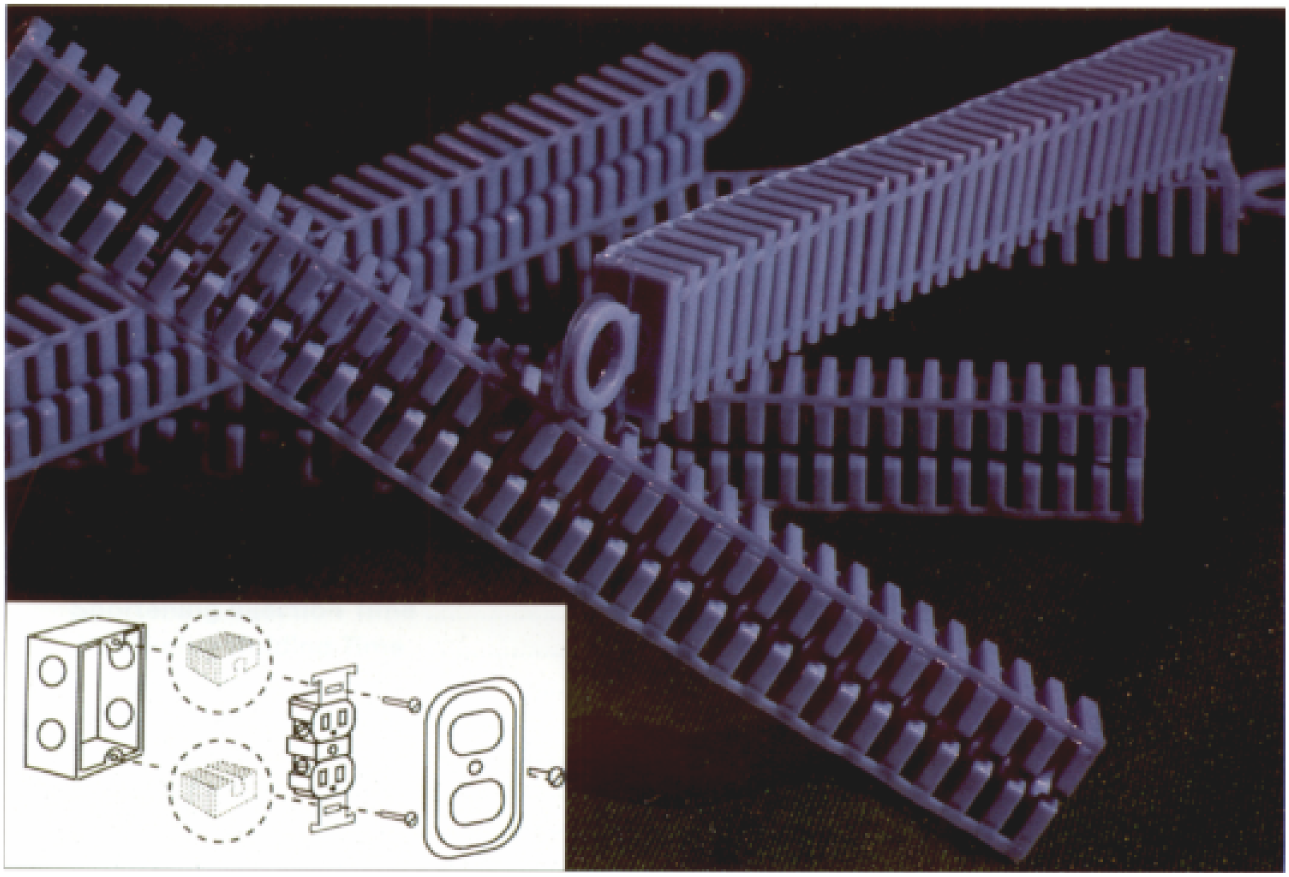
Injection Molding Guide

Hytrel®

thermoplastic polyester elastomer



DuPont Hytrel® thermoplastic polyester elastomer was specified for these spacers for loosely fitting, recessed electrical receptacles because of its flexibility, toughness, resilience, and creep resistance.

Start
with
DuPont

Identity and Trademark Standards

Guidelines for Customer Use—Joint ventures and authorized resellers

Only joint ventures and resellers who have signed special agreements with DuPont to resell DuPont products in their original form and/or packaging are authorized to use the Oval trademark, subject to the approval of an External Affairs representative.

Guidelines for Customer Use—All other customers

All other customer usage is limited to a product signature arrangement, using Times Roman typography, that allows mention of DuPont products that serve as ingredients in the customer's products. In this signature, the phrase, "Only by DuPont" follows the product name.

Hytrel® only by DuPont   or   Hytrel®
Only by DuPont

A registration notice ® or an asterisk referencing the registration is required. In text, "Only by DuPont" may follow the product name on the same line, separated by two letter-spaces (see above example). When a DuPont product name is used in text, a ® or a reference by use of an asterisk must follow the product name. For example, "This device is made of quality DuPont Hytrel® thermoplastic polyester elastomer for durability and corrosion resistance."

*Hytrel® is a DuPont registered trademark for its brand of thermoplastic polyester elastomer.

Rev. August. 1995

DUPONT®

Hytrel®

thermoplastic polyester elastomer

Table of Contents

Hytrel®
Injection Molding Guide

General Information

Hytrel® is the DuPont registered trademark for its family of thermoplastic polyester elastomers. Hytrel® offers a unique combination of mechanical, physical and chemical properties that qualifies it for demanding applications. The various grades of Hytrel® exhibit a wide range of properties and an easy processability.

This report provides detailed guidelines for injection molding of Hytrel®. It reviews the type of equipment as well as the processing conditions necessary to achieve high quality parts and high productivity.

Product Description

Hytrel® combines many of the most desirable characteristics of high-performance elastomers and flexible plastics. It features: exceptional toughness and resilience; high resistance to creep, impact and flex fatigue; flexibility at low temperatures; and good retention of properties at elevated temperatures. In addition, it resists deterioration from many industrial chemicals, oils and solvents.

Hytrel® is a block copolymer, consisting of a hard (crystalline) segment of polybutylene terephthalate and a soft (amorphous) segment based on long-chain glycols. Properties are determined by the ratio of hard to soft segments and by the make-up of the segments. Most grades of Hytrel® do not contain or require additives to enhance their properties, except for specific applications.

The grades of Hytrel® are grouped into four categories, by performance:

1. ***General purpose grades*** which exhibit versatile processing characteristics, are lowest in cost and are suitable for many applications;
2. ***High-performance grades*** generally provide an extra margin of mechanical properties for the more demanding applications;
3. ***Specialty grades*** provide special properties or processing characteristics for particular applications;
4. ***Concentrates*** contain relatively high concentrations of specific property-enhancing additives for blending with other grades of Hytrel®.

Hytrel® is supplied as cylindrical to oval-shaped pellets (approximately 3.2 mm [0.125 in] in diameter by 3.2 mm [0.125 in] long). They are packaged in 25 kg (55 lb) multi-wall paper bags with a moisture barrier inner wall. Palletized units contain 40 bags, or 1000 kg net weight, wrapped in polyolefin film on disposable wooden pallets.

Most grades of Hytrel® are also available in 750 kg (1,650 lb) bulk boxes with a moisture resistant liner.

Property data sheets on currently available grades can be obtained through your local sales office listed at the end of this bulletin or through your DuPont sales representative or are also available through the World Wide Web, www.dupont.com/enggpolymers/americas. A summary of the molding grades is given in **Table 1**.

Table 1
Molding Grades of Hytrel® Engineering Thermoplastic Elastomer

Grade of Hytrel®	Nominal Hardness,[1] Durometer D	Flexural Modulus,[2] MPa (psi)	Description
General Purpose			
G3548W	35	32.4 (4,700)	Best balance of cost and performance
G4074	40	65.5 (9,500)	
G4078W	40	65.5 (9,500)	
G4774, G4778	47	117 (17,000)	
G5544	55	193 (28,000)	
High-Performance			
4056	40	62 (9,000)	Provide an extra measure of strength and service to meet the needs of the most demanding applications in a wide range of hardnesses
4069	40	55 (8,000)	
4556	45	94 (14,000)	
5526	55	207 (30,000)	
5556	55	207 (30,000)	
6356	63	330 (48,000)	
7246	72	570 (83,000)	
8238	82	1210 (175,000)	
Specialty			
3078	30	28 (4,000)	Very soft, flexible grade
5555HS	55	207 (30,000)	Heat-stabilized grade of 55D Hytrel®
HTR6108	60	193 (28,000)	Low permeability grade
HTR8068	46	174 (25,200)	Flame retardant grade

[1] ASTM D2240 (ISO 868)

[2] ASTM D790 (ISO 178), at room temperature

Note: All of the values reported in this table are based on ASTM methods. ISO methods may produce different test results due to differences in test specimen dimensions and/or test procedures.

Table 2 shows several attributes of the product range that should be considered in injection molding. Certain grades depending on typical composition may, however, not fall exactly into these generalizations.

Table 2
Characteristics of Hytrel®

	Soft Grades 30–47 Durometer D	Hard Grades 55–82 Durometer D
Crystallinity	–	+
Shrinkage	–	•
Thermal stability	+	++
Wide processing window	+	++
Melt temperature	•	+
Mold temperature	–	•
Cycle time	•	short

– low, • medium, + high

Melt Properties

Hytrel® engineering thermoplastic elastomer has good flow characteristics. The melt viscosity and, hence, the melt flow varies depending on the composition of the resin. The melt viscosities of various grades of Hytrel® versus temperature are shown in **Figures 1–3**.

Figure 1. Apparent Melt Viscosity versus Temperature—General Purpose Hytrel® Grades at Shear Rate of 1000 s^{-1}

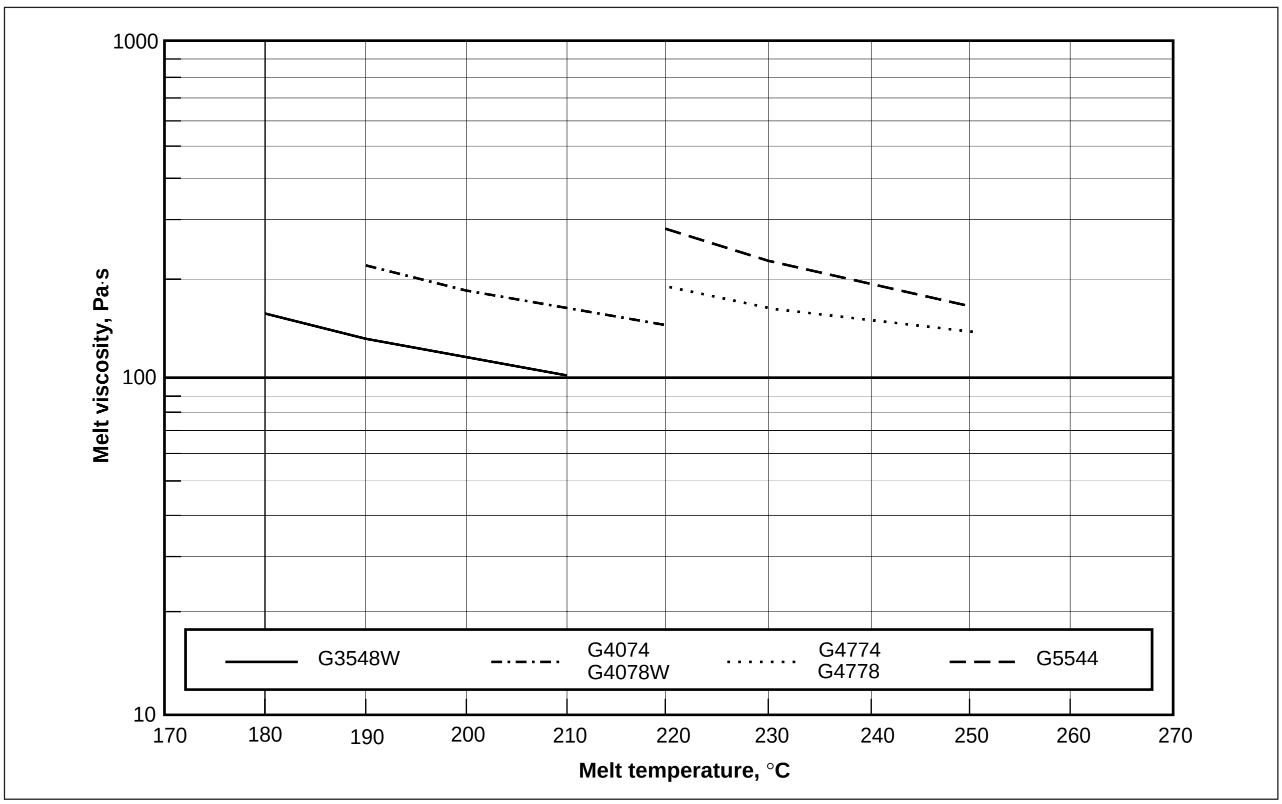


Figure 2. Apparent Melt Viscosity versus Temperature—High-Performance Hytrel® Grades at Shear Rate of 1000 s^{-1}

