Tormach® Injection Molder

Operator Manual

-

Questions or comments?
Please email us at: info@tormach.com

UM10158_Inj_molder_0915A
Document Part Number 32079

1 Product Specification

1.1 Product Scope

The Tormach PCNC Injection Molder is a plastic injection molding head that is designed to be used as an accessory to the PCNC series of milling machines for developing small prototype molds. The PCNC Injection Molder can be used to mold plastics up to 900°F and has a 1 oz shot capacity.

2 Safety

This product is an accessory for Tormach PCNC mills. Review and be comfortable with the safety guidelines and operating procedures of the PCNC mill prior to using the Injection Molder.

In addition to safety hazards associated with operating a CNC milling machine, there are several specific safety concerns to be aware of when operating the Injection Molder:

- **Burn Hazards.** Several areas of the Injection Molder, and in particular the heater box, can become quite hot during operation. Do not touch hot areas and wear gloves to protect from inadvertent burns. Allow proper time for the Injection Molder to cool prior to removing it for storage
- **Vapors for Melting Plastic.** Make sure to provide adequate ventilation. Many plastics will burn if heated beyond its melting temperature.
- **Pinch Hazards.** Be aware of the machine environment. Keep fingers away from springs and other pinch hazards.

3 Basic Operation

3.1 Installing the Proximity Switch

Prior to first use, it is necessary to install the proximity switch to the Ram unit. To install the proximity switch, carefully thread it on to the bracket as shown. Adjust the position to maintain approximately .050" clearance between the sensor and target collar.

Figure 1. Insert the Proximity Sensor and hand tighten the retention nuts

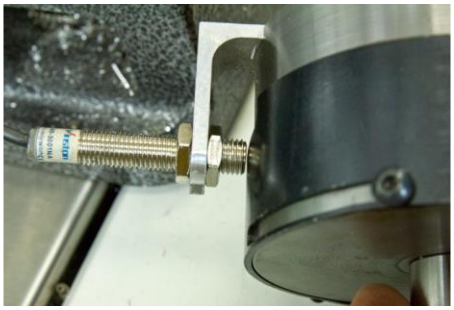


Figure 2. Attach the Sensor Bracket the Ram as shown.

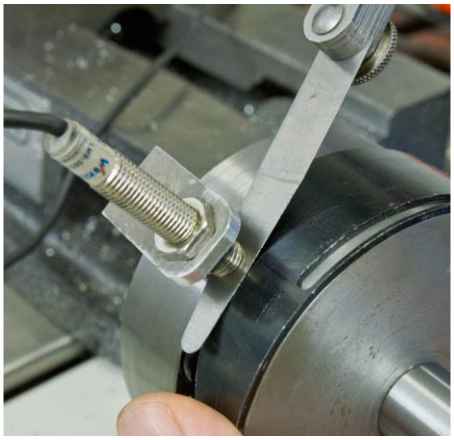


Figure 3. Adjust Sensor position; use feeler gauge or similar as reference

3.2 Mounting the Injection Molder on your PCNC

First, fasten the spindle Clamp to the outside diameter of your PCNC Spindle:

1. Slip the collar over the nose of the spindle. If the fit is tight, use a large screw driver to pry open the collar slightly.
2. Push the collar up firmly against the head casting
3. Secure the Clamp by tightening the M10 bolt to tighten the clamp

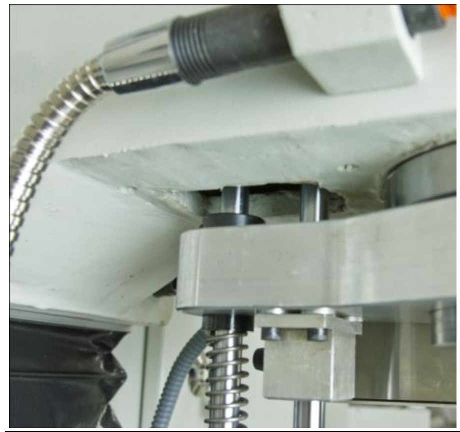


Figure 4. Position Injection Molder so rods enter spindle cavity as shown.

Figure 5. Use a screw driver to pry apart collar if fit is too tight.

Figure 6. Tighten Hex nut to secure position

3.3 Installing the Ram

The Ram has a ¾" straight shank designed for use with the Tormach Tooling System™ (TTS). To install:

1. Insert the ram into the spindle. To do this first loosen the drawbar and then insert the ram shank into the spindle. Position the ram so that the sensor does not interfere with the collar and tighten the drawbar to secure the ram in the spindle.
2. Plug the DIN connector into the Accessory port on the PCNC operator panel.
3. Place the Funnel on the Heater Box
4. Plug the controller into the 115VAC power source. <u>Do not plug the controller into the accessory (coolant) outlet on the PCNC.</u>

Figure 7. Insert Ram as shown. Adjust sensor position to avoid interference.

Figure 8. Tighten drawbaw to secure the ram.

