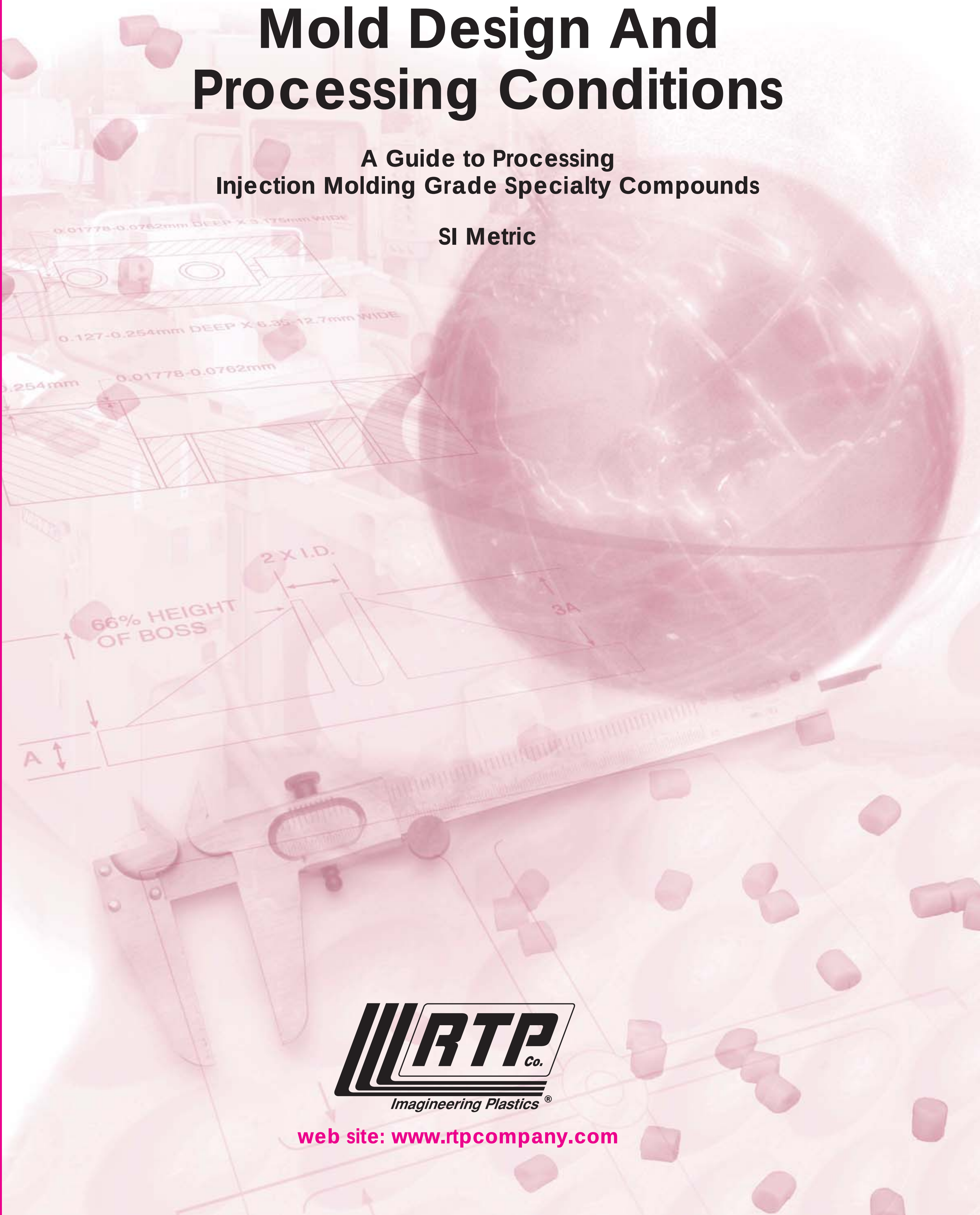
CONDENSED

Mold Design And Processing Conditions

A Guide to Processing Injection Molding Grade Specialty Compounds

SI Metric



RTP Co.
Imagineering Plastics®

web site: www.rtpcompany.com

Operating Conditions

Standard Molding Equipment

Machine Type: We recommend using a screw injection machine for molding reinforced thermoplastics. A screw injection machine improves melt homogeneity, reduces variations in the molded parts, and minimizes degradations and cold spots of the polymer melt.

Machine Size: The machine should be sized to use approximately 50-70% of machine barrel capacity per shot. This maintains a short barrel residence time and eliminates material degradation. The machine should have adequate clamp pressure to obtain 55-110 N/mm^2 of projected surface area. Generally, a reinforced material requires 50-70% higher clamping pressures than a nonreinforced polymer of the same type.

Molding Conditions

Drying: Successful molding of reinforced thermoplastics requires adequate drying. Inadequate drying can cause extremely erratic molding conditions and less than perfect molded parts. Excessively wet materials outgas and can undergo a viscosity change during processing. This may cause brittleness, blisters, voids, silver streaking and poor surface finish. RTP Company materials are dried prior to packaging in moisture resistant containers. However, we recommend thoroughly drying the materials in a dehumidifying type dryer. This is important with hygroscopic materials but can also be essential for non-hygroscopic materials. Condensed surface moisture can dramatically affect high temperature molded parts. The recommended drying times are provided as guidelines; however, an actual moisture check is necessary.

Barrel Temperature: Typically the rear zone/zones are set 6-12° C cooler than the front zone and nozzle. Some modifications may be needed depending on part size and configuration.

Melt Temperature: Use the Processing Conditions Chart in this guide for recommended starting temperatures.

Mold Temperature: Use the Processing Conditions Chart in this guide for recommended starting temperatures. Normally, reinforced materials require higher mold temperatures than nonreinforced materials. Higher mold temperatures will achieve a smoother, more blemish-free surface by providing a resin rich skin on reinforced materials.

Injection Pressure: Injection pressure should be set low initially and increased to the point of filling the part just short of causing flash. Maximum pressure without flash generates optimum physical properties for your RTP Company material.

Back Pressure: Low back pressure (approx. 0.34 MPa) minimizes fiber breakage and property deterioration.

Injection Speed: Generally, the fastest possible cavity fill time is best. This minimizes glass orientation and maximizes weld line integrity

Screw RPM: The lowest possible rpm is recommended to minimize fiber breakage and screw recovery should be set accordingly. Slower rpm's result in a more uniform melt by minimizing shear heat buildup.

Tips To Improve Surface Conductivity

Compounds With Conductive Fillers

The parts may look beautiful but not be functional in static dissipative applications. The following processing conditions are important to successfully molding good, conductive parts.

1 All resins containing carbon black should be dried.

2. Slower fill rates improve surface conductivity. Fill rates less than 25.4 mm per second generally work best. Decreasing the fill speed has little effect when using subgates.

