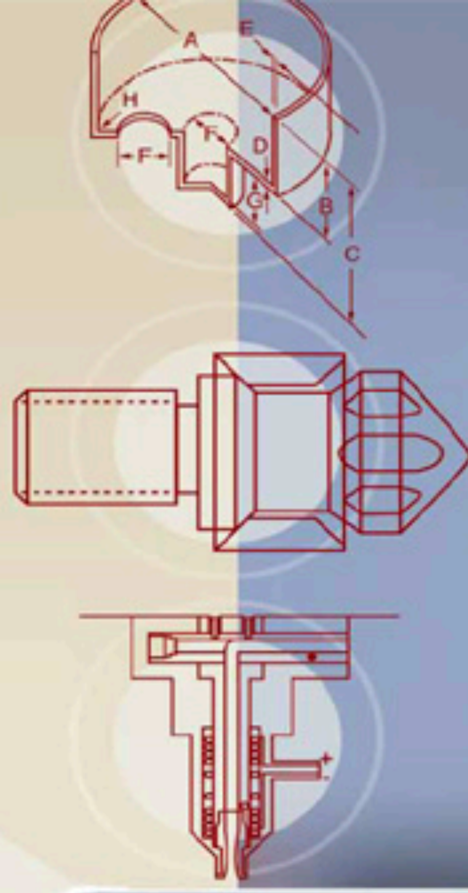
BASF Plastics

Injection Molding Processing Guide for Ultramid® Polyamide (Nylon)



BASF

The Chemical Company

English/Metric Conversion Chart

To Convert English System	To Metric System	Multiply English Value by...
DISTANCE		
inches	millimeters	25.38
feet	meters	0.30478
MASS		
ounce (avdp)	gram	28.3495
pound	gram	453.5925
pound	kilogram	0.4536
U.S. ton	metric ton	0.9072
VOLUME		
$inch^3$	$centimeter^3$	16.3871
$inch^3$	liter	0.016387
fluid ounce	$centimeter^3$	29.5735
quart (liquid)	$decimeter^3$ (liter)	0.9464
gallon (U.S.)	$decimeter^3$ (liter)	3.7854
TEMPERATURE		
degree F	degree C	(°F - 32)/1.8 = °C
PRESSURE		
psi	bar	0.0689
psi	kPa	6.8948
ksi	MN/m^2	6.8948
psi	MPa	0.00689
ENERGY AND POWER		
in lb_f	Joules	0.113
ft lb_f	Joules	1.3558
kW	metric horsepower	1.3596
U.S. horsepower	kW	0.7457
Btu	Joules	1055.1
Btu in/(hr ft^2 °F)	W/m °K	0.1442
VISCOSITY		
poise	Pa s	0.1
BENDING MOMENT OR TORQUE		
ft lb	N m	1.356
DENSITY		
lb/in^3	g/cm^3	27.68
lb/ft^3	kg/m^3	16.0185
NOTCHED IZOD		
ft lb/in	J/m	53.4

Table of Contents

Chapter 1

Introduction

Chapter 1: Introduction

General Information

Ultramid resin is available as uniform pellets predried to a very low moisture content. Depending upon the particular grade, it is available in 55 lb. (25 kg) bags, or 1500 lb. (680 kg) corrugated boxes. Each of the two containers includes a moisture barrier liner to maintain a low moisture level in the Ultramid nylon. Open containers should be resealed to maintain low moisture levels. Most grades are available in heat stabilized (HS), black (BK-102), and UV stabilized black (BK-106) formulations. Standard and custom colors are also available upon request.

Ultramid Nylon

The majority of the Ultramid family is based on nylon polyamide) 6. BASF is a fully integrated supplier of nylon 6. This includes full responsibility for production of feedstocks to the compounding, manufacturing, and distributing of hundreds of grades of resin.

Ultramid nylon 6 is one of the most versatile and performance-proven engineering thermoplastics. Impact modification of Ultramid nylon results in an extremely flexible and impact-resistant product, while reinforcement produces high strength, stiffness, and dimensional stability.

Another attribute of Ultramid nylon is ease of processability. It is known for its wide processing window in both extrusion and injection molding processes and for its ability to achieve a resin-rich, uniform surface appearance, even with high levels of reinforcement.

The Ultramid product line also includes nylon 6, 6/6 for film extrusion products and nylon 6,6 for injection molding products.

Ultramid Homopolymers

Ultramid homopolymers are used in a wide variety of extrusion and injection molding applications. Major attributes include strength, toughness, and excellent chemical and abrasion resistance.

Standard Grades

Standard products include 8200 a medium viscosity injection molding grade; 8202, a low viscosity injection molding grade; 8270, a modified, ultra high viscosity grade for extrusion and blow molding applications; 5202, a low viscosity injection molding grade based on nylon 6,6.

Alpha Grades

Alpha grades differ in crystalline structure compared to standard Ultramid grades. This results in increased strength, stiffness, and heat distortion temperature combined with faster set up time. Grades include 8202C, a low viscosity, highly crystalline injection molding grade; 8202CQ, a low viscosity, improved productivity injection molding grade; 8203C, an intermediate viscosity, highly crystalline tubing and cable liner extrusion grade; and 5202CQ, a low viscosity, improved productivity injection molding grade based on nylon 6,6.

Rotational Molding Grades

Ultramid 8280 nylon and 8281 nylon are specifically tailored for rotational molding. 8280 exhibits excellent strength and toughness. 8281 is a plasticized rotomolding resin that offers increased flexibility.

Impact Modified Ultramid Nylon

Impact modification of Ultramid nylon results in a series of polymers containing various levels of enhanced toughness and flexibility combined with excellent chemical and thermal resistance.

Impact-modified injection molding grades include 8253, which exhibits improved dry-as-molded toughness over conventional nylon 6 while maintaining excellent strength and stiffness characteristics; 8255, which offers a high degree of flexibility combined with toughness; 8351, a high impact, faster cycling grade; and Ultratough Nylon BU50I, offering high impact strength and ductility to -40° C (-40° F).

Reinforced Ultramid Nylon

Fiberglass or a combination of fiberglass and mineral reinforcement enhances the performance characteristics of Ultramid nylon molding compounds.

Fiberglass Reinforced Grades

Fiberglass reinforcement improves Ultramid nylon's strength, stiffness, dimensional stability, and performance at elevated temperatures. Glass reinforced grades include HMG10, 50% glass, high modulus; HMG13, 63% glass, high modulus; SEG7, 35% glass; 8230G, 6% glass reinforcement; 8231G, 14% glass; 8232G, 25% glass; 8233G, 33% glass; 8234G, 44% glass; 8235G, 50% glass; HPN™ 9233G, 33% glass reinforced, improved productivity; and 5233G, 33% fiberglass based on nylon 6,6.

Fiberglass Reinforced, Impact Modified Grades

Combining fiberglass reinforcement along with impact modification produces compounds that offer increased dry-as-molded impact while maintaining excellent strength and stiffness properties. Products include TG3S, 15% glass, impact modified; TG7S, 34% glass, impact modified; 8331G, 14% glass, impact modified; 8332G, 25% glass, impact modified; 8333G HI, 33% glass, high impact, improved productivity and surface appearance; 8334G, 40% glass reinforced, impact modified; and HPN 9333G, 33% glass reinforced, impact modified, improved productivity.

Mineral Reinforced Grades

Mineral reinforcement enhances strength and stiffness properties while maintaining typical chemical resistance associated with Ultramid nylon. Mineral reinforced products include 8260, 40% mineral, chrome plateable; 8360, 34% mineral; and 8362, 34% mineral, impact modified; and HPN 9362, 40% mineral reinforced, impact modified, improved productivity.

Mineral/Glass Reinforced Grades

Mineral and glass reinforcement leads to products with an excellent balance of mechanical properties combined with warpage resistance. Mineral/Glass reinforced grades include SEGM35 H1, 40% glass/mineral reinforced; 8262G, 20% mineral/glass reinforced; 8266G, 40% mineral/glass reinforced; and 8267G, 40% mineral/glass reinforced.

Chapter 2

Health, Safety, and Environmental Considerations for Processing Ultramid Nylon

Chapter 2: Health, Safety, and Environmental Considerations for Processing Ultramid Nylon

Safety is an important consideration during the processing of any thermoplastic resin. While the molding of Ultramid resins is generally considered safe, failure to take adequate precautions in the following areas may lead to personal injury.

Housekeeping

Slips and Falls
Ultramid nylon resin comes in cylindrical-shaped pellets. When spilled on a floor, they can be dangerous. To avoid falls from pellet spills or leaks, sweep up or vacuum material and place it in a container for possible reuse or disposal.

Material Handling

Cuts
The product may be packaged in drums, bags or gaylords. Gloves are recommended when handling drums to avoid cuts during manual movements and when removing the ring seals, releasing the locking rings, and removing the drum lids. The rigid paper used in bags or the corrugated construction of the gaylords may also cut the skin. Cutting tools should have a protected cutting edge to guard against lacerations. (Please see section on *Personal Protective Equipment,* page 7).

Thermal Hazards

Burns
Due to the temperatures necessary to process nylon resins, injection molding machine parts and equipment may be hot and cause burns on contact with the skin. In addition, contact with molten resin from normal operation or unexpected occurrences may result in burns involving any exposed areas of the body. Operators should wear personal protective equipment. Injection molding machinery suppliers also provide purge guards which help to protect the operator from burns from spattering molten polymer. (Please see section on *Personal Protective Equipment,* page 7).

Thermal Processing Hazards

Thermal processing of thermoplastic materials is always accompanied by the release of fumes and vapors. At recommended processing temperatures, these fumes and vapors will generally be low boiling volatiles. In the case of Ultramid products, the fumes will be principally caprolactam monomer. Caprolactam vapor can cause irritation of the eyes, nose, throat, and skin at sufficiently high concentrations. Proper ventilation should be provided at all possible emission points on the injection molding machine to minimize exposure of volatiles to equipment operators.

Thermal Decomposition

The recommended processing temperatures for Ultramid resins have been optimized to provide excellent processability and performance characteristics. Normal processing temperatures may range from 450° F to 590° F (230° C to 310° C). Please reference Chapter 7: *Processing Ultramid Nylon* for further specific information. Excessively high processing temperatures can lead to thermal decomposition. Thermal breakdown may create a complex mixture of organic and inorganic compounds which may be flammable, toxic, and/or irritating. The components generated can vary depending on colorants, specific temperature, exposure time, and other environmental factors. Proper ventilation should be provided at all possible emission points on the injection molding machine to minimize exposure of volatiles to equipment operators. Injection molding machinery suppliers also provide purge guards containing limit switches that prevent operation of the press while in the open position. This helps to minimize exposure to any volatiles emitted by the molten extrudate, in addition to protecting the operator from burns from spattering molten polymer. Under no circumstances should the protective circuitry provided on processing equipment be altered to allow operation while guards are in the open position.

Personal Protective Equipment

Proper personal protective equipment should be worn depending upon conditions that exist in the molding facility.

Eye and Face Protection

Safety glasses containing side shields should be worn as a minimum when working in a plastics processing facility. A face shield should be considered for additional protection when purging or working near molten material. Contact lenses should not be worn when processing materials for extended time periods since their permeable structure may absorb vapors which can cause eye irritations.

Hands, Arms, and Body

Gloves are recommended when handling or opening drums to avoid abrasion hazards from sharp surfaces. When handling molten polymer or hot parts, insulated gloves should be worn. Arm protection should be considered for additional protection against hot surfaces, such as machine barrels and tooling, and when handling hot parts.

Respiratory Protection

In dusty conditions, a mechanical filter respirator should be worn. If exposed to thermal processing fumes or vapors in excess of permissible exposure levels, an organic vapor respirator is suggested. Respiratory protection for the conditions listed above should be approved by NIOSH.

Foot Protection

Safety shoes should be worn when working in areas where heavy objects are being moved, such as transporting or setting molds.

Material Safety Data Sheets (MSDS)

MSDS are supplied by BASF for all Ultramid products. The MSDS provide health, safety, and environmental data specific to Ultramid nylon grades or resin families. The MSDS include information concerning first aid measures, hazards information, precautions/procedures, personal protective equipment, physical and reactivity data, hazardous ingredients, and environmental data. Also included on the MSDS is a Product Safety Department contact where additional information can be obtained. MSDS can be found on our website at www.plasticsportal.com/usa

Chapter 3

General Part Design

Chapter 3: General Part Design

Section Thickness

Uniformity in wall thickness is critical when designing parts to minimize warpage, distortion, internal stresses and cycle times. Figure 3A shows several examples of proper design. When non-uniform section thickness is unavoidable, gradual blending should be used between the sections as shown in Figure 3B. In general, using the thinnest wall allowable, based on the expected function of the part, will help to reduce cycle time and material costs. Typical wall thickness for Ultramid resin parts range from .040 inch (1mm) to .200 inch (5mm).

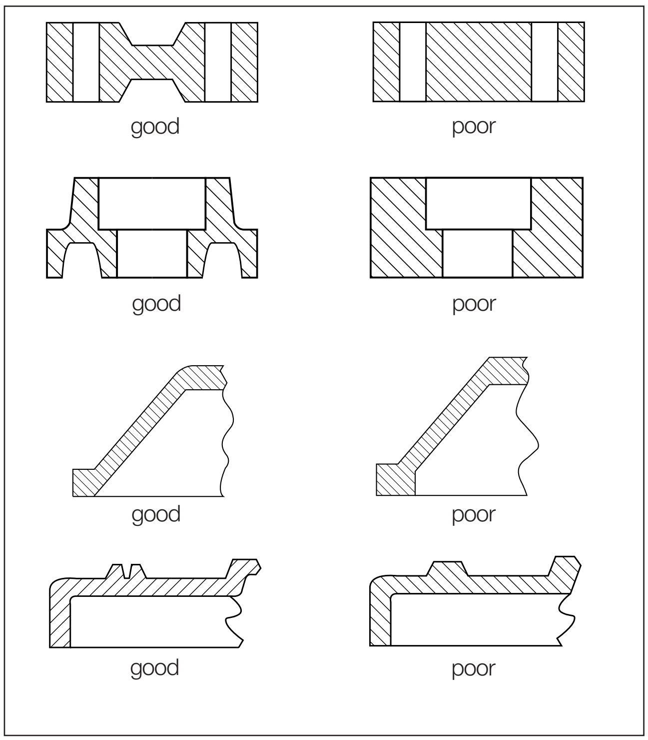


Figure 3A

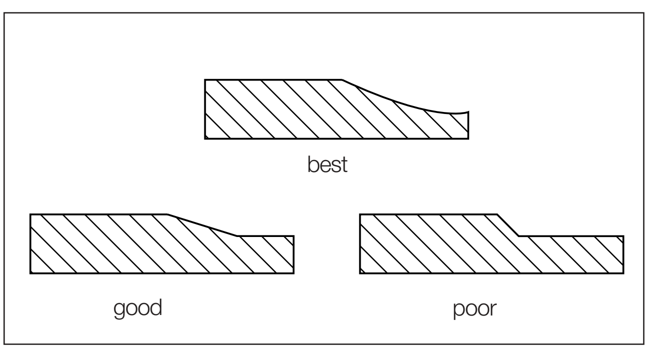


Figure 3B

Undercuts

When necessary, undercuts producing 2% strain are allowable for reinforced Ultramid nylon grades. If properly designed, undercuts producing up to 8% strain are possible with unreinforced grades. However, wall thickness, part design and mold temperature are factors which can influence ease of part ejection from the mold. Parts with undercuts should be thoroughly inspected after molding for unwanted damage or aesthetic flaws in the undercut area.

Recommended Radii

Sharp corners act as stress concentrators and often contribute to part failure. In addition, sharp corners prevent smooth flow when filling. For these reasons, parts should be designed with generous radii and fillets wherever possible. A minimum radius of .020" (0.5mm) is recommended at all sharp corners and larger radii are generally beneficial when possible. However, making the radius too large will cause a sink mark on the opposite surface due to the greater mass of material. Figure 3C shows the relationship between stress concentration and fillet radius. As the stress concentration factor (K) increases, the part becomes more prone to failure. The preferred value of R/T is 0.6 for most parts designed with Ultramid resin.