3.4 Checking for Ram/Receiver Interference

Your Injection Molder is designed with a chamfered ram and receiver so absolute alignment between the ram and receiver is not crucial; however, you should verify relative alignment each time prior to use. To check relative alignment, slowly jog the ram down manually and take care to observe that the ram is able to pass into the receiver without issue. It is normal for the receiver to move slightly as the ram enters the receiver. If you notice severe alignment issues:

1. Re-adjust the heater box position in the XY-plane by loosening and rotating the collar
2. Re-adjust the heater box position in the YZ-plane by loosening the bolts holding heater box

3.5 Adjusting Maximum Heater Box Position

Occasionally, it may be necessary to shorten the maximum Heater Box Z position if the Injection Molder will not fit between the mold and spindle head. This is especially true if a tall mold or vise is used, or if the molder is used on a smaller machine, like the PCNC 770.

To adjust, carefully loosen the set screw on the clamping collar. Compress the spring to push the heater box closer to the Clamping collar and retighten the clamping collar, being carefully as stabilize the unit as the spring force increases. This will allow you to shorten the maximum heater box position by up to 3 inches.

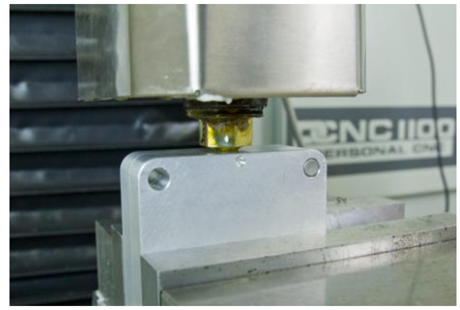


Figure 9. Position nozzle above mold sprue.

Figure 10. To check for interference, slowly jog ram into receiver.

Figure 11. Ram should pass without issue. If severe interference exists, adjust ram position and/or heater box alignment.

3.6 Setting Ram Pressure

Your Injection Molder uses a compliant ram. This design, when properly adjusted, allows for consistent pressure to be applied to the mold through the entire injection time. Ram pressure is set by positioning the ramp on the collar relative to the proximity sensor. For a high injection pressure, set the collar to HIGH. For a low injection pressure, set the collar to LOW. After setting the position, tighten the set screws to secure the target collar.

During the initial part of the injection stage, the die springs are compressed when the ram comes into contact with the plastic in the cylinder; this creates pressure inside the cylinder. This pressure forces plastic out through the nozzle and into the mold cavity. As this happens, the die springs relax, which helps to maintain a consistent back pressure inside the cylinder.

If the die springs are allowed to completely relax, pressure inside the cylinder will decrease towards the end of the injection stage. This may result in an incomplete shot. Too much pressure, on the other hand, will require additional clamping to avoid mold separation during injection and also make it difficult to release the mold and eject the part. By adjusting the Ram pressure in combination with dwell time and ram speed, you should be able to find correct conditions for most applications using trial and error.

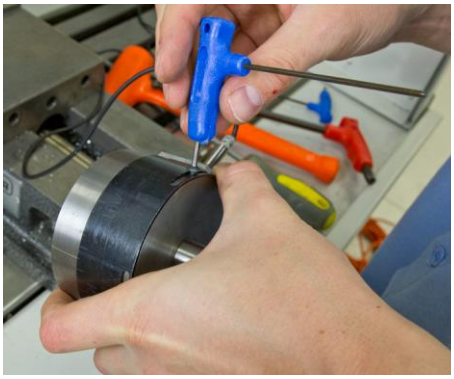


Figure 12. Loosen Collar set screw

Figure 13. HIGH position

Figure 14. LOW position

3.7 Operating the Temperature Controller

The temperature control system consist of a PID, Band Heater, and Thermocouple

3.7.1 Basic Heater Operation (Autotune)

Your Injection Molder uses Closed Loop PID with a K-type Thermocouple and 500W Heater to control the temperature of the heater box. This PID features an autotune mode that is suitable for most applications. When in Autotune mode, only the SET POINT needs to be adjusted. The controller will monitor and switch the band heater on/off to maintain temperature at or near the setpoint target.

3.7.2 Disabling Autotune

To disable Autotune mode and enter programming mode, press and hold the ↑ or ↓ key for 5 seconds. Once in programming mode, you may press and hold the ↑ or ↓ key for 5 seconds at any time to exit. With Autotune disabled, the following parameters can be adjusted using the ↑ or ↓ keys. To advance to the next parameter, press SET.