3. Higher melt temperatures and slower fill speeds tend to layer the carbon black particles on the part surface. A more conductive carbon black part typically has a more mottled, dull surface. A glossier surface is more resin rich and usually less conductive.

4. Decreased packing usually increases surface conductivity. Short shots can have much higher levels (one or two magnitudes) of surface conductivity, especially if measured at the edge of the flow front where the carbon particles lay closest to the surface.

5. Typical molding is usually less conductive near the gate and more conductive away from the gate, (i.e. 1K - 2K ohms with carbon black).

6. Mold temperature does not have a significant effect on conductivity.

7. Surface conductivity is usually independent of back pressure and screw speed.

Compounds With Conductive Fibers

Conductive materials based on carbon fiber typically require opposite molding conditions from conductive fillers.

1. Use the base resin drying requirements for carbon fiber reinforced compounds.

2. Increasing the fill speed usually increases surface conductivity 10^5 to $>10^4$ ohms/sq. (Caution: Excess fill speeds can break up carbon fibers and reduce mechanical properties.) All corners in the flow path should have generous radii.

3. With carbon fiber, high melt temperatures decrease surface resistivity (thereby increasing conductivity) because they increase the degree of packing. Short shots in carbon fiber are typically non-conductive.

4. A typical carbon fiber molding is more conductive near the gate.

5. Mold temperature does not have a significant effect on conductivity.

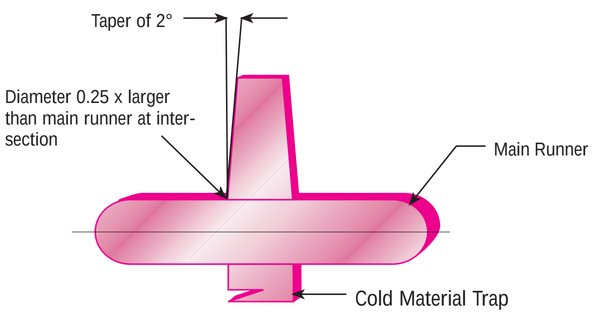
6. Back pressure and screw speed should be kept as low as possible, i.e. 0.17-0.34 MPa and 20-40 rpm respectively.

Mold Design

This guide is a condensed version. For more comprehensive information refer to RTP Company's Comprehensive Mold Design and Processing Conditions or contact our marketing department at 507-454-6900.

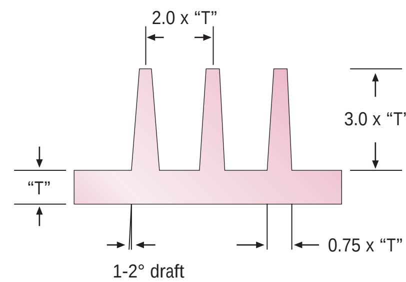
Sprue/Cold Material Trap Design

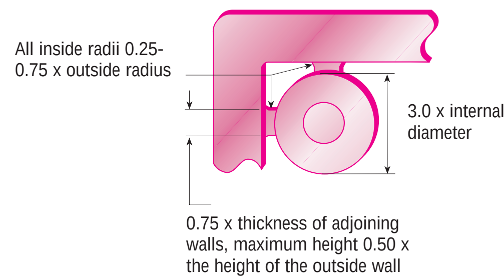
Sprues connect the nozzle of the injection molding machine to the main runner or cavity. The sprue should be as short as possible to minimize material usage and cycle time. The bushing should have a smooth, tapered internal finish that has been polished in the direction of the draw to ensure clean separation of the sprue and the bushing.



Ribs

Ribs should follow the proportional thickness guidelines shown below. If the rib is too thick in relation to the part wall, you may experience sinks, voids, warpage, weld lines, and longer cycle times. Position ribs in the line of flow to improve filling and prevent air entrapment.

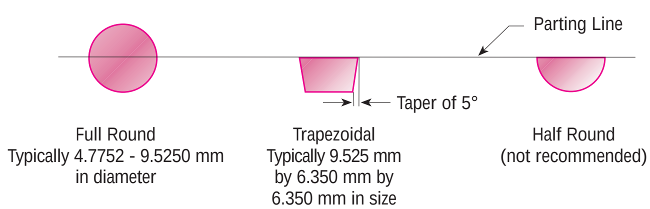




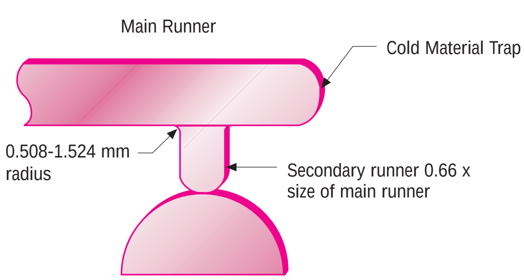
Runner Design

Runner systems convey the melted plastic from the sprue to the gate or part. The most efficient profile for a runner is circular (full-round). A less expensive, yet adequate, profile is a trapezoid, with tapers as shown in the diagram to ensure a good volume-to-surface area ratio. Half rounds are not recommended because of their poor perimeter to area ratio.

Main Runner - A channel that connects the sprue to the gate or part.



Secondary runner - A runner system located between the main runner and the molded part.



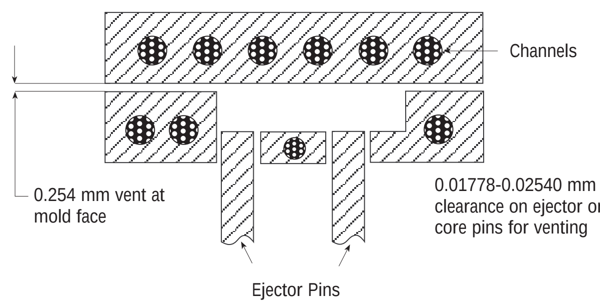
Key

Plastic Resin (material)

Tool Steel

Cooling

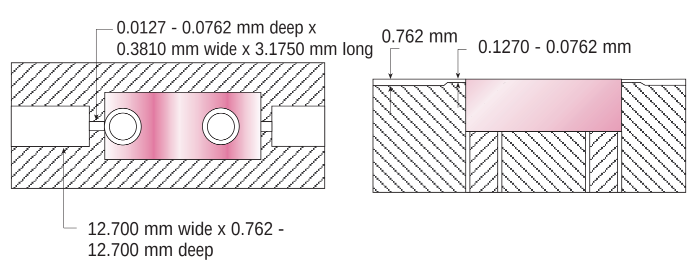
Molds must be provided with adequate cooling to take advantage of the faster cooling rates of reinforced compounds. Poor cooling results in rising mold temperatures and longer cycle times. Inadequate heating can result in voids, shorts and poor surface finish. Cooling and heating channels should be located directly in the mold inserts and cores if mold design permits.



Ejector Pins: Should be located on the heaviest sections of the part to minimize distortion when it leaves the core They should be balanced as much as possible over the part's surface. Reinforced thermoplastic require more pins due to lower mold shrinkage and greater potential for drag during ejection.