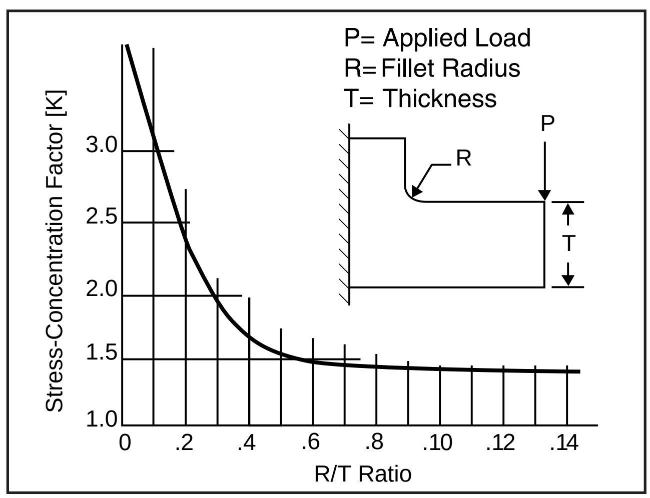


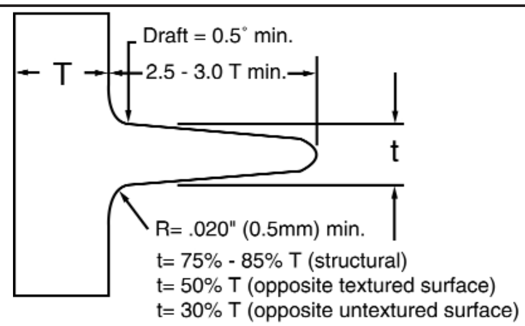
Figure 3C

Draft Angles

For most parts molded from Ultramid resin, a draft angle of 1° per side is required for facilitating part ejection. However, draft angles as low as 0.5° (requires highly polished surfaces) and as high as 1.5° per side are not uncommon depending on part design and complexity. In general, larger draft angles make it easier to eject the part from the mold, especially parts with deep pockets, tall ribs, or heavy textures.

Ribs

Ribs are effective design features which add strength and often facilitate flow during filling. However, proper design is important as ribs sometimes cause sink marks or aesthetic irregularities. Ribs should only be used when needed for stiffness and strength. In structural parts where sink marks are of no concern, rib base thickness (t) can be between 75% - 85% of the adjoining wall thickness (T). For appearance parts, where sink marks are objectionable, rib base thickness (t) should not exceed 50% of the adjoining wall thickness (T) if the outside surface is textured and 30% if not textured. In addition, ribs should include proper draft and a base radius of at least .020" (0.5mm) as shown in Figure 3D.



Rib Design

Figure 3D

Tolerances

Figure 3E shows the SPI tolerance standards for nylon resins, including Ultramid nylon. This is a general guide for design, based on typical parts produced by several different molders and resin suppliers. The numbers shown should not be interpreted as final design specifications for all applications, but rather as a reference for nylon parts similar to the example shown, molded under normal conditions. Parts with non-standard designs or complex geometry should be evaluated on an individual basis.

Many factors must be considered when maintaining tolerances in molded parts, such as processing conditions, mold/part design, and end-use environment. In general, parts with many close tolerance requirements will be more difficult to produce consistently than parts with fewer or less critical requirements.

Effect of Processing on Tolerances

Control of tolerances will be influenced by molding conditions. Since shrinkage can directly affect dimensional change, it is important to provide adequate pressure during the filling and packing stages. In addition, machine consistency, temperature control, and cycle time must be carefully maintained to prevent dimensional shift.

Process conditions typically have more effect on the shrinkage of unreinforced and impact modified grades of Ultramid nylon than on reinforced grades.

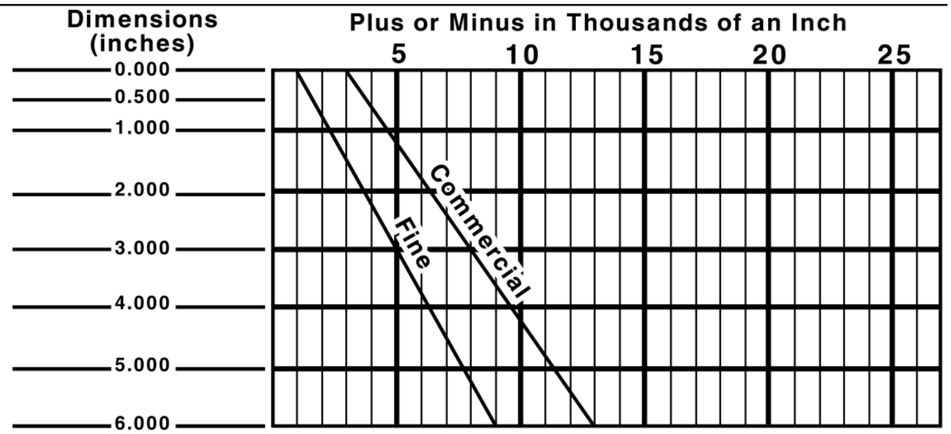
Effect of Mold Design on Tolerances

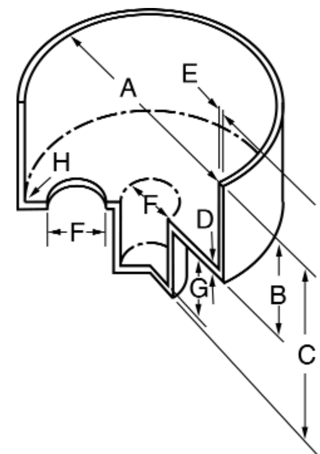
The complexity of the mold has a direct influence on control of part tolerance. Family or multi-cavity molds with non-uniform runner systems and/or slides and cams should be given special consideration since tolerances will be harder to maintain under such circumstances. Gates and runners must be large enough to provide good packing pressure and thereby minimize shrinkage. After many cycles, mold wear can also contribute to dimensional shift, especially with mineral and/or glass fiber reinforced grades. Proper tooling selection should be considered when high volume production is expected. (Please see *Tooling Considerations*, Chapter 5.)

Standards & Practices of Plastics Molders	Material Polyamide (Nylon) (PA)

Note: The *Commercial values* shown below represent common production tolerances at the most economical level. The *Fine values* represent closer tolerances that can be held but at a greater cost. Addition of reinforcements will alter both physical properties and dimensional stability. Please consult the manufacturer.

Drawing Code	Dimensions (inches)	Comm. ±	Fine ±
A=Diameter (See note #1) B=Depth (See note #3) C=Height (See note #3)	8.000 to 12.000 for each additional inch add (inches)	0.003	0.002
D=Bottom Wall	(See note #3)	0.004	0.003
E= Side Wall	(See note #4)	0.005	0.003
F=Hole Size Diameter (See note #1)	0.000 to 0.125	0.002	0.001
	0.126 to 0.250	0.003	0.002
	0.251 to 0.500	0.003	0.002
	0.501 and over	0.005	0.003
G=Hole Size Depth (See note #5)	0.000 to 0.250	0.004	0.002
	0.251 to 0.500	0.004	0.003
	0.501 to 1.000	0.005	0.004
H=Corners, Ribs, Fillets	(See note #6)	0.021	0.013
Flatness (See note #4)	0.000 to 3.000	0.010	0.004
	3.001 to 6.000	0.015	0.007
Thread Size (Class)	Internal	1	2
	External	1	2
Concentricity	(See note #4)(F.I.M.)	0.005	0.003
Draft Allowance per side		1.5°	0.5°
Surface Finish	(See note #7)		
Color Stability	(See note #7)		





Reference Notes

1. These tolerances do not include allowance for aging characteristics of material.
2. Tolerances are based on 0.125" (3.175mm) wall section.
3. Parting line must be taken into consideration.
4. Part design should maintain a wall thickness as nearly constant as possible. Complete uniformity in this dimension is sometimes impossible to achieve. Walls of non-uniform thickness should be gradually blended from thick to thin.
5. Care must be taken that the ratio of the depth of a cored hole to its diameter does not reach a point that will result in excessive pin damage.
6. These values should be increased whenever compatible with desired design and good molding techniques.
7. Customer-Molder understanding is necessary prior to tooling.

Figure 3E

Table reprinted from S.P.I.'s Standards & Practices of Plastics Molders. Guidelines For Molders and Their Customers. Material reproduced with the permission of S.P.I.

Chapter 4

The Injection Molding Machine

Chapter 4: The Injection Molding Machine

Machine Selection

Selecting an injection molding machine with proper design is critical to molding a quality part and ensuring economic success.

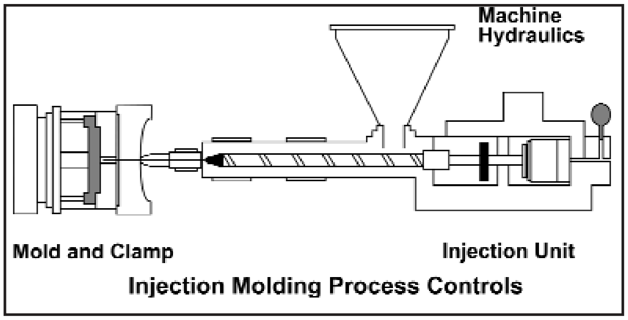


Figure 4A

Below are four important molding machine points to consider when molding Ultramid nylon. These issues are discussed in detail in this chapter.

- Proper Screw and Barrel Selection
- Proper Nozzle Tip Type
- Condition and Type of Check Valve
- Clamp Requirements

Screw and Barrel Selection

Screw Design

The screw performs the following functions in the injection molding process.

1. Conveys the material through the barrel.
2. Mixes the material to the proper molten state.
3. Compresses the material to maximum density.
4. Forces the material into the mold.

Adjustments to the injection process often involve the screw to some degree. The screw, when performing its function, provides important contributions to the overall process. Below are several process parameters that are affected by screw design.

1. Material melting profile
2. Melt temperature
3. Material mixing
4. Shearing of the resin

Therefore, it is important to be aware of the type of screw that is being used for each application. As mentioned above, in most cases Ultramid products can be processed with general purpose screws that are supplied by the machine manufacturer.

However, when molding Ultramid products for extended periods of time in a production environment, specific barrel and screw designs are recommended for maximum machine wear resistance and proper material plastication. Depending on the level of reinforcement in the material, different barrel and screw combinations are recommended by BASF. Refer to Figure 4B for recommended screw and barrel composition.

Ultramid Material Type	Wear Environment	Base Material	Screw Flight O.D.	Root	Barrel Liner
Homopolymers and unfilled polymers	Standard	ANSI 4140 or Nitrided steel	Nickel alloy	Chrome plated	Bimetallic A
Filled material < 20% loading	Abrasive	Nitrided steel	Tungsten carbide + Nickel alloy	Nitrided steel	Bimetallic B
Highly filled material >20% loading	Highly Abrasive	Bimetallic screw	Tungsten carbide composites	Tungsten carbide composites	Bimetallic B
	Bimetallic A	Chromium-modified boron-iron alloy containing 5 to 7% nickel			
	Bimetallic B	Tungsten Carbide Composite			

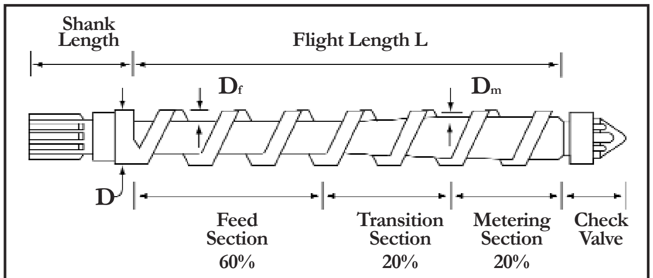
Figure 4B. Barrel and Screw Recommendations for Ultramid **Products.**

Screw design is critical to ensure that proper melt quality is achieved. The two critical parameters to be aware of when molding Ultramid nylon products are the L/D ratio and the compression ratio. These are defined in Figure 4C and shown in Figure 4D.

$$\frac{L}{D} = \frac{\text{Flight length of screw}}{\text{Outside diameter of screw}}$$

$$\text{Compression Ratio} = \frac{\text{Depth of feed section}}{\text{Depth of metering section}} = \frac{D_f}{D_m}$$

Figure 4C



Typical Screw Configuration

Figure 4D

The recommended L/D ratio and compression ratio for molding Ultramid nylon are listed in Figure 4E. When possible, it is recommended that the minimum L/D be 20:1. This will ensure proper melt dispersion.

Ultramid Nylon Material Type	L/D	Ideal Compression Ratio
Unreinforced homopolymers	min. 20:1	3:1
Reinforced material	min. 20:1	2.5:1

Figure 4E

Symptoms of Wear

There are several symptoms of cylinder, screw, and valve wear which can be observed in the molding process.

1. Significant amount of screw rotation during injection indicates a worn barrel and/or check valve. This allows a backflow of melt over the ring, causing the screw flights to counter rotate.
2. Inability of screw to hold a cushion usually indicates a worn cylinder or valve.
3. Excessive recovery time required.
4. Defective, streaked, splayed, or non-uniform parts due to poor melt quality resulting from worn components.
5. Difficulty in achieving consistent color change from plastic hanging up in worn areas of cylinders and screws.
6. Front and center barrel heats may override settings.
7. Inconsistent shot size.

Barrel Sizing

When choosing a press in which to run a mold, it is important to check the shot size as a percentage of the total barrel capacity. Most barrels are rated in ounces of general-purpose styrene (GPPS). To know the shot capacity for any material other than GPPS, the GPPS ounce rating must be converted to the density of the other material. For Ultramid resins, one must insert the specific gravity (S.G. – a measure of density) of the Ultramid resin grade into the formula shown in Figure 4F. The specific gravity is easily obtained from the data sheet for the given Ultramid nylon grade.

$$\text{Barrel Size (oz GPPS)} \times \frac{\text{S.G. Ultramid}}{\text{S.G. GPPS}} = \text{Barrel Size (oz Ultramid)}$$

Figure 4F

For example, if you have a molding machine with a 64-ounce barrel and you want to know the barrel capacity using Ultramid 8233 nylon, the barrel capacity would be calculated as follows:

$$64 \text{ (oz GPPS)} \times \frac{1.38}{1.06} = 83.2 \text{ (oz Ultramid)}$$

Figure 4G

The total shot weight, in ounces (all parts, including runners), is then divided by the Ultramid nylon ounces found in figure 4G. This will yield the percentage of shot size. When injection molding Ultramid products, it is recommended that the shot size not exceed 75% of barrel capacity. Shots larger than 75% may not allow the material to thoroughly melt and mix. On the other hand, a shot size of less than 30% of the barrel capacity is not recommended. This may lead to extended material residence time in the barrel which can, in turn, lead to material degradation, part brittleness and discoloration. This is especially true in products using cadmium-free pigment systems.

On occasions when a mold is already being run in a press, this ratio can be observed by comparing the linear shot size being used to the maximum shot size available and calculating the percent comparison.

Nozzle Tip Type

Reverse taper nylon tip nozzles are recommended when molding Ultramid products. The reverse taper will minimize material drool and stringing which can be encountered when molding crystalline resins. The reverse taper design is suggested when molding unreinforced nylon, and may be used with reinforced grades. However, nozzle bore diameters are recommended to be approximately 25% larger for reinforced Ultramid products than for unreinforced Ultramid nylon due to the higher material viscosities encountered. Typically, general purpose nozzles are recommended when using reinforced grades. Temperature control on the nozzle should be controlled separately from other barrel zones. Extended nozzles may require two or more zones of control. Reference Chapter 7: *Processing Ultramid Nylon* for proper barrel temperature settings.

Below are examples of a reverse taper nozzle design. Figure 4H shows a one-piece nozzle. Figure 4I shows a removable nozzle tip that may be used on general purpose nozzle bodies.

For ease of sprue removal, attempt where possible to design the sprue bushing diameter 0.005"–0.030" (.125mm– .75mm) larger than the nozzle tip orifice diameter.

