SABIC
Innovative
Plastics™

سابك
sabic

Injection molding processing guide

Sharing our futures

Contents

Contents

Injection molding trouble shooting guide for LNP* engineering compounds

Problem	Cause	Remedy
Brittleness		
	Wet material	• Review drying procedure
	Overheating	• Reduce barrel/nozzle temperature
	Molded-in-stresses	• Increase barrel/nozzle temperature
	Poor part design	• Eliminate sharp corners
	Weld-lines	• Increase injection pressure
		• Increase melt temperature
Warped parts		
	Part temperature differential	• Check mold cooling system
	Excessive shrinkage	• Increase part packing
	Orientation of material	• Change gate location
	Poor part design	• Add ribs or part thickness to improve stiffness
	Ejection problem	• Check for uniform wall thickness
		• Increase cooling time
		• Reduce mold temperature
		• Increase ejector pin area
Flashing		
	Inadequate clamp tonnage	• Use a larger machine
	High Injection Pressure	• Reduce injection pressure
	Misaligned platen	• Align platen
	Excessive vent depth	• Review mold venting
Burn marks		
	Air trapped in cavity	• Improve mold venting
	Barrel or nozzle overheating	• Check heater controls
	Shear heat	• Reduce injection speed
	Contamination	• Purge barrel
		• Clean hopper dryer
	Hang-up in molding machine	• Remove and clean screw
Weak weld-lines		
	Insufficient venting	• Improve cavity venting
	Injection speed or mold temperature too low	• Increase injection rate and tool temperature
	Incorrect gate location	• Relocate gate or add overflow tab
Sticking in mold		
	Over packing	• Reduce injection pressure
		• Reduce injection speed
	Mold design	• Check for undercuts
		• Inspect ejector system
		• Increase draft in tool
Surface imperfections (frosty surface or white marks)		
	Injection speed too low	• Increase injection speed
	Cold melt/cold mold	• Increase barrel temps/mold temperature
	Wet material	• Review drying procedures
Sinks or voids		
	Holding pressure or time too low	• Increase hold pressure or time
	Insufficient feed	• Increase shot size
	Gate freezing off or located improperly	• Check gate dimension and location
Gate blush		
	Cold material	• Add cold slug well in runner
		• Increase melt temperature
	Melt fracture	• Reduce injection speed
		• Increase gate size
		• Add radius to gate
Dimensional inconsistency		
	Shot to Shot variation	• Maintain adequate cushion
	Melt temperature variation	• Check for worn check ring
	Inadequate packing	• Check heater bands/controllers
		• Increase hold time
		• Enlarge gate to prevent premature freeze-off

Part design

Performance of parts made from LNP* composites depends on the compound properties, part design, and the molding process. Good part design is critical for meeting application structural requirements and molding productivity. Assembly requirements should also be considered during the design stage.

Design process

The design process can be simplified by following a three stage approach where material, design and fabrication decisions are made in parallel. SABIC Innovative Plastics engineers should be consulted during the early stages of the design process. Additional information on each of the above steps can be found in the LNP Speciality Compound Design Guide. The following guidelines are reminders of good design practice aimed at producing quality injection molded parts.

Preliminary stage
- Define requirements
- Establish conceptual geometry
- Select materials
- Select fabrication method
- Perform feasibility analysis
- Decision to move forward

Engineering stage
- Complete detailed part design
- Fabrication decisions
- Material decisions
- Prototype testing
- Evaluation and redesign

Manufacturing stage
- Design, build and evaluate a tool
- Cavity filling analysis
- Manufacturing equipment selection
- Part testing
- Customer evaluation

Design for injection molding
Part geometry is critical to achieving a well-molded part. Considerations include

Wall thickness
Uniform wall thickness throughout the part and attention to nominal thickness is important. (See guidelines below)

Uniformity
- Residual stresses (warpage, sinking, chemical resistance)
- Mechanical properties (strength, impact resistance)

Nominal thickness
- Agency approvals (flammability)
- Processibility (flow, length, cycle time)
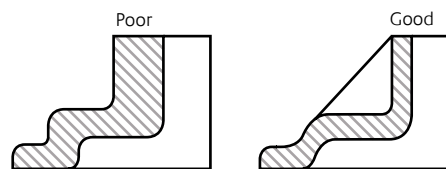- Maximum thickness based on polymer system

Coring
Core thick areas of the part to maintain uniform wall sections and even cooling.
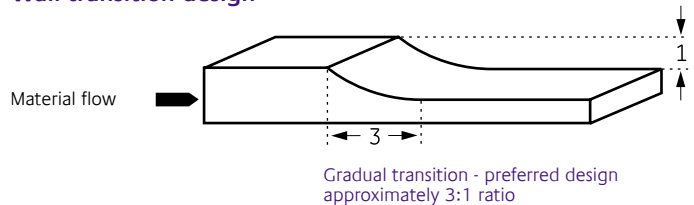
Shrinkage and tolerances
Typical shrinkage of glass-fiber-reinforced compounds will be one-third to one-half that of non-reinforced resin. LNP recommends starting with a prototype tool to determine exact shrinkage, particularly on parts with complex shapes or drastic variations in wall thickness. Parts molded from compounds which exhibit anisotropic shrinkage characteristics (reinforced, crystalline resins) should also be prototyped initially or molded on a "surrogate" tool in order to predict critical shrinkage results.
In general, reinforced compounds can be molded to tighter tolerances than unfilled materials. Holding tight tolerances can significantly increase the cost of a molded part since designing for close tolerances may add steps to the manufacturing process or require higher tooling costs compared to coarse tolerances. For more information on tolerances and warpage, see page 25.

Variable wall section diagram
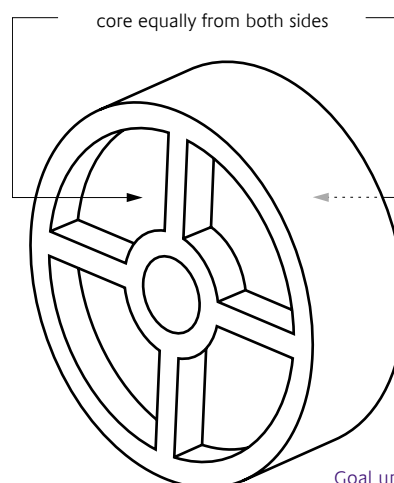

Poor   Good

Wall transition design

Material flow

1

3

Gradual transition - preferred design
approximately 3:1 ratio

Coring

core equally from both sides

Goal uniform wall with even pressure distribution and even cooling

Radii

Sharp corners cause stress concentrations and should be avoided. See guidelines below for proper corner radii.

Draft

All part walls should have 2°–3° draft per side whenever possible, with a minimum of 1° draft since reinforced compounds shrink less than neat resins. Unfilled compounds should maintain 1/2° draft per side minimum. Textured surfaces require an additional 1° draft per side for every 0.001" depth of texture.
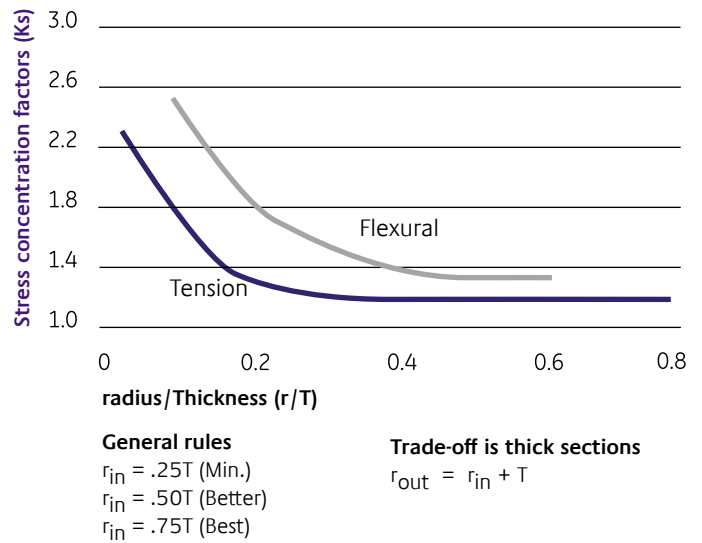
Weld-lines

Proper tool design (gate location, etc.) can help minimize the formation of weld-lines. If they cannot be avoided, they should be located in the area of the part where minimal applied stress is expected.

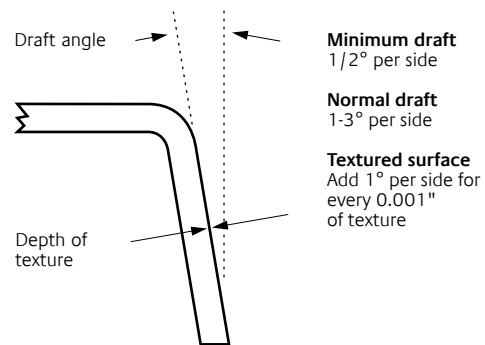Our experience with LNP* composites and weld-lines has shown that

- Weld-line tensile strength of filled or reinforced compounds depends partly on the inherent weld-line integrity of the base resin

- Fibrous reinforcement, which orients parallel to the weld-line, can cause dramatic loss of strength in the weld-line area. The amount of decrease is directly related to the volume of reinforcement. Particulate fillers do not effect weld-line strength to such a degree

- Part thickness has little or no effect on weld-line tensile strength, although thicker walls usually reduce stress

- Molding variables – other than hold time — do not significantly effect weld-line strength

- Optimized venting should be provided at the weld-line to maximize weld-line strength

- Overflow tabs do not improve weld-line strength enough to justify the corresponding increase in cost

Ribs/bosses/gussets

Ejection problems or thick sections (sink marks) may result from improperly designed ribs, bosses, or gussets. Long core pins may overheat or deflect if not designed properly. Unsupported pins should have an L/D of < 5/1. Copper alloys may be used to provide better cooling of long core pins.

General rules

r_{in} = .25T (Min.)
r_{in} = .50T (Better)
r_{in} = .75T (Best)

Trade-off is thick sections

r_{out} = r_{in} + T

Draft

Minimum draft
1/2° per side

Normal draft
1-3° per side

Textured surface
Add 1° per side for every 0.001" of texture

Ribs

Gussets

Bosses

Controlled by stress in boss with t as optimum t = .50T to .75T
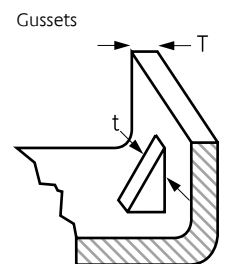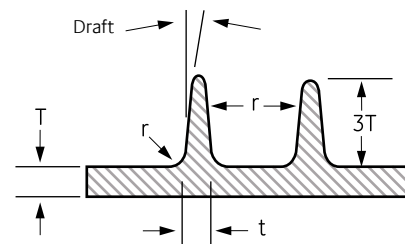
Materials of construction

Choosing the proper mold steel is dependent on the application. For prototyping, it may not be necessary to use a hardened tool steel. In most instances, a pre-hardened steel or aluminum is used to minimize cost and allow easy modification of the tool during the prototype stage. These softer metals also allow molding of sufficient test pieces and often pre-production parts as well. Quite often a pre-hardened tool steel such as P-20 or NAK[R]-55, is used when making very large molds, as the hardening of tool steels in large applications becomes very impractical.
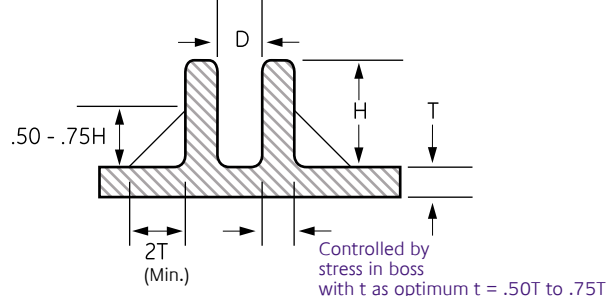
Higher production quantities require hardened tool steels for cores and cavities. S-7, H-13, and often stainless steel 420 are the most commonly used steels. S-7 is an excellent mold steel and can provide long production runs. When high melt and mold temperatures are required, the steel of choice will be H-13. H-13 is also used to produce hot runner manifolds. H-13 has very high tempering temperatures and can withstand high mold processing temperatures without loss of hardness.

In cases where high wear and abrasion-resistance are required, or in cases where the environment produces a lot of condensation, stainless steel will be the mold steel of choice. A2, ASP23 or D-2 steels may be used as cavity inserts in areas of high wear.

All tool steels can be protected with some kind of plating to protect against wear, abrasion and corrosion. Only stainless steel can be repaired through welding and machining. On steels which have been plated, repairs can only take place after the plating has been removed. After repair, it will then have to be reapplied.

Suggested mold steels

Steel	Rockwell hardness (C Scale)	
SS420	30-35	Melt processible fluoropolymer composites
H13	48-54	
S7	54-56	Thermocomp*, Stat-Kon* composites
D2	56-58	Lubricomp*, and Verton* composites

The steels listed above can be improved for corrosion resistance using electroless nickel or chrome plating.

Surface finish

Depending on the requirements for part aesthetics, customer needs and performance functions, the surface of a molded part may vary. From a SPI #1 mirror-like high gloss finish to a textured finish produced using a photo-etching technique, almost any type of molded-part surface finish is possible in an LNP* composite. It is important to understand that certain materials perform better with certain mold surface finishes. As an example, polypropylene releases better out of a mold with a matte finish as opposed to a high polish. A high gloss part is difficult to achieve in a highly filled resin.

Venting

To maximize mold performance, venting is an important function during the molding cycle. As thermoplastic materials enter the mold cavity, the air in the cavity needs to be expelled. Vents are placed usually in the areas last to fill, near knit lines, as well as on the runner system. Additional venting along the parting line perimeter will improve overall venting significantly. Trapped air in a mold manifests itself as a burn mark on the molded part. Basic rule of thumb air in mold must be able to escape at the same rate plastic enters cavity.

Vent depth varies with the materials used — usually amorphous thermoplastics require deeper vents as viscosity is higher.

Venting

Thermocomp resins	Venting depth	
Styrenics (A, B, C, N series)		
Polycarbonate		
PEI/PEEK/PPO		
PSUL/PUS/PAS	0.0015 – 0.003"	Increase depth to 0.03" after a 0.03" land length
Thermoplastic polyurethane		
Amorphous nylon		
Polyester elastomer		
Polyethylene		
Polypropylene		
Nylons	0.0005" – 0.001"	
(H, I, P, Q, R, S series)		
Acetal		
PBT/PET		
PPS	0.0005" – 0.001"	

Single and multi-cavity

There are three basic types of molds most commonly used in the industry, and specifically with LNP* compounds. The type used most often is a two-plate, cold runner configuration shown in Figure 1. Material is injected through the sprue bushing, runner system into the part. After cooling, the entire molded piece is ejected (including gate, runner, and sprue). The parts are then removed, and the remaining runner and sprue are either discarded or recycled. Shown in Figure 2 is a three-plate mold with a cold runner. Using this design, the material is injected through a sprue and runner system into the part cavity. After cooling, the part is separated from the gate and runner as the mold opens. The parts fall free from the runner. Sprue, gate and runner still need to be recycled or discarded. Figure 3 shows electrically heated hot runner manifold tools. The manifold and gate probe channels are heated to a temperature at or near the polymer melt temperature in order to keep the material in a molten state. As a result, there is no sprue, gate and runner waste to contend with. These systems are more expensive compared to the conventional two or three-plate designs, however, they provide better thermal control of the material and little to no waste.