Venting

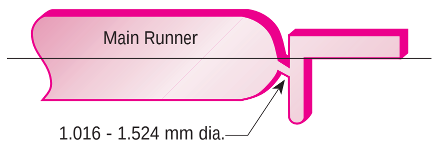
Proper venting of cavities is very important. Inadequate venting can result in gas burns, poor weld line strength and nonfilled parts. Too much venting can result in excessive flash and poor weld lines due to inadequate pressure buildup. Venting should primarily be located at the last point of fill and where weld line occur. Vent size depends on the viscosity of the polymer and can vary from 0.01270 mm to 0.07620 mm deep. Venting can also be used around knockout pins, moving cores and mold inserts.



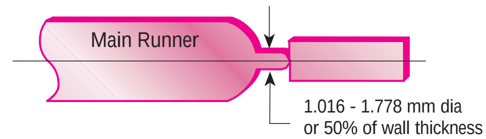
Gate Design

The gate serves as the entrance to the cavity and should be designed to permit the mold to fill easily. A cavity can have more than one gate. Gates should be small enough to ensure easy separation of the runner and the part but large enough to prevent early freeze-off of polymer flow, which can adversely affect the consistency of part dimensions. A variety of gate designs and locations are shown below:

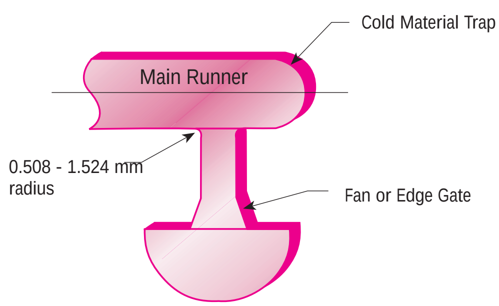
Submarine or Tunnel Gate - An edge gate located below the parting line or molded surface.



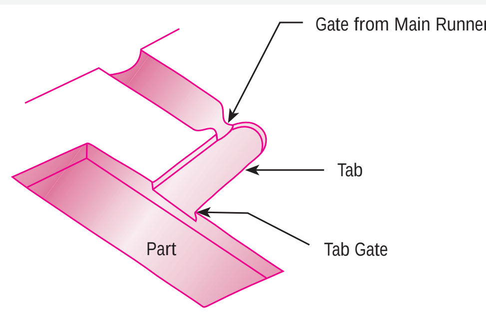
Pinpoint or Restricted Gate - A restricted opening between the runner and molded part. Normally used with thin wall parts.



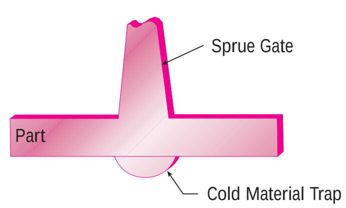
Fan or Edge Gate - A common gate located in the sidewall of the part to prevent restriction of resin flow. Normally used with multi-cavity, two-plate molds.



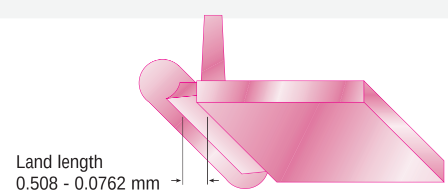
Tab Gate - Used for melt orientation when a large volume is needed for mold fill. The tab helps avoid surface splotches due to high shear, direct gating, or jetting.



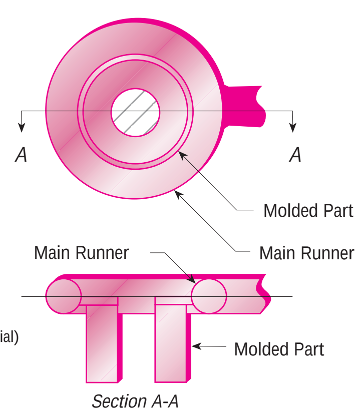
Sprue Gate - Recommended for single cavity molds requiring symmetrical filling. Usually used with circular parts.



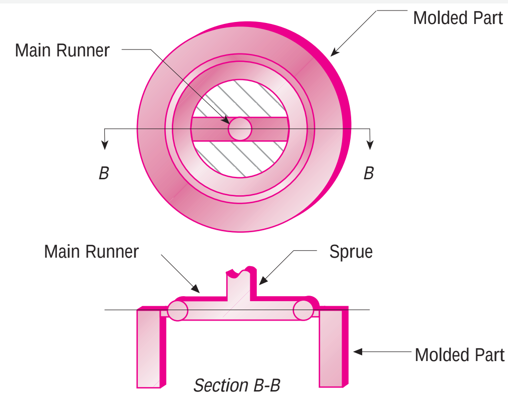
Flash Gate - A long, shallow, rectangular edge gate.



External Ring Gate - A system used when concentricity and a smooth interior surface are important. Can be used in multi-cavity molds.



Internal Ring Gate - A system used with large circular parts when concentricity and smooth outer surface are required. Can only be used with single-cavity molds.



Key

Plastic Resin (material)

Tool Steel

Troubleshooting Guide

SUGGESTED REMEDIES Perform in numerical order by column	Blisters	Brittleness	Excessive Flash	Gas Burns	Oversized Part	Poor Surface Finish	Poor Weld Lines	Short Shots	Silver Streaking	Sink Marks	Undersized Part	Voids	Warping
Change Gate Location							8						6
Clean Mold Faces			4	5		6							
Clean Vents	5			2			5	11				12	
Check for Material Contamination	6	4							4				
Check for Uneven Mold Temperature													1
Check Mold Faces for Proper Fit			5										
Dry Material	1	6	6	6		7			1			11	
Increase Amount of Material							4	1		8	7	10	
Increase Back Pressure						5		6			6		
Increase Clamp Pressure			2										
Increase Cooling Time										10			9
Increase Holding Pressure						8		12		11	1		
Increase Injection Hold Time							2			2	2	2	
Increase Injection Pressure						2	1	2		1		1	2
Increase Injection Speed						3	9	3	2		8		
Increase Injection Time										12	5		
Increase Mold Temperature		7			5	1	3	7			9	8	3
Increase Size of Gates								8	6	4	10	4	
Increase Size of Runners								9		5	11	5	
Increase Size of Sprue								10		6		6	
Increase Size of Vent				4			6	4					
Locate Gates Near Heavy Cross Sections										7		7	
Raise Material Temperature					7	4	10	5			4		
Redesign Ejection Mechanism													10
Reduce Amount of Regrind		5											
Reduce Back Pressure		2							7				
Reduce Cylinder Temperature	2	1	8	3	4				3				7
Reduce Holding Pressure			7		3								8
Reduce Injection Pressure			3		2								
Reduce Injection Speed	3		1	1	1	9	7		8	3		3	
Reduce Mold Temperature										9	3	9	4
Reduce Molded Stress		8											5
Reduce Overall Cycle Time					6								
Reduce Screw Speed	4	3							5				