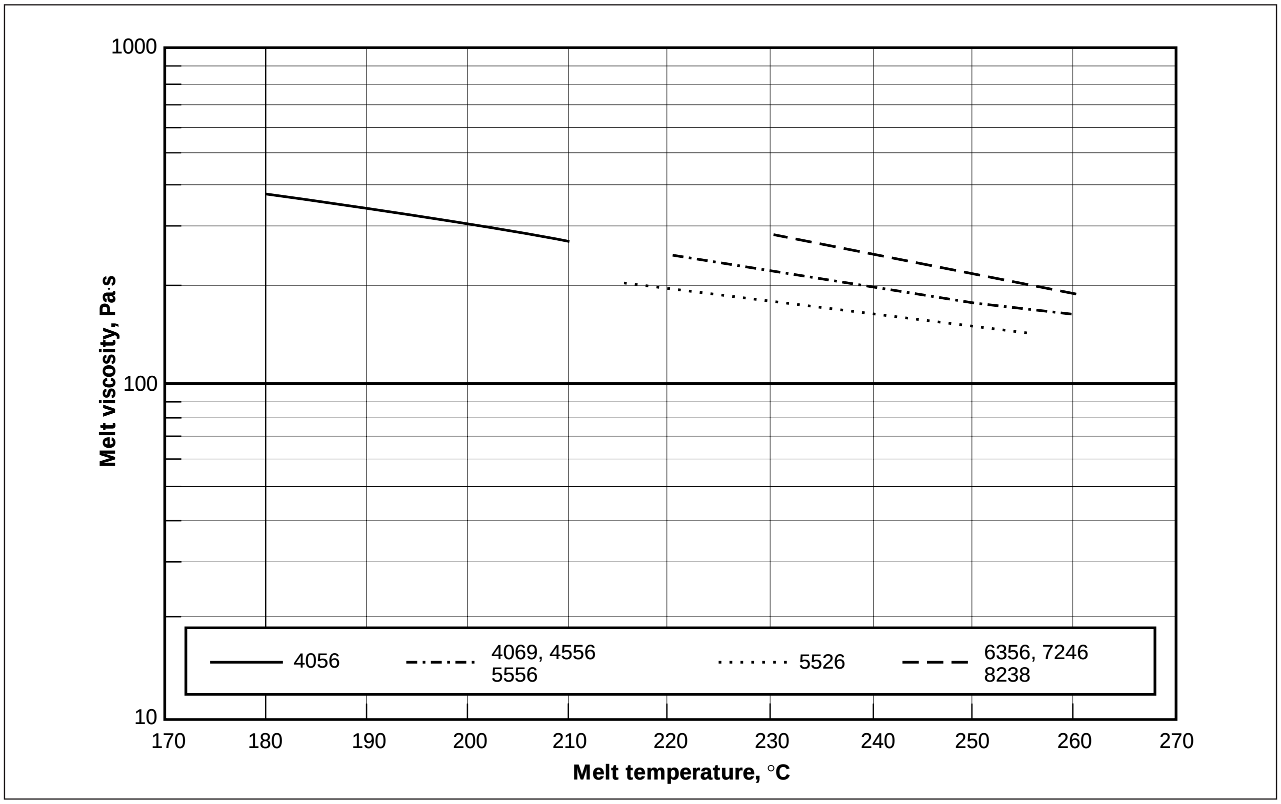
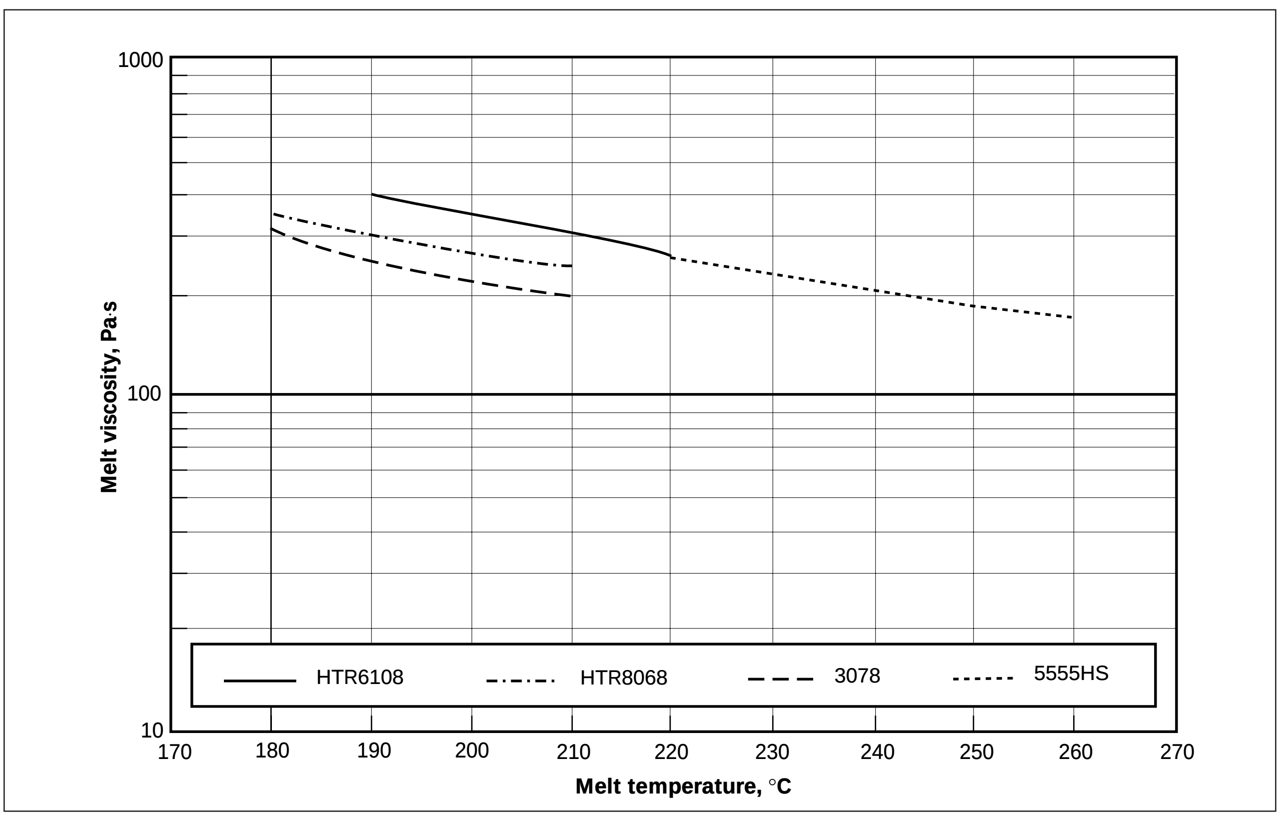


Figure 3. Apparent Melt Viscosity versus Temperature—Specialty Hytrel® Grades at Shear Rate of 1000 s^{-1}



Thermal Properties

When handled properly, Hytrel® has outstanding thermal stability. In the melt under normal operating conditions, the evolution of gaseous by-products is minimal for most grades. This thermal stability combined with a chemically pure polymer with no plasticizers minimizes problems such as change of viscosity with hold-up time in the injection unit or formation of black specks. An exception to this is during the molding of flame retardant grades or use of concentrates containing additives, where processing outside the optimum recommended conditions can more readily accelerate thermal decomposition.

Melt temperatures and some thermal properties of the various grades are shown in **Table 3**. Melt temperatures are a function of the ratio of hard to soft segments. Generally, the greater the hard segment content (and thus, hardness), the higher the melting temperature.

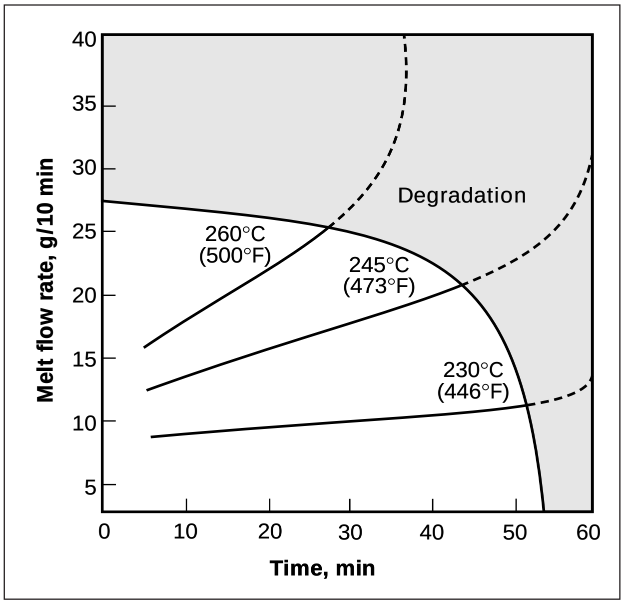
Table 3
Thermal Properties of Hytrel®

	T_m		T_m (complete)		T_c		H_f	T_g	
Grade	**°C**	**(°F)**	**°C**	**(°F)**	**°C**	**(°F)**	**J/g**	**°C**	**(°F)**
General Purpose									
G3548W	156	(312)	180	(356)	107	(225)	8	–40	(–40)
G4074	170	(338)	190	(374)	120	(248)	17	–37	(–35)
G4078W	170	(338)	190	(374)	120	(248)	17	–37	(–35)
G4774, G4778	208	(406)	225	(437)	170	(338)	27	–45	(–49)
G5544	215	(419)	230	(446)	173	(343)	33	–35	(–31)
High-Performance									
4056	150	(302)	170	(338)	70	(158)	12	–32	(–26)
4069	193	(379)	210	(410)	112	(234)	14	–50	(–58)
4556	193	(379)	220	(428)	115	(239)	24	–45	(–49)
5526	203	(397)	220	(428)	147	(297)	26	–18	(0)
5556	203	(397)	220	(428)	145	(293)	26	–20	(–4)
6356	211	(412)	230	(446)	155	(311)	31	0	(32)
7246	218	(424)	232	(450)	162	(324)	35	25	(77)
8238	223	(433)	235	(455)	170	(338)	37	40	(104)
Specialty									
3078	170	(338)	200	(392)	78	(172)	4	–60	(–76)
5555HS	203	(397)	216	(421)	166	(330)	26	–18	(0)
HTR6108	168	(334)	193	(379)	66	(151)	22	20	(68)
HTR8068	169	(336)	185	(365)	140	(284)	15	–	–

T_m: Melting Temperature (Peak of DSC endotherm)
T_c: Crystallization Temperature (Peak of DSC exotherm)
T_m (complete): Extrapolated End Point of Melting Curve, DSC endotherm
H_f: Heat of Fusion
T_g: Glass Transition Temperature

The thermal stability of these polymers permits exposure at melt temperatures for prolonged periods with minimum degradation. **Figure 4** shows the melt flow rate for Hytrel® 7246 after exposure at various temperatures for periods up to one hour. The modest change in melt flow rate indicates a high thermal stability of the resin.

Figure 4. Thermal Stability at Processing Temperatures, Moisture Content <0.1%



Material Handling

Drying

Hytrel® thermoplastic polyester elastomer must be dried prior to processing. It is critical to ensure that the resin is dry during processing to make quality parts that would give good service performance.

Hytrel® is resistant to hydrolysis. It does not react with moisture in the air, but will absorb the moisture if left exposed. Equilibrium moisture levels as determined by ASTM D570 depend on grade (see **Table 4**).

At temperatures substantially above the melting point, excess moisture causes hydrolytic degradation of the polymer. Such degradation results in poor physical properties and brittleness. No visual defects may be apparent but poor in-service performance can occur, particularly at low temperatures.

Generally, no degradation of the polymer or imperfections in the molding or extrusion occur if the moisture content is less than 0.10%, the maximum moisture specification for all injection molding grades of Hytrel®. When dry polymer is subjected to 50% relative humidity, 0.10% moisture increase occurs in about 2 hr, whereas at 100% relative humidity, it occurs in less than 1 hr (see **Figure 5**). Therefore, pellets so exposed should be redried before use.

When drying Hytrel®, dehumidified air ovens are recommended. Effective drying with such ovens takes place in 2–3 hr at 100°C (210°F) or overnight at 70°C (160°F).* Drying ovens without dehumidifiers may be used but will require 4–6 hr or more, depending on the quantity being dried. Even then, these ovens may not be adequate during periods of high humidity. Prolonged drying (>12 hr) at 100°C (210°F) is not recommended.

* It is critical to ensure that the dehumidifying medium is dry prior to the drying of Hytrel®.

