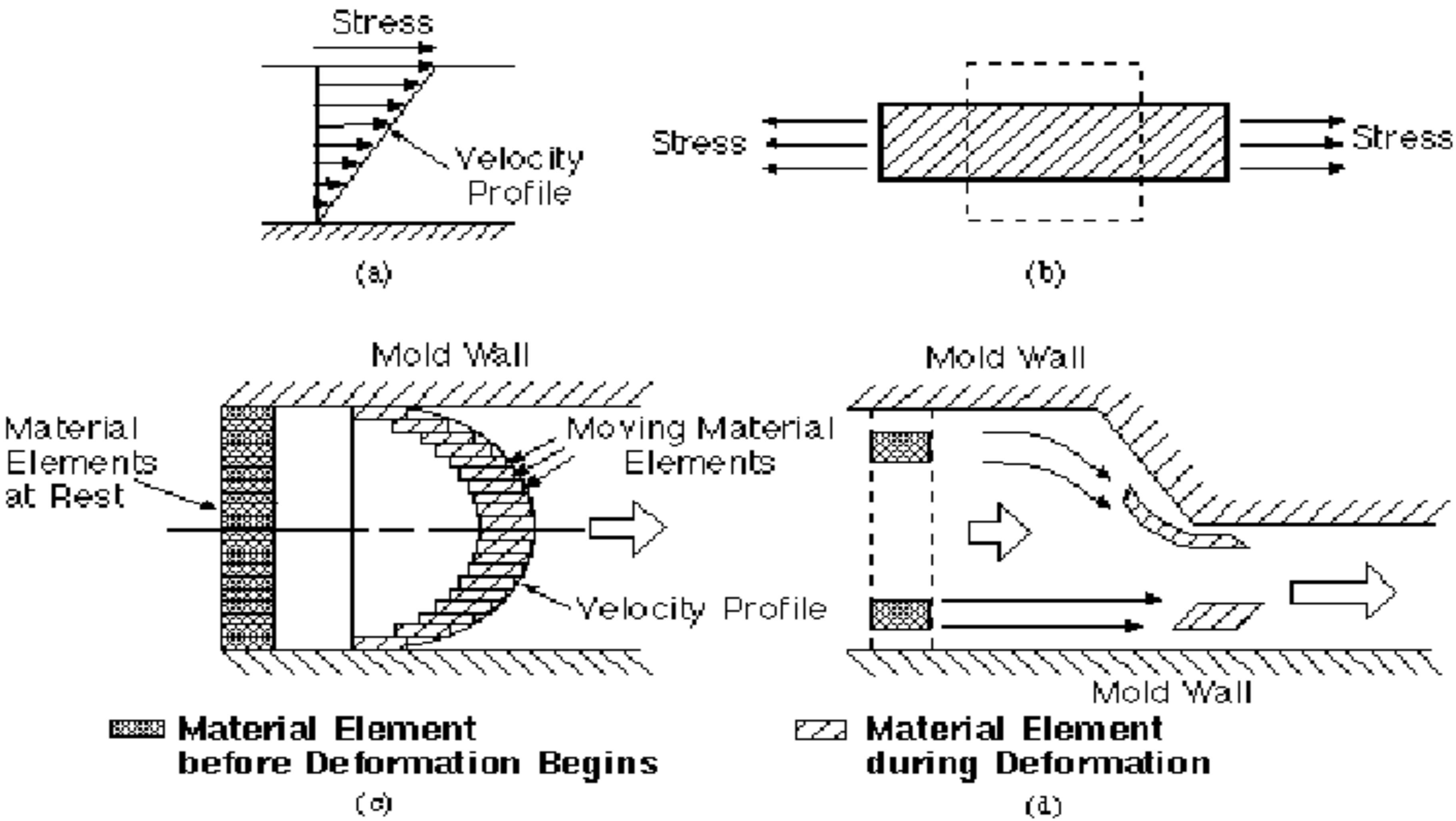
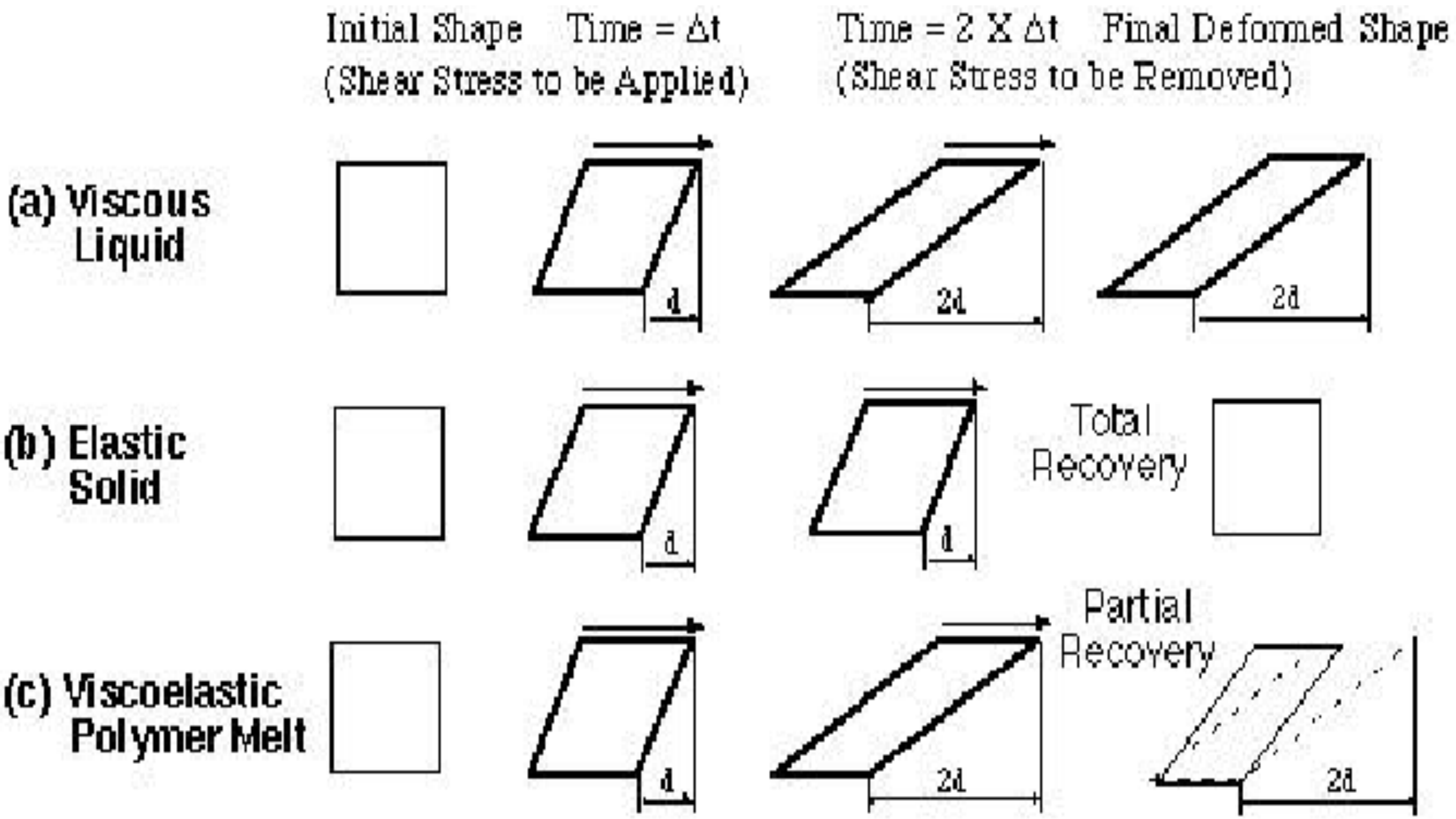
MKR1163 PLASTICS DESIGN AND PROCESSING

INTRODUCTION TO MOLD DESIGN

Review of Plastic Flow



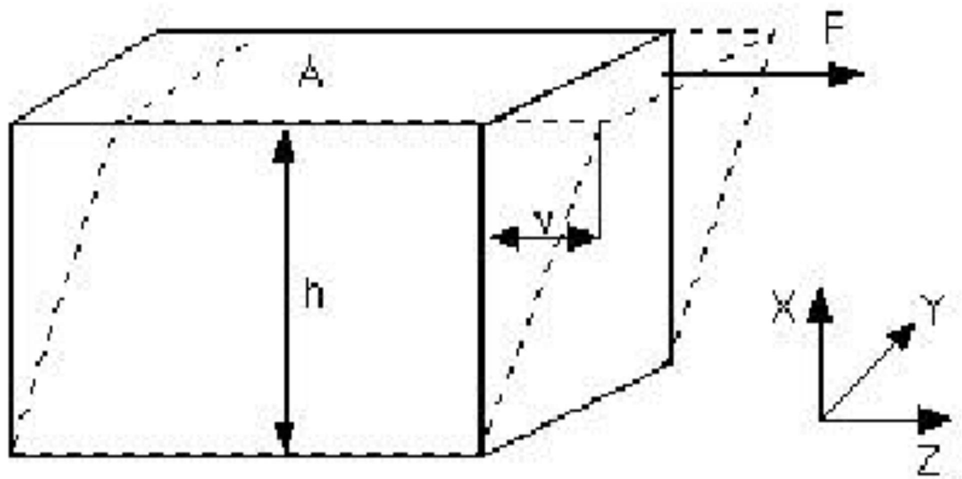

Initial Shape
Time = Δt
(Shear Stress to be Applied)
Time = 2 X Δt
Final Deformed Shape
(Shear Stress to be Removed)
(a) Viscous Liquid
d
2d
2d
(b) Elastic Solid
d
d
Total Recovery
(c) Viscoelastic Polymer Melt
d
2d
Partial Recovery
2d