World Headquarters:
580 East Front St.• PO Box 5439
Winona, MN 55987-0439
(507) 454-6900 • (800) 433-4787
FAX (507) 454-2089
Web site: www.rtpcompany.com
Email: rtp@rtpcompany.com

Manufacturing Locations:
Winona, MN
South Boston, VA
Dayton, NV
Beaune, France
Fort Worth, TX • Indianapolis, IN

ISO 9002 COMPLIANT FACILITIES

Specialty Compounds

Processing Conditions

RTP SERIES	POLYMER TYPE		TEMPERATURE °C		DRYING		DEW POINT °C	MOISTURE CONTENT %
			MELT	MOLD	TEMP (°C)	TIME (HRS)		
100	POLYPROPYLENE	(PP)	191-232	32-66	79	2	NA	NA
200	NYLON 6/6†	(PA)	277-299	66-107	79	4	-18	0.20
200A	NYLON 6†	(PA)	243-279	54-93	82	2	-18	0.20
200B	NYLON 6/10	(PA)	277-299	66-107	79	2	-18	0.20
200C	NYLON 11	(PA)	224-288	38-66	79	4	-18	0.20
200D	NYLON 6/12†	(PA)	249-285	60-93	79	4	-18	0.20
200E	NYLON, AMORPHOUS	(PA)	271-299	66-99	79	4	-34	0.10
200F	NYLON 12†	(PA)	221-274	66-104	79	4	-40	0.10
200H	NYLON 6/6 IMPACT MODIFIED	(PA)	277-299	66-107	79	4	-18	0.20
300	POLYCARBONATE†	(PC)	288-316	82-121	121	4	-29	0.02
400	POLYSTYRENE†	(PS)	210-249	38-66	82	2	NA	NA
500	STYRENE ACRYLONITRILE†	(SAN)	238-279	52-82	82	2	NA	NA
600	ACRYLONITRILE BUTADIENE STYRENE†	(ABS)	204-238	63-85	82	2	-18	0.10
700	HIGH DENSITY POLYETHYLENE	(HDPE)	193-232	21-66	79	2	NA	NA
700A	LOW DENSITY POLYETHYLENE	(LDPE)	193-232	21-66	79	2	NA	NA
800	ACETAL	(POM)	182-218	79-107	121	2	-32	0.15
900	POLYSULFONE	(PSU)	332-371	93-149	135	4	-32	0.15
1000	POLYBUTYLENE TEREPHTHALATE†	(PBT)	238-271	79-107	121	4	-29	0.03
1100	POLYETHYLENE TEREPHTHALATE†	(PET)	260-298	135-162	121	4	-40	0.01
1200	POLYURETHANE THERMOPLASTIC ELASTOMER†	(TPUR)	185-218	38-60	107	6	-18	0.01
1300	POLYPHENYLENE SULFIDE	(PPS)	307-329	135-177	149	6	NA	0.04
1400	POLYETHERSULFONE†	(PES)	343-377	135-177	149	6	-32	0.04
1500	POLYESTER THERMOPLASTIC ELASTOMER†	(TPE)	210-238	21-49	93	2-4	NA	NA
1700	POLYPHENYLENE OXIDE, MODIFIED	(PPO)	249-288	66-93	93	2	-18	0.10
1800	ACRYLIC	(PMMA)	182-218	79-107	93	4	-18	0.02
1800A	POLYCARBONATE/ACRYLIC ALLOY	(PC/PMMA)	238-266	32-66	82	3-4	-18	0.02
2100	POLYETHERIMIDE†	(PEI)	354-399	135-177	149	4	-29	0.04
2200	POLYETHERETHERKETONE†	(PEEK)	349-399	163-218	149	3	-29	0.10
2300A	RIGID THERMOPLASTIC POLYURETHANE†	(RTPU)	221-243	52-93	107	4-6	-32	0.01
2300C	RIGID THERMOPLASTIC POLYURETHANE†	(RTPU)	238-260	93-121	132	4-6	-32	0.01
2500	POLYCARBONATE/ABS ALLOY†	(PC/ABS)	243-274	52-93	93	4	-29	0.02
2700	STYRENIC THERMOPLASTIC ELASTOMER	(TES)	182-232	16-38	79	2	NA	NA
2800	OLEFINIC THERMOPLASTIC ELASTOMER†	(TEO)	182-210	16-66	79	2	-18	0.03
3000	POLYMETHYLPENTENE	(PMP)	266-304	66-93	79	2	NA	NA
3100	PERFLUOROALKOXY	(PFA)	343-385	149-232	121	2	NA	NA
3200	ETHYLENE TETRAFLUOROETHYLENE	(ETFE)	293-343	66-149	121	2	NA	NA
3300	POLYVINYLIDENE FLUORIDE	(PVDF)	210-288	82-104	121	2	NA	NA
3400-3	LIQUID CRYSTAL POLYMER	(LCP)	332-366	66-121	149	8	-29	NA
3400-4	LIQUID CRYSTAL POLYMER	(LCP)	363-399	66-93	149	8	-29	NA
3500	FLUORINATED ETHYLENE-PROPYLENE	(FEP)	343-385	93+	121	2-4	NA	NA
4000	POLYPHTHALAMIDE†	(PPA)	302-329	135-163	79	6	-32	0.05
4000A	POLYPHTHALAMIDE HOT WATER MOLDABLE	(PPA)	329-343	66-163	79	6	-29	0.10
4200	THERMOPLASTIC POLYIMIDE†	(TPI)	399-416	177-232	204	6	-40	0.01
4300	POLYSULFONE/POLYCARBONATE ALLOY	(PSU/PC)	282-327	66-99	121	4	-29	0.02
4400	NYLON, HIGH TEMPERATURE	(NHT)	310-343	135-163	79	4	-40	0.10
4500	ALIPHATIC POLYKETONE	(PK)	221-260	82-149	60	4	-32	0.02
4600	SYNDIOTACTIC POLYSTYRENE	(SPS)	293-327	71-149	82	2	-29	0.02
4700	POLYTRIMETHYLENE TEREPHTHALATE	(PTT)	232-260	87-120	127	4-6	-40	0.01

†Hygroscopic Materials

Technical Support: 800-433-4787 (International callers +1-507-454-6900)