Table 4
Water Absorption at 23°C after 24-hr Immersion

Grade of Hytrel®	Equilibrium Moisture Level, % after 24 hr
General Purpose	
G3548W	5.0
G4074	2.1
G4078W	3.0
G4774, G4778	2.5
G5544	1.5
High-Performance	
4056	0.6
4069	0.7
4556	0.6
5526	0.5
5556	0.5
6356	0.3
7246	0.3
8238	0.3
Specialty	
3078	3.0
5555HS	0.7
HTR6108	0.2
HTR8068	1.9

Figure 5. Moisture Absorption at Ambient Temperature

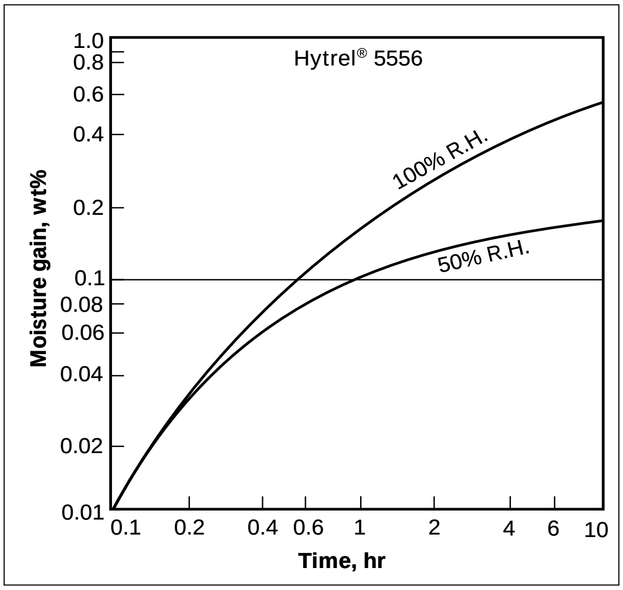
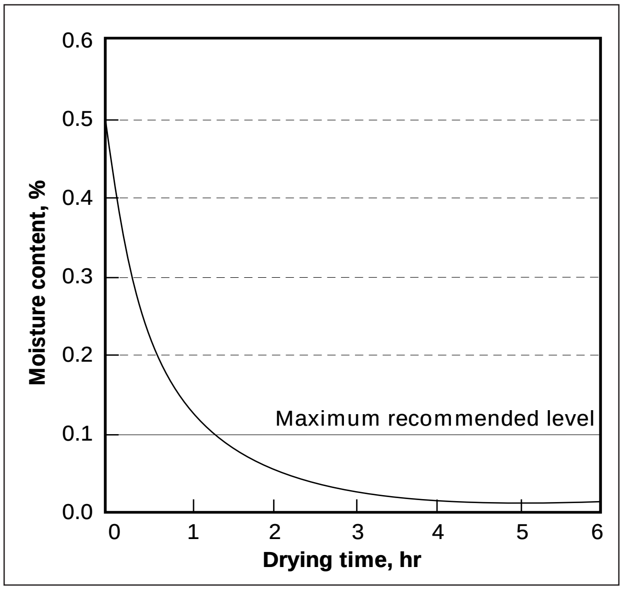


Table 5 and **Figure 6** show recommended drying conditions for Hytrel®.

Figure 6. Drying Guidelines with Dehumidified Hopper Dryer at 110°C. Dew Point –30°C



The air flow rate is very important. For each pound per hr of resin dried, 0.8 to 1 cubic ft per min (CFM) of air is required. Depending on dryer design, lower air rates will significantly reduce the resin temperature in the hopper. The dew point of air entering the hopper must be –18°C (0°F) or lower throughout the drying cycle in order to adequately dry Hytrel® resins.

Table 5
Drying Conditions for Hytrel®

	Drying Temperature	Drying Time
Dehumidified hopper	100°C (210°F)	2–3 hr
Dehumidified hopper	70°C (160°F)	overnight
Air circulating oven	100°C (210°F)	4–6 hr (in dry weather)

In an air circulating oven it may not be possible to achieve the recommended moisture level during high humidity conditions, thus they are not recommended.

The upper limits of these suggested drying times are particularly appropriate for the harder grades which give up absorbed moisture less readily.

Purging

Low or high density polyethylene resins can be used for purging Hytrel® thermoplastic polyester elastomer. Since some degree of degradation does take place with time, it is recommended to purge the cylinder when the machine is shut down. The venting of gases which may be generated at high temperatures or long residence times should be considered. To prevent cross-contamination proper cleanup and purging is always recommended before and after molding.

Regrind

Regrind can be used to a level of 25% without a significant drop in properties. However, the quality of regrind is essential to retain mechanical properties. The following points should be carefully considered:

- Keep the thermal history of regrind as short as possible to maintain the high quality of the polymer.
- Use grinders with properly adjusted, sharp knives shaped for polyethylene cutting to produce clean regrind with a minimum amount of fines.
- Regrind should be about the same size as the virgin granules.
- Excessive amounts of fines should be removed.
- Degraded or contaminated regrind must be discarded.
- To prevent any contamination, the grinder should be thoroughly cleaned prior to grinding operation.
- All regrind needs to be dried before molding.

Caution! All data presented in this section applies to the regrinding of unpainted parts. Since paint systems vary from part to part, regrind levels for a specific primed and/or painted part application must be examined on an individual basis.

Unpainted Hytrel® resins display excellent melt stability and property retention after repeated regrind. The effects of up to 4 passes (100% regrind) through an injection molding machine on tear resistance, tensile strength and break elongation for Hytrel® 5556 is given in **Figures 7** to **9**.

A key factor in use of regrind is resin drying. Reground polymer, as with virgin material, must be dried to <0.10% water to insure adequate property retention.

Figure 7. Effect of Regrind on Tear Resistance

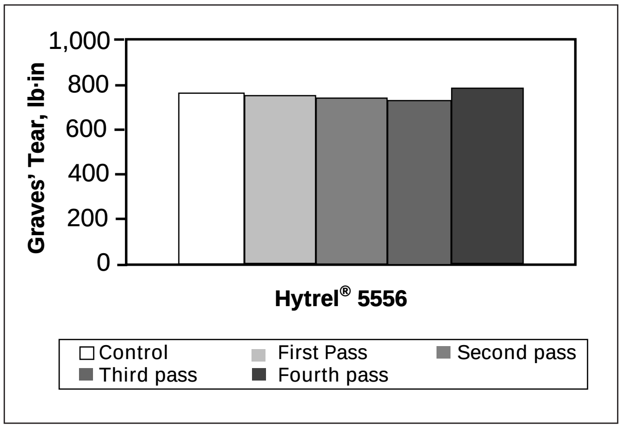


Figure 8. Effect of Regrind on Tensile Strength at Break

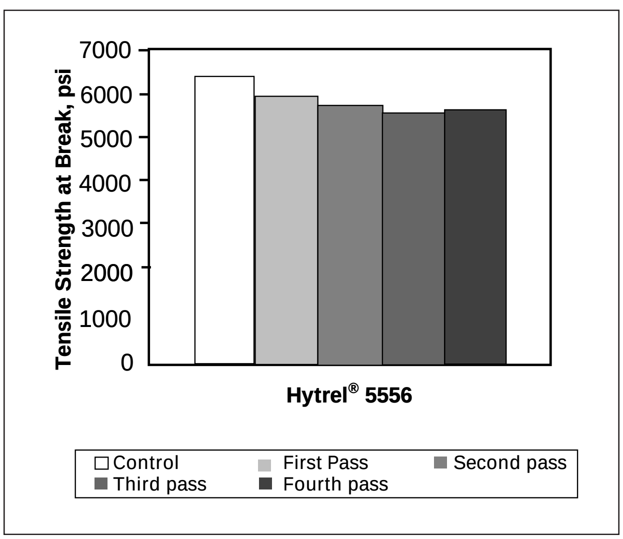
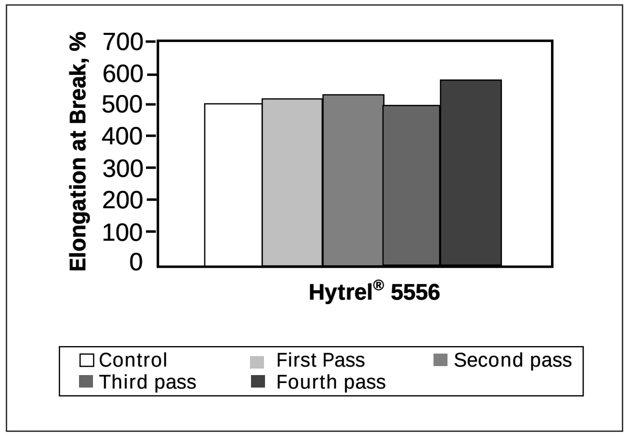


Figure 9. Effect of Regrind on Break Elongation



Melt index inversely correlates with product molecular weight. High melt index numbers represent significant molecular weight degradation and corresponding product property loss. Refer to bulletin "Hytrel® Rheology and Handling."

Adding Concentrates and Pigments

Concentrates

DuPont offers the following concentrates for use with Hytrel®. These concentrates are:

Hytrel® 10 MS – Hydrolytic stabilizer

Hytrel® 20 UV – Ultraviolet light stabilizer

Hytrel® 30 HS – Heat stabilizer

Hytrel® 40 CB – Carbon black concentrate

Hytrel® 51 FR – Flame retardant concentrate suitable for softer grades of Hytrel (hardness <55D)

Hytrel® 52 FR – Flame retardant concentrate suitable for stiffer grades of Hytrel (hardness ≥55D)

In all cases, the concentrates are designed to be tumble blended with any grade of Hytrel® before molding to enhance specific properties. Further information and recommended let-down ratios for blending of the concentrates is given in the relevant individual product data sheets, and Hytrel® Design Guide, Module V.

These Hytrel® concentrates have some hazards or off-gases associated with the specific additives they contain in addition to the Hytrel®. Therefore, these products require some special precautions for handling and processing.

Please refer to the Design Guide, Module V, the appropriate Material Safety Data Sheets, and bulletin, “Handling and Processing Precautions for Hytrel®.”

Pigments

Hytrel® grades for molding are available in a natural color. However, for many applications, it may be desirable to add pigments, or color concentrates, at the molding machine. The most convenient way to do this is by means of a color masterbatch, in granule form. Ideally, these masterbatches should be based on a low melting grade of Hytrel®.

The masterbatches can be cube blended at the molding machine, allowing for proper blending time and predrying. The compatibility and dispersion of the carrier resin used in masterbatch is very critical.

- **Masterbatches based on PE or PVC carriers are not recommended.** (Specific color matching and testing requirements should be directed to the masterbatch suppliers).

General Precautions Relating to Compounding

Materials with a pH of less than 7, such as acidic clays with pH 4.5 to 5.5, promote decomposition of Hytrel®. Compounding of Hytrel® with acidic pigments, lubricants, or additives should therefore be avoided.

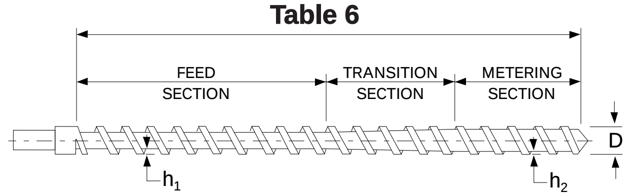
Molding Equipment

Hytrel® can be molded on standard injection molding machines. Even when the standard grades of Hytrel® degrade, no corrosive products are formed and equipment does not need to be specially corrosion resistant.

Screw Design

General purpose screws with a gradual transition zone are recommended. To avoid excessive shear of the polymer or bridging of the elastomeric pellets, screw compression ratio should not exceed 3:1 to 3.5:1 and the metering zone should be relatively deep, from 2.5 to 3.0 mm (0.100 to 0.120 in) for a 60 mm (2.36 in) screw. For a more uniform polymer melt and mixing, screw L/D (length to diameter) should be at least 20:1, and the compression ratio at least 2.5:1. Suggested screw designs are summarized in **Table 6**.

Table 6



Gradual Transition Screw

Screw Diameter (D), mm (in)	Feed Zone Depth (h_1), mm (in)	Metering Zone Depth (h_2), mm (in)
38 (1.5)	6.35 (0.250)	2.03 (0.080)
51 (2.0)	8.13 (0.320)	2.54 (0.100)
63 (2.5)	9.65 (0.380)	2.79 (0.120)
89 (3.5)	10.16 (0.400)	3.17 (0.125)

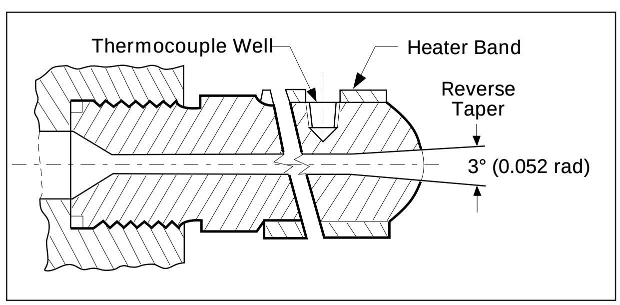
For injection molding machines, the following general configurations are suggested:

	Upper Range	Lower Range	Recommended
Feed Section	60%	33-1/3%	50%
Transition Section	33-1/3%	20%	25%
Metering Section	33-1/3%	20%	25%

Nozzle Design

Nozzles with reverse taper as shown in **Figure 10** are recommended for processing Hytrel®. Shut off nozzles are not required because Hytrel® does not drool at normal operating temperatures. Because the polymer melt is generally more viscous than other semicrystalline thermoplastics, nozzle diameter (and runner system) should be dimensioned somewhat larger.