Parameter	Display Setting	Factory Default	Notes
Lower Temperature Limit	LSP	0.00	0C is maximum value for K-type Thermocouple
Upper Temperature Limit	USP	1200	1200C is maximum value for K-Type Thermocouple. Adjust the upper limit may be necessary, especially if large system oscillations occur. Think off this as "Coarse" Adjustment. Adjust Limit so it is approximately 20% above desired temperature
AL1 (Alarm 1) Hysteresis	HY1	1.00	Not Used
AL2 (Alarm 2) Hysteresis	HY2	1.00	Not Used
Decimal Point Setting	dP	0	Should not be a need for finer measurement in Application
Password Setting	ScY	015	When LcK parameter is set to 010, User must enter this value in ScY parameter to operate PID
AL1 (Alarm 1) Set Range	AL1		Alarm Mode 1 / Not Used
AL1 (Alarm 1) Mode	A01		Alarm Mode 1 / Not Used
AL2 (Alarm 2) Set Range	AL2		Alarm Mode 2 / Not Used
AL2 (Alarm 2) Mode	A02		Alarm Mode 2 / Not Used
Modification Value	PVF	0.00	Display Value = Measured Value-Modification Value. Changing this parameter will adjust the value displayed; however, it will not have any effect on temperature control. Range is ±100
Input Signal Selection	InP	K	Input Signal is K-type Thermocouple
Proportional Term	P	3.00	Range is .01 - 3600. If P=OFF, then controller is operating in simple on/off mode; no temperature feedback is being used. "Fine Adjustment"
Integral Term	I	240.0	OFF = Cancel Integral Time. This setting may be adjusted. Temperature Oscillates around setpoint, then increase this term. If the Temperature is stable but never reaches set point, decrease this term
Derivative Term	d	oFF	OFF = Cancel Derivative Time. Derivative term is not required in typical use.
Control Direction	oUd	HEAt	Sets PID as Heater Control
Control Hysteresis	HYS	1.00	Does not generally need adjustment
Output Control Time	CtL	020	Does not generally need adjustment
interval	Pc	5.00	Not Used
Cooling Output Gap	Cdb	0.00	Not Used
Cooling Output Control Mode	C_t	20	Not Used
Temperature Unit	c-F	F	Choose F for "Farenheit" or C for Celsius
Parameter Lock Code Setting	LcK	0	000 = Unlocked. 010 = Locked

4 Working with Injection Molder Software

The Injection Molder software is included with PathPilot software and uses a subset of the available PCNC control functions. Review the PCNC operator manual for general controller operation.

4.1 Accessing Injection Molder Software

Once installed, the Injection Molder screen can be accessed from the Settings tab; check the Enable Injection Molder box.

Figure 15. Accessing the Injection Molder

4.2 Injection Molder Screen Overview

The Injection Molder screen contains a select subset of PathPilot commands.

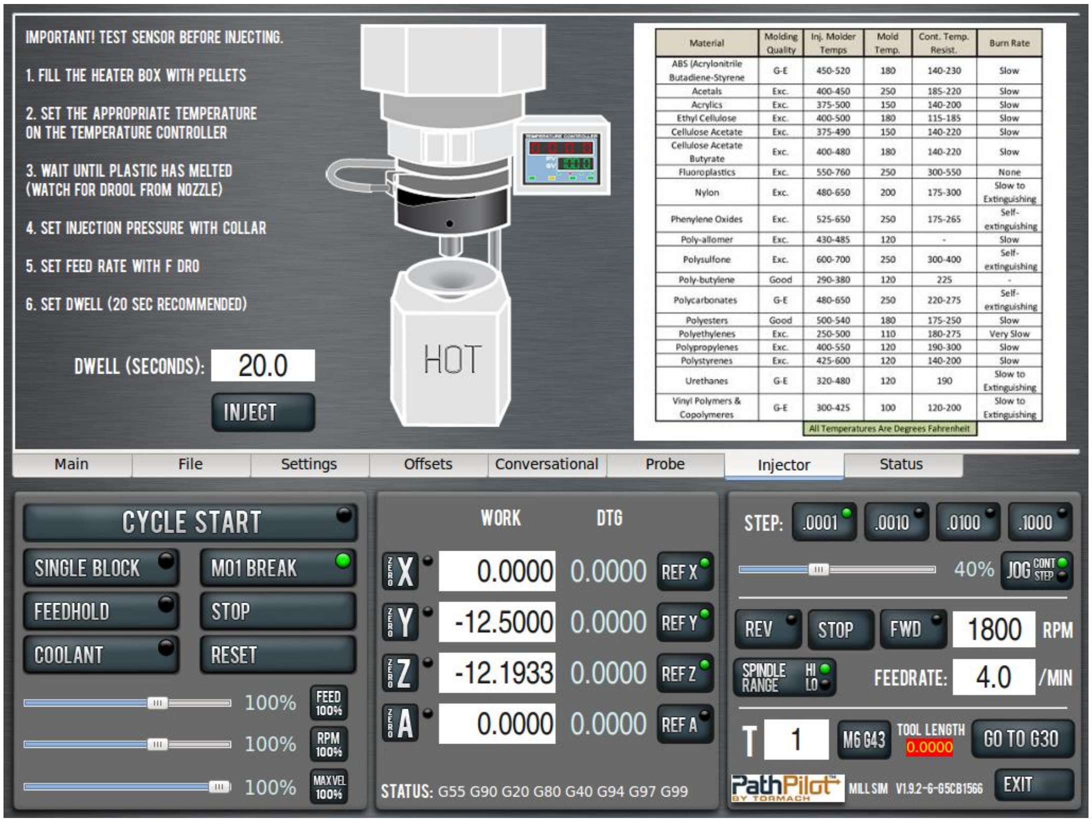


Figure 16. Injection Molder Screen Set

4.3 Position DRO

Figure 17. Position DROs

The current location of the Injection Molder is indicated in the XYZ DROs.