Hot runners

Full hot runner molds or "runnerless" molds have many advantages but need to be carefully designed to ensure proper control of melt delivery to the cavities. While providing virtually no waste during the molding operation, runnerless tools can also be vulnerable to problems during start-up, particularly with semi-crystalline LNP compounds, which are prone to fast freeze-off. Guidelines to follow include

Figure 1
Two-plate cold runner mold

Top clamping plate
Water cooling lines
Cavity plate 1
Cavity and molded part
Mold seperates
Runner
Core plate
Core
Support plate
Knockout pins
Ejector housing
Ejector plates
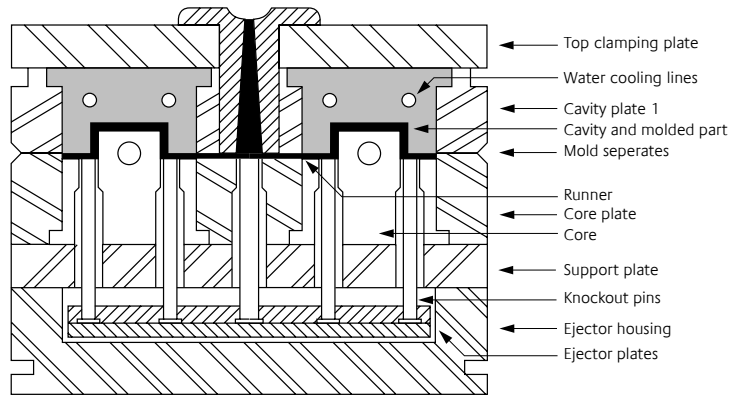
Figure 2
Three-plate mold system "opened" position

P.L. 1
P.L. 3
P.L. 2

Top clamp plate
Undercut pin
Floating plate
Secondary sprue
Spring
Cavity plate
Piece part
Pull rod
Stripper plate
Stripper plate guide pin
Return pin
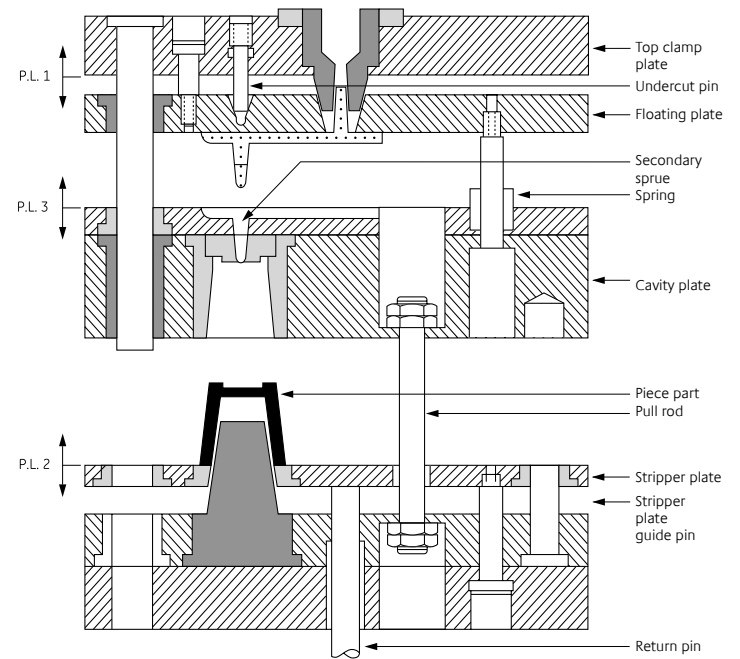
Three-plate mold system "closed" position

P.L. 1
P.L. 3
P.L. 2

Top clamp plate
Primary sprue
Back-up plate spring
Cavity plate
Stripper plate guide pin
Pull rod
Return pin
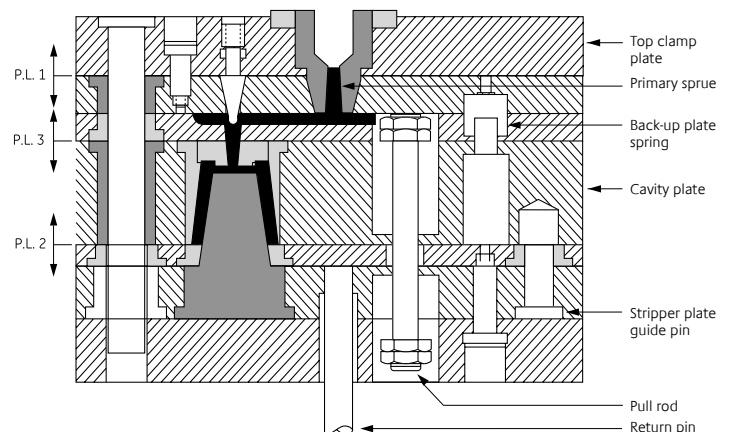
- Use only a well-balanced hot runner manifold system built specifically for the application

- Depending on materials used, hot runners can be designed to gate directly onto a part with an open gate drop, use a valve gate for vestige-free molding, or gate into a small surface runner

- Heated runner paths and hot drops should provide a streamlined flow path to the cavity and be externally heated

- When using long glass fiber reinforced resins, gate vestiges are unavoidable

- Use complete systems from one hot runner manufacturer rather than different components from different manufacturers

- Gate dimensions depend on amount and type of material displaced, part size/configuration and wall thickness

- Contact SABIC Innovative Plastics engineers for more specific information on hot runner designs

Figure 3
Hot runner manifold tool
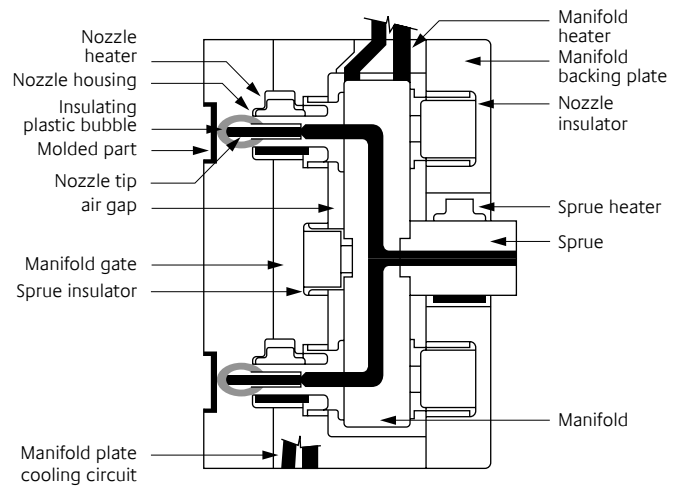
A guide to tool surface enhancements commonly used in the plastics processing industry

The chart listed below provides molders and tool builders with several common options for treating tool steel to provide corrosion protection or improve abrasion resistance. Other solutions may be available.

|	Application coating	Application process	Substrates	Hardness	Benefits
PTFE infused into metallic coatings	**Poly-Ond**				
	Nickel phosphorous impregnated with polymers. baked to cure and set	Electroless nickel deposition sprayed or dipped in PTFE, baked to cure and set	Almost any material — steel, aluminum, brass, bronze, cast iron	50 R_C (68-70R_C) if baked at 400°C (750°F for 1 hour.)	May reduce cycle times and reject rates. May reduce or eliminate need for release sprays. May prolong tool life and increases hardness
Metallic platings	**Industrial hard chrome**				
	Chromium plus trace amounts of oxides and hydrogen.	Electrolysis deposition. High density.	Most metals, many other surfaces.	68–70 R_C	Can salvage worn or mismachined parts by restoring dimensions. Good adhesion. Hard, slippery surface.
	Electroless nickel				
	Nickel alloy, with varying phosphorous content to provide needed properties.	Deposited by chemical reduction without electric current.	Ferrous metals, aluminum and copper alloys.	48 R_C as applied. Heat-treated up to 70R_C	Adheres well. High hardness and corrosion resistance make it popular for use in molds. Uniformly deposited to close tolerances. Good lubricity.
	Titanium nitriding				
	Thin-film, high temperature coating.	Negative ionized process. Physical vapor deposition.	Most steels, stainless steel, beryllium copper.	85 R_C 2,300 Vickers 0.003 HV	Reduces friction. Uniform thickness of thin coating will not affect dimensional tolerances. Good for glass-reinforced resins
Thin film, hardness coatings	**Diamond black**				
	Boron carbide thin film with tungsten disulfide. (no Pb, P, or S).	Magnetron sputtering at low temperature.	A-2, A-6, S-7, M-2 D-2, H-13, Stavax, aluminum, BeCu (no Pb, P or S).	93–95 R_C 4,700 Knoop	Can apply to micro finishes. Hardness increases tool life and eases cleaning. Provides good lubricity and improves corrosion resistance.

Sprues, runners, and gates
Sprue bushing
With conventional molds, the hot material is introduced into the mold with a sprue bushing. A runner system transfers the material through a gate into the cavity. Proper design and dimensions of sprue bushings are critical to achieving proper melt distribution and ejection of the molded part. The sprue "O" diameter should be approximately 20% larger than the exit diameter of the nozzle to prevent hang-up of the sprue when the mold opens. Guidelines for sprue dimensions are shown below. "O" dimensions are also critical in relation to part and runner size.

In place of a conventional sprue bushing, a heated sprue bushing can be used. Initial cost of a heated sprue or "hot" bushing may be higher than a "cold" bushing, but it will minimize the amount of waste generated by eliminating the sprue attached to each part or runner system. With hot bushings, control of the melt temperature entering the runner system is more accurate.
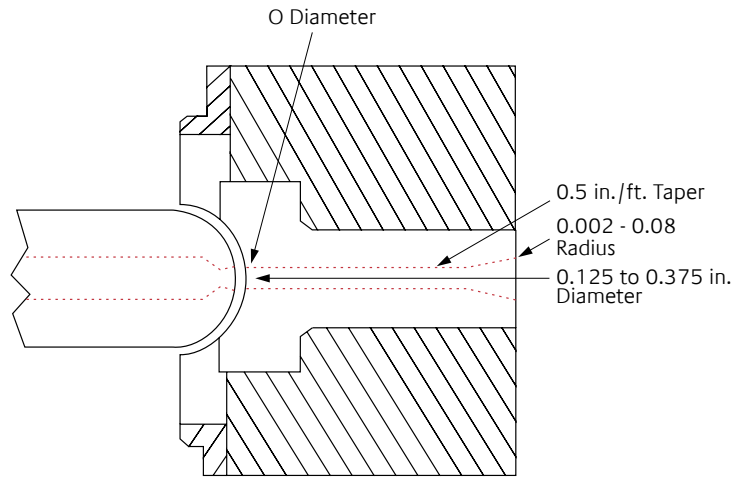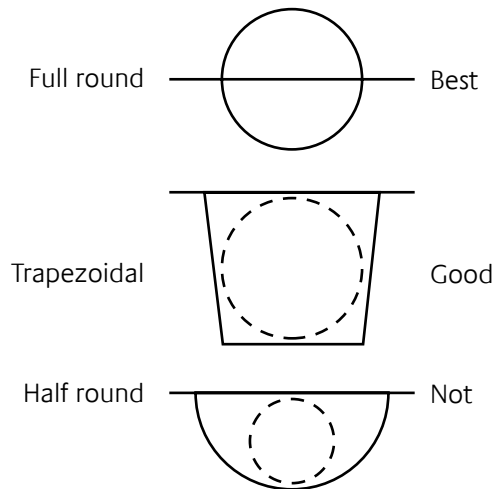
Runner design
Runners should be as short as possible to reduce unnecessary pressure drops. Full round cross sections are best. If the runner must be placed on one half of the mold, use a trapezoidal shape with a round bottom. Half round and flat runners are not very efficient and are not recommended.

Runners should provide balanced, unrestricted flow to the part or parts (if multi-cavities are used). Generally, every 90 degree turn in a runner requires a 20% decrease in diameter. This needs to be taken into consideration when designing the runner system. Round runners typically range from 0.125" – 0.375" diameter (3.2 - 9.5mm).
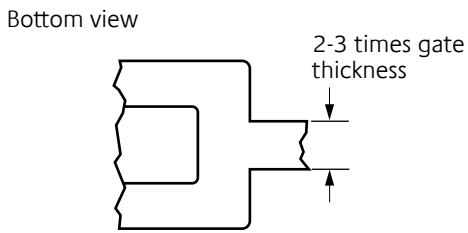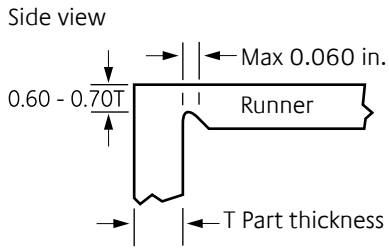
Gates
Gating design of parts for LNP* composites should provide ample processing latitude without creating molded-in stresses. Depending on what gate type is used (i.e., sub, edge, tab, etc.), a gate still has fundamental dimensions which control fill rate, amount of material flowing into the cavity, and the rate of part solidification. Wall thickness governs the size of the gate while part geometry governs gate location on the part. There are three critical dimensions on a gate depth, width, and land. Details on several gating designs are shown on the next page.

Gate size/location
• Minimum 0.040" (1mm) gate size, 0.100" (2.5mm) for Verton* long fiber composites
- Full round or trapezoidal
- 50 – 60% of wall thickness for crystalline composites
- 50 – 75% of wall thickness for amorphous composites
• Preferably into thickest wall section
- Locate gate directly opposite wall or boss to prevent back-fill and "jetting" (except for Verton long fiber composites)
- Tunnel gates are not recommended for Verton composites.

Sprue bushing

O Diameter

0.5 in./ft. Taper
0.002 - 0.08 Radius
0.125 to 0.375 in. Diameter

Runner configuration

Full round — Best

Trapezoidal — Good

Half round — Not

Full round or trapezoidal recommended
Half round not recommended due to high surface area

Detail of rectangular edge gates

Side view

0.60 - 0.70T

Max 0.060 in.

Runner

T Part thickness

Bottom view

2-3 times gate thickness

Conventional gate
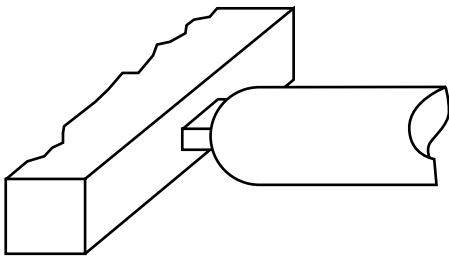
Characteristics
- 1/4" to 1/2" the cross sectional area of the runner
- Visual defect is large
- Small pressure loss
- Plastic freezes slowly

Max. 0.060 in.

Land

0.025 - 0.100 in.