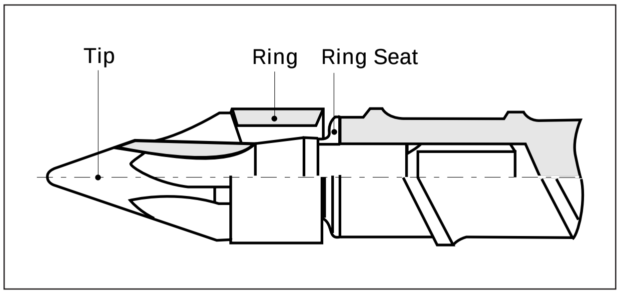
Figure 10. Reverse Taper Nozzle



Non-return valves ensure consistency of shot weight and cavity pressure from shot to shot.

Figure 11 shows the recommended non-return valve design.

Figure 11. Non-Return Valve



Clamping Force

The force required to hold a mold closed against injection pressure depends on:

- Injection pressure to fill the mold and pack-out parts
- Injection speed required for filling the part
- The dimensions of parts and runner at the mold parting line
- Mold details—laterial slide actions, core pull requirements, etc.
- Precision and tolerance requirements.

Most well-built molds can be adequately clamped if the machine has 48–69 MPa (3.5-5.0 tons/in^2) based on the projected area, since injection pressures are seldom over 100 MPa (14 kpsi) and fill rates are moderate. Higher available clamping pressure gives more freedom in choosing suitable molding parameters.

Molding Conditions

Melt Temperature

The melt temperature should be taken directly from the molten polymer (using a needle pyrometer) and should be checked periodically during production. Typical melt temperatures for various grades of Hytrel® are given in **Table 7**. Recommended cylinder temperatures are included.

Because Hytrel® has a good thermal stability, melt temperature can be increased up to 20°C in standard grades (see **Figure 13**) to improve the filling of thin parts. When higher than recommended melt temperatures are used, the cylinder temperature profile should be adapted (see following paragraph).

Cylinder Temperature Profile

To minimize sticking of pellets on the screw and when higher than recommended melt temperatures are used, a rising cylinder temperature profile (lower rear temperature) is normally preferred. Occasionally, a decreasing cylinder temperature profile can be used to reduce screw torque or to improve melt homogeneity.

As a general guideline, the graph in **Figure 12** can be used to define the optimum cylinder temperature profile, as based on hold up time (HUT) and percent of stroke used. Screw design, however, should also be taken into consideration.

Nozzle Temperature

The nozzle temperature should be adjusted to prevent freeze-off or drool. For optimum performance it should be controlled independently at a point near the orifice (see **Figure 10**). To prevent drooling in certain cases, the use of pressure relief (suck-back) is recommended.

Mold Temperature

Mold temperature is measured with a thermocouple directly on the cavity’s surface.

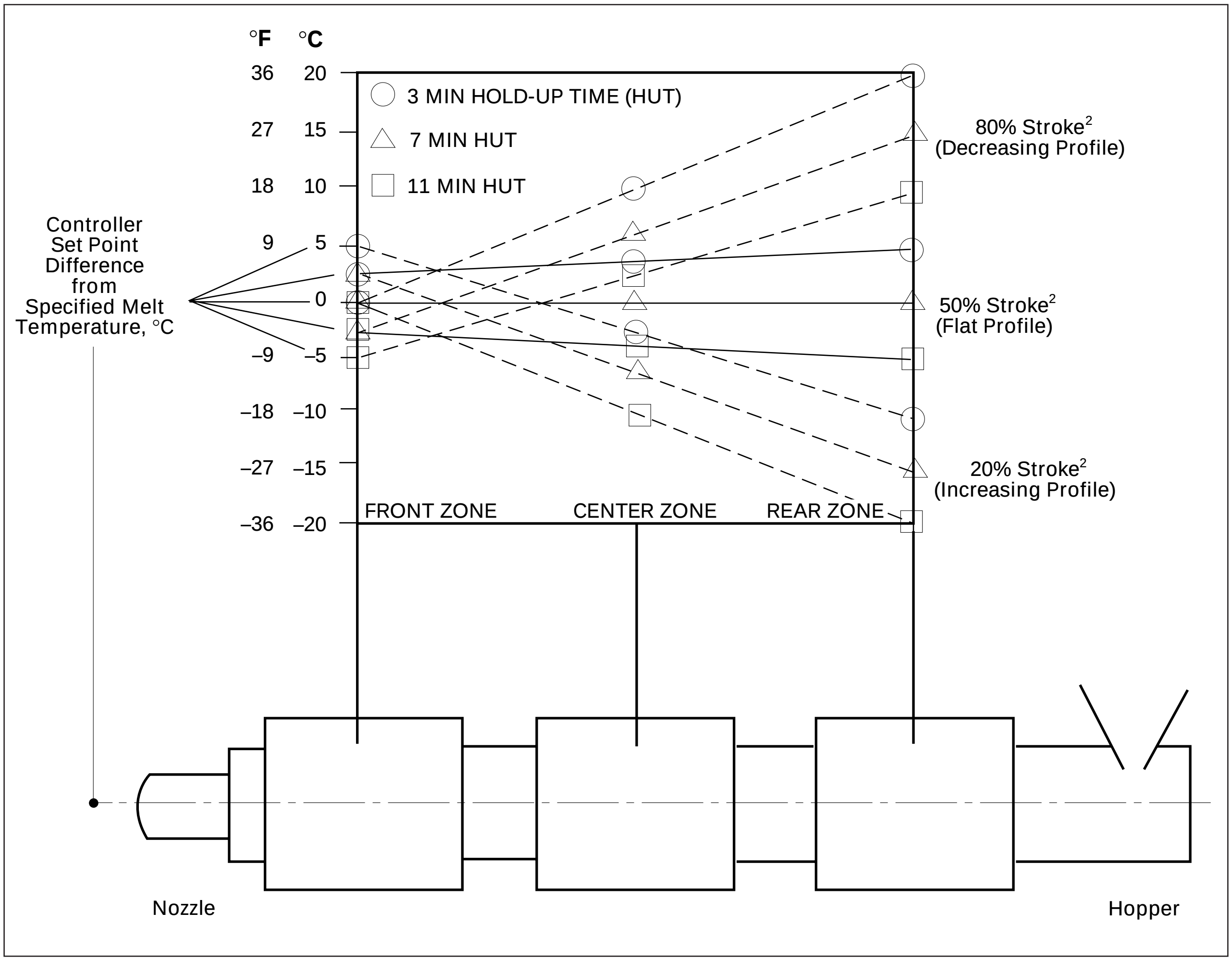
Recommended mold temperature for all grades is 45°C (113°F). Mold temperature has little effect on mechanical properties. The main effect is on shrinkage (see page 16).

Lower mold temperatures will reduce cycle time and improve ejection, particularly with the softer grades.

Higher mold temperatures will improve surface appearance.

Figure 14 shows the influence of mold temperature on the flow length at recommended melt temperature.

Figure 12. Cylinder Settings[1] for a Specified Melt Temperature—Recommended Controller Set Points from Target Value



[1] Barrel residence time, shot size and desired melt temperature each influences the barrel settings for optimum melt quality.
[2] The percent stroke refers to the portion of the actual machine shot capacity.

Injection Speed

Injection speed varies with part thickness and geometry. Thin parts should be filled rapidly before the polymer cools. In general, higher fill rates will improve surface finish, but too high rates may cause jetting or turbulence that may result in surface defects.

Injection and Holding Pressure

The injection pressure should be set to the minimum pressure required for filling the cavity.

For the harder grades of Hytrel® (above 55D) the hold pressure can be set equal to the injection pressure. For the softer grades (below 47D) the hold pressure should be set to follow a decreasing pressure profile. Excessive hold or injection pressure can result in overpacking and sticking in the mold cavity especially with the softer grades.

High pressure will reduce the apparent mold shrinkage, but can increase flash.

Figure 15 shows the flow length of various grades and **Figure 16** shows the effect of injection pressure on the flow length.

Table 7
Recommended Optimum Melt Temperatures and Injection Molding Setup Conditions for Hytrel®

Hytrel Grade	Recommended Optimum Melt Temperature	Melt (Stock) Temperature Range	Typical Cylinder Temperatures			
			Nozzle	Front	Center	Rear
G3548W 4056	190°C (375°F) 180°C (355°F)	180–205°C (355–400°F)	180°C (355°F)	190°C (375°F)	190°C (375°F)	165–190°C (330–375°F)
3078 HTR6108 HTR8068	205°C (400°F) 205°C (400°F) 205°C (400°F)	190–210°C (375–410°F)	190°C (375°F)	205°C (400°F)	205°C (400°F)	180–205°C (355–400°F)
G4074 G4078W	200°C (390°F) 200°C (390°F)	190–220°C (375–425°F)	190°C (375°F)	205°C (400°F)	205°C (400°F)	180–205°C (355–400°F)
4069, 4556 5556, 5526 5555HS	230°C (445°F) 230°C (445°F) 230°C (445°F) 230°C (445°F)	220–250°C (430–480°F)	220°C (430°F)	235°C (455°F)	235°C (455°F)	205–235°C (400–455°F)
G4774, G4778 G5544 6356	240°C (465°F) 240°C (465°F) 240°C (465°F)	235–260°C (455–500°F)	235°C (455°F)	245°C (475°F)	245°C (475°F)	220–245°C (430–475°F)
7246 8238	245°C (475°F) 250°C (475°F)	240–260°C (465–500°F)	240°C (465°F)	245°C (475°F)	245°C (475°F)	220–245°C (430–475°F)

Figure 13. Influence of Melt Temperature on Flow Length

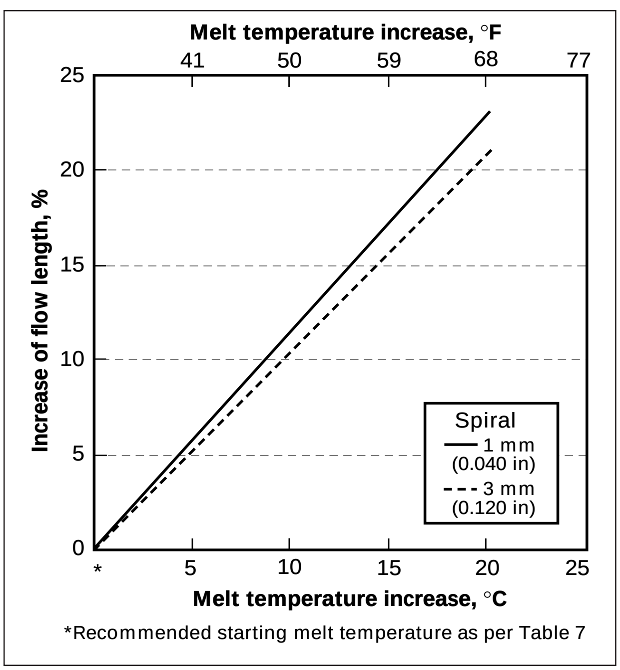


Figure 14. Influence of Mold Temperature on Flow Length (at Recommended Melt Temperature)

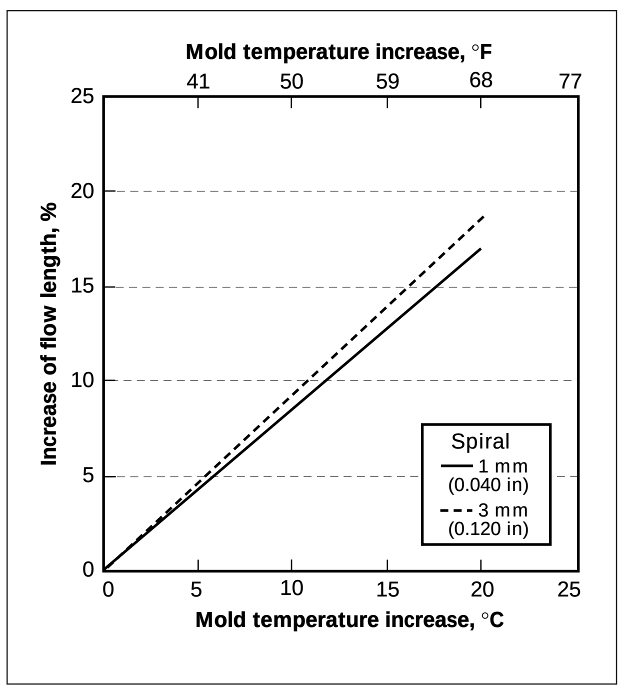
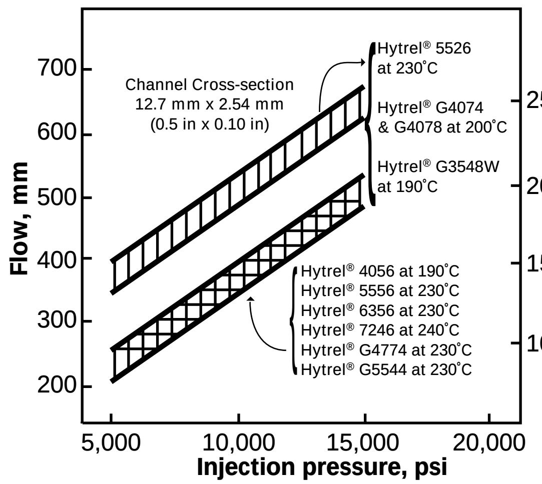


