Morgan Injection Molder

G-100T

Machine Purpose: Inject melted thermoplastic into die

Safety: **Must wear safety glasses while operating machine. Keep.** Beware of objects that dangle and could get caught in moving parts. Beware of hot plastic, moving parts, and crushing forces. **This machine has no built-in safety sensors.** Always operate machine with safety grating closed.

Materials: Thermoplastics

Machine Specs: 6 cu in/4oz, 20 Ton Clamping Pressure, 12,000 psi injection pressure, 8”x11”x7.5” mold area, 0-800 F (430 C)

Accessories: Heated bed, injection speed control, antidrool nozzle, digital model of machine available

Design Limitations: 6 in^2 max shot (total volume), 4-20 in^2 parting line cross section dependent of material choice.
High Flow: 115 in^2 of cross section
Medium Flow: 8 in^2 of cross section
Low Flow: 2.5 in^2 of cross section

Instruction Required: Individual checkout scheduled via help session, group classes to come

Determining Clamping Force/Part Viability for this machine

Always include a clamping safety factor of 10% clamping force, and 10% extra for every inch over 1 inch of depth.

Cross section x tons per inch (injection pressure) = clamp force

Parting line cross section of 2x2 (4 square inches) with styrene requiring a 1-2 ton per inch clamping force would require 4-8 tons of clamping force + 10% safety: 4.4-8.8 tons of clamping force. While the same part in polycarbonate would require 5 tons per square inch would need 20 tons of clamping force, the top range of the machine, and thus would no be recommended for this machine as it would outside the clamping pressure with a safety factor added.

For a quick calculation visit http://www.custompartnet.com/calculator/clamp-force

Material	Ton per in^2 low (injection Pressure)	Ton per in^2 high (Injection Pressure)
ABS	2.70	4.41
HIPS	1.07	2.20
CAB	1.07	2.20
HDPE	1.64	2.77
LDPE	1.07	2.20
NYLON 6	4.41	5.47
NYLON 6 (30% GF)	4.98	6.04
NYLON 6/6	4.41	5.47
NYLON 11	1.64	2.20
NYLON 12	1.64	2.20
PEEK	2.20	4.41
POLYCARBONATE	3.27	5.47
POLYESTER PBT	3.27	4.98
PPS	2.13	3.20
PET	2.77	5.33
PMMA ACRYLIC	2.20	4.41
POM	3.27	5.47
POLYPROPYLENE - H	1.64	2.77
POLYSTYRENE	1.07	2.20
PVC	2.20	3.27
SAN	2.77	3.27

Go-No/Go Injection Mold

If you answer no to any of these questions below then the object is not injection moldable on this machine.

Is the object cross section smaller then 6”x9”?

Is the volume of the object greater then 6 in^2?

Is the area cross section more then 18 square inches?

If you multiple the area by injection pressure needed for material selection listed below does it exceed 18 tons? (20 tons with safety factor of 10%)

Mold Considerations

- Molds should have 1 inch of metal around all sides away from cavity
    - If you object is a 2x2x1 inch object your mold will need to be a minimum of 4x4x3 inches
- All faces should have a taper
- Rounded Corners will flow better then sharp corners
- Injections will typically stick to the male side of a mold
- At least .002" airway channels are need to allow air to escape while injecting (material dependent)
- Air escape channels should be design from the furthest distance from the injection point as possible
- Its always a good practice to have replaceable components to a mold. As parts wear out it will be easier/cheaper to repair/replace part of a mold rather then the whole thing.

Operations:

1. Turn on control box (switch on side) & select temperature control settings for material to be used.

Figure 1Control Box & Temp Control

2. Lift Orange barrel safety gate and fill barrel with granules of material of choice. (polypropylene is a good test material)

‘

Figure 2 Barrel Guard

Initial Set Up

a. Close barrel safety gate (machine won’t extrude with gate open)
b. Close mold safety gate
c. Once barrel and nozzle are up to temperature press inject to purge barrel several times to remove any residual material from last operation.
d. Reload and rerun as necessary until material comes out clean.

Figure 3 Table/Mold Guard

3. Chose the timer valve by rotating the knob clockwise until the indicator stops at 0, do not close too tightly as damage may occur to the needle and seat.
4. Select clamp force and injection pressure.
    a. Setting pneumatics pressure (based on chart for material choice)
    b. Setting clamp forces
        i. Set clamp forces by rotating the orange knurled knob.

