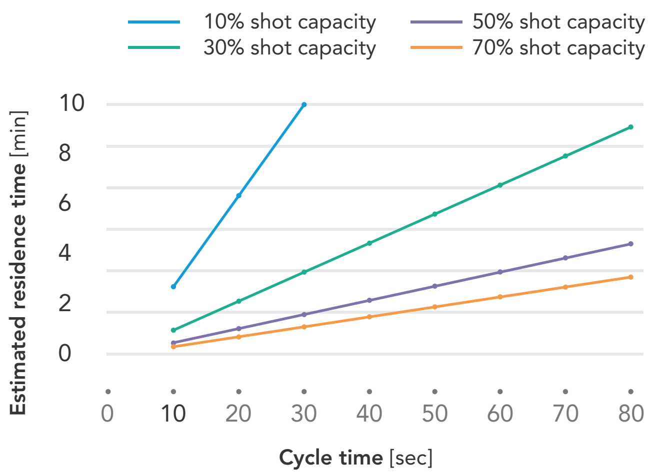
- Injection time
- Cushion
- Hydraulic pressure at transfer position

A screw with a compression ratio of 2.5 – 3.5 to 1 and a length to diameter (L/D) ratio of 18 – 22 is recommended.
A reverse taper nozzle is also recommended.

Choose a press that provides a clamp pressure of at least 8 kpsi, 545 bar, or 55 MPa (4 tons per square inch) of projected part area and a shot capacity 1.5 to 3.3 times the shot size. A press of this size minimizes residence time in the barrel, because each shot will use 30 to 70 % of the shot capacity. Figure 3.1 provides a tool for estimating residence time from cycle time and shot capacity utilization.
It is generally recommended that the estimated residence time should be no more than six minutes. Excessive residence time can result in resin degradation which leads to drooling, flash, plate out, burn marks, and poor mechanical properties of the molded part.

The injection molding machines need to be properly maintained. The barrel to screw clearance should be monitored regularly to make sure it still conforms to the manufacturer’s specification. The check ring should be checked routinely for excessive wear.

Figure 3.1
Effect of shot capacity utilization and cycle time on residence time[1]



[1] *Recommended residence time is six minutes or less*

Mold Temperature Control

Depending on the grade of Amodel® resin being processed, either a water or oil heater will be required to maintain the proper mold temperature. Amodel® A-1000 series grades require mold temperatures of 135 °C (275 °F) and therefore require oil circulation units. All other Amodel® resins can use water units. The temperature controller should be capable of maintaining heat transfer fluid temperature to +/- 2 °C (3 °F). The pump should have adequate volume capacity to create turbulent flow of the fluid through the mold.

The hoses leading from the pump to the tool should be of adequate size and covered with a braided metallic sheath. Thermal insulation on hoses will prevent heat loss from the system. Seals and O-rings in the system must withstand the temperatures required.

Alarms on the mold temperature system should include high and low temperature and high pressure. An automatic high-pressure shutoff is recommended to protect molding personnel.

Processing Conditions

Starting Point Machine Settings

Table 4.1 provides the machine settings for setting up the injection molding machine to mold the various Amodel® resin grades.

Barrel Temperatures

Although the given barrel temperatures may be adjusted, the goal is to achieve the melt temperature shown in the table.

The actual temperature settings will depend on the shot capacity to shot size ratio. If a large portion of the resin inventory will be used for each shot, the rear temperatures should be increased. If only a small portion of the resin inventory will be used, the middle temperatures should be reduced. Extended residence time will result in degradation of the polymer and unacceptable molded part properties. Holding the melt at temperatures above 350 °C (660 °F) may result in polymer degradation and should be avoided.

The nozzle temperature should be adjusted to compensate for freeze-off or drool. The nozzle temperature should be set as high as possible to minimize the cold slug, yet low enough to prevent drool.

The mold temperature controllers should be set to maintain the mold temperature at or above the minimum temperature shown in Table 4.1. Higher temperatures can be used to improve surface appearance or allow for easier fill of the mold cavity, but cycle times may be lengthened. Parts with very thin-wall sections may require higher mold temperatures to achieve optimal crystallinity.

Molding Cycle Settings

For this discussion, the molding process can be envisioned as three distinct steps:

- Polymer injection or mold filling
- Packing and holding or part densification
- Cooling and screw recovery

Polymer Injection or Mold Filling

The mold-filling step is the portion of the cycle at high or injection pressure; it ends when the pressure is reduced to the lower holding pressure. This step of the process can be controlled using a variety of process control methods.
These include:

- Controlling the time at a constant injection pressure
- Controlling the pressure with a cavity pressure sensor
- Controlling the injection rate until the screw position reaches a set point

The recommended method is to control the injection rate (designated injection velocity), and to switch to holding pressure at a set screw position when using a press with this capability. The advantage of this method is that a controlled volume of resin is being delivered to the cavity at a specified rate. Generally, fast injection rates are recommended.

To use this method, the proper screw position for switching from injection pressure to hold pressure must be determined. This position should be such that the part is about 95 % filled; the remainder of the part should be filled and packed out with holding pressure. This method should allow any remaining gas in the tool to be vented without burning. An efficient method for determining the correct screw position for the transition to holding pressure is as follows:

1. Set holding (low) pressure to zero.
2. Set screw forward (injection) velocity to 5 to 13 cm/sec. (2 to 5 in./sec.).
3. Mold several shots and observe the parts.The objective is to find the screw position at which the part will be almost filled but not packed out.
4. If the parts appear completely filled, move the transition point numerically higher (smaller shot volume).
5. If the parts are very short, adjust the transition point numerically lower (larger shot volume).