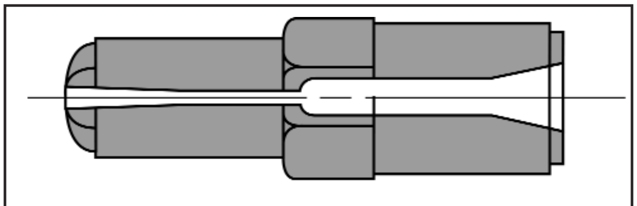

Nylon Reverse Taper Nozzle

Figure 4H

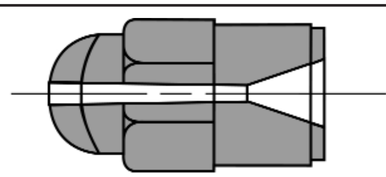

Nylon Reverse Taper Nozzle Tip

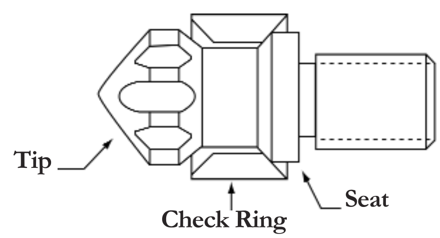
Figure 4I

Check Valve

BASF recommends that the non-return check valve be of the "Free-Flowing" ring design type. This design, while ensuring a consistent shot size, tends to wear the best while running materials containing higher filler content. Maintaining the check valve in a proper working condition is critical to ensuring quality and consistent moldings. There are several issues that may result if the check valve is not functioning properly.

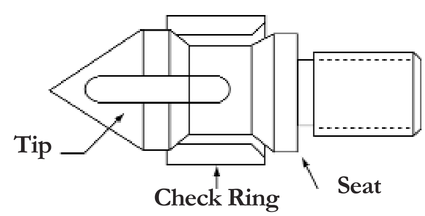
1. Lack of ability to maintain a cushion, resulting in forward screw slippage.
2. Inconsistent shot size.
3. Dimensional inconsistency in parts.
4. Sink marks due to lack of pack pressure.
5. Surface imperfections from splay, whitening, or mineral bloom.
6. Potential to degrade material.
7. Screw rotation upon injection.
8. Possible override of barrel temperature settings.

Below are acceptable types of check ring design that have been used successfully when processing Ultramid resin.



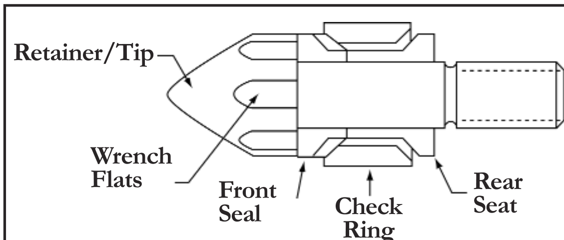
Standard Valve

Figure 4J



Free-Flow Valve

Figure 4K



Four-Piece Valve

Figure 4L

Clamp Requirements

Clamp Sizing

It is important to have adequate clamp force to maintain a fully closed tool during the injection process. High injection pressure, which is often required to fill a cavity, must be offset by pressure exerted by the clamp on the parting line of the tool. If sufficient clamp pressure is not present, the following may result:

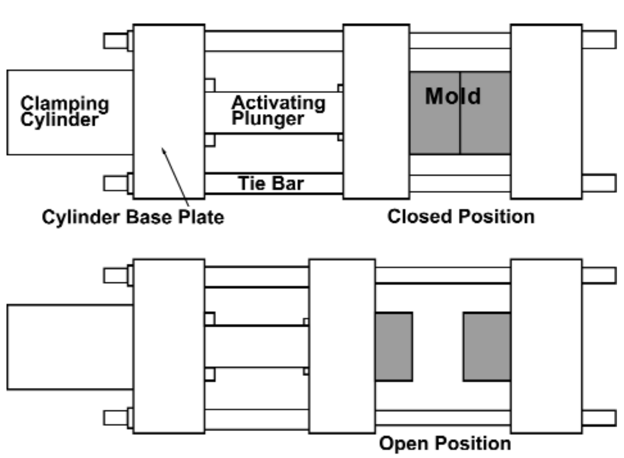
1. Flash at parting line.
2. Peened over parting line resulting from repeated flash.
3. Inability to mold a part that is fully packed out.

Often, larger machines are equipped with proportionally sized injection barrels and clamping systems. However, injection molding machines can be equipped with oversized platens. This will allow larger molds with a relatively small shot size to be processed in a smaller, more economical molding machine.

Traditional clamping systems for injection molding machines fall into two main categories. These include the following types of clamps:

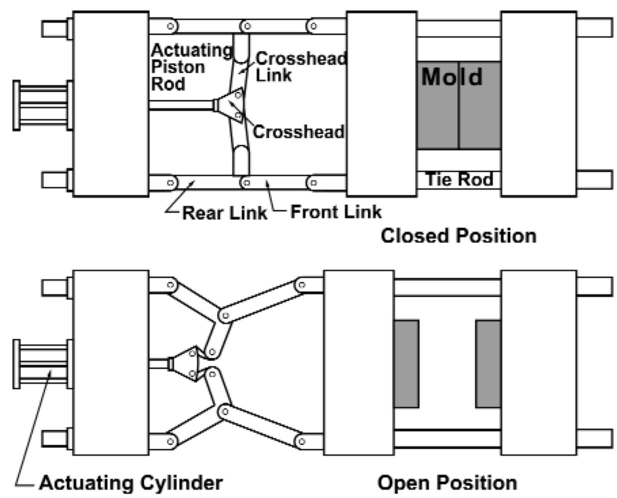
1. Hydraulic Clamp
2. Toggle Clamp

The two main clamping mechanisms and the advantages and limitations of each are shown in Figures 4M and 4N, respectively.



Hydraulic Clamp

Figure 4M



Toggle Clamp

Figure 4N

Clamp Style	Advantages	Limitations
Hydraulic	Fast mold set up. Easily read clamp pressure. Low maintenance. Low platen deflection. Force concentrated at center of platen.	Requires large volume of hydraulic oil. Energy Inefficient. Must overcompensate due to compressibility of oil. Not floorspace efficient.
Toggle	Less expensive. Fast clamp motion. Energy efficient. Auto decelerated clamping.	Requires more maintenance. Clamping force may not be concentrated at center of platen. Difficult to adjust.

Figure 4P

Clamp Force and Cavity Pressure

It is important to verify that the clamp pressure of the molding machine is capable of maintaining a tightly closed mold during the injection process. In other words, the total clamp force exerted must exceed the opposing total force generated during injection. Since the clamping force is usually known, it is important to determine what the injection pressure exerted in the cavity of the tool may be and multiplying this by the projected area of the part and runner. The same equation may also be used to determine if clamp force is sufficient. This is a function of two components; the projected area parallel with the line of draw in the tool and the type of material used to mold the part. The projected area may be calculated by measuring the part. Below is a table of cavity pressure estimates for Ultramid products using the equation provided. The value obtained is the *minimum value* recommended for clamp tonnage in the injection molding machine.

Ultramid Nylon Type	Cavity Pressure Estimate
Unreinforced Polymers	2 – 3 tons/in^2
Reinforced Materials	3 – 5 tons/in^2

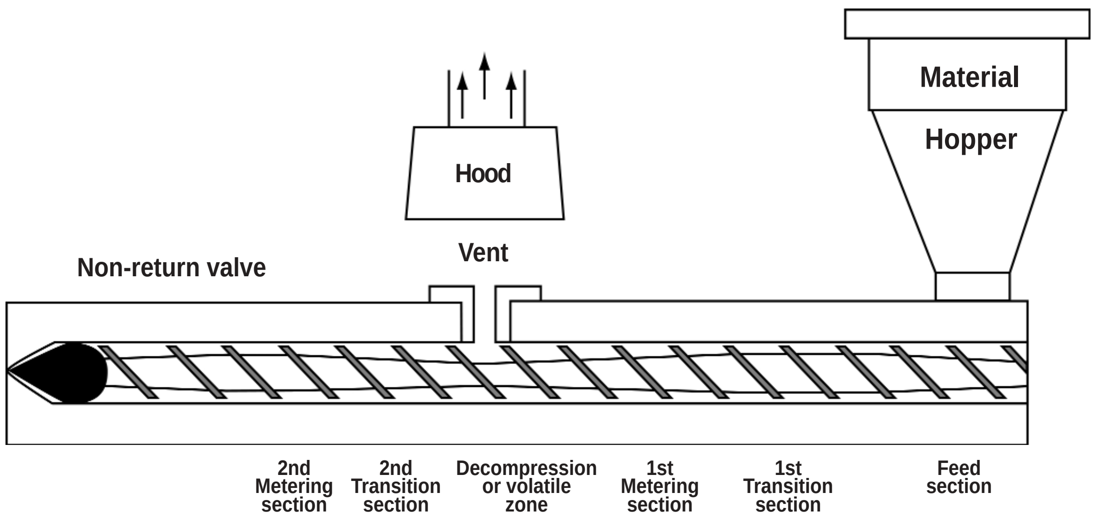
$$\text{Minimum Clamp Force Required (tons)} = \text{Part Projected Area (measured) } (in^2) \times \text{Cavity Pressure Estimate (see above) } (tons/in^2)$$

Figure 4Q

Vented Barrels

Vented barrels are commonly used as a method of removing gases (mainly moisture from hygroscopic materials). The basic concept involves melting the material through the first transition and metering section of the screw and then depositing the material into a decompression zone. At this point, most of the moisture in the material is released from the barrel through a vent. The resin is then processed through a second transition and metering section prior to passing through the check ring assembly. Below is a sketch of a typical vented barrel configuration.

If a vented barrel is selected, BASF recommends the use of a longer screw. Recommended L/D ratio for this application ranges from 26:1 to 32:1. The longer screw will facilitate producing a homogeneous melt. In addition, a hood placed above the vent is recommended to remove the volatiles from the molding facility.



Typical Vented Barrel Configuration

Figure 4R

Many molders prefer vented barrels when processing Ultramid products as a way of removing the moisture from the material. However, in cases where the material contains a very high level of moisture, the vented barrel process is not capable of removing *all* of the volatiles. Below are advantages and disadvantages of using a vented barrel.

Advantages
1. Possibly eliminate pre-drying of the material.
2. Assists in reducing gas entrapment in the tool cavity.
3. Possibly avoid the cost of drying equipment for most Ultramid products, but may require a starve feeder.
4. Easier and quicker material changes.

Disadvantages
1. Potentially greater variation in part quality due to inconsistent levels of moisture in the material.
2. Risk of partially clogged vent ports resulting in poor part quality.
3. Vented barrels may result in longer cycle times and inability to operate successfully at full injection stroke.
4. Longer residence time in barrel possibly leading to resin degradation in smaller shot sizes.

Temperature Control

Heaters surrounding the barrel heat the material in the screw channel by means of electricity, and in some instances hot oil or steam. In addition to this conducted heat, it is important to note that the material is also subjected to shear heat developed by the mechanical working of the material in the barrel by the screw.

A minimum of three heater control zones for the barrel corresponding to the three functional zones of the screw is recommended. However, in most cases additional heater zones are likely when using a larger barrel. Thermocouples are often used and recommended as a temperature feedback to the controller. Maintaining accurate temperature control together with a feedback system will assist in maximum processing potential. Such a feedback system has also been used in data recording for statistical process control purposes.

Chapter 5

Mold & Tooling Considerations

Chapter 5: Mold & Tooling Considerations

Tool Steel Materials

The careful selection of the tool steel for the construction of an injection mold designed for Ultramid resins is important to ensure the long-term durability of the tool. Many of the Ultramid materials contain high levels of filler reinforcement which tend to wear on the surface of the tool. High wear is most effectively countered with high surface hardness. Selecting the right tool steel can increase the useful service life of the tool as well as greatly improve the maintenance of any texturing or graining of the mold surface. The following tool steels are recommended when constructing cavity and core sections.

Material	Recommended AISI-SAE Steel Designation	Tool Steel Characteristics	Typical Hardness as Finished (Rockwell C)
Unfilled Polymers	P20	Medium alloy mold steel	30 – 36 Rc
Reinforced Materials	S7	Shock resisting tool steel	54 - 56 Rc
	H13	Hot work tool steel (Cr based)	50 - 52 Rc
	420	Stainless steel	50 - 52 Rc

Figure 5A

For increased wear resistance, the tools may be hardened, plated, or surface treated.

If the tool is to be grained and surface treated to improve wear resistance, it is recommended that any surface or hardness treatment be performed after the texture process.

Texturing and Surface Finish

Ultramid products, when molded in tooling containing a textured, grained, or polished surface, will reproduce the surface of the tool. All of the tool steels that are recommended for use with Ultramid products can be chemically etched or textured. However, prior to texturing, the mold should be heat treated. This will result in a finer grain structure of the steel which will result in a smoother surface to etch into. Typical texture depths range between .0004" (0.01mm) and .005" (.125mm).

To ensure ease of part ejection and reduce the chance for streaks and scuff marks, the following rule is recommended for incorporating draft into part walls containing a texture.

1° draft + 1.5° draft for each 0.001" (0.025mm) grain depth

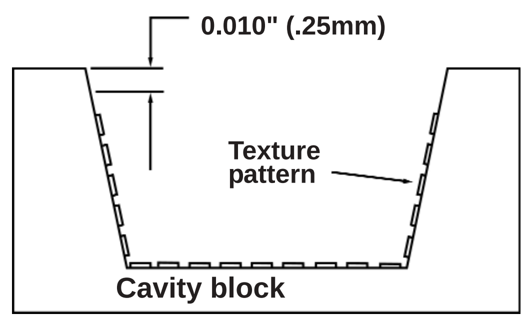
Below is a table of information regarding more commonly used textures and the recommended draft angle.

TEXTURING: Draft Angle Requirements

Grain Type:	Grain Depth: inch (mm)	Min. Draft Angle:
Turf	0.004" (0.0875)	1 + 6 = 7
	0.003" (0.075)	1 + 4.5 = 5.5
	0.002" (0.05)	1 + 3 = 4
Naples	0.0033" (0.083)	1 + 5 = 6
	0.0026" (0.067)	1 + 4 = 5
	0.002" (0.05)	1 + 3 = 4

Figure 5B

In addition (as shown in figure 5C), where possible maintain an area around the parting line perimeter of .010" (0.25mm) without the textured pattern. This will protect the shut off region at the parting line.



Typical Textured Cavity

Figure 5C

Below are several suggestions for tool design when specifying a texture or grain.

1. Prior to texturing, heat treating the tool is recommended.
2. To ensure a consistent texture, the depth of the heat treat into the steel should exceed that of the texture.
3. In order to ensure proper release of the part from textured side walls, do not exceed a depth of etching of .001 inches (.025mm) per 1.5° draft.
4. Texturing of the core half of the mold is not recommended based on potential part release problems.

Sprue Bushing

The sprue bushing is the material entry port into the mold. The injection molding machine nozzle interfaces with the sprue bushing. For ease of part and runner system ejection from the tool, a minimum taper of 1.5°–3.5° is recommended over the length of the sprue bushing. It is also recommended that the sprue be polished in the draw direction.

When molding Ultramid products, it is also suggested that the minimum diameter of the opening of the sprue bushing at the nozzle interface be at least 0.118" (3.0mm).

For best results, ensure that the machine nozzle orifice diameter be less than the sprue bushing inner diameter by .005" – .030" (.125mm – .75mm). This condition will ensure that a smooth transition occurs as the material enters the tool, thereby not creating a shear condition or a pressure drop which can lead to improper packing of the part and surface appearance problems.

The sprue diameter at the intersection of the primary runner should be at least equal to or greater than the diameter or depth of the runner.

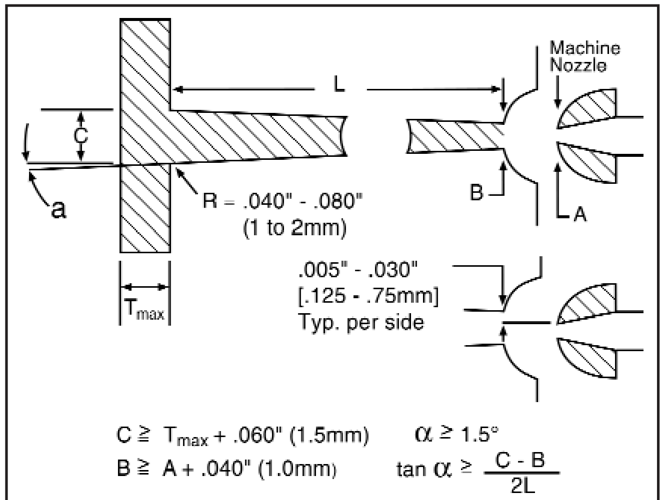
The overall dimensions of the sprue depend primarily on the dimensions of the component to be molded and especially its wall thickness. Following are general guidelines to be considered.