4.4 Setting Feedrate

To set the feed rate, enter it into the Injection Feedrate DRO. Press “Enter” to confirm

4.5 Setting Dwell

To set the dwell time, enter it into the Dwell DRO. Press “Enter” to confirm.

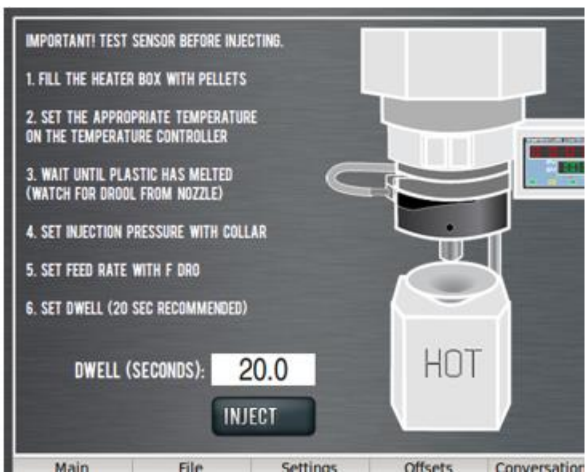


Figure 18. Feedrate and Dwell Settings

4.6 Testing Proximity Sensor

Important: *Test the proximity sensor each time that you use the Injection Molder. To do this, simply take a thin steel object, such as the blade of a flathead screwdriver, and place it near the sensor face. You should see the sensor LED light on the control screen.*

4.7 Start/Stop Injection

To start an Injection, click inject (see **Figure 18**).

To abort or prematurely stop an injection, click stop (see **Figure 16**). Alternatively, you can also press “RESET” or hit the E-Stop button to immediately stop an injection (see **Figure 16**).

Click “Exit” button to leave the injection screen set and return to the default mill screens (see **Figure 16**).

4.8 Advanced Functions

The following functions are optional convenience features.

4.8.1 MDI (Manual Data Input)

Use the MDI input to input G-Code commands not supported by the main screen set. Use this feature with care to so as to not crash the Injection Molder attachment.

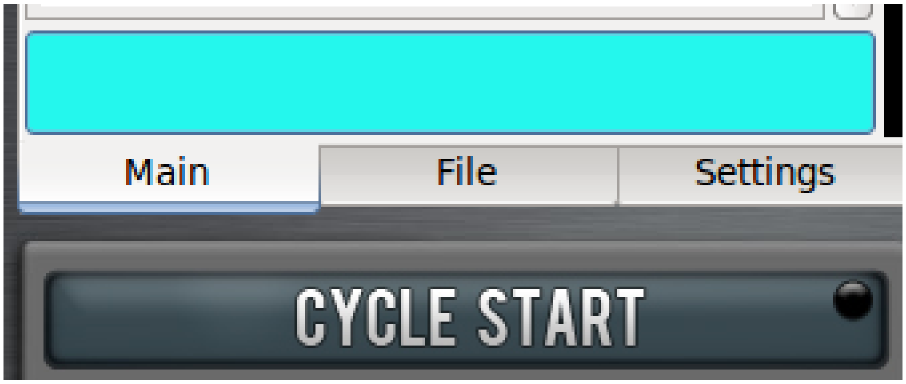


Figure 19. MDI Line

4.8.2 Work Offsets

Use the Arrow keys to cycle through work offsets G54-G59. When using multiple molds, you may find it useful to use a separate work offset to store the location of each mold. Click the “GOTO X0Y0” button move to position X0Y0 for the work offset of the position selected.