5° to 20° taper

1/8 to 1/4 of runner diameter

*

Angle of tunnel gate should not exceed 15°-20° off of vertical for stiff materials

Gates to prevent jetting

Impinging edge gate

Section

Bottom view

Overlap gate

Side view

Bottom view

Tab gate

Part

Tab

Gate

Runner
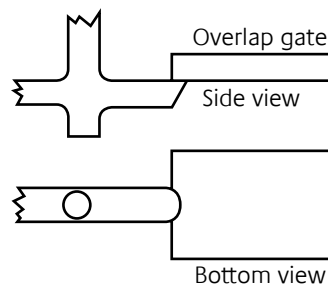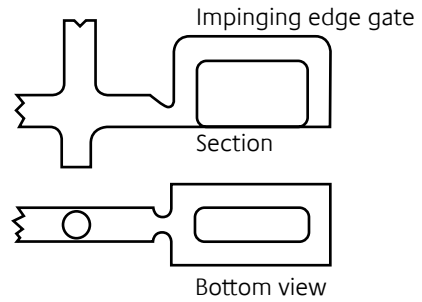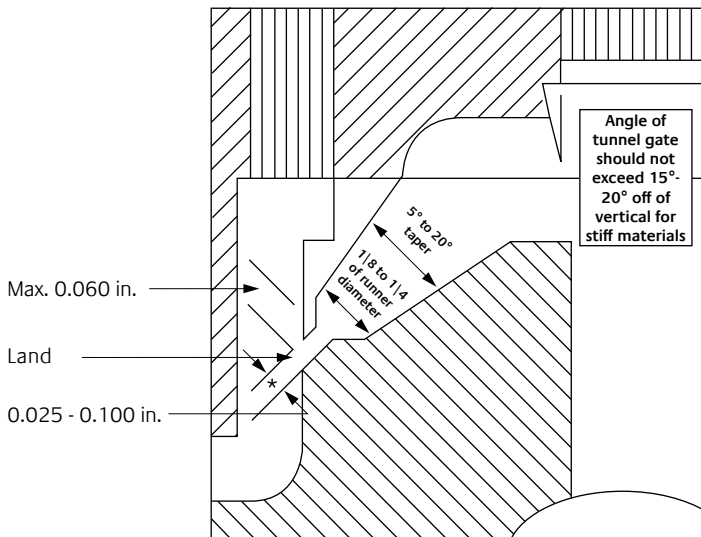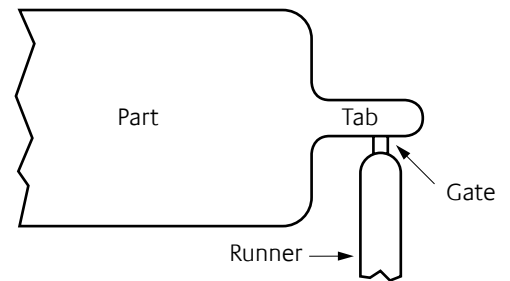
Basic design features

LNP's* engineering resins process best on in-line reciprocating screw injection molding machines. Ram or plunger type equipment should be avoided because they do not provide uniform melt homogeneity.

Injection molding machines selected should provide a means of monitoring and controlling all molding parameters with good accuracy. Part quality is maximized when individual control of injection and holding pressures, ram position and velocity, back pressure and screw speed are provided. Good barrel and nozzle temperature control is also important since optimum melt temperature control is important for achieving low thermal stress levels in the molded part.

Closed loop (PID) temperature controls should be utilized whenever possible.

Select a molding machine that will utilize approximately 40–70% of the machine's rated barrel capacity — converted to reflect the higher density of an LNP composite vs. general purpose polystyrene. Smaller shot weights are possible but may limit the processing latitude of the composite and increase the possibility for thermal degradation due to longer residence time in the barrel.

Clamp tonnage requirements for LNP composites range from 3–6 tons/in^2 (42–83 MPa) with increased clamp force required for the higher viscosity composites.
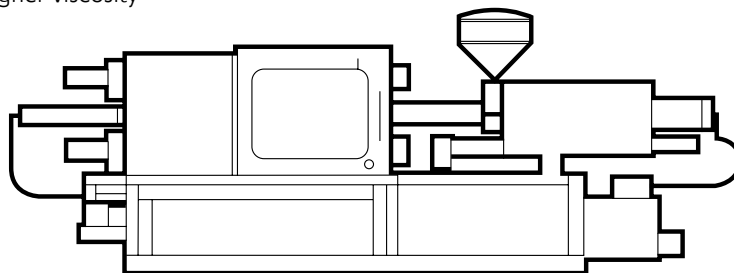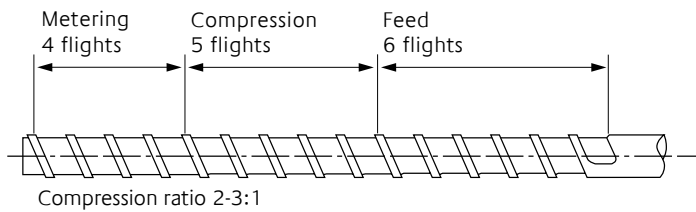
Barrel and screw features

The following are important design points when selecting a machine for injection molding of LNP's composite materials

- Choose a single-stage plasticating unit utilizing an abrasion resistant, bimetallic barrel liner. Two-stage, vented barrel units should be avoided unless prior experience has demonstrate good results.

- While vented barrel processing may reduce the amount of drying required for unfilled, hygroscopic resins such as polycarbonate or nylon, reinforced and/or filled composites may hinder the devolitalization process and cause hydrolytic degradation.

- The injection screw should be constructed of materials that provide a balance of abrasion resistance and corrosion protection. Options for construction include 4140 HT Steel with abrasion resistant coating such as UCAR*, Stellite* or Colmonoy* on screw flights. Note Stellite #6 should not be used on flight tips due to galling.

  CPM9V* Note For screws less than 35mm (< 1 inch) this approach may not be cost-effective.

- The screw flights should not be chrome plated since reinforced or filled composites will abrade or chip the plating in a short period.

- A square-pitch screw having a minimum L/D ratio of 18/1 is suggested. While "general purpose" type screws, supplied with most molding machines and having a compression ratio of 3.0:1 are suitable, compression ratios of 2.0–2.5:1 are preferred since they help reduce the amount of fiber attrition occurring and minimize shear heat input to the composites. A rapid transition (nylon type) screw should be avoided when molding LNP amorphous composites since they can impart excessive shear heat.

- A minimum of three separately controlled barrel heating zones should be employed plus separate control for the nozzle zone.

- A "free-flow" type check ring utilizing a fully-fluted tip is recommended. The free-flow non-return valve provides an unrestricted flow path and no "deadspots" to the melt. Fiber attrition will be minimized using a free-flowing tip, particularly in Verton* long glass fiber composites where maximum fiber length in the molded part helps achieve better strength and impact properties. Ball check valves and positive shut-off devices are not recommended when processing LNP composites due to their restrictive nature.

Reciprocating screw injection machine

Screw drive motor

Mold
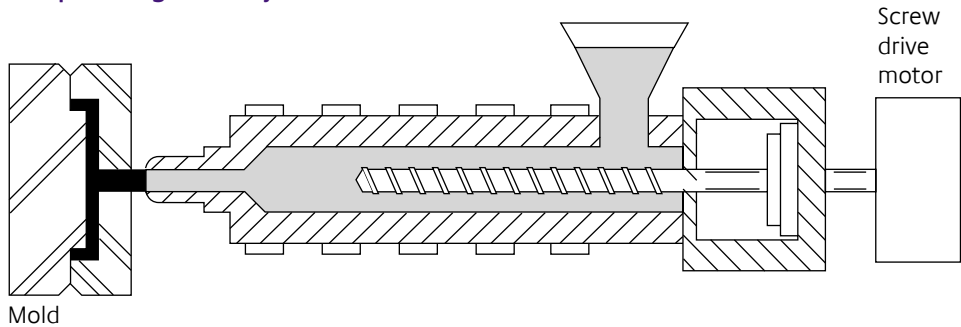
Screw types general purpose

Metering
4 flights

Compression
5 flights

Feed
6 flights

Compression ratio 2-3:1

- Suitable for all Thermocomp* composites
  (generally 2-2.5/1 for amorphous composites)

Nylon screw

Metering
4 flights

Compression
1 1/2-2 flights

Feed
9 - 10 flights

Compression ratio 3 - 4:1 minimum 15:1, length diameter ratio
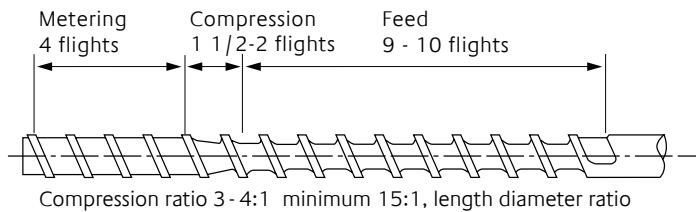
- Specifically for nylons
- Not recommended for amorphous composites

Enlarged "free-flow" non-return valve

Body / retainer

Ring

Seat

Radiused flow path
to reduce shear and
material hang-up

Matched interface
to prevent material
hang-up

Highly polished
surfaces to maintain
smooth flow along
with geroulsy radius
flutes

Large flow
through area
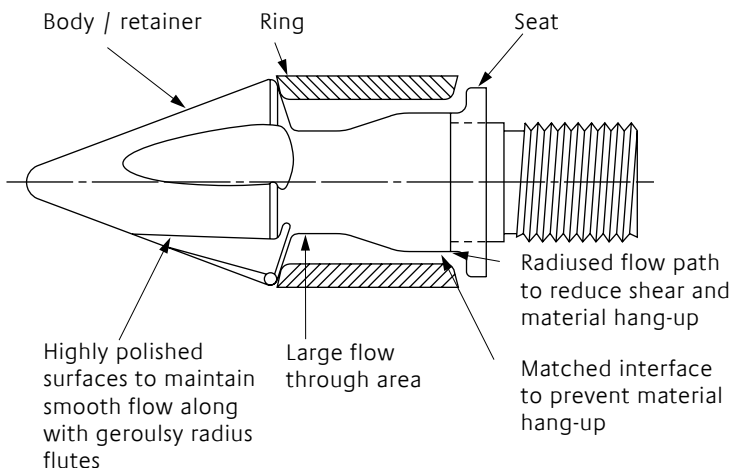
Non-return valves (screw tips)

Injecting ring

Sliding ring
Preferred for
Thermocomp
composites

Plasticizing seat

Pin

Ball

Ball type
Can cause
material
"hang-up"

Hole

Injecting ring

Free-flow
Preferred for
Verton* long fiber
composites

Plasticizing seat
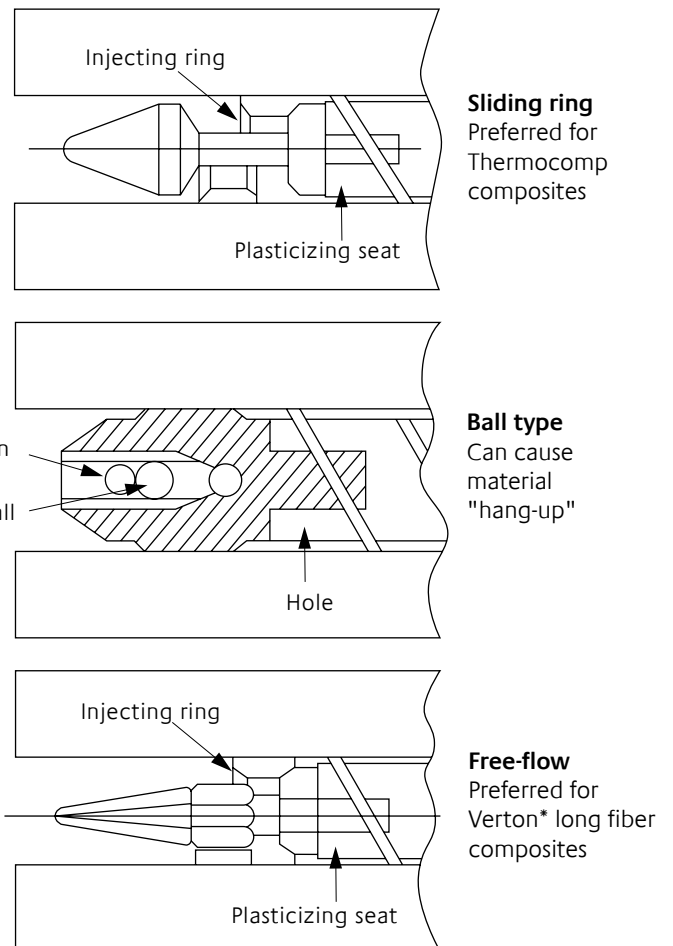
Nozzle design

Nozzles should be as short as possible and offer an unrestricted flow path. An open channel (full bore) design is preferred over a tapered channel (nylon type nozzle) in order to minimize pressure loss and shear heat to the composite. Nozzle exit-land L/D ratio should be 1:1 for a straight land. A reverse taper discharge can be added to provide sprue break-off within the nozzle. Positive shut-off nozzles should be avoided. The nozzle radius must match the radius of the sprue bushing where it will mate.

The discharge diameter for short nozzles should be a minimum of 0.187 (4.75mm) and 20% smaller than the sprue bushing diameter. Nozzles should be equipped with a full-length heater band and separate thermocouple control. Avoid variac type controllers since they lack the accuracy that proportional type controllers provide.

Mold temperature control

Accurate temperature control of the mold cavity walls is important and has a direct influence on the production of quality molded parts. Separate control of each mold half is necessary for best results.

Low mold temperatures will promote rapid cooling and higher output rates at the expense of molded part quality. Rapid cooling of the melt affects internal stresses, degree of orientation, post-molding shrinkage, warpage, and part surface quality negatively and should be avoided. Water, oil and or electric cartridge heaters are used depending on the LNP* compound molded as shown in the adjacent diagram below.

The mold should be insulated from machine platens in order to maintain uniform temperature control of the molding surfaces.

For more information on cooling, see page 27.

Nozzle design

General purpose

$^1/_2$" or $^3/_4$" radius
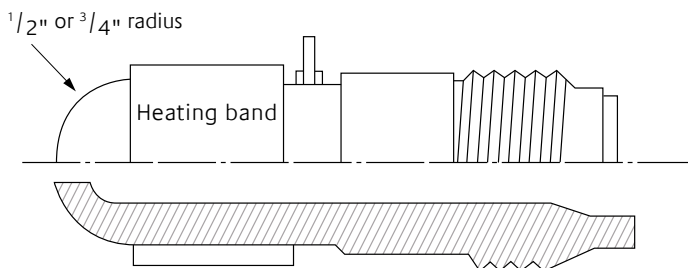
• Can be used for most Thermocomp* composites
• Low viscosity resins (e.g. Nylon) may need decompression/screw pull-back

Reverse taper

Heating band

2-3"
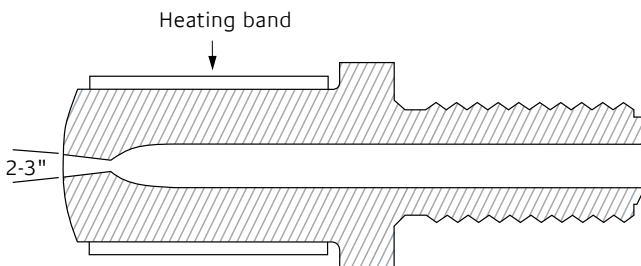
Mold temperature control (suggested)

100°F/38°C	Polyethylene
	Polypropylene
	Polyurethane
	PolyesterElastomer
	Polystyrene
150°F/66°C	Nylon 11
	Nylon 6/12
	Nylon 6/6
	Nylon 12
	Nylon 6
	Acetal
	ABS
	PBT
	SAN
	Nylon 6/10
	Super tough nylon
200°F/93°C	Polycarbonate
	amorphous nylon
250°F/121°C	PET
	PPS
300°F/150°C	PES/PAS
	PEI
	Polysulfone
350°F/177°C	PEEK
	PEK
	Melt processible fluoropolymers

Water

Oil or electric heat
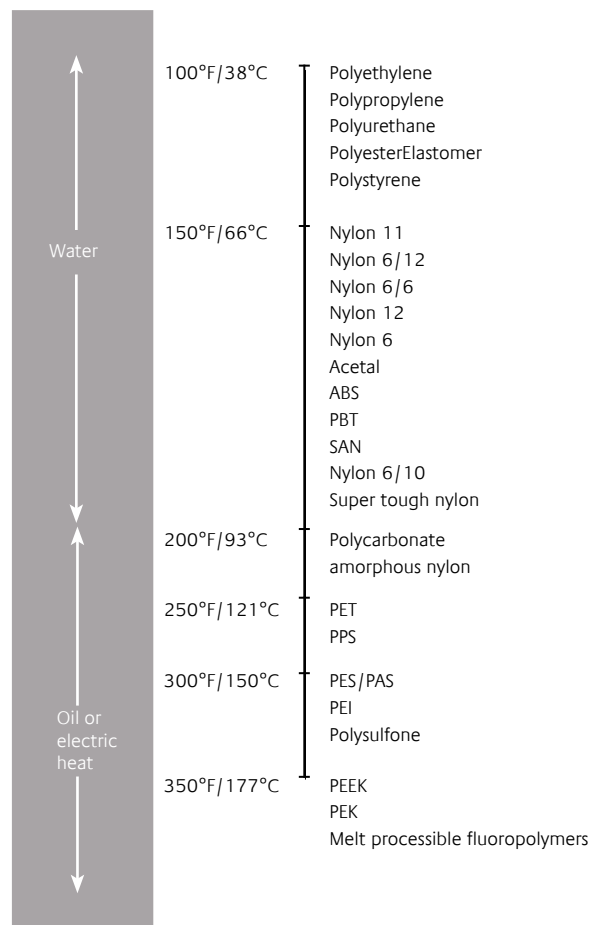
Material handling and preparation
Drying
Most LNP* compounds require thorough drying prior to injection molding to assure maximum properties in the molded part. The drying requirements followed for the base resins are generally the same for the compounds produced from them. Resins that are hygroscopic and must be dried include

- ABS
- PBT
- Polysulfone
- PPA

- Nylon (all nylons)
- Polycarbonate
- Polyethersulfone
- Polyetherimide

Drying is recommended for the compounds made from these resins since the pellets may retain surface moisture. Acetal processibility has demonstrated improved processing (less volatiles) after 2–4 hours drying at 200°F (93°C).