A. The sprue must not freeze before any part cross section in order to permit sufficient transmission of holding pressure.

B. The sprue should demold easily and consistently.

C. It is very important that the radius on the machine nozzle match that of the sprue bushing.

The figure below presents recommendations for sprue design.



Typical Sprue Bushing Design Recommendation

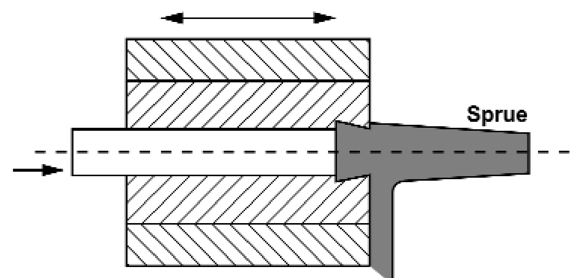
T_{max} = Maximum runner thickness
Dia. A = Diameter of opening at end of machine nozzle
Dia. B = Diameter of sprue at machine nozzle interface
Dia. C = Diameter of sprue bushing at part intersection
L = Overall length of sprue

Figure 5D

It is important that the sprue bushing bore be properly draw polished for ease of demolding and ensuring that the part will run on an automatic cycle consistently without the sprue sticking. Grinding and polishing in a pattern perpendicular to the direction of ejection results in undercuts, which may detrimentally affect the ejection of the sprue.

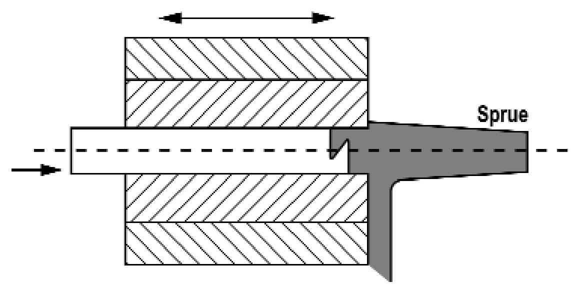
Sprue Puller

Typically, the sprue and the molded part are removed from the cavity at the same time, with both remaining on the moveable (or core) half of the mold. In multi-cavity molds, where a cold runner system is employed, a sprue puller on the moveable half of the mold is recommended. This will ensure that the sprue remains on the moveable half of the tool when the mold opens. A sprue puller is designed with an intentional undercut of various types depending on design, as shown in figures 5E through 5H. The reverse taper sprue puller is recommended, as it typically functions best and will also act as a cold slug well.



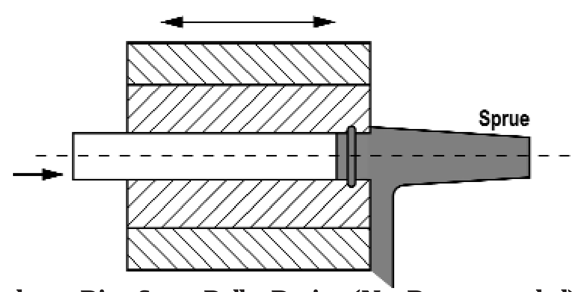
Reverse Taper Sprue Puller Design (Best)

Figure 5E



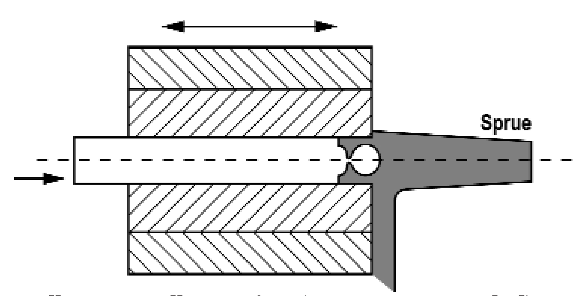
“Z Puller” Sprue Design (Good)

Figure 5F



Undercut Ring Sprue Puller Design (Not Recommended)

Figure 5G



Ball Sprue Puller Design (Not Recommended)

Figure 5H

Cold Slug Well

Use of cold slug wells is always recommended in cold runner systems. A cold well is designed to catch any material in the tip of the machine nozzle that may have cooled below the melting point and begun to solidify. If injected into the part, this cold slug can lead to surface imperfections, such as jetting and gate blush, and also result in a potentially weakened part. Examples of typical cold slug well placement and design are included in Figures 5I and 5J.

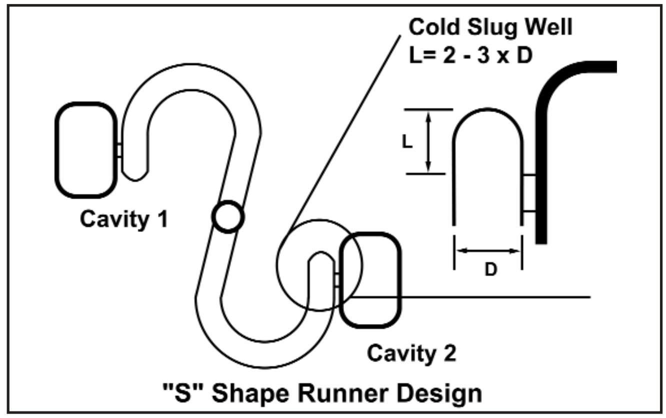


Figure 5I

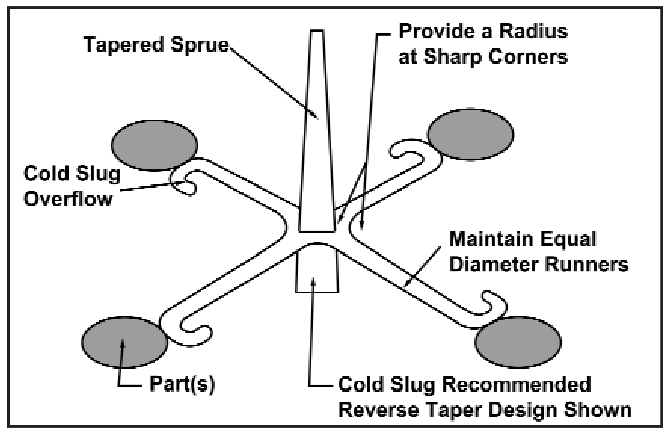
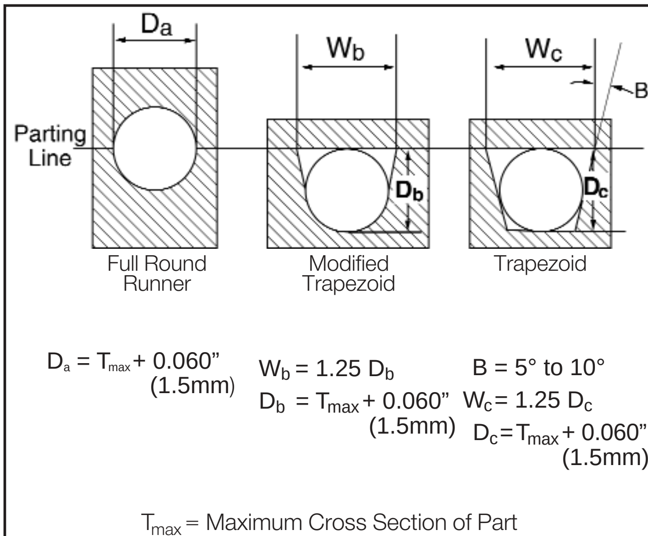


Figure 5J

Runner Systems

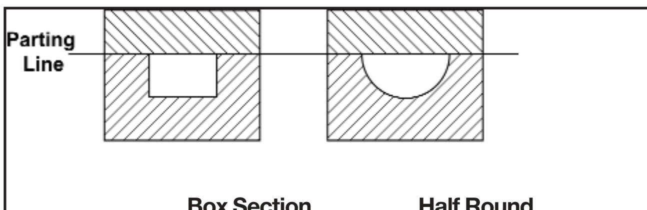
The runner design is a very important phase of tooling design. There are several objectives which the runner must perform to ensure the quality requirements of most parts. Both cold runner and hot runner manifold systems can be used when molding Ultramid products. Insulated runners are NOT recommended. Suggested criteria for runner design are listed below:

1. Minimize restrictions to flow in the runner system, such as inconsistent cross section.
2. Design for ease of part ejection.
3. Overall length should be as short as possible to reduce losses in material pressure and temperature and excessive regrind generation.
4. Make runner cross section large enough whereby runner freeze-off time exceeds that of the gate. This ensures that proper hold pressure is applied.
5. The runner system should not be the limiting factor when reducing cycle time.
6. Minimize rate of runner weight to part weight without conflicting with other guidelines.



Recommended Runner Designs

Figure 5K



Unfavorable Runner Designs

Figure 5L

Runner Style	Advantages	Disadvantages
Full Round	1. Smallest surface to cross section ratio. 2. Slowest cooling rate. 3. Low heat and frictional loss. 4. Center of channel freezes last; maintains hold pressure.	Matching into cavity/core difficult.
Modified Trapezoid	1. Easier to machine; usually in one half of tool only. 2. Offers similar advantages of full round.	More heat loss and scrap compared to full round.
Trapezoid	Easy to machine.	More heat loss than modified trapezoid.
Box Section	Easy to machine.	1. Reduced cross section efficiency. 2. Reduced ability to transfer pressure. 3. Difficult to eject.
Half Round	Easy to machine.	1. Smallest cross-sectional area. 2. Most inefficient runner design. 3. Poor pressure transmission into cavity. 4. Generates more regrind.

Figure 5M

Cold Runner Design

The full round runner is recommended. This type allows for the most efficient material flow and tends to induce the least chilling effect on the material. Trapezoid and modified trapezoid runners are also feasible. However, resin flow to the part is less efficient with these designs. They also tend to induce more of a chilling effect on the material in the runner and generate more regrind. The half round runner is not suggested because it does not provide for optimum flow and it causes the greatest chilling effect on the resin. If the material in the runner freezes prematurely, the material in the cavity will not be adequately packed, which may lead to excessive shrinkage or other problems.

In addition, to ensure part to part consistency, the runner length from the sprue to each cavity should be of the same diameter and length. By balancing the cavities in this fashion, you will ensure that each cavity receives equal flow and pressure simultaneously.

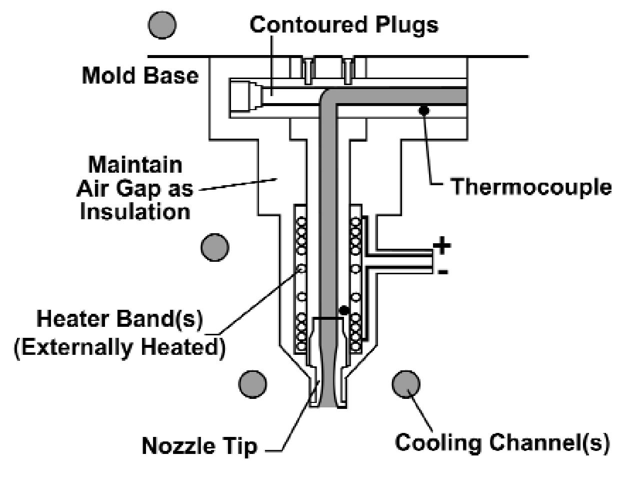
The table below shows suggested runner diameters and corresponding runner lengths and part thicknesses.

Primary Runner Diameter	Maximum Length	Maximum Part Thickness
0.125" – 0.187" (3.18mm – 4.75mm)	6.0" (152mm)	0.187" (4.75mm)
0.25" – 0.312" (6.35mm – 7.94mm)	12.0" (304.8mm)	0.50" (12.7mm)
0.375" (9.53mm)	15.0" (381mm)	0.75" (19.05mm)

Table 1A

Hot Runner Design

Both hot runner and cold runner systems can be utilized when molding Ultramid products. When using a hot runner system, the resin is injected from the machine barrel into a heated manifold network within the tool. An externally heated hot manifold system is recommended for Ultramid resins. This manifold commonly directs the material through a series of heated channels to the location of the gate in order to fill out the part.



Hot Runner and Nozzle System

Figure 5N

Suggestions for designing hot runner systems for optimum performance:

1. When machining the melt passage in the manifold, take care not to create any dead spots where material may hang up. Over time, this material may degrade and contaminate the material flowing through the system.
2. Placement of heaters in the manifold design is critical to ensuring uniform heat transfer in the melt, thereby avoiding a cold spot in the system which may lead to freeze-off or uneven fill patterns.
3. Reduce contact areas between the hot runner manifold system and the tool steel. Heat transfer to the mold from the manifold should be minimized.
4. Where possible, attempt to locate cooling lines away from the manifold system. This can also lead to undesirable heat transfer out of the manifold.
5. The use of a temperature insulator is recommended between the mold base and the machine platens to reduce heat loss (especially on the stationary half).
6. To eliminate the chance for electrical interference, which may lead to false thermocouple readings, try to locate the heater wire leads away from the thermocouple wires.
7. Proper water cooling around the gate orifice is key to control. Also critical is a separate zone of temperature control for the gate area of the hot manifold tip.

Gate Design

The gate connects the part to the runner. It is usually the smallest cross section in the entire system. Designing the gating concept for a part is highly dependent on both tool design and part geometry. Often, the most desirable gating scenario is not feasible due to tooling or part design limitations. Equally important to molding a successful part is the location of the gate on this part.

As a guide, the following gating configurations are presented:

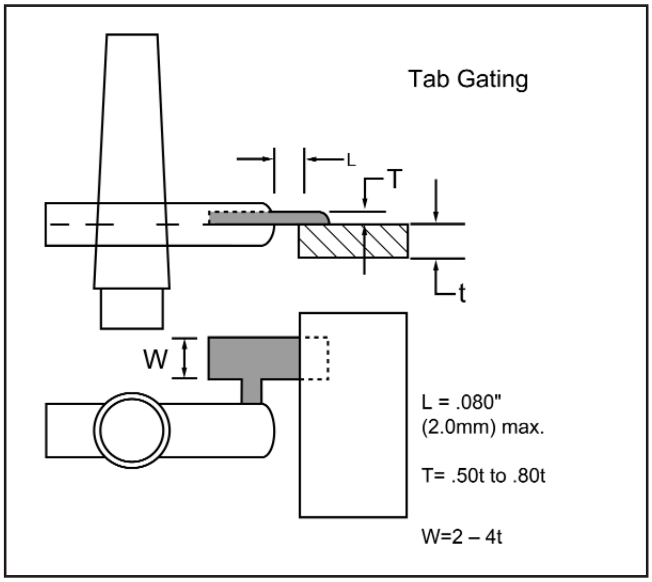


Figure 5P

Tab or film gating, Figure 5P, is often used where flatness is critical or in large surface areas where warpage may be a concern. Due to the nature of this type of gate, a post molding operation is typically required to properly remove the gate vestige.

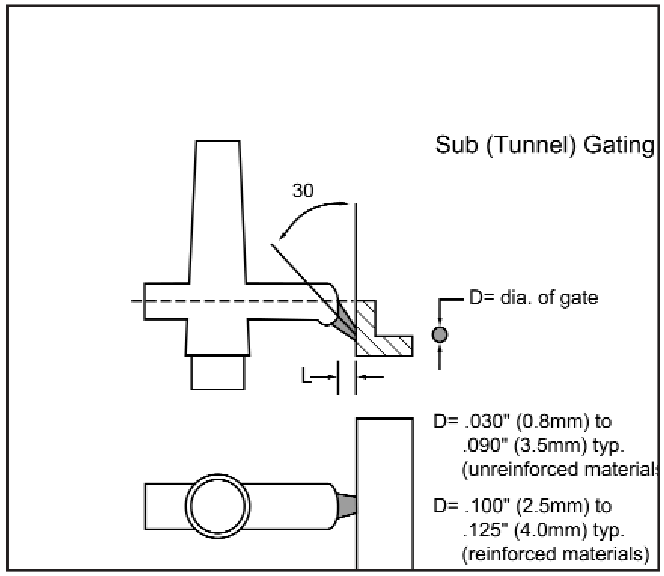


Figure 5Q

Sub gating, Figure 5Q, can be designed to provide automatic degating of the part from the runner system during ejection. Sub gate size is highly dependent on both part size and tooling limitations. Each application should be thoroughly reviewed.