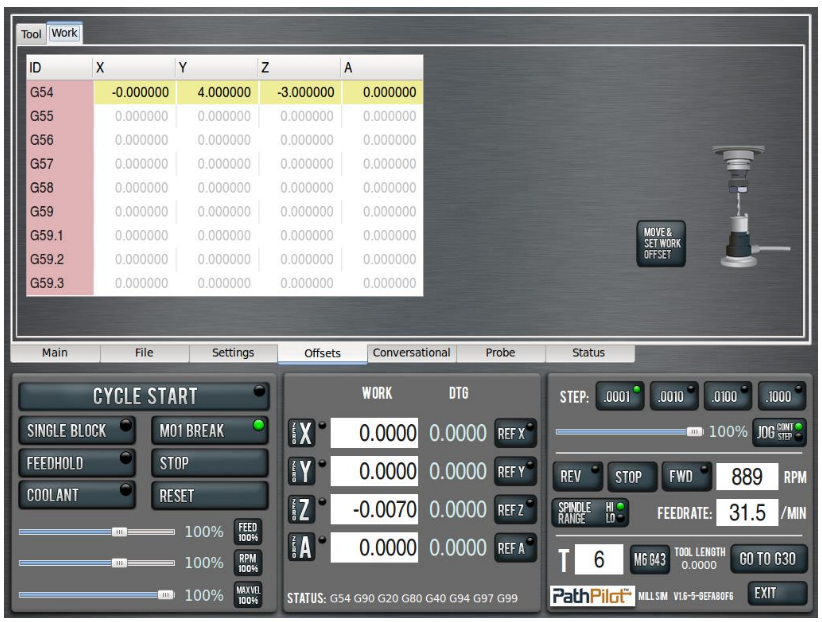


Figure 20. Multiple Work Offset Sections

5 Making your own Mold Blanks

Making your own mold blanks is not difficult, but does require attention to detail to achieve good results. The following procedure is one example of how to produce your own Aluminum Mold Blanks. This design uses .250" pins that are press fit into one half of the mold; the methods presented can be adapted to produce other designs as well.

Materials Required:

- 2 Aluminum Blocks for Mold Halves (slightly oversize from final dimensions)
- .250" Dowel Pins

Recommended Tooling:

- Spot Drill / Chamfer Tool
- Fly Cutter
- .375" Carbide End Mill
- Screw Machine Drill "Size D"
- .249 Reamer
- .251 Reamer

Workholding:

- Machinists Vise
- Parallels or Step Jaws
- Workstop

5.1 Sample Procedure for Making Mold Blanks

1. Set up in the vise

Small molds can be held comfortably in a machinist's vise. It is important to support the work from underneath. Step Jaws are an excellent choice, but parallels can also be used as well. This will leave an approximately .100" shelf that can be removed after the part is flipped over during Operation 2.

Using a workstop is advantageous since you'll be repeating each procedure twice, once for each mold half, and the workstop will helpful in quickly position work during the second iteration.

2. Square each block to uniform size

Take care to square each block to a uniform size. This is critical for properly aligning mold halves

3. Spot the pin hole location in 2 corners

This design uses two pins to align the mold halves. The pins are positioned at opposite corners. Use a spot drill to mark the location of each hole, referring to the drawing for exact location.

4. **Drill the Holes**

   Once the holes have been spotted, the next step is to drill them out. Using a stubby drill such as a Screw Machine Drill with an ER toolholder or similar will minimize deflection and tool run-out and produce straight holes.

5. **Ream the Press fit holes**

   For best results, use a reamer to finish the holes so they are slightly undersize. Using a .249" reamer will result in a slight interference fit between the hole and the .250" pin.

6. **Profile the Setup (Rough/Finish)**

   With an End Mill, profile the perimeter to size. This is best done first with a roughing operation that leaves the perimeter approximately .005" oversize, which can be removed with a final finishing step.

7. **Face the top (mating) surface**

   Use a flycutter (or similar) to skim cut the surface until it is flat. You may need to repeat this several times to achieve a completely flat surface.

8. **Chamfer Pin Holes / Edge Break Hole Perimeter**

   Using the chamfer drill, break the edges of the holes. This is important as it allows the pins to be easily pressed into the mold half. Alternatively, you could also use a hand chamfer tool to break the edges.

9. **Flip the part and remove the remaining material from the back side**

   Turn the part over in the vise. Use a roughing mil to remove the majority of the remaining material before using a flycutter to finish the final surface.

10. **Use the endmill to create a small notch on the edge of the mold.**

    This is very helpful in providing leverage to separate the mold halves with light pressure from a screwdriver after they are assembled.

Make the second half in a similar manner.

At this point, it is appropriate to repeat steps 1-8 to create the second half, this time substituting the Slip fit Reamer for the Press Fit reamer in Step 4. Remember, the top (mating) surface of this mold half the opposite face from the first half.

11. **Press fit the dowel pins into the appropriate holes**.

You can do this with an arbor press, but you can use the vise to accomplish this: Place the dowel pins in the slip fit holes, then position the press fit mold half so it is aligned with the pins. Place both halves carefully in the vise, and close the vise until pins are forced into the press fit half of the mold. Using this method will allow you to avoid pushing the pins in crookedly.

6 Making your First Mold: Poker Chip

This first mold project is designed to demonstrate the basic steps to make a simple mold. A mold needs 3 parts:

1. **Cavity** – basically the negative of the part that you wish to make
2. **Vents** - A vent is a shallow passage that provide a way for air to escape from the mold.
3. **Sprues** – This is the passage through which liquid plastic is introduced into the cavity

Molds may also have optional components. There include cores and inserts. Please see the appendix for some examples of these applications.

6.1 Cavity

Once you have a suitable mold blank, the first step is to make the cavity. For our poker chip mold, the cavity is a simple circular pocket, 1” in diameter and .050” deep.

Because the mold cavity must be split between both sides of the mold, an important challenge in mold making is ensuring that the mold halves correctly line up. Care must be taken to make sure the programming reference point (i.e. work offset) of one mold half can be related to its partner half.

