Polycarbonate (PC) Injection Molding — Complete Setup Guide

Processing parameters, mold design, defect prevention & optimization checklist

Key Points

Polycarbonate (PC) requires precise settings due to its high viscosity, heat sensitivity, and hygroscopicity. Proper drying and temperature control are critical to prevent silver streaks, bubbles, and internal stress. PC is known for its exceptional strength-to-weight ratio and optical transparency.

I. Core Processing Parameters

Temperature Settings

Parameter	Value	Range / Notes
Melt Temperature	270–320 °C (518–608 °F)	260–340 °C (500–644 °F)
Target Melt Temp	315 °C (600 °F)	Typical starting point
Barrel – Rear (Feeding)	260–280 °C	580–600 °F
Barrel – Center (Compression)	280–300 °C	580–620 °F
Barrel – Front (Metering)	300–320 °C	600–620 °F
Nozzle Temperature	280–300 °C	580–600 °F
Mold Temperature	80–120 °C (176–248 °F)	80–150 °C (180–250 °F)
Target Mold Temp	100 °C (210 °F)	Standard; 120 °C for thin/ optical

Critical Warning: Never exceed 340 °C (decomposition threshold). Signs of degradation: darkened color, silver streaks, dark spots, bubbles, and degraded mechanical properties.

Drying Requirements (Mandatory)

Parameter	Value
Drying Temperature	120 °C (250 °F)
Drying Time	4+ hours minimum (3–5 hours typical)
Target Moisture	<0.02% (some grades up to 0.04%)
Dryer Type	Dehumidifying oven/hopper dryer (dew point ≤ −30 °C)
Hopper Temp After Drying	70–80 °C (prevent re-absorption)

Critical: PC must be dried to <0.02%. Undried material causes:

- Decreased mechanical strength
- Bubbles and crazes on the surface

- Silver streaks
- Poor transparency

II. Pressure & Speed Settings

Parameter	Value	Range / Notes
Injection Pressure	80–120 MPa	850–1400 kg/cm² (max up to 2400 kg/cm²)
Thin-Walled Parts	120–145 MPa	High pressure required
Injection Speed	Medium to Slow	Multi-stage: Slow → Fast → Slow
Hold Pressure	30–75% of injection	30–75 kg/cm²
Back Pressure	50–100 PSI	3–14 kg/cm² (10–15% of injection)
Screw Speed	30–60 RPM	40–70 RPM recommended; target 50
Shot Size	40–60% of capacity	Max 70–80% (never >75%)
Cooling Time	20–60 seconds	Varies by part thickness

Multi-stage injection: Slow → Fast → Slow prevents air trapping and ensures smooth cavity filling.

III. Machine Requirements

Screw Design

Parameter	Specification
Type	General-purpose screw, chrome-plated
L/D Ratio	15–20:1 (optimal 20:1)
Compression Ratio	1.5–3.0:1
Sections	50% feed · 25% compression · 25% metering
Check Ring	Sliding ring (at least 3.2 mm gap)
Screw Type	Reciprocating screw required

Clamping Force

Parameter	Value
Required Tonnage	0.47–0.78 tons/cm² (3–5 tons/in²) of projected area
Machine Size	40–60% of machine capacity (reduce 10% if measured in PS ounces)

Nozzle Design

Feature	Specification
Type	Flow-through nozzle, reverse taper
Tip Opening (min)	≥4.5 mm diameter
For parts ≥5.5 kg	≥9.5 mm diameter
Gate Comparison	0.5–1 mm smaller than gate diameter, ~5 mm length

IV. Mold Design Requirements

Runners & Gates

Feature	Recommendation
Design	Short and thick (minimize pressure loss)
Gate Type	Round or fan gate (avoid pin-point)
Gate Size	As large as possible
Cavity Surface	Mirror polish #8000+ (no pits or scratches)

Venting

Location	Depth	Width
Parting Lines	0.015–0.03 mm	5–8 mm
Corners	0.015–0.03 mm	5–8 mm

Critical: Poor venting causes short shots, gas lines, burning, and fogging.

Ejection System

Parameter	Setting
Support	Multi-point, balanced
Speed	Slow and uniform
Avoid	Whitish marks and cracks

V. Material Properties & Shrinkage

Property	Value
Linear Shrinkage	0.5–0.7% (0.005–0.007 in/in)
Volume Shrinkage	6.63% (PC among the lowest)
Transparency	88–90% (optical clarity)
Strength	Exceptional (≈250× stronger than glass)
Heat Resistance	Up to 135 °C continuous
Melt Flow Index	5–25 g/10 min (varies by grade)