Figure 15. Snakeflow at Processing Temperature



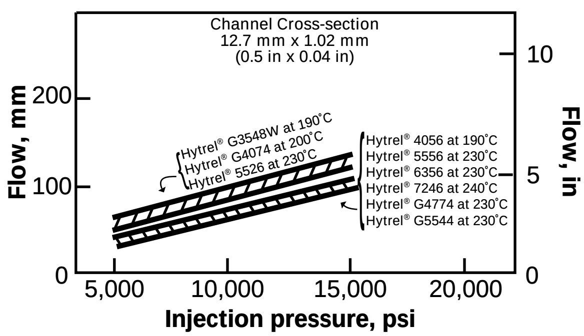
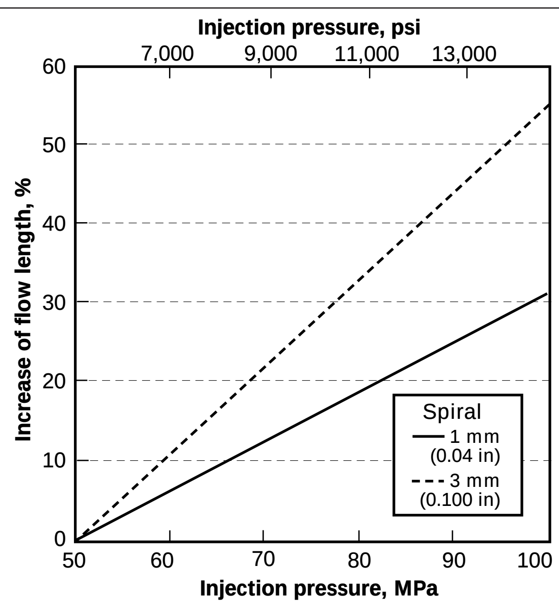


Figure 16. Influence of Injection Pressure on Flow Length (at Recommended Melt Temperature)



Other processing parameters are standard

Screw Forward Time (SFT)

The holding pressure should be maintained for the time necessary to avoid sink marks and for the gate to seal. This depends strongly on the grade of Hytrel®. In general, the screw forward time for harder grades is shorter:

72D–82D	4–5 sec/mm (101–127 sec/in) (for parts up to 4 mm thick)
44D–63D	5–6 sec/mm (127–152 sec/in)
35D–40D	7–8 sec/mm (178–203 sec/in)

The screw forward time has a strong influence on shrinkage.

Screw Rotation Speed/Back Pressure

Molten polymer throughput is controlled primarily by the speed (rpm) of the screw. For Hytrel® without additives, a medium to fast speed of 100 rpm is usually adequate. Some back pressure (0.34 to 0.55 MPa [50 to 80 psi*]) can be used to improve melt homogeneity. If additives (e.g., color concentrates) are being mixed in, higher screw speed and higher back pressure may be required to obtain adequate mixing.

Cycle Time

Molding cycle time is dependent on part size and on polymer melt and mold cavity temperatures. Cycle time ranges from 15 sec for thin parts to 3 min for thick parts. For a simple part of 6 mm (0.25 in) thickness, a cycle time of 1 to 1.5 min is a good starting point. The major elements of cycle are:

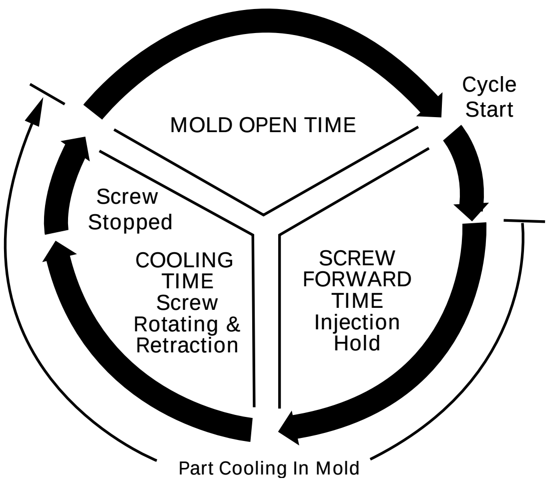
- screw forward time
- cooling time
- mold open time, as shown in **Figure 17**.

Screw forward time involves injection plus the time that the screw is held forward by hydraulic pressure. During cooling time, the screw rotates and retracts, and the moldings cool sufficently to be ejected when the mold opens.

In an ideal case, the molding cycle will be determined largely by screw forward time. The screw forward time, which is a function of part thickness and changes slightly with mold temperature, will be long enough for the part to freeze. This results in maximum weight, minimum mold shrinkage, and minimum tendency to form sinks or voids. Screw forward time needs to exceed gate freeze time by only 1–2 sec. Cooling time is determined by the

*gauge pressure

Figure 17. The Molding Cycle



ability of the part to eject without distortion. Screw speed should be set to provide a screw retraction time of a few sseconds less than cooling time.

Quality Moldings on Fast Cycle

To optimize part quality while attaining a cycle that will produce the maximum number of usable parts per machine per hour, you have to follow a step-by-step plan. Without such a plan, the parts molded may not be the best or the cycle may be longer than necessary. Before doing anything to change the cycle, a molder must find out from the end user or quality control supervisor what factors will cause a part to be accepted or rejected. Appearance, strength and dimensions may all be involved in determining what is required of the final molded part. Molding parts with excessive precision can make them too expensive because of wasted cycle time.

No attempt to determine the quality of parts should be made before a full shot has been made and parts are well packed out. Be sure that the correct material is being used, that it has been dried if necessary and that there is an adequate supply. Check to see that the machine and mold are operating smoothly. A few hours spent correcting a fault in a mold or machine can pay for itself quickly in faster cycles.

Shortening Injection Time

Start your adjustment of the molding cycle by purposely making the cycle longer than necessary and then ascertain that all parts produced are within specifications. Next, weigh indivdual parts—on a precision scale if possible. Record individual cavity weights. To assure quality parts, weights must remain near the maximum even as the screw forward time is shortened. Careful experimentation to determine exactly how long the screw must be held forward to make acceptable parts is the next and key step in establishing the shortest cycle.

While evaluating the effect of a shorter screw forward time, avoid any change in the length of the overall cycle. Shortening the cycle at this point could change the temperature of the mold and of the resin in the cylinder. On most machines this will require that any time subtracted from the injection phase must be at least temporarily added to the hold phase. Thus, the overall cycle is, for the time being, constant. Continue reducing injection time until some fault is found in the part such as non-flatness, a dimensional shortcoming or a loss in part weight. At that point, move the time back a notch at a time—increasing the injection time—until the problem disappears. Let the machine run for several shots to see that quality is being maintained. You have now established the minimum period for the injection phase of the molding cycle, although a minor adjustment upward may still be necessary to assure quality after you have completed the hold time adjustments covered below.

Shortening Hold Time

Leaving the injection timer on this newly established setting, begin reducing hold time by a few seconds each time and allow conditions to stabilize. Collect and label a few samples at each setting for later measurement. As the cycle gets shorter, screw rotational speed may need to be increased and the mold coolant temperature decreased. Continue reducing hold time until an obvious fault apears.

If not accomplished at an earlier stage, use this fine tuning period to also minimize mold open time by:

- Shortening mold opening stroke.
- Maximizing opening and closing clamp speeds, with slow clamp and break movements only enough to protect the mold.
- Looking for potential faults or delays in the hydraulic circuits and timers.
- Eliminating ejection problems or installnng sweeps or robot actuators for part and runner removal.

Good record keeping is very useful for trouble-shooting after the fact. DuPont recommended data record form is attached for information at the end of the brochure.

Mold Design

The following paragraphs stress some important aspects that should be considered when designing a mold for parts made of Hytrel®.

Materials of Construction

No special metals are required for most grades since Hytrel® has no corrosive action on the alloys commonly used for injection molds and cavities. An exception to this is when molding any flame-retardant compound: when operating above the recommended process temperature or when employing long residence times, corrosion-resistant equipment should be considered.

Mold Surface Finish

Textured and mat finished cavity surfaces minimize the effects of flow lines or marks and scratches on the part. Highly polished, plated mold finishes may cause difficulty in ejecting the soft grades of Hytrel® (below 47D).

Sprue Bushing Design

An incorrectly designed sprue bushing frequently causes sprue sticking and unnecessary cycle delays. The diameter of the sprue at the smaller end should be equal to the diameter of the runner it feeds. Standard bushing should have a taper of at least 2.5° (0.044 rad), but larger tapers result in less sprue sticking.

A properly mated injection nozzle and sprue bushing facilitates ejection of the sprue. The diameter of the hole in the nozzle should be 0.5–1 mm (0.02–0.04 in) less than that of the sprue bushing. Since Hytrel® is elastomeric, sprue pullers with a generous undercut (e.g., “Z” pullers, sucker pin, or offset undercut type) are needed for sprue removal.

Runners

Runners should be streamlined to reduce turbulence. A full round or trapezoidal runner should be used whenever possible to minimize pressure drop and for ease of removal. A trapezoidal runner should have its depth not less than 75% of its width. Runner systems should have a balanced layout. Runner section depends primarily on the rheology and freezing characteristics of the polymer melt, the runner length and the thickness of the part. To improve the flow and to facilitate ejection, the surface of the runners should be smooth but not polished.

Runnerless molding, both insulated and hot runner, may also be used. Sufficient heating capacity and control must be provided to ensure that neither freezing nor overheating occur. This will prevent unnecessary cycle interruptions and possible polymer degradation.

Gates

Fan gates, flash gates and tab gates (see **Figure 18**) are recommended in order to minimize flow lines and distortion at the gate. For thick section moldings, sprue gates are usually required to eliminate sinks.

Tunnel gates (see **Figure 18**) as small as 0.5 mm (0.02 in) can be used. The land length should be kept as short as possible (0.5–1 mm [0.02–0.04 in]) and the edges of the gate should be sharp to help break the gate. If the gate is large in diameter or the edges are radiused, the gate may be difficult to break off (especially with soft grades).

Gate dimension is important. Gates too small will require high injection pressure and will result in high shear forces. Oversized gates will require longer hold pressure time to avoid flow back and sink marks, or degating problems especially with the softer grades (below 47D). In general, gate thickness should be half of the part thickness. For parts less than 1.5 mm (0.06 in) thick, the gate should have the same thickness as the part. Gate lands should be between 0.5–1 mm (0.02–0.04 in).

To avoid sink marks and filling problems, the gate should be located in the thickest section of the part.