The most reliable way to do this is by establishing work offset at the center of the dowel pin hole location. To do this, you’ll need to center the spindle over the center line of the dowel hole. There are several techniques to do this:

1. With a touch probe – use the hole center function as described in the PCNC Operator’s manual
2. With a dial test indicator – Use a spindle mounted dial test indicator to swing indicate the center of the hole.
3. With a drill bit sized to the hole diameter – Carefully position a drill bit so that it can pass into the hole unobstructed.

Once the spindle is centered, set your work offset as described in the PCNC Operators manual.

6.2 Vents

For simple mold design, a basic simple venting strategy is as follows:

.001” deep x .050” wide perimeter channel

Plus

(11) .001” deep x .050” wide vents radiating towards the mold edges and equally spaced.

More sophisticated mold designs require other venting strategies.

6.3 Sprue

The final step in making a mold is to create the sprue. This is best done in the following manner

1. Fit the mold halves together and securely hold the mold in the vise
2. Center the spindle over the parting line of the mold
3. Use a 90 deg countersink or spot drill to create a shallow tapered hole to receive the Injection Molder nozzle (approximately .300-.400" in depth is adequate).
4. Use a twist drill to complete the sprue passage into the cavity. A sprue of diameter between .100 and .200" is generally adequate.
5. Remove the molds for the vise and separate them. Clean out any chips, etc that have fallen into the cavity.

6.4 Polishing the Mold

Polishing the mold is optional, but recommended for best results. A typical polishing sequence for an aluminum mold is as follows

1. Remove tool marks with a 120 grit ultra soft stone. **Use stoning oil.** Thoroughly clean the surface when complete to remove the any loose grit before proceeding to the next step.
2. Repeat step one, this time with a 320 grit ultra soft stone until all the scratched from the 120 grit stone have been removed.
3. Repeat twice more, this time with 400 grit. Followed with a final polish of 600 grit aluminum oxide wet dry sandpaper.
4. Use a diamond polishing paste to finish, such a 15 micron blue compound. Use a diamond lapping oil. You can also use a brush fitted to an electric rotary tool for this step. You can continue to step down to by using finer pastes is a better finish is required.

7 Your First Shot

1. Fill the heater box cylinder with pellets
2. Set the appropriate temperature on the Temperature Controller. For LDPE, choose 450°F
3. Wait for the plastic to melt. Watch for drool from the nozzle; this is an indication that the plastic is ready. Monitor so the plastic doesn't burn
4. Set the injection collar midway between LOW and HIGH
5. Set the RAM SPEED to 20 IPM
6. Set the DWELL to 20 sec

After the first shot, remove the mold and assess part quality. Adjust molding parameters as necessary.

Troubleshooting Part Quality Issues:

Short Shots or Surface Wrinkles • Increase ram speed • Increase injection pressure • Increase mold temperature • Increase ram pressure time • Increase runners or gate size • Increase mold venting • Increase time between shots	Warping of Parts • Reduce mold temperature • Increase ram pressure time • Increase time between shots • Reduce material temperature • Check part design for section variations
Sinks • Increase ram pressure time • Increase ram speed • Enlarge gate • Reduce material temperature • Increase mold venting	Dimension Variation Shot to Shot • Establish and maintain cycle time • Keep mold temperature constant • Maintain constant material temperature • Maintain constant injection pressure • Increase mold venting
Discoloring • Decrease material temperature • Reduce time between shots • Thoroughly dry material • Thoroughly preheat material	Part too Small • Increase injection time • Increase injection pressure • Reduce material temperature • Enlarge gate
Gassing • Decrease material temperature • Thoroughly dry material • Preheat material thoroughly	Part too Large • Reduce injection pressure • Decrease injection time
Weld Marks • Increase mold venting • Increase material temperature • Increase gate size • Move gate or use multiple gate • Increase injection speed and pressure • Add overflow puddle	Surface Streaking • Raise mold temperature • Thoroughly preheat material • Thoroughly dry material • Reduce injection rate