Web site: www.rtpcompany.com • Email: rtp@rtpcompany.com

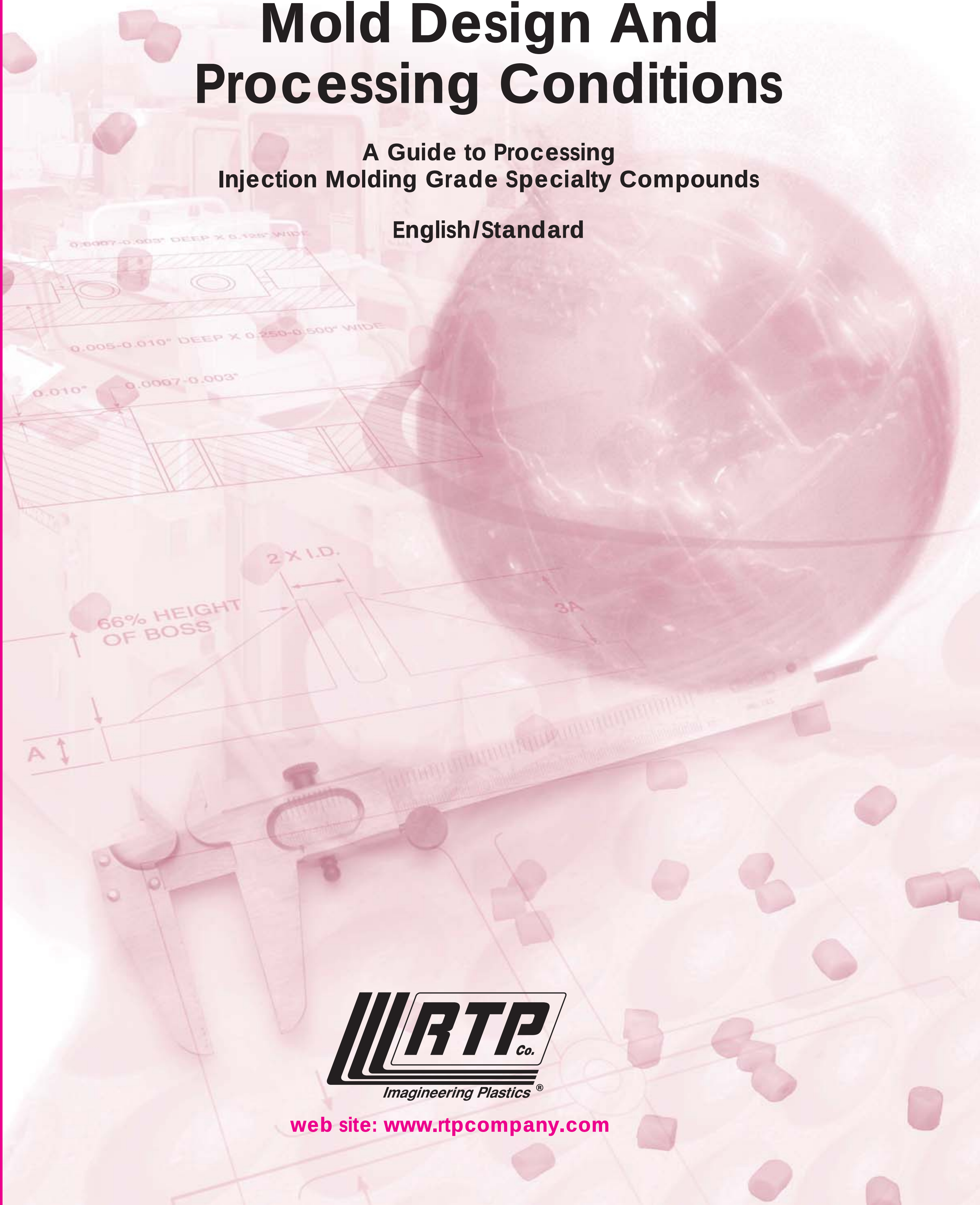
World Headquarters: Winona, MN

CONDENSED

Mold Design And Processing Conditions

A Guide to Processing Injection Molding Grade Specialty Compounds

English/Standard



Imagineering Plastics®

web site: www.rtpcompany.com

Operating Conditions

Standard Molding Equipment

Machine Type: We recommend using a screw injection machine for molding reinforced thermoplastics. A screw injection machine improves melt homogeneity, reduces variations in the molded parts, and minimizes degradations and cold spots of the polymer melt.

Machine Size: The machine should be sized to use approximately 50-70% of machine barrel capacity per shot. This maintains a short barrel residence time and eliminates material degradation. The machine should have adequate clamp pressure to obtain 4-8 tons per square inch of projected surface area. Generally, a reinforced material requires 50-70% higher clamping pressures than a nonreinforced polymer of the same type.

Molding Conditions

Drying: Successful molding of reinforced thermoplastics requires adequate drying. Inadequate drying can cause extremely erratic molding conditions and less than perfect molded parts. Excessively wet materials outgas and can undergo a viscosity change during processing. This may cause brittleness, blisters, voids, silver streaking and poor surface finish. RTP Company materials are dried prior to packaging in moisture resistant containers. However, we recommend thoroughly drying the materials in a dehumidifying type dryer. This is important with hygroscopic materials but can also be essential for non-hygroscopic materials. Condensed surface moisture can dramatically affect high temperature molded parts. The recommended drying times are provided as guidelines; however, an actual moisture check is necessary.

Barrel Temperature: Typically the rear zone/zones are set 10-20° F cooler than the front zone and nozzle. Some modifications may be needed depending on part size and configuration.

Melt Temperature: Use the Processing Conditions Chart in this guide for recommended starting temperatures.

Mold Temperature: Use the Processing Conditions Chart in this guide for recommended starting temperatures. Normally, reinforced materials require higher mold temperatures than nonreinforced materials. Higher mold temperatures will achieve a smoother, more blemish-free surface by providing a resin rich skin on reinforced materials.

Injection Pressure: Injection pressure should be set low initially and increased to the point of filling the part just short of causing flash. Maximum pressure without flash generates optimum physical properties for your RTP Company material.

Back Pressure: Low back pressure (approx. 50 psi) minimizes fiber breakage and property deterioration.

Injection Speed: Generally, the fastest possible cavity fill time is best. This minimizes glass orientation and maximizes weld line integrity.

Screw RPM: The lowest possible rpm is recommended to minimize fiber breakage and screw recovery should be set accordingly. Slower rpm's result in a more uniform melt by minimizing shear heat buildup.

Tips To Improve Surface Conductivity

Compounds With Conductive Fillers

The parts may look beautiful but not be functional in static dissipative applications. The following processing conditions are important to successfully molding good, conductive parts.

1 All resins containing carbon black should be dried.

2. Slower fill rates improve surface conductivity. Fill rates less than 1 inch per second generally work best. Decreasing the fill speed has little effect when using subgates.

3. Higher melt temperatures and slower fill speeds tend to layer the carbon black particles on the part surface. A more conductive carbon black part typically has a more mottled, dull surface. A glossier surface is more resin rich and usually less conductive.

4. Decreased packing usually increases surface conductivity. Short shots can have much higher levels (one or two magnitudes) of surface conductivity, especially if measured at the edge of the flow front where the carbon particles lay closest to the surface.

5. Typical molding is usually less conductive near the gate and more conductive away from the gate, (i.e. 1K - 2K ohms with carbon black).