Table 4.1
Starting point molding conditions

Parameter	A - 1000 AS - 1000 Series	A - 4000 AS - 4000 Series	A - 6000 Series	ET - 1000 AT - 1000 Series	HFFR-4000 Series	AT - 1100 Series	AT - 6100 Series	A - 8000 AS - 8000 Series	A - 9000 AS - 9000 Series	BIOS Series
Drying Instructions[1]										
Drying temperature [°C (°F)]	120 (248)	120 (248)	120 (248)	110 (230)[2]	120 (248)	110 (230)[2]	110 (230)[2]	120 (248)	120 (248)	120 (248)
Drying time [hours]	4	4	4	4	4	4	4	4	4	4-6
Molding Conditions										
Target melt temperature [°C (°F)]	320 – 345 (608 – 653)	330 – 345 (626 – 653)	325 – 340 (617 – 644)	320 – 330 (608 – 626)	330–340 (626–644)	320 – 330 (608 – 626)	320 – 330 (608 – 626)	340-360 (644-680)	340-365 (644-689)	325-345 (617-653)
Barrel temperatures [°C (°F)]										
Rear zone	310 (590)	315 (599)	310 (590)	300 (572)	310 (590)	310 (590)	310 (590)	315 (599)	315 (599)	315 (599)
Middle zone	315 (599)	320 (608)	315 (599)	310 (590)	320 (608)	315 (599)	315 (599)	325 (617)	325 (617)	325 (617)
Front zone	320 (608)	325 (617)	320 (608)	315 (599)	325 (617)	320 (608)	320 (608)	335 (635)	340 (644)	330 (626)
Nozzle temperature[3]	320 (608)	325 (617)	320 (608)	315 (599)	325 (617)	320 (608)	320 (608)	335 (635)	340 (644)	330 (626)
Mold temperature	> 135 (275)	> 80 (176)	> 80 (176)	< 90 (194)	> 80 (176)	> 135 (275)	> 80 (176)	>145 (293)	>170 (338)	>145 (293)
Injection speed	Moderate to high	High	High	Moderate	High	Moderate	Moderate	Moderate to high	Moderate to high	Moderate to high
Fill time [seconds]	1-3	1-2	1-2	2-4	0.5-2	1-3	1-3	1-3	1-3	1-3
Injection pressure [bar (kpsi)]	700 – 1,500 (10 – 22)	700 – 1,500 (10 – 22)	700 – 1,500 (10 – 22)	700 – 1,500 (10 – 22)	600 – 1,500 (9 – 22)	600 – 1,500 (9 – 22)	600 – 1,500 (9 – 22)	700-1500 (10-22)	700-1500 (10-22)	600-1500 (9-22)
Hold pressure [bar (kpsi)]	350 – 800 (5 – 12)	350 – 800 (5 – 12)	350 – 800 (5 – 12)	350 - 800 (5 – 12)	350 – 800 (5 – 12)	350 – 800 (5 – 12)	350 – 800 (5 – 12)	350-800 (5-12)	350-800 (5-12)	350-800 (5-12)
Hold time[4] [seconds/mm]	3	1	1.5	3	1	3	1.5	3	2	3
Back pressure [bar (psi)]	< 5 (72)	< 5 (72)	< 5 (72)	< 5 (72)	< 5 (72)	< 5 (72)	< 5 (72)	<5 (72)	<5 (72)	<5 (72)
Screw speed [m/s (rpm)]	< 0.3 (150)	< 0.3 (150)	< 0.3 (150)	< 0.3 (150)	< 0.3 (150)	< 0.3 (150)	< 0.3 (150)	<0.3 (150)	<0.3 (150)	<0.3 (150)

(1) Air used for drying must have a dew point below – 30 °C (– 22 °F)
(2) Drying these grades above 110 °C (230 °F) can result in pellet clumping
(3) Adjust downward if drooling occurs
(4) Calculate hold time in seconds by multiplying seconds/mm by maximum part thickness in mm

Alternatively, the cavity pressure can be used to determine the correct transition set point. The cavity pressure will increase when the part is filled. Therefore, the transition set point can be adjusted by increasing it until a pressure increase is indicated, then reducing it incrementally until no cavity pressure increase is observed. The success of this method is dependent on the correct placement of the pressure transducers in the cavity. Transducers should be placed in the last-to-fill area of the cavity.

This transition set point will produce a part that is nearly filled but not packed out. Once the appropriate transition set point has been determined, packing or holding pressure should be applied to complete the filling and pack the part out. Where velocity and position controls aren't available, injection pressure and timers should be adjusted to fill the part quickly, usually in 1 to 2 seconds.

Packing and Holding

The filling of the mold cavity with polymer is completed during the packing and holding portion of the process. The continued application of pressure achieves maximum part density. The variables are holding pressure and time.

During the injection portion of the cycle, the injection rate for the particular grade of Amodel® resin was set using the value shown in Table 4.1. The pressure required to achieve that rate should be observed and the holding pressure should initially be set to one-half of that value. The holding pressure should be as high as possible without flashing the cavity to ensure optimal densification of the molded part.

The hold time will depend upon a number of factors, including the thickness of the part, the gate dimensions, the mold temperature, and the resin crystallization rate. Using the multiplier provided in the setup chart to determine the initial hold time, based on the maximum part thickness, will give an initial estimate of the appropriate hold time. Actually, the best way to determine the hold time is experimentally. Weigh parts until increasing the hold time does not increase the part weight. Lack of full densification in molded parts may result in performance problems including warpage, uneven shrinkage, and/or cracks.

Cooling

During the cooling step, the part is getting stiff and strong enough to be ejected without warpage and/or deformation from the ejector pins. Simultaneously, the screw rotates, picking up material for the next cycle. The speed at which the screw rotates should be between 100 and 200 rpm, with just enough back pressure to ensure a uniform shot size. The rate at which the screw should travel is given in the setup chart.

If long cooling times are used, the polymer's residence time in the barrel can become excessive, resulting in polymer degradation. A screw delay (added time interval between the end of pack/hold and screw recovery) can help to minimize degradation. If ejection problems occur, check for proper draft on the tool as well as for undercuts or lack of polish on ejection surfaces.

Mold Temperature and Control

It is necessary to maintain a uniform temperature across the surface of the mold. This is generally accomplished by circulating a heat transfer medium through a series of channels cut into the tool. This heat transfer medium is usually a blend of water and ethylene glycol when the mold temperature is to be 93 °C (200 °F) or below, or an oil when the mold temperature needs to be above 93 °C (200 °F).

Consideration must also be given to mold features, such as cores that may be thermally isolated and require additional thermal control. There are several methods of handling these situations. The easiest method is to cut an additional cooling channel in the cavity detail and supply cooling medium through the length of the core. In situations where the detail is too small to cut such a channel, a thermally conductive pin made of copper-beryllium may be placed inside the core to facilitate heat transfer. The consequence of not controlling hot spots in a tool will include extended cycle times, as well as parts sticking in the tool.

Insert Molding

The practice of molding inserts into parts has become widespread. These inserts may include conductors, lead frames, bearings, bushings, torque limiters, and structural elements. There are several considerations when incorporating inserts into a molded part.

The insert must be able to withstand the temperature of the molten plastic. Bearing lubricants and platings must be capable of withstanding high temperature. Tin platings, for example, are not generally suitable for insert molding with Amodel® resin because the melting point of tin is below the processing temperature of the resin.

The inserts must be held in place by some means to ensure that the force of the injected plastic does not shift their position to an undesirable location. This can be accomplished by using locating pins on both halves of the tool. Inserts that are meant to be encapsulated in the part should be held in place by locating pins that are retracted during injection. The timing of the locating pin retraction should be experimentally determined and based on screw position for consistency. The insert must also be held in the tool by some means to ensure that it is not dislodged by the action of the mold closing. This is more of an issue in horizontal clamping machines than in vertical clamping machines.

Although the shrinkage of the plastic around an insert is generally sufficient to hold the insert in place, it is often desirable to place some detail on the insert surface to create a mechanical bond. Any angular detail on a conductor will accomplish this and a knurled surface on a straight insert will ensure a tight bond. When molding with large metallic inserts, it is sometimes advantageous to preheat the insert. Heating the insert will thermally expand it and help prevent cracking of the part, because the resin and insert will cool and shrink at a more similar rate.

Mold Releases

The use of mold release sprays or additives with Amodel® resins is not recommended. Mold releases can cause a variety of problems, such as surface blemishes, adhesive bonding difficulties, electrical property issues, mold deposit buildup, and process inconsistencies. Mold release should not be necessary, except perhaps for starting up a new mold.

