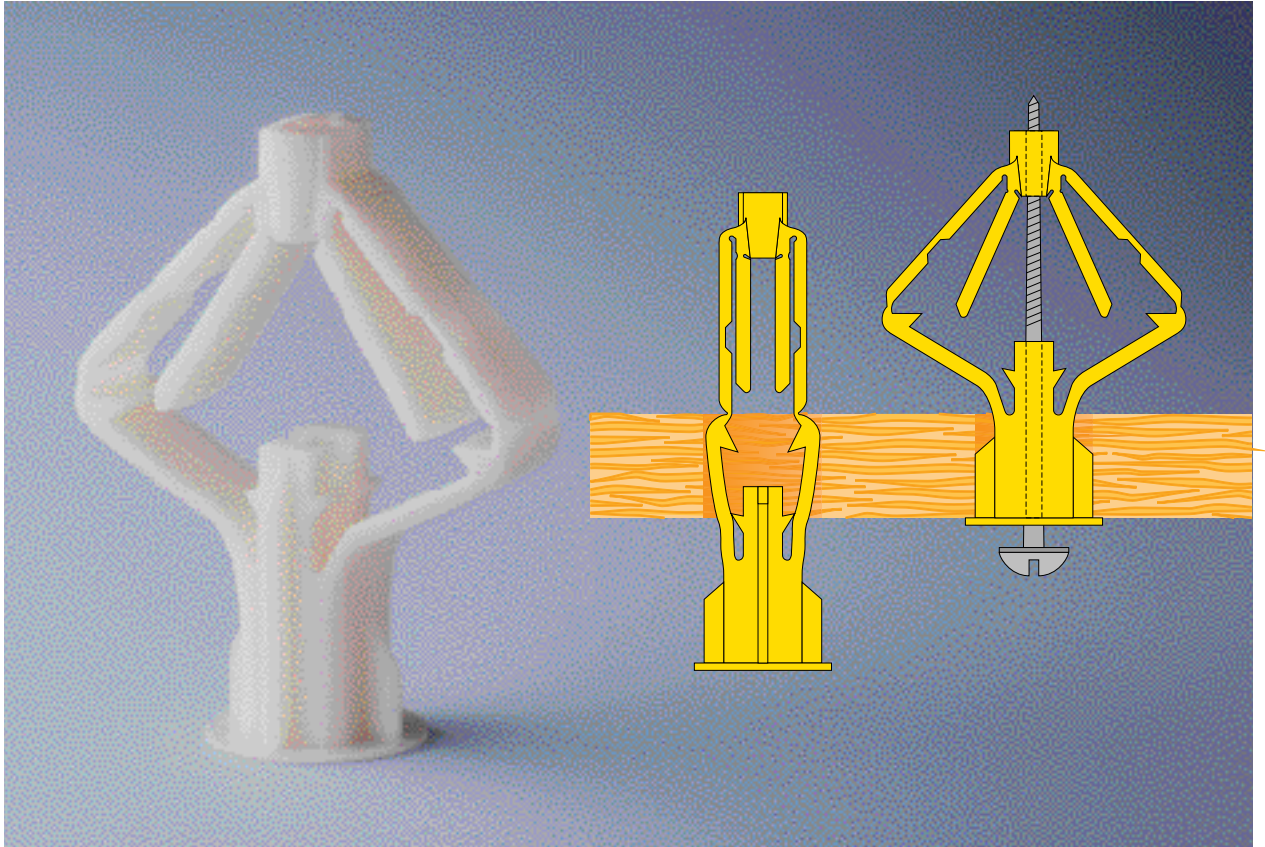




# Hytrel®

Engineering Thermoplastic Elastomer

## Injection moulding guide



Start  
with DuPont  
Engineering Polymers

## Table of contents

Injection moulding of HYTREL® .....	2
Handling precautions .....	2
General information .....	3
Product description .....	3
Product line.....	3
Melt properties .....	3
Thermal properties.....	6
Material handling.....	7
Drying.....	7
Purging .....	8
Regrind.....	8
Moulding equipment.....	8
Screw design.....	8
Nozzle design .....	8
Moulding conditions .....	9
Melt temperature .....	9
Cylinder temperature profile .....	9
Nozzle temperature .....	9
Mould temperature .....	10
Injection speed.....	10
Injection and holding pressure.....	10
Screw forward time (SFT).....	11
Screw rotation speed/back pressure.....	11
Cooling time .....	11
Mould design .....	11
Materials of construction .....	11
Mould surface finish.....	11
Sprue bushing design.....	11
Runners.....	11
Gates.....	11
Venting.....	12
Undercuts.....	12
Part ejection.....	12
Shrinkage and post moulding shrinkage .....	13
Post moulding shrinkage .....	14
Injection moulding trouble shooting guide .....	14

## Injection moulding of HYTREL®

The various grades of HYTREL® engineering thermoplastic elastomer exhibit a wide range of properties and an easy processability. Parts made of HYTREL® can be produced on standard injection moulding, extrusion, blow moulding, casting and rotational moulding equipment.

Film manufacturing techniques include casting, extrusion, and blown film. Post processing operations may include thermoforming, heat setting, bonding, welding, machining or painting.

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

## Handling precautions

All safety practices normally followed in the handling and processing of thermoplastic polymers should be followed for HYTREL® engineering thermoplastic elastomer. The polymer is not hazardous under normal shipping and storage conditions. During processing, particularly if recommended temperatures and holdup times are exceeded to any great degree, HYTREL® may degrade and decompose with evolution of gaseous products. Potential hazards from these gaseous decomposition products include “blow-back” through the hopper, fire and exposure to toxic vapours (principally tetrahydrofuran).

As with all thermoplastics, thermal burns from contact with molten polymer are a potential hazard. Before processing HYTREL® observe the precautions recommended.

Compounding ingredients or additives may present hazards in handling and use. Before proceeding with any compounding or processing work, consult and follow label directions and handling precautions from suppliers of all ingredients.

## General information

HYTREL® engineering thermoplastic elastomer offers an unique combination of mechanical, physical and chemical properties that qualify them for demanding applications.

### Product description

HYTREL® engineering thermoplastic elastomer is available in pellet form, packaged in 25 kg multi-wall paper bags with a moisture-barrier inner wall. Selected grades are also available in 500 kg corrugated paper board boxes with a moisture barrier loose liner. The 3 mm diameter pellets flow well in hoppers and material handling equipment.

Property data sheets on currently available grades can be obtained through your local sales office listed at the end of this bulletin or your sales representative.

### Product line

#### Grades

##### Standard

G3548 L	} Best balance of cost and performance in a wide range of hardnesses.
G4074	
G4774	
G5544	
6358	
7248	
8238	

##### High performance

4056	} These provide an extra measure of strength and service life to meet the needs of the most demanding applications.
4069	
4556	
5526	
6356	
7246	

##### Specialty

G4078 W	Standard grade. Containing improved colour stable antioxydant
5555 HS	Offers the highest heat ageing resistance
6359 FG	Food grade

##### Concentrates

10 MS	Hydrolytic stabilizer
20 UV	UV stabilizer for colours other than black
30 HS	Heat stabilizer
41 CB	Carbon black concentrate
51 FR and 52 FR	Flame retardant concentrate

The following table shows several attributes of the product range that should be considered in injection moulding. Certain grades depending on typical composition may however not fall exactly into these generalizations.

Table 1 Characteristics of HYTREL®

	Soft grades 30-47 Shore D	Hard grades 55-82 Shore D
Crystallinity	–	+
Melt viscosity	+	•
Shrinkage	–	•
Chemical resistance	+	++
Thermal stability	+	++
Wide processing window	+	++
Melt temperature	•	+
Mould 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 and compared to several other thermoplastic resins are shown in Figure 1.