Polymer Melt Viscosity

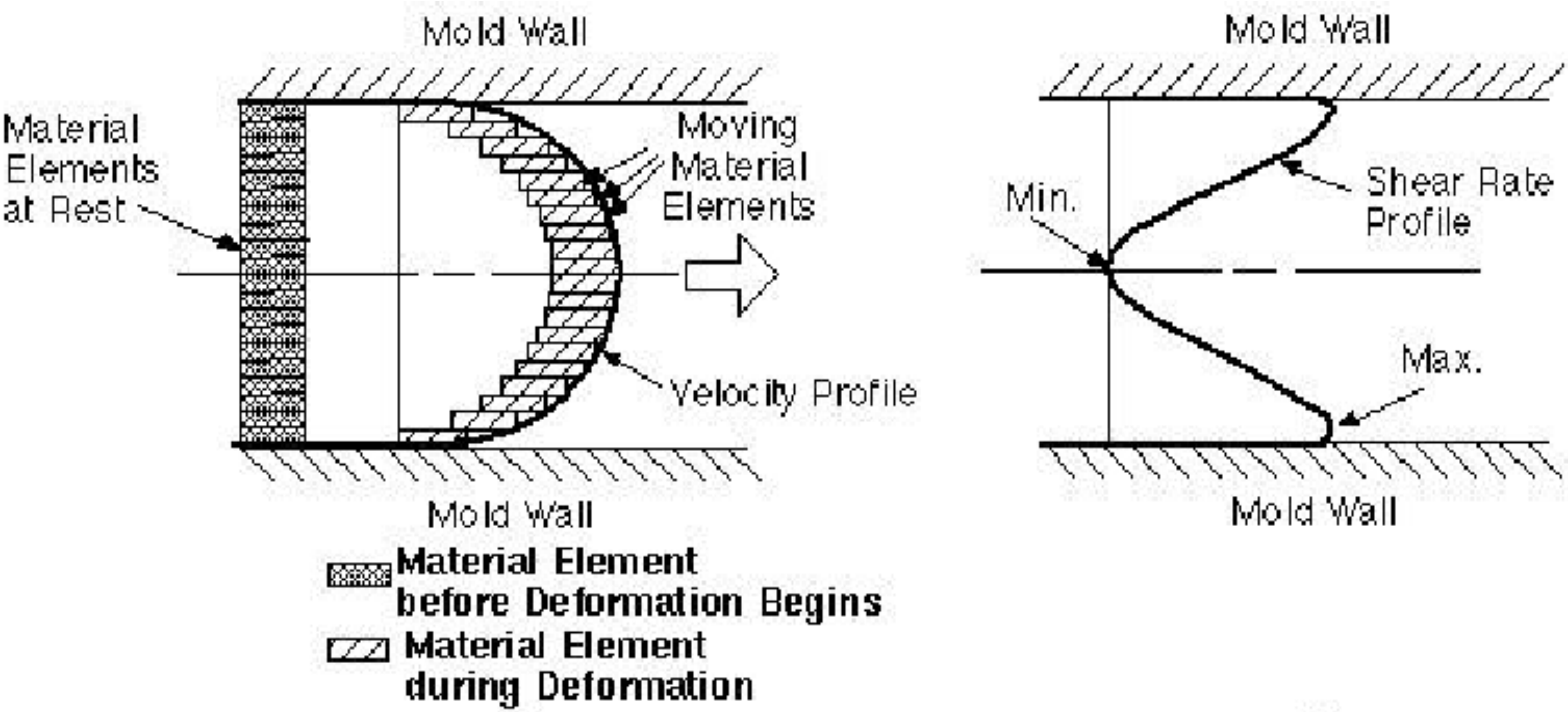
$$\text{Viscosity} = \frac{\text{Shear Stress}}{\text{Shear Rate}} \quad (D.1)$$

where

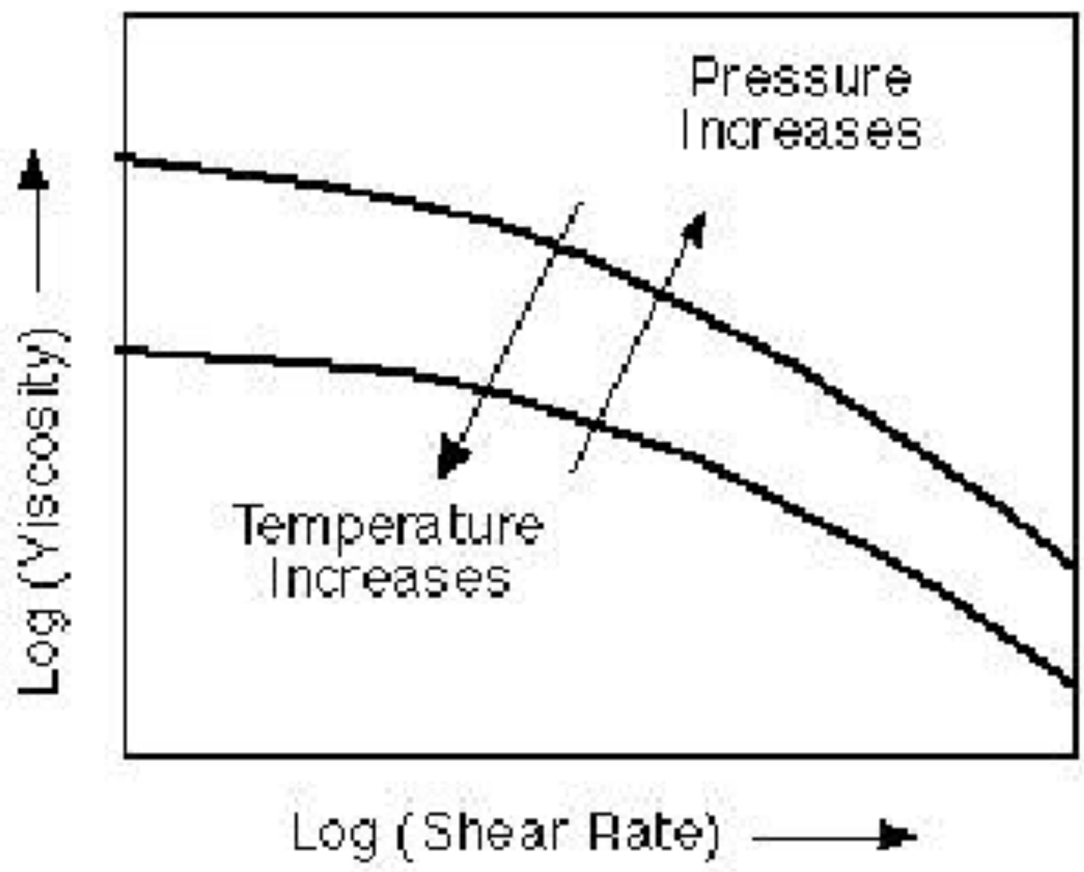
$$\text{Shear Stress} = \frac{\text{Force (F)}}{\text{Area (A)}} \text{ and Shear Rate} = \frac{\text{Velocity (v)}}{\text{Height (h)}} \quad (D.2)$$



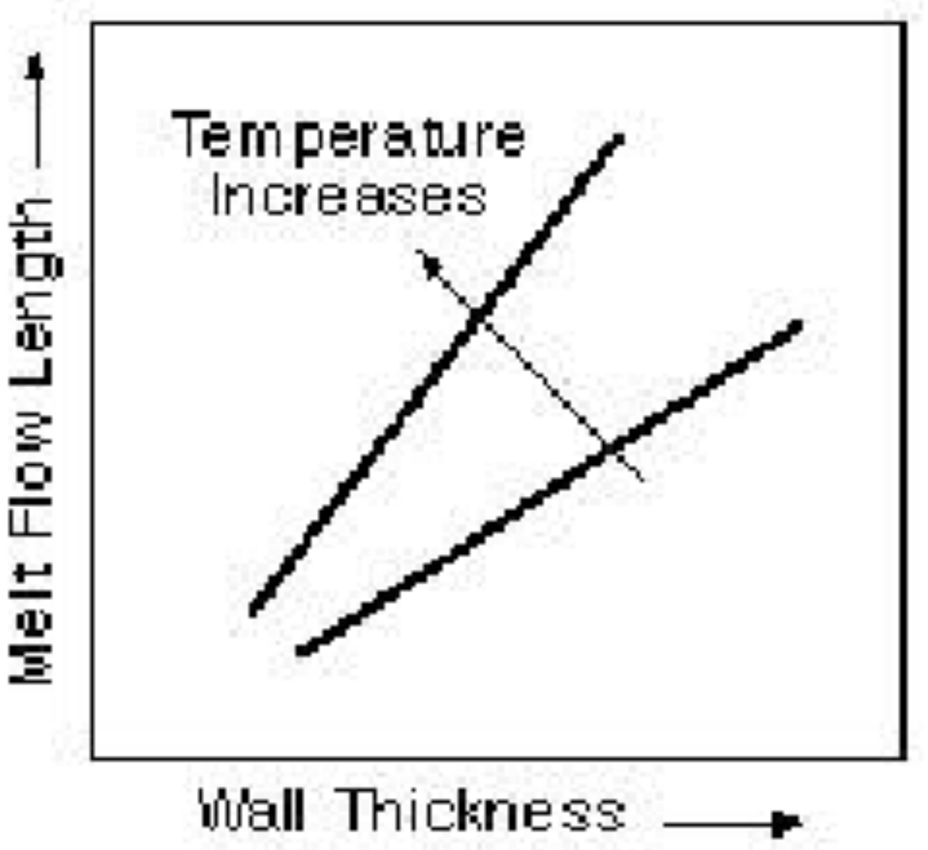
Velocity Profile and Shear Rate Distribution