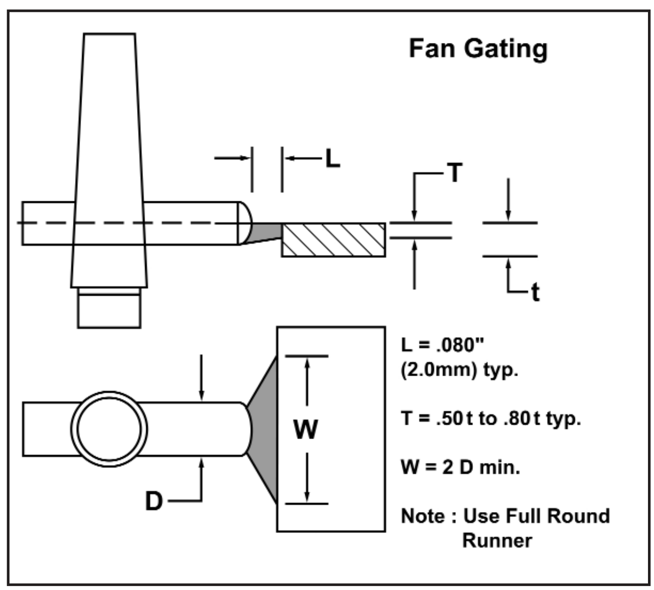


Figure 5R

The fan gate or the edge gate, shown in Figures 5R and 5S, is often used to feed flat, thin sections which will tend to allow the material to flow across the cavity in a uniform fashion. It also has proven successful in reducing warpage. The gate cross-sectional area should always be less than the cross-sectional area of the runner. The edge gate is the most common gate type used and generally presents a good compromise between ease of part filling and gate removal.

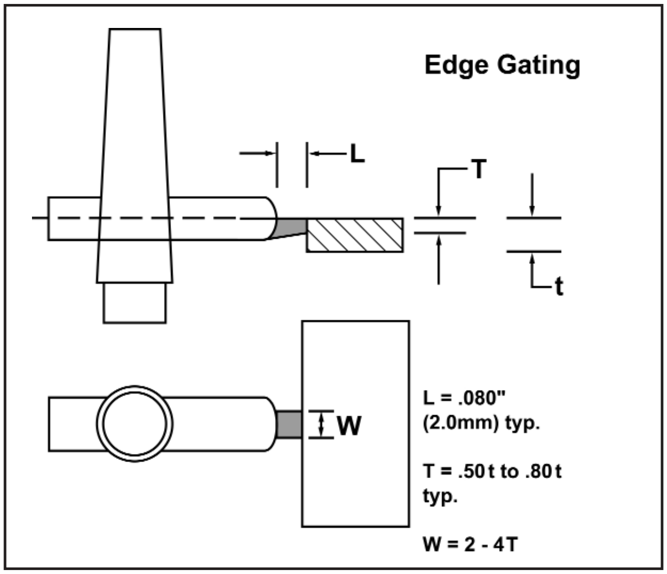
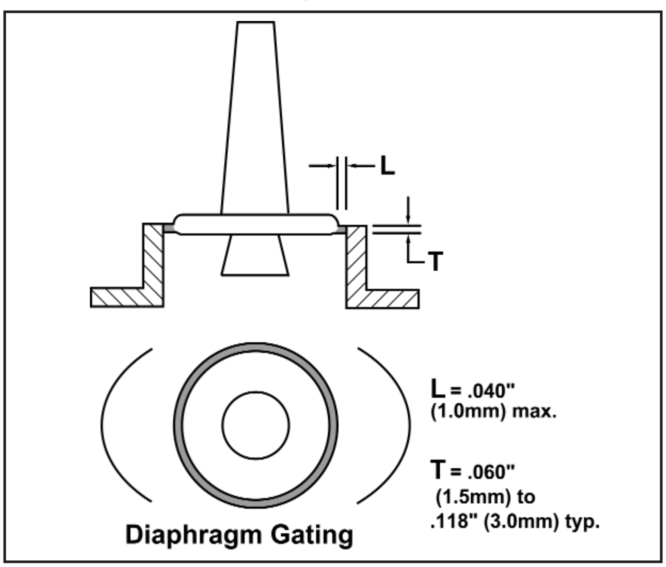


Figure 5S

The diaphragm gate, shown in Figure 5T, is used when molding cylindrical parts requiring a high level of concentricity and weld-line strength. However, due to the nature of this type of gate, a post mold degating operation is typically required.

Figure 5T



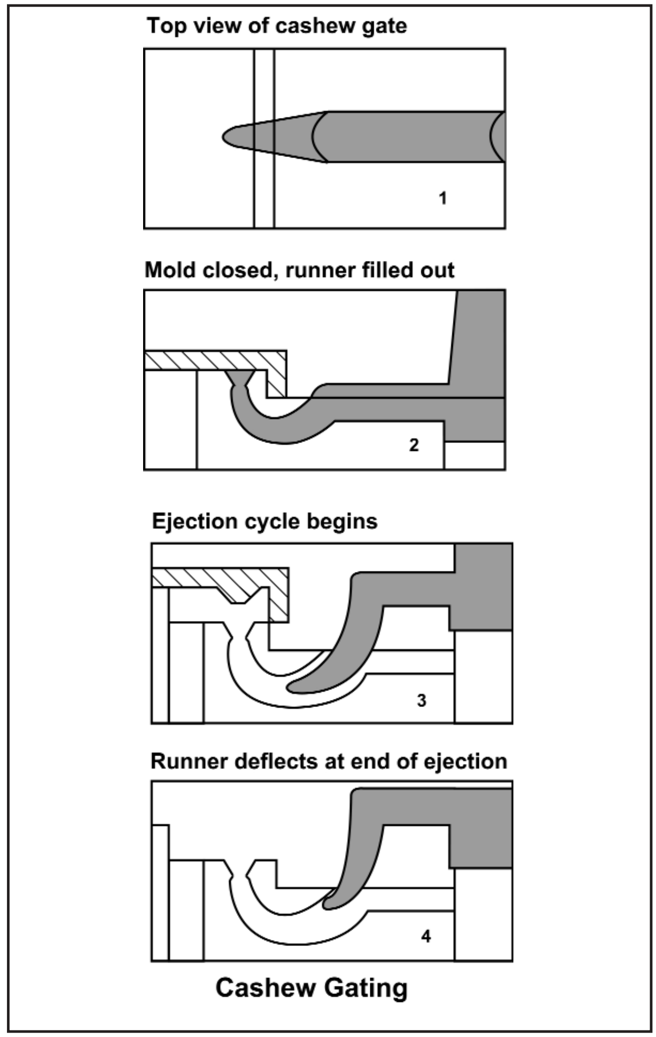


Figure 5U

Cashew gating can be highly effective when using more flexible materials. Due to the required degree of bending of the runner, as shown in sketches 3 and 4 of Figure 5U, filled or stiffer materials are not recommended for use with this type of gate as they often break during ejection. This could lead to broken ---pieces of plastic becoming lodged in the tool, plugging the gate on the next shot. Therefore, it is recommended that only unfilled materials be used with this type of gating.

Gate Sizing

Figure 5V shows the recommended gate thicknesses for both filled and unfilled grades of Ultramid resins for typical part thickness cross sections. These values can be applied when using all types of gates.

Recommended Gate Thicknesses

Typical Part Thickness	Unfilled Ultramid Nylon	Filled Ultramid Nylon
Up to 0.060" (1.5mm)	0.040" (1.0mm)	0.040" – 0.060" (1.0 – 1.5mm)
Up to 0.125" (3.2mm)	0.060" – 0.090" (1.5 – 2.3mm)	0.060" – 0.125" (1.5 – 3.2mm)
Up to 0.187" (4.7mm)	0.090" - 0.125" (2.3 – 3.2mm)	0.125" – 0.187" (3.2 – 4.7mm)
Up to 0.250" (6.4mm)	0.125" – 0.187" (3.2 – 4.7mm)	0.187" – 0.250" (4.7 – 6.4mm)

Figure 5V

If the gate is to be enlarged for increased material flow and pressure transfer, focus on increasing gate thickness rather than the width. Keeping the gate to a square (or round) cross section will be the most efficient.

Gate Location

Gate location is critical because it ultimately determines the direction of the material flow within the cavity. In many cases, this has an effect on the following factors:

- Shrinkage
- Physical properties
- Distortion or warpage
- Part appearance

The above factors are a result of the orientation of the molecular structure of the material or any fillers that may be present. Note that:

1. The amount of orientation is higher in thin walled moldings.

2. Higher strength and impact resistance values are observed in the direction of flow while sections in the perpendicular direction may exhibit reduced toughness.

Therefore, prior to designing the injection mold and gate location(s), the mode of stress loading that the part will experience should be determined.

The placement and quantity of gates required will have an effect on the overall flow length and orientation of the material. Typically, the maximum flow length from each gate for Ultramid nylon in an injection mold is 15 inches (381mm). This value is highly dependent on runner diameter, runner length, part geometry, and part thickness.

The following suggestions may be used when proposing gating location concepts for Ultramid products:

1. Direct incoming flow against the cavity wall or core to minimize gate blush and jetting.

2. Avoid gating that will cause melt fronts to converge such that air is entrapped. Where possible, attempt to direct flow fronts and air toward vents.

3. If possible, position the gate location at the thickest (low pressure) section of the part. Always try to flow from thick to thin sections.

4. Select gate location to obtain the best strength relative to loading. Tensile and impact strength are highest in direction of flow, especially with filled or reinforced Ultramid nylon.

5. The gate should be positioned away from any area of the part that will be subject to impact or bending stress. The gate area tends to contain high residual stresses from the filling process may become a likely site for fracture initiation.

6. Minimize weld lines especially in impact or highly stressed areas. Locate weld lines to thicker areas on the part.

7. In multiple cavity tooling applications, it is imperative that each gate among the various cavities be of the same size (diameter, thickness, etc.). This will ensure equal pressure and flow to each cavity.

8. If possible, locate the gate in an inconspicuous area of the part where finishing will not be required.

Venting

During the filling of the cavity with Ultramid resins, the melt has to displace the air which is contained in the cavity. If there is nowhere for this air to go, it may compress, forming a pressure head that will resist the flow of plastic. As the air compresses, it will also heat. In some cases the air can reach temperatures that exceed the ignition temperature of the plastic and volatiles that are present. This results in a burn line where the trapped air contacts the plastic, which can produce an undesirable charred blemish on the surface of the part and may even oxidize and erode the mold.

Part geometry, position in the mold, and gating location all have a big impact on the venting.

Suggestions for locating and designing vents into tools designed for Ultramid nylon:

1. Tooling inserts may be incorporated at ribs or part sections that tend to trap air. The mere existence of the insert may alleviate a trapped air problem.

2. When molding thin walled parts <.125" (3.0mm) with a high injection rate, the cavity should be vented close to the gate as well as at the extremity of flow. Air removal from the tool reduces the chance of a significant pressure build-up.

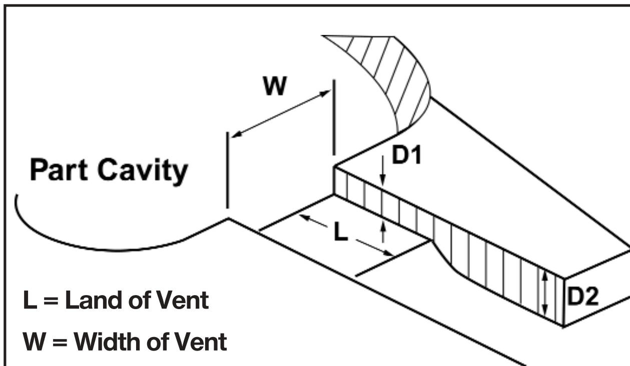
3. Avoid venting to internal pockets in the tool. Vent to atmosphere.

4. Thick walled > .125" (3.0mm) parts with high surface appearance requirements may require extra venting.

Suggested concepts which may be used for incorporating vents:

1. Parting line vents.
2. Ejector pins (flats ground on pins).
3. Add venting pins.
4. Incorporating inserts at sections that trap air.
5. Add an overflow well.
6. Sintered metal inserts.

Figure 5W shows a sketch of a typical parting line vent and the corresponding dimensions are shown in Figure 5X.



L = Land of Vent
W = Width of Vent
D1 = Depth of Vent
D2 = Depth of Relief

Typical Parting Line Vent Design

Figure 5W

Material Type	Vent Dimensions L	W	D1	D2
Unfilled	0.03" – .06"	.375" – .5"	.0005" – .001"	0.01"
	0.75 – 1.5mm		9.5 – 12.5mm	.013 –
.025mm	0.25mm			
Mineral Filled	0.03" 0.75mm	.375" – .5" 9.5 – 12.5mm	.001" – .002" .05 – .05mm	0.01" 0.25mm
Glass Filled	0.03" 0.75mm	.375" – .5" 9.5 – 12.5mm	.001" – .002" .03 – .05mm	0.01" 0.25mm

Figure 5X

The figures below show two venting schemes that are commonly used when relieving vents to atmosphere. Figure 5Y shows the use of vent channels from the parting line to the edge of the tool. The venting scheme in Figure 5Z ensures a positive shut off area outside the parting line area and then relieving the entire parting line. This latter design is referred to as continuous venting.

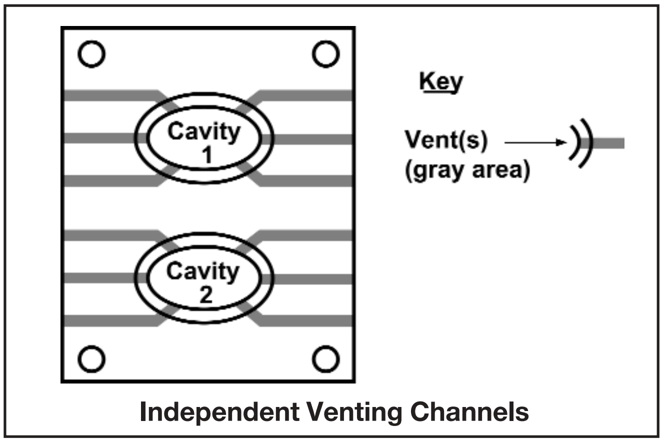


Figure 5Y

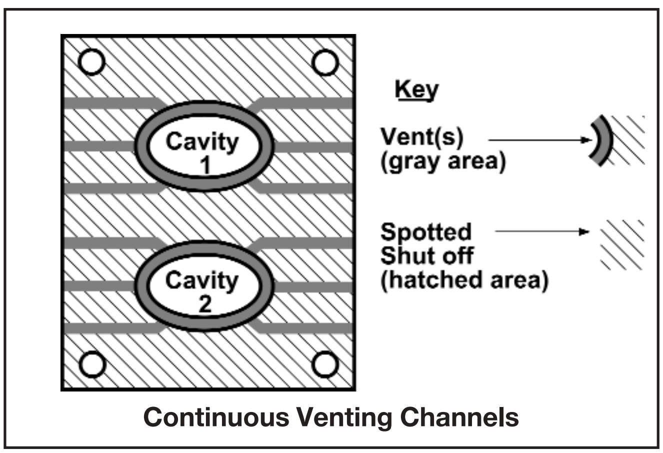


Figure 5Z

Cooling

To ensure optimum molding cycles while maintaining part surface requirements and mold filling capability, a stable mold temperature should be determined and maintained. This is accomplished by incorporating cooling lines throughout both the cavity and core of the mold. It is important that the temperature variation throughout both halves of the part cavity be kept to a minimum. When designing the tool, take into account potential hot spots such as hot runner manifolds and gate locations as well as cold areas such as the area last to fill. Hot spots and cool spots often require extra water channels to maintain temperature consistency throughout the tool.