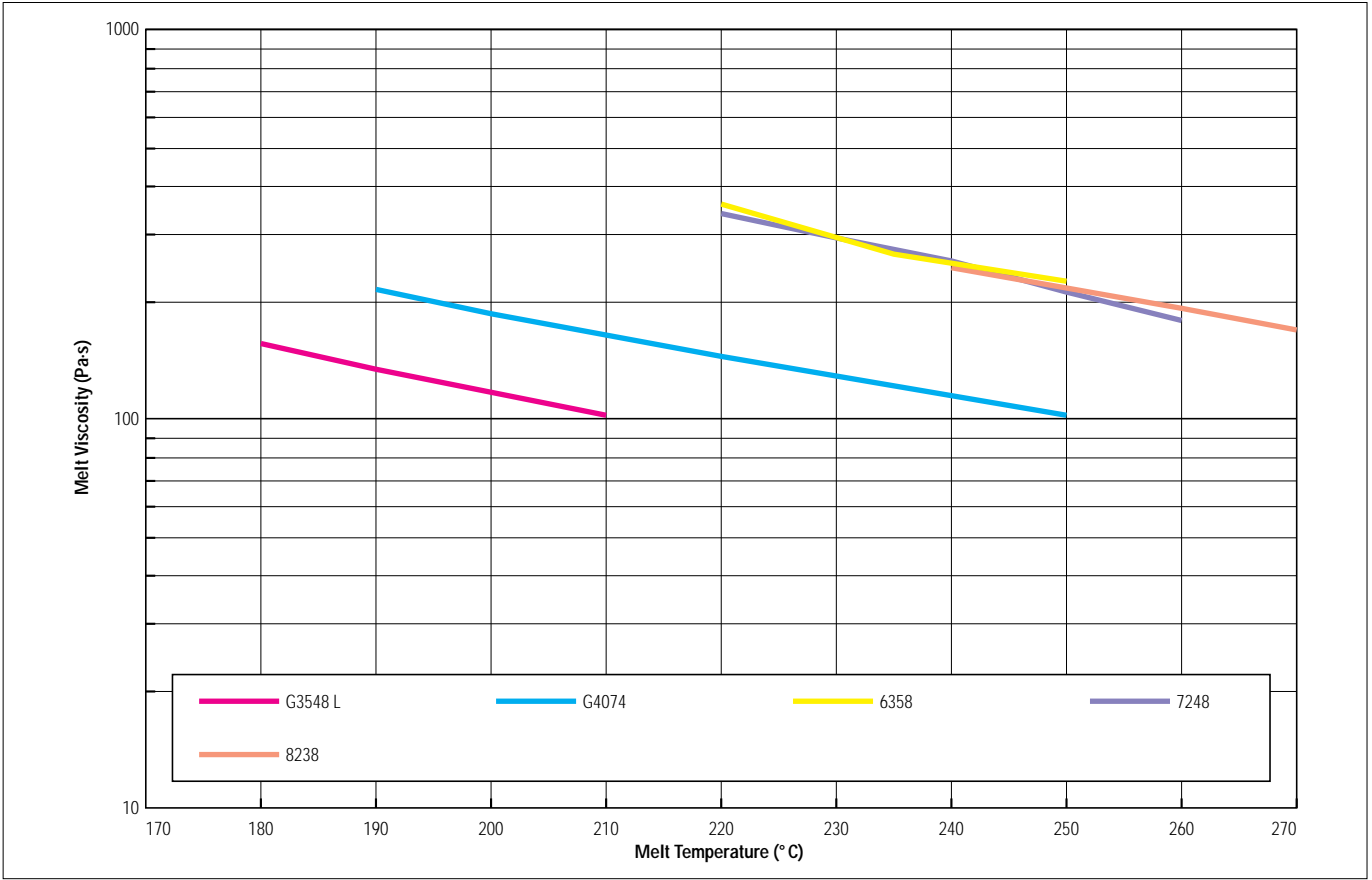


Fig. 1a Apparent melt viscosity vs. temperature. Standard grades at Shear Rate of 1000 s<sup>-1</sup>

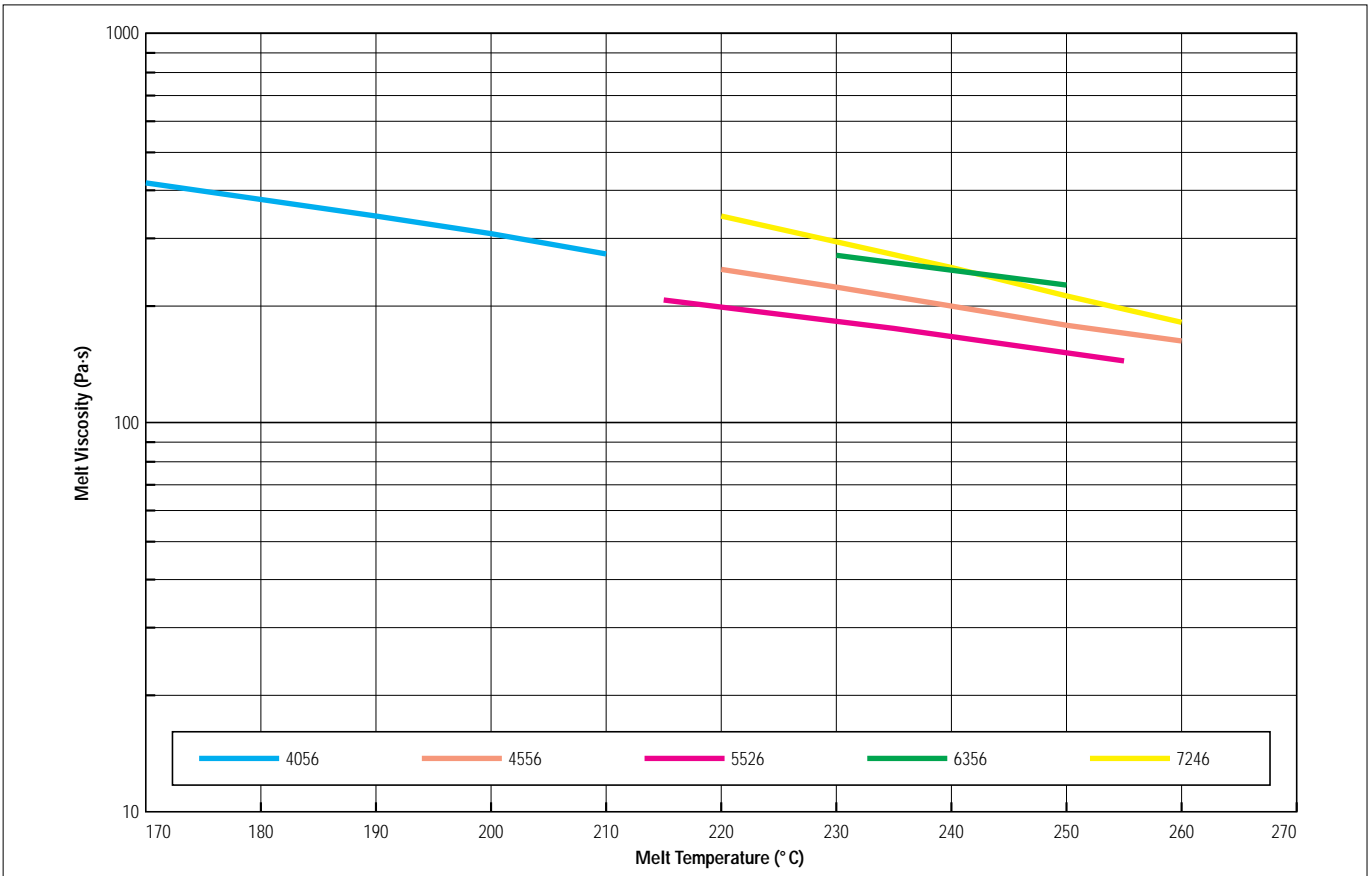


Fig. 1b Apparent melt viscosity vs. temperature. High performance grades at Shear Rate of 1000 s<sup>-1</sup>

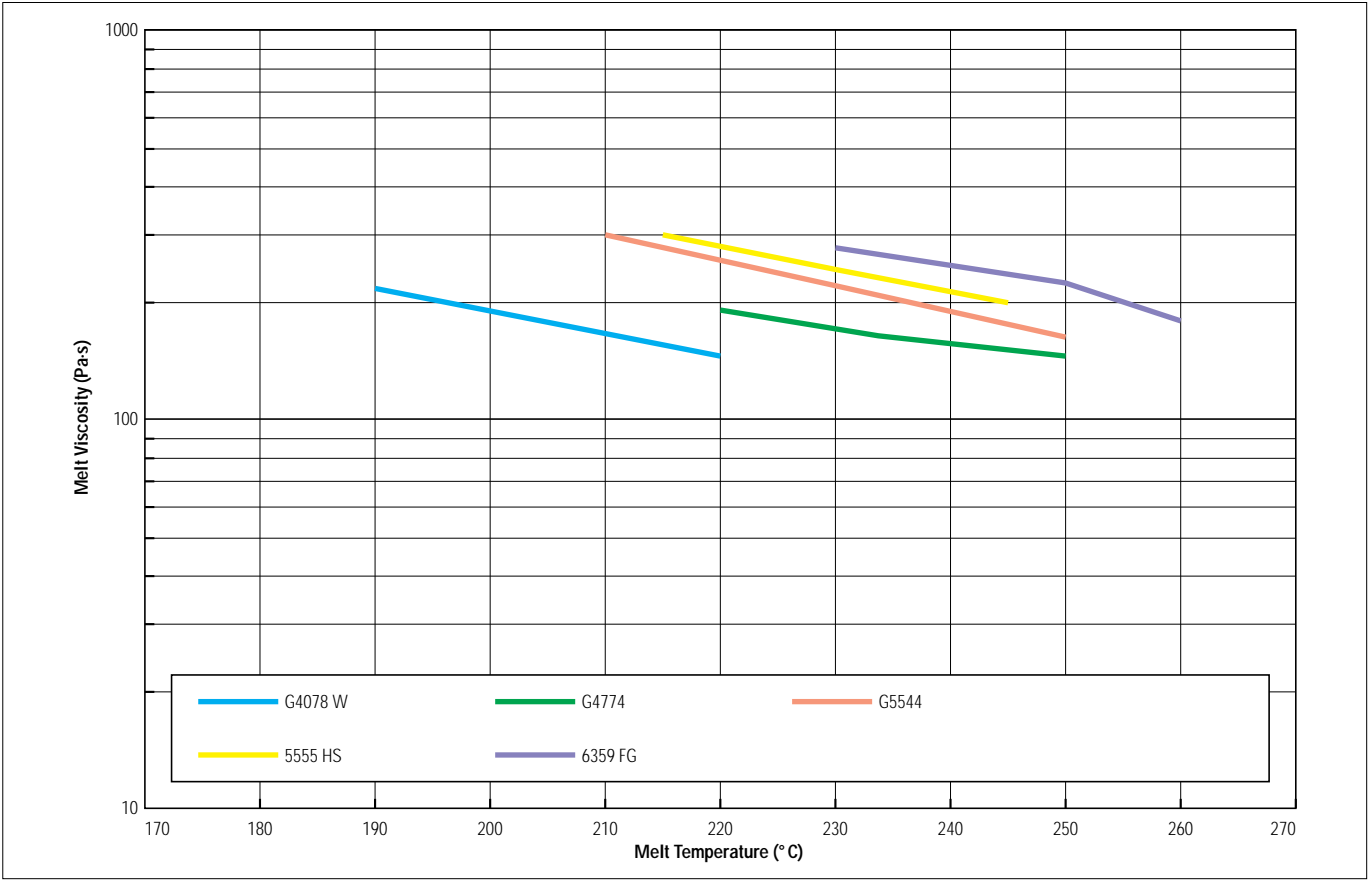


Fig. 1c Apparent melt viscosity vs. temperature. Specialty grades at Shear Rate of  $1000 \text{ s}^{-1}$