If part ejection is difficult, the mold design and processing conditions should be reviewed. Make sure that there are sufficient ejector pins with enough area. Also make sure that the mold has sufficient draft. Inspect the mold for damage that may be causing an undercut. Verify that the cycle settings include sufficient cooling time.

Regrind

Since Amodel® resin is thermoplastic, sprues and runners can be ground, mixed with unprocessed (virgin) resin, and reprocessed into useful molded articles. This ground material is commonly referred to as regrind. Properly done, the use of regrind is an economical and environmentally sound practice. End-use requirements and specifications should be considered when introducing regrind into a process.

Material Suitable for Regrinding

In general, sprues and runners generated in the molding process comprise the majority of the regrind source. Certain non-conforming molded parts may also be used, provided they don't contain any foreign objects such as molded-in inserts and they aren't contaminated with oil, grease, or dirt. Don't use degraded parts or sprues and runners that have burn marks.

Amount of Regrind

Ideally, the amount of regrind used should equal the amount generated in the molding process. It is important that the level of regrind be consistent.

For example, if the sprue and runners comprise 20 % of the molded shot weight, then 20 % is the amount of regrind that should be used. While a higher level, say 25 %, may still provide useful parts, the process doesn't generate enough regrind to maintain 25 % on a consistent basis. If regrind levels vary, the molding process and the quality of the molded parts may also vary.

Generally, regrind levels of 25 % are widely accepted. In some cases, even higher levels will provide acceptable parts. As a practical matter, regrind levels should usually not exceed 50 %. Extremely high levels of regrind may result in molded part quality issues.

Regrind studies have been conducted to demonstrate that the quality of parts produced with moderate levels of regrind is acceptable. These studies are summarized in Figures 6.1 through 6.3. These graphs show that both tensile strength and impact resistance are maintained when 25 % regrind was used both with a structural glass-reinforced grade and a flame-retardant grade of Amodel® resin.

Figure 6.1
Effect of 25 % regrind on tensile strength of Amodel® AS-1133 HS

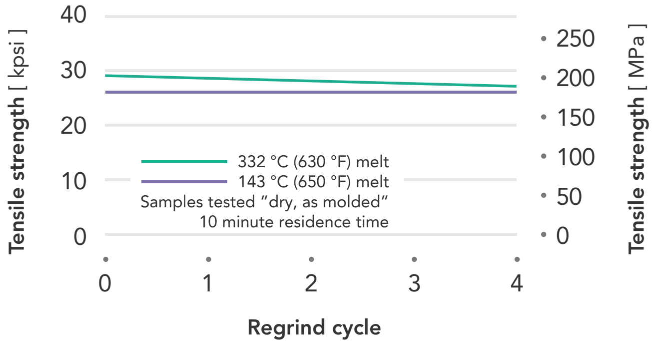


Figure 6.2
Effect of 25 % regrind on tensile strength of Amodel® AS-1145 VO

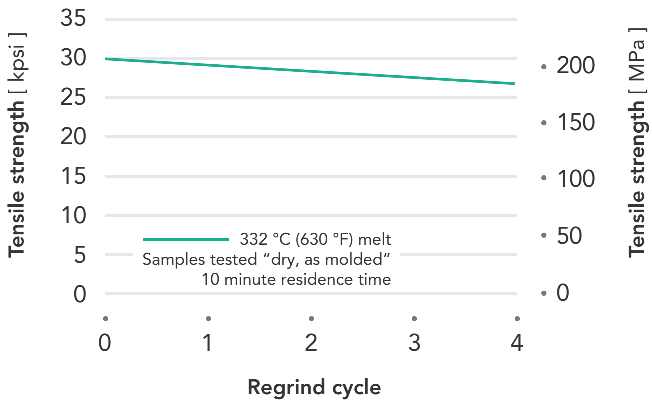
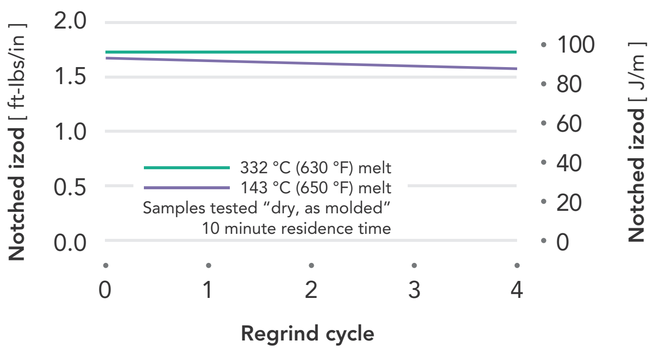


Figure 6.3
Effect of 25 % regrind on Izod impact of Amodel® AS-1133 HS



Molding Process Effects

Since the regrind material has been previously processed through the molding machine, it may have a slightly lower molecular weight. This may manifest itself in the molding process as a lower melt viscosity. While this effect is not always noticeable, it may be necessary to adjust molding conditions to compensate. A small decrease in second stage, or holding, pressure is usually appropriate.

The potential for reduction in melt viscosity is an important reason why consistent amounts of regrind should be used. If the regrind level varies over time, it is possible that the variations in the melt viscosity will require numerous molding machine adjustments to maintain molded part quality.

If the regrind, either before or after grinding, is allowed to absorb atmospheric moisture and isn't dried properly, a significant, unmanageable drop in melt viscosity will occur. Regrind should be handled and dried in the same manner as virgin resin. Flash on parts, nozzle drool, and/or excessive out gassing may be signs of wet resin or regrind.

Regrind Usage Schemes

Continuous regrind addition

Some molders believe that regrind should be discarded after a certain number of passes through the molding machine. While that may be prudent with some materials, it isn't necessary when using Amodel® resin and a proper regrind usage plan. There are two ways of analyzing this process: the experimental and the theoretical. From the experimental standpoint, studies have shown that continuous use of regrind at 25 % has no significant deleterious effect on part properties (Figures 6.1 through 6.3).

From the theoretical standpoint, consider that the amount of regrind from the previous generation is reduced by the fraction of regrind being used. For example, consider a situation where 25 % regrind is being used.

- First pass, 75 % virgin is mixed with 25 % regrind, so 25 % of the resin has been through the molding machine once before
- Second pass, 75 % virgin is mixed with 25 % regrind from the first pass, so 6.25 % has been through the machine twice
- Third pass, 75 % virgin is mixed with 25 % regrind from the second pass, so 1.56 % has been through the machine three times
- Fourth pass, 75 % virgin is blended with 25 % regrind from the third pass, so only 0.39 % has been through the machine four times
- Fifth pass, the amount of regrind being molded for the fifth time is less than 0.1 %

Therefore, the constant dilution with virgin resin allows regrind to be continuously used.

Cascading regrind use

Don't use the cascading method. In this method, all of the regrind is collected and run at 100 %. The regrind from this run is used again at 100 % and the regrind from that run is then discarded. This method will produce three sets of parts (virgin, first generation, and second generation) with three differing sets of properties. It also requires three different sets of molding conditions since the viscosity of the materials will be different. This procedure is not recommended.