Inconsistent mold temperatures may lead to the following problems:

1. Non-uniform part surface finish.

2. Non-uniform part shrinkage and warpage.

3. Lack of part dimensional control.

4. Potential binding of tightly fitting cavity and core sections.

In general, it is advisable to maintain less than a 20° F (11° C) differential in tool steel temperatures over the molding surface of a large mold and 5° F (3° C) with smaller tools. Designing for a tighter tool temperature range will assist in providing a greater processing window. Figure 5AA shows a typical design of a cooling channel layout arranged around the part surface. To maintain a stable process, tool temperature can be best controlled by incorporating a sufficient amount of water channels and placing them approximately .5 inch (13mm) from the part surface. Also, where the design permits, place the lines no more than 2 inches (51mm) apart for maximum control.

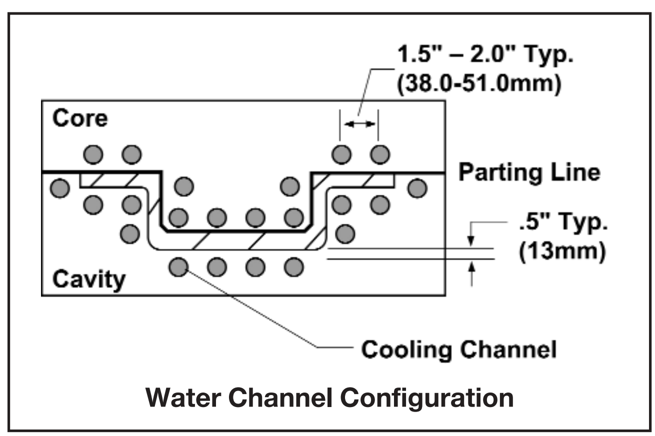


Figure 5AA

Often, part designs make the installation of adequate cooling lines difficult. In those cases, techniques other than a typical cooling line can be utilized, such as:

1. Heat pipes
2. Air pipes
3. Bubblers
4. Baffles

In addition, alloy materials such as beryllium copper offer thermal conductivity values upwards of 10 times that of standard P-20 steel. These materials can be inserted in areas of the tool where water lines may be difficult to install, while maintaining tool durability. For best efficiency, water lines should always go through a portion of the beryllium copper insert.

The following suggestions are recommended when designing and maintaining cooling systems of the injection mold:

1. Use a pyrometer to check that the cavity surfaces of the tool are at the desired temperature.
2. Minimize looping of hoses to maximize cooling efficiency.
3. Blow out water lines with compressed air to remove any foreign matter that may have collected.
4. Attempt to flow opposite directions (horizontal to vertical) between cavity and core.
5. Prior to mold start up, check all waterlines to ensure coolant flow is occurring.
6. Ensure that the flow pattern through the tool meets requirements for turbulent flow. Cooling simulation packages can be used to evaluate this.
7. To ensure operator safety, allow hot molds to cool prior to removing the mold from the molding machine.
8. Maintain and frequently check for wear or weakening of cooling hoses. With conventional rubber hoses, do not exceed 200°F (95°C) water temperature to ensure operator safety and the long-term integrity of hoses. Temperatures in excess of 200° F (95°C) require steel-braided hose.

Shrinkage

Shrinkage is the difference between the dimensions of the cooled part and the tool. Ultramid nylon 6 products are considered semi-crystalline products. Having this inherent characteristic, Ultramid products tend to exhibit shrinkage resulting from a decrease in part volume as material crystal-lization occurs. Resins with higher filler content will have less shrinkage. When designing the injection mold, it is important to specify the proper material shrinkage in order to achieve a part that meets your dimensional requirements.

Below are several factors which may affect the shrinkage of an injection molded component:

1. Location and size of the gate(s). Shrinkage is usually greater in the cross flow direction.
2. Part designs that contain large variations in cross section thickness can lead to unequal stresses throughout the part and tend to cause differential shrinkage and warpage to become more pronounced.
3. Increased glass content as a filler reinforcement will tend to lower the shrinkage of the material.
4. Random glass orientation, which can result from tool design scenarios containing multiple gates, can be effective in leading to a more uniform part shrinkage condition.
5. Thicker areas shrink more than thinner areas.

Refer to Figure 5BB which suggests the typical material shrinkage values for the Ultramid products. Nonetheless, there are many conditions, including part design and material processing, that can affect part shrinkage. As an accurate prediction mechanism, prototyping is often the most valuable tool for determining precise part shrinkage. Please contact a BASF Technical Development Engineer for assistance in determining material shrinkage.

Category Product	Ultramid	Shrinkage
General Purpose Homopolymers	8200 HS	0.012
	8202 HS	0.012
Flexible and Impact Resistant	8253 HS	0.012
	8254 HS	0.013
	8350 HS	0.014
	8351 HS	0.014
	BU50I	0.018
Reinforced for High Stiffness and Strength	8230G HS	0.008
	8231G HS	0.005
	8232G HS	0.004
	8233G HS	0.003
	8234G HS	0.002
	8235G HS	0.002
	8262G HS	0.008
	8266G HS	0.004
	8267G HS	0.004
	8331G HS	0.005
	8332G HS	0.004
	8333G HS	0.003
	8334G HS	0.002
	HMG10	0.002
	HMG13	0.002
	SEG7	0.003
	SEGM35	0.004
	TG3S	0.004
	TG7S	0.003
Mineral Reinforced	8260 HS	0.009
	8360 HS	0.010
	8362 HS	0.010

Tested samples were 5.0" x 0.5" x 0.125" (127mm x 12.7mm x 3.2mm) molded bars. These values are presented as a guide. Shrinkage values may be different depending on the actual application, including part design, mold design, and processing variables.

Figure 5BB

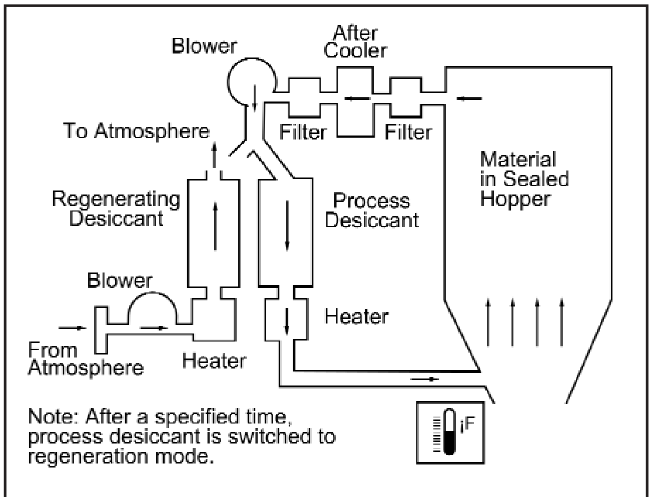
Chapter 6

Auxiliary Equipment

Chapter 6: Auxiliary Equipment

Material Drying

Since Ultramid resins are hygroscopic materials, they must be in a dry condition prior to injection molding. (Reference Chapter 7: *Processing Ultramid Nylon* for drying recommendations). Therefore, the dryer is an important piece of equipment. For optimum drying conditions, BASF recommends a closed loop desiccant dryer. A typical dessicant drying system is depicted in figure 6A.



Typical Dessicant Dryer Bed

Figure 6A

Dryers should provide uniform and even distribution of air through the hopper. The critical temperature when measuring the heated air is the temperature at the entrance to the hopper. **Recommended air flow into the hopper is one cubic foot of air per minute for every pound per hour of use.** If 200 lbs of material are consumed per hour, the airflow to the hopper should be 200 cubic feet of air per minute.

Below are several guidelines for using and maintaining dryer systems:

- Clean the filters on the dryers on a regular basis to ensure that the specified airflow is maintained.
- Avoid loading regrind with high levels of fines or small dust-like particles into the dryer.
- Using a dewpoint meter, check the dewpoint of the air entering the hopper. The recommended dewpoint should range between -20° F and -40° F (-30° C and - 40° C).
- Change and maintain the desiccant in the dryer per the manufacturers recommendations.

Mold Temperature Control

Mold (water) heaters are commonly used to maintain a consistent temperature throughout the injection mold. In most cases, water is used as the heat transfer media, however, ethylene glycol mixtures as well as oil are commonly used. Occasionally, two heaters are required per mold to allow different mold temperatures between the cavity and the core.

Selecting the proper heater to use for a particular mold is very important to ensure an efficient process. An undersized heater may be less of an upfront cost but the operating costs may outweigh the original savings. This is due to the fact that in many undersized cases, the heater will be required to run at 100% capacity for extended periods of time.

Mold heaters are commonly rated in tons. One ton equals the ability to transfer 12,000 BTU/hr. To calculate the appropriate size thermolator for each mold, the formula in Figure 6B can be used.

$$\frac{A \times (B - C)}{12{,}000} = \text{Required Tonnage}$$

A = Actual material used in lbs/hr
B = Temperature of melt (°F)
C = Temperature of part when it comes out of the mold

Figure 6B

Sufficient water flow through the tool is also critical to ensure that an efficient heat transfer process is occurring. This is demonstrated in Figure 6C. As the gallons per minute of flow (GPM) through the tool decreases, the difference between the tool inlet and outlet temperatures (Delta T) must increase dramatically. Therefore, to run at a lower flow rate through the tool and still achieve the results of running with a higher flow, the inlet coolant temperature must be set significantly lower. This results in lower mold heater settings, which may be costly to operate.

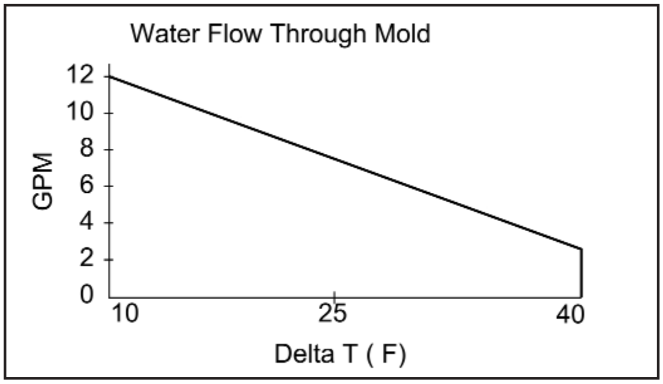


Figure 6C

Following are several other suggestions that may be helpful when designing the coolant system for a specific tool:

1. Typical temperature losses through the coolant handling system is 20%. This includes the coolant lines, mold base, and platens.

2. Antifreeze does not transfer heat as well as water.

3. If antifreeze is needed, use only *inhibited* ethylene glycol with high corrosion resistance. A straight ethylene glycol can become acidic and corrosive with use and may form a gel within two years in open air systems.

4. Where equal heat transfer is desired, ensure that equal coolant flow is occurring between the coolant channels. This may be accomplished by the use of adjustable flow regulators.

Granulators

Many molding applications allow the use of regrind back into the process. This will require the use of material grinders. Many styles of granulators have been used successfully with Ultramid resins. To ensure a consistent regrind blend, granulator maintenance is often critical. In addition to maintenance, proper safety should be observed in accordance with the manufacturer's recommendations.

The following are suggestions to maintain a consistent regrind blend while maintaining the equipment in proper working order:

1. Following material changes, the granulator should be cleaned and vacuumed to remove any foreign material that may contaminate the blend.

2. Blades should be sharpened and screens cleaned regularly to ensure a consistent regrind pellet size.

3. When grinding filled materials, for prolonged periods, components, such as blades, rotors and screens, should be made of hardened materials to resist the erosive effects of these more abrasive plastic compounds.

4. In many instances, ear protection is recommended when operating a granulator.

Chapter 7

Processing Ultramid Nylon

Chapter 7: Processing Ultramid Nylon

Processing Conditions

Ultramid resin has very good molding characteristics due to its excellent ability to flow. Since Ultramid products have a wide processing window, the following conditions are typical recommended settings that can be used at machine set up. Included in this chapter are the key processing variables that may be manipulated in order to control the molding process.

Drying

Ultramid resins are hygroscopic materials and are supplied dry. Therefore, care should be taken to ensure that the material remains in an enclosed container prior to molding as nylon will tend to absorb moisture from the atmosphere over time. As supplied from BASF in sealed boxes and bags, Ultramid resin is usually dry enough for injection molding. However, a moisture analysis of the material is always a good idea to ensure that the material is within the recommended moisture range. Figure 7A shows the moisture content recommendations for injection molding Ultramid resins.

Ultramid Product	Recommended Moisture Content (by weight)
Unreinforced homopolymers	0.10 – 0.20%
<20% Reinforced	0.10 – 0.16%
>20% Reinforced	0.06 – 0.12%

Figure 7A

In order to ensure proper material properties, the moisture content should not be dried to moisture levels below 0.02%.

If the material contains higher levels of moisture than listed on the chart, drying may be required to ensure proper processing and desired part quality. A dryer (as discussed in Chapter 6) containing a desiccant system is highly recommended for best results. Figure 7B gives the drying recommendations for Ultramid products. These values are for 100% virgin products.

The data presented are general guidelines for moisture content. Refer to the particular resin's property data sheet for more specific recommendations.

Recommended Dryer Settings			
	Dryer Inlet Air Temperature	Dryer Dewpoint	Drying Time
Ultramid Nylon 6	180°F (82°C)	<-20°F (-30°C)	4 hours
NOTE: Not to exceed 250°F (121°C) for two hours.			

Figure 7B

Moisture absorption and moisture loss is a function of pellet surface area. In other words, the smaller the pellet, the more surface area per weight, and therefore, the more moisture that it will absorb or lose in a given period of time. Therefore, when molding regrind, which may contain larger sized pellets, additional drying time at 180° F (82°C) may be required.

To ensure that the material is dried for the recommended time, the equation in Figure 7C may be used to calculate the residence time of the material in the dryer for a continuously running process. To perform this calculation, the following variables are needed:

1. Total shot weight (including sprue and runner)
2. Total cycle time
3. Total dryer capacity

These values are then entered in the appropriate position in the equations below:

A $$\frac{\text{Shot Weight (lbs)}}{\text{Total Cycle Time (sec)}} \times \text{3600 (sec/hr)} = \text{Dryer Thruput Rate} \left(\frac{\text{lbs}}{\text{hr}}\right)$$

B $$\text{Total Dryer Capacity (lbs)} \div \text{Dryer Thruput Rate} \left(\frac{\text{lbs}}{\text{hr}}\right) = \text{Residence Time in Dryer}$$

Figure 7C

Below are key recommendations for maintaining an acceptable moisture level for injection molding Ultramid products:

1. Prior to molding, store the material in a container that is sealed from the atmosphere.
2. Minimize the distance from the dryer to the machine hopper and/or dryer hopper in an effort to minimize heat loss through the connecting hoses.
3. Check the drying equipment for crimped or cracked hoses which may reduce the heat transfer to the hopper.

Melt Temperature

The actual melt temperature is influenced by barrel temperature settings, screw design, screw RPM, back pressure, material residence time and shear heat through the nozzle. Since it is difficult to estimate the effect that each variable has on the process, the actual melt temperature should be measured with a pyrometer. Actual measurement of the melt temperature will ensure process repeatability. The barrel temperature profiles in Figures 7D through 7I are representative for processing of the various categories of resins.