- Acetal
- Polypropylene
- PPO/PS

— Polystyrene
— HDPE
— SAN

The drying parameters listed below are applicable to all LNP compounds. Drying temperatures, however, vary with each resin type as illustrated in the table below.

- Minimum drying time 3–4 hours with a 0°F (-18°C) or less dewpoint
- A machine-mounted, dehumidifying hopper dryer equipped with a closed-loop circulating air system is preferred. When bulk drying, ensure resin remains dry via positive flow of dry air through the conveying sytem
- The hopper dryer should be equipped with a diffuser cone to ensure proper distribution of air flow and allow "plug flow" of material through the hopper
- Batch drying can be accomplished with air circulating, desiccant tray dryers with trays filled to a depth of no more than 1" (25mm). Material dried in this manner should be placed in a sealed hopper and residence time kept to a minimum

Conveying
LNP compounds are readily conveyed via vacuum hopper loaders. When a dry colorant is used, an in-line proportioner should be used below the hopper since dry colorant will clog desiccant hopper systems. Pre-blending of a pigment master batch should be done only when the concentrate is in pellet form. When Verton* is processed with a color concentrate, pellets should be sized to 0.4375 in. (11mm) to match Verton pellet size and minimize separation. LNP also recommends using LNP supplied color concentrates (Colorcomp*) for improved property retention.

1 drying required prior to molding
160°F – 170°F (71°C – 77°C)
Nylon 6/6
Nylon 6
Nylon 6/10
Super Tough Nylon
Nylon 6/12
Nylon 11
Nylon 12
ABS
SAN
225°F (107°C)
Thermoplastic Polyurethane
230°F (110°C)
Polybutylene Terephthlate
Amorphous Nylon
250°F (121°C)
Polycarbonate
275°F (135°C)
Polyethylene Terephthlate
300°F (150°C)
PEK
PEEK
PES/PAS
PEI
PSUL

2 drying recommended
(will prevent surface blemishes due to volatiles)
170°F (77°C)
Polyethylene
Polypropylene
Polystyrene
200°F (93°C)
Acetal
240°F (116°C)
PPO
250°F (121°C)
PPS
300°F (150°C)
Melt Processible Fluoropolymers

For more information on drying, see page 28.

Regrind
The use of reclaimed sprues, runners, and non-contaminated parts is permissible with most LNP composites depending on the color and physical property requirements of the finished part. Use a conventional scrap grinder equipped with 0.25 in. to 0.375 in. (6.4–9.6mm) screen. A maximum of 20% regrind can be blended with virgin material without appreciable loss of properties or color. If end use properties are important or critical part tolerances are required, the use of any regrind should be fully qualified prior to production. Be sure to keep regrind free of contamination, and dry before recycling. Blend regrind with virgin material before adding it to the hopper dryer. Regrind may require longer drying times as the reground compound will tend to absorb more moisture than virgin pellets due to the larger surface area of the regrind. For more information on regrind, see page 20.

Machine preparation
Prior to introducing LNP compounds, the molding machine barrel should be thoroughly cleaned either by purging or mechanically cleaning the cylinder with brass wool. If the screw is removed, clean it also and check carefully for nicks, cracks, or excessive wear. The check ring should also be inspected for any abnormalities. To check for cleanliness without removing the screw, purge with an unfilled, amorphous resin such as polystyrene or polycarbonate and look for foreign particles or discoloration in the air shot. Any evidence of contamination in the first several molded parts may suggest that the cleaning procedure should be repeated.

Process parameters

Typical start-up processing conditions are shown in the following table along with basic trouble shooting tips. A more comprehensive trouble shooting guide can be found in the last section of this brochure. Optimum parameters should be developed for each job from these initial settings. Set screw rpm to be within the cycle of your process. Excessive screw speed will damage glass fibers. A fast injection rate will promote the best surface finish in fiber reinforced compounds but may cause property variations in Lubricomp* and Stat-Kon* composites due to shear effects on the additives. Cavities should be filled as rapidly as possible to minimize fiber orientation and enhance weld-line integrity. Mechanical properties of reinforced compounds are optimized by packing the part as much as possible.

Higher than average mold surface temperatures should be used to maximize flow length and obtain a good surface finish. A hotter mold can be maintained for a reinforced compound without lengthening the cycle. When running a new tool for the first time with an LNP* composite, always start with less than maximum pressure and limit the shot size to avoid over packing the part. For more information on process start-up, see page 27.

Melt temperature

In general, melt temperatures for reinforced composites are 30–60°F higher than the unfilled resin. The compound stock temperature should be monitored during the normal processing cycle with a needle pyrometer by taking air shots through the nozzle. If a small shot size makes accurate readings difficult, take several shots and use the first purge to preheat the needle prior to subsequent readings.

Injection pressure

Injection pressure of 10–15,000 psi (70–105 MPa) is normally adequate for reinforced composites. Lower first stage pressure can be expected for unfilled compounds. The first stage (boost) pressure influences the surface quality, orientation, and mechanical stressing of the melt, therefore, excessive pressure should be avoided. Holding pressure in the range of 50–75% of first stage pressure will provide compression of the melt in the cavity for replication of the mold surface. Holding pressure should be set just high enough in order to prevent voids, sink marks and control shrinkage.

Back pressure

Screw back pressure should be kept as low as possible with 25–50 psi (170-345 kPA) usually being sufficient. The back pressure serves to displace air in the screw feed section, improve melt homogeneity, and facilitate heating of the melt. Adjustments to the back pressure will be apparent in the stock temperature developed. A surplus of back pressure will result in overheating of the melt and excessive fiber breakage.

Typical processing conditions for glass reinforced compounds

	Cylinder temp °F/°C	Melt temp °F/°C	Mold temp °F/°C.
ABS	400-480 / 205-250	500 / 260	180 / 82
SAN	400-520 / 205-270	500 / 260	180 / 82
Polystyrene	380-500 / 195-260	475 / 246	150 / 66
Modified PPO	450-600 / 230-315	570 / 299	200 / 93
Polycarbonate	540-630 / 282-332	600 / 315	200 / 93
Polyethersulfone (PES)	650-715 / 343-380	680 / 360	300 / 150
Polysulfone	620-700 / 327-370	660 / 349	300 / 150
Polyetherimide	650-715 / 343-380	680 / 360	300 / 150
Polypropylene	380-440 / 193-227	440 / 227	100 / 38
Polyethylene	380-440 / 193-227	430 / 220	100 / 38
Acetal	350-420 / 177-215	410 / 210	200 / 93
Polyester	430-500 / 220-260	470 / 243	225 / 107
Nylon 6	480-550 / 249-288	520 / 270	200 / 93
Nylon 6/6	500-570 / 260-299	560 / 293	225 / 107
Polyphthalamide (PPA)	575-625 / 300-330	625 / 330	275 / 135
Nylon 6/10	460-525 / 238-274	520 / 270	200 / 93
Nylon 6/12	460-525 / 238-274	520 / 270	200 / 93
Nylon 11	400-520 / 205-270	450 / 230	120 / 50
Nylon 12	380-520 / 193-270	450 / 230	175 / 80
Toughened Nylon	480-550 / 238-288	560 / 293	225 / 107
Amorphous Nylon	480-550 / 238-288	560 / 293	225 / 107
Polyphenylene Sulfide (PPS)	540-630 / 282-232	620 / 327	275 / 135
Polyetherketone (PEEK)	660-740 / 348-394	710 / 377	+300 / +150

Injection speed

In general, the injection fill rate should be as rapid as possible, especially for thin-walled parts. Thick walled parts should be filled using moderately slow injection rates. Excessive injection rates may cause undesirable high shear rates to be developed. Evidence of excessive fill speed might appear as gate blush, jetting, or discolored streaking.

Cushion

A minimal amount of cushion should be provided. Normally a range of 0.125"–0.25" (3.2-6.4mm) cushion will allow for adequate compensation of shot to shot variation. A minimal amount of cushion will also provide better pressure transfer on the melt, minimize over packing and prevent excessive shrinks and voids.

Screw speed

A screw speed between 30–60 rpm is sufficient for most LNP compounds. Slightly higher rotation speeds can be used for small diameter <1.5" (38mm) screws. Optimum screw speed selection can be reached by adjusting screw return stoppage to occur just prior to the mold open sequence. High screw speeds can result in over heating of the melt and increased residence time in the molding machine.

Nozzle temperature

Full-length nozzle heater bands should be controlled separately from the front barrel zone. Closed-loop, thermocouple controlled rather than a Variac-type heater will help achieve a uniform melt temperature. The nozzle temperature setting should approximate the desired melt temperature and may be lower for semi-crystalline compounds (i.e. nylons, acetal). Drooling and nozzle shear heat compensation is regulated by adjustments to the nozzle heater. Melt decompression may also be used to reduce drooling.

Mold temperature

Individual and accurate control of each mold half in the ranges shown on the process chart assures the best results. The effects of lower mold temperature outlined under the Mold Temperature Control section should be kept in mind. The effects of the high range of mold temperature options is shown below.

Large cores or small diameter core pins may require lower temperatures or special cooling control to aid part ejection (prevent sticking).

Higher mold temperature

Increases	Decreases
Crystallinity (crystalline resins)	Molded in stresses (all resins)
Shrinkage (all resins)	Impact strength (crystalline resins)
Heat distortion temperature (crystalline resins)	

Cycle time

Cooling time is the major portion of the total molding cycle. The cooling requirements are dependent on the part thickness and the level of filler in a composite. Glass fiber and carbon fiber reinforced LNP* composites will cool faster than unfilled compounds due to the higher thermal diffusivity of these materials.

Mold release agents

The application of external mold release agents to tool surfaces should be kept to a minimum. Initially, part sticking can be corrected by making adjustments to pressure, temperature or cycle time. In addition, draft angles, mold surface finish, and ejector pin area may require attention. Intermittent part sticking can often be overcome by moderate application of a mold release spray. Since some mold release sprays may have a detrimental effect on the properties and surface appearance of LNP composites, compatibility testing is necessary prior to their use. In some cases, formulation adjustments are possible, permitting internal release agents to be added to the composite during compounding. Internal releases will reduce the tendency for part sticking.

Regrind

Details on the addition of regrind to virgin material can be found in the Material Handling Section. Remember to keep regrind clean and discard any degraded, discolored or contaminated parts.

Recommended purge materials for LNP compounds

The proper selection of a purge material for a given compound depends highly on the melt temperature of base resin in the compound you wish to purge. The following table is a guide for purging compounds with base resins that fall within the following melt temperature range. Refer to table on page 13 for base resin melt temperature.

Melt temperature range of compounds base resin	Recommended purge material
550°F / 288°C and below Melt temperature	HDPE, GP polystyrene or ground cast acrylic
550°F–650°F / 288°C–340°C Melt temperature	HDPE or polycarbonate
650°F / 340°C and ABOVE Melt temperature	HDPE must be extrusion grade w/ melt index 0.3–0.35 g / 10 min. Glass reinforced polycarbonate

When purging dark compounds a 20–30% glass filled version of the above recommended purge material will help scour the barrel and will aid in removing the pigment residue. The glass reinforced version should be followed by its unfilled analog to eliminate any glass residue that may remain.

When purging high heat materials (>550°F / >288°C), begin purging at processing temperatures and reduce barrel temperatures to approximately 500°F (260°C) while continuing to purge. Once barrel temperatures have stabilized at 500°F (260°C), use one of the recommended low temperature (≤550°F / ≤288°C) purging materials as the final purge. This will help reassure that virtually all traces of the high heat polymer are removed, therefore reducing the possibility of contamination (black specs) in future molded parts.

Shutdown procedure
The recommended procedure for shutdown after molding LNP* composites differs depending on the length of shut down.

Complete shutdown
At the completion of the molding run, all traces of compound should be thoroughly purged from the barrel with HDPE, general purpose styrene or commercially available purging compound. Be aware that fumes from these purging materials will intensify when used after a high temperature LNP composite due to the higher barrel temperatures employed, which may be above 550°F (288°C). Dry, high-molecular weight polycarbonate is more stable than the above materials and may be used to facilitate more efficient barrel purging when barrel temperatures are above 550°F (288°C). When purging begins, the barrel heats should be lowered gradually to approximately 420°F (215°C). After the purge appears clean, the screw is left in the forward position. Barrel and nozzle heats can then be turned off.

Breaks in production
When it becomes necessary to halt production for short periods of time, the cylinder should be purged at regular intervals to prevent degradation of the LNP composite. If a longer delay is anticipated (i.e. 15 minutes), shut off the resin feed at the hopper, lower barrel temperatures, and thoroughly purge the cylinder.

Material changeover
Immediate changeover to a dissimilar compound requires complete purging of the plasticating cylinder. If any doubt exists about compatibility of trace amounts of LNP composite and the new compound, mechanical cleaning is advised.

LNP* Thermocomp* resin

Reinforced and filled Thermocomp composites are readily processed on reciprocating screw injection molding machines. Slightly higher injection pressures and barrel temperatures can be anticipated due to the lower melt flow (higher viscosity) of reinforced compounds compared to unfilled engineering resins. Experience has shown that marginal increases in pressure and/or melt temperature will help control compound melt viscosity and facilitate filling thin-walled cavities. Suggested start-up processing conditions can be found on page 13 (Typical Processing Conditions for Glass Reinforced Compounds).

A summary of processing guidelines for standard filled and reinforced Thermocomp composites can be found below. Additional considerations for special Thermocomp materials follows this section.