Part Specifications and Qualification

Sometimes a part drawing will specify "100 % Virgin" or "No regrind allowed". In these cases it is advisable to make the end user aware of the economic benefit of using regrind. When prototyping parts, it is a good idea to provide samples containing the desired level of regrind. This will allow the customer to perform testing on parts that will be representative of production. If parts containing regrind are not provided until the part is fully qualified, it may be necessary to perform additional time-consuming and expensive testing to verify the acceptability of regrind.

Optimizing Regrind Use

Remove contamination

Oil, grease, and dirt on parts cannot be tolerated in the regrind stream. Foreign materials must not be allowed to contaminate the regrind. If the identity of a material is not certain, do not risk using it. A very small amount of undesirable material can contaminate a large quantity of virgin resin. The grinder must be cleaned thoroughly when changing materials. Magnets placed above the feed throat of the molding machine can remove ferrous metals, such as chipped pieces of grinder blades.

Classify by size

To allow good mixing of the regrind and virgin resin, the regrind particle size should be as close to the virgin resin pellet size as possible. Oversize pieces of regrind may not feed well and bridge in the hopper. Dust from the grinder tends to cling to the sides of the hopper and require high maintenance on the vacuum loader filters. An excellent method of controlling particle size is to use a vibratory dual screen unit with an upper 4 mesh screen and a lower 40 mesh screen. Particles that cannot pass through the 4 mesh are recycled to the grinder. Fine particles that pass through the 40 mesh screen should be discarded. Material that stays between the two screens is suitable for regrind.

Dry

As noted earlier, the regrind must be dry for proper molding. A handling method that eliminates the need for special drying procedures is to use the regrind in a "closed loop" system as shown in Figure 6.4. In such a system, the sprue and runner are conveyed to the grinder and size classifier as soon as they are generated. The regrind is then placed back into the hopper dryer before it can pick up any atmospheric moisture.

Figure 6.4
Closed-loop regrind system diagram

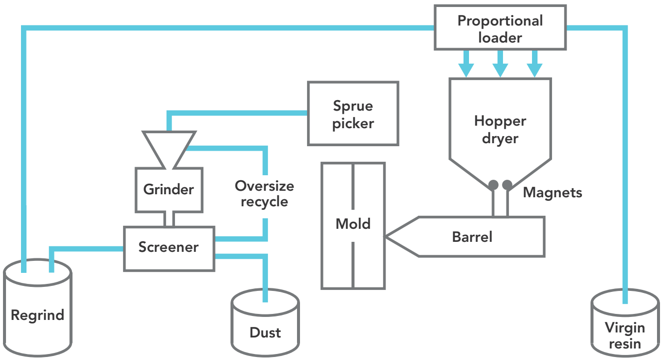
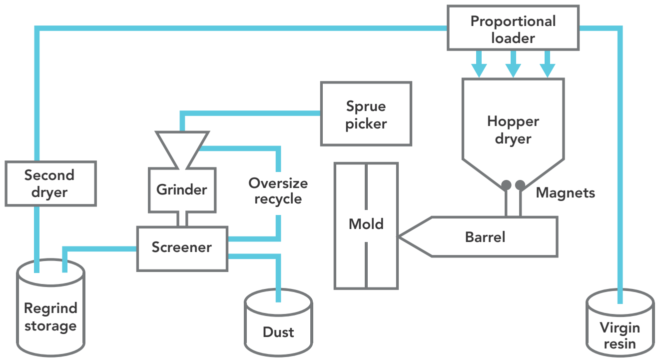


Figure 6.5
Open loop regrind system diagram



Blend completely with virgin resin

In order to maintain shot-to-shot (and part-to-part) consistency, the regrind must be mixed efficiently with the virgin resin. If done off-line, this may be accomplished with drum tumblers or a similar method. Figure 6.4 shows a proportional loader on the hopper dryer which meters volumetric amounts from the virgin and regrind sources. The loader has a controller which can vary the time that each source is drawn from, thereby adjusting the percentage of regrind as necessary.

It is not acceptable to achieve a 25 % regrind level, for example, by placing 75 pounds of virgin resin in the hopper, then placing 25 pounds of regrind on top. In that case, one would be producing 75 % of their parts with 100 % virgin and 25 % of their parts with 100 % regrind.

Use constant amounts

To maintain a stable process, the same amount of regrind should be used all the time. Ideally, the amount used will equal the amount generated. As discussed earlier, if the regrind level changes over time, it is possible that the process and the molded part quality may also change. The proportional loader will draw from both the regrind and virgin sources and control the regrind amount at the desired ratio.

Purging

Purging is the process of removing the resin from the barrel of the injection molding machine. When changing from one resin to another, the resin remaining in the barrel is purged prior to the introduction of the second resin. Purging is also important for start-up and shut-down periods when the resin may be exposed to relatively long lengths of time at high temperature. Typically, thermally sensitive resins are purged and replaced with more stable resins on machine shut-down. Purging can also be used to clean the barrel and screw of degraded resin. Purging recommendations for use with Amodel® resins for both routine and non-routine circumstances are provided.

Purging in Routine Circumstances

At the end of a molding run, or if the molding process is to be interrupted for longer than 10 minutes, it is advisable to remove the Amodel® resin from the molding machine barrel. Failure to remove the resin during an interrupted process may cause the resin to become degraded, contaminating subsequent molded parts.

Proper safety procedures should be followed at all times. All machine guards and covers must be in place when purging. Personal protection equipment must be worn. Purged materials are very hot and should be handled with proper protective equipment. It is also advisable that a purge barrier be placed against the sprue bushing to protect the tool. The purge barrier should be constructed of material able to withstand purging conditions. Purge materials should be disposed of properly, in accordance with local regulations.

Cycle interrupt purge procedure

If the cycle is to be interrupted for longer than 10 minutes, a one-step purge can be accomplished as follows:

1. Shut off the resin feed at the hopper throat.
2. Move the barrel carriage away from the sprue bushing.
3. Purge the screw until empty and no more Amodel® resin can be purged.
4. Add high-density polyethylene (HDPE) to the feed throat.
5. Purge barrel with HDPE until the purge runs clear. When restarting the molding process, purge with Amodel® resin until an acceptable shot is obtained.

Daily shut-down procedure

When molding is complete for the day, or when the molding will be suspended for several hours, and you intend to start-up with the same grade of Amodel® resin, the Cycle Interrupt Purge Procedure may be used as follows:

1. Repeat steps 1 to 5 of the previous process.
2. Turn off the barrel heaters.

Changeover procedure

When molding is complete and you intend to change to another grade of material, the following procedure is suggested:

1. Shut off the resin feed at the hopper throat.
2. Move the barrel carriage away from the sprue bushing.
3. Purge the screw until empty and no more Amodel® resin can be purged.
4. Add an appropriate high-temperature purge material to the feed throat. Several commercially available purge materials* are rated to withstand the temperatures used here. The purge material used must be rated for use at 400 °C (750 °F) minimum. Follow the instructions provided by the manufacturer of the purge material.
5. Purge with the high-temperature purge material until no more Amodel® resin is visible in the purge shots.
6. Add HDPE to hopper throat.
7. Purge with HDPE until the purge runs clear.
8. Turn off the barrel heaters.