Figure 4 Timer/Speed/Pressure Controls

Figure 6 Clamping Force Gauge

Figure 5.1 Pilot Force Gauge

    c. Setting Injection Speed
        i. Adjust pilot valve pressure to a 6 (green zone, good starting point, Figure 5.1)
        ii. Set injection pressure on separate regulator installed (Figure 7)
        iii. Then set the speed at the flow control valve at the top of rear of machine. (Figure 8)

Note: The injection pressure and speed required for any application depends on many factors such as wall thickness, flow distance, gate size, and plastic being used. Its best to start low and slow and increase speed then injection pressure until a good part is produced. Most applications work between 3000-6000psi. Pressure above 6k may be required if a part is large or has thin wall sections. Also, some thermoplastics require high injection pressure due to their composition.

Figure 7 Injection Speed Control

Figure 8 Injection Speed Control

5. Place the mold on the table of the Morgan-Press, taking care to locate the mold so that the nozzle seat is beneath the nozzle. Use the table center location stud supplied whenever possible. (If preheat plate is to be used, locate this on the table center locator stud and place mold on the pre-heat plate locator stud, yellow circle Figure 9)

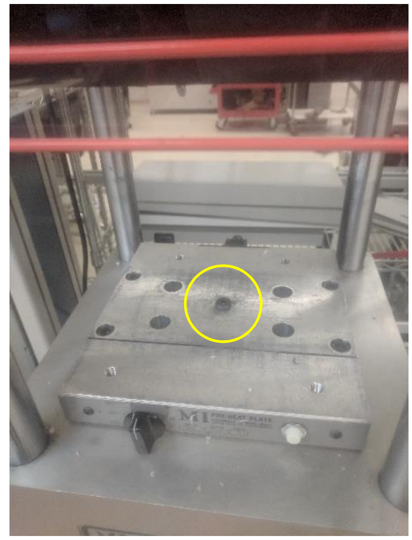


Figure 9 Centering Stud on Heating Plate

Adjusting table height

1. On the Morgan-Press units with toggle clamping mechanisms, it is a combination of air pressure plus mechanical adjustment of the table height which yields the ultimate clamp force. This adjustment is similar to tightening or loosening a pair of vice grip pliers where the screw adjustment is the table height and squeeze on the handle is the air cylinder

    Note: **<u>if a clamp force of greater than 10 tons is needed the upper platen assembly must be properly used!!!</u>** Applying a clamp force of greater than 10 tons directly against the nozzle may result in damage to the Morgan-Press. **<u>Clamping against the nozzle above 10 tons constitutes misuse of the equipment and affects warranty.</u>**

2. Set clamp force to minimum that will actuate the table (2-3 tons). Raise the table under power to full up stroke position by pushing the Clamp Control knob (orange knob Figure 4) with the table guard down (Figure 3). If mold does not reach the nozzle, the table must be adjusted upward by rotating the Elevating Spindle (Figure 10) with the 3/8 allen key. The elevating spindle should pushed in to engage the gears during the height adjustment the pulled out before clamping. Turn the spindle clockwise to lower and CCW to raise table. For short molds, (5” or less) a simple spacer made of parallel aluminum plates can be used.

    If the mold touches the nozzle but does not allow the full toggle stroke, the table must be adjusted downward using the Elevating spindle and Allen key. The maximum mold or stack height is 7.5” inches. DO NOT ATTEMPT TO ADJUST TABLE HEIGHT WHILE THE MOLD IS IN THE CLAMPED POSITION. Rather pull out the Clamp Control Valve to lower the table with the toggle power, then rotate the spindle. Continue up or down adjustment until mold gently touch the nozzle when toggle is in full up position.

Figure 10 Height Adjustment

3. To attain full clamp force on the mold, adjust clamp pressure gauge to desired setting. The Table must then be mechanically adjusted slightly upward in the same manner as above until the toggle lock comes to a full position while moving over center with a definite “thunk” sound. This is the adjustment which is similar to increasing the tension on a pair of vice grips. This position can be attained after two or three trials. It may be necessary to re-adjust clamp after several operating cycles due to the heat expansion of the mold and equipment plus any nozzle retightening that may be required.
    a. See proper nozzle tightening procedure
4. When done with clamp adjustment, disengage the table elevating gear by pulling the spindle out.