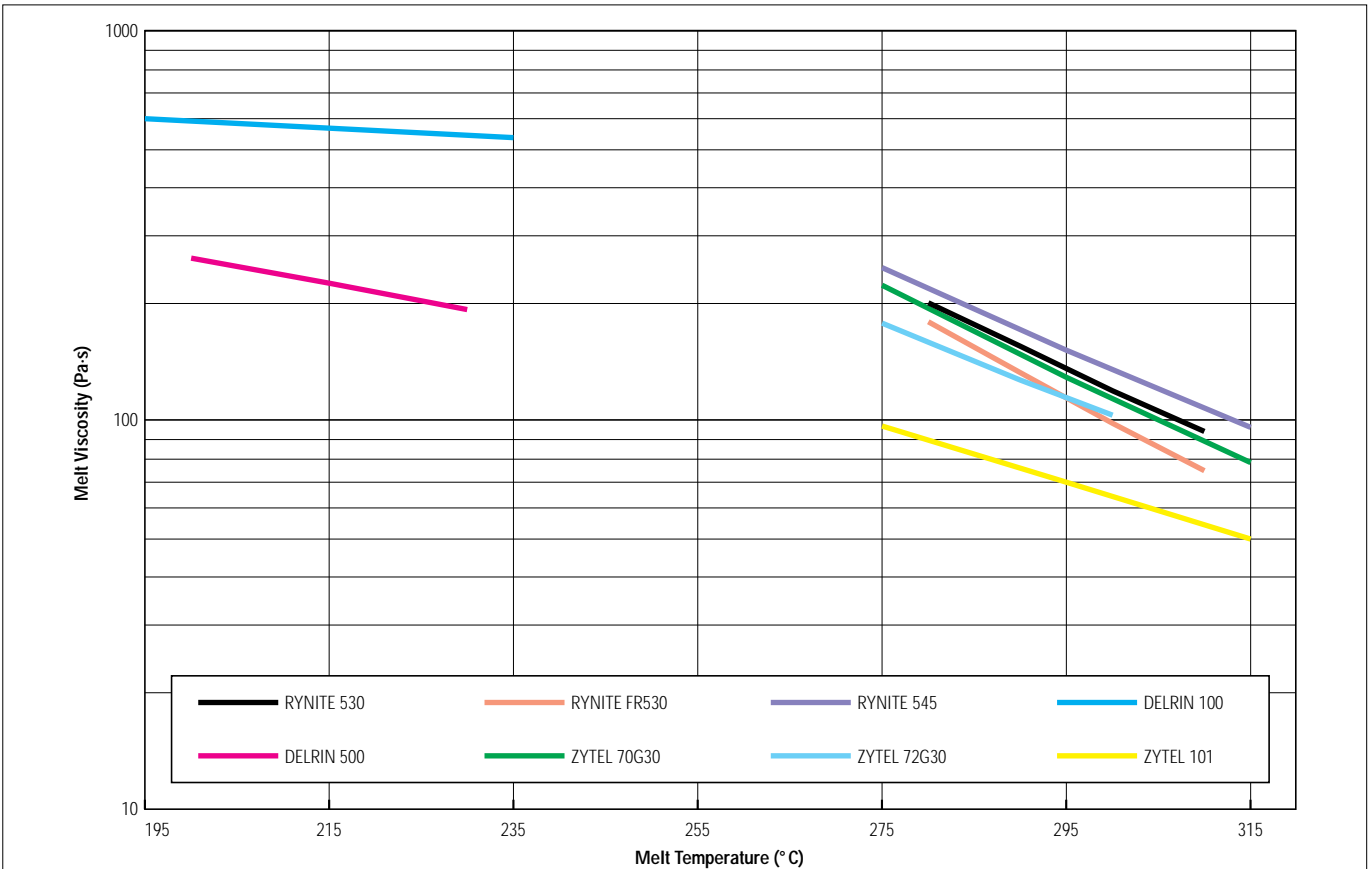


Fig. 1d Apparent melt viscosity vs. temperature. Other engineering polymers at Shear Rate of  $1000 \text{ s}^{-1}$

## Thermal properties

When handled properly, HYTREL® engineering thermoplastic elastomer has an outstanding thermal stability. In the melt under normal operating conditions, no gaseous by products are evolved. This thermal stability combined with a chemically pure polymer with no plasticizers and little additives minimizes problems such as change of viscosity with hold-up time in the injection unit cylinder or formation of black specks.

The thermal stability of HYTREL® engineering thermoplastic elastomer allows a higher flexibility during processing. Even after a short machine stoppage (10-15 minutes), production can be resumed without purging and still achieve acceptable parts.

Typical melt temperatures and some thermal properties of the various grades are shown in Table 2.

Table 2: **Thermal properties of HYTREL® engineering thermoplastic elastomer**

Grade	Tm (°C)	Tc (°C)	Tg (°C)	Hf (J/g)
<b>Standard</b>				
G3548 L	156	107	-24	8
G4074	173	120	-37	17
G4774	208	170	-17	27
G5544	215	173	-34	33
6358	213	155	0	31
7248	219	170	+25	35
8238	223	170		37
<b>High performance</b>				
4056	148	70	-32	12
4069	195	112	-51	14
4556	193	115		24
5526	202	147	-18	26
6356	213	155	0	31
7246	219	170	+25	35
<b>Specialty</b>				
G4078 W	173	120	-38	17
5555 HS	202	166	-18	26
6359 FG	213	155	2	31

Tm: Melting point  
 Tc: Crystallization point  
 Tg: Glass transition point  
 Hf: Heat of fusion

The thermal stability of these polymers permits exposure at melt temperatures for prolonged periods with minimum degradation. Figure 2 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.

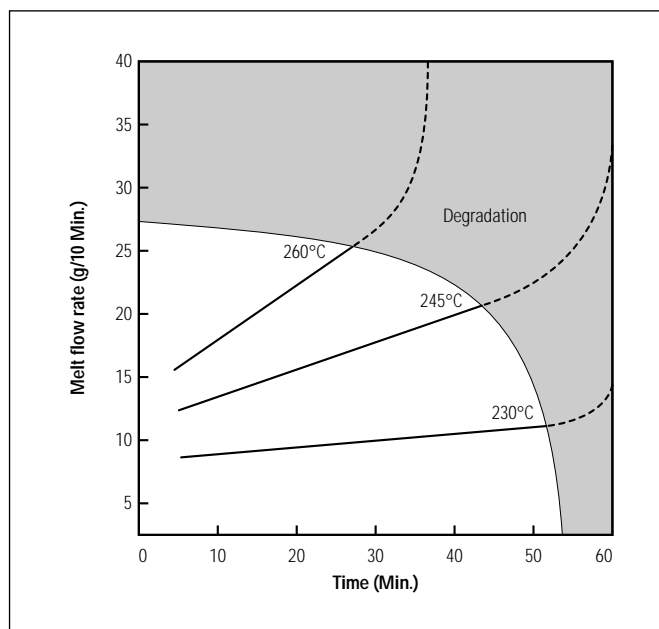


Fig. 2 **Degradation**

# Material Handling

## Drying

HYTREL® engineering thermoplastic elastomer can be used directly from undamaged sealed bags and may not need to be dried prior to moulding. As manufactured, these polymers are dried to a moisture content below 0,1% and packaged in special moisture resistant bags.

However, since there is a possibility of damaged or open bags, and in any case when using regrind, a desiccant hopper dryer should be used to insure mouldings of high quality. This will also protect against moisture pickup during processing. Dehumidified hopper dryers reduce and control the resin moisture content and improve quality.

HYTREL® engineering thermoplastic elastomer is hygroscopic (as are all polyesters) and if left exposed will absorb moisture from the air. At temperatures substantially above the melting point, excess water (more than 0,1% for all grades) causes hydrolytic degradation of the polymer. Such degradation does not appear as visual defects, but results in poor physical properties, brittleness and poor in-service performances particularly at low temperatures.