6. Mold temperature does not have a significant effect on conductivity.

7. Surface conductivity is usually independent of back pressure and screw speed.

Compounds With Conductive Fibers

Conductive materials based on carbon fiber typically require opposite molding conditions from conductive fillers.

1. Use the base resin drying requirements for carbon fiber reinforced compounds.

2. Increasing the fill speed usually increases surface conductivity 10^5 to $>10^4$ ohms/sq. (Caution: Excess fill speeds can break up carbon fibers and reduce mechanical properties.) All corners in the flow path should have generous radii.

3. With carbon fiber, high melt temperatures decrease surface resistivity (thereby increasing conductivity) because they increase the degree of packing. Short shots in carbon fiber are typically non-conductive.

4. A typical carbon fiber molding is more conductive near the gate.

5. Mold temperature does not have a significant effect on conductivity.

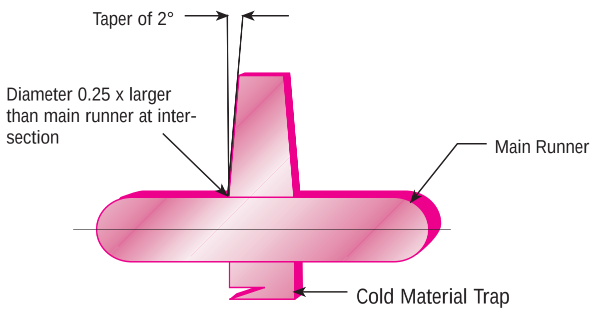
6. Back pressure and screw speed should be kept as low as possible, i.e. 25-50 psi and 20-40 rpm respectively.

Mold Design

This guide is a condensed version. For more comprehensive information refer to RTP Company's Comprehensive Mold Design and Processing Conditions or contact our marketing department at 507-454-6900.

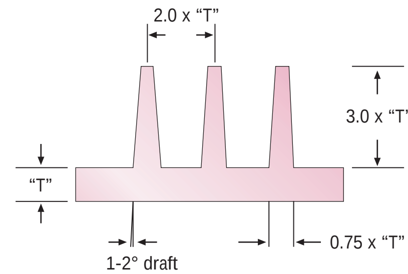
Sprue/Cold Material Trap Design

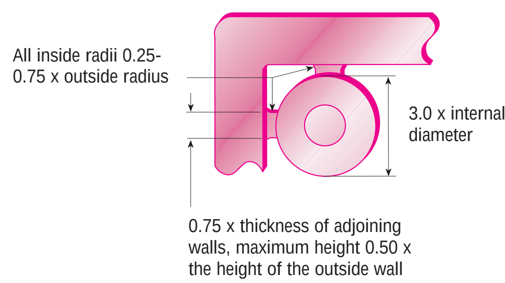
Sprues connect the nozzle of the injection molding machine to the main runner or cavity. The sprue should be as short as possible to minimize material usage and cycle time. The bushing should have a smooth, tapered internal finish that has been polished in the direction of the draw to ensure clean separation of the sprue and the bushing.



Ribs

Ribs should follow the proportional thickness guidelines shown below. If the rib is too thick in relation to the part wall, you may experience sinks, voids, warpage, weld lines, and longer cycle times. Position ribs in the line of flow to improve filling and prevent air entrapment.

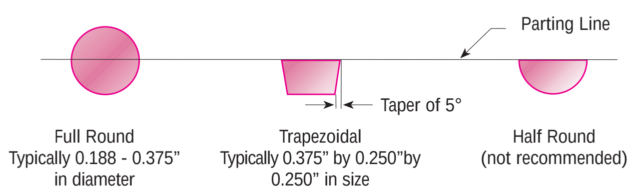




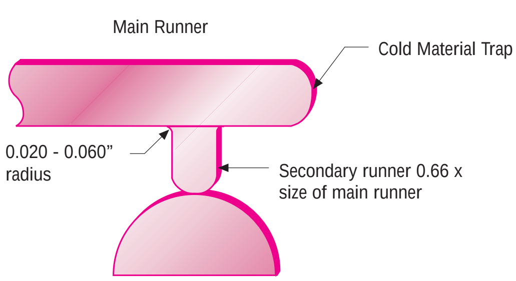
Runner Design

Runner systems convey the melted plastic from the sprue to the gate or part. The most efficient profile for a runner is circular (full-round). A less expensive, yet adequate, profile is a trapezoid, with tapers as shown in the diagram to ensure a good volume-to-surface area ratio. Half rounds are not recommended because of their poor perimeter to area ratio.

Main Runner - A channel that connects the sprue to the gate or part.



Secondary runner - A runner system located between the main runner and the molded part.

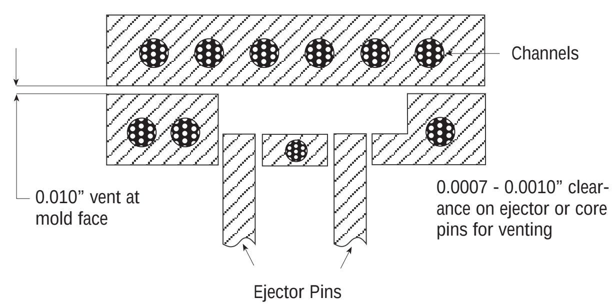


Plastic Resin (material)

Tool Steel

Cooling

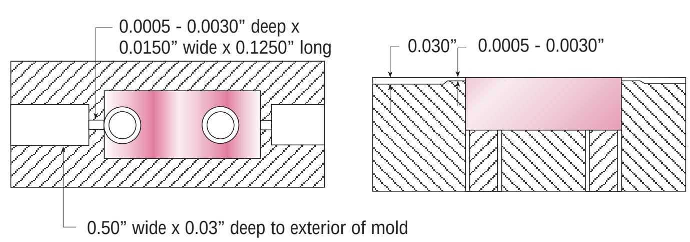
Molds must be provided with adequate cooling to take advantage of the faster cooling rates of reinforced compounds. Poor cooling results in rising mold temperatures and longer cycle times. Inadequate heating can result in voids, shorts and poor surface finish. Cooling and heating channels should be located directly in the mold inserts and cores if mold design permits.



Ejector Pins: Should be located on the heaviest sections of the part to minimize distortion when it leaves the core They should be balanced as much as possible over the part's surface. Reinforced thermoplastic require more pins due to lower mold shrinkage and greater potential for drag during ejection.

Venting

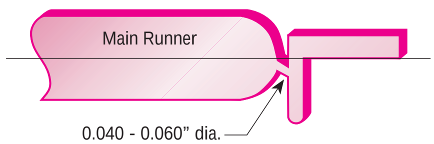
Proper venting of cavities is very important. Inadequate venting can result in gas burns, poor weld line strength and nonfilled parts. Too much venting can result in excessive flash and poor weld lines due to inadequate pressure buildup. Venting should primarily be located at the last point of fill and where weld line occur. Vent size depends on the viscosity of the polymer and can vary from 0.0005" to 0.0030" deep. Venting can also be used around knockout pins, moving cores and mold inserts.