Note: Clamp adjustment should be periodically monitored during operation. It may be necessary to re-adjust the table platen height due to heat expansion and mechanical vibration. The proper clamp force is obtained only when the toggle mechanism positively locks over center with a definite “thunk”.

Upper Platen Assembly Use and Setup (done after adjusting height section if clamping above 10 tons)

1. Position platen box assembly so that the bottom of the lower plate is well above the nozzle
    a. Tighten bolts until seated on all for corners (green circle Figure 11)
    b. Loosen Allen bolts on both sides until steel plate can slide (blue circles Figure 11)
    c. Push Plate until seated upward, lock holding bolts (blue Circles Figure 11), nozzle should now be protruding beneath platen face)

Figure 11 Upper Platen Assembly

2. Raise mold into position (with less then 10 tons of clamping pressure!) using orange clamping button.
3. Lower the Upper Platen until its resting on the face of the mold (loosen the allen bolts locking it in plate, green circles figure 11), unscrew the adjusting bolts until they touch the top of the Upper Platen (green circles figure 11).
4. Retighten suspension screws so the platen box will still move by twisting the adjustment bolts, but don’t drop under its own weight.

5. Unclamp mold
6. Lower platen box by unscrewing the 4 height adjustment bolts (green circles Figure 11) ½ turn. This is an initial adjustment to insure that the major clamping forc4 is against the platen not the nozzle.
7. Set clamp force for actual desired tonnage for molding.
8. By turning all 4 height adjustment bolts raise or lower the Upper Platen box until proper tonnage action (clunk/vice grip lock action) is achieved. (Usually ¼-1/2 additional turn to each bolt is required.
9. Tighten Suspension Screws Snuggly when appropriate height achieved.

Running First Part

1. Ensure Barrel Full of granules, Nozzle and Barrel to full operating temperature and clamping alignment/upper platen (if necessary) are set up properly.
2. Close all safety guards (hopper/granule door & Mold/platform cage)
3. Raise the table to clamped position
4. Set clamping pressure is set desired tonnage
5. Set Injection pressure is set at desired tonnage

    Note: You will most likely need to adjust clamping pressure, injection pressure, injection speed to suit your mold.

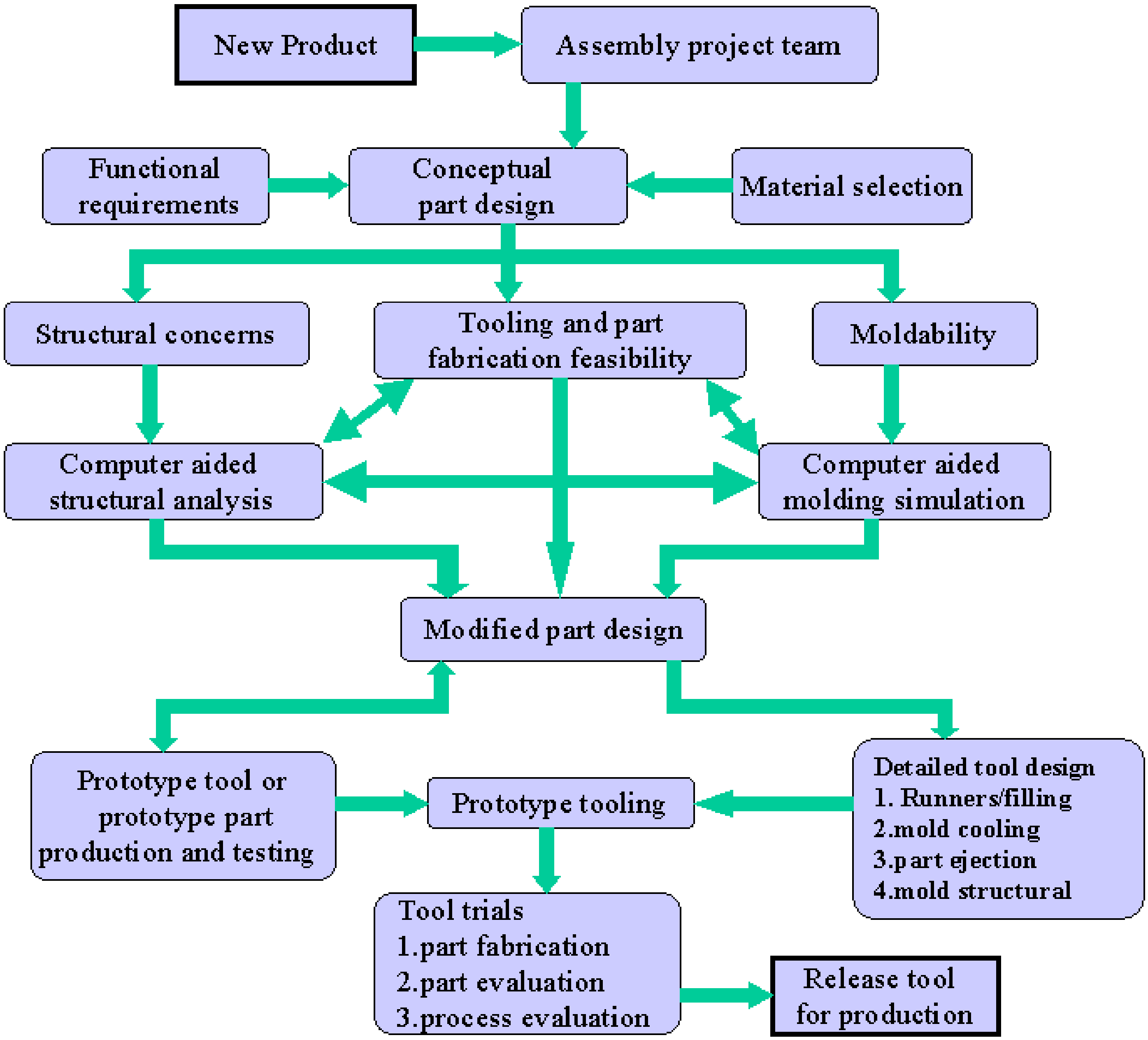
6. Set Timer to zero for first several runs
7. Once ready press and hold the Push to Inject button to inject plastic into cavity.
8. Once estimated mold release time has passed release injection button
9. Release push to clamp button, if necessary push black button on right side of machine to lower table
10. Pull mold apart evaluate and adjust settings as necessary. Run again.