** A suitable high-temperature purge material will typically consist of a mixture of fractional melt flow high-density polyethylene and acrylic (PMMA). Other materials are available that will also perform well.*

Purging in Non-Routine Circumstances

Situations occur without warning that interrupt the molding process. Purging is usually required because the cycle interruption will often result in resin degradation, which can affect part quality.

Loss of electrical power to molding machine

When power is lost, proceed as follows:

1. Turn all barrel and nozzle heater controls to the "off' position.

2. When power is again available, retract the barrel carriage away from the mold and turn the barrel and nozzle heaters to the on position. As the heaters heat the barrel to processing temperature, some degradation of the Amodel® resin is likely. As a result of this degradation, nozzle drool and out gassing may occur.

3. Increase ventilation. Install protective shields to prevent contact with hot resin.

4. When the barrel has heated sufficiently, follow the previously described *Cycle Interrupt Purge Procedure* to remove all the Amodel® resin from the barrel. Do not attempt to accelerate the process by increasing the temperatures beyond the process set points.

5. Once all resin has been purged, the molding process may then be restarted as desired.

Loss of barrel or nozzle heater band

A heater band may fail resulting in a partial freeze-off of the barrel.

1. If this occurs, turn off all heater bands.

2. Manually decompress the screw and allow the barrel to cool.

3. When the barrel has cooled, replace the defective heater band and turn on heat.

4. After the barrel has come up to temperature, purge using the previously described *Cycle Interrupt Purge Procedure*.

5. Restart the molding process.

Loss of thermocouple control on a heater band

A thermocouple may become defective, resulting in an overheating situation.

1. If this occurs, turn off the defective zone and follow the previously described *Cycle Interrupt Purge Procedure*.

2. Turn off all barrel heaters.

3. When cool, replace the defective thermocouple.

4. Restart the molding process.

Nozzle obstruction

The molding machine nozzle may become obstructed for two reasons: the resin solidifies (freezes) in the nozzle or the nozzle is plugged by a broken screw tip, check ring, or other physical obstruction. Because resin solidification is more common, try steps 1 and 2 first.

1. Verify that the nozzle heater and thermocouple are in good working order.

2. If so, raise the nozzle temperature in increments of 5 °C (10 °F) until the resin flows freely. Do not raise the nozzle temperature above 350 °C (660 °F). If the nozzle temperature doesn't rise sufficiently, it may be necessary to install a larger heater band and/or insulate the nozzle. Do not use an external heat source, such as a torch, because localized overheating may result in dangerous pressure build-up. Hot resin may squirt out when the pressure is released.

3. If raising the temperature up to 350 °C (660 °F) does not clear the obstruction, it is likely that the nozzle is physically plugged. Turn off all barrel heaters.

4. When the barrel has cooled, remove the nozzle and the obstruction. Do not attempt to remove the nozzle while the resin is still molten as a pressure build up may be released.

5. Replace the cleared nozzle and turn on the barrel heaters. When the barrel temperature reaches the set point, purge using the previously described *Cycle Interrupt Purge Procedure*.

Loss of molding machine function

In general, if any function of the molding machine, such as clamp operation, hydraulic pressure, or other mechanical function, is not in proper working order, good safety practice dictates that the malfunction be repaired before the molding process continues. If molding must be stopped for more than ten minutes to facilitate such repairs, the *Cycle Interrupt Purge Procedure* should be followed.

Mold Design

Tool Steels and Surface Treatments

As with any engineering resin, the number and quality of parts that a tool is expected to produce will dictate the tool steel selection. For high-volume production, the initial expense of high-quality tooling will be a sound investment.

Generally, the common tool steels, such as H-13, S-7 and P-20, are acceptable for constructing injection molds. When molding glass- or mineral-reinforced resins, abrasion resistance is required and H-13 performs best. Soft metals, like aluminum, should be avoided – even for prototyping. Tool steel should be hardened prior to production; however, it is wise to sample the mold before hardening so that final dimensional cuts can be made easily.

After determining the general mold dimensions using the published shrinkage values for the resin grade being considered, it is prudent to cut all mold dimensions "steel safe." That is, mold components designed to form internal features on parts (cores) should be cut oversized and mold components designed to form external features should be cut with smaller than expected dimensions. After the initial material sampling, molded parts can be measured and final mold dimensions can be adjusted to produce the desired part dimensions.

While an excellent surface appearance may not be required, it is necessary to remove all machining marks from the mold to ensure proper part ejection. All surfaces should be polished in the direction of ejection. Textured surfaces are permitted for cosmetic parts; however, undercuts are not permitted. Additional information is available on "Draft" on page 23 in the section that discusses the use of draft to aid part ejection.

Plating of tool steel is not generally required. If a high-gloss, durable surface is required, it may be obtained with a surface treatment such as high-density chrome plating or titanium nitride treatment. Numerous other mold surface coatings and/or treatments are commercially available. While we have not completely investigated all of them, we haven't yet found any that offer a significant long-term improvement over high-density chrome or titanium nitride.

While Amodel® resin is not chemically aggressive toward tool steel, wear and abrasion can occur, especially with the glass and mineral/glass-filled grades. Typically, wear will occur in high shear areas of the mold, such as gates, corners, and areas inside the cavity that are initially contacted by the injected resin.

When designing the mold, wear should be considered when choosing gate location and cavity layout. The use of gate inserts and easily interchangeable internal cavity components at expected wear locations will minimize downtime for repairs.

Mold Types

There are several types of molds that can be used to mold Amodel® resins. These include two-plate, three-plate and hot-runner systems. All of these can contain manual or hydraulic slides and other required features. Amodel® resins generally do not lend themselves to compression, transfer, or blow molding.

Two-Plate Molds

Two-plate, or A-B molds, are the simplest and most common of all mold types. They use a stationary A-side and a movable B-side. The molten resin is injected through the sprue in the A-side, along a runner system on the parting line into the mold cavity or cavities, usually cut into the B-side.

Because the ejector system is generally designed to eject the molded part from the movable B-side, it is necessary that the part and the sprue and runner system remain in the B-side when the mold opens. To ensure that this occurs, it is common practice to cut a cold slug well, the runner system, and the majority of the cavity into the B-side.

The cold slug well, normally cut into the B-side opposite the sprue, has two functions. First, it collects the leading edge of the injected shot, which usually contains a "cold slug" of resin from the nozzle tip, and it prevents that material from entering the mold cavity. Second, by virtue of a slight undercut, the cold slug well will pull the sprue out of the A side as the mold opens. Placing an ejector pin in the B side at the cold slug well will eject the sprue.

The runner system is usually cut into the movable side of the mold as well. Ejector pins in the runner system which are partially below the surface, known as "sucker pins" ensure that the runner stays in the movable side. These sucker pins can contain slight undercuts.To ensure that the molded part itself stays in the movable side on mold opening, it is common practice to have the majority of the part formed in that side. Detail in the stationary side should be kept to a minimum. When it is necessary to have significant detail in the A-side, plans for positive ejection out of the A-side, such as a spring-loaded ejector system, are advisable.