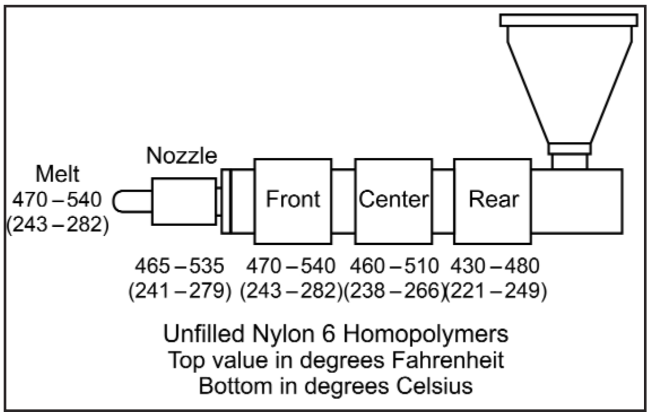


Figure 7D

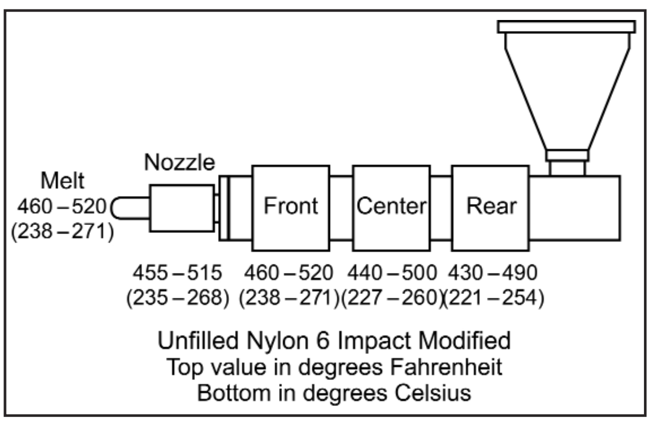


Figure 7E

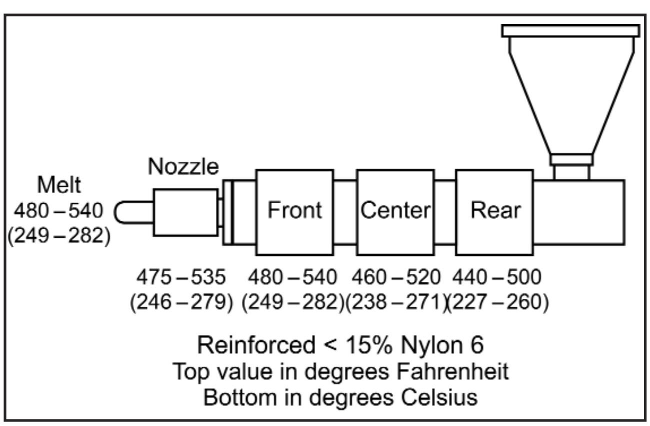


Figure 7F

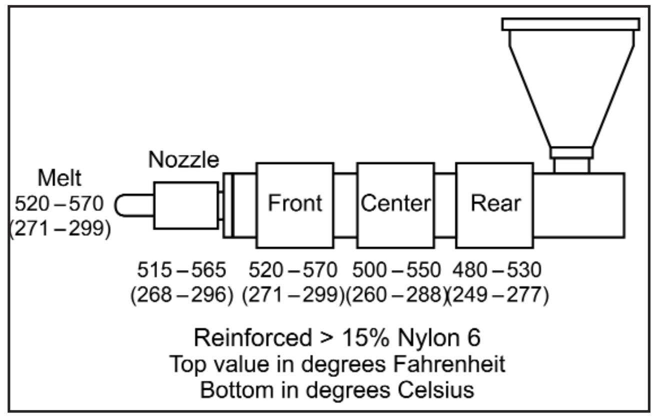


Figure 7G

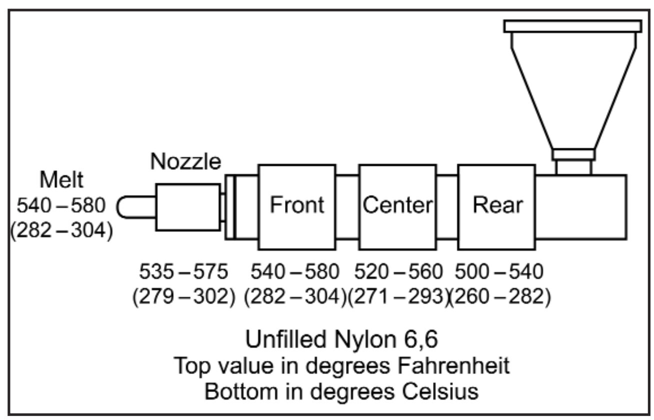
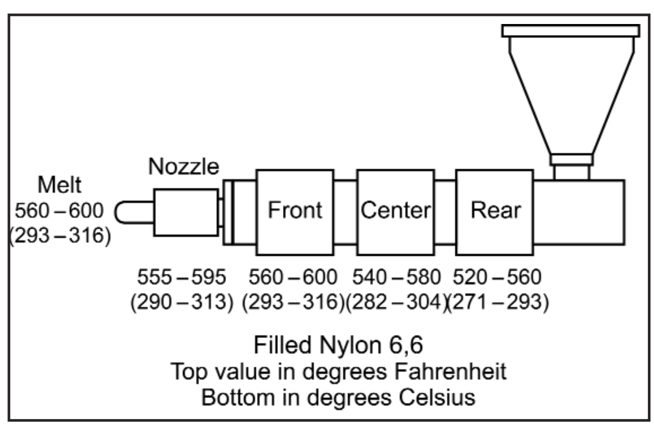


Figure 7H



Hot Runner Systems

The purpose of a hot runner system is to maintain heat in the material and not add, nor remove, heat. When molding Ultramid resins through a hot runner system, use standard injection molding processing guidelines. Recommended manifold temperature settings should be similar to the front or center zones of the machine barrel.

Residence Time

Residence time of Ultramid resin in the barrel at processing temperatures should be minimized to 3–5 minutes. Residence time may be calculated by using the equation in Figure 7J.

Calculating Residence Time

$$\frac{\text{Total Shots}}{\text{Barrel}} \times \frac{\text{Cycle Time (sec)}}{60} = \text{Residence Time (minutes)}$$

Figure 7J

This calculation (7J) will be a rough estimate only, as at any given time there is significantly more material in the barrel than 100% of the maximum shot size.

Mold Temperature

Ultramid nylon 6 should always be molded in temperature-controlled molds. Recommended mold temperatures range from 50° F to 200° F (10° C to 93° C). For best properties and cycle time, a mold temperature of 180° F (82° C) is recommended. When molding parts that require good aesthetics, a mold temperature between 180° F to 200° F (82° C to 93° C) is suggested.

Ultramid nylon 6,6 resins should also be molded in a temperature-controlled mold. Recommended mold temperatures range from 110° F to 220° F (43° C to 104° C).

Mold temperature will influence cycle time, part dimensions, warpage and mechanical properties. The effects of varying mold temperature when molding Ultramid resins are listed in Figure 7K. In some cases, it may be required to elevate or reduce the mold temperature to attain the desired results.

Mold Temperature

Lower	Higher
Minimal shrink	Maximum material shrink
Reduced cool (cycle) time	Increased flow
Higher molded-in stress	Improved knit line strength
Less than optimum surface	Reduced molded-in stresses
Improved impact properties	Improved surface appearance

Figure 7K

The ability to control mold temperature is often highly dependent on tool design. If possible, all surfaces of the tool should be maintained at a consistent temperature. This will ensure consistent properties throughout the part. Below are additional suggestions for setting mold temperature:

1. When possible, use a pyrometer to verify that the mold temperature is at the desired temperature and consistent throughout both halves of the tool.

2. Avoid a temperature differential between the halves of the tool greater than 75° F (24° C). This will reduce the risk of tool steel interference or binding from temperature-induced expansion in the mold.

3. A temperature differential between the mold halves may induce the part to warp toward the hotter side of the tool.

Injection Pressure

The actual required injection pressure will depend on many variables, such as melt and mold temperatures, part thick-ness, geometry, and flow length. Generally, low to medium pressures are desirable to maintain material properties, appearance criteria, and cycle time.

Holding Pressure and Pack Pressure

Holding pressure is the pressure transferred through the melt into the part cavity following the filling of the mold. The change from injection pressure to holding pressure is commonly called the transfer point. Holding pressures are normally 1/2 to 2/3 of the maximum injection pressure and should take effect after the cavity is filled.

Holding pressure should be maintained until the gate freezes off. Applying pressure beyond this point will not affect the part. To estimate when the gate is freezing off, adjust the process to mold a full consistent shot. Begin by molding without a hold time and incrementally with each successive shot increase hold time by one second. Weigh each part and plot the weight on a curve of part weight vs. hold time. When the curve begins to level off, showing a consistent part weight, the gate freeze off time can be noted. The first point at which the curve becomes horizontal is the freeze off temperature.

General Holding Pressure Guidelines

Increasing	Decreasing
Reduces shrinkage	Reduces part sticking
Reduces sink marks	Increases shrinkage
Reduces warpage	Reduces sprue sticking
Potential to flash parting line	Induces surface gloss
Reduces gloss on grained parts	Possibly reduces part strength

Figure 7L

Back Pressure

Back pressure on a screw results when its backward movement (screw recovery) is restricted. Back pressure is always recommended to ensure a consistent shot size and homogeneous melt. Higher pressures may be required for more intensive mixing but may induce higher melt temperature, glass breakage (reinforced materials), and increased cycle time due to hotter melt temperatures and slower screw recovery. Typical back pressure settings for Ultramid products are 25 to 100 psi. Increasing back pressure as a substitute for a proper heat profile or an inadequate screw design is not recommended.

Injection Speed

Generally speaking, reinforced Ultramid resins should be injected with high velocity rates because of the inherent crystalline nature. A greater range of injection speeds can be utilized with reinforced Ultramid nylon grades.

Fast injection provides for longer flow, improved pack condition, and better surface aesthetics when molding reinforced grades. Filling the cavity at a faster rate will allow the material to crystallize at a uniform rate in the tool which tends to result in lower molded-in stress.

Slower fill speeds may be required when filling through gate designs where jetting or gate blush is occurring. Slow fill is also used in tools with poor venting to eliminate part burning and in parts with thick cross sections to reduce sink marks and voids.

Programmed or profiled injection has been proven successful when molding parts with non-uniform wall stock to reduce voids and gas entrapment burning. It has also proved to be an advantage when molding through subgates and pinpoint gates.

Slow injection speeds at the start of injection can be used to eliminate/reduce gate blush, jetting and burning of the material.

Cushion

The use of a cushion of material at the end of the screw stroke is highly recommended when molding Ultramid products. Typically, a small cushion is acceptable for any Ultramid resin to promote shot to shot consistency. A cushion of 0.100"–0.25" (2.5mm–6.35mm) is recommended. The inability to hold a consistent cushion is usually indicative of a worn non-return valve. On injection molding machines with process controllers, the cushion can often be maintained automatically since the process controller will monitor screw location and can compensate for any deviation in cushion.

Maintaining a cushion as suggested will assist in the processing of the material in the following ways:

1. Helps to maintain consistent physical properties throughout the molded part.

2. Aids in ensuring dimensional reproducibility, weld line integrity and control of sink marks.

3. Assists maintaining a consistent surface quality.

Screw Rotation

Screw recovery speeds that will permit screw rotation during 75–90% of the cooling time are recommended. This will prevent excessive melt temperature increases and maintain a homogeneous melt temperature.

Screw Decompression

Decompression or "suck back" is the intentional pulling back of the screw and polymer from the nozzle area to prevent drool. It is usually accomplished by time or screw position settings. The result is introducing air to the molten plastic, which cools and may oxidize the plastic. Therefore, it is recommended to minimize the use of decompression.

The nylon reverse taper machine nozzle has been used successfully to minimize drool and stringing. Please refer to Chapter 4 of this guide for more information on nozzle tips.

Purging

The barrel should be purged if the process will be shut down or idled for any length of time. For short interruptions, one need only purge several shots, but longer shut downs require a complete purging or emptying of the barrel. It is always a good idea to purge the first several shots at start up to reduce the chance of contamination from previous processing.

Regrind

Ultramid nylon, like many thermoplastic materials, may be used with its own regrind. Typical levels of regrind range from 25–30% if the initial molding did not cause thermal degradation or severe glass breakage (in filled materials) to the Ultramid product.

Below are several suggestions when using regrind:

1. Mix regrind back into the virgin material type from which it was originally molded.

2. It is important that the material to be reground be free from oil, grease or dirt, and show no signs of degradation.

3. Regrind that contains excessive quantities of fines or dust-like particles may result in molding problems such as burning or splay. Try to select grinder screens that will minimize fines.

4. It is advantageous to try to make the particle size of the regrind as close to the initial pellet size as possible. This will allow for ease of blending and for consistent drying of the material.

5. Prior to processing the regrind, ensure that the moisture level is similar to that of the virgin material. This will help in processing.

6. If the regrind material does become wet it is usually better to dry the material off line in a separate dryer.

7. Some color shift may occur when using regrind. It is important to dry the regrind at 180° F (82° C) maximum to reduce the amount of color shift.

8. To maintain a consistent regrind blend, it is recommended to use your regrind back into the process as it is generated.

9. If regrind is to be stored for future use, store it in a container with a moisture barrier.

Pre-Colored Ultramid Nylon

Ultramid nylon resins can be supplied in numerous custom and standard compounded colors. Ultramid resins can also be easily colored by dry-blending natural resin with commercially available color concentrates. Please call BASF for a list of commercial sources.

Processing Cadmium-Free vs. Cadmium Colors

BASF has established itself as a leader in "cad-free" color technology for nylon 6. At present, cad-free pigments, especially bright reds, bright oranges, and bright yellows do not have as high a melt stability as cadmium pigments. For this reason, care should be taken when molding cad-free color Ultramid resins, especially when melt temperatures of greater than 570° F (299° C) are required. We recommend the following for good color repeatability in cad-free colors:

1. Do not dry the bright cad-free colors above 180° F (82° C) and do not dry for an extended period of time (over 10 hours). If needs require longer drying, reduce dryer set-point to 130° F (55° C) after 8 hours.

2. Minimize residence time in the barrel. A shot size of 50% to 60% of barrel capacity is ideal. If the shot size is less than 50% of barrel capacity, profile barrel temperatures with lower settings in the middle and rear zones.

3. Establish the molding process with the lowest melt temperature that yields a good part.

4. Ensure that the check valve is holding a cushion. A free- flow check ring is recommended. Ball check valves tend to have dead spots where material may hang-up for several shots and lead to discolored streaks in molded parts.

5. Ensure that the clearance between the screw and barrel inner diameter is not excessive. Any leakage around the check valve will excessively shear heat the material.

6. Attempt to design-out or remove all instances where shear may be induced in the melt (i.e. thick to thin wall transitions and sharp corners).

7. Acceptable regrind levels may be lower than those typically used for non-cadmium-free pigment systems. This is due to the inherently lower thermal stability of cad-free pigments.

8. Adjust screw RPM so that screw rotation lasts for a minimum of 80% to 90% of the cooling time. This will reduce thermal history on the material.

9. Nozzle bore, sprue bushing bore, gate thickness, and diameter should all be as large as possible to minimize shear heating during injection.

10. Avoid the use of a reverse taper nozzle if possible. This will also reduce shear during injection.

Chapter 8

Ultramid Nylon Troubleshooting Guide for Injection Molding

Chapter 8 : Ultramid Nylon Troubleshooting Guide for Injection Molding

Introduction

The purpose of the tables that constitute the major part of this chapter are to identify the broad categories of molded-part deficiencies and molding problems that can arise in injection molding. Under each category, possible causes are identified and remedies suggested. While the suggestions are specifically made for parts molded from Ultramid nylon resins, many of them have more general applicability.

Responsibility for the proper molding of a part lies with the molder, who applies knowledge and skill to achieve a satis-factory result from a complex interplay of factors including:

- Choice of Material
- Mold Design
- Melt Stability
- Moisture Sensitivity
- Material Stress Behavior
- Throughput Rate
- Machine Characteristics
- Aesthetics
- Dimensions

Ultramid nylon molding compounds are formulated to perform well in both the end-product and in the fabrication process. BASF provides a generous array of Ultramid engineering plastics from which to choose. Assistance with the selection of the most appropriate Ultramid nylon molding polymer for your needs is always available from BASF. You will find our address and phone number at the end of this guide.

With its resident expertise and comprehensive resources and equipment, the BASF customer support staff has built a strong base of technical experience and data on injection molding. We continue to perform studies on injection molding and investigations into the performance properties of our nylon resins. Our technical personnel are always ready to share their knowledge with you.

Calling for Technical Assistance

When it is necessary to call the technical service staff for help, please have the following information available:

1. Type of problem, i.e. molding defect, part failure.
2. Material type, color number, and lot number.
3. Material handling procedures, regrind percent used, drying times and temperatures.
4. Injection molding parameters. Actual melt and mold temperatures, actual fill time, injection pressures and times, back pressure , screw RPMs , cooling time, clamp tonnage, shot size versus barrel size.
5. Miscellaneous: Nominal wall thickness, gate size, number of cavities, balance of flow to each cavity, etc.