- In-line reciprocating screw injection molding machines work best.
- Reinforcements and fillers will help to promote shorter cycles with Thermocomp composites.
- Utilize 40–70% of the rated barrel capacity for best results.
- Nozzles should be heated and provide an unrestrictive flow path. Open channel (full bore) nozzles are preferred over tapered (nylon) type nozzles. A reverse taper discharge design will assist sprue-break.
- "Free-flow" type check rings are preferred. Positive shut-off devices should be avoided.
- Sprue bushings need to be large, well-tapered, and highly polished to promote mold release.
- Short, full-round runners will help maximize cavity pressure and improve part appearance.
- Fast injection rates serve to maximize physical properties and part surface finish.
- Low screw recovery RPM (40–70), and minimal back pressure 25 psi (170 kPa) will help to minimize fiber attrition.

Gating

Gate size and location are extremely important in controlling shrinkage and molded-part dimensions. The following guidelines provide direction toward gating parameters.

- Locate gate to achieve a balance of flow in all directions, or to achieve flow along the axis of the most critical dimensions.
- Gate into the thickest section of the cavity so material flows from thick to thinner sections.
- Gate thickness should be not less than 50–75% of the adjacent wall, and gate width two to three times the thickness. Tunnel gates should be no smaller than 0.040" (1mm) diameter.
- Gate land length should be 0.060"(1.5mm) maximum.

Regrind

The reinforcing effect of glass or carbon fibers decreases with repeated processing because of fiber breakdown. Also, thermal degradation will cause a decrease in mechanical properties, particularly impact strength. Where molded components need to meet specific demands on tensile or impact strength, the use of regrind is not recommended until thorough testing is carried out. Fiber breakdown and thermal degradation can also effect the dimensions of molded parts. Care should be taken with the addition of regrind on parts where tight dimensional tolerances must be met. Should the use of regrind be possible, always use a set ratio of regrind and virgin material.

A maximum of 20% regrind should not be exceeded if part physical properties are important.

For more information on regrind, see page 20.

Flame retardant compounds

LNP's flame retardant composites demonstrate excellent injection molding processibility. In those thermoplastics which are not inherently flame retardant, the flame retardant additives act as flow aids and increases the flow length of the material. This enables the molder to fill thin-walled parts and to achieve a good molded part surface finish. Excessively high cylinder temperatures and long residence times can destroy flame retardancy and cause mold corrosion. Molds designed to run flame retardant compounds should be safeguarded against possible mold corrosion with a surface treatment such as chrome or electroless nickel. To minimize the possibility of developing corrosive by-products, these compounds should be molded with stock temperatures as low as possible. LNP FR compounds can be molded with melt temperatures of 50–100°F (10–38°C) lower than the standard, non-FR, glass fiber reinforced equivalent due to the increased flow that FR additives impart.

Proper pre-drying of FR composites is critical too since these materials are more sensitive to both hydrolytic and thermal degradation than non-FR compounds. Regrind levels should not exceed 10% and should be fully qualified to ensure that mechanical, dimensional and agency (UL/CSA) requirements are maintained.

LNP* Thermocomp* HSG compounds

High Specific Gravity compounds require special consideration when injection molding due to the low volume of base resin in the composite. With specific gravities up to 10, HSG compounds contain a high weight percent of environment-friendly fillers that give molded parts a solid, "metal" feel. It is critical to maintain low melt temperatures with HSG composites (compared to melt temps used when molding "standard" products), since excessively high melt temperatures may cause separation of the filler and resin possibly leading to screw seizure. The melt emanating from the barrel nozzle should appear stiff and homogenous. If full parts cannot be achieved after initial start-up, gradual increases to fill pressure are suggested rather than increases to compound stock temperature.

Materials for screw, barrel, and mold construction with Thermocomp HSG compounds are the same as those for all other LNP compounds (covered in previous sections). The fillers used in HSG to impart high specific gravity are no more abrasive than glass fibers and our experience has shown that screw and barrel wear can be minimized by following the guidelines suggested in the Barrel and Screw features—Machine Considerations. Since HSG materials are prone to fast set-up, small pin point gates—<0.040" are not recommended since this type of gate can lead to premature gate freeze-off. Hot runner systems work well with HSG compounds providing the guidelines on page 6 are followed. The high thermal conductivity of HSG compounds requires that individual hot runner components provide good thermal insulation properties—especially in the nozzle tip area - in order to prevent temperature control fluctuations.

Melt processible fluoropolymer

Injection molding of LNP compounds based on FEP, E/TFE, PFA, PVDF, and E/CTFE should be carried out on reciprocating screw equipment. Corrosion resistant materials should be used for processing equipment that comes in contact with the molten fluoropolymer compound. Mold cavities should be protected with a corrosion resistant surface treatment as well. Good temperature control of barrels and molds is important since melt temperatures in excess of 700°F and mold temperatures up to 500°F can be anticipated when molding the MPFP compounds.

Suggested molding conditions for LNP MPFP compounds are shown in the table below. Before initial startup, thorough heat soaking of all equipment should be allowed to ensure optimum temperatures are met prior to plasticating resin. Screw RPMs should be slow (40–60) to minimize shear heating of the compound.

Molded parts will have a rough or frosty appearance if injection speed is too fast, and a rippled surface if too slow. The surface effect of reinforcements and additives should not be confused with melt fracture, which is caused by too rapid an injection rate. The critical shear-rate at which melt fracture occurs is temperature- and pressure-dependent. Increased melt and/or nozzle temperature will reduce chances of melt fracture. A fast injection rate in the case of FP-E, FP-C, and FP-V series compounds, combined with proper mold temperature will typically produce the best surface. For FP-F and FP-P series, a smooth surface results from slower fill rates and correct mold temperature. Extremely hot molds should not be used for thick-wall sections, since increased post-mold shrinkage will result.

Clean out and shutdown procedures are similar to those for standard LNP compounds, except that complete removal of the screw is recommended unless another MPFP production run is scheduled to follow immediately. After thorough purging, remove the screw and clean all parts with a wire brush.

Base resin	FEP	PFA	E/TFE	E/CTFE	PVDF
Series name	FP-F	FP-P	FP-E	FP-C	FP-V
Equipment factor					
Rear cylinder, °F/°C	600-625 / 315-330	600-650 / 315-343	525-575 / 274-302	510-530 / 265-277	380-420 / 193-216
Center cylinder, °F/°C	625-650	625-650	575-625	520-540	400-440
Front cylinder, °F/°C	650-680	650-725	575-625	530-550	430-450
Nozzle, °F/°C	680-720	700-725	625-650	550	475
Mold temperature, °F/°C	200-400	300-500	Ambient to 375	Ambient to 225	Ambient to 200
Stock temperature, °F/°C	625-700	625-67	575-625	540-550	400-450
Injection speed	Slow	Slow	Moderately fast	Moderately fast	Moderately fast

LNP* Verton* compounds

Verton long fiber composites contain glass fibers approximately 11mm in length. In order to help achieve maximum performance in the molded part, the following guidelines are recommended

- Materials of construction for injection molding screws, barrels and tooling with Verton are the same as standard glass fiber reinforced LNP composites

- Design of the IM screw and screw-tip is critical to reduce the amount of glass fiber attrition that occurs in the molding machine. A screw compression ratio of approximately 2.5:1 (maximum) together with a "free-flow" screw tip will help reduce fiber breakage. Mixing screws are not recommended.

- The molding machine nozzle, sprue bushing, and tool runner system need to be constructed with generous dimensions, no sharp corners and fullround runners to achieve best results. Hot runner systems have been used successfully with Verton composites. Open gate drop designs work best.

- Minimize screw speeds and back pressures in order to help reduce fiber attrition. 30–70 RPM and 25–50 psi (170-350 kPa) screw speed and back pressure (respectively) are recommended.

- A minimum gate thickness of 0.100" (2.5mm) is recommended for Verton composites in order to minimize glass fiber attrition and achieve optimum part physical properties.

LNP Lubricomp* and Lubriloy* compounds

LNP compounds that contain lubricant fillers and/ or reinforcements are injection molded using the same guidelines as the Thermocomp* composites. Particular attention to the following details will ensure that wear properties are optimized

- Thorough drying of Lubricomp composites is critical since moisture will promote excessive volatiles and possible separation of the lubricant additive.

- Generous, unrestrictive flow paths through the runners and gates will ensure parts are molded with optimum dispersion of the lubricant additive. Avoid pinpoint gates.

- Moderate injection rate is suggested to reduce the shear heating of the material.

- Good venting practices are important with Lubricomp and Lubriloy composites, particularly those containing PTFE. Vents should be incorporated into the runners, core and cavities, and the intersection of flow fronts. Ejector-pin vents will permit blind holes to be vented.

LNP* Stat-Kon* compounds

LNP compounds that contain conductive fillers or reinforcements are injection molded using the same guidelines as the Thermocomp* composites. Particular attention to the following details will ensure that electrical properties are optimized

- Stat-Kon compounds based on hygroscopic base resins such as polycarbonate, nylon, or polyester require very thorough drying since degradation due to moisture makes these impact sensitive compounds too brittle for some applications.
- Generous, unrestrictive flow paths through the runners and gates will ensure parts are molded with optimum dispersion of the conductive additive.
- Excessive shear heating of the material due to restrictive runners or gates will promote separation of the conductive additives possibly leading to poor part-surface finish and/or inconsistent electrical properties.
- Careful control of compound melt temperature is needed to prevent thermal degradation and further loss of impact of these impact sensitive compounds.
- Low screw recovery RPM (30–70), and minimal back pressure 25 psi (170 kPa) will help to minimize fiber attrition.

LNP Faradex* and EMI-X* compounds

LNP composites that contain conductive fibers to impart shielding characteristics can be processed on standard injection molding equipment using the guidelines for standard grades. Attention to the following details will help ensure optimum shielding properties

- Pre-compounded carbon fiber composites demonstrate optimal shielding performance utilizing moderate to fast injection rates, low back pressure and low screw speeds to minimize fiber length attrition.
- Stainless steel fiber dry blends perform best using slow to moderate injection rates, 50–200 psi (345-1380 kPa) back pressure, and moderate screw speeds to optimize dispersion.
- Hopper magnets should be removed prior to molding Faradex composites which are dry-blends.
- Tooling materials for Faradex compounds follow the same guidelines as standard LNP reinforced composites.

LNP Stat-Loy* compounds

Stat-Loy compounds are injection molded using the same guidelines as standard LNP composites. Attention to the following details will help ensure optimum dissipative properties in the molded part

- The alloy components that impart permanent anti-static properties to Stat-Loy also lower the base-resin viscosity and provide good flow characteristics for injection molding.
- Lower melt temperatures and lower injection pressures (compared to a standard GF reinforced grade) are typical for Stat-Loy composites due to easy flow properties.
- Excessive injection rate should be avoided since high shear rates may effect part surface cosmetics.
- Avoid small gates with Stat-Loy due to the shear sensitivity of the compound. Best results are found with a minimum gate thickness of 0.060" (1.5mm).
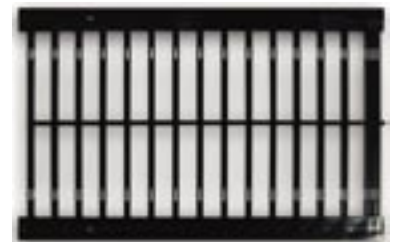
Gas-assist molding (GAM)

The GAM process or gas injection molding process has been used successfully with LNP* composites to mold hollow bodied parts. Hollow gas injection moldings are produced by the controlled injection of an inert gas (N_2) into the molten polymer melt compound through the runner system or into the part. The gas forms a continuous channel through the less viscous, thicker sections of the melt. Some of the many advantages of gas-assist molding include

- Curved hollow sections
- Weight reduction
- Cycle time reduction
- Elimination of warpage
- Increased stiffness and functionality

The key parameters of the hollow gas molding process include the following

- Gas packing pressure
- Gas delay time
- Gas hold time

Gas packing pressure

will vary depending on part geometry. In our experience you should be in the range of 400–800 psi.

Gas delay time

is the time that gas injection is delayed from the start of plastic injection. Our experience has shown this to be true when the cavity has achieved a degree-of-fill of approximately 75%.

Gas hold time

is the time gas pressure is applied to the molding after plastic injection. This time is also referred to as the gas packing time.

Injection compression molding (ICM)

ICM is an injection molding technique where the melted polymer is injected into a partially open mold. The mold closes, compresses, and distributes the melt throughout the cavity, thus completing the filling and packing stage. Compression can be either simultaneous with, or sequential to injection of polymer.

The ICM process is advantageous to molding Verton* long glass fiber reinforced composites since it helps preserve the fiber lengths and thereby increase the physical properties of the finished part.

Additional features of ICM include

- Weld-line strength improvement
- Significantly lower injection pressure
- Reduces residual molded-in stress
- Allows down-gauged thinner walled parts
- Facilitates one-step molded-in cover-stock laminations

ICM requires modification of existing equipment or selection of the ICM option on new machines. Equipment requirements include precise clamp positioning, accurate shot size control on injection, and speed control of the secondary clamp or compression action. Additional software and monitoring equipment may also be required depending on age of existing equipment. LNP has customer support facilities in Exton, PA that can assist customers with the following

- Part design engineering
- Processing expertise
- Injection press conversion expertise
- Technical service / mold trial assistance

Thin wall injection molding

It is generally accepted that a thin wall part using an engineering resin, is defined as having a wall thickness between 0.020" (0.5mm) and 0.080" (2mm), and a flow length to thickness ratio (L/T) greater than 75. At 0.040" (1mm) this equates to at least 3" of flow. This definition is challenging for most engineering composites, especially amorphous resins such as polycarbonate. The processing, tool design and part design rules for thin wall parts change since thinner wall sections dissipate heat rapidly during injection creating a higher skin/core ratio. This translates into extremely high fill pressure, 15,000–35,000 psi (100–240 MPa) and extremely short fill times (<0.75 sec.) to produce the part. The consequences of high fill pressures, applied for very short times, require that molding equipment have

- High pressure hydraulics for injection and clamp (accumulators and high torque pumps may be required). Fill pressure of 30,000 psi (200 MPa) can demand 4–6 tons/inch² of clamp.
- High resolution sensors to control hydraulics.
- Rugged machine with platens thick enough to limit deflection to 0.0005" (0.013mm).
- Possibly a smaller barrel to limit shot capacity to 40–70%.

The consequences of high pressure on tooling are

- Tool deflection must be minimized with thicker plates (support plates 2–3" / 50–75mm), extra support pillars, and a more rugged assembly.
- Cavity-core shifting needs to be studied. Core interlocking mechanisms may be required to prevent core deflection.
- Extra venting is necessary. Vacuum venting may allow faster fill speeds to be achieved.
- A high flow melt-feed system, together with numerous large gates will promote easier fill and reduce core shift. Hot runners are almost mandatory.
- High quality hardened tool steels are required.
- Ejector pin diameter should be large (2X normal) and numerous to facilitate part ejection. Low friction mold coatings will promote part ejection (see page 6).
- Thermal expansion of the entire tool needs to be equal. This requires like steels for cavity/ core plates and possible heating/cooling of ejector plates.

Structural foam molding (SFM)

Thermoplastic structural foam molding is an option with several LNP* composites that can provide much larger parts having higher strength-to-weight ratios and better dimensional stability than solid injection-molded parts. Tooling costs can be lower and in some cases, aluminum molds can be used with thermoplastic SFM.

