# Recommendations for Injection Molding Stanyl® PA46

## DRYING

Stanyl® resins are packaged at a moisture level equal to or less than 0.10%).

- Resins should be dried to a moisture level equal to or less than 0.05% before injection molding
- Hopper dryers or ovens that circulate heated and dehumidified air are strongly recommended. Dew points should be maintained between -30 and -40˚C (-22 & -40˚F).

## GENERAL DRYING GUIDELINES (Use Dehumidifying Drier)

| Moisture Content | Time | Temp. |
|---|---|---|
| 0.05 - 0.2% | 2-4 h | 80˚C (175˚F) |
| 0.2 - 0.5% | 4-8 h | 80˚C (175˚F) |
| > 0.5% | ≤100 h | 80˚C (175˚F) |
| | or 24 h | 105˚C (220˚F) |

## SCREW ROTATION SPEED

For Stanyl resins, fast screw rotation speeds can be used.

- The circumferential speed of the screw should not exceed 6500/D RPM where D is the screw diameter in mm.

Screw rotation speeds may be lowered in order to:

- minimize shear
- maximize homogeneity
- reduce glass fiber length breakdown (for filled grades).

- The minimum screw rotation speed gives a plasticizing time that is just within the cooling time

## BACK PRESSURE

Keep back pressure low in order to prevent:

- nozzle-drooling
- long plasticizing times.

- Recommended effective pressure 20-100 bar (machine settings less than 50 psi).

## INJECTION SPEED

High injection speeds are required in order to:

- prevent crystallization in the mold during injection phase
- maximize glass fiber orientation (for filled grades)
- obtain better surface finish.

- Adequate mold venting is required to avoid burning at the end of the flow path (due to diesel effect).

## COOLING PHASE

Because of the fast solidification of Stanyl, cooling time can be very short. For this reason the plasticizing time will be the determining step for cycle time. A wide range of molding temperatures can be used:

- For dimensional stability and flow, mold temperatures above 80°C (175°F) are recommended
- For optimal crystallinity for unreinforced grades and optimal mechanical performance, mold temperatures above 120°C (250°F) are recommended
- Holding time and pressure depend on gate and runner design and shape of molded article.

## PROCESSING STANYL HIGH FLOW

Due to their improved flow, Stanyl HF grades exhibit less shear heating and LCP settings may be applied:

- For optimal mechanical performance and flow, the temperature of the melt should be around 315-325°C (615°F)
- For dimensional stability, a 120°C (250˚F) mold temperature is recommended
- Increased temperature settings in the compression/metering zone in order to achieve proper melt temperature
- Lower temperature at the feed zone but at least 280°C (535°F)
- High screw rotation speeds
- Low effective back pressure ±10 bar (0 – 50 psi machine settings)
- High injection speed
- Control part fill with screw position or start with low or moderate pressure to avoid overpacking. Use low or moderate holding pressures in case of cushion variations
- Check for drooling at the nozzle and adjust nozzle temperature if necessary, but not too low! Use a reverse tapered polyamide nozzle.

## SAFETY

For all safety aspects we refer to our Material Safety Data Sheets, which are available upon request from your local DSM Sales Office.

For all thermoplastics:

- Wear face shield and heat resistant gloves when handling a hot melt (e.g. when purging)
- Dust masks must be used during regrinding operations.

## TROUBLE SHOOTING

See the section on troubleshooting for a full listing of injection molding problems, their causes and remedies.

*When problems cannot be solved with the aid of these guidelines, please contact our local Sales representative through your DSM Sales Office.*

# Trouble Shooting

PROBLEM
**Cause:**
- ❑ remedy

NOZZLE DROOL
**Material is wet:**
- ❑ use dried material (< 0.05% moisture )

**Melt temperature too high:**
- ❑ reduce nozzle temperature
- ❑ reduce cylinder temperature (check melt temperature with probe)
- ❑ reduce residence time in cylinder (shorter cycle time) or use smaller barrel size
- ❑ reduce screw speed

**(Back) pressure too high:**
- ❑ reduce back pressure
- ❑ use screw decompression

**Wrong nozzle design:**
- ❑ improve temperature control of nozzle
- ❑ use reverse tapered nozzle
- ❑ use nozzle with smaller orifice diameter