At normal processing conditions, little degradation of the polymer occurs if the moisture content is below 0,1%. When dry polymer from open bags or from the injection machine's hopper is exposed to 50% relative humidity air, an increase of 0,1% moisture will occur in about 2 hours, while at 100% relative humidity it occurs in less than an hour.

Therefore, granules so exposed should be redried before use. Figure 3 shows the water absorption rate for HYTREL® 5556. This pickup rate will depend on the moisture equilibrium level of each grade as shown in Table 3. The higher the equilibrium level, the faster the pickup rate.

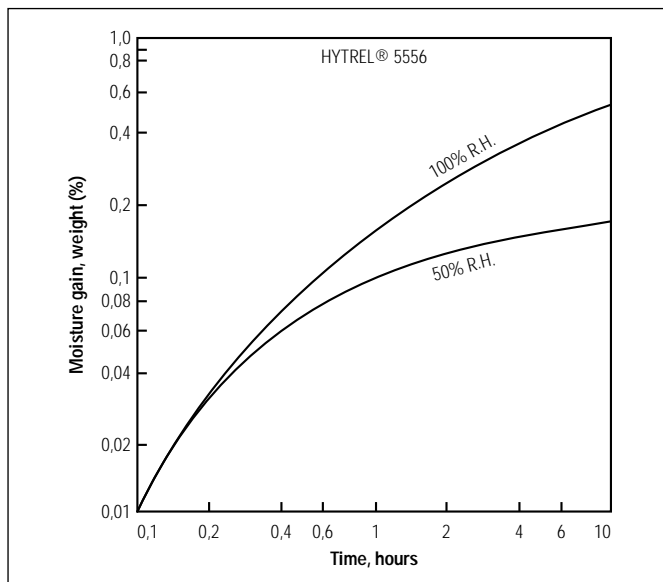


Fig. 3 Moisture absorption at ambient temperature

Table 3: Water absorption at 23° C after 24 hours immersion.

Grade	Equilibrium moisture level (% after 24 hrs)
<b>Standard</b>	
G3548 L	3,6
G4074	2,5
G4774	2,5
G5544	1,5
6358	0,3
7248	0,3
8238	0,3
<b>High performance</b>	
4056	0,6
4069	0,7
4556	0,7
5526	0,5
6356	0,3
7246	0,3
<b>Specialty</b>	
G4078 W	2,5
6359 FG	0,3

Table 4 and Figure 4 show recommended drying conditions for HYTREL®.

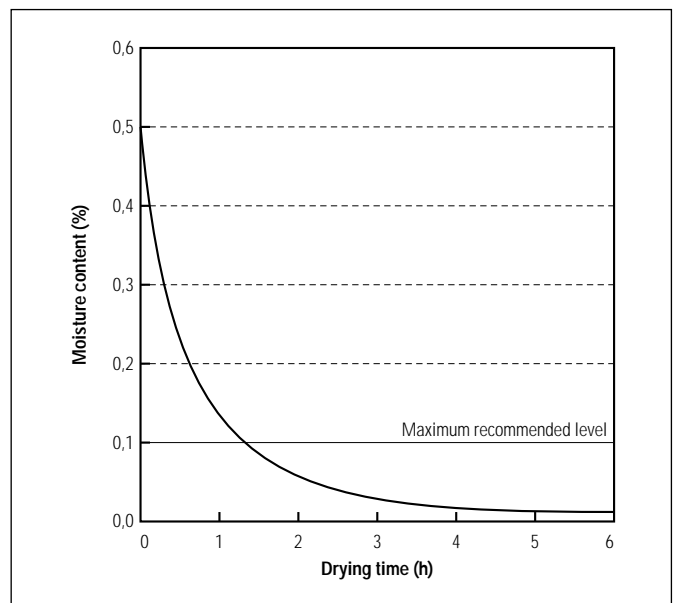


Fig. 4 Drying guidelines with dehumidified hopper dryer at 110° C. Dew point - 30° C

Table 4 Drying conditions for HYTREL®

	Drying temp.	Drying time
Dehumidified Hopper	110° C	2-3 hours
Air circulating oven	110° C	4-6hours (in dry weather)

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

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® engineering thermoplastic 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.

### Regrind

Regrind can be used to a level of 50% 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.
- All regrind needs to be dried before moulding.

### Recommended usage levels for regrind of HYTREL®

Type of HYTREL®	Melt flow rate <sup>1</sup> (g/10 min.)	Max. regrind usage (%)	Max. allowable melt flow rate of regrind (g/10 min.)
4056	5,3 at 190° C	25	10
		50	08
5526	18 at 220° C	25	31
		50	25
5556	7 at 220° C	25	15
		50	11
6356/58	8,5 at 230° C	25	14
		50	11
5555 HS	8,5 at 220° C	25	15
		50	12
7246/48	12,5 at 240° C	25	21
		50	17
8238	12 at 240° C	25	21
		50	17

1) ASTM Method 1238, 2,16 kg load.

## Moulding equipment

HYTREL® engineering thermoplastic elastomer can be moulded on standard injection moulding machines. Even when HYTREL® degrades 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 jamming of the elastomeric pellets, screw compression ratio should not exceed 2,5:1 to 3,0:1 and the metering zone should be relatively deep, from 2,5 to 3,0 mm for a 60 mm screw. For a more uniform polymer melt and mixing, screw L/D (length to diameter) should be at least 20:1.

### Nozzle design

Standard open nozzles as shown in Figure 5 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 semi-crystalline thermoplastics, nozzle diameter (and runner system) should be dimensioned somewhat larger.

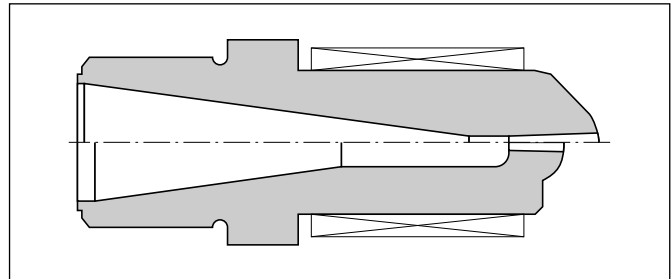


Fig. 5 Open nozzles recommended for moulding

Figure 6 shows the recommended back-flow valve design.

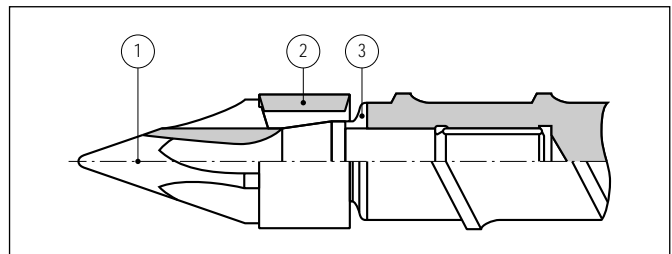


Fig. 6 Back flow valve

## Moulding conditions

### Melt temperature

The melt temperature is 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 5.

Because HYTREL® has a good thermal stability, melt temperature can be increased up to 20° C (see Figure 8) 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 7 can be used to define the optimum cylinder temperature profile.

### 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 5). To prevent drooling in certain cases, the use of pressure relief (suck-back) is recommended.

Table 5 Recommended melt temperatures for HYTREL®

Grade	Melt temperature
Standard	
G3548 L	190 ± 10° C
G4074	200
G4774	230
G5544	240
6358	240
7248	245
8238	250
High performance	
4056	180 ± 10° C
4069	230
4556	230
5526	230
6356	240
7246	245
Specialty	
G4078 W	200 ± 10° C
5555 HS	230
6359 FG	240