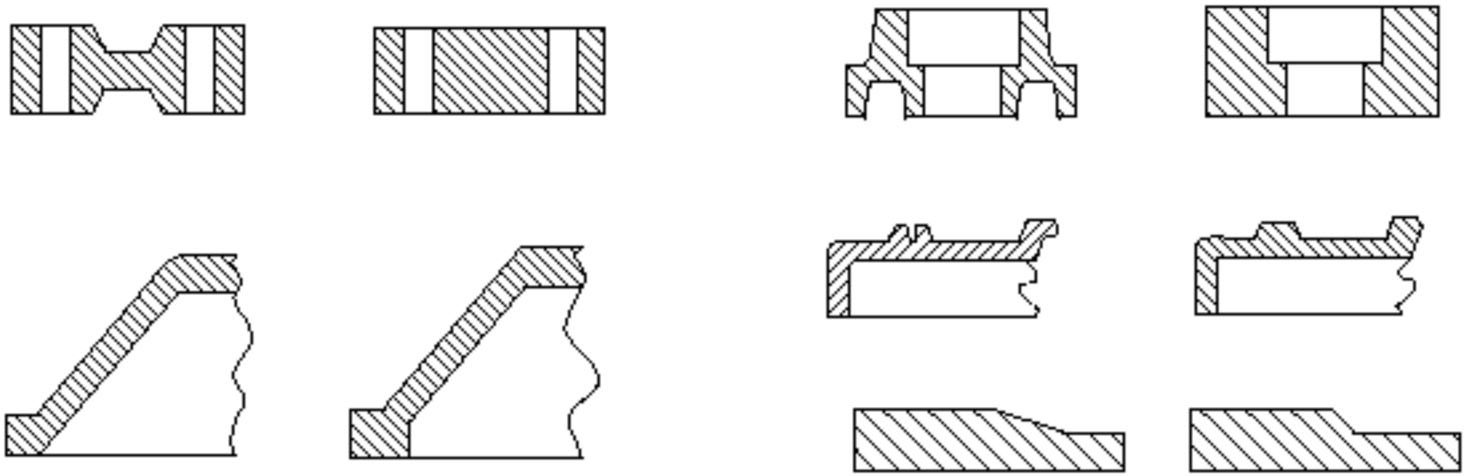
Factors Effecting Viscosity



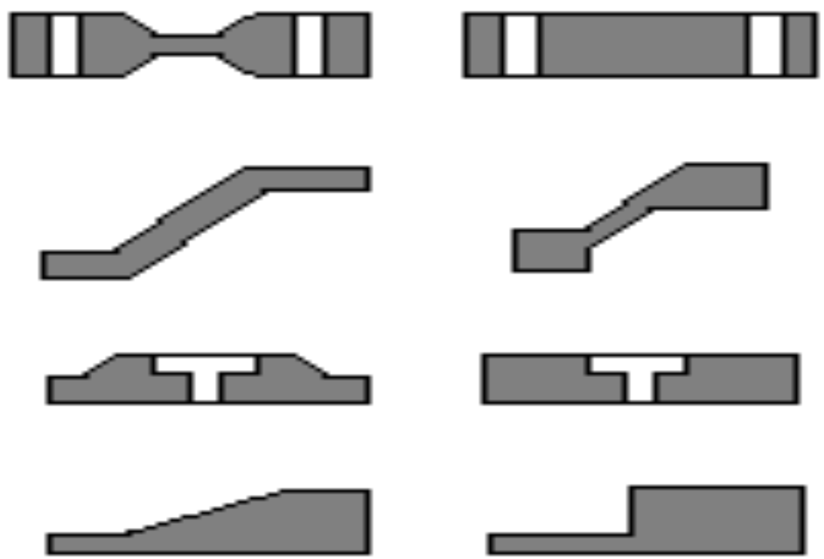
Melt Flow Length



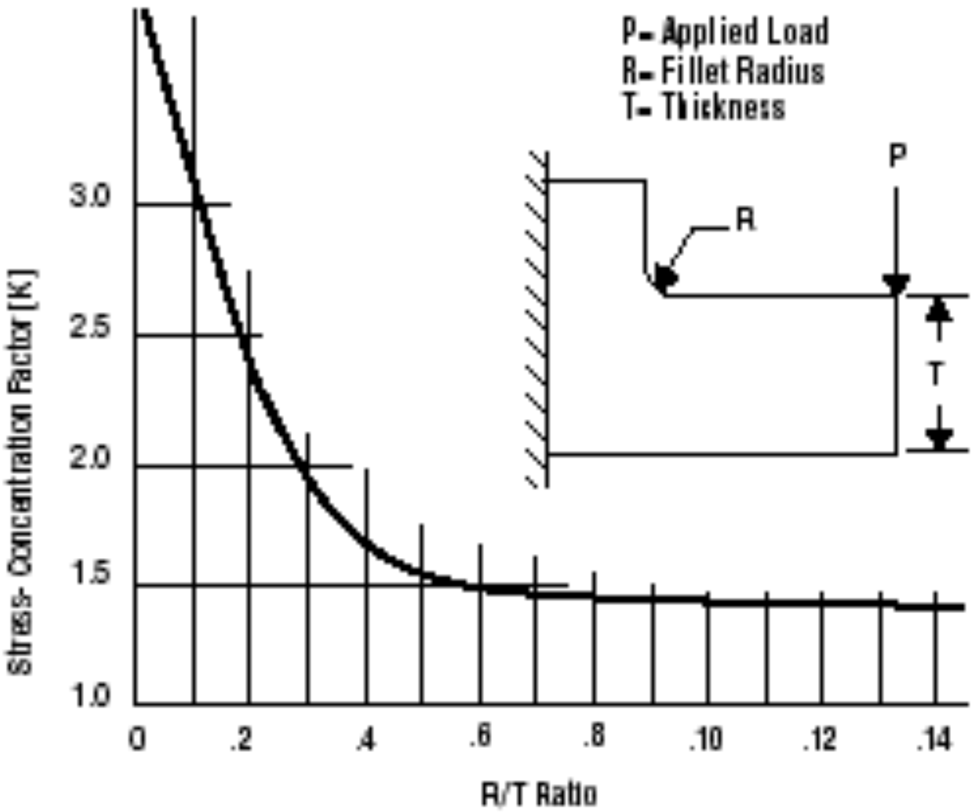
Part Design Heuristics



Section Thickness - Uniformity is critical to minimize warpage, distortion and internal stress

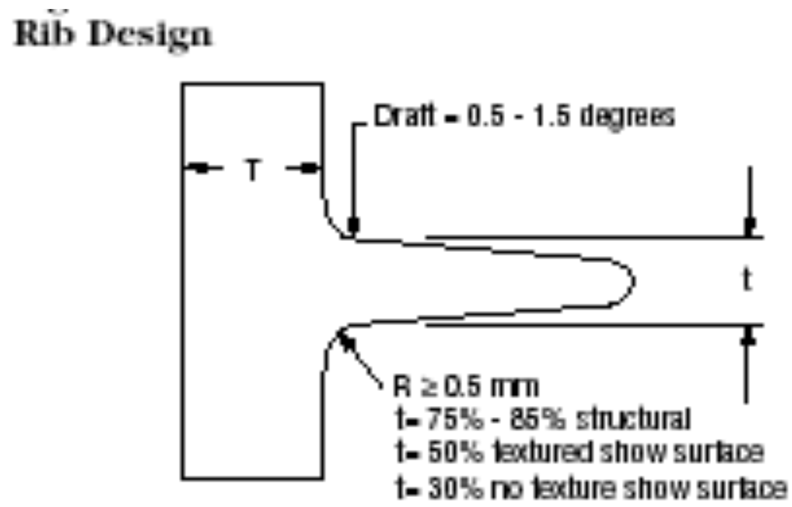


Radii - sharp corners act as stress concentrators. A min radius of 0.5 mm is recommended.



Draft Angles - 0.5 to 1.5o is required to facilitate part ejection

Ribs - add strength
and facilitate flow



Tolerances - must provide good packing to minimize shrinkage. Multi-cavity molds with non-uniform runner system are harder to maintain tolerances

Standards & Practices of Plastics Moulders

Material Polyamide (PA)

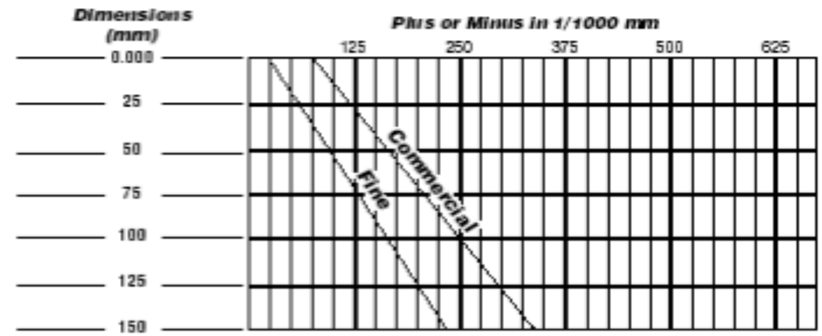
Note: The *Commercial values* shown below represent common production tolerances at the most economical level. The *Fine values* represent closer tolerances that can be held but at a greater cost. Addition of reinforcements will alter both physical properties and dimensional stability. Please consult the manufacturer.

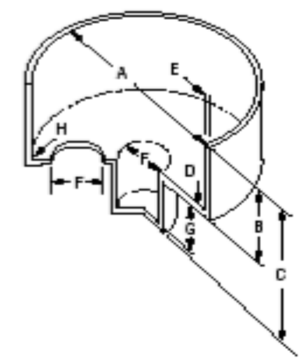
Drawing Code

A=Diameter (See note #1)

B=Depth (See note #3)

C=Height (See note #3)