Three-Plate Molds

Three-plate molds are a modification of the two-plate system in which a center plate is added between the movable and stationary plates. This center plate isolates the sprue and runner system from the parts. The runner system is formed between the stationary and the center plates and the molded parts are formed between the center and movable plates. When the mold opens, the parts remain and are ejected from the movable plate. The runner system and sprue, however, are broken from the molded parts and remain between the center and stationary plates. A spring loaded ejector system in the center plate ejects the runner.

There are several aspects of this system that make it more attractive than the two-plate system. First, degating is accomplished during the part ejection process, not as a secondary operation. Second, there is far greater freedom in selecting the number and the location of the gates by the placement of the gate drops through the center plate. Larger parts may be gated at several locations for ease of filling.

Hot Runner Systems

Hot runner systems replace the cold sprue and runner system with an electrically heated manifold that keeps that portion of the shot in a molten state. Resin is injected into the cavities on the B-side directly from the manifold drops. The most significant advantage of a hot runner system is material efficiency. Since no sprue and runner are molded, 100 % of the resin is in the part.

Thermal management of hot runner systems for crystalline materials such as Amodel® PPA is critical. Excessive residence time in the manifold should be avoided as it can result in material degradation. Separate temperature controllers for each drop on the manifold must be used. The position of the controlling thermocouple for each heat source in the manifold and drop should be close to the resin and between the heat source and the resin.

In general, mold design should be as simple as possible. Consideration should be given to part ejection as well as gate location when determining cavity positioning. Thermal management of the mold is also critical. The circulating heat transfer medium should be designed to maintain uniform cavity temperature. Failure to address these issues can result in part ejection problems.

Balancing Cavity Layout

In any multi-cavity mold, a balanced cavity arrangement is critical to molding quality parts. This means that all of the cavities must contain the same volume and fill at the same time. An unbalanced mold will over-pack some cavities while others are under-packed. A balanced mold fills all cavities at the same rate with the same pressure, ensuring uniform parts. Usually, this is accomplished by placing the cavities equidistant from the sprue with identically sized runners. The flow path should be the same length to each cavity.

Family molds are sometimes constructed to mold two or more different shaped parts in a single mold. This type of mold is often impossible to balance and should be avoided. When economics require that different cavities be placed in a single-mold base, the runner system should be equipped with cavity shutoffs. If acceptable parts cannot be molded together, cavity shutoffs allow each part to be molded individually.

Thermal Management

Since the process of thermoplastic injection molding essentially consists of injecting a molten resin into a cavity and allowing it to solidify there before ejection, it is very important that the temperature of the mold be controlled properly. This is generally accomplished by circulating a heat transfer medium through channels in the mold. Water is usable for mold temperatures up to 93 °C (200 °F), but oil is required for higher temperatures. The mold temperature required depends upon the resin grade being processed.

Electric cartridge heaters should not be used. Although they have the capability to heat the mold, they cannot remove heat from the mold. Since the polymer being injected into the mold is considerably hotter than the cavity, the excess heat needs to be removed. This is especially true in thermally isolated areas such as core pins where heat may build up and cause ejection problems. Thermally conductive copper-beryllium pins may be inserted into these areas to facilitate heat transfer.

The heat transfer channels in the mold should be located equidistant from each cavity and the flow designed so that each cavity is exposed to the same amount and temperature of fluid. The flow pattern past the cavities should be designed to be concurrent rather than sequential. The internal lines carrying the heat transfer fluid should be sized, within the limits of the available flow rate, to create turbulent flow to maximize heat transfer.

Runner Systems

The purpose of the runner system is to provide a channel connecting the sprue and the cavity. To avoid wasting the material in the sprue and runner system, they are generally ground and reused. Typically mixing 25 % ground sprues and runners with 75 % virgin resin is allowed. Therefore, the most material-efficient mold design will be achieved when the sprue and runner weight is no more than 25 % of the total shot weight.

Reducing the weight of the runner can be accomplished by minimizing both the length of the runner and the runner surface-to-volume ratio. A full round runner has the lowest surface-to-volume ratio. It is the most efficient runner, but is difficult to fabricate. A trapezoidal runner with a 10 % slope increases the weight of the runner system by about 25 %, but it is considerably easier to machine.

The size of the runner will depend on the length of the flow path and the grade of resin being used. In all cases, the major axis of the runner should be larger than the thickest section of the part to ensure that the runner does not freeze off before the part is fully densified.

When a runner splits to fill two or more cavities, the total volume of all secondary runners should be no greater than the volume of the primary runner. This will ensure that the velocity of the melt front is not decreased.

Cold slug wells should be used at every turn in the runner system as well as at the base of the sprue. These wells will serve to remove the leading edge of the advancing melt and prevent the introduction of cold material into the cavity.

Since the material in the runner system is usually re-used, the runner system should be well-vented to prevent burn marks. Venting the runner will also allow the gas in the runner to exit out of the runner rather than having to be vented out through the cavity.

Keeping the runner system on the proper plate during mold opening can be accomplished with sprue pullers and sucker pins that provide slight undercuts. Generous ejector pins on the runner system will ensure positive ejection from the mold.

Gating

All conventional gating types are suitable for processing Amodel® resins, including hot runner systems.

The considerations for selecting a gating method should include gate location for optimal densification, gate removal method, generation and use of regrind, and cosmetic requirements.

Sprue Gating

Sprue (direct) gating is most often used with hot runner systems and is often used on prototype molds as well. This method places the mold cavity directly in line with the sprue or under the hot runner drop. The major advantages of this method include system simplicity and minimized runner volume and flow length. The disadvantages of sprue gating include the potential for cold slugs to be visible on the part and the need to remove the sprue or hot runner vestige, which generally involves a post machining operation or a manual operation by the operator at the press.

Edge Gates

Edge gates are by far the most common gate type. These gates are used with a conventional sprue and runner system. The runner introduces the resin to the mold cavity along the parting line. Cold slug wells are placed in the runner system to keep cold slugs out of the parts. An undercut is generally placed in the movable side of the mold to act as a sprue puller. Slight undercuts can also be placed in the gate, where it meets the part, to facilitate runner/part separation.

Advantages of edge gates include ease of fabrication, modification, maintenance, and trouble-free operation. Cold slug wells eliminate cold slugs in parts. The disadvantage of this gating method is the generation of scrap, some of which may be ground and reused. Gate inserts are strongly recommended for ease of gate replacement if excessive wear occurs.