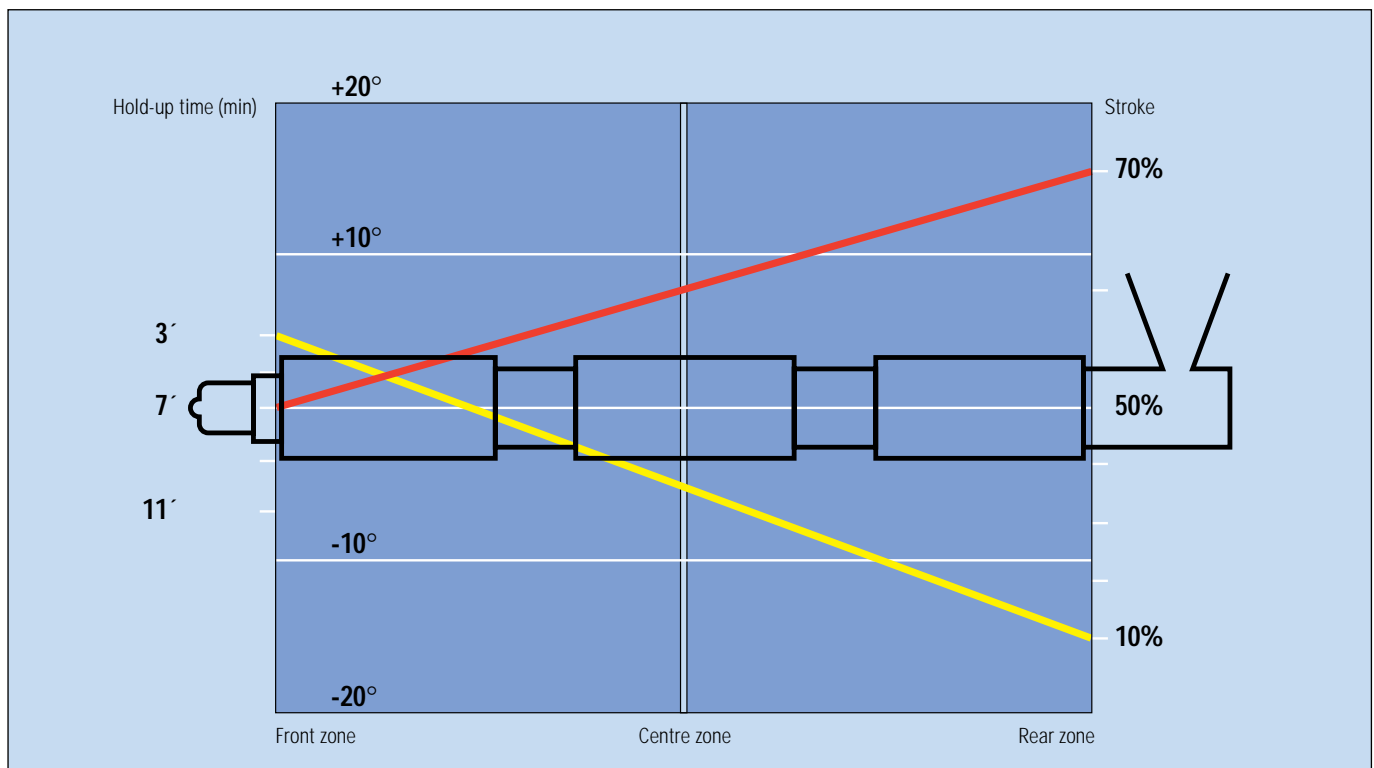


Fig. 7 Cylinder temperature profile for a constant melt temperature

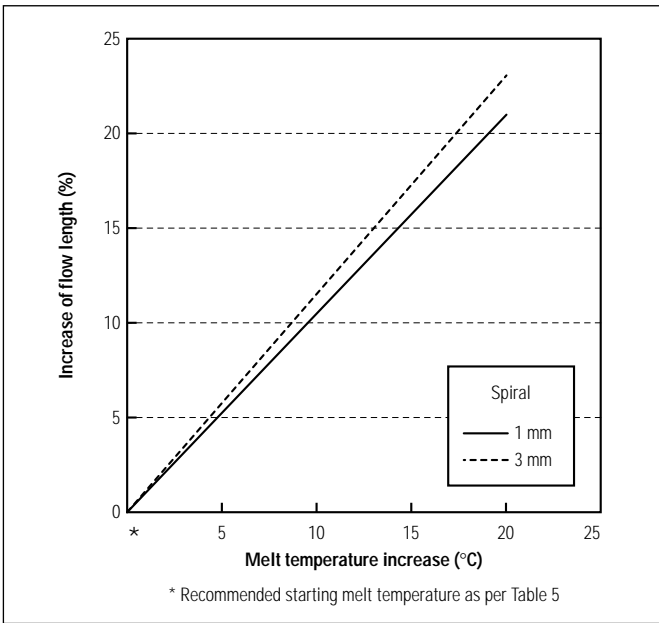


Fig. 8 Influence of melt temperature on flow length

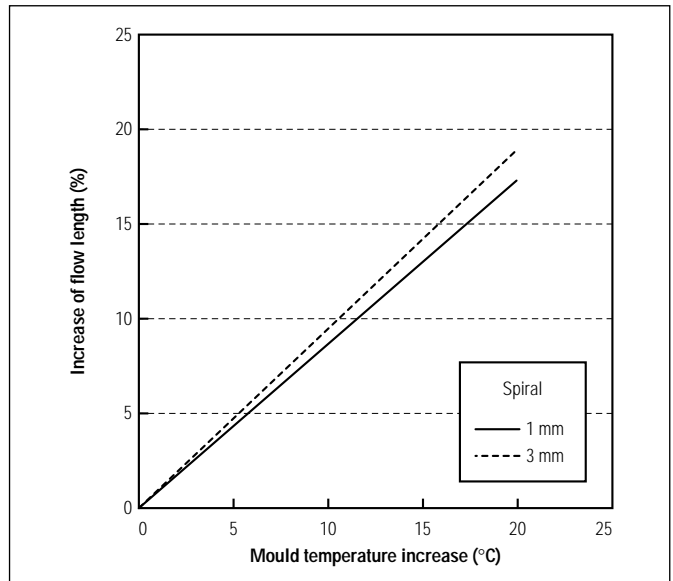


Fig. 9 Influence of mould temperature on flow length (at recommended melt temp.)

### Mould temperature

Mould temperature is measured with a thermocouple directly on the cavity's surface.

Recommended mould temperature for all grades is 45°C. Mould temperature has little effect on mechanical properties. The main effect is on shrinkage (see page 13).

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

Higher mould temperatures will improve surface appearance.

### 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 mould cavity especially with the softer grades.

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

Figure 10 shows the flow length of various grades and Figure 11 the effect of injection pressure.

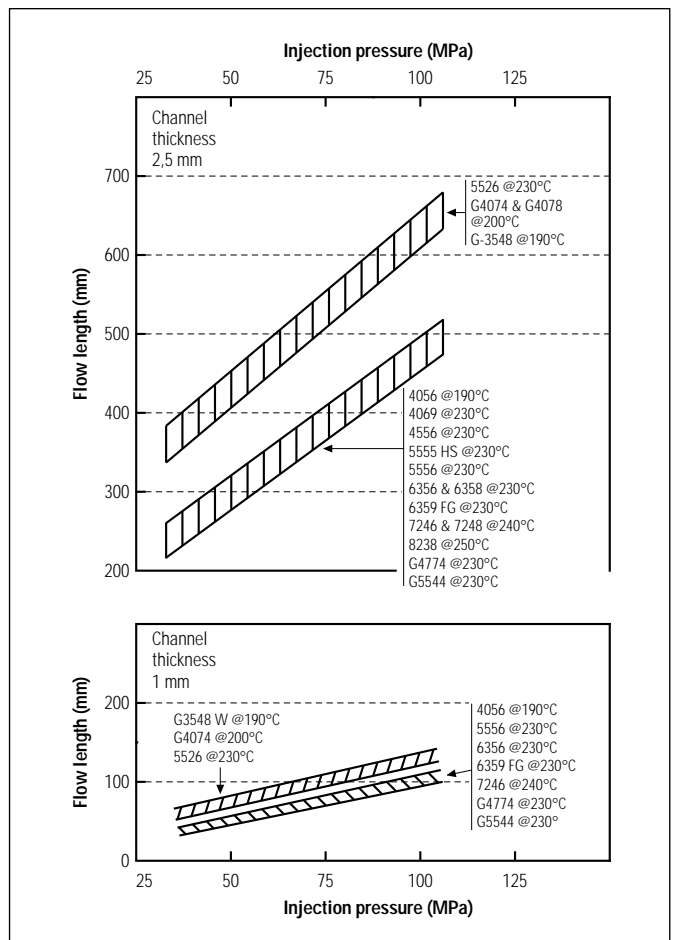


Fig.10 Snake Flow at processing temperature

### 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:

72 D – 82 D	4-5 s/mm (for parts up to 4 mm thick)
55 D – 63 D	5-6 s/mm
35 D – 47 D	7-8 s/mm

The screw forward time has a strong influence on shrinkage.

### Screw rotation speed/back pressure

To avoid shearing of the polymer, the screw peripheral speed should not exceed 0,2 m/s. Some back pressure (0,2 to 1 MPa) can be used to improve melt homogeneity. If additives (e.g. colour concentrates) are being mixed in, lower screw speed and higher back pressure may be required to obtain adequate mixing.

### Cooling time

For the harder grades, cooling time is generally not necessary and should be set 1 or 2 seconds longer than the plastification time. For the softer grades, cooling time should be set as low as practical for smooth ejection.

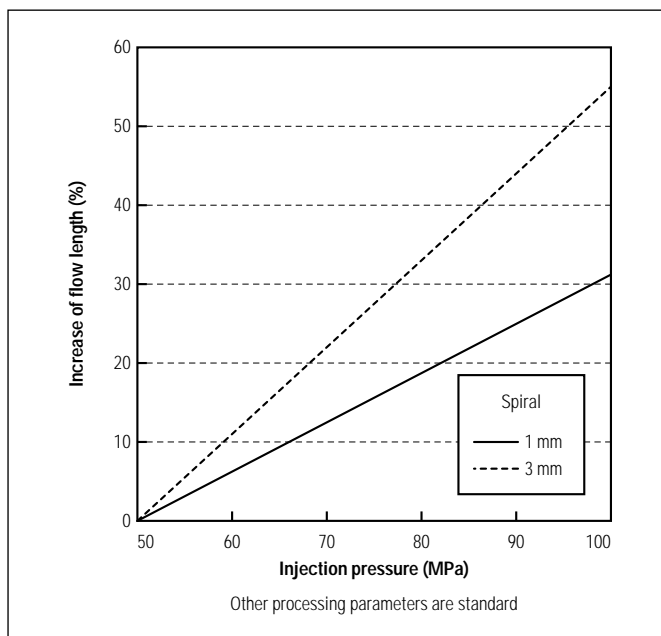


Fig. 11 Influence of injection pressure on flow length (at recommended melt temp.)