NOZZLE FREEZE-OFF
**Temperatures too low:**
- ❑ increase nozzle temperature
- ❑ increase melt temperature (do not exceed 320°C (610°F) for standard grades or 330°C (625°F) for HF grades)
- ❑ retract nozzle from mold after plasticizing

**Wrong nozzle design:**
- ❑ improve temperature control of nozzle
- ❑ increase reverse taper of nozzle orifice
- ❑ use nozzle with larger orifice diameter

NOZZLE SPITTING
**Degradation of material leads to gas formation:**
- ❑ use dried material (< 0.05% moisture)
- ❑ reduce residence time
- ❑ reduce melt temperature, not too low [>305°C (580°F)]

MOLDING CONTAINS UNMELTED GRANULES
**Insufficient energy input:**
- ❑ increase barrel temperature
- ❑ increase back pressure
- ❑ increase screw speed
- ❑ preheat granulate to 80-100°C (175-210°F)
- ❑ increase barrel size

SCREW SLIP
**Low friction or premature melting:**
- ❑ reduce temperature in rear zone
- ❑ preheat granulate
- ❑ decrease back pressure
- ❑ decrease screw speed
- ❑ purge to cool down

SHORT SHOTS
**Insufficient material:**
- ❑ check hopper content
- ❑ adjust feed setting
- ❑ check the transfer point

**Insufficient flow:**
- ❑ increase injection pressure/speed
- ❑ check melt temperature: if necessary increase temperature

**Too much resistance:**
- ❑ increase diameter of gate, runner, sprue and nozzle orifice
- ❑ increase venting
- ❑ use more gates

FLASH
**Insufficient locking force:**
- ❑ increase clamping force
- ❑ reduce injection speed/pressure
- ❑ use profiled injection speed/pressure
- ❑ check the transfer point
- ❑ reduce holding pressure
- ❑ reduce melt temperature

**Over-molding:**
- ❑ reduce feed setting
- ❑ check the transfer point
- ❑ reduce holding pressure
- ❑ check if mating area is clear

**Mold problems:**
- ❑ clean or enlarge vents
- ❑ check parting line
- ❑ check for wear in mold

SINK MARKS AND/OR VOIDS
**Too much shrinkage:**
- ❑ increase holding time
- ❑ increase holding pressure
- ❑ decrease injection speed for thick sections
- ❑ increase injection speed for thin sections
- ❑ increase mold temperature

**Mold problems:**
- ❑ increase diameter of gate, runner, sprue and nozzle orifice
- ❑ change position of the gate towards thickest section
- ❑ use more gates

WELD LINES
**Incomplete mixing of two melt fronts:**
- ❑ increase injection speed
- ❑ increase holding pressure and time
- ❑ improve venting at weld line area
- ❑ increase melt temperature
- ❑ increase mold temperature
- ❑ add an overflow well at the weld line area
- ❑ change position of the gate to move weld line to less critical area
- ❑ increase diameter of gate, runner, sprue and nozzle orifice

**Air traps:**
- ❑ decrease injection speed
- ❑ improve venting
- ❑ improve flow
- ❑ change location of the gate

BURN MARKS
**Diesel effect due to compressed air:**
- ❑ improve venting
- ❑ decrease injection speed
- ❑ decrease melt temperature
- ❑ change location of the gate

FLOW LINES, DELAMINATION
**Melt temperature too low:**
- ❑ increase melt temperature
- ❑ increase mold temperature
- ❑ increase injection pressure/speed
- ❑ increase diameter of gate, runner, sprue and nozzle orifice

**Contamination of the granulate with other polymers:**
- ❑ use clean virgin granulate

SILVER STREAKS, SPLAY MARKS, MICA EFFECT
**Wet material:**
- ❑ use dried material

**Degraded material:**
- ❑ reduce residence time (cycle time) or use a smaller barrel
- ❑ reduce melt temperature (check nozzle temperature)
- ❑ reduce screw speed
- ❑ reduce back pressure
- ❑ increase gate diameter
- ❑ reduce injection speed/ pressure
- ❑ avoid/minimize decompression

**Air trapped in the melt:**
- ❑ increase back pressure
- ❑ reduce screw speed
- ❑ reduce injection speed/pressure
- ❑ reduce temperature in rear zone
- ❑ improve mold venting