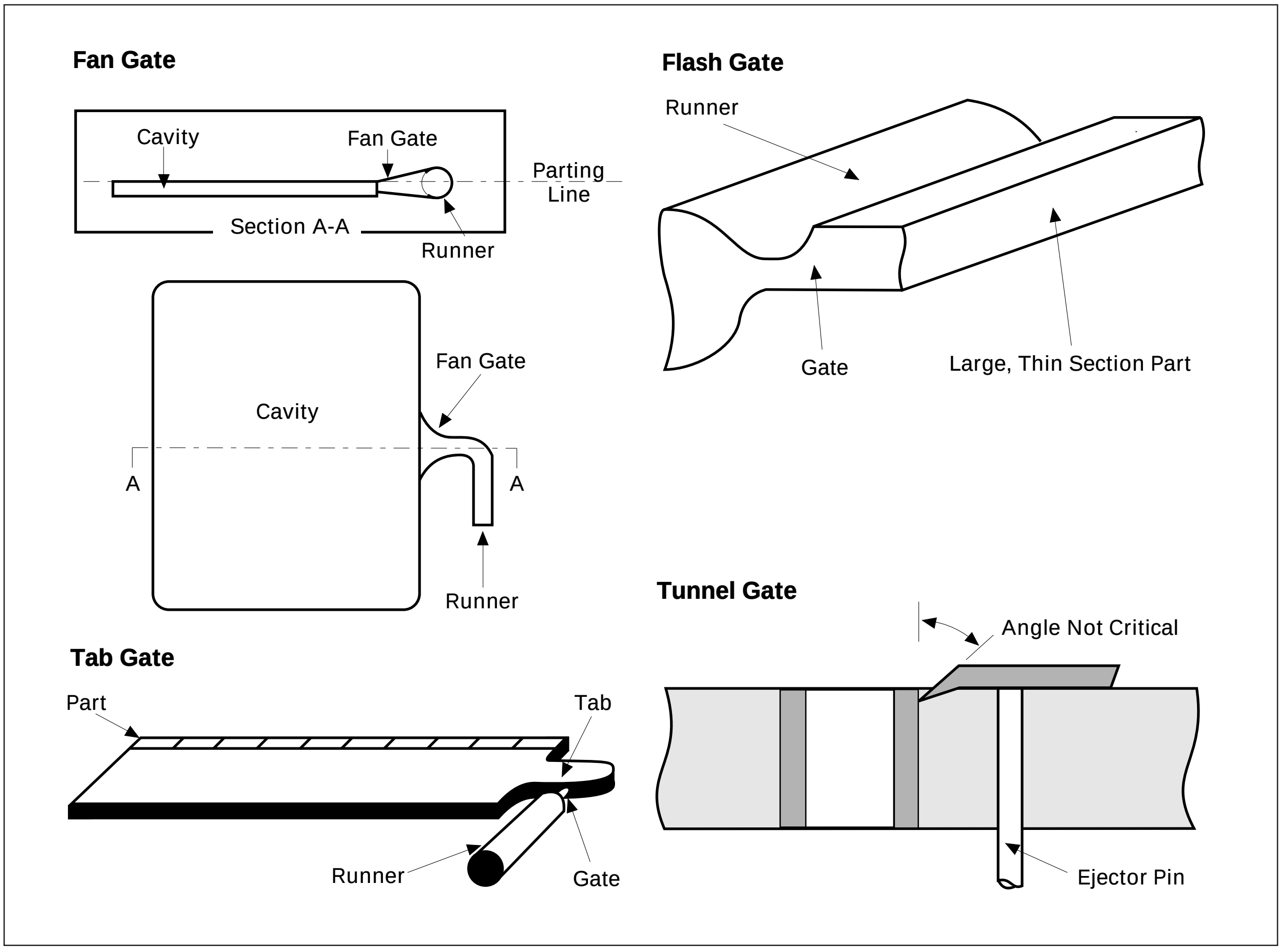
Venting

Venting provides a path for the escape of air from the cavity as melt displaces it. Flow into any cavity can be seriously reduced by inadequate venting of the cavity. (When runners are long or large in diameter, they should be vented as well.) This is important since fast cavity fill rates are commonly used with Hytrel®. The vent opening into the mold should be broad but thin. Vent openings up to 6 mm (0.25 in) wide should not be deeper than 0.04 mm (0.0015 in) to minimize danger of flash. Vents are positioned at points of final fill to prevent burning of the part from trapped air which can be compressed to very high temperatures. Sometimes, air entrapment cannot be predicted before initial mold trials, so frequently vents must be added after molds are released for production.

Undercuts

The depth of undercut that can be stripped from a mold will vary with the size and shape of the part, overall cycle, mold temperature and the grade of Hytrel® used. Undercuts should be radiused generously to aid ejection and should be no more than

Figure 18. Gate Design



0.8 mm (0.03 in) deep. Placing the undercut near the ejection or stripper plate helps to avoid distortion of the part on demolding.

Part Ejection

Ample draft, 0.5° to 2° (0.009 to 0.035 rad) taper per side, will ease ejection especially when a core is removed from a deep part or when a part is removed from a deep cavity. When a mold must have very little or no draft, stripper plates are recommended for ejection. When pin ejectors are used, they should have a large surface area and act on the thickest sections of the part. Ejector mechanism should be located to provide uniform stripping of the part from the mold.

If the part is small, the knockouts should be shaped proportionately to the part (i.e., ring, disc, etc.). If the part is large, use 13–25 mm (0.5–1 in) diameter pins if design permits.

Undercuts should have room to flex during ejection.

To reduce possible sticking problems, a matted surface finish on molds is preferable when molding the softer grades of Hytrel® (below 47D).

Shrinkage and Post-Molding Shrinkage

The shrinkage of Hytrel® in injection molding depends on numerous factors such as:

- Grade of Hytrel®
- Molding conditions (injection pressure, SFT, mold temperature, etc.)
- Part geometry and thickness
- Mold design, runner, sprue system, gate size

The shrinkage is measured at room temperature, and at 50% relative humidity on standard test specimen 24 hr after molding. Shrinkage increases significantly after molding, but tends to reach a maximum after 24 hr.

This section will provide some information on how shrinkage varies with these parameters. Unless stated, these shrinkage values were obtained on test specimens of 3.2 mm (0.125 in) thickness molded at standard conditions:

Mold temperature: 45°C (115°F)

Melt temperature: as recommended in Table 7
Injection pressure: 70 MPa (10,150 psi)
SFT: optimum

Table 8 gives the nominal shrinkage values for various grades, obtained under these standard conditions.

Most grades of Hytrel® show typical nominal mold shrinkage from 0.5 to 3%, on a 3.2 mm (0.125 in) truck part. Thick-walled parts exhibit higher shrinkage, while thinner parts exhibit low shrinkage with an exception of very thin parts.

Table 8
Shrinkage of Hytrel®
(ASTM D955)
(measured on standard test specimen, in flow direction 3.2 mm [0.125 in] thick, molded at recommended conditions)

Grade of Hytrel®	Shrinkage, %
Standard	
G3548W	0.5
G4074	0.8
G4078W	0.8
G4774, G4778	1.4
G5544	1.6
High-Performance	
4056	0.5
4069	0.8
4556	1.1
5526	1.4
5556	1.4
6356	1.5
7246	1.6
8238	1.6
Specialty	
5555HS	1.4

Figures 19–21 show the influences on shrinkage of different injection molding parameters. They will provide a general guideline to help in predicting the shrinkage. Nevertheless, they cannot give an exact value. **The shrinkage evaluation for precision parts should be made on a prototype tool.**

The shrinkage values given in the following figures should be added to or subtracted from the nominal shrinkages given in **Table 8** in order to get a first approximation of the final shrinkage.

Figure 19. Influence of Mold Temprature on Shrinkage

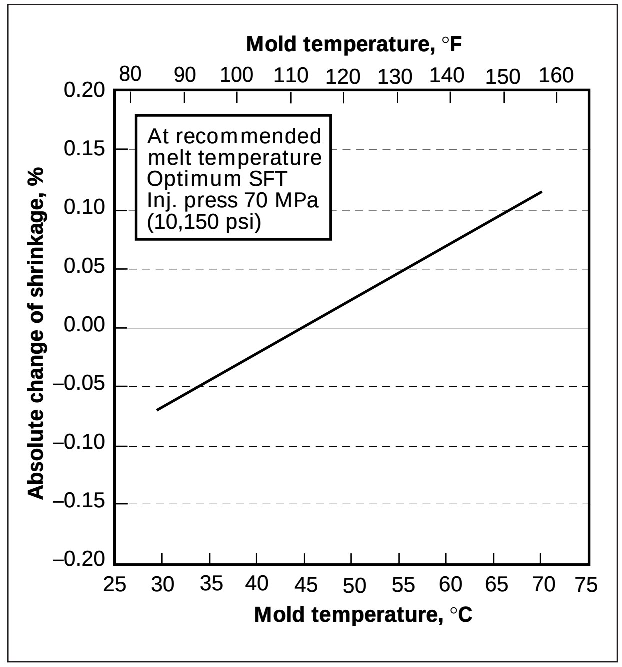


Figure 20. Influence of Injection Pressure on Shrinkage

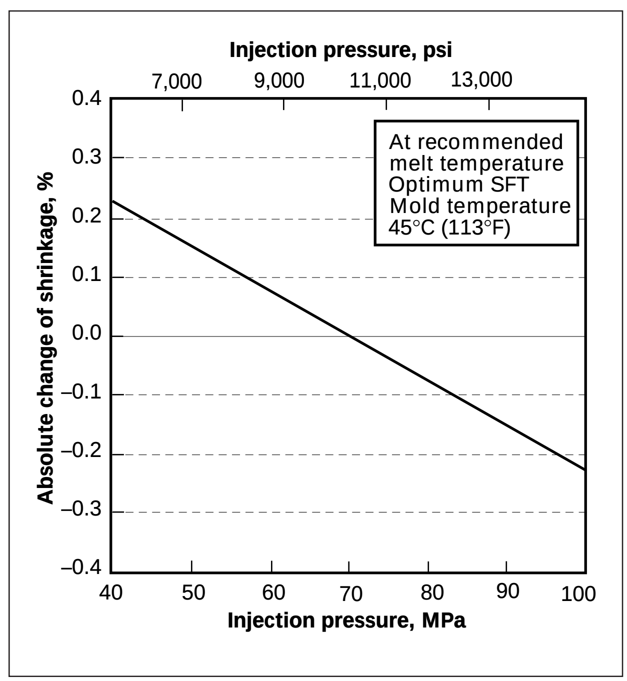
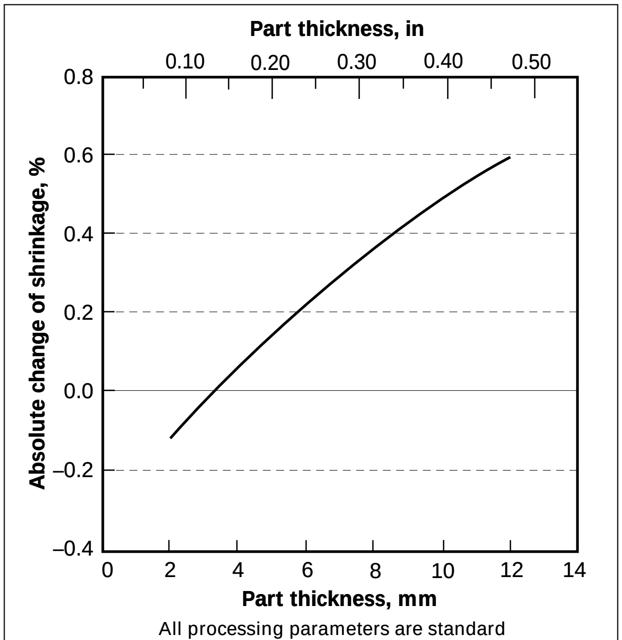


Figure 21. Influence of Part Thickness on Shrinkage



For example, an approximation of the shrinkage of a part made of Hytrel® can be done as follows:

Nominal shrinkage of Hytrel 5526:	1.40% (**Table 8**)
Part is molded using a 64°C mold temperature (versus 45°C):	+0.08% (**Fig. 19**)
Part is molded using an injection pressure of 900 bar (versus 700 bar):	–0.15% (**Fig. 20**)
Part has a thickness of 2 mm (versus 3.2 mm):	–0.13% (**Fig. 21**)
The **approximation** of the total shrinkage is:	**1.20%**

Post-Molding Shrinkage

Post-molding shrinkage is measured after annealing parts at 120°C (248°F) for 4 hr. Even for the stiffer and more crystalline grades, the absolute value of post-molding shrinkage for parts molded at recommended conditions is low, less than 0.1%.

Overmolding

Overmolding (or insert molding) is a process in which a thermoplastic material is molded directly onto a second thermoplastic material (the insert). Hytrel® generally overmolds best when used with other grades of Hytrel®; however, other materials like PBT, polycarbonate, ABS can be used with good adhesion and processibility.

The optimum process requires that the insert grade have a relatively low melting point (<190°C [<374°F]), and preferably with a broad melting point for slower crystallization. In order to achieve a good bond, the material used as the overmold should be injected at least 30°C (86°F) higher in melt temperature, and be processed at a somewhat higher temperature than is typically used for injection molding. This ensures that the higher melting resin can melt the surface of the insert, thus establishing a good bond.

If these requirements cannot be met, then the design may incorporate a mechanical bond (molded-in mechanical locking devices) or design with some flash or projection which can melt together to form a bond. The insert can also be mechanically abraded or may even require an adhesive to achieve a good bond. In all cases, the insert should be dry and free of grease and oil which could interfere with a good bond.

Operating Precautions

All safety practices normally followed in the handling and processing of thermoplastic polymers should be followed for Hytrel® polyester elastomer.

DuPont is not aware of any health hazards connected with Hytrel® as shipped to customers. This product information deals with potential hazards connected with melt processing of these resins.

Skin Contact

As with all thermoplastics, serious thermal burns can result from skin contact with melt at normal processing temperatures of 175° to 260°C (347° to 500°F). This could happen when moisture and other gases that generate pressures can eject molten polymer through the nozzle or hopper. Heat resistant gloves and long sleeved clothing should be worn when exposure to molten polymer is likely. In the event molten polymer does contact the skin, immediately cool the polymer and burned areas of skin with cold water or ice. Do not attempt to peel the polymer from the skin. Call a physician for medical treatment of the burn.

Pressure Buildup

Most thermoplastic polymers, including Hytrel®, can decompose to give gaseous products when heated for a prolonged period of time. These gases can generate high pressures if confined. Pressure buildup can be high and rapid, and metal parts could rupture and injure personnel.

Avoid the following which can cause Hytrel® to degrade to form gaseous decomposition products:

- Excessively high temperature
- Wet resin
- Molten polymer held in the machine too long at normal operating temperatures of 175° to 260°C (347° to 500°F)
- Contaminants that promote decomposition at normal operating temperatures.