## Mould design

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

### Materials of construction

No special metals are required since HYTREL® has no corrosive action on the alloys commonly used for injection moulds and cavities.

### Mould surface finish

Textured and mat finished cavity surfaces minimize the effects of flow lines or marks and scratches on the part. Highly polished, plated mould 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°, but larger tapers result in less sprue sticking.

A properly mated injection nozzle and sprue bushing facilitates ejection of the sprue. The nozzle diameter should be 0,5-1 mm 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 moulding, both insulated and hot runner, may also be used. Sufficient heating capacity and control must be provided to insure 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 12) are recommended in order to minimize flow lines and distortion at the gate. For thick section mouldings, sprue gates are usually required to eliminate sinks.

Tunnel gates (see Figure 12) as small as 0,5 mm can be used. The land length should be kept as short as possible (0,5-1 mm) 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 a 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 thick, the gate should have the same thickness as the part. Gate lands should be between 0,5-1 mm.

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 mould should be broad but thin. Vent openings up to 6 mm wide should not be deeper than 0,04 mm 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 mould trials, so frequently vents must be added after moulds are released for production.

### Undercuts

The depth of undercut that can be stripped from a mould will vary with the size and shape of the part, overall cycle, mould temperature and the grade of HYTREL® used.

Undercuts should be radiused generously to aid ejection and should be no more than 0,8 mm deep. Placing the undercut near the ejection or stripper plate helps to avoid distortion of the part on demoulding.

### Part ejection

Ample draft, 0,5° to 2° 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 mould 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 mould.

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 diameter pins if design permits.

Undercuts should have room to flex during ejection.

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

With these grades, a slip or antiblock agent (a fatty amide dusted on the pellets at a rate of 0,2-0,3%) may be beneficial in injection of parts.

Nucleating agents such as talc dusted onto pellets at a 0,5% level increase the crystallization rate by about 20%.

A dry fluorocarbon type release can be used to aid part ejection. Non silicon type release agents are recommended if parts are to be painted.

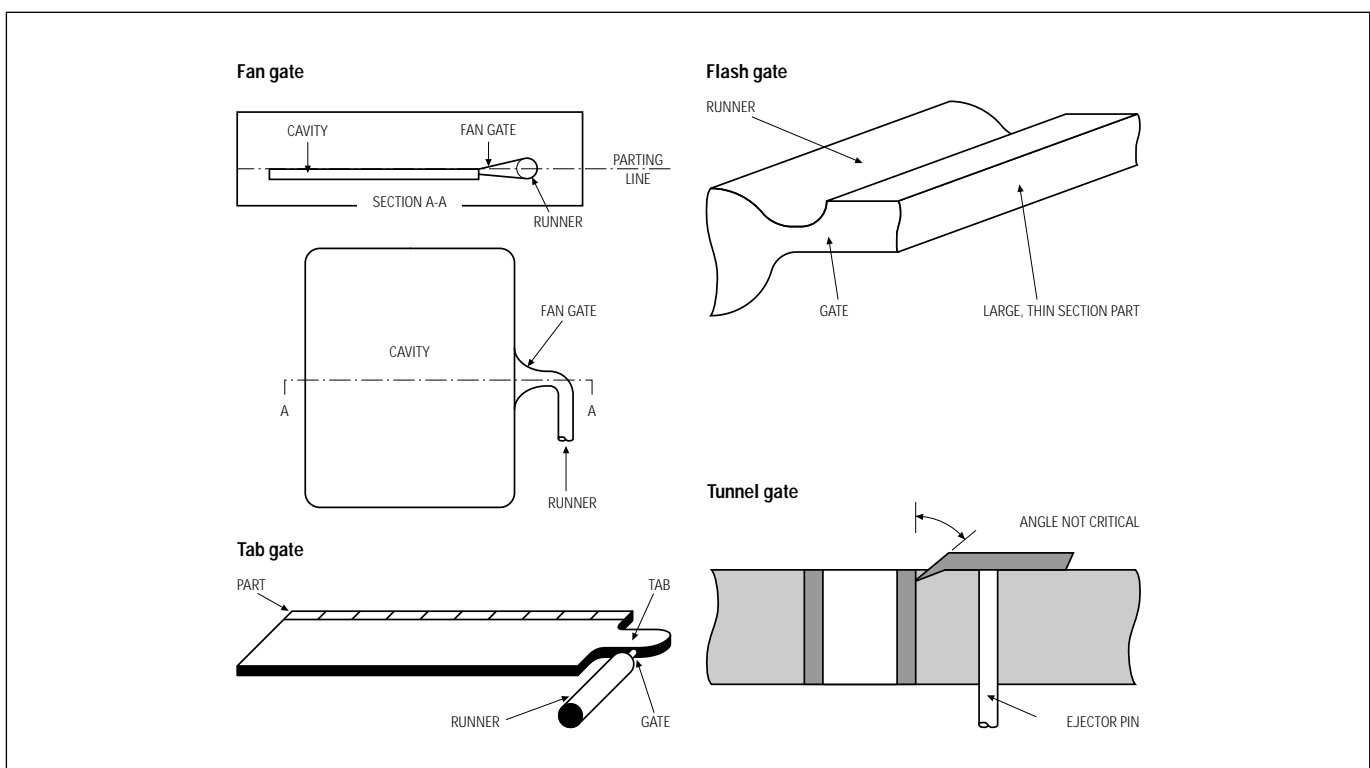


Fig. 12 Gate Design

## Shrinkage and post moulding shrinkage

The shrinkage of HYTREL® engineering thermoplastic elastomer in injection moulding depends on numerous factors such as:

- Grade of HYTREL®
- Moulding conditions (injection pressure, SFT, mould temperature...)
- Part geometry and thickness

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

This chapter 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 thickness moulded at standard conditions:

**Mould temperature: 45° C**

**Melt temperature: as recommended in Table 5**

**Injection pressure: 70 MPa**

**SFT: optimum**

Table 6 gives the nominal shrinkage values for various grades, obtained under these standard conditions.

The following figures show the influences on shrinkage of different injection moulding 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 or subtracted to the nominal shrinkages given in Table 6 in order to get a first approximation of the final shrinkage.

Tableau 6 **Shrinkage of HYTREL®**  
(measured on standard test specimen,  
moulded at recommended conditions)

Grade	Shrinkage (%)
Standard	
G3548 L	0,8
G4074	0,8
G4774	1,6
G5544	1,7
6358	1,6
7248	1,7
8238	1,8
High performance	
4056	0,2
4069	0,8
4556	1,1
5526	1,1
5556	1,4
6356	1,6
7246	1,7
Specialty	
G4078 W	0,9
5555 HS	1,3
6359 FG	1,6