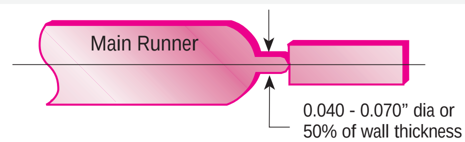
Gate Design

The gate serves as the entrance to the cavity and should be designed to permit the mold to fill easily. A cavity can have more than one gate. Gates should be small enough to ensure easy separation of the runner and the part but large enough to prevent early freeze-off of polymer flow, which can adversely affect the consistency of part dimensions. A variety of gate designs and locations are shown below:

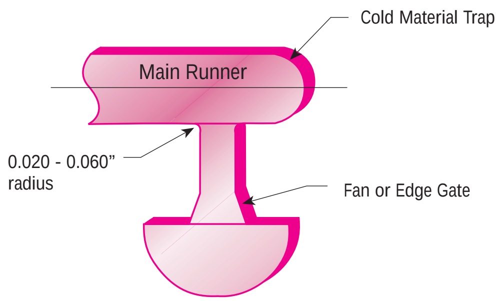
Submarine or Tunnel Gate - An edge gate located below the parting line or molded surface.



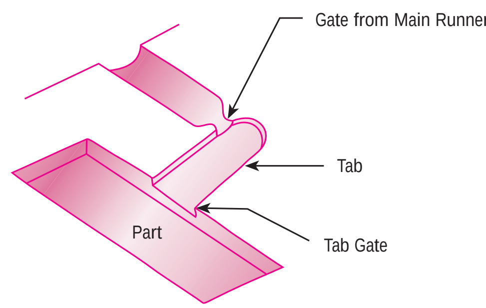
Pinpoint or Restricted Gate - A restricted opening between the runner and molded part. Normally used with thin wall parts.



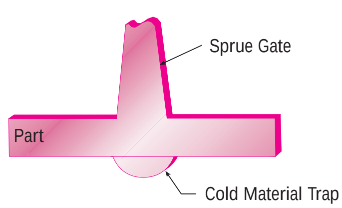
Fan or Edge Gate - A common gate located in the sidewall of the part to prevent restriction of resin flow. Normally used with multi-cavity, two-plate molds.



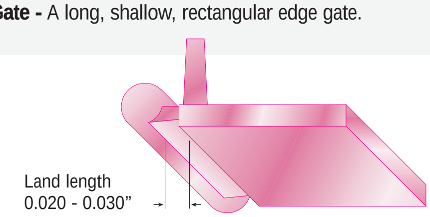
Tab Gate - Used for melt orientation when a large volume is needed for mold fill. The tab helps avoid surface splotches due to high shear, direct gating, or jetting.



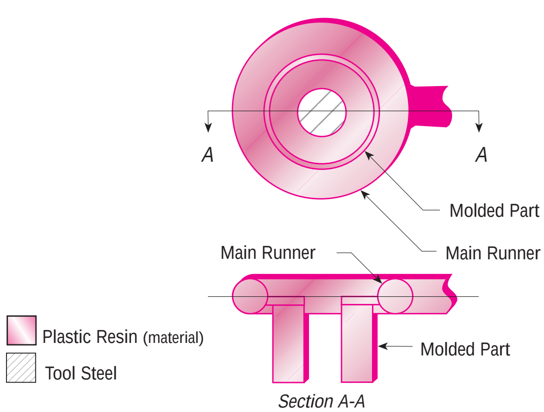
Sprue Gate - Recommended for single cavity molds requiring symmetrical filling. Usually used with circular parts.



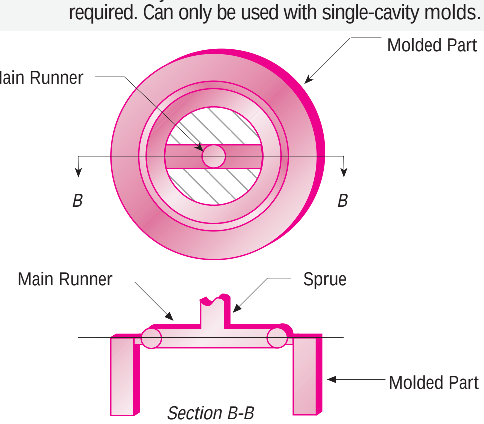
Flash Gate - A long, shallow, rectangular edge gate.



External Ring Gate - A system used when concentricity and a smooth interior surface are important. Can be used in multi-cavity molds.



Internal Ring Gate - A system used with large circular parts when concentricity and smooth outer surface are required. Can only be used with single-cavity molds.



Troubleshooting Guide

PROBLEM

SUGGESTED REMEDIES
Perform in numerical order by column

Suggested Remedies	Blisters	Brittleness	Excessive Flash	Gas Burns	Oversized Part	Poor Surface Finish	Poor Weld Lines	Short Shots	Silver Streaking	Sink Marks	Undersized Part	Voids	Warping
Change Gate Location							8						6
Clean Mold Faces			4	5		6							
Clean Vents	5			2			5	11				12	
Check for Material Contamination	6	4							4				
Check for Uneven Mold Temperature													1
Check Mold Faces for Proper Fit			5										
Dry Material	1	6	6	6		7			1			11	
Increase Amount of Material							4	1		8	7	10	
Increase Back Pressure						5		6			6		
Increase Clamp Pressure			2										
Increase Cooling Time										10			9
Increase Holding Pressure						8		12		11	1		
Increase Injection Hold Time							2			2	2	2	
Increase Injection Pressure						2	1	2		1		1	2
Increase Injection Speed						3	9	3	2		8		
Increase Injection Time										12	5		
Increase Mold Temperature		7			5	1	3	7			9	8	3
Increase Size of Gates								8	6	4	10	4	
Increase Size of Runners								9		5	11	5	
Increase Size of Sprue								10		6		6	
Increase Size of Vent				4			6	4					
Locate Gates Near Heavy Cross Sections										7		7	
Raise Material Temperature					7	4	10	5			4		
Redesign Ejection Mechanism													10
Reduce Amount of Regrind		5											
Reduce Back Pressure		2							7				
Reduce Cylinder Temperature	2	1	8	3	4				3				7
Reduce Holding Pressure			7		3								8
Reduce Injection Pressure			3		2								
Reduce Injection Speed	3		1	1	1	9	7		8	3		3	
Reduce Mold Temperature										9	3	9	4
Reduce Molded Stress		8											5
Reduce Overall Cycle Time					6								
Reduce Screw Speed	4	3							5				

World Headquarters:
580 East Front St. • PO Box 5439
Winona, MN 55987-0439
(507) 454-6900 • (800) 433-4787
FAX (507) 454-2089
Web site: www.rtpcompany.com
E-mail: rtp@rtpcompany.com