General design guide for injection molded part

1. Design Process [after Robert A. Malloy, Plastic Part Design for Injection Molding, Hanser Gardner, 1994]

There are a number if different approaches that can be taken when developing a new product. Historically, new parts or products were developed using a Sequential Engineering approach to design, which begins with a new product idea generated by marketing groups and ends up with the manufacturing stage. The problem with this approach is that it may not result in an optimum design, and will no doubt take more time and money than would be required using a more Concurrent Engineering approach.

The concept of concurrent engineering can be illustrated by the following figure. This "Parallel" or "Concurrent Engineering" approach to product design reduces development time, improves quality, and minimizes the potential for unanticipated production or performance problems.



Concurrent Design Process for Plastic Injection Molded Parts (after Malloy)

2. General Guidelines for injection molding[Douglas M. Bryce]

(a). Determine required clamp force

*clamp force=projected area * injection pressure*

The projected area of the part can be found based on the geometry of the part. The thickness is important only if it is more than 1 inch. For every inch of thickness over 1 inch, the total clamp force must be increased by 10 percent.

The injection pressure must vary with the flow ability of the material. The typical value for injection pressure is between 2 to 8 tons per square inch (or 28,000~110,000 KPa). As a rule of thumb, 4 or 5 tons/inch^2 (55,000~69,000 KPa) can be used for most products. For example, if polycarbonate has been selected, then the injection pressure could be 5 tons per square inch.

Too low a clamping force can lead to flash, or non-filled parts; too high a clamping force can lead to mold damage.

(b). Determine the injection molding cost for a specific product

For calculating actual manufacturing cost, the following information is needed:

1. Material cost.

The material cost can be determined using the following three-step formula:

(1). Determine the total volume of the part ($inch^3$).

total volume = volume of the runner system + volume of the part.

(2). Determine the weight per unit volume ($lb/inch^3$).

(3). Determine the cost per unit weight.

$cost = (cost / lb) * (lb / inch^3) * (inch^3)$

2. Machine cost.

The total machine cost is determined by the machine hourly rate (the hourly cost of machine and operator) and overall cycle time (so-called gate-to-gate cycle time).