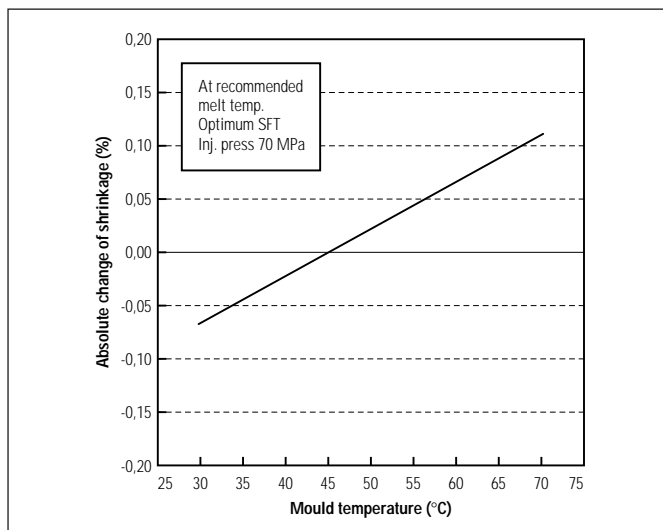


Fig. 13 Influence of mould temperature on shrinkage

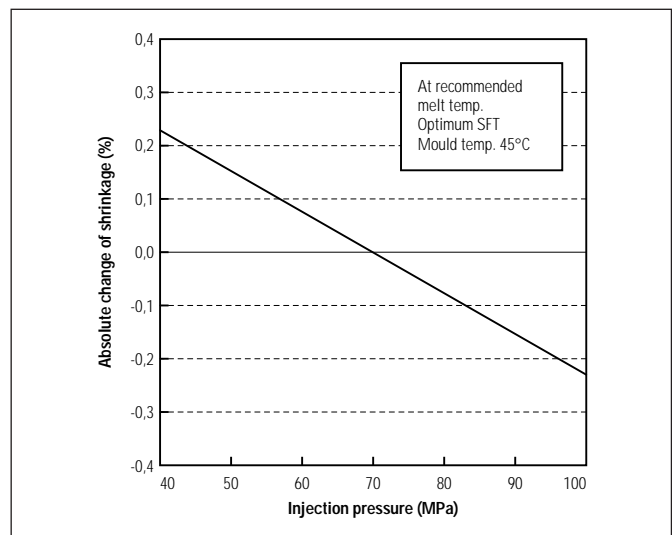


Fig. 14 Influence of injection pressure on shrinkage

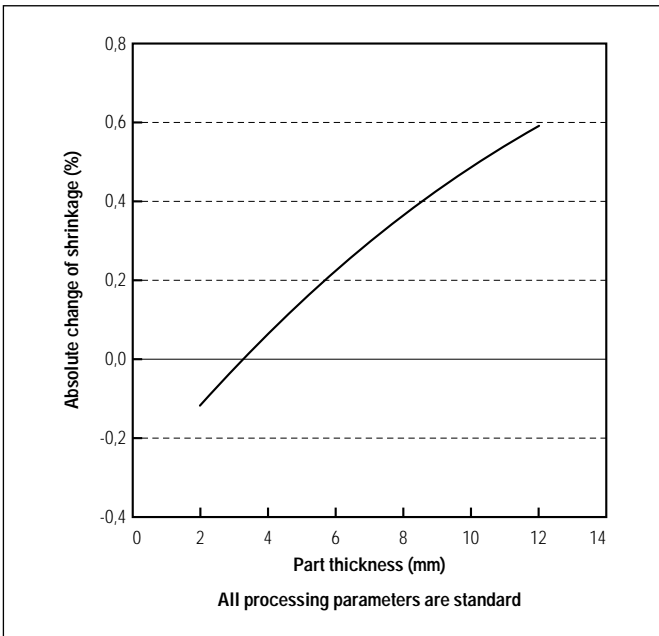


Fig. 15 Influence of part thickness on shrinkage

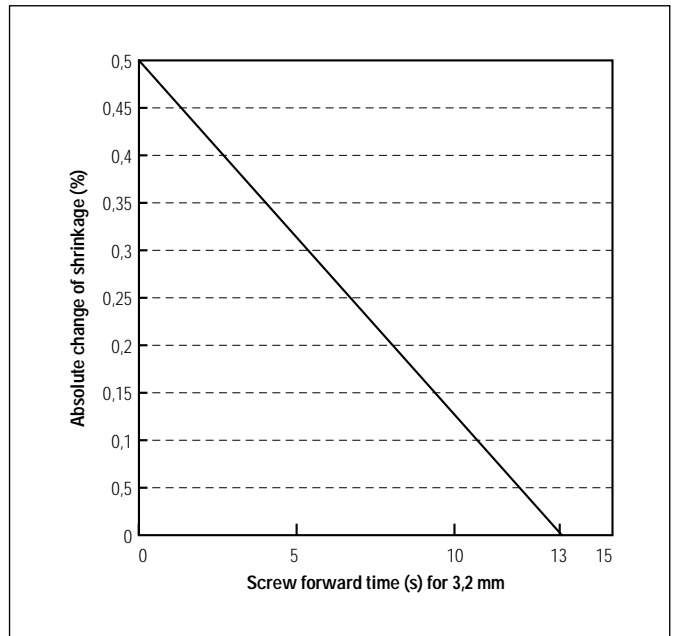


Fig. 16 Influence of screw forward time on shrinkage for 55 to 80 shore D grades

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

Nominal shrinkage of HYTREL® 5526:	1,10% (Table 6)
Part is moulded using a 65° C mould temp. (vs. 45° C):	+ 0,08% (Table 13)
Part is moulded using an injection pressure of 90 MPa vs. 70:	- 0,15% (Table 14)
Part has a thickness of 2 mm (vs. 3,2):	- 0,13% (Table 15)
The <b>approximation</b> of the total shrinkage is:	<b>0,90%</b>

It is strongly recommended that a prototype test cavity be user under the expected production moulding conditions to determine actual part shrinkage.

We recommended to apply a holding pressure to avoid sink marks (see Screw Forward Time chapter) and the shrinkages values are given for an optimum SFT.

It may happen that the Screw forward time is not effective in a part area located far from the gate. For this reason, the Fig. 16 show the influence of the SFT on the absolute change of shrinkage.

### Post moulding shrinkage

Post moulding shrinkage is measured after annealing parts at 120° C for 4 hours. Even for the stiffer and more crystalline grades, the absolute value of post moulding shrinkage for parts moulded at recommended conditions is low, less than 0,1%.

### Injection moulding trouble shooting guide

This section identifies various problems which might be experienced during the injection moulding of HYTREL® engineering thermoplastic elastomers. 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 inter related. 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 at the end of this bulletin.

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

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
<b>III. Short shots – after a period of successful injection moulding operations.</b>	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 moulded 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.
	D. Polymer is sticking in the feed throat.	Increase cooling of the feed throat. Reduce temperature of the rear zone.
<b>IV. Short shots – occur periodically during injection moulding operations.</b>	A. Cylinder temperature controller is cycling too broadly.	Consult technical service representatives for the temperature controllers used on the injection moulding machines.
	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.
<b>V. Flashing.</b>	A. Injection pressure is too high.	Reduce injection pressure.
	B. Too much material is being injected into the mould.	Reduce shot size or run without a pad, Note Section VII, Item A. Reduce pressure to pack out.
	C. Material is too hot.	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 mould.	Inspect land areas, etc. of the mould carefully and clean where necessary.

<b>PROBLEM</b>	<b>POSSIBLE CAUSE</b>	<b>SUGGESTED SOLUTION</b>
<b>V. Flashing.</b> <i>(cont.)</i>	F. Mould surfaces, cores and/or cavity inserts are out of register.	Remove mould, overhaul and correct the register.
	G. Mould or platens are warped.	Check and overhaul if necessary.
	H. Clearance in vents, knockouts, etc. is too great.	Check clearance and adjust as necessary. Clearance should not be more than 0,038 mm.
	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 in depth.
	J. Injection pressure is unevenly distributed in the mould.	Cavity and runner layout should be balanced.
<b>VI. Ejection difficulties.</b>	K. Projected cavity area is too large for the available clamping pressure.	Shift to a press with greater available clamping pressure. Reduce the number of cavities.
	A. Excessive flashing.	See suggested solutions for Section V.
	B. Material too highly packed in the cavity (mainly with large gates).	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 mould. 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 vapour-hone mould core and core pins in the direction of ejection.
	D. Parts stick to the mould due to highly polished surfaces.	Check suggestions in Section VI, Item C. Use internal or external mould release. Use matte finish on the mould cavity.
E. Mould conditions:		
1. Mould surfaces are scratched and marred.	Overhaul and polish the mould surfaces.	
2. Draft or taper on cavity walls, cores or sprues is not great enough.	A minimum of 0,017 rad [1°] 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.	

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
<b>VII. Warpage or part deformation.</b>	A. Moulded-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-homogeneous.	Increase temperatures of the cylinder and/or screw speed.
	B. Part is being ejected while still too hot.	Reduce temperature of the mould. 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 deep. Use ribs and bosses of minimum thickness.
G. Mould cooling is inadequate. (Capacity of the cooling system is too low, cooling circuits in the mould 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.	
H. Moveable mould components (cores) have shifted or become misaligned.	Realign.	
I. Runner system is inadequate.	Redesign.	

<b>PROBLEM</b>	<b>POSSIBLE CAUSE</b>	<b>SUGGESTED SOLUTION</b>
<b>VIII. Excessive shrinkage.</b>	A. Gates not frozen off.	Increase time injection ram is forward.
	B. Effective injection pressure in the cavities is too low. <ol style="list-style-type: none"> <li>1. Gates are too small or improperly designed.</li> <li>2. Runner system is improperly designed (diameters and layout are incorrect).</li> <li>3. Melt temperature is too low.</li> <li>4. Flow rate of material is too low.</li> <li>5. Nozzle orifice is too small.</li> </ol>	<p>Increase size of gates and/or shorten length of lands.</p> <p>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.</p> <p>Check actual temperature of the melt with a needle pyrometer. If necessary, increase temperatures of the cylinder.</p> <p>Use polymer with a higher melt index, if possible.</p> <p>Use nozzle with a larger orifice.</p>
	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. Mould temperature is too high.	Reduce temperature of the mould.
	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. Moulding conditions not optimized.	See "Shrinkage ".
	<b>IX. Sinks, shrink marks, voids, bubbles.</b>	A. With the exception of Item B-4, the causes shown for Section VIII generally apply.
B. Moisture content of the polymer is too high.		Dry the polymer.

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
<b>X. Burning, charring or black specks.</b>	A. Material is too hot.	Reduce temperatures of the cylinder. Shorten time of cycle.
	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 moulding 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 moulded 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 <sup>®</sup> until the machine is free of other polymers or remove the screw and nozzle and clean thoroughly.
<b>XI. Degradation.</b>	A. Material is overheated.	Reduce temperatures of the cylinder. Shorten time of cycle. (See Item G, following).
	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. Try virgin material.
	E. Improper shut down 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.

<b>PROBLEM</b>	<b>POSSIBLE CAUSE</b>	<b>SUGGESTED SOLUTION</b>
<b>XI. Degradation. (cont.)</b>	F. Moisture content of the polymer is too high.	Dry the regrind and polymer.
	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 noule at the desired melt temperature and all other temperatures as low as operable.
	H. Stagnation of material in the cylinder, noule, or nozzle valve.	Inspect the cylinder. Eliminate dead spots (streamline) as necessary.
<b>XII. Dimensional variations.</b>	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 moulding polymers of HYTREL <sup>®</sup> , it is suggested that the shot size not exceed 75% of the machine's plasticizing capacity.
	E. Mould temperatures inadequately controlled.	Check coolant for temperature variations. Install temperature controller if needed. Check location of coring.
<b>XIII. Surface defects on the moulded article.</b>	A. Mould lubricant used excessively.	Clean mould surfaces thoroughly. Use lubricant sparingly for 40D polymer or not at all (wipe on – don't spray on).
	B. Moisture on cavity surfaces.	Wipe mould surface thoroughly with a rag moistened with alcohol. Raise the mould temperature. Apply anti-condensation material to the outer surface of the mould base. Check for coolant leaks.