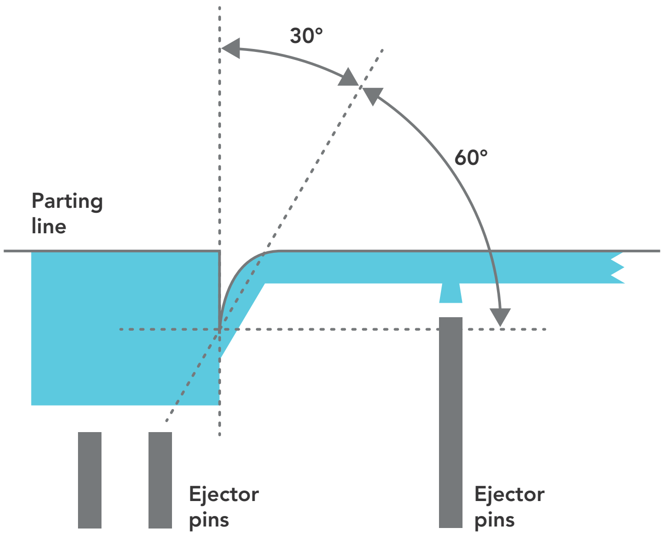
Diaphragm Gates

Diaphragm gating is used almost exclusively for molding circular parts without knit lines. A high degree of flatness can also be attained with this method when using a fiber-reinforced grade that may be prone to warpage when using other gate types. Similar to sprue gating, a machining operation will be required to remove the gate.

Tunnel or Submarine Gates

Tunnel or submarine gating is another popular method because it is self-degating. Tunnel gates employ a conventional parting line runner system similar to a standard edge gate. In close proximity to the mold cavity, however, the runner tunnels beneath the parting line and gates into the part below the mold parting surfaces, as illustrated in Figure 8.1. Upon ejection, the molded part and runner/gate are separated by the tool steel itself. The angle of the drop is critical in ensuring that the runner will eject properly and not become stuck in the mold. Due to the high modulus of the reinforced grades of Amodel® resin, a maximum angle of 30 ° perpendicular to the parting line of the mold is recommended for those grades. Unfilled grades with lower modulus may use less severe drop angles.

Figure 8.1
Tunnel gates



The main advantage of tunnel gates is the automatic degating feature. A potential disadvantage is the possibility of an irregular gate vestige. Gate inserts are strongly recommended for use with tunnel gates.

Hot Runner Molds

Hot runner molds are extremely popular due to their material efficiency. Hot runner molds can be combined with any of the previous gating methods, though they are most often direct-gated. Hot runner molds are often provided as a turnkey system; however, certain aspects of the design are important for successful operation with Amodel® resin. The hot runner manifold channels should be free-flowing with no sharp corners or dead spots.

Molten resin tends to stagnate in corners or dead spots, resulting in degradation, which eventually contaminates the melt and the parts.

Temperature control of the hot runner mold is critical. Each drop in the mold should have its own thermocouple and heat source. Placement of the thermocouple should be between the heat source and melt channel, allowing accurate thermal management without degradation of the resin.

Gate Dimensions

The dimensions of a gate depend on several factors, including the size of the part, its thickness, the type of gate being used, and the grade of resin. In general, the smallest gate dimension should be 50 % of the wall thickness at the point being gated into to ensure optimal densification. If the gates are too small, parts may be under-packed, shrink erratically, have internal porosity or sink marks, and have poor mechanical performance.

Rectangular edge gates should be 1.5 to 2 times as wide as they are deep and proportional to the part thickness as noted above.

Tunnel gates should be a minimum 0.5 mm (0.020 in.) on the minor axis, increasing in size for larger parts.

Three-plate molds should use proportionally sized gates, but no less than 0.5 mm (0.020 in.) in diameter for small parts and no more than 3.2 mm (0.125 in.) in diameter for large parts. Very large gates used with three-plate molds may cause degating problems.

Gate Location

Gates should always be located in the thickest section of the part to allow flow from thick to thin sections. Cosmetic considerations may dictate gate locations; however, it is inadvisable to flow through a thin section to a thicker section. Other factors that may influence gate location include knit line location, flatness requirements, and the number of gates required to fill the part.

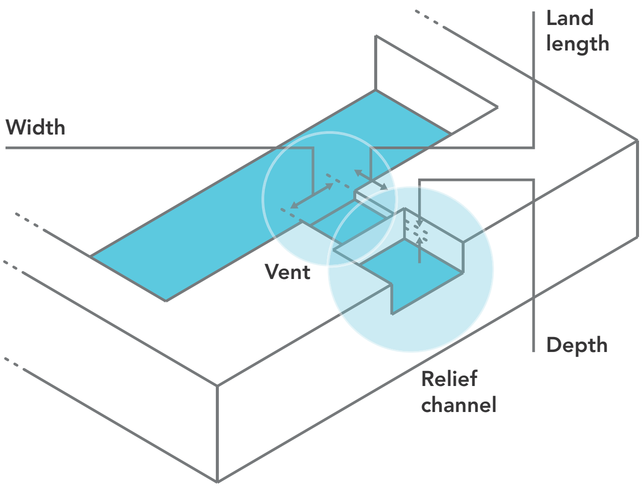
Venting

Vents in a mold cavity, as shown in Figure 8.2, allow the gas (air) present in the cavity to escape as the resin fills. Inadequate venting may result in the gas being compressed in the cavity, which may then heat up to the point of causing burn marks on the part and a deposit on the mold surface. This is known as dieseling. Poor venting may also result in poor weld-line strength and the inability to completely fill the cavity. The relatively fast injection rates required by most Amodel® grades require that a considerable amount of venting be used in the tooling.

Figure 8.2
Vents

Standard vent dimensions:

Depth: 0.04 mm (0.0015 in.) to 0.06 mm (0.0025 in.)
Width: 3.2 mm (0.125 in.) minimum
Land length: 0.8 mm (0.03 in.) minimum to 1.6 mm (0.06 in.)
Relief channel depth: 1.3 mm (0.05 in.) minimum
The relief channel should extend to the edge of the tool.



Vent location depends on the cavity layout and can be accurately predicted by flow simulation. Short shots can also be used to find areas for venting. In general, vents should be placed opposite the gate, at expected knit lines, and at various locations on the parting line, such that the total volume of vents is equal to about 25 % of the cavity perimeter. Venting below the parting line can be accomplished by incorporating vents at ejector pins.

Venting on core pins and in deep cavities also assists part ejection by breaking the vacuum formed and preventing part warpage.

If production runs are often interrupted to clean deposits from the mold, additional or deeper vents may help solve the problem.

Molding Tolerances

The dimensional variability of a molded part is a combination of the dimensional variability in the cavities, the variability due to the molding process, and the variability in material shrinkage.

Material shrinkage is a function of the filler type and content. The higher the level of filler, the lower the shrinkage. Fillers with an aspect ratio, such as glass fiber, will have different shrinkages in the flow and transverse direction. Consult the data sheet for the shrinkage of a particular grade.

Relatively good tolerance capability is achievable with Amodel® resins, provided molding process variations are minimized.

The actual tolerance depends on the dimension. The larger the dimension, the larger the tolerance needed. Generally a tolerance of +/- 0.2 % of a given dimension can be held. For very large dimensions, it may be possible to hold better than 0.2 %. For very small dimensions, assume +/- 0.05 mm (0.002 in.) as the best tolerance.

For statistical tolerance capability, it is advisable to double the above targets.

Part Ejection

There are several important factors to consider in planning for the ejection of parts from a mold cavity. The majority of molding problems are traceable to avoidable ejection problems.