The dominant effect in determining cycle time is the time it takes to cool the part, and cooling time depends on wall thickness. The cooling time for a plate-like part of thickness, h, can be estimated using this formula: [Tim .A. Osswald]

$$t_{cooling} = \frac{h^2}{\pi * \alpha} * ln(\frac{8}{\pi^2} * \frac{T_m - T_w}{T_d - T_w})$$

T_m-melt temperature
T_w-mold(wall) temperature
T_d-average part temperature at ejection

α -thermal diffusivity

The following table provides some information about the wall thickness and corresponding overall cycle time.

Wall thickness, in. (mm)	Overall cycle time, seconds
0.060(1.5)	18
0.075(1.9)	22
0.100(2.5)	28
0.125(3.2)	36

Wall thikness, the melt temperature, and mold wall temperature as well as the final part temperature when it is ejected all have effects on the cooling time. The melt temperature is usually available from the manufacturer. The suggested melt temperature and mold wall temperature are listed in the tables below.

Material	Melt Temperature(F)
Acetal (copolymer)	400
Acetal (homopolymer)	425
Acrylic	425
ABS (medium-impact)	400
Cellulose acetate	385
Nylon (Type 6)	500
Polyallomer	485
Polyamide-imide	650
Polyarylate	700
Polybutylene	475
Polycarbonate	550
Polyethylene (Low density)	325
Polyethylene(High density)	400
Polypropylene	350
Polystyrene (general purpose)	350
Polystyrene (medium-impact)	380
Polystyrene (high-impact)	390
PVC (rigid)	350
PVC (flexible)	325

Material	**Mold Temperature(F)**
Acetal (copolymer)	200
Acetal (homopolymer)	210
Acrylic	180
ABS (medium-impact)	180
Cellulose acetate	150
Nylon (Type 6)	200
Polyallomer	200
Polyamide-imide	400
Polyarylate	275

Polybutylene	200
Polycarbonate	220
Polyethylene (Low density)	80
Polyethylene(High density)	110
Polypropylene	120
Polystyrene (general purpose)	140
Polystyrene (medium-impact)	160
Polystyrene (high-impact)	180
PVC (rigid)	140
PVC (flexible)	80

(c). Estimate the gate-to-gate cycle time

The following table shows the typical time estimates. The actual cycle time is less than the sum of these values, because there are overlaps between some operations. As shown, the cooling time is the most important part.

Parameter	Average Value(second)
Gate closing time	1
Mold closing time	4
Initial injection time	3
Injection hold time	5
Cooling time	12
Screw return time	8
Mold open time	4
Ejection time	1
Part removal time	2
Mold inspection,clean,spray,etc.	2

(d). Determine the wall thickness

(1). Design guidelines:

- All walls should be equal thickness if possible.
- If a thicker wall is needed, a gentle transition should be specified.
- To avoid sink marks, ribs should be 2/3 the wall thickness, gussets should be 1/2 the wall thickness.
- Sharp corners should be eliminated by using radii.

(2). The wall thickness is mainly determined by the flow ability of the plastic. The ability to flow also determines how far a plastic can be injected for a specific wall thickness of product.

The approximate maximum flow-path-to-thickness ratio of some common thermoplastics are listed here:

ABS	175:1
Acetal	140:1
Acrylic	130~150:1
Nylon	150:1
Polycarbonate	100:1
Polyethylene(low density)	275~300:1
Polyethylene(high density)	225~250:1
Polypropylene	250~ 275:1
Polystyrene	200~250:1
Polyvinyl Chloride, rigid	100:1

The following table is a listing of common materials and the wall thickness they can flow through.

This table shows that an easy-flow material (such as crystalline nylon) allows thinner wall.

Material	Minimum (in./mm)	Maximum (in./mm)
ABS	0.030/0.762	0.125/3.175
Acetal	0.015/0.381	0.125/3.175
Acrylic	0.025/0.635	0.250/6.350
Nylon(amorphous)	0.030/0.762	0.125/3.175
Nylon(crystalline)	0.015/0.381	0.125/3.175
Phenolic	0.045/1.143	1.000/25.400
Polycarbonate	0.040/1.016	0.400/10.160
Polyester(TP)	0.025/0.635	0.125/3.175
Polyester(TS)	0.040/1.016	0.500/12.700
Polyethylene(HD)	0.020/0.508	0.250/6.350
Polyrthylene(LD)	0.030/0.762	0.250/6.350
Polypropylene	0.025/0.635	0.300/7.620
PPO	0.030/0.762	0.400/10.160
Polystyrene	0.030/0.762	0.250/6.350
PVC	0.040/1.016	0.400/10.160