Manufacturing Locations:
Winona, MN
South Boston, VA
Dayton, NV
Beaune, France
Fort Worth, TX • Indianapolis, IN

ISO 9002 COMPLIANT FACILITIES

Specialty Compounds

Processing Conditions

RTP SERIES	POLYMER TYPE		TEMPERATURE ⁰F		DRYING		DEW POINT ⁰F	MOISTURE CONTENT %
			MELT	MOLD	TEMP (⁰F)	TIME (HRS)		
100	POLYPROPYLENE	(PP)	375-450	90-150	175	2	NA	NA
200	NYLON 6/6†	(PA)	530-570	150-225	175	4	0	0.20
200A	NYLON 6†	(PA)	470-535	130-200	180	2	0	0.20
200B	NYLON 6/10	(PA)	530-570	150-225	175	2	0	0.20
200C	NYLON 11	(PA)	435-550	100-150	175	4	0	0.20
200D	NYLON 6/12†	(PA)	480-545	140-200	175	4	0	0.20
200E	NYLON, AMORPHOUS	(PA)	520-570	150-210	175	4	-30	0.10
200F	NYLON 12†	(PA)	430-525	150-220	175	4	-40	0.10
200H	NYLON 6/6 IMPACT MODIFIED	(PA)	530-570	150-225	175	4	0	0.20
300	POLYCARBONATE†	(PC)	550-600	180-250	250	4	-20	0.02
400	POLYSTYRENE†	(PS)	410-480	100-150	180	2	NA	NA
500	STYRENE ACRYLONITRILE†	(SAN)	460-535	125-180	180	2	NA	NA
600	ACRYLONITRILE BUTADIENE STYRENE†	(ABS)	400-460	145-185	180	2	0	0.10
700	HIGH DENSITY POLYETHYLENE	(HDPE)	380-450	70-150	175	2	NA	NA
700A	LOW DENSITY POLYETHYLENE	(LDPE)	380-450	70-150	175	2	NA	NA
800	ACETAL	(POM)	360-425	175-225	250	2	-25	0.15
900	POLYSULFONE	(PSU)	630-700	200-300	275	4	-25	0.15
1000	POLYBUTYLENE TEREPHTHALATE†	(PBT)	460-520	175-225	250	4	-20	0.03
1100	POLYETHYLENE TEREPHTHALATE†	(PET)	500-570	275-325	250	4	-40	0.01
1200	POLYURETHANE THERMOPLASTIC ELASTOMER†	(TPUR)	365-425	100-140	225	6	0	0.01
1300	POLYPHENYLENE SULFIDE	(PPS)	585-625	275-350	300	6	NA	0.04
1400	POLYETHERSULFONE†	(PES)	650-710	275-350	300	6	-25	0.04
1500	POLYESTER THERMOPLASTIC ELASTOMER†	(TPE)	410-460	70-120	200	2-4	NA	NA
1700	POLYPHENYLENE OXIDE, MODIFIED	(PPO)	480-550	150-200	200	2	0	0.10
1800	ACRYLIC	(PMMA)	360-425	175-225	200	4	0	0.02
1800A	POLYCARBONATE/ACRYLIC ALLOY	(PC/PMMA)	460-510	90-150	180	3-4	0	0.02
2100	POLYETHERIMIDE†	(PEI)	670-750	275-350	300	4	-20	0.04
2200	POLYETHERETHERKETONE†	(PEEK)	660-750	325-425	300	3	-20	0.10
2300A	RIGID THERMOPLASTIC POLYURETHANE†	(RTPU)	430-470	125-200	225	4-6	-25	0.01
2300C	RIGID THERMOPLASTIC POLYURETHANE†	(RTPU)	460-500	200-250	270	4-6	-25	0.01
2500	POLYCARBONATE/ABS ALLOY†	(PC/ABS)	470-525	125-200	200	4	-20	0.02
2700	STYRENIC THERMOPLASTIC ELASTOMER	(TES)	360-450	60-100	175	2	NA	NA
2800	OLEFINIC THERMOPLASTIC ELASTOMER†	(TEO)	360-410	60-150	175	2	0	0.03
3000	POLYMETHYLPENTENE	(PMP)	510-580	150-200	175	2	NA	NA
3100	PERFLUOROALKOXY	(PFA)	650-725	300-450	250	2	NA	NA
3200	ETHYLENE TETRAFLUOROETHYLENE	(ETFE)	560-650	150-300	250	2	NA	NA
3300	POLYVINYLIDENE FLUORIDE	(PVDF)	410-550	180-220	250	2	NA	NA
3400-3	LIQUID CRYSTAL POLYMER	(LCP)	630-690	150-250	300	8	-20	NA
3400-4	LIQUID CRYSTAL POLYMER	(LCP)	685-750	150-200	300	8	-20	NA
3500	FLUORINATED ETHYLENE PROPYLENE	(FEP)	650-725	200+	250	2-4	NA	NA
4000	POLYPHTHALAMIDE†	(PPA)	575-625	275-325	175	6	-25	0.05
4000A	POLYPHTHALAMIDE HOT WATER MOLDABLE	(PPA)	625-650	150-325	175	6	-20	0.10
4200	THERMOPLASTIC POLYIMIDE†	(TPI)	750-780	350-450	400	6	-40	0.01
4300	POLYSULFONE/POLYCARBONATE ALLOY	(PSU/PC)	540-620	150-210	250	4	-20	0.02
4400	NYLON, HIGH TEMPERATURE	(NHT)	590-650	275-325	175	4	-40	0.10
4500	ALIPHATIC POLYKETONE	(PK)	430-500	180-300	140	4	-25	0.02
4600	SYNDIOTACTIC POLYSTYRENE	(SPS)	560-620	160-300	180	2	-20	0.02
4700	POLYTRIMETHYLENE TEREPHTHALATE	(PTT)	450-500	190-250	260	4-6	-40	0.01

†Hygroscopic Materials

This information is intended to be used only as a guideline for designers and processors of modified thermoplastics for injection molding. Because injection mold design and processing is complex, a set solution will not solve all problems. Observation on a "trial and error" basis may be required to achieve desired results. No information supplied by RTP Company constitutes a warranty regarding product performance or use. Any information regarding performance or use is only offered as suggestion for investigation for use, based upon RTP Company or other customer experience. RTP Company makes no warranties, expressed or implied, concerning the suitability or fitness of any of its products for any particular purpose. It is the responsibility of the customer to determine that the product is safe, lawful and technically suitable for the intended use. The disclosure of information herein is not a license to operate under, or a recommendation to infringe any patents.

Technical Support: 800-433-4787 (Local callers 507-454-6900)
Web site: www.rtpcompany.com • E-mail: rtp@rtpcompany.com
World Headquarters: Winona, MN

2/2000