VI. Critical Precautions & Defect Prevention

Common Defects & Solutions

Defect	Cause	Solution
Silver Streaks / Bubbles	Moisture >0.02%	Improve drying; verify <0.02% with moisture analyzer
Internal Stress	Low mold temp, high pressure	Higher mold temp (100–120 °C), lower hold pressure, annealing
Sink Marks	Insufficient packing, low mold temp	Higher packing pressure, mold temp ~120 °C, optimize gate
Flow Marks	Slow injection, wrong gate	Faster injection (thin parts), optimize gate design
Crazes / Cracks	High stress, fast ejection	Annealing (120–130 °C × 30–60 min), slower ejection, higher mold temp
Darkened Color	Temperature >340 °C	Strict temp control; never exceed 340 °C
Short Shots	Poor venting, low pressure	Improve venting, raise injection pressure
Poor Transparency	Overheating, dirty mold	Clean mold, proper drying, avoid >340 °C, polish cavity #8000+

Internal Stress Control (Critical for PC)

- **Annealing:** 120–130 °C for 30–60 minutes, slow cooling
- **Reduces:** >80% of internal stress
- **Prevents:** Cracking during storage, assembly, and use
- **High mold temp:** 100–120 °C reduces residual stress significantly

VII. Process Optimization Checklist

Temperature Control

- Barrel sectional: Feeding 260–280 °C, Compression 280–300 °C, Metering 300–320 °C
- Nozzle temp: 280–300 °C (slightly lower than front barrel)

- For thin-walled: higher temp within 270–320 °C
- For optical lenses: lower melt temp + higher mold temp (120 °C)
- Real-time temperature monitoring: ≤340 °C at all times

Injection Parameters

- Multi-stage speed: Slow → Fast → Slow
- Pressure: 80–120 MPa (thin-walled: 120–145 MPa)
- Hold pressure: 30–75% of injection (low preferred)
- Cooling: 20–60 seconds (sufficient for wall thickness)

Material Handling

- Drying: 120 °C for ≥4 hours (dehumidifying dryer)
- Moisture check: confirm <0.02% with a moisture analyzer
- Hopper: insulated, maintain 70–80 °C
- Re-dry if material is unused for >4 hours
- Screen: remove impurities, agglomerates, dust
- Avoid mixing different grades or batches

Environment

- Clean, dust-free, temperature-controlled (20–25 °C)
- Relative humidity: <60%

VIII. Special Applications

Thin-Walled Parts (<2 mm)

Parameter	Setting
Temperature	280–310 °C (higher for flow)
Injection Pressure	120–145 MPa (high)
Injection Speed	Fast (overcome flow resistance)
Mold Temp	100–120 °C (higher for fill)

Thick-Walled Parts (>5 mm)

Parameter	Setting
Injection Speed	Slow (prevent bubbles/dents)
Hold Time	Longer (gate freeze-off)
Mold Temp	100–120 °C (reduce sink marks)
Melt Temp	Lower (280–300 °C)

Optical Lenses (High Transparency)

Parameter	Setting
Melt Temp	Lower (280–300 °C)
Mold Temp	Higher (120 °C)
Gate Design	Optimized for uniform flow
Packing Pressure	High (reduce sink marks)
Annealing	120–130 °C × 30–60 min

Glass / Mineral Filled PC

Parameter	Setting
Injection Speed	Fast (recommended)
Temperature	280–320 °C
Mold Temp	90–110 °C

IX. Quick Setup Summary

Parameter	Standard Setting
Drying	120 °C × 4+ hrs (<0.02% moisture)
Melt Temp	270–320 °C (max 340 °C)
Mold Temp	80–120 °C (100 °C standard, 120 °C for thin/optical)
Injection Pressure	80–120 MPa (thin: 120–145 MPa)
Injection Speed	Medium-Slow (Slow-Fast-Slow multi-stage)
Hold Pressure	30–75% of injection (low preferred)
Back Pressure	50–100 PSI (10–15%)
Screw Speed	30–60 RPM (target 50)
Shot Size	40–60% (max 70–80%)
Cooling Time	20–60 seconds
Venting	Depth 0.015–0.03 mm, width 5–8 mm
Shrinkage	0.5–0.7%
Annealing	120–130 °C × 30–60 min

X. Critical Warnings

- **Drying MANDATORY:** Must dry to <0.02% (use moisture analyzer)
- **Temperature limit:** Never exceed 340 °C (decomposition)
- **Venting essential:** Poor venting = defects (short shots, burning)

- **Annealing recommended:** 120–130 °C × 30–60 min eliminates stress
- **High mold temp:** 100–120 °C reduces stress significantly
- **Stable temps:** Fluctuation within ±5 °C
- **Regrind limit:** Max 20% (higher levels reduce mechanical properties)
- **Moisture protection:** Keep insulated during processing

Compiled by Topworks Plastic Mold · plasticmoulds.net · myplasticmold.com
Parameters cross-checked against industry sources (ZetarMold, Fictiv, FOW Mould, Boyan Manufacturing, 3ERP, CSMFG, Ruicheng).