Brittleness

	Possible Cause	Suggested Remedy
1.	Melt temperature too low.	Increase melt temperature (weak weld lines).
2.	Material overheated, resulting in molecular breakdown.	a. Decrease melt temperature. b. Residence time in cylinder excessive– use smaller barrel. c. Decrease overall cycle. d. Reduce back pressure.
3.	Contamination by foreign material or excessive pigment usage.	a. Inspect resin for contamination (replace if contaminated). b. Purge injection cylinder thoroughly. c. Keep hopper covered. d. Review material handling procedures for regrind usage. e. Reduce filler or pigment loading.
4.	Excessive amounts of regrind.	a. Reduce regrind % mixed with virgin. b. Regrind level dependent upon application: general rule – 25–30%. c. Keep hopper covered. d. Review material handling procedures for regrind usage. e. Reduce filler or pigment loading.
5.	Injection rate too slow.	a. Increase injection or first stage pressure. b. Increase boost time.
6.	Improper gate location or size.	a. Relocate gate away from potential stress area. b. Increase gate size to obtain optimum filling.
7.	Moisture in material during processing.	a. Review material handling to eliminate moisture pick up. b. Dry material prior to molding. c. Utilize hopper dryers.
8.	Dry-as-molded properties.	Moisture condition parts to increase toughness.

Bubbles, Voids

Possible Cause	Suggested Remedy
1. Excessive internal shrinkage.	a. Increase packing pressure. b. Increase injection forward time. c. Increase gate thickness. d. Minimize, or core out, heavy sections in part design. e. Increase feed, ensure cushion. f. Replace check valve if cushion cannot be held.
2. Melt temperature too high.	a. Decrease melt temperature. b. Lower back pressure. c. Lower screw RPM.
3. Moisture in material.	a. Review material handling to eliminate moisture pick up. b. Dry material prior to molding. c. Utilize hopper dryers. d. Review percent of regrind.
4. Air entrapment.	a. Add mold venting. b. Relocate gate. c. Reduce clamp pressure to allow parting line vents to work.
5. Condensation on mold surface.	a. Wipe mold surface thoroughly with solvent. b. Increase mold temperature.

Burn Marks

Possible Cause	Suggested Remedy
1. Melt temperature too high.	a. Decrease melt temperature. b. Lower back pressure. c. Lower screw RPM.
2. Air entrapped in mold.	a. Vent cavity at final point of fill. b. Decrease first stage pressure or injection speed. c. Relocate gate. d. Clean vents and/or enlarge vents. e. Reduce clamp pressure to allow parting line vents to work.
3. Injection rate too fast.	a. Decrease injection rate. b. Decrease first stage pressure. c. Decrease boost time. d. Enlarge gates.
4. Moisture in material.	a. Review material handling to eliminate moisture pick up. b. Dry material prior to molding. c. Utilize hopper dryers.

Cracking, Crazing

Possible Cause	Suggested Remedy
1. Packing excessive material into the mold.	a. Decrease packing pressure. b. Decrease shot size. c. Increase transfer position (to lower injection peak pressure). d. Decrease injection time.
2. Non-uniform or too cold a mold temperature.	a. Increase mold temperature. b. Supply uniform cooling to the cavity.
3. Knockout system poorly designed.	Redesign knockout system for balanced ejection forces.
4. Inadequate draft angles or excessive undercuts.	Rework mold.

Dimensional Variations

Possible Cause	Suggested Remedy
1 Non-uniform feeding of material.	a. Adjust temperature profile for optimum feeding. b. Increase shot size to maintain uniform cushion. c. Replace check valve if cushion cannot be held.
2. Large variation in cylinder temperature due to inadequate or defective controllers.	Replace or calibrate controllers.
3. Unbalanced runner system, resulting in non-uniform cavity pressure.	a. Increase holding pressure to maximum. b. Increase injection rate. c. Balance/increase runner and gate sizes to provide uniform filling.
4. Insufficient packing of part.	Increase injection forward time and/or pressure to ensure gate freeze off.
5. Regrind not uniformly mixed with virgin.	a. Review regrind blending procedure. b. Decrease percentage of regrind.
6. Molding conditions varied from previous run.	a. Check molding records to ensure duplication of process conditions.
7. Part distortion upon ejection.	See Part Sticking in Mold, page 53.

Discoloration, Contamination

Possible Cause	Suggested Remedy
1. Material overheated in injection cylinder.	a. Decrease melt temperature. b. Decrease overall cycle. c. Residence time in cylinder excessive for shot size – use smaller barrel. d. Decrease nozzle temperature. e. Decrease screw RPM. f. Decrease back pressure. g. Check calibration of cylinder controllers. h. Check barrel and nozzle heater bands and thermocouples.
2. Burned material hanging up in cylinder, nozzle (black specks), or check ring.	a. Purge injection cylinder. b. Remove and clean nozzle. c. Remove and inspect non-return valve for wear. d. Inspect barrel for cracks or gouges. e. Decrease injection rate.
3. Material oxidized by drying at too high temperature.	Reduce drying temperature to 180°F (82°C).
4. Contamination by foreign material.	a. Keep hopper covered. b. Review material handling procedures for virgin and regrind. c. Purge injection cylinder.

Excessive Cycle Time

Possible Cause	Suggested Remedy
1. Poor mold cooling design.	a. Increase mold cooling in hot spots. b. Ensure fast turbulent flow of water through cooling channels.
2. Platen speeds excessively slow.	a. Adjust clamp speeds to safely open and close quickly. b. Low pressure close time excessive, adjust clamp positions and pressures to safely and efficiently open and close mold.
3. Melt temperature too high.	Decrease melt temperature.
4. Mold temperature too high.	Decrease mold temperature.
5. Screw recovery time excessive.	a. Check machine throat and hopper for blockage or bridging. b. Check for worn screw and barrel, especially in the feed zone.

Flashing

Possible Cause	Suggested Remedy
1. Material too hot.	a. Decrease melt temperature. b. Decrease mold temperature. c. Lengthen cycle time.
2. Injection pressure too high.	a. Decrease injection pressure. b. Decrease boost time. c. Decrease injection rate. d. Increase transfer position (to lower injection peak pressure).
3. Excessive packing of material in cavities.	Decrease packing pressure.
4. Projected area too large for available clamping force.	Use larger tonnage machine.
5. Mold clamping pressure not properly adjusted.	a. Increase clamping pressure. b. Check mold parting line for obstruction. c. Check platen parallelism.
6. Uneven or poor parting line and	a. Remove mold, carefully inspect and repair parting lines, cavities and cores which do not have positive shut off. b. Add support for mold cores and cavities.
7. Non-uniform cavity pressure due to unbalanced filling.	a. Balance/increase runner and gate sizes to obtain uniform filling. b. Properly balance cavity layout for maintaining uniform cavity pressure.

Flow Lines

Possible Cause	Suggested Remedy
1. Melt temperature too low.	Increase melt temperature.
2. Mold temperature too cold.	Increase mold temperature.
3. Gate size too restrictive, causing jetting.	a. Increase gate size. b. Decrease injection rate.
4. Material impinging on cavity wall or core.	a. Decrease injection rate. b. Relocate gate.
5. Non-uniform wall thickness.	Redesign part to obtain a more uniform wall thickness to provide for optimum filling.
6. Insufficient mold venting.	Improve mold venting.

Lamination

Possible Cause	Suggested Remedy
1. Melt temperature too low.	Increase melt temperature.
2. Mold temperature too cold.	Increase mold temperature.
3. Injection rate too low.	a. Increase first stage pressure. b. Increase boost time.
4. Holding pressure too low.	Increase packing pressure.
5. Gate size too small.	Increase gate size for improved filling.
6. Contamination.	See Discoloration, Contamination, page 50.

Nozzle Drooling

Possible Cause	Suggested Remedy
1. Nozzle temperature too hot.	a. Reduce nozzle temperature. b. Decrease melt temperature. c. Reduce back pressure. d. Increase screw decompression. e. Use enough screw RPM's to allow screw to recover using approximately 90% of the cooling time.
2. Wrong nozzle design.	Use reverse taper nozzle.
3. Moisture in material.	a. Review material handling to eliminate moisture pick up. b. Dry material prior to molding. c. Use hopper dryers.

Part Sticking in Mold

Possible Cause	Suggested Remedy
1. Overpacking material in mold.	a. Decrease first stage injection pressure. b. Decrease boost time. c. Decrease injection forward time. d. Decrease packing pressure. e. Increase injection transfer position (to lower injection peak pressure).
2. Improper finish on mold.	Draw polish mold to proper finish.
3. Insufficient draft on cavities and sprue.	Polish and provide maximum allowable draft.
4. Knockout system poorly designed.	a. Redesign knockout system for balanced ejection forces. b. Review operation of knockout system, plates not opening in proper sequence.
5. Core shifting and cavity misalignment.	a. Realign cavities and cores. b. Add interlocks to mold halves.
6. Undercuts in mold and possible surface imperfections.	a. Repair and polish. b. If undercut is intentional, redesign or reduce.
7. Non-uniform cavity pressure in multi-cavity mold.	Redesign runners and gates for balanced filling of cavities.
8. Molded parts too hot for ejection.	a. Increase cooling time. b. Decrease melt temperature. c. Decrease mold temperature.
9. Molded parts sticking to stationary half of mold.	a. Redesign sprue puller. b. Apply mold release. c. Increase nozzle temperature. d. If parts remain on wrong side of mold, undercut other side or try differential mold temperatures.

Short Shots

Possible Cause	Suggested Remedy
1. Melt temperature too low.	Increase melt temperature.
2. Mold temperature too cold.	Increase mold temperature.
3. Injection pressure too low.	a. Increase first stage pressure. b. Increase boost time. c. Increase injection speed.
4. Insufficient feed.	a. Increase shot size to maintain a constant cushion. b. Inspect non-return valve for wear.
5. Insufficient injection forward time.	a. Increase injection forward time. b. Increase injection rate.
6. Entrapped air causing resistance to fill.	a. Provide proper venting. b. Increase number and size of vents.
7. Restricted flow of material to cavity.	a. Increase gate size. b. Increase runner size. c. Use nozzle with larger orifice.
8. Unbalanced flow to cavity in multi-cavity mold.	a. Increase gate size. b. Redesign runner to provide balanced flow.

Splay (Silver Streaking)

Possible Cause	Suggested Remedy
1. Excess moisture in material during processing.	a. Review material handling to eliminate moisture pick up. b. Dry material prior to molding. c. Utilize hopper dryers.
2. Melt temperature too high	a. Decrease barrel temperature. b. Decrease nozzle temperature.
3. Excessive shear heat from injection.	a. Decrease injection rate. b. Reduce screw RPM. c. Increase runner size and/or gates. d. Check for nozzle obstruction.
4. Air entrapment.	a. Reduce screw decompression. b. Improve mold venting.
5. Condensation and/or excessive lubricant on mold surface.	a. Increase mold temperature. b. Clean mold surface with solvent. c. Use mold release sparingly.
6. Moisture condensing in feed section of barrel.	a. Decrease throat cooling. b. Increase rear zone temperature setting.

Sprue Sticking

Possible Cause	Suggested Remedy
1. Improper fit between nozzle and sprue bushing.	Nozzle orifice should be smaller than sprue bushing orifice.
2. Insufficient taper on sprue bushing.	Increase taper.
3. Rough surface of sprue bushing.	Eliminate imperfections and draw polish surface.
4. Sprue puller design inadequate.	a. Redesign sprue puller and increase undercut. b. Increase sprue diameter if too small for strength. c. Decrease sprue diameter if too large for cooling.
5. Overpacking material in sprue.	a. Decrease packing pressure. b. Decrease injection forward time. c. Use machine sprue break.
6. Nozzle temperature too low to provide clean break.	a. Increase nozzle temperature. b. Use reverse taper nozzle.

Surface Imperfections (Glass On Surface, Mineral Bloom)

Possible Cause	Suggested Remedy
1. Melt temperature too low.	Increase melt temperature.
2. Mold temperature too cold.	Increase mold temperature.
3. Insufficient packing pressure.	Increase packing pressure.
4. Injection rate too slow.	a. Increase first stage pressure. b. Increase boost time. c. Increase injection speed.
5. Insufficient material in mold.	a. Increase shot size and maintain constant cushion. b. Inspect non-return valve for wear. c. Decrease injection transfer position (thereby increasing the peak pressure).
6. Water on mold surface.	a. Increase mold temperature. b. Repair any water leaks.
7. Excessive lubricant on mold surface.	a. Clean mold surface with solvent. b. Use mold release sparingly.
8. Moisture in material.	a. Review material handling to eliminate moisture pick up. b. Dry material prior to molding. c. Use hopper dryers.
9. Insufficient venting.	Provide adequate vents.

Warpage

Possible Cause	Suggested Remedy
1. Molded part ejected too hot.	a. Decrease melt temperature. b. Decrease mold temperature. c. Increase cooling time. d. Cool part in warm water after ejection. e. Utilize shrink fixture.
2. Differential shrinkage due to non-uniform filling.	a. Increase injection rate. b. Increase packing pressure. c. Balance gates and runners. d. Increase/decrease injection time. e. Increase runner and gate size.
3. Differential shrinkage due to non-uniform wall thickness.	a. Provide increased cooling to thicker sections. b. Increase cooling time. c. Operate mold halves at different temperatures. d. Redesign part with uniform wall sections.
4. Knockout system poorly designed.	Redesign knockout system for balanced ejection forces.
5. Melt temperature too low.	Increase melt temperature to pack out part better.
6. Glass fiber orientation.	Relocate gate.

Weld Lines (Knit Lines)

Possible Cause	Suggested Remedy
1. Melt temperature too low.	Increase melt temperature.
2. Mold temperature too cold.	Increase mold temperature.
3. Insufficient pressure at the weld.	a. Increase first stage injection pressure. b. Increase boost time. c. Increase packing pressure. d. Increase pack time. e. Increase injection speed.
4. Entrapped air unable to escape from mold fast enough.	a. Increase or provide adequate vents at the weld area. b. Decrease injection rate.
5. Excessive lubricant on mold surface plugging vents.	a. Clean mold surface with solvent. b. Use mold release sparingly.
6. Distance from gate to weld line too far.	a. Relocate or use multiple balanced gates. b. Cut overflow tab in mold to improve weld line strength.
7. Injection rate too slow.	a. Increase injection speed. b. Increase first stage injection pressure. c. Increase boost time.

Processing Quality Checklist

↑ Increase

↓ Decrease

	RAPID RESPONSE							SLOWER RESPONSE				
	Back Pressure	Boost Pressure	Injection Rate	Mold Open Speed	Packing Pressure	Screw Decompression	Screw Forward Time	Cycle Time	Melt Temperature	Mold Temperature	Nozzle Temperature	Possible Causes and/or Solutions
Bubbles/Voids		↑			↑	↓	↑		↓	↑		Improve venting, Inc. gate size, Min. thick sections
Burn Marks		↓	↓				↓		↓			Improve venting, Relocate gate
Discoloration	↓		↓					↓	↓		↓	Purge barrel/Clean screw/Barrel/Nozzle
Distortion Upon Ejection				↓	↓							Check mold surface for smooth release
Erratic Screw Retraction	↑								↑			Check for screw wear
Flash	↓	↓	↓		↓				↓	↓		Mold needs adjustment/Clamp tonnage too low
Flow Lines			↓						↑	↑		Increase gate size, Check venting
KO Pin Penetration				↓	↓			↑				Poor mold cooling
Lamination		↑	↑		↑				↑	↑		Contaminated material, Increase gate size
Nozzle Drool	↓					↑			↓		↓	Use reverse taper nozzle
Part Sticking in Mold		↓	↓		↓		↓	↑	↓	↓	↑	Check for damaged mold surfaces
Poor Weld Lines		↑	↕		↑				↑	↑		Improve venting, Relocate gate, Clean vents
Short Shots		↑	↑		↑		↑		↑	↑		Increase gate size, Increase shot size
Shot to Shot Variation			↑		↑		↑					Non-return valve leakage
Sink Marks		↑			↑		↑		↕			Increase gate size
Splay Marks			↓		↑	↓			↓	↑	↓	Wet material
Sprue Sticking					↓		↓				↑	Damaged sprue bushing, Increase taper
Surface Blemish		↑	↑		↑				↑	↑		Wet material
Unmelted Pellets	↑							↑	↑			Check heater bands
Warpage			↕		↑		↓	↑	↕	↓		Check cooling line location
White Spots	↓	↑	↓			↓				↑		Wet material

www.plasticsportal.com/usa

BASF
The Chemical Company