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
XIII. Surface defects on the moulded article. ( <i>cont.</i> )	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. Trappe dair.	Reduce temperature of <b>rear</b> cylinder.
	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 four hours 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 mould 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. Materials is too cold.	Raise temperatures of the cylinder and/or mould.	
3. Injection speed is too slow.	Increase speed of injection.	
4. Mould is too cold.	Increase temperature of mould.	
5. Flow rate of the material is too low.	Use polymer with a higher melt index, if possible.	
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.	

PROBLEM	POSSIBLE CAUSE	SUGGESTED SOLUTION
<b>XIII. Surface defects on the moulded article.</b> <i>(cont.)</i>	H. Weak weld lines due to:	
	1. Material being too cold at the point of weld.	Raise temperatures of the cylinder and/or mould.
	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 mould release.	Clean the mould. Use mould release sparingly or not at all.
<b>XIV. Poor colour dispersion.</b>	A. Because of poor mixing ...	
	1. In a ram machine.	Use a dispersion nozzle or premix before moulding. 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 colour 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 moulding.
	2. Base polymer is not compatible with HYTREL®.	Check with the concentrate supplier or Du Pont. 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.	

## For further information on Engineering Polymers contact :

### Benelux

Du Pont de Nemours (Belgium)  
Antoon Spinoystraat 6  
B-2800 Mechelen  
Tel. (15) 44 14 11  
Telex 22 554  
Telefax (15) 44 14 09

### Bulgaria

Du Pont Bulgaria  
18, Tchermi Vrah Blvd.  
Bldg. 2, Floor 5  
BG-1407 Sofia  
Tel. (2) 66 59 13/66 57 36  
Telex 24 261 DUPONT BG  
Telefax (2) 65 63 29/66 56 11

Česká Republika a  
Slovenská Republika  
Du Pont - Conoco CS Spol. S.R.O.  
Palác Kultury  
5, Kvetna 65  
CZ-14009 Praha 4  
Tel. (02) 422 641 to 422 646  
Telex 121 834 DUPO C  
Telefax (02) 61 21 50 51

### Danmark

Du Pont JET Danmark A/S  
Roskildevej 163  
DK-2620 Albertslund  
Tel. 436 236 00  
Telex 33 263 DENDUPONT  
Telefax 436 236 17

### Deutschland

Du Pont de Nemours  
(Deutschland) GmbH  
DuPont Straße 1  
D-61343 Bad Homburg  
Tel. (06172) 87 0  
Telex 410 676 DPD D  
Telefax (06172) 87 27 01

### Egypt

Medgenco International Trade Co.  
13, El Bostan Street  
ET-Cairo  
Tel. (02) 392 78 66  
Telex 93 742 MK UN  
Telefax (02) 392 84 87

### España

Du Pont Ibérica S. A.  
Edificio L'Ilia  
Avda. Diagonal 561  
E-08029 Barcelona  
Tel. (3) 227 60 00  
Telefax (3) 227 62 00

### France

Du Pont de Nemours (France) S.A.  
137, rue de l'Université  
F-75334 Paris Cedex 07  
Tel. 01 45 50 65 50  
Telex 206 772 dupon  
Telefax 01 47 53 09 67

### Hellas

Du Pont Agro Hellas S.A.  
12, Solomou & Vas. Georgiou Street  
GR-152 32 Halandri, Athina  
Tel. (01) 680 16 14  
Telefax (01) 680 16 11

### Israël

Gadot  
Chemical Terminals (1985) Ltd.  
22, Shalom Aleichem Street  
IL-633 43 Tel Aviv  
Tel. (3) 528 62 62  
Telex 33 744 GADOT IL  
Telefax (3) 282 717

### Italia

Du Pont de Nemours Italiana S.p.A.  
Via Aosta 8  
I-20063 Cernusco sul Naviglio (Mi)  
Tel. (02) 25.302.1  
Telefax (02) 92 107 845

### Magyarország

Serviced by Interowa.  
See under Österreich.

### Maroc

Deborel Maroc S. A.  
40, boulevard d'Anfa - 10°  
MA-Casablanca  
Tel. (2) 27 48 75  
Telex 23 719 BOUKBEN  
Telefax (2) 26 54 34

### Norge

Nordic Polymers A/S  
Niels Leuchsvei 99  
N-1343 Eiksmarka  
Tel. 67 14 28 70  
Telefax 67 14 30 44

### Österreich

Interowa  
Führer-Haimendorf KG  
Bräuhausgasse 3-5  
A-1050 Wien  
Tel. (01) 512 35 71  
Telex 112 993 IROWA A  
Telefax (01) 512 35 71 12/  
512 35 71 31

### Polska

Du Pont Conoco Poland Sp.z o.o.  
Intraco Bldg - Floor 13  
ul. Stawki 2  
PL-00193 Warszawa  
Tel. (2) 635 04 01  
Telex 812 376 DUP PL  
Telefax (2) 635 07 61

### Portugal

ACENYL  
Rua do Campo Alegre, 672 - 1°  
P-4100 Porto  
Tel. (2) 69 24 25/69 26 64  
Telex 23 136 MACOL  
Telefax (2) 600 02 07

### Romania

Serviced by Interowa.  
See under Österreich.

### Russia

E. I. du Pont de Nemours & Co. Inc.  
Representative Office  
B. Palashevsky Pereulok 13/2  
SU-103 104 Moskva  
Tel. (095) 956 38 50  
Telex 413 778 DUMOS SU  
Telefax (095) 956 38 53

### Schweiz/Suisse/Svizzera

Dolder AG  
Immengasse 9  
Postfach 14695  
CH-4004 Basel  
Tel. (061) 326 66 00  
Telex 962 306 DOL CH  
Telefax (061) 326 62 04

### Slovenija

Serviced by Interowa.  
See under Österreich.

### Suomi/Finland

Du Pont JET Oy  
Piispankalliotie 17  
P.O. Box 136  
FIN-02201 Espoo  
Tel. +358-9-88 72 61  
Telefax +358-9-88 72 62 00

### Sverige

Du Pont Conoco Nordic AB  
Torshamnsgatan 35  
S-164 86 Kista (Stockholm)  
Tel. (08) 750 37 00  
Telex 12 410 DUPONO S  
Telefax (08) 750 90 41

### Türkiye

Du Pont Products S.A.  
Turkish Branch Office  
Sakir Kesebir cad. Plaza 4  
No 36/7, Balmumcu  
TR-80700 Istanbul  
Tel. (212) 275 33 83  
Telex 26541 DPIS TR  
Telefax (212) 211 66 38

### Ukraine

Du Pont de Nemours  
International S.A.  
Representative Office  
3, Glazunova Street  
Kyiv 252042  
Tel. (044) 294 9633/269 1302  
Telefax (044) 269 1181

### United Kingdom

Du Pont (U.K.) Limited  
Maylands Avenue  
GB-Hemel Hempstead  
Herts. HP2 7DP  
Tel. (01442) 34 65 00  
Telefax (01442) 24 94 63

### Argentina

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

### Brasil

Du Pont do Brasil S.A.  
Al. Itapecuru, 506 Alphaville  
06454-080 Barueri-São Pablo  
Tel. (5511) 421-8468/8556

### Asia Pacific

Du Pont Kabushiki Kaisha  
Arco Tower, 14th Fl.  
8-1, Shimomeguro 1-chome  
Meguro-ku, Tokyo 153  
Tel. (03) 5424 6100

### South Africa

Plastamid  
P.O. Box 59  
Elsies River 59  
Cape Town  
Tel. 27 (21) 592 12 00  
Telefax 27 (21) 592 14 09

### USA

DuPont Engineering Polymers  
Barley Mill Plaza, Building #22  
P.O. Box 80022  
Wilmington, Delaware 19880  
Tel. (302) 892 0541  
Telefax (302) 892 0737

Requests for further information  
from countries not listed above  
should be sent to:

Du Pont de Nemours  
International S.A.  
2, chemin du Pavillon  
CH-1218 Le Grand-Saconnex  
Geneva  
Tel. (022) 717 51 11  
Telex 415 777 DUP CH  
Telefax (022) 717 52 00

### Internet location :

<http://www.dupont.com/>

This information corresponds to our current knowledge on the subject. It is offered solely to provide possible suggestions for your own experimentations. It is not intended, however, to substitute for any testing you may need to conduct to determine for yourself the suitability of our products for your particular purposes. This information may be subject to revision as new knowledge and experience becomes available. Since we cannot anticipate all variations in actual end-use conditions, DuPont makes no warranties and assumes no liability in connection with any use of this information. Nothing in this publication is to be considered as a license to operate under or a recommendation to infringe any patent right.

**Caution:** Do not use in medical applications involving permanent implantation in the human body. For other medical applications, see "DuPont Medical Caution Statement", H-51459.

Start with  
DuPont  
Engineering Polymers



Hytrel®

Only by DuPont