Danger signals which warn that the polymer is beginning to degrade and decompose are: frothing and spitting of molten polymer at the nozzle, pronounced odor, discolored polymer, or badly splayed parts.

If extensive decomposition occurs, heat should be shut off and the machine should be purged. Medium viscosity polyethylene resin is recommended for purging. High pressure usually will result from decomposition unless the gas can escape from the feed port. Accordingly, when molding Hytrel®, it is important that the operator be familiar with the factors which can cause decomposition, the danger signals which warn of this problem, and the action that should be taken to prevent pressure buildup.

Fire

Hytrel resins can ignite and burn if exposed to temperatures in excess of 300°C (572°F) or to open flame. Therefore, polymer temperature should be kept within the recommended processing range to minimize this potential. Heat to the molding machine should be shut off if any delay in processing greater than 30 min is anticipated.

A principal gaseous decomposition product of Hytrel® is tetrahydrofuran (THF). Formation of THF vapors can occur under any of the conditions mentioned in the previous section. While THF is a potential fire or explosion hazard (open cup flash point –14°C [7°F]) the danger of fire through ignition of THF or other flammable vapors resulting from the decomposition of Hytrel® can be minimized by providing good ventilation in the molding area. It is recommended that masses of molten polymer from purging be quenched with water to minimize degradation and the formation of gaseous decomposition products.

Fumes

Under normal processing conditions the amount of gaseous decomposition is minimal so fumes are usually not a problem. Decomposition is dependent on high processing temperature and excessive time that the polymer is held at that temperature. Decomposition can occur either within molding equipment or in pools such as those formed when purging the molding machine.

Thermal decomposition of Hytrel® resin produces THF and other gaseous products including acetaldehyde and acrolein (see **Table 9**). *These fumes can be hazardous.* The 8-hr time weighted average (TWA) for maximum permissible exposure limit (PEL) for THF is 200 ppm; its 15-min short-term exposure limit (STEL) is 250 ppm. The 8-hr TWA for acetaldehyde is 100 ppm; the 15-min STEL is 150 ppm. The 8-hr TWA for acrolein is 0.1 ppm; the 15-min STEL is 0.3 ppm.

Depending on the time and temperature conditions, the fumes evolved during thermal degradation of Hytrel® may contain:

- Pyrolysis products such as ethylene and benzene
- Oxidation products such as carbon dioxide, carbon monoxide, water, ethers, alcohols, and aldehydes including acrolein and crotonaldehyde

The recommended standard practice of ventilation in the molding area prevents fumes from being discharged and accumulated. Do not rely on odor detection to determine the effectiveness of the ventilation since the exposure limit for acrolein is below the concentration at which its odor can be detected.

Good ventilation should also be provided during any regrind operations and during the burnout of any equipment containing Hytrel®.

Table 9
Exposure Limits for Decomposition Products of Hytrel®

Decomposition Product of Hytrel®	8-hr TWA, ppm	15-min STEL, ppm
Tetrahydrofuran	200	250
Acetaldehyde	100	150
Acrolein	0.1	0.3

Additional precautions must be taken to avoid processing above the recommended process temperatures and holdup times when molding some specialty resins, such as a flame retardant grade, or when using additive-containing concentrates. Before proceeding with any compounding or processing work, consult and follow label directions and handling precautions from suppliers of all ingredients. Additional precautions for Hytrel® resins and concentrates can be found in bulletin, "Handling and Processing Precautions for Hytrel®" and the appropriate Material Safety Data Sheets.

Spills

Spilled pellets can be very slippery underfoot and should be swept up immediately and disposed of properly.

Waste Disposal

Preferred options for disposal are (1) recycling, (2) incineration with energy recovery, and (3) landfill. The high fuel value of this product makes option (2) very desirable for material that cannot be recycled. Forced draft incineration gives good combustion and converts Hytrel® to CO_2, water and trace components. However, incineration is not recommended for any materials containing flame retardant. Disposal must be in accordance with applicable federal, state/provincial and local regulations.

Drying

Since Hytrel® can be dried at temperatures as high as 110°C (230°F), contact with hot hoppers, ovens, or air lines could result in severe burns. Insulation of these components is recommended.

Static Electricity

If Hytrel® is handled in the vicinity of flammable vapors or dust, then adequate grounding is recommended.

Injection Molding Troubleshooting Guide

This section identifies various problems which might be experienced during the injection molding of Hytrel® engineering thermoplastic elastomer. It also lists the most likely causes of these problems and suggests possible solutions. In all cases, listings are in the order of most likely occurrence. In addition, more than one description of the same or a similar problem is sometimes given because problems can be interrelated. All suggested solutions should be followed until the problem is resolved. If the problem cannot be solved by following these suggestions, please contact the nearest DuPont Engineering Polymers sales office, provided on the back page of this bulletin.

Troubleshooting Guide		
PROBLEM	**POSSIBLE CAUSE**	**SUGGESTED SOLUTION**
I. Short shots—at the start of injection molding operation—injection ram is bottoming	A. Shortage of material	• Check the injection stroke and increase as necessary. • Increase hold time or rpm. • Be sure the feed hopper has sufficient material and that the shut-off gate is open. • Check the feed system for blockage and bridging. • See that the air and power supply to the weigh feeder (if used) are turned on.
	B. Machine capacity is too small	• If none of the above provides sufficient feed, it will be necessary to: (a) place the mold in a larger shot capacity press; or (b) block off some of the mold cavities.
	C. Polymer melt is slipping past the screw (ram)	• Use a non-return screw tip. • Check the non-return tip and barrel for excessive wear or a jammed ring valve. • Reduce temperature of the polymer melt.
II. Short shots—at the start of the injection molding operation—injection ram is not bottoming	A. Injection time is too short	• Increase time of injection.
	B. Injection pressure is too low	• If ram is completely stopped before the end of the injection cycle, increase injection pressure. Operate at maximum injection speed (higher boost pressure). • Provide sufficient venting for each mold cavity.
	C. Cylinder temperature is too low	• If the machine is at maximum injection pressure, raise the cylinder temperatures. • Check actual temperature of the melt with a needle pyrometer.
	D. Heater bands on the nozzle or cylinder are inoperative	• Check all heater bands for proper operation with a pyrometer or clamp-on ammeter.

Troubleshooting Guide

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
II. Short shots—at the start of the injection molding operation—injection ram is *not* bottoming *(continued)*	E. Nozzle, sprue or gates are blocked or frozen	• Check orifices of the nozzle, sprue and gates for foreign or unplasticized material.
	F. Excessive resistance to flow in the sprue bushing, runners, vents and/or gates	• Enlarge these flow paths as necessary consistent with machine shot capacity and sufficient melt velocity to preclude premature freezing.
	G. Material viscosity is too high (melt index is too low)	• Increase temperature of the melt. • Use resin with a lower viscosity, if possible. (See bulletin, "Rheology and Handling.")
III. Short shots—after a period of successful injection molding operations	A. Check Items D and E in Section II	• See suggested solutions for Section II, Items D and E.
	B. Loss of injection pressure	• Check hydraulic system for defective pumps or valves. • Check for low oil level. • Check for overheated oil supply, possibly due to loss of coolant or plugged heat exchanger.
	C. Venting is inadequate (usually accompanied by burned or charred spots on molded part)	• Check for blockage of vents.
	D. Shortage of material	• See suggested solutions for Section I, Item A.
	E. Interrupted feed	• Clear bridging in the feed throat.
	F. Polymer is sticking in the feed throat	• Increase cooling of the feed throat. • Reduce temperature of the rear zone.
IV. Short shots—occur periodically during injection molding operations	A. Cylinder temperature controller is cycling too broadly	• Consult technical service representatives for the temperature controllers used on the injection molding machines. • Increase overall cycle time.
	B. Cycles are inconsistent	• Check all timers with a stopwatch for consistent timer control. • Check time ram is in motion. (Inconsistent time indicates melt is non-uniform.) • If on semi-automatic cycle, check for variations in operator-controlled portion of the cycle. • Check hydraulic system for sticking solenoid valves. • Check if the ring shut-off valve on the non-return screw tip is worn or clogged.

Troubleshooting Guide

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
V. Flashing	A. Injection pressure is too high B. Too much material is being injected into the mold C. Material is too hot	• Reduce injection pressure. • Reduce shot size or run without a pad. Note Section VII, Item A. • Reduce pressure to pack out. • Reduce temperature of the melt.
	D. Clamp end of press is out of adjustment	• Reset the toggles and/or increase clamp pressure.
	E. Flash or foreign material is acting as a high spot on mating surfaces of the mold F. Mold surfaces, cores and/or cavity inserts are out of register	• Inspect land areas, etc., of the mold carefully and clean where necessary. • Remove mold, overhaul and correct the register.
	G. Mold or platens are warped H. Clearance in vents, knockouts, etc., is too great	• Check and overhaul if necessary. • Check clearance and adjust as necessary. • Clearance should not be more than 0.038 mm (0.0015 in).
	I. Venting is insufficient or blocked thereby forcing material from the cavity area	• Inspect the vents and clean if necessary. • Increase width of the vents. Vents should not be more than 0.038 mm (0.0015 in) in depth.
	J. Injection pressure is unevenly distributed in the mold K. Projected cavity area is too large for the available clamping pressure	• Cavity and runner layout should be balanced. • Shift to a press with greater available clamping pressure. • Reduce the number of cavities.
VI. Ejection difficulties	A. Excessive flashing B. Material too highly packed in the cavity (mainly with large gates)	• See suggested solutions for Section V. • Reduce injection pressure and/or hold pressure. • Reduce size of the shot pad. • Reduce time that injection ram is forward.
	C. Pieces deform during ejection (part is too soft)	• Increase time of the overall cycle. • Reduce temperature of the mold. • Increase diameter and number of knockout pins. • Use rubber type sprue puller or sucker pins with more undercut. • Incorporate air ejection in conjunction with mechanical methods. • Sand blast or vapor-hone mold core and core pins in the direction of ejection.

Troubleshooting Guide		
PROBLEM	**POSSIBLE CAUSE**	**SUGGESTED SOLUTION**
VI. Ejection difficulties *(continued)*	D. Parts stick to the mold due to highly polished surfaces	• Check suggestions in Section VI, Item C. • Use internal or external mold release. • Use matte finish on the mold cavity.
	E. Mold conditions:	
	1. Mold surfaces are scratched and marred	• Overhaul and polish the mold surfaces.
	2. Draft or taper on cavity walls, cores or sprues is not great enough	• A minimum of 1° (0.017 rad) taper on long cores or cavities is required.
	3. Undercuts are improperly designed	• Undercuts should not have sharp angles but should be tapered to ease ejection.
	4. Sprue bushing and nozzle orifice are misaligned	• Align nozzle and sprue bushing.
VII. Warpage or part deformation	A. Molded-in stresses are too high due to:	
	1. Excessive packing of the cavity	• Reduce injection pressure. • Increase venting. Operate without a shot pad (with the ram bottoming). If shrinkage is a concern, note suggestions in Section VIII, Item E.
	2. Cavities being filled too slowly	• Increase temperatures of the cylinder and/or ram speed (boost pressure).
	3. Melt temperature being too low or non-homogenous	• Increase temperatures of the cylinder and/or screw speed.
	B. Part is being ejected while still too hot	• Reduce temperature of the mold. • Increase time of the overall cycle. • Reduce temperatures of the cylinder. • Consider use of shrink or cooling fixtures.
	C. Ejector mechanism is improperly designed	• Redesign. Use knock-out pins with larger area or use stripper plates.
	D. Part is improperly designed (non-uniform walls)	• Redesign. Use walls with a more uniform thickness or gradual changes in thickness.
	E. Gates are improperly located and/or designed	• Redesign or relocate the gates. Gate into thickest sections toward longest flow path.
	F. Undercuts, ribs, bosses, threads, etc., are improperly designed	• Redesign. Undercuts should be radiused and no more than 0.8 mm (0.03 in) deep. Use ribs and bosses of minimum thickness.
	G. Mold cooling is inadequate (Capacity of the cooling system is too low, cooling circuits in the mold halves are not balanced, heat transfer is poor)	• Increase capacity of cooling. • Modify coring to give adequate cooling. Locate coring closer to the cavity surface.