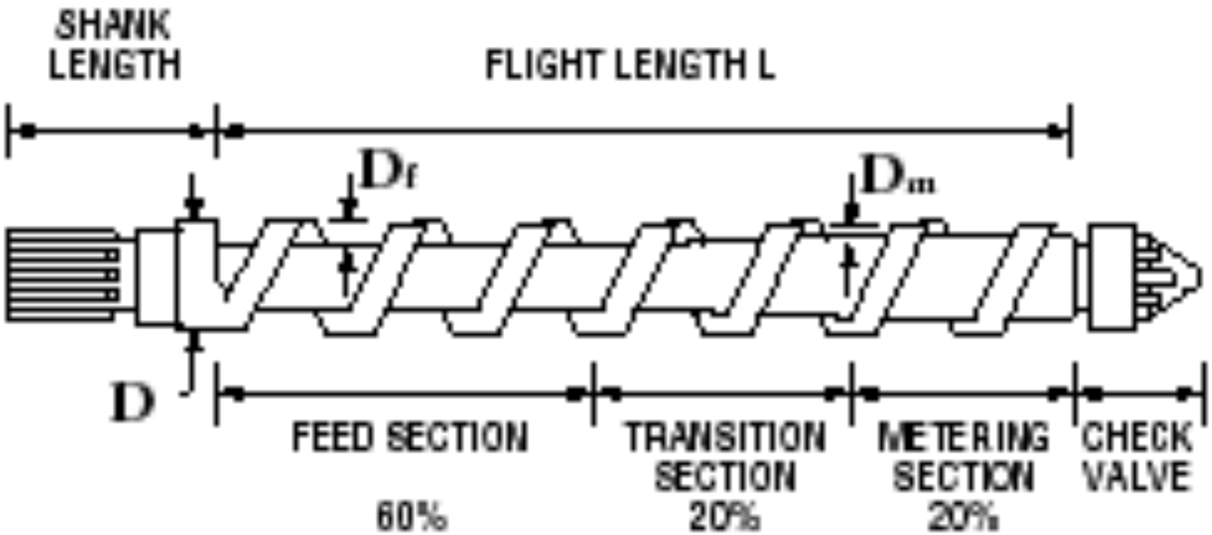
Drawing Code	Dimensions (mm)	Comm. ±	Fine ±
		0.076	0.05
D=Bottom Wall	(See note #3)	0.1	0.076
E= Side Wall	(See note #4)	0.127	0.076
F=Hole Size Diameter (See note #1)	0.000-3.175	0.05	0.025
	3.176-6.35	0.076	0.05
	6.36-12.7	0.076	0.05
	> 12.8	0.127	0.076
G=Hole Size Depth (See note #5)	0.000-6.35	0.1	0.05
	6.36-12.7	0.1	0.076
	12.8-25.4	0.127	0.1
H=Corners, Ribs, Fillets	(See note #6)	0.53	0.33
Flatness (See note #4)	0.000-76.2	0.254	0.1
	76.3-152.4	0.38	0.18
Thread Size (Class)	Internal	1	2
	External	1	2
Concentricity	(See note #4)(F.I.M.)	0.127	0.076
Draft Allowance per side		1.5°	0.5°
Surface Finish	(See note #7)		
Color Stability	(See note #7)		

Reference Notes

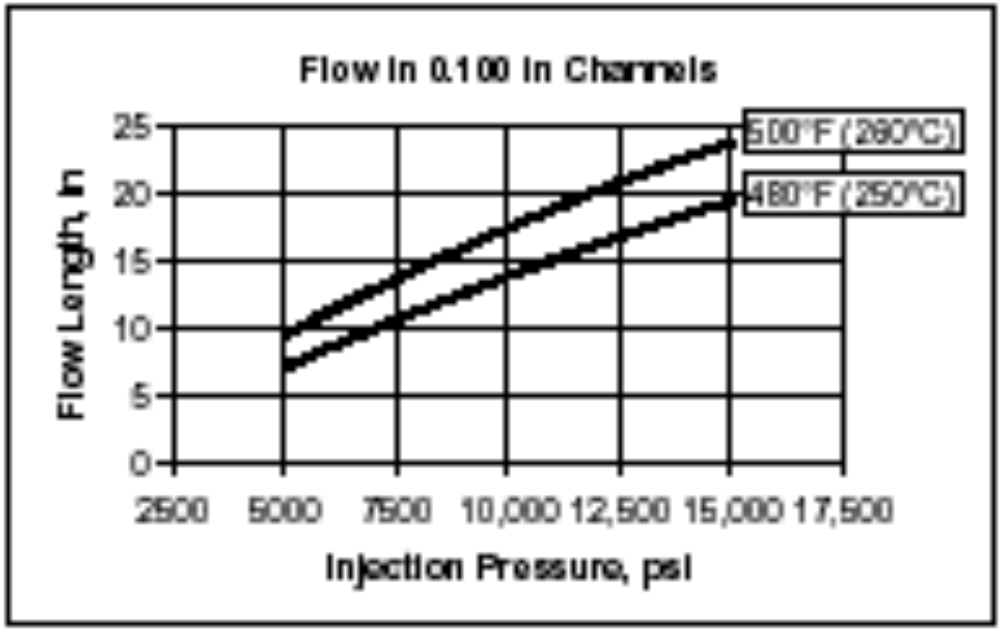
1. These tolerances do not include allowance for aging characteristics of material.
2. Tolerances are based on 3.175 mm wall section.
3. Parting line must be taken into consideration.
4. Part design should maintain a wall thickness as nearly constant as possible. Complete uniformity in this dimension is sometimes impossible to achieve. Walls of non-uniform thickness should be gradually blended from thick to thin.
5. Care must be taken that the ratio of the depth of a cored hole to its diametre does not reach a point that will result in excessive pin damage.
6. These values should be increased whenever compatible with desired design and good moulding techniques.
7. Customer-Moulder understanding is necessary prior to tooling.

Machine Selection

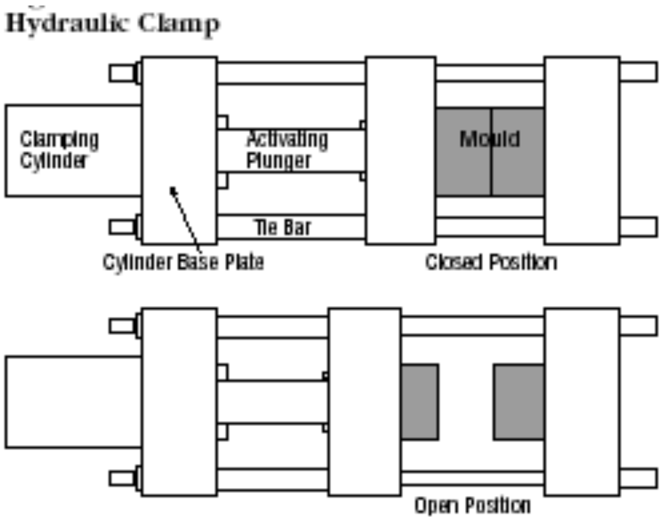
Screw and Barrel Sizing - L/D Ratio, Compression Ratio, Barrel Size in GPPS

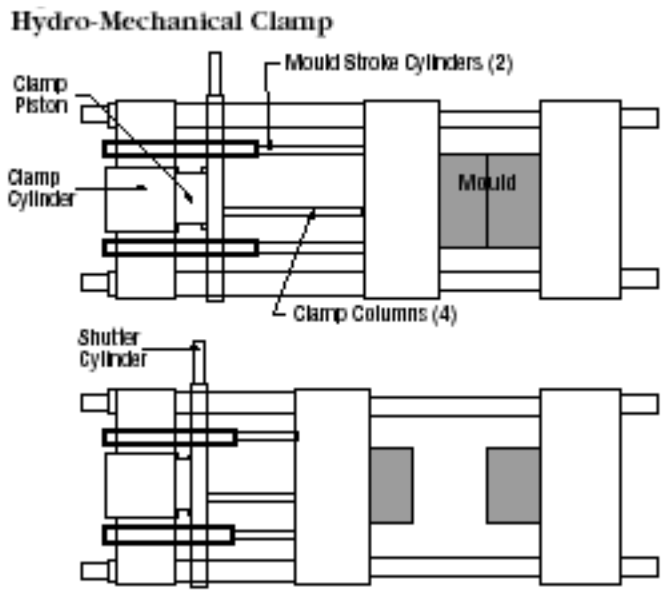


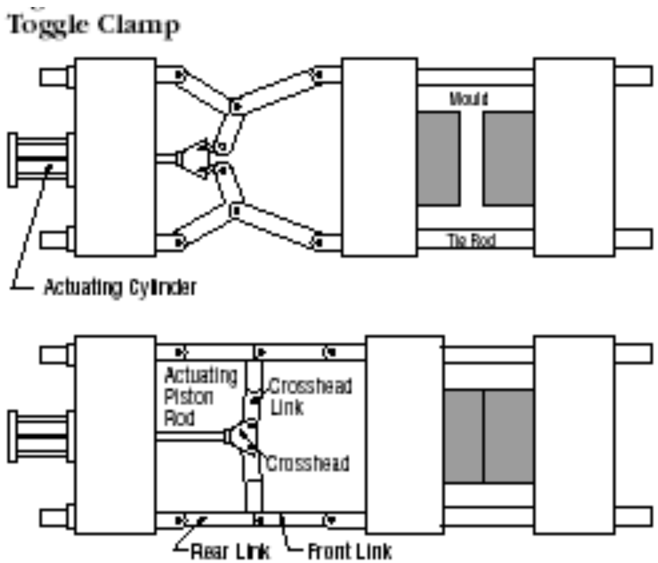
Injection Pressure Requirement



Clamp Size - insufficient clamp pressure will results in Flash and Inability to mold a component that fully packed out







Clamp Types

Clamp Style	Advantages	Limitations
Hydraulic	Fast mould set-up. Easily read clamp pressure. Low maintenance. Low platen deflection. Force concentrated at center of platen.	Requires high volume of hydraulic oil. Energy Inefficient. Must overcompensate due to compressibility of oil. Not floorspace efficient.
Toggle	Less expensive. Fast clamp motion. Difficult to adjust. Energy efficient.	Requires more maintenance. Force not concentrated at center of platen.
Hydro-Mechanical	Floorspace efficient. Force concentrated at center of platen. Requires low volume of hydraulic oil. Fast clamp motion.	Expensive.

MOULD & TOOLING CONSIDERATIONS

- Tool Steel Materials
- Texturing and Surface Finish
- Sprue Bushing
- Sprur Puller
- Runner System
- Cold Runner Design
- Hot Runner system
- Gate Design
- Gate Location
- Venting
- Cooling
- Shrinkage

Tool Materials

Selecting the right steel can increase the servicelife of the mold as well as greatly improve the maintenance of any texturing or graining of the part surface.

Material	Recommended Steel	Steel Characteristics	Typical Hardness (Rockwell C)
Homopolymers & Unfilled Copolymers	AISI P20	Medium alloy mold steel	30 - 36 Rc
Reinforced Materials	AISI S7 AISI H13 AISI 420	Shock resistance Hot work (Cr based) Stainless Steel	54 - 56 Rc 50 - 52 Rc 50 - 52 Rc

Surface Finish

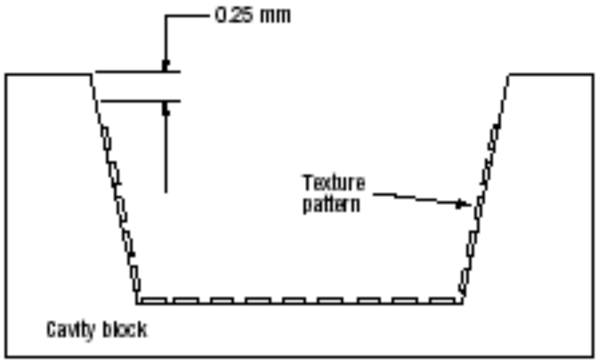
- The mold should be heat treated prior to texturing. Typical texture depths range between 0.01 mm to 1.25 mm.
- The ensure ease of part ejection and reduce the chance for streaks and scuff marks, draft should be incorporated into walls containing a texture.