Draft

Draft refers to adding a taper that allows the part to become free from the mold after only short travel of the ejector pins. The amount of taper is commonly called the draft angle. Although draft angles may not generally be regarded as part of the ejection system, they are essential for the ejector system to function properly. Minimum draft angles for Amodel® resins are 0.5 to 1 ° for external mold surfaces and 1 to 1.5 ° for internal surfaces or cores.

When dimensional tolerances do not allow the recommended draft, alternate methods should be investigated, such as dividing the molded part between the movable and stationary mold surfaces or splitting core pins. Both of these methods can reduce the dimensional change due to draft by half.

When the part surface is textured, draft angles need to be increased by 1 ° for each 0.025 mm (0.001 in.) of texture depth.

Mold Polish

It is essential that the eject surfaces of molds be polished in the direction of ejection, which is known as draw polishing. Undercuts are not permitted because they cause parts to stick. Mold polishing should be a regular part of preventative mold maintenance.

Ejector Pins

The most common method for part ejection is the use of ejector pins attached to an ejector plate on the movable half of the mold. When the mold opens, the ejector pins move forward and push the part out of the mold cavity. The placement and size of the ejector pins are extremely important to ensure reliable ejection of parts. Using undersized or too few ejector pins can result in the molding defect called witness marks or unnecessarily long molding cycle times.

Ejector pins should be placed uniformly around the part to ensure even ejection. Additional pins should be placed at the deepest feature of the cavity. These pins will make sure the part is pushed out of the cavity uniformly. Without them, the deep features will be pulled out of the cavity by the shallow sections of the part, which may result in part distortion or breakage. When a deep cavity feature of the part is in the form of a rib, a blade or rectangular ejector pin should be considered. Ejector pins should be as large as possible to minimize the force per unit area of ejection.

Stripper Plates

When a part design calls for numerous cores, especially with low draft angle tolerance requirements, a stripper plate can be used to pull the cores back into the movable half of the mold before the ejector pins engage. Stripper plates can often be spring actuated, minimizing the complexity of design.

Stationary Plate Ejectors

When the complexity of a part dictates that there is considerable detail on both the movable and stationary sides of the mold, there may be the undesirable tendency for the part to remain on the stationary side of the mold as it opens. This can generally be remedied with the use of an undercut (sprue puller) on the movable side under the sprue. In extreme cases, a spring-loaded ejector plate on the stationary side of the mold forces the part to stay in the movable side. The guidelines for movable side ejectors apply here.

Collapsible Cores

When undercuts are required on an inner diameter, a collapsible core may be used. Due to the complexity and high maintenance of these cores, their use should be avoided whenever possible.

Mold Maintenance

Proper mold maintenance is critical to maintaining part quality and maximizing mold life.
The recommended minimum maintenance program is shown in Table 8.1. Individual molds may require additional maintenance items.

Table 8.1
Mold maintenance recommendations

Every Shift	Daily	Every 20,000 Cycles or Every Production Run (Whichever is Less)	Every 100,000 Cycles	On Returning Mold to Service From Storage
Inspect parts for broken cores or ejector pins as well as drag marks indicating cavity damage.	Inspect tool for worn, bent or loose ejector pins.	Remove tool from press.	Disassemble all plates and clean with safety solvent.	Using an approved solvent, remove the mold preservative.
Inspect parts for burn marks, which may indicate vent clogging.	Clean all non-parting line vents.	Drain and clean all oil/water lines. Ensure that lines are free flowing.	Check all components for wear. Replace as necessary.	Check tool for damage during storage.
Observe tool as it opens and closes for irregular motion.		If tool will be out of service, clean with safety solvent and spray with rust preventative before storage.	Lubricate all moving parts as required.	Check oil/water lines for leakage.
Clean tool surface with mold cleaner, especially vents.		Check that all clamps, bolts and plates are tight and in good repair.	Check gates and vents for proper dimensions.	Provide last shot from previous run to operators for quality comparisons.
		If tool is to be put into storage, bag the last shot and store with the tool as representative of the quality of parts being produced.	Check all gaskets, seals and O-rings for integrity.	
			Check ejector system for alignment.	

Troubleshooting

Table 9.1 is a chart to assist with troubleshooting the molding of Amodel® resin. The remedies have been grouped into two categories, the process variables and the tooling or equipment variables. The process variables can be applied during a molding run. Changing a tooling or equipment variable normally requires that the process be shut down to fix the problem. There may be reluctance to shut down in order to fix a problem, but often that may be the best approach.

Table 9.1
Troubleshooting Guide for Amodel® PPA

Problem	Process Parameters: Ensure resin dryness	Back pressure	Barrel termperature	Cooling time	Cushion	Hold pressure and time	Injection pressure	Injection speed	Mold temperature	Nozzle temperature	Screw recovery speed	Shot size	Tooling and Equipment: Cavity venting	Increase clamp pressure	Increase draft	Increase gate size	Change gate location	Clean and polish mold	Insulate nozzle	Use reverse taper nozzle	Polish sprue bushing	Comments
Brittle parts – wet resin	1+																					Maximum moisture 0.10 %
Brittle parts – cold resin			2+			1+	5+	6+	7+	3+						4						Minimum moisture 0.03 %
Brittle parts – degradation		2−	1−	4−	5−						3−											Reduce residence time
Burn marks	4+		5−				2−	3−	7−	6−			1+			8	9					Vents 0.03 mm to 0.06 mm
Ejector marks				1+		4−	3−		2−						5			6				
Flash			4−			3−	2−						5−	1								
Flow lines		5+	1+				2+	3+	4+													
Jetting							3−	2−									1					
Knit lines			4+			6+	2+	3+	5+								1					
Nozzle drools	2+	5−	4−	7−	8−					1−	6−									3		
Nozzle freezes			3+						4+	1+									2	5		Retract injection unit
Parts stick				5+		3−	2−		6±						7			1				Do not overpack
Plate out on tool, vents	2+		4−		7−				3−		6−		1+									
Screw recovery	1+	2±	4±								3±											Check screw for wear
Short shots			4+				2+	3+	6+			1+	7+			5						
Sink marks			4+			1+	3+		2+							5						
Splay marks	1+						3−	4+								2						
Sprue sticks				3+		1−	2−			4+										6	5	Do not overpack
Surface poor			4+			3+		2+	1+									6		5		
Voids						1+	3+		2+							5						
Warpage				3+		2+			1+			4+				5	4					

Apply the remedies in numerical order: + Increase, – Decrease, ± Increase or Decrease

Index

Progress beyond

Specialty Polymers

Worldwide Headquarters
SpecialtyPolymers.EMEA@solvay.com
Viale Lombardia, 20
20021 Bollate (MI), Italy

Americas Headquarters
SpecialtyPolymers.Americas@solvay.com
4500 McGinnis Ferry Road
Alpharetta, GA 30005, USA

South America
SpecialtyPolymers.Brazil@solvay.com
Estrada Galvão Bueno, 5.505 - Batistini
09842-080 - São Paulo, SP - Brazil

Asia Headquarters
SpecialtyPolymers.Asia@solvay.com
No.3966 Jindu Road
Shanghai, China 201108

www.solvay.com