Troubleshooting Guide		
PROBLEM	**POSSIBLE CAUSE**	**SUGGESTED SOLUTION**
VII. Warpage or part deformation *(continued)*	H. Moveable mold components (cores) have shifted or become misaligned	• Realign.
	I. Runner system is inadequate	• Redesign.
VIII. Excessive shrinkage	A. Gates not frozen off	• Increase time injection ram is forward.
	B. Effective injection pressure in the cavities is too low	
	1. Gates are too small or improperly designed	• Increase size of gates and/or shorten length of lands.
	2. Runner system is improperly designed (diameters and layout are incorrect)	• Increase size of runners to decrease resistance to polymer flow. Runners should be sized so they maintain a relatively constant shear rate for the required volume of flow. Gates should be sized for proper freeze-off.
	3. Melt temperature is too low	• Check actual temperature of the melt with a needle pyrometer. If necessary, increase temperatures of the cylinder.
	4. Flow rate of material is too low	• Use polymer with a higher melt index, if possible. (See bulletin, "Rheology and Handling.")
	5. Nozzle orifice is too small	• Use nozzle with a larger orifice.
	C. Injection pressure is too low	• Increase pressure of injection slowly until borderline flash conditions are reached. Note suggestions in Section VII, Item A.
	D. Mold temperature is too high	• Reduce temperature of the mold.
	E. Not enough material in the cavity	• Increase size of shot to obtain a very slight pad. Note suggestions in Section VII, Item A.
	F. Dwell time is too short	• Increase time injection ram is forward.
	G. Molding conditions not optimized	• See page 17.
IX. Sinks, shrink marks, voids, bubbles	A. With the exception of Item B-4, the causes shown for Section VIII generally apply	• See suggested solutions for Section VIII. • Resins with a low flow rate may assist in minimizing voids in heavy sections of easy to fill parts. Also, the use of *slow* speeds for the injection ram might help in the control of sinks and voids.
	B. Moisture content of the polymer is too high	• Dry the polymer. For suggested drying procedures see bulletin, "Rheology and Handling."
X. Burning, charring or black specks	A. Material is too hot	• Reduce temperatures of the cylinder. • Shorten time of cycle.

Troubleshooting Guide		
PROBLEM	**POSSIBLE CAUSE**	**SUGGESTED SOLUTION**
X. Burning, charring or black specks *(continued)*	B. Molten resin is exposed to air in the machine due to starving the feed section or entraining air in the screw feed	• Keep a reserve of resin in the hopper to avoid starving the feed section. • Reduce screw speed on the screw injection molding machines to obtain melt in the feed section of the screw before significant compression.
	C. Vents are inadequate or blocked	• Inspect and clean vents. • Vent at point where polymer is burning.
	D. Material is entering the cavities too rapidly	• Sufficient venting normally corrects this problem. If this doesn't solve the problem, try reducing the injection ram speed. (See suggestions Section II, Item B.)
	E. Material is hanging up in the heating cylinder and/or nozzle (generally indicated by specks or streaks in the molded item)	• Clean the nozzle and cylinder with purge compound or disassemble. Polymer flow path should be streamlined with no dead spots for polymer hang-up.
	F. Regrind is of questionable quality	• Segregate and check the regrind critically for contamination, excessive moisture or degraded polymer. Try virgin material.
	G. Previous polymer or purge material has not been completely removed	• Purge with Hytrel until the machine is free of other polymers or remove the screw and nozzle and clean thoroughly.
XI. Degradation	A. Material is overheated	• Reduce temperatures of the cylinder. • Shorten time of cycle.
	B. Thermocouple is burned out	• Check all thermocouples for proper operation.
	C. Temperature controller is malfunctioning	• Check for sticking relays. • Check for sluggish or stuck meter movements in all controllers. • Calibrate controllers. • Check for controllers which may be connected to the wrong heaters.
	D. Regrind is of questionable quality	• Segregate and check regrind critically for contamination, excessive moisture or degraded polymer. Measure the melt index of each polymer feed component. (See bulletin, "Rheology and Handling.") Try virgin material.
	E. Improper shutdown procedures were used (over weekends or periods of interrupted production)	• Purge machine thoroughly until degraded (low viscosity) polymer has been discharged. Rule-of-thumb is that polymer hold-up is four times the maximum shot capacity of the machine.

Troubleshooting Guide

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
XI. Degradation *(continued)*	F. Moisture content of the polymer is too high	• Dry the regrind and polymer. For suggestions on drying, see bulletin, "Rheology and Handling."
	G. Polymer residing in the barrel too long	• Change to a smaller capacity machine. Shot should be between 25% and 75% of the machine capacity. If a smaller capacity machine is not available, use a temperature profile with the front zone and nozzle at the desired melt tempera ture and all other temperatures as low as operable.
	H. Stagnation of material in the cylinder, nozzle, or nozzle valve	• Inspect the cylinder. Eliminate dead spots (streamline) as necessary.
XII. Dimensional variations	A. Non-uniform feed due to: 1. Variation in machine operation 2. Variation in the material	 • Check operation of the feed mechanism. • Check pellet size for variations. Check feed throat for obstructions or sticking polymer.
	B. Cylinder temperatures are cycling too broadly	• See suggestions in Section IV, Item A.
	C. Cycles are inconsistent	• See suggestions in Section IV, Item B.
	D. Machine capacity is too small	• See suggestions in Section I, Item B. For consistency of dimensions when molding polymers of Hytrel, it is suggested that the shot size not exceed 75% of the machine's plasticizing capacity.
	E. Mold temperatures inadequately controlled • Check location of coring.	• Check coolant for temperature variations. Install temperature controller if needed.
XIII. Surface defects on the molded article	A. Mold lubricant used excessively	• Clean mold surfaces thoroughly. Use lubricant sparingly for 40D polymer or not at all (wipe on—don't spray on).
	B. Moisture on cavity surfaces	• Wipe mold surface throughly with a rag moistened with alcohol. • Raise the mold temperature. • Apply anti-condensation material to the outer surface of the mold base. • Check for coolant leaks.

Troubleshooting Guide		
PROBLEM	**POSSIBLE CAUSE**	**SUGGESTED SOLUTION**
XIII. Surface defects on the molded material *(continued)*	C. Material conditions	
	1. Contamination by foreign material	• Inspect rework material thoroughly. • Use care in handling materials and caution in keeping foreign materials clear of hopper and work area.
	2. Bubbles due to:	
	a. Trapped air	• Reduce temperature of *rear* cylinder. • Increase back pressure.
	b. Moisture is condensing on cold pellets when they are moved into a warm, humid processing area	• Use a dryer to remove condensed moisture. Store pellets in the processing area for a minimum of 4 hr prior to use.
	c. Moisture absorbed in the polymer	• Dry the resin. Use a hopper dryer.
	D. Delamination due to:	
	1. Contamination of the material	• Check the material for foreign matter.
	2. Material being too cold	• Increase temperatures of the cylinder.
	E. Pigment poorly dispersed	• See suggestions in Section XIV, Item A.
	F. Cloudy or hazy surfaces—low gloss	
	1. Injection pressure is too low	• Increase pressure of injection. See suggestions in Section VII, Items A-1 and A-2.
	2. Injection speed is too low	• Increase speed of injection. See Section X, Item D.
	3. Effective injection pressure in the cavities is too low	• See suggestions in Section VIII, Item B.
	4. There is moisture on the mold and/or pellet surfaces	• Dry the resin. See suggestions in Section XIII, Items B and C.
	G. Flow lines (ripple pattern)	
	1. Gate design and/or location is not correct	• Redesign and/or relocate gate.
	2. Material is too cold	• Raise temperatures of the cylinder and/or mold.
	3. Injection speed is too slow	• Increase speed of injection.
	4. Mold is too cold	• Increase temperature of mold.
	5. Flow rate of the material is too low	• Use polymer with a higher melt index, if possible. (See bulletin, “Rheology and Handling.”)
	6. Polymer melt is jetting into the cavity	• Decrease speed of injection. Correct design and/or location of gate.
	7. Polymer melt is non-uniform	• Regrind or additives are not well dispersed in the virgin polymer.

Troubleshooting Guide		
PROBLEM	**POSSIBLE CAUSE**	**SUGGESTED SOLUTION**
XIII. Surface defects on the molded article *(continued)*	H. Weak weld lines due to:	
	1. Material being too cold at the point of weld	• Raise temperatures of the cylinder and/or mold.
	2. Material flowing too slowly to the point of weld	• Increase injection speed.
	3. Weld line being too far from the gate	• Improve venting or install an overflow tab.
	4. Effective injection pressure in the cavities is too low	• See suggestions in Section VIII, Item B.
	5. Use of excessive mold release	• Clean the mold. Use mold release sparingly or not at all.
XIV. Poor color dispersion	A. Because of poor mixing . . .	
	1. In a ram machine	• Use a dispersion nozzle or premix before molding. Change to a screw machine for better mixing.
	2. In a screw machine	• Increase the head or back pressure and/or screw rpm. Use a high shear or mixing screw or mixing nozzle.
	B. Because of the pigment . . .	
	1. Particles are too coarse	• Grind the pigment or obtain as a powder.
	2. Feed is non-uniform	• Use a color feeder. Preblend pigment and polymer.
	3. Pigment is difficult to disperse	• Use predispersed pigment concentrate.
	C. Because of the concentrate . . .	
	1. Letdown ratio is too great	• Use a lower ratio, letdowns greater than 25 to 1 are difficult by injection molding.
	2. Base polymer is not compatible with Hytrel	• Check with the concentrate supplier or DuPont. Use Hytrel as the base polymer.
	3. Pigment concentration is too high	• Use a concentrate with a lower level of pigment at a lower letdown ratio, for example, 15 to 1 rather than 25 to 1.

DuPont Engineering Polymers
MOLD INSPECTION AND REPAIR RECORD

MOLD NAME ______________________ MOLD NUMBER ______________________

PART NUMBER(S) MOLD SIZE:

CUSTOMER'S MOLD NUMBER ______________________ ________ X ________ X ________
length width height

FITS FOLLOWING MACHINES ______________________

______________________ DATE RECEIVED ______________________

SPECIAL SET-UP EQUIPMENT REQUIRED ______________________

Mark (√) if item is O.K. for next run. Mark (R) if item must be repaired before next run. Give brief explanation.

MOLD CONDITION													DESCRIBE MAINTENANCE REQUESTED	DATE	DESCRIBE MAINTENANCE COMPLETED	DATE
Inspec. date	Sprue brushing	Runner	Puller (sprue)	"A" cavity and cores	"B" cavity and cores	Ejectors	Cooling flow	Side Action	Vents	Wiring and heaters	Others	Others				

DuPont Engineering Polymers

MOLDING DATA RECORD

JOB	MOLD DESCRIPTION	MACHINE SETUP INSTRUCTIONS	SPECIAL INSTRUMENTATION
OPERATORS	SCREW USED		SAFETY CHECK
ENGINEERS	PRESS NO.	NOZZLE #	

DATE / TIME	Run Number	Resin	Lot Number	TEMPERATURES (°C) (°F)					MOLD			PRESSURES (MPa) (psi)				CYCLE TIMES (sec)									WEIGHTS (g)		REMARKS
				Rear	Center		Front	Nozzle	Fixed	Movable	Melt	Injection 1st Stage	Injection 2nd Stage	Clamp (Tons)	Back	Injection	Hold	Open	Overall	Booster	Ram in Motion	Screw Retraction	Pad (Inches)	RPM	Full Shot	Part Only	

COMMENTS ON MOLDING OPERATIONS, START-UP, ETC.

PAGE NO

Start with DuPont

For more information on Engineering Polymers: (800) 441-0575

http://www.dupont.com/enggpolymers

For Automotive Inquiries: (800) 533-1313

U.S.A.

East

DuPont Engineering Polymers
Chestnut Run Plaza 713
P.O. Box 80713
Wilmington, DE 19880-0713
(302) 999-4592

Automotive

DuPont Engineering Polymers
Automotive Products
950 Stephenson Highway
Troy, MI 48007-7013
(248) 583-8000

Asia Pacific

DuPont Asia Pacific Ltd.
P.O. Box TST 98851
Tsim Sha Tsui
Kowloon, Hong Kong
852-3-734-5345

Canada

DuPont Canada, Inc.
DuPont Engineering Polymers
P.O. Box 2200
Streetsville, Mississauga
Ontario, Canada L5M 2H3
(905) 821-5953

Europe

DuPont de Nemours Int'l S.A.
2, chemin du Pavillon
P.O. Box 50
CH-1218 Le Grand-Saconnex
Geneva, Switzerland
Tel.: ##41 22 7175111
Telefax: ##41 22 7175200

Japan

DuPont Kabushiki Kaisha
Arco Tower
8-1, Shimomeguro 1-chome
Meguro-ku, Tokyo 153
Japan
(011) 81-3-5434-6100

Mexico

DuPont S.A. de C.V.
Homero 206
Col. Chapultepec Morales
11570 Mexico D.F.
(011 525) 722-1456

South America

DuPont America do Sul
Al. Itapecuru, 506
Alphaville—CEP: 06454-080
Barueri—São Paulo, Brasil
Tel.: (055-11) 7266-8531/8647
Fax: (055-11) 7266-8513
Telex: (055-11) 71414 PONT BR

DuPont Argentina S.A.
Avda.Mitre y Calle 5
(1884) Berazategui-Bs.As.
Tel.: (541) 319-4484/85/86
Fax: (541) 319-4417

Printed in U.S.A.
[Replaces: H-70214 (96.10)]
Reorder No.: H-81091 (00.1)

DuPont Engineering Polymers