Recommendation:

1.5 degrees draft for each 0.025 mm grain depth + 1 degree draft

To protect shut off region at the parting line, maintain an area around the parting line perimeter of 0.25 mm without the textured pattern

Typical Textured Cavity



Texture Specification Suggestions:

i. Heat treatment of mold prior to texturing
ii. The depth of heat treat into the steel should exceed that of the texture to ensure consistent texturing
iii. For proper part release from the textured side walls, do not exceed a depth of etching of 0.02 mm maximum per 0.5 degree draft
iv. Texturing of the core half of the mold is not recommended

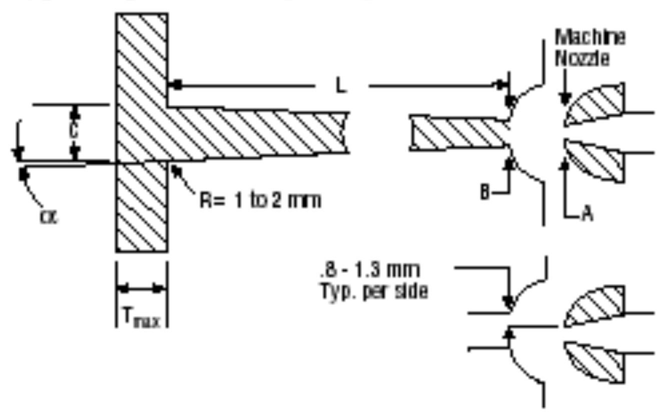
Sprue Bushing

For ease of part and runner system ejection, a minimum taper of 1.5 to 3.5 degree is recommended over the length of the sprue bushing.

Inner diameter of machine nozzle < Outer diameter of sprue bushing. Why?

Grinding and polishing only in a direction normal to the direction of demolding. Why?

Typical Sprue Bushing Design Recommendation



$C \geq T_{max} + 1.5$ mm

$B \geq A + 1.0$ mm

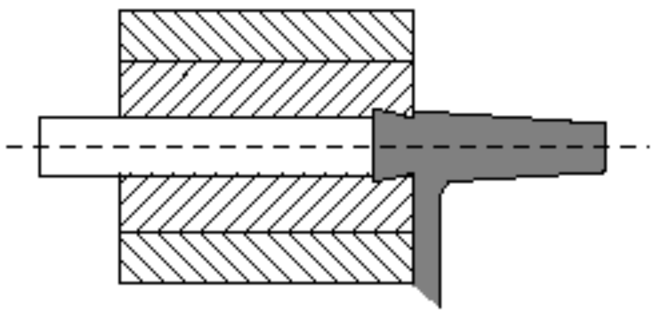
$\alpha \geq 1.5°$

$\tan \alpha \geq \frac{C - B}{2L}$

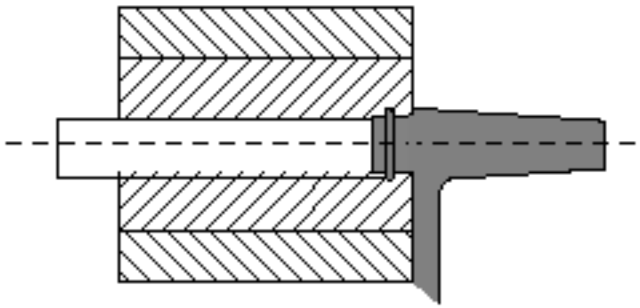
T_{max} = Maximum runner thickness.
Dia. A = Diameter of opening at end of machine nozzle.
Dia. B = Diameter of sprue at machine nozzle interface.
Dia. C = Diameter of sprue bushing at part intersection.
L = Overall length of sprue.

Sprue Puller

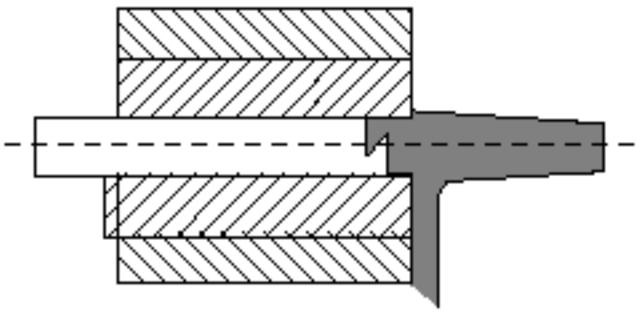
Backdraft Taper Sprue Puller Design



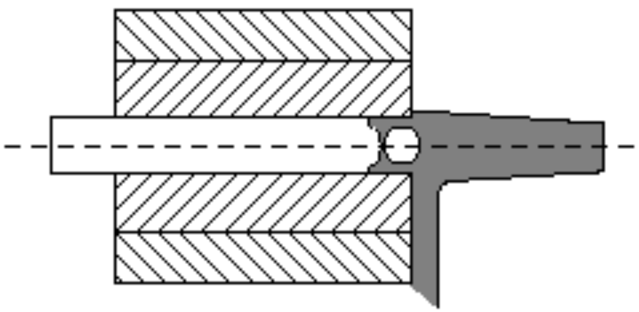
Undercut Ring Sprue Puller Design



Z Pin Sprue Puller Design

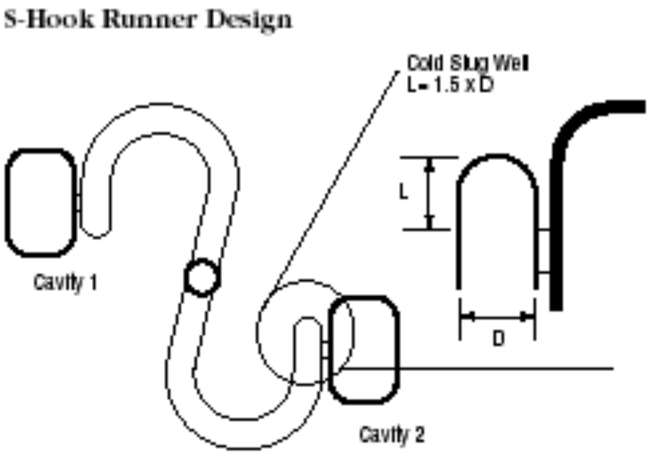


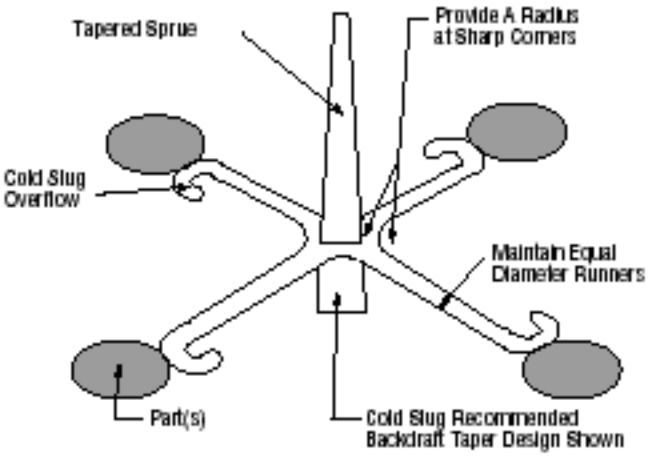
Ball Sprue Puller Design



In a multi-cavity mold (runner system), sprue puller is incorporated to ensure sprue removal from stationary side of the mold. Also acts as Cold Slug well. Cold Slug?

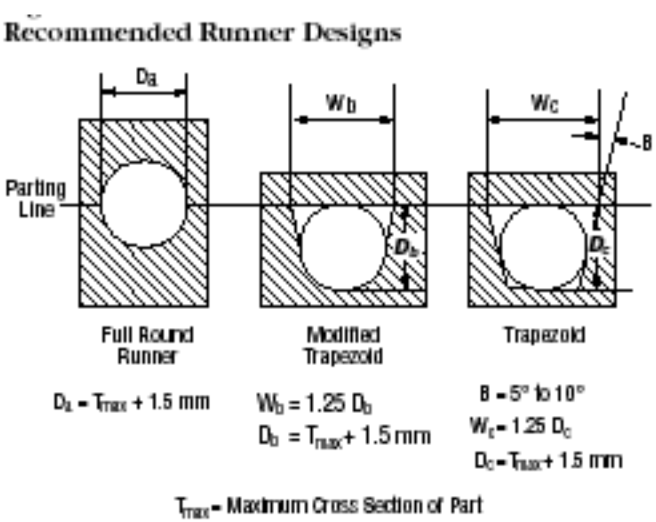
Cold Slug Well

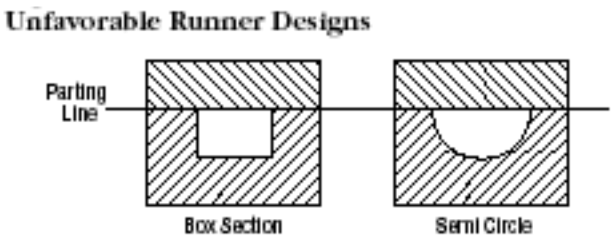




Cold slug can lead to surface imperfections and weaken the part structure. Slug catcher?

Runner System





Runner Style	Advantages	Disadvantages
Full Round	Smallest surface to cross section ratio. Slowest cooling rate. Low heat and frictional loss. Center of channel freezes last; maintains hold pressure.	Machining into cavity/ core difficult.
Modified Trapezoid	Easier to machine; usually one half of tool only. Offers similar advantages of full round.	More heat loss and scrap compared to full round.
Trapezoid	Easy to machine.	More heat loss than modified trapezoid.
Box Section	Easy to machine.	Small cross section Reduced ability to transfer pressure.
Semi-Circle	Easy to machine.	Smallest cross-section. Most inefficient runner design. Poor pressure transmission into cavity.