## DISCOLORATION, BROWN STREAKS

**(locally) Overheated material:**
- ❑ check for dead spots; nozzle and/or hot runner and non-return valve
- ❑ reduce residence time (cycle time) or use smaller cylinder
- ❑ reduce melt temperature
- ❑ reduce nozzle temperature
- ❑ check injection unit for wear, causing excessive shear
- ❑ check for contamination, also drier, hopper, etc.
- ❑ reduce injection speed
- ❑ reduce decompression

**Contamination of the granulate:**
- ❑ clean injection unit
- ❑ use clean virgin granulate

## BLACK STREAKS

**Contamination of the granulate:**
- ❑ clean injection unit
- ❑ use clean virgin granulate

**(locally) Overheated material:**
- ❑ see 'Discoloration, Brown Streaks'

**Too low melt temperature:**
- ❑ increase cylinder temperature, especially at hopper side

## BRITTLENESS

**Degraded material:**
- ❑ reduce residence time (cycle time) or use smaller cylinder
- ❑ reduce melt temperature
- ❑ reduce screw speed
- ❑ increase gate diameter
- ❑ reduce amount of regrind

**Too cold material:**
- ❑ increase melt temperature (above 305˚C (580˚F) for standard grades; above 315˚C (600˚F) for HF grades)

**Wet material:**
- ❑ use dried material

**Internal stresses:**
- ❑ reduce holding pressure
- ❑ check the transfer point
- ❑ increase mold temperature

**Non-homogeneous melt:**
- ❑ increase back pressure

**Contamination of the granulate:**
- ❑ check for contamination

**Components are dry:**
- ❑ moisture condition components

**Design problems:**
- ❑ avoid sharp corners, apply radii

## MATTE SURFACE

**Insufficient mold finish:**
- ❑ Poor mold surface

**Unreinforced grades:**
- ❑ increase mold temperature
- ❑ reduce thermal hot spots in mold (opposite gate)
- ❑ increase melt temperature
- ❑ increase injection speed

## FROSTING "GLASS FIBERS ARE VISIBLE"

**Glass fiber reinforced grades:**
- ❑ increase mold temperature
- ❑ increase injection speed
- ❑ increase melt temperature
- ❑ increase holding pressure

**Wet material:**
- ❑ use dried material

## DELAMINATION, BLISTERING

**Wet material:**
- ❑ use dried material

**Internal shear stresses:**
- ❑ use moderate injection speed
- ❑ increase mold temperature
- ❑ increase melt temperature
- ❑ increase gate diameter

**Contamination:**
- ❑ use clean virgin material
- ❑ clean injection unit

## MOLD EJECTION PROBLEMS

**Overpacking:**
- ❑ reduce injection speed
- ❑ check the transfer point
- ❑ reduce holding pressure/time

**Mold design problems:**
- ❑ check for mold damage or undercuts
- ❑ use draft angle
- ❑ use lubricated grades or add 0.05-0.1% lubricant based on polyethylene wax
- ❑ change mold temperature (depending on shape of article)

## WARPAGE

**Differential shrinkage (unreinforced grades):**
- ❑ adjust mold temperature of each mold half (high temperature = high shrinkage)
- ❑ assure uniform wall thickness
- ❑ increase cooling time

**Glass fiber reinforced grades:**
- ❑ change location of the gate
- ❑ reduce injection speed
- ❑ increase mold temperature
- ❑ increase melt temperature
- ❑ use mineral filled / low warpage material

**Internal stresses:**
- ❑ reduce holding pressure
- ❑ reduce holding time
- ❑ check the transfer point
- ❑ increase mold temperature
- ❑ use HF grades

## CYCLE TIME

Due to the fast crystallization of Stanyl short holding and cooling times are possible. Compared to other high performance thermoplastics this results in shorter cycle times (up to 50% reduction)

| Wall Thickness | 1mm (0.04in) | 2mm (0.08in) | 3mm (0.12in) |
|---|---|---|---|
| Cooling Time* | 5 sec | 7 sec | 9 sec |

* Actual cooling time will depend onpart geometry & dimensional quality requirements as well as the tool design.

## HOT RUNNERS

When processing Stanyl with hot runners, keep in mind these basic rules:

- Central bushing heated separately
- Manifold heated from both sides
- Tip with thermocouple in front (near gate)
- Only use external heated system.