(e). Determine the runner system

(1). General guidelines for runner and gate design:

- The runner cross section diameter also depends upon the type of plastic being molded. High-viscosity(very stiff) materials require larger-diameter runners than low-viscosity materials.
- The longer the flow path the plastic must travel along, the larger the runner diameter must be at the start.
- Right-angled turn in a runner system requires an additional 20% increase in the diameter to compensate for pressure drops.
- A part should be gated into its thickest section, from thick to thin, never the reverse.
- Cavity sets should be located as close to the sprue as possible to minimize travel time and distance.

(2). Runner diameters for some common materials

	Runner Diameter (in./mm)		
Material	Runner Length(in./mm) 3 /76.2	Runner Length(in./mm) 6/152.4	Runner Length(in./mm) 10/254
ABS	0.093/2.4	0.109/2.8	0.156/3.9
Acetal	0.062/1.6	0.093/2.4	0.125/3.1
Acrylic	0.125/3.1	0.156/3.9	0.187/4.7
Cellulose acetate	0.093/2.4	0.109/2.8	0.156/3.9
Cellulose acetate butyrate	0.093/2.4	0.109/2.8	0.125/3.1
Ionomer	0.062/1.6	0.093/2.4	0.125/3.1
Nylon 66	0.062/1.6	0.078/1.9	0.093/2.4
Polycarbonate	0.125/3.1	0.156/3.9	0.203/5.1
Polyethylene	0.062/1.6	0.093/2.4	0.125/3.1
Polypropylene	0.062/1.6	0.093/2.4	0.125/3.1

Polyphenylene oxide	0.125/3.1	0.156/3.9	0.203/5.1
Polyphenylene sulfide	0.125/3.1	0.156/3.9	0.203/5.1
Polysulfone	0.156/3.9	0.187/4.7	0.218/5.5
Polystyrene	0.093/2.4	0.109/2.8	0.125/3.1
Rigid PVC	0.125/3.1	0.187/4.7	0.250/6.3

(3).Hot runners

The purpose for hot runner system is to reduce the overall cycle times. The advantage of the hot runner system is that the runner does not have to be included in the calculation of cycle times. The cooling portion of the molding cycle only applies to the molded part and the overall cycle can be much shorter than if runners were included.

3. Parameter Change versus Property Effect [Douglas M. Bryce]

What is the best setting for the injection pressure, back pressure, melt temperature and mold temperature, etc.?
-It all depends on the material being molded and the type of mold being used, as well as the status of the injection machine and environmental conditions. Generally, the effect of parameters on the product properties would be:

Parameter	Property Effect
Injection Pressure(+)	Less shrinkage, higher gloss, less warp, harder to eject
Injection Pressure(-)	More shrinkage, less gloss, more warp, easier to eject
Back Pressure(+)	Higher density,more degradation, fewer voids
Back Pressure(-)	Lower density, less degradation, more voids
Melt Temperature(+)	Faster flow, more degradation, more brittle, flashing
Melt Temperature(-)	Slower flow, less degradation, less brittle, less flashing
Mold Temperature(+)	Longer cycle, higher gloss, less warp, less shrinkage

Mold Temperature(-) Faster cycle, lower gloss, greater warp, higher shrinkage

References

- *C-MOLD Reference Manual,* CAE, Ithaca, New York.
- Tim A. Osswald, *Polymer Processing Fundamentals*, Hanser, Munich, 1998.
- Robert A. Malloy, *Plastic Part Design for Injection Molding*, Hanser, Munich, 1994.
- Douglas M. Bryce, *Plastic Injection Molding: Manufacturing Process Fundamentals*, Society of Manufacturing Engineers, Dearborn, 1996.
- Douglas M. Bryce, *Plastic Injection Molding: Material Selection And Product Design Fundamentals*, Society of Manufacturing Engineers, Dearborn, 1997.
- Douglas M. Bryce, *Plastic Injection Molding: Mold Design and Construction Fundamentals*, Society of Manufacturing Engineers, Dearborn, 1998.
- Dominick V. Rosato, Donald V. Rosato and M. G. Rosato, *Injection Molding Handbook*, Kluwer Academic Publishers, Amsterdam, 2000