Objectives of Runner System Design:

i. Minimize restrictions of flow in the runner such as inconsistent cross section
ii. Design for ease of part ejection
iii. Short as possible to minimize loses in pressure and temperature
iv. Runner cross section large enough to equal/exceed gate freeze-off time. Why?
v. Runner system is not the limiting factor when reducing cycle time. Meaning?
vi. Minimize runner weight to part weight

Runner Balancing

The runner length from the sprue to each cavity should all be of the same diameter and length. This will ensure part to part consistency. How?

Recommended runner diameters and corresponding runner length and part thickness:

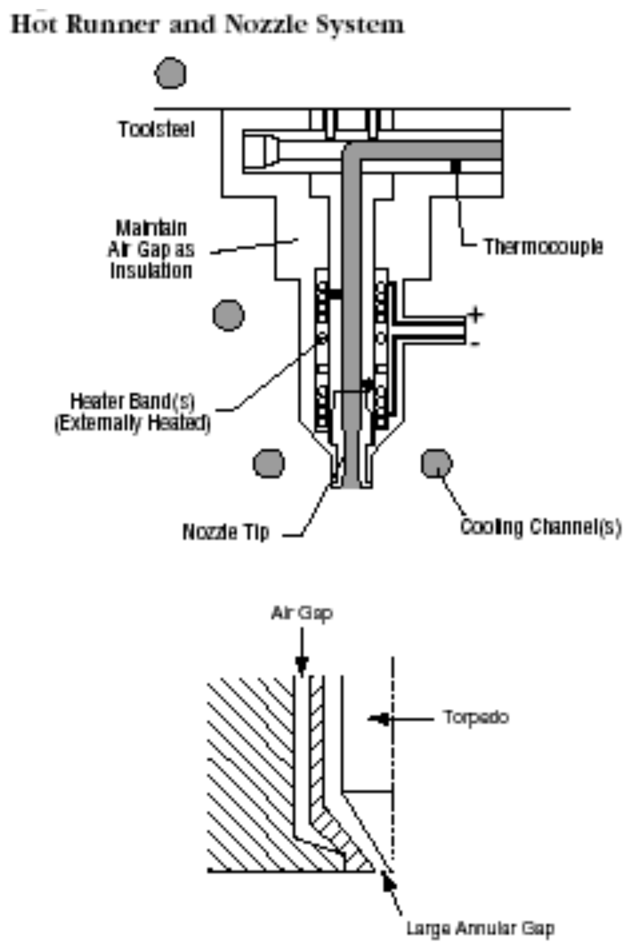
Primary Runner Diameter	Maximum Length	Maximum Part Thickness
3.2 mm-4.8 mm	150 mm	4.8 mm
6.4 mm-8.0 mm	300 mm	12.5 mm
9.5 mm	380 mm	19.0 mm

Hot Runner Design

Why Hot Runner?

Design Recommendations:

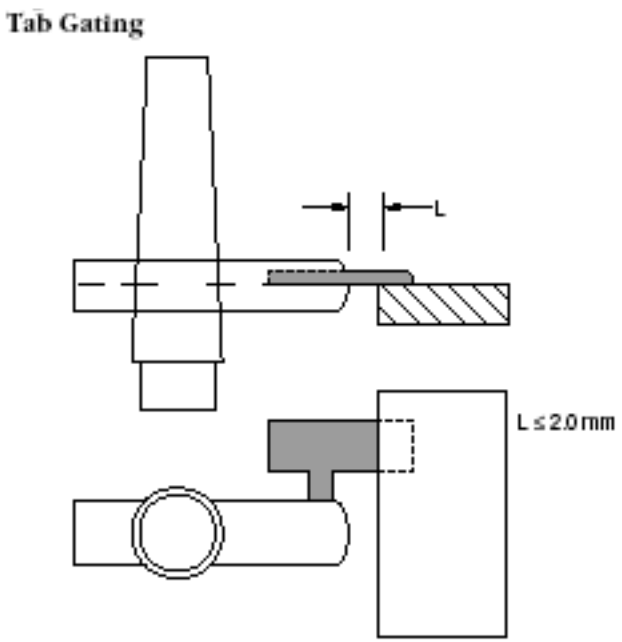
- avoid traps
- avoid cold spot
- minimize heat transfer to mold steel
- avoid undesirable heat transfer from hot runner/ manifold



Gate Design

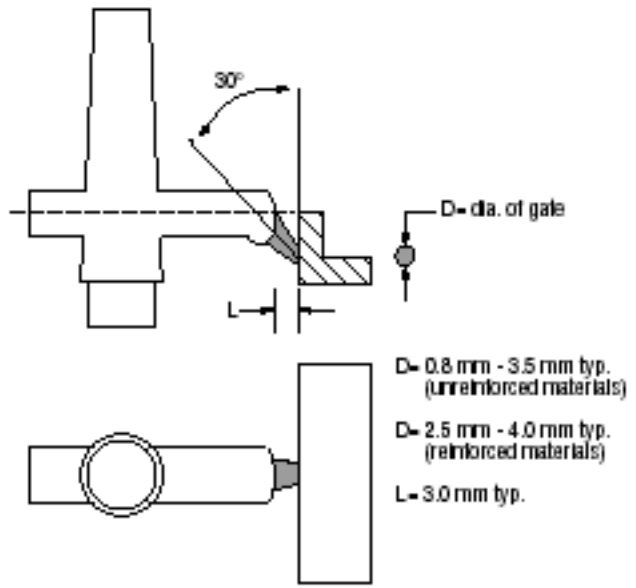
smallest cross section - depend on part geometry

Tab or film gating - used where flatness is critical or for large surface area (warping concern). Post injection removal of gate is required.

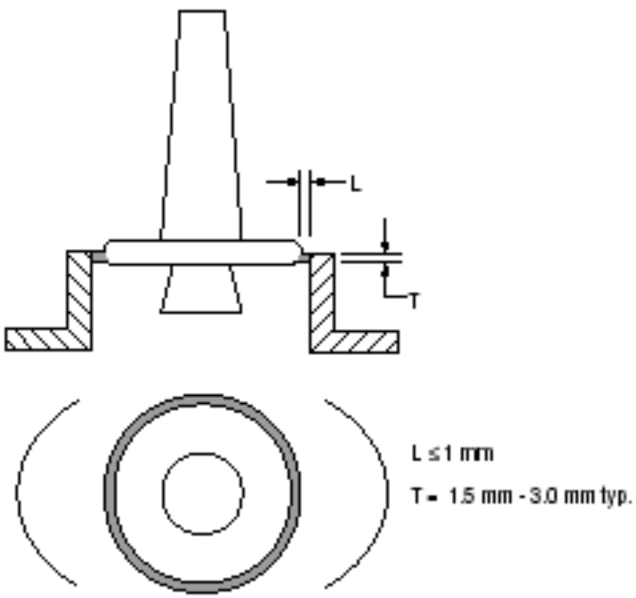


Sub-gating can be designed to provide automatic de-gating of the part from the runner system during the ejection cycle. dependent on part size.

Sub (Tunnel) Gating



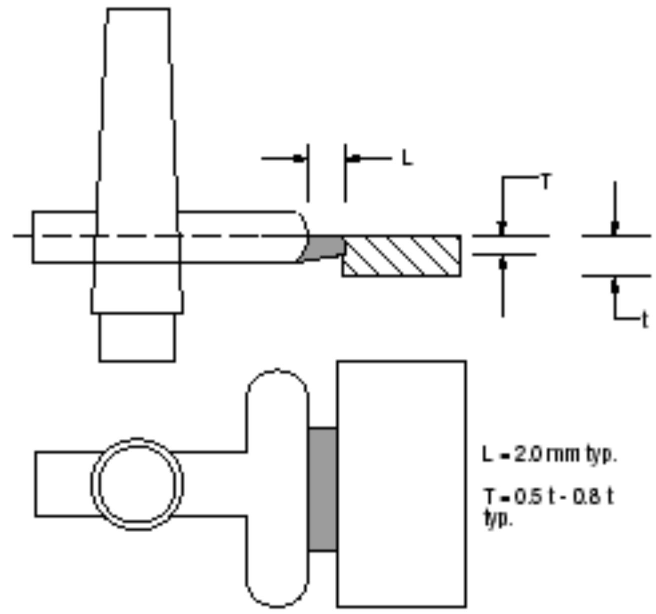
Diaphragm Gating



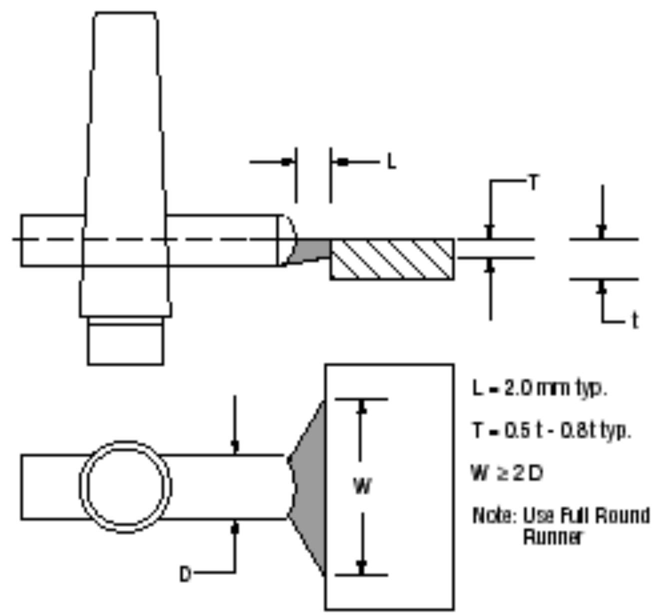
Diaphragm gate is used when molding cylindrical parts requiring a high level of concentricity and weld strength. Post molding de-gating is required.