**Suggestion:**
Start up with a PA66 GF, then increase temperatures to PA46 settings. Don't stop the machine, and change the polymer material to the required PA46 material. After several shots Stanyl can be molded properly.

## SAFETY WHEN PROCESSING HOT RUNNERS

During short production stops the injection unit must be pulled back, the screw should be in the most forward position and when downtimes are relatively long, thorough purging with HDPE (MFI <1) is required.

Hot runner temperatures should be decreased during longer production stops.

Take care when the hot runner is blocked due to a frozen nozzle, tip, adapter or sprue bushing. If the melt is present for a longer period of time, it will lead to degradation. This may be followed by pressure build up in the barrel, nozzle and the hot runner system. Be aware that under these conditions a sudden outburst of molten material can take place. Always wear safety protection whenever working inside a mold opening, or when working on the nozzle!

## SALES OFFICES

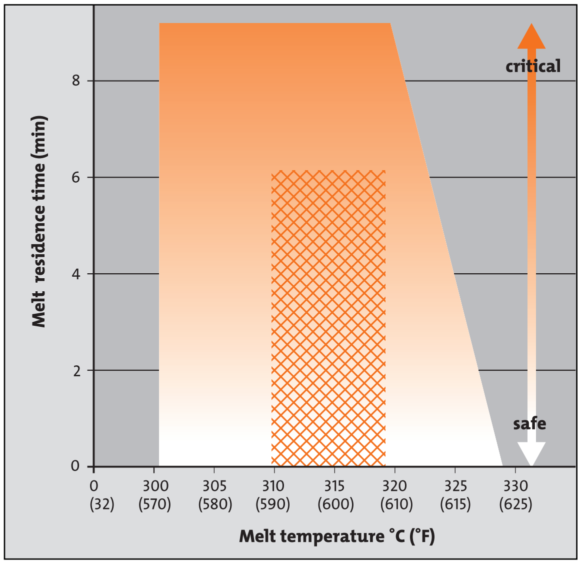
**North America**
DSM Engineering Plastics
P.O. Box 3333
2267 West Mill Road
Evansville, IN 47732-3333
Tel. 812 435 7500
Fax 812 435 7702
www.dsmep.com

**Europe**
DSM Engineering Plastics
Poststraat 1
6130 AA Sittard
The Netherlands
Tel. 31 46 477 04 50
Fax 31 46 477 3959
www.dsmep.com

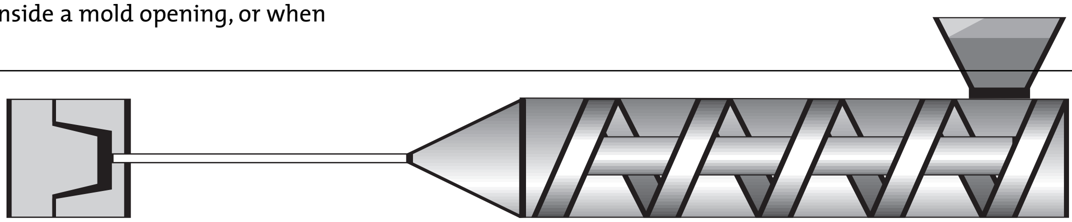
**Asia Pacific**
DSM Engineering Plastics
Suite A, 10th Floor
Century Ba-Shi Building
No. 25 Middle Chong Qing Road
Shanghai 200020, China
Tel. 86 21 6386 3080
Fax 86 21 6386 2198
www.dsmep.com

01/05 Web

## PROCESSING WINDOW




## TEMPERATURE SETTINGS



| Stanyl | Mold | Melt | Nozzle* | Front | Center | Rear |
|---|---|---|---|---|---|---|
| **Standard Grades** | 80-120°C (175-250°F) | 300-320°C (580-610°F) | 280-300°C (535-570°F) | 300-320°C (570-610°F) | 300-320°C (570-610°F) | 280-320°C (535-610°F) |
| **High Flow Grades** | 60-120°C (140-250°F) | 315-330°C (600-625°F) | 280-300°C (535-570°F) | 315-330°C (600-625°F) | 310-325°C (590-620°F) | 280-310°C (535-590°F) |

* The correct nozzle temperature should show a neat sprue end and no drooling from the nozzle. This temperature is usually 5 to 20 degrees lower than the actual polymer temperature. Be careful during the start-up for nozzle freeze-off.