



RECOMMENDED PROCESSING GUIDELINES FOR INJECTION MOLDING

Styrene Acrylonitrile Butadiene Terpolymer

Drying: Drying is recommended for 2-4 hours at 160-180F to achieve optimum clarity in molded parts. SAN copolymers will absorb up to 0.6% moisture. Therefore, it is recommended to dry to a level of < 0.1% moisture in a dessicant drier to achieve a dew point less than -20°F (-29°C).

Processing: Recommended clamp tonnage for injection molding SAN is 2.0-3.0 tons per square inch of projected area. The machine size is a factor of the polymer volume required to fill the mold. Shot size between 40% and 75% of total machine capacity is recommended. Reciprocating screw injecting molding machine barrel, nozzle and mold temperature setting are recommended as follows:

		°F	°C
Barrel temperatures:	Feed	320 - 360	160 - 182
	Compression	360 - 430	182 - 221
	Metering	380 - 450	193 - 232
	Nozzle Temperature:	380 - 450	193 - 232
	Melt Temperature:	380 - 420	193 - 216
Mold Temperature:		100-170(front)	38-77
		80-140(rear)	27-60

NOTE: Excessive barrel or melt temperatures or long residence times in barrel can cause material to yellow causing finished part color shifts. Mold temperature is dictated by part and mold design.

Screw Design: Most general screw designs will work with this material. Avoid harsh mixing type screws that may cause material degradation. Screw speed should be slow and adjusted to allow complete recovery just prior to mold opening. Maintain a cushion of 1/8th" or less after injection to provide good dimensional reproducibility and weld line integrity.

Non-Return Valve: Sliding check ring designed NRV is recommended with SBN-150 material. If a NRV sliding check ring style valve is used, use a small amount of the dcrew decompression (screw suckback) to help set the slip ring prior to the next shot. Excessive screw decompression can cause splay. The use of a ball type check valve can cause material degradation.

Shot weight: 40-80% of rated barrel capacity. Barrel temperatures may need to be modified to adjust for differences from the recommended shot size versus barrel capacity ratio. Long residence times in the barrel should be avoided.

Injection pressure: Is dependent on part size and mold design. An injection pressure which is too low may result in sink marks; while if too high, demolding may be more difficult and/or flash may occur. Typical standard injection pressures range between 600 and 1100 psi.

Injection Velocity: Dictated by mold and part design. Generally medium to fast fill speeds.

Back Pressure: 75-150 psi is generally sufficient to insure a consistent shot density to mix material and colorants well. Excessive back pressure can cause higher melt temperatures and cause material degradation.

Purge: Use Polystyrene or SAN to purge machine after colors or before starting to insure a clean machine.

The information contained herein is provided for general reference purposes only. Please note that the above information pertains only to natural resins and may change with the addition of colorants or other additives. Network Polymers Inc. makes no guarantee or warranty and does not assume liability with respect to the accuracy of completeness of the information presented, or the product results in any specific instance and hereby expressly disclaims any implied warranties of suitability or fitness for a particular purpose or any other warranties or representations whatsoever expressed or implied. Nothing herein shall be construed as a recommendation to use any product in conflict with patents covering any material or its use. No license is implied or in fact granted under the claims of patent.