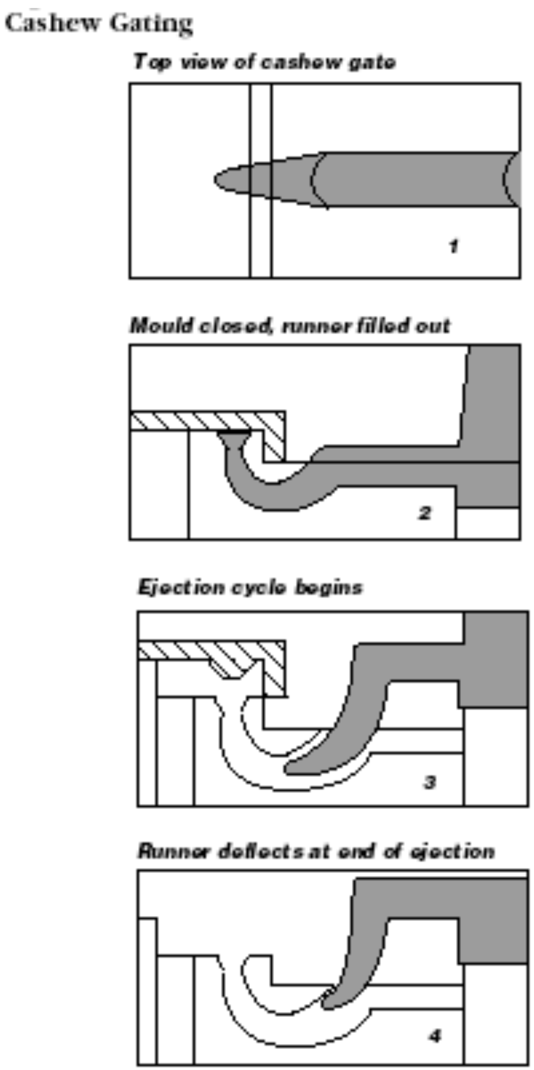
Fan/edge gate is used to feed flat, thin sections. Proven to reduce warping.

Edge Gating



Fan Gating





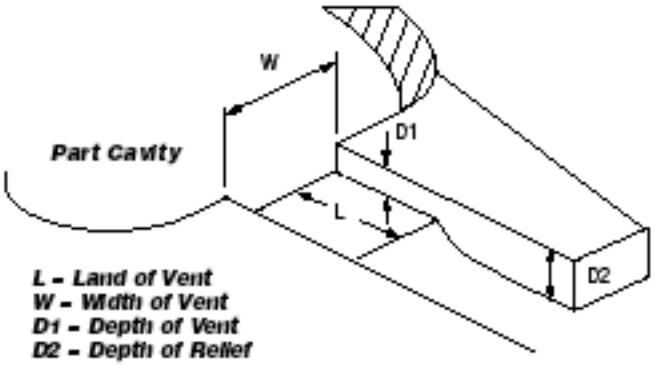
Cashew gating can be highly effective when using flexible material. Only for unfilled polymers.

Gate Position

The gate should be located at the thickest wall section to ensure that holding pressure remains effective. Gate location effects shrinkage, distortion/warpage and physical properties. tensile and impact strength are highest in the direction of flow, esp. for filled materials.

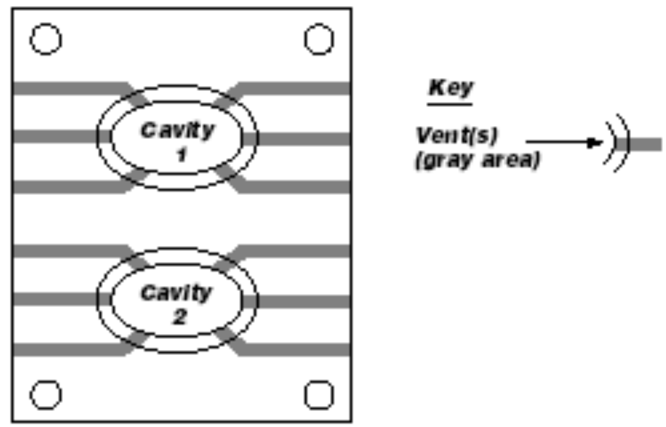
Venting

Typical Parting Line Vent Design

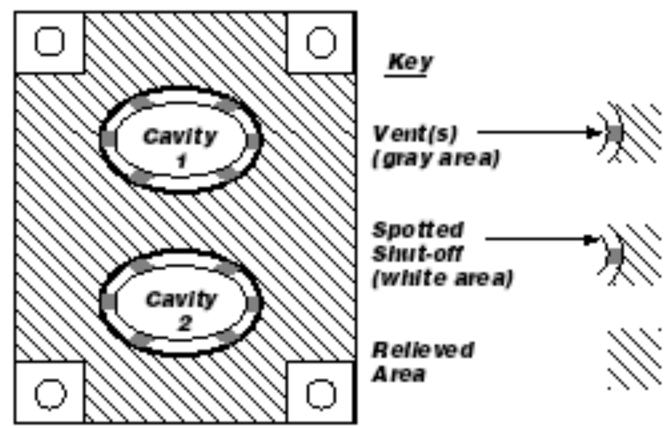


Material Type	Vent Dimensions L	W	D1	D2
Unfilled	0.75-1.5 mm	9.53-127 mm	0.013-0.025 mm	0.5 mm
Mineral Filled	0.75 mm	9.53-12.7 mm	0.05 mm	1 mm
Glass Filled	0.75mm	9.53-12.7mm	0.05mm	1 mm

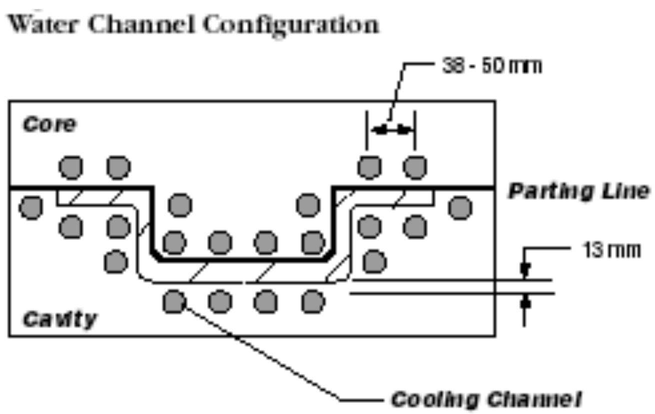
Independent Venting Channels

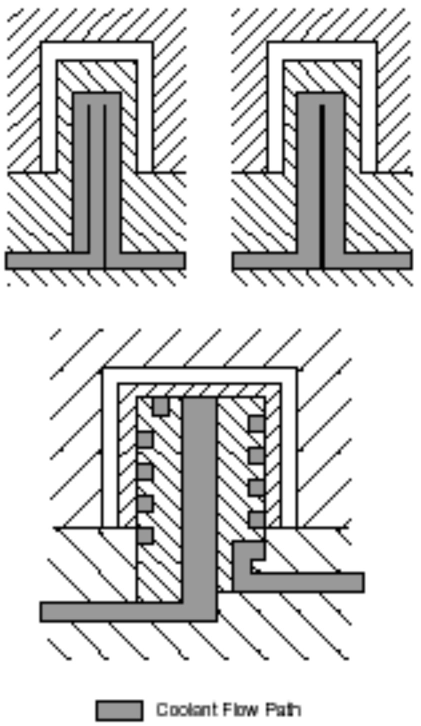


Continuous Venting Example



Cooling Channels





Inconsistent mold temperatures may lead to:

i. non-uniform part surface finish

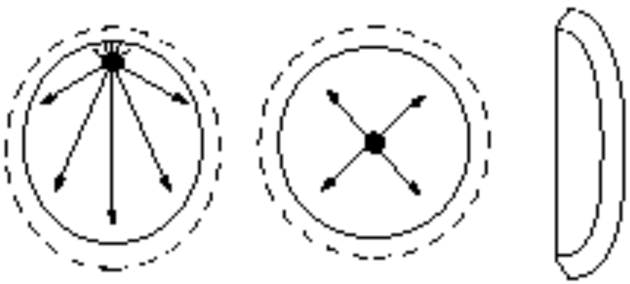
ii. Non-uniform part shrinkage and warpage

iii. lack of control of part dimensions

iv. potention binding of tightly filling cavity

Shrinkage

Gating Relationship to Scrinkage Patterns



Eccentric Gating

Centric Gating

Shrink can result in part concavity

Effect of Mold Temperature on Shrinkage
(3″ × 5″ × 0.125″ Plaques)

	Mold Shrinkage, %		
Mold Temperature	100°F (38°C)	150°F (66°C)	200°F (93°C)
Unfilled Resins	1.4	1.6	1.8
30% Glass-reinforced Resins			
Flow (length)	0.17	0.19	0.21
Transverse (width)	0.65	0.75	0.85
Super Tough Resins	2.0	2.2	2.4

Effect of Part Thickness on Shrinkage
(Mold Temperature 150°F [66°F])

	Mold Shrinkage, %	
Part Thickness	0.125 in	0.250 in
Unfilled Resins	1.6	2.0
30% Glass-reinforced Resins		
Flow (length)	0.19	0.35
Transverse (width)	0.75	0.85
Super Tough Resins	2.0	—

When designing mold, it is important to specify material shrinkage.

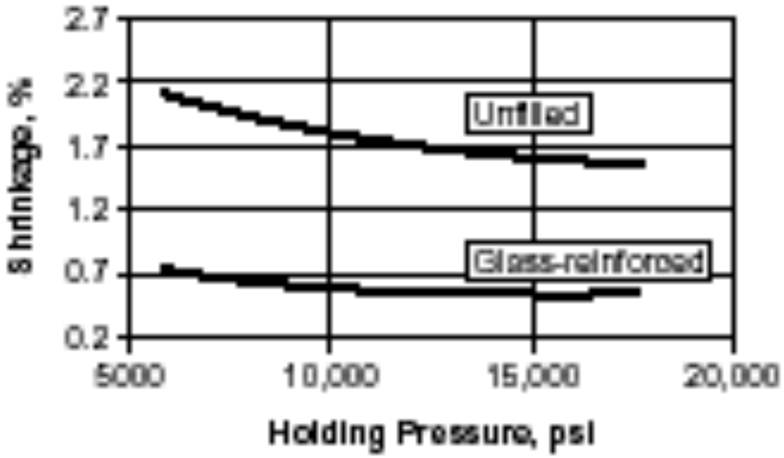
Factors may affect the shrinkage:

i. Location and size of gates

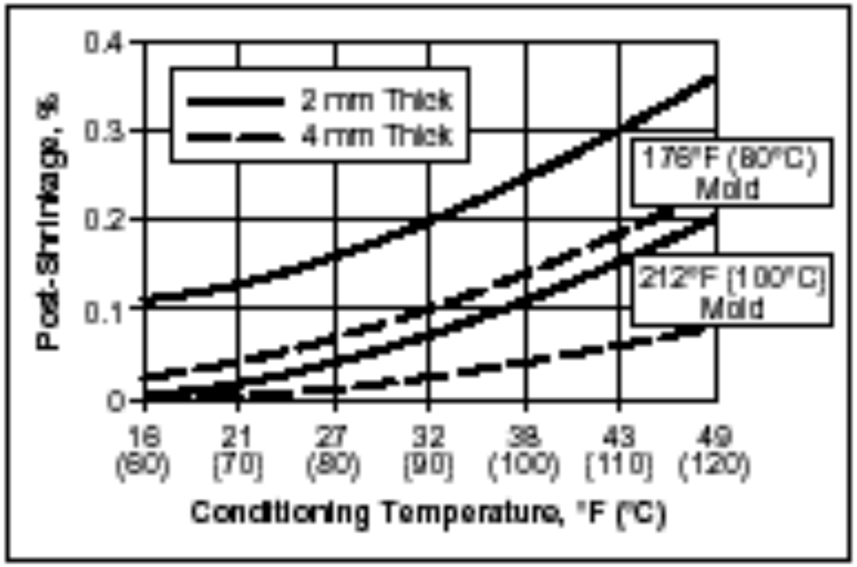
ii. Part designs - variations in thickness

iii. Increased filler content

iv. Filler orientation - multiple gates



2.7
2.2
1.7
1.2
0.7
0.2
Shrinkage, %
Unfilled
Glass-reinforced
5000
10,000
15,000
20,000
Holding Pressure, psi



0.4
0.3
0.2
0.1
0
Post-Shrinkage, %
2 mm Thick
4 mm Thick
176°F (80°C) Mold
212°F (100°C) Mold
16 (60)
21 (70)
27 (80)
32 (90)
38 (100)
43 (110)
49 (120)
Conditioning Temperature, °F (°C)

Brittleness

Possible Cause	Suggested Remedy
1. Melt temperature too low.	a. Increase melt temperature.
2. Material overheated, resulting in molecular breakdown.	a. Decrease melt temperature. b. Residence time in cylinder excessive for size-use smaller barrel. c. Decrease overall cycle. d. Reduce back pressure.
3. Contamination by foreign material or excessive pigment usage.	a. Inspect resin for contamination. b. Purge injection cylinder thoroughly. c. Keep hopper covered. d. Review material handling procedures for regrind usage. e. Reduce filler or pigment loading.
4. Excessive amounts of regrind.	a. Reduce regrind mixed with virgin. b. Regrind level dependent upon application: general rule-25-30%. c. Keep hopper covered. d. Review material handling procedures for regrind usage. e. Reduce filler or pigment loading.
5. Injection rate too slow.	a. Increase injection or first stage pressure. b. Increase boost time.
6. Improper gate location or size.	a. Relocate gate away from potential stress area. b. Increase gate size to obtain optimum filling.
7. Moisture in material during processing.	a. Review material handling to eliminate moisture pick-up. b. Dry material prior to moulding. c. Utilize hopper dryers.
8. Dry as molded properties.	a. Moisture condition parts to increase toughness.

Bubbles, Voids

Possible Cause	Suggested Remedy
1. Excessive internal shrinkage.	a. Increase packing pressure. b. Increase injection forward time. c. Increase gate thickness. d. Minimise heavy sections in part design. e. Increase feed, ensure cushion. f. Replace check valve if cushion cannot be held.
2. Melt temperature too high.	a. Decrease melt temperature.
3. Moisture in material.	a. Review material handling to eliminate moisture pick-up. b. Dry material prior to moulding. c. Utilise hopper dryers. d. Review percent of regrind.
4. Air entrapment.	a. Add mould venting. b. Relocate gate. c. Reduce clamp pressure to allow parting line vents to work.
5. Condensation on mould surface.	a. Wipe mould surface thoroughly with solvent. b. Increase mould temperature.

Burn Marks	
Possible Cause	**Suggested Remedy**
1. Melt temperature too high.	a. Decrease melt temperature.
2. Air entrapped in mould.	a. Vent cavity at final point of fill. b. Decrease first stage pressure or injection speed. c. Relocate gate. d. Clean vents and/or enlarge vents. e. Enlarge gates. f. Reduce clamp pressure to allow parting line vents to work.
3. Injection rate too fast.	a. Decrease first stage pressure. b. Decrease boost time. c. Decrease injection rate.
4. Moisture in material.	a. Review material handling to eliminate moisture pick-up. b. Dry material prior to moulding. c. Utilise hopper dryers.

Cracking, Crazing	
Possible Cause	**Suggested Remedy**
1. Packing excessive material into the mould.	a. Decrease packing pressure. b. Decrease shot size.
2. Non-uniform or too cold a mould temperature.	a. Increase mould temperature. b. Supply uniform cooling to the cavity.
3. Knockout system poorly designed.	a. Redesign knockout system for balanced ejection forces.
4. Inadequate draft angles or excessive undercuts	a. Re-work mould.

Dimensional Variations

Possible Cause	Suggested Remedy
1 Non-uniform feeding of material.	a. Adjust temperature profile for optimum feeding. b. Increase shot size to maintain uniform cushion. c. Replace check valve if cushion cannot be held.
2. Large variation in cylinder temperature due to inadequate or defective controllers.	a. Replace or calibrate controllers.
3. Unbalanced runner system, resulting in non-uniform cavity pressure.	a. Increase holding pressure to maximum. b. Increase injection rate. c. Increase gate size progressively from sprue to provide uniform filling.
4. Regrind not uniformly mixed with virgin.	a. Review regrind blending procedure. b. Decrease percentage of regrind.
5. Moulding conditions varied from previous run.	a. Check moulding records to ensure duplication of process conditions.
6. Part distortion upon ejection.	a. See "Sticking in mould."

Discoloration, Contamination

Possible Cause	Suggested Remedy
1. Material overheated in injection cylinder.	a. Decrease melt temperature. b. Decrease overall cycle. c. Residence time in cylinder excessive for shot size-use smaller barrel. d. Decrease nozzle temperature. e. Decrease screw RPM. f. Decrease back pressure. g. Check calibration of cylinder controllers. h. Check barrel and nozzle heater bands and thermocouples.
2. Burned material hanging up in cylinder or nozzle (black specks).	a. Purge injection cylinder. b. Remove and clean nozzle. c. Remove and inspect non-return valve for wear. d. Inspect barrel for cracks or gouges. e. Decrease injection rate.
3. Material oxidized by drying at too high a temperature.	a. Reduce drying temperature to 80°C.
4. Contamination by foreign material.	a. Keep hopper covered. b. Review material handling procedures for virgin and regrind. c. Purge injection cylinder.

Excessive Cycle Time

Possible Cause	Suggested Remedy
1. Poor mould cooling design.	a. Increase mould cooling in hot spots. b. Ensure fast turbulent flow of water through cooling channels.
2. Platen speeds excessively slow.	a. Adjust clamp speeds to safely open and close quickly. b. Low pressure close time excessive, adjust clamp positions and pressures to safely and efficiently open and close mould.
3. Melt temperature too high.	a. Decrease melt temperature.
4. Mould temperature too high.	a. Decrease mould temperature.
5. Screw recovery time excessive.	a. Check machine throat and hopper for blockage or bridging. b. Check for worn screw and barrel especially in the feed zone.

Warpage

Possible Cause	Suggested Remedy
1. Moulded part ejected too hot.	a. Decrease melt temperature. b. Decrease mould temperature. c. Increase cooling time. d. Cool part in warm water after ejection. e. Utilise shrink fixture.
2. Differential shrinkage due to non-uniform filling.	a. Increase injection rate. b. Increase packing pressure. c. Balance gates and runners. d. Increase/decrease injection time. e. Increase runner and gate size.
3. Differential shrinkage due to non-uniform wall thickness.	a. Provide increased cooling to thicker sections. b. Increase mould cooling time. c. Operate mould halves at different temperatures. d. Redesign part with uniform wall sections.
4. Knockout system poorly designed.	a. Redesign knockout system for balanced ejection forces.
5. Melt temperature too low.	a. Increase melt temperature to pack out part better.
6. Glass fibre orientation.	a. Relocate gate.

Processing Quality Checklist

▲ Increase
▼ Decrease

	Rapid Response							Slower Response				
	Back Pressure	Boost Pressure	Injection Rate	Mould Open Speed	Packing Pressure	Screw Decompression	Screw Forward Time	Cycle Time	Melt Temperature	Mould Temperature	Nozzle Temperature	Possible Causes and/or Solutions
Bubbles/Voids		▲			▲	▼	▲		▼	▲		Improve venting, Increase gate size, Min. thick sections
Burn Marks		▼	▼				▼		▼			Improve venting, Relocate gate
Discoloration	▼		▼					▼	▼		▼	Purge barrel/Clean screw/Barrel/Nozzle
Distortion upon Ejection				▼	▼							Check mould surface for smooth release
Erratic Screw Retraction	▲								▲			Check for screw wear
Flash	▼	▼	▼		▼				▼	▼		Mould needs adjustment/Clamp tonnage too low
Flow Lines			▼						▲	▲		Increase gate size, Check venting
KO Pin Penetration				▼	▼			▲				Poor mould cooling
Lamination		▲	▲		▲				▲	▲		Contaminated material, Increase gate size
Nozzle Drool	▼					▲			▼		▼	Use reverse taper nozzle
Part Sticking in Mould		▼	▼		▼		▼	▲	▼	▼	▲	Check for damaged mould surfaces
Poor Weld Lines		▲	▲▼		▲				▲	▲		Improve venting, Relocate gate, Clean vents
Short Shots		▲	▲		▲		▲		▲	▲		Increase gate size, Increase shot size
Shot to Shot Variation			▲		▲		▲					Non-return valve leakage
Sink Marks		▲			▲		▲		▲▼			Increase gate size
Splay Marks			▼		▲	▼			▼	▲	▼	Wet material
Sprue Sticking					▼		▼				▲	Damaged sprue bushing, Increase taper
Surface Blemish		▲	▲		▲				▲	▲		Wet material
Unmelted Pellets	▲							▲	▲			Check heater bands
Warpage			▲▼		▲		▼	▲	▲▼	▼		Check cooling line location