Experience has shown that SFM works best with glass and carbon fiber reinforced Thermocomp*, Stat-Kon* and Verton* composites as well as certain specialty grades of unreinforced Lubricomp* and Colorcomp. Chemical blowing agents (CBA) are typically used to achieve density reductions up to 30%, sink control in thick part areas, or warp reduction.

LNP can supply (exothermic) CBA concentrates — Foam-Kon* 20 and Foam-Kon 28 — that are blended with molding compounds at levels between 2–3% to produce parts with a solid-skin/ foam-core structure. Highest density reduction is achieved with a minimum wall thickness of 0.250" (6.35mm). Endothermic CBAs have also been used successfully with LNP composites and have shown some benefits over exothermic CBAs.
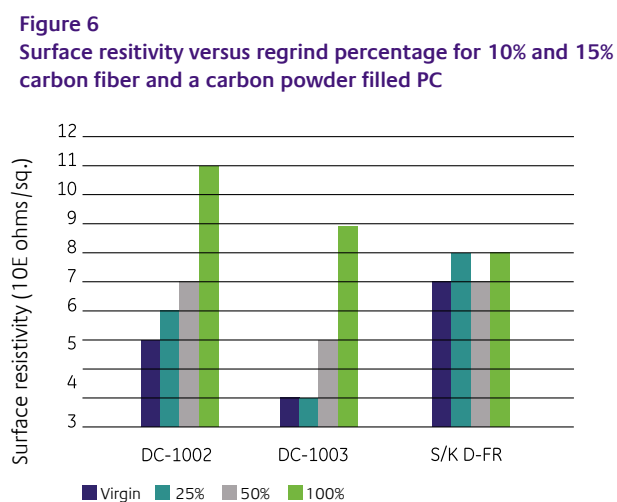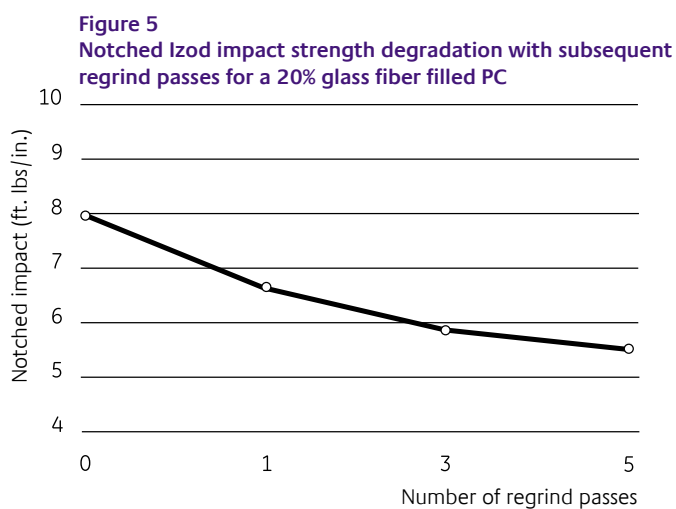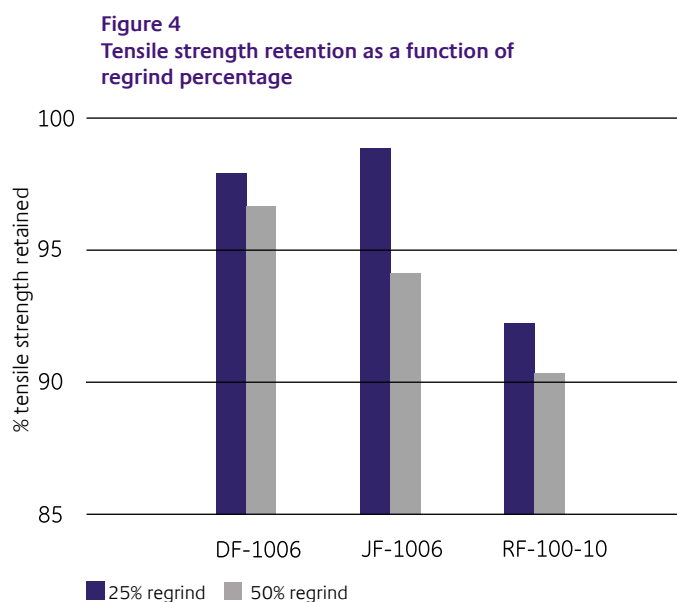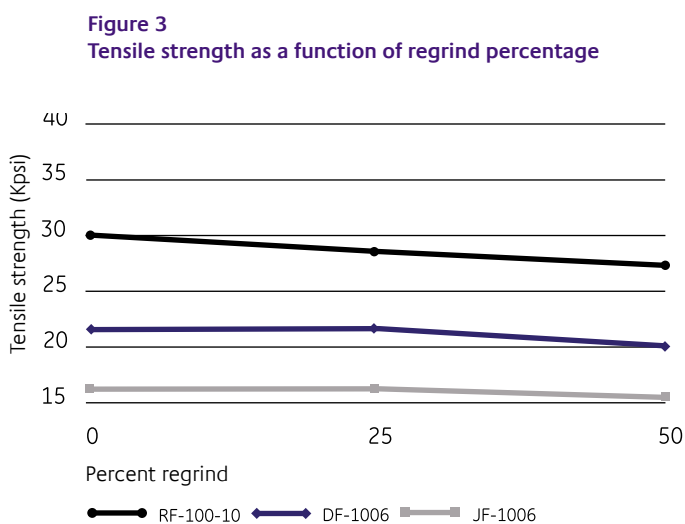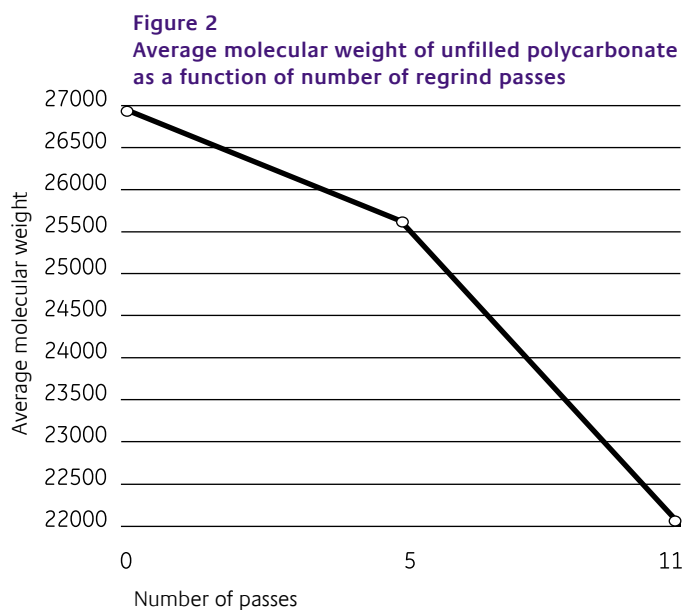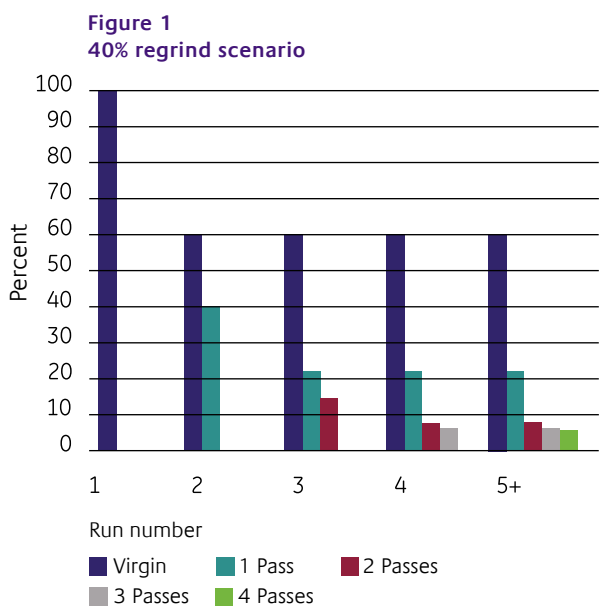
Regrind

The use of regrind in parts made from thermoplastic composites provides some obvious economic and environmental benefits. The question most customers have is what level of regrind is appropriate for a particular material? Unfortunately, the answer is not as simple as running a series of experiments on test specimens and picking an acceptable level of physical property degradation. Each part is unique. When a specific part design and thermoplastic material is chosen and approved there exists some margin of safety between the properties by design and those that result in part failures. Thus, regrind usage must be limited such that the properties of the part do not dip to the point of failure. What can be learned from laboratory tests involving regrind is its effect on the physical and electrical properties with respect to a variety of material and filler combinations.

The first step in this study involves an understanding of the regrind process. Most high volume applications that generate sprues and runners generate a continuous source of regrind. Thus, the regrind generated (label it run 5+) is a blend of virgin material and regrind from an earlier run (run 4). That regrind (from run 4) is also a blend of virgin material and regrind from even earlier (run 3) and so on. Figure 1 shows how the various generations are blended in a continuous 40% regrind situation. In this situation, 6.4% of the material has been through three or more passes. Table 1 lists the percentages of material that has seen three or more passes for a variety of regrind rates.

As the thermoplastic composite goes from its first pass to its nth pass, there are two basic modes for degradation. The first is the polymer degradation that results from multiple heat histories. The second is the degradation that the filler experiences as a result of the additional heat histories, and from the physical destruction that results from molding and grinding operations.

Table 1

	Regrind rate					
	10%	20%	25%	30%	40%	50%
Percentage of material with three or more passes	0.1	0.8	1.6	2.7	6.4	12.5

Figure 1
40% regrind scenario

Figure 2
Average molecular weight of unfilled polycarbonate as a function of number of regrind passes

Figure 3
Tensile strength as a function of regrind percentage

Figure 4
Tensile strength retention as a function of regrind percentage

Figure 5
Notched Izod impact strength degradation with subsequent regrind passes for a 20% glass fiber filled PC

Figure 6
Surface resitivity versus regrind percentage for 10% and 15% carbon fiber and a carbon powder filled PC

Each polymer reacts to heat histories in a unique manner. Many undergo chain-scission resulting in lower viscosity and reduced physicals. Other polymers crosslink resulting in an increase in viscosity. Regardless of the mechanism, they all react at different rates. In an experiment of 100% regrind polycarbonate, the molecular weight was measured as a function of the number of regrind passes (Figure 2).

The molecular weight continually decreases with an increasing number of passes. These data were based on properly dried polycarbonate. Polymers that have accelerated degradation rates when exposed to water at processing temperatures can be expected to degrade at a much faster rate. Since all chemical reactions are a function of time and temperature, the degree of degradation can also be expected to be a function of the processing temperature and the residence time. Glass reinforced grades of polycarbonate will experience higher rates of molecular weight degradation relative to the unreinforced grades as a result of viscous heating.

The physical property loss of the polymer that resulted from the thermal degradation will be imparted to the thermoplastic composite. Thus, in polycarbonate, properties that are a function of molecular weight, such as impact strength, will be lower in regrind blends.

Fillers that are sensitive to temperature will behave in a similar manner to the polymers. For instance, flame retardant are chemicals that undergo a reaction that is time and temperature dependent. It would be expected that materials that contain flame retardant would see a continuously diminishing level of retardancy with each regrind pass. The extent of the degradation would be different for each formulation.

Most fillers, however, are not sensitive to temperatures at which the polymers are processed. Their susceptibilities to the regrind process are primarily a result of the physical destruction of the filler. Fillers that are particulate in nature will not be as severely affected by the regrind process. Glass, carbon, stainless steel and aramid fibers will experience a much greater degree of destruction than PTFE, carbon powder, or mineral fillers. Within the fiber filler's group, carbon fibers, being the most brittle, will show greater fiber length attrition than will aramid or stainless steel fibers. Composites compounded with fibers that provide minimal reinforcement, such as stainless steel, will not experience as much property degradation as those that contain highly reinforcing fillers. Again, the extent to which a thermoplastic composite will exhibit loss of physical properties will be dependent on the filler type and the property in question.

Figures 3 and 4 show the loss in tensile strength with increasing amounts of regrind. The data used in these examples utilized single pass regrind only. Regrind blends from continuous regrind operations will experience greater fiber attrition and increased loss of tensile strength. Data have shown that fiber length attrition occurs in the first few regrind passes. As the fibers get shorter, the likelihood that they will be broken in future regrind passes diminishes (see Figure 5). Thus, while fiber length attrition is asymtotic in the first few passes, polymer degradation is continuous.

Fillers that are not affected by regrind operations will not see any appreciable shift in the properties it imparts to the composite. Stat-Kon* D-FR is a carbon powder based polycarbonate. Since the carbon powder will not be affected by the regrind process, the surface resistivity it imparts to the composite will not change. Whereas, the carbon fiber based products Stat-Kon DC-1002 and DC-1003 will see increased surface resistivity as a result of the fiber attrition (see Figure 6). Similar retention of wear and limiting PV properties can be expected with unreinforced, PTFE and or silicone filled polymers such as Lubricomp* DL-4040 and RL-4040 (there will continue to be some physical property loss as a result of polymer degradation as previously discussed).

Now that the effects of using regrind in thermoplastic composites have been discussed, the formulation for determining safe amounts of regrind is further complicated by the huge variation in the injection molding and regrind processes from application to application. Factors such as residence time, back pressure, screw speed, injection speed, injection pressure, gate size (resulting in shear heating), melt temperatures and water content all can affect the heat history of the polymer. Similarly, back pressure, screw speed, injection speed, injection pressure, gate size, melt temperatures, runner size and length, screw configurations, grinding equipment and the grinder sieve size all can affect the amount of fiber length attrition. The only method for determining a safe amount of regrind is through thorough analysis of the entire process and a complete understanding of which properties are critical to the application.

With all of the changes in the composite's molecular weight and fiber filler's length with increasing usage of regrind, expect the process variation to increase as well. This variable is very difficult to determine. It takes time and a large number of lots to determine the extent of the effect.

Material selection can also be manipulated to increase the success rate of using regrind in an application. For example, if a particular part generates 40% waste that is intended for use as regrind, but tests at 40% regrind levels have shown that the loss in impact strength leads to part failures, recommend a higher glass loading or high impact grades to compensate for the expected loss in impact strength. Similarly, in conductive composites, carbon powder can be recommended to minimize the increase in surface resistivity.

When an opportunity for using regrind in an application exists, carefully examine all of the factors that will affect the properties of the composite. Compare them with the requirements of the application. Make material adjustments if necessary and put it to work. The economic benefits are hard to ignore.

Mold cooling

In order to remove a molded part from the mold, the material must be sufficiently cooled to provide ejection without distortion.

Adequate mold cooling can be considered to have occurred if the part surface is hard enough to prevent ejector pins from penetrating.

Proper mold cooling involves
- The ability to uniformly control the rate of cooling
- Getting the heat energy out of the material and into the mold
- Conducting it away by a circulating fluid

Mold cooling guidelines
- Cooling rate is thickness dependent
- Put cooling lines exactly where needed
- Cooling fluid must be close to the surface
- Design cooling system first — add ejector pins later
- Insure turbulent flow of cooling media for maximum heat transfer. Sharp turns in the cooling circuit and high speed (volume of flow) will promote turbulent flow
- If possible, channel each cavity and core
- Cooling should be uniform in each mold half

### Injection molding set-up and process optimization procedure
The following method has been found to be useful to determine the optimum molding parameters for injection molded parts. It is often referred to as a mold filling/gate freeze-off study. The following steps should be followed after establishing the correct melt and mold temperature

1. Set mold heater so that cavity and core surface temperature is in the correct range for that material. Measure and record mold surface temperature with hand pyrometer.