8 Troubleshooting

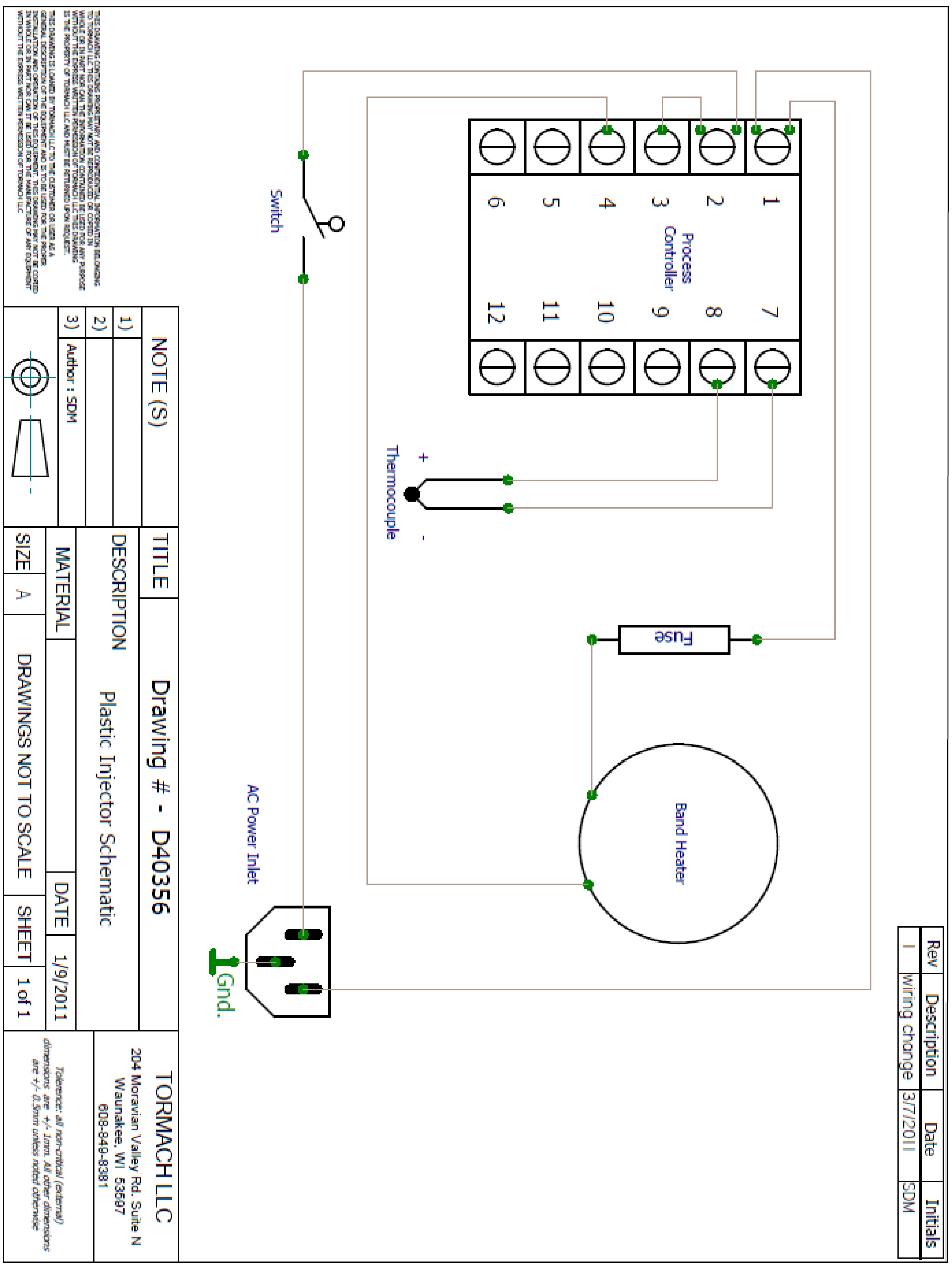
No Heat			
Possible Cause	Probability	Action to Identify Cause	Discussion
Temperature Controller is not Plugged in	High	Make sure that power cord is plugged in a power is	Unit requires 120VAC power. Do Not plug unit into mill accessory port.
Fuse is Blown	Medium	Check fuse	Disconnect power before checking or replacing fuses
Band Heater is bad	Low	Check Continuity across heater leads	
Controller is bad	Low		

Proximity Sensor Does Not Work			
Possible Cause	Probability	Action to Identify Cause	Discussion
Sensor is too far from target	High	Wave the tip of a steel screwdriver in front of the proximity sensor and confirm that “Test Indicator Before Move” LED can be indicated.	Use a feeler gauge or similar to adjust the position of the sensor to within .50” (1.2mm) of target collar.
4th Axis homing is disabled	High	Wave the tip of a steel screwdriver in front of the proximity sensor and confirm that “Test Indicator Before Move” LED can be indicated.	Check to see that 4th Axis Homing is enabled using the PCNC Config Utility. PCNC Config can be accessed via the Windows Start Menu on your PCNC Controller.
Sensor is damaged	Low	Wave the tip of a steel screwdriver in front of the proximity sensor and confirm that “Test Indicator Before Move” LED can be indicated	If no indication, sensor is damaged and needs replacement.

Temperature Controller is not Functioning Correctly			
Possible Cause	Probability	Action to Identify Cause	Discussion
Temperature Controller is configured wrong	High	Review Section 2.5	
Thermocouple is damaged or loose	Low	Inspect thermocouple wiring and location	Welded tip of Thermocouple should be located next to band heater for best results.
Thermocouple is mis-wired.	Low	Reverse thermocouple connections	Thermocouples must be wired with correct polarity

Computer Communication cannot be Established			
Possible Cause	Probability	Action to Identify Cause	Discussion
Mill Communication Issues	High	Machine LED on Mill cannot be turned on	Review PCNC operators Manual, Table 3.1