Molding techniques
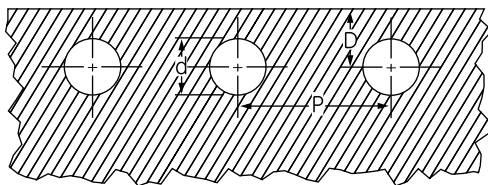
d - Diameter of water line
D - Depth of water line (D = d to 2d)
P - Pitch (P=3d to 5d)

Baffle

Highly conductive insert

Blade

In

In
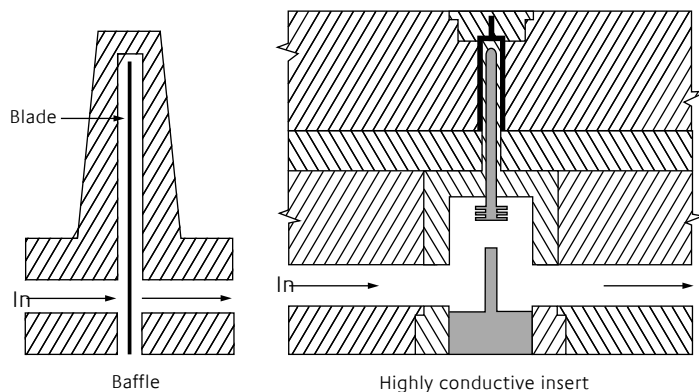
2. Set barrel heaters to a ramp profile, lower at the feed throat and higher at the front zone. Load material and make several air shots. Check melt temperature with needle pyrometer. Adjust barrel heater set-points and screw back pressure accordingly to achieve a melt temperature in the correct range for that material. Measure and record the melt temperature.

3. Set injection pressure to the high limit so it does not influence injection speed negatively.

4. Set injection speed to maximum for crystalline materials, medium fast for amorphous materials.

5. Set holding pressure and time to zero.

6. Start making short shots without any cushion, then incrementally increase shot volume until the part is approximately 95% full. This becomes the transfer point (time). Begin weighing parts.

7. Set packing pressure 20%-30% below first stage pressure and continue weighing parts (use only 1-2 sec. pack time). Increase shot volume until a cushion is developed. Incrementally increase pack time until part weight does not increase. At this point, gate freeze-off has been reached.

8. Set the cooling time to permit the parts to be ejected without the ejector pin pushing into the parts.

Be sure to record the process conditions and mark the parts accordingly in order to correlate part performance and dimensional stability with a specific set of process parameters.

Doing it right drying
One secret of successful processors is proper drying. If material isn't properly prepared for molding, it literally becomes "garbage" in the barrel because the moisture breaks down the polymer at the temperatures required to process the plastic. This can occur for many different polymers, including
- Nylon (all types)
- Polycarbonate
- Polyesters (PBT, PET, etc.)
- Polysulfone, PES, PEEK
- Polyurethanes, etc.

There are 5 keys to remember to insure good drying
- Pellet/regrind incoming moisture level
- Air temperature
- Dewpoint/dessicant
- Residence time
- Air flow

Incoming pellet moisture
To afford repeatable drying performance, it is important to prevent wide swings in incoming moisture. While this is largely the manufacturer's concern, the processor must be aware of procedures that can influence this. Key among these are storage and regrind usage.

Regrind of hygroscopic materials will continue to pick up moisture if stored while exposed to the atmosphere. In summer, with the increase in humidity, it is all the more important to properly store regrind to prevent excessive moisture pickup. The best way to use regrind is to recycle it at the machine using a grinder with a proportioning loader. In this way, the regrind will not be "contaminating" the pellets with excessive moisture, as can happen if regrind sat for months in a warehouse.

Temperature
The important thing to remember is that the temperature that matters is not at the output of the dryer, but at the input of the hopper. Make sure that temperature is monitored at the hopper input, not dryer output! It is not uncommon when running at very hot temperatures (above 225°F/107°C) for significant heat loss to occur over a 10 ft. length of uninsulated dryer hose. So, make sure the hose leading to the hopper is insulated!

Typical closed dehumidifying system
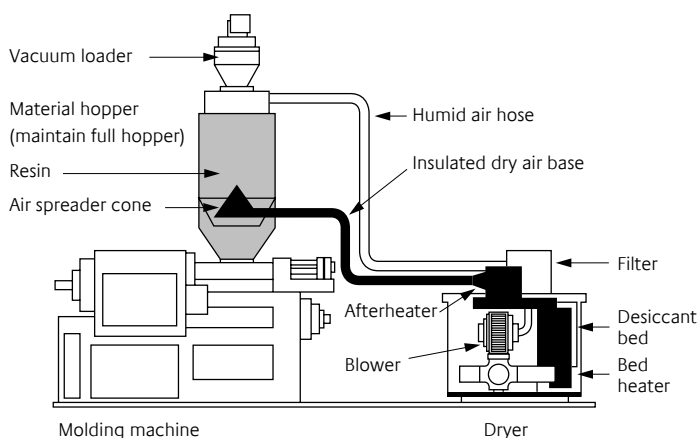
Residence time

The hopper should be properly sized so that for a specific production rate there is adequate residence time in the hopper. For most resins, 4–6 hours residence time is required. Typically, hotter air temperatures can reduce residence time.

Example A single cavity tool produces a 45.4 gram part every 36 seconds. What is the throughput rate? For a 6-hour hopper residence time, how large should the hopper be?

    45.4g = 0.1 lb. 36 sec. = 1/100 hr. = 0.01 hr.
    (0.1 lb.)/(36 sec.) = 0.1 lb./0.01 hr. = 10 lb./hr. throughput rate
    10 lb./hr. 6 hrs. = 60 lbs. hopper capacity

Airflow

If the dryer cannot produce adequate air flow, the resin cannot be effectively dried, and required residence times dramatically increase. Make sure the filters are cleaned. A good rule of thumb would be anywhere from twice a week to once every 2 weeks.

Hopper design

A tall cylindrical hopper equipped with a diverter cone and baffles is preferred to a short, square hopper. Material or air channeling will be minimized when using a properly designed hopper.

Moisture analysis

LNP* recommends determining the moisture content of compounds prior to molding using a commercially available moisture analyzer. This will help confirm proper function of the drying equipment.

Dessicant/dewpoint

Monitoring of dewpoint is the single most important factor a processor can do to assure that the drying system is working properly. Dewpoint is a measure of the moisture content of the warm air. Monitoring dewpoint can spot the following common problems associated with malfunctioning dryers

- Bad dessicant/dessicant contamination (dessicant loses effectiveness with time and number of regeneration cycles).
- Air leakage into the close loop system
- Inoperative regeneration heating elements
- Incorrect blower rotation
- Dessicant assembly not regenerating properly.

Dewpoint should be 0°F (-18°C) or lower. Continuous monitoring is suggested to assure that all multiple bed dessicant units are regenerating properly. Dewpoints higher than 0°F (-18°C) may dry material, but required residence times are extended. In extreme cases, if the dewpoint is not low enough, the resin never achieves the required level of dryness.

Percent moisture versus time
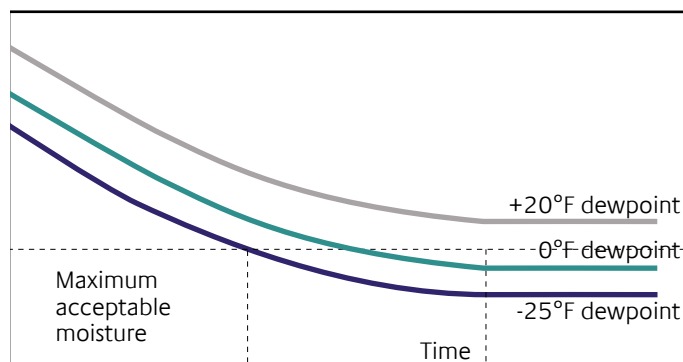
Tolerances

According to Shigley "Among the effects of design specification on cost, those of tolerances are perhaps most significant. Tolerances in design influence the productivities of the end product in many ways, from necessitating additional steps in processing to rendering a part completely impractical to produce economically. Tolerances cover dimensional variation, surface-roughness range, and also the variation in mechanical properties..."

Tolerances for injection-molded parts are controlled by six variables – material shrinkage, gating, part geometry, tool quality, tool tolerance and processing. Tool tolerances are unique to the mold manufacturing method, and they usually are held constant for any dimension. Processing (mold temp., melt temp., gate freeze-off time, etc.) is usually established for other reasons. Therefore, assuming that normal tool tolerances can be held, and processing is optimized and held constant (closed loop controls are recommended), there are four main variables which control injection molded tolerances. Of these four, material shrinkage is the most important. Figure 1 illustrates a summary of these four variables and the reasons why these variables are important.

Figure 1 Four major molded-part tolerance variables

Material shrinkage	• large or small • anisotropic or isotropic • predictable with thickness
Gating	• proper location and number • minimize distance to tolerance dimension • adequate and even pressure distribution • proper size and type • minimize anisotropic fiber orientation • balanced runner system
Part geometry	• dimension size • parting-line dimensions • linear and diametrical dimensions • dimensions which restrict material shrinkage • wall thickness • radii, draft, uniform wall, etc. • proper cooling for even mold temperature
Tool quality	• proper tool material • proper venting • amount of total rework allowed • minimal tool deflection

To compare materials, consider a one-inch linear dimension, measured in the flow direction, not across a parting line, which has proper cooling, venting and a gate close by. A practical minimum tolerance for materials which shrink 0.006 in./in. in both directions would be ±0.002". Such materials include many neat amorphous thermoplastic resins, cast zinc, cast magnesium and cast aluminum. Cast alloys, however, can hold ±0.001" for every additional inch, while neat amorphous resins remain at ±0.002" for each additional inch for reasonable flow distances for the material. Amorphous resins reinforced with 30% glass fiber have a practical minimum tolerance of ±0.001" for the inch dimension mentioned above and ±0.001" for each additional inch for reasonable flow distances. Crystalline resins reinforced with 30% glass fiber have a practical minimum tolerance of ±0.002" for the inch dimension and ±0.002" for each additional inch.

The tolerances previously mentioned are considered practical minimums. That is, a geometry can be molded within these tolerances and still be fairly complex yet not be exactly ideal from a tolerance point-of-view. Under ideal conditions, for even simple geometries, attempts can be made at tolerances of 50% of the above tolerances. Further, standard tolerances can be found by adding ±0.001" to minimum tolerances, and coarse tolerances can be determined by adding ±0.002" to minimum tolerances.

Understanding thermoplastic part warpage

One of the most difficult phenomenon to predict when designing an injection molded thermoplastic part is the degree of part warpage. Significant part warpage can result in the part being out of specification (tolerance) or totally nonfunctional in the application. For these reasons it is desirable to minimize the amount of warpage in an injection molded part. To minimize part warpage it is important to understand the cause of part warpage. An injection molded thermoplastic part will warp when internal stresses in the part reach a level high enough to overcome inherent part stiffness and cause part deformation. Differential shrinkage is the primary cause of these internal or residual stresses in an injection molded part. Differential shrinkage in a part can be caused by a variety of factors. When evaluating part warpage the four areas that must be considered are material, part design, tool design and processing.

Material

The material selected for an application can have a dramatic effect on the dimensional stability of the final part. It is more difficult to achieve a dimensionally stable part using materials with very high shrinkage values. Due to their lower shrinkage values, amorphous resins are generally chosen over crystalline resins when tight tolerances are required. More importantly than the magnitude of shrinkage is the degree of isotropic shrinkage in the material. If a material shrinks anisotropically (e.g., shrinkage in transverse direction is different than shrinkage in flow direction) then differential shrinkage in the part will result. This differential shrinkage in the part may cause part warpage. A material which shrinks isotropically will minimize differential shrinkage and stress in the part and maximize part dimensional stability. The following table illustrates the effect of differential material shrinkage on part warpage.

Notice how the crystalline acetal material exhibits much greater warpage due to the higher shrinkage values and, more importantly, the large differences between flow and transverse shrinkage. Warpage in the amorphous polycarbonate materials is low due to the low shrinkage values and more isotropic shrinkage behavior. The addition of fibrous reinforcements will generally increase part warpage even though overall material shrinkage is lower. Fiber orientation during cavity filling will increase anisotropic (differential) shrinkage as the fibers align in the flow direction. This results in a significant reduction in shrinkage in the flow direction with little shrinkage reduction in the transverse direction. This is evident in the following data for the 10% and 30% glass fiber reinforced materials.

Base resin	Modifier	Flow shrinkage (in./in.)	Transverse shrinkage (in./in.)	Warpage A/D
Acetal	Unfilled	0.020	0.016	0.075
Acetal	10% GF	0.011	0.013	0.030
Acetal	30% GF	0.004	0.015	0.300
Polycarbonate	Unfilled	0.005	0.005	0.001
Polycarbonate	10% GF	0.003	0.003	0.001
Polycarbonate	30% GF	0.001	0.003	0.003

Please refer to LNP's* "Predict Shrinkage and Warpage of Reinforced and Filled Thermoplastics" for more material data.

Part design

The design of the plastic part can also have a significant effect on dimensional stability. If an injection molded part is designed with very non-uniform wall sections then the part may experience differential shrinkage problems. Differential shrinkage occurs in non-uniform wall parts due to differences in the timing of shrinkage and the amount of shrinkage in the part. Thicker areas of the part will cool slower than thinner areas due to the greater thermal mass of the thicker sections. As a result, the thinner sections of the part will cool and shrink before the thicker sections. This results in differential shrinkage in the part. This differential shrinkage is due to the timing of shrinkage in the part.

Differing levels or amounts of shrinkage will also exist in the part as the thicker sections will shrink more than thinner sections. This effect is amplified in crystalline materials. The degree of crystallinity through the part will be effected by wall thickness variations. Thicker sections will be slower cooling and have a higher degree of crystallinity than faster cooling thinner sections of the part. Sections with higher crystallinity will shrink more than sections with lower crystallinity levels resulting in differential shrinkage in the part. This differential shrinkage is due to differences in the amount of shrinkage in the part.

Parts designed with non-uniform wall sections can experience severe differential shrinkage due to differences in both the timing of shrinkage and the amount of shrinkage. As mentioned above, differential shrinkage causes stress in the part and can lead to part warpage. The figure to the right illustrates this phenomenon.

As variations in part wall thickness increase the chance of part warpage also increases. Designing injection molded thermoplastic parts with uniform wall sections is recommended for this very reason.

One area of part design which can effect part warpage relates to overall part stiffness. A stiffer part will be more resistant to warpage than a more flexible design. Internal stresses in a part are the cause of part warpage or deformation. If the part is sufficiently stiff then these stresses can be withstood and the part will not deflect or warp a measurable amount. The use of ribs or gussets is the most effective way to increase part stiffness without increasing wall thickness.

Tool design

Improper tool design can also cause part dimensional stability problems. The gate location(s) chosen for a part is critical to minimizing warpage. An ideal gate location is one which provides balanced cavity filling with the melt simultaneously reaching all of the last areas of the cavity to fill. Two detrimental effects — overpacking and differential cooling — may result if cavity filling is unbalanced. Overpacking and differential cooling cause differential shrinkage in the part which can lead to part warpage.
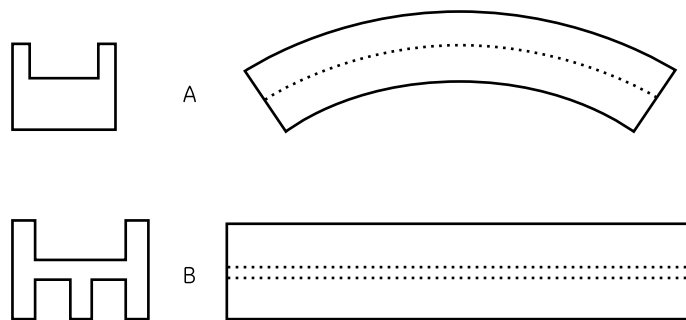
A = Non-Uniform wall design results in part warpage
B = Uniform wall design minimizes in part warpage

Warpage due to differential material shrinkage


$$\frac{A}{D} = \text{Warp}$$
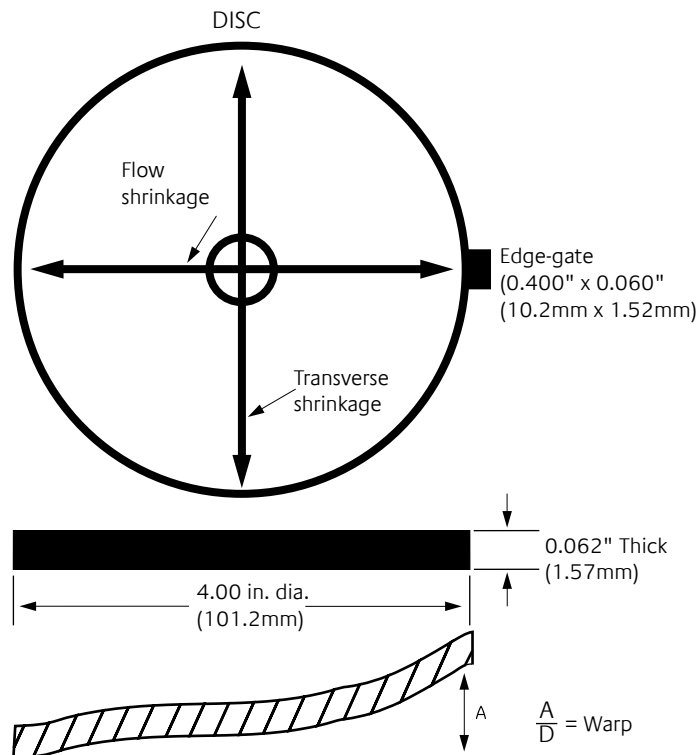
Overpacking can occur when cavity filling is unbalanced. Unbalanced cavity filling occurs when certain areas of the cavity fill prematurely before the rest of the cavity has filled. As the melt in the prematurely filled regions of the cavity begins to cool, more melt is forced into these regions due to the high first stage pressure left on to fill the remainder of the cavity. As a result, the prematurely filled regions of the cavity will experience overpacking. Highly packed areas of the part will shrink less than areas of lower packing resulting in differential shrinkage in the part. This differential shrinkage is due to differences in the amount of shrinkage in the part.

Differential cooling can also occur when cavity filling is unbalanced. The melt in the prematurely filled regions of the cavity will begin to cool as the remainder of the cavity fills. The melt in these regions is sitting idle and will rapidly lose heat while cavity filling is completed. Consequently, at the instant of fill the melt in these regions will be at a lower temperature than the rest of the melt in the cavity. This temperature differential will be dependent on the cavity fill time and the temperature difference between the melt and the mold steel. The melt in the prematurely filled regions of the cavity will shrink and cool sooner than melt in other areas of the cavity. This results in differential shrinkage in the part. This differential shrinkage is due to the timing of shrinkage in the part.

Proper gate location and number of gates is also critical during the packing phase of the injection molding process. By using multiple gates, and keeping flow lengths short, a more uniform and even packing pressure can be applied throughout the cavity. An even packing pressure distribution in the cavity will promote uniform part shrinkage. Uniform shrinkage in the part will help to minimize part warpage.

The mold cooling system is a critical area of the tool design which can have a significant effect on overall part dimensional stability. A mold cooling system which does not maintain a uniform mold temperature during processing can also result in differential cooling of the part. The melt in cooler areas of the cavity will cool and shrink before the melt in the hotter areas of the cavity. This results in differential shrinkage in the part again due to differences in the timing of shrinkage. Differences in the amount of shrinkage throughout the part may also occur due to an improperly designed mold cooling system. The melt in hotter areas of the cavity will cool slower resulting in an increase in shrinkage. This is particularly true for crystalline resins as the slower cooling rate will increase crystallinity and subsequent shrinkage. A properly designed mold cooling system will cool all the melt in the cavity to the ejection temperature at the same time.

An excellent example of differential cooling resulting in part warpage can be found with a box shaped geometry. During processing the inside corners of the box will remain hot due to greater heat transfer from the melt to the corners of the box core. This greater heat transfer is due to the corners of the box core being surrounded by melt on three sides. Due to this temperature difference, the corners of the box will cool and shrink later than the sidewalls of the box. The sidewalls are pulled inward as the corners cool and shrink relative to the previously frozen sections. The amount of shrinkage in the corners may also be greater, particularly for crystalline resins. This phenomenon is illustrated in the figures below.

Typical cooling circuits

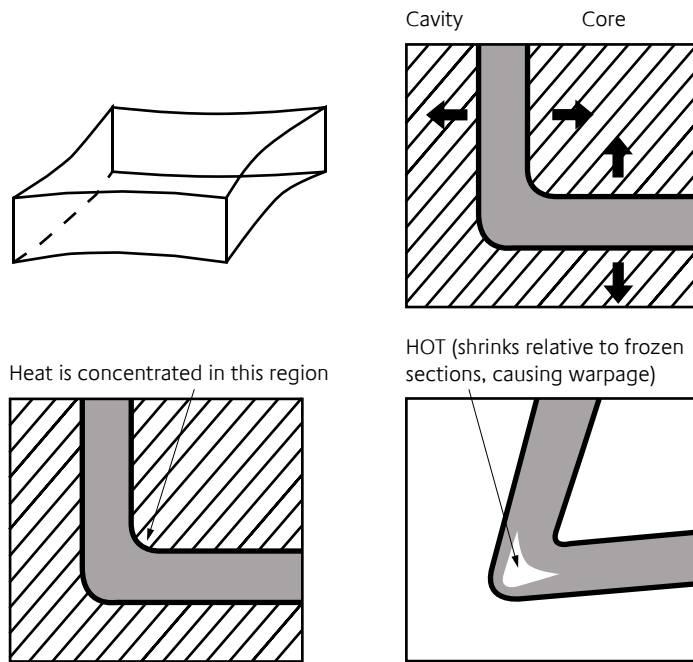Cooling circuits for uniform heat transfer
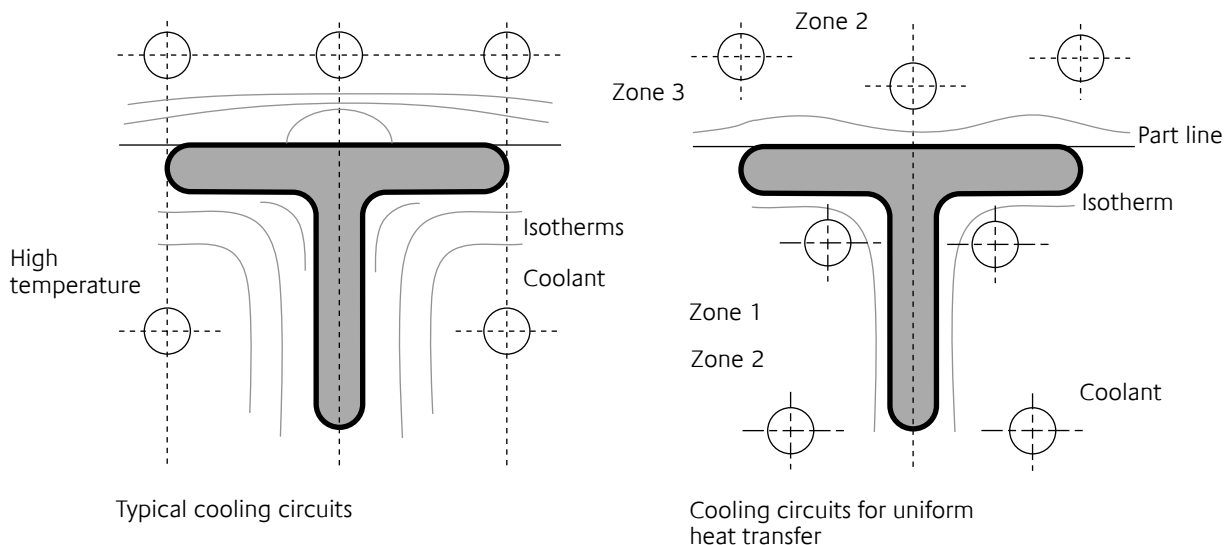
A properly designed mold cooling system will effectively remove heat from the corners of the box to maintain a uniform overall mold temperature. The illustration below is an excellent example of preferentially cooling hotter areas of the mold to maintain a uniform overall mold temperature. Notice how the coolant channels have been moved closer to the hotter areas of the cavity (corners, thick section) to provide uniform mold cooling.

Processing

Proper processing, specifically cavity fill time, is also a critical factor in achieving dimensionally stable parts. If the cavity fill time is too short (eg., cavity filled too quickly) a significant amount of molded-in stress can exist in the part. Excessively fast cavity filling produces high shear rates which causes substantial melt shearing and subsequent high levels of shear stress in the part. This high level of molded-in stress can cause part warpage. This is the one area where the fundamental cause of warpage is not differential shrinkage.

If cavity fill times are too long, differential melt cooling may result. An excessively long cavity fill time results in a large melt temperature drop as cavity filling is completed. A non-uniform melt temperature distribution in the cavity at the instant of fill will result due to this large melt temperature drop. A non-uniform melt temperature distribution at the instant of fill is undesirable and will result in differential cooling of the part. The melt in the last areas to fill will begin to cool and shrink sooner than the melt near the gate(s) resulting in differential shrinkage in the part. This differential shrinkage is due to differences in the timing of shrinkage.

Effectively packing the part is also an important factor for producing parts with minimal warpage. Overpacking or underpacking the part during the packing phase of the injection molding process can lead to dimensional stability problems. Areas near the gate are subjected to a higher level of packing pressure. When flow lengths are long, areas furthest from the gate will experience less packing pressure. These lower packed areas will shrink more than areas closer to the gate. As a result, the part will shrink differentially and may warp due to differences in the amount of shrinkage in the part.

Cavity   Core

Heat is concentrated in this region

HOT (shrinks relative to frozen sections, causing warpage)

Part warpage and dimensional stability problems can also occur from underpacking the part. Underpacking of the part is generally caused by either premature gate freeze-off or a hold time which is too short. The end result is that the melt cools and shrinks in the cavity without experiencing any applied pressure. No additional melt is forced into the cavity to account for the volumetric shrinkage of the material. Underpacking generally results in an increase in overall part shrinkage and tendency to warp.

In summary, although warpage in injection molded thermoplastic parts is very difficult to predict, the following suggestions are provided to aid in minimizing part warpage
  • Utilize a material which shrinks more isotropically
  • Design the part with uniform wall thicknesses
  • Increase part stiffness through ribbing and gussets
  • Position gate(s) for balanced cavity filling and minimal flow lengths
  • Optimize the mold cooling system to maintain a uniform mold temperature throughout the tool
  • Ensure that cavity fill time is appropriate
  • Effectively pack part by
    – insuring that gate freeze-off is not premature
    – providing for adequate hold time

LNP Colorcomp compounds

Colorcomp pre-colored unfilled engineering resins are excellent candidates for the OEM or molder with lot releases of 110 to 40,000 pounds, critical color accuracy requirements and/or a need for short lead times. Colorcomp resins can be manufactured from virtually any thermoplastic resin in the LNP product line, in addition to select trademark resins from other major suppliers. A full line of special effects resins is available. All Colorcomp resins are manufactured to meet QS/ISO standards for lot traceability.

LNP Konduit* compounds

Composites of thermally conductive fillers and engineering thermoplastics, Konduit compounds have up to 2 to 10 times more thermal conductivity than traditional unfilled resins, while remaining electrically insulative, plus CLTEs similar to many metals. Konduit composites may reduce thermal rise and increase the efficiency of clutch coils, motors, transformers and many other coil wound systems.

LNP Lubricomp* compounds

Lubricomp internally lubricated compounds offer inherent lubrication through the addition of PTFE, silicone, aramid fiber and/or other materials to a wide variety of engineering thermoplastics. Lubriloy* compounds, a family of proprietary lubricated alloys, offers properties approaching PTFE-lubricated materials at reduced cost. These products may find use in demanding wear applications in the business machines, automotive, medical, appliance and industrial markets.

LNP Starflam* compounds

Most Starflam flame retardant (FR) nylon compounds meet the requirements for halogen and red phosphorous free materials, and can be formulated to comply with voluntary ECO labels such as Blue Angel and TCO. They also provide higher impact strength, faster cycle times, lower specific gravity and less mold corrosion versus brominated and chlorinated FR systems. UL flame retardancy ratings of V-2 to V-0 can be typical on 1.6mm thickness samples. Starflam compounds are laser markable and offer high relative thermal index, comparative tracking index and Glow wire test electrical properties. They may be suitable candidates for electrical and electronic applications, including mini circuit breakers, contactors, connectors, switch gear housings, relays and motor housings.

LNP Stat-Kon* compounds

Stat-Kon electrically conductive compounds may provide economical and reliable solutions against electrostatic buildup. Faradex* compounds offer EMI/RFI shielding and ESD protection, eliminating the need for most special coatings or paints. Stat-Loy* composites contain permanent anti-static additives that are non-humidity dependent and non-migratory. Formulated for ease of processing, these compounds can be injection molded or extruded. Common applications include automotive fuel delivery systems, electronic and electrical equipment/instruments, business machines and more.

LNP Thermocomp* compounds

Thermocomp glass and/or carbon fiber reinforced compounds offer enhanced mechanical and thermal properties, including exceptional resistance to high temperature, fatigue, creep, impact and chemicals. Thermotuf* composites have been impact modified for additional toughness. The Thermocomp line also includes High Specific Gravity (HSG) compounds, melt processible fluoropolymer compounds and Exceptional Processing (EP) compounds for thin wall molding. Products from the Thermocomp line are typically used in automotive functional components, business machines, electrical/electronic components, consumer goods, appliances and industrial applications.

LNP Verton* compounds

Verton composites combine nylon, polypropylene, polyphthalamide and other engineering thermoplastics with long reinforcing fibers using SABIC Innovative Plastics pultrusion process, which may provide an outstanding balance of cost and performance in structural applications. Specifically, these remarkably lightweight materials offer exceptional mechanical properties, combining rigidity with outstanding strength and resistance to impact failures. Verton composites find use in demanding structural applications, primarily in the automotive, industrial and recreational markets, and frequently replace die-cast metal.

Americas
Headquarters
SABIC Innovative Plastics
One Plastics Avenue
Pittsfield, MA 01201
USA
T 800 845 0600
T 413 448 5800
F 413 448 7731

European Headquarters
SABIC Innovative Plastics
Plasticslaan 1
PO Box 117
4600 AC
Bergen op Zoom
The Netherlands
T +31 164 292911
F +31 164 292940

Technical Answer Center
T 0800 1 238 5060 *toll free
(if mobile disruption +36 1 238 5060)

Pacific
Headquarters
SABIC Innovative Plastics
1266 Nanjing Road (W)
Unit 902-907, Plaza 66
200040 Shanghai
China
T +86 21 3222 4500
F +86 21 6289 8998

Email
productinquiries@sabic-ip.com