8.1 Electrical Schematic



8.2 Parts List

Spare Parts	
31921	Proximity Sensor Assembly
31937	Band Heater
32040	Nozzle
32038	Funnel
32061	Control Box Subassembly
32059	Thermocouple, K-type
32065	Ram Subassembly
30171	Rocker Switch
30258	AC Power Inlet
30182	5x20mm Fuse, 6.3A Fast Blow
32277	Temperature Controller
30191	Cord with NEMA 5-15P Plug
32049	Set Screw Collar
32050	Poly Urethane Bumper
32044	Return Spring
Accessories	
32374	LDPE Plastic Pellets, 1lbs.
32075	Mold Blank with Dowel Pins
32076	Cleaning Rod, Nozzle
32077	Ram Scrapper Tool
32066	Purge Block Tool
32708	Manual

APPENDIX: EXAMPLE APPLICATIONS

The following sections are re-published with permission from:

Cutting Costs in Short-Run Plastics Injection Molding
Written by Morgan Industries, Inc.
3311 East 59th Street
Long Beach CA, 90805
Copyright April 1998

INTRODUCTION

The plastics industry is one of the fastest growing major industries in the world. Every year there is an increase in the amount of plastics used in all types of products. A good example of this is the percent of plastics used in today's automobiles compared to 15 years ago.

The continued rate of growth in the industry hinges on the development of improved and new thermoplastics with greater physical properties. This has opened the door to applications never thought possible before. These are emerging both as product innovations and as existing products converted from materials such as metal, glass, wood, or paper to plastics for competitive and economic advantage.

The plastics injection molding field, at large, is volume oriented. Vendor sources, particularly mold makers and custom molders, are geared to long tool life and high volume production. Plastics equipment manufacturers, likewise, have concentrated almost exclusively on fully automatic, sophisticated injection molding machines whose economics lie in single runs of 100,000 parts or more and in multi-shift operations.

Definition of Short-Run Plastics

In order to prevent any misconception, we are defining short-run as production in which annual requirements for a particular part range between one and 25,000 parts. Typical of these are nonstandard or engineered plastic parts whose piece cost is presently above $.25. Recognizing the wide substitution possible between metals and plastics, it is not necessary to limit applications to parts which are presently in plastic. As your experience grows, you will want to analyze the merits of converting a number of non-plastic parts or components to this material.

Economics of Short-Run

Chapter V sets forth a wide number of examples of simple tools designed for various plastic parts. Cost savings are in an impressive range of 44 to 94 percent and the reduction of lead times is equally significant.

These examples have been analyzed on a total cost basis amortizing the tool over a single year's production. At the same time, we should point out that the tool life is assumed to meet three to five years of production.

The cost savings over alternate production methods are realized through the following features:

– Tools designed and built as Class B or C tooling (see Chapter II, page 10). All areas which do not require finish or tolerances are completed in the rough; heavy cost cutting is concentrated on the tool base, ejector systems, use of inserts and the reduction or elimination of heating and cooling systems.

– These tools are meant to operate according to predetermined quantity requirements. For example, if 50-100 large parts per set-up are intended, the tool plates might be bolted together rather than be built to withstand high clamp pressures.

– Likewise, the tools are designed to last the life of a short-run tool rather than several million parts. The life factor is geared to anticipate 2-5 year production requirements as well as to anticipate part or product obsolescence.

– Very often, as an added means of maintaining the total economics of a year's production run, a tool will be designed to complete about 80 percent of the part detail. Limited machining of the parts on simple jigs is then required to complete them. This approach is also used to achieve extremely tight or critical tolerances.

On balance, the economics of the proposed short-run tools lie in flexible design to meet intended production requirements in terms of quantity, quality, product obsolescence and labor input.

CHAPTER ONE

INJECTION MOLDING PROCESS

THEORY OF INJECTION MOLDING

The theory of injection molding can be reduced to four simple individual steps: Plasticizing, Injection, Chilling, and Ejection. Each of those steps is distinct from the others and correct control of each is essential to the success of the total process.

Plasticizing describes the conversion of the polymer material from its normal hard granular form at room temperatures, to the liquid consistency necessary for injection at its correct melt temperature.

Injection is the stage during which this melt is introduced into a mold to completely fill a cavity or cavities.

Chilling is the action of removing heat from the melt to convert it from a liquid consistency back to its original rigid state. As the material cools, it also shrinks.

Ejection is the removal of the cooled, molded part from the mold cavity and from any cores or inserts.

Repetition of these basic steps in sequence is the process of injection molding.

THE PRACTICE OF INJECTION MOLDING

The practice of injection molding varies from the theory, only in as much as process limitations and available equipment affect it.

Step by step the process as practiced is as follows:

Material Preparation

Before the thermoplastic material is plasticized, it may be necessary to do some simple yet essential preparation.

Drying

Many materials are hygroscopic and will absorb moisture from the atmosphere. This moisture may be absorbed into the granules, it may adhere to the outer surface of the material, or both. It is absolutely essential that this moisture be driven off before plasticizing is attempted. The heat of plasticizing, usually from 350°F to 700°F, will cause any moisture present to vaporize. The steam and pressure generated will cause the plasticized material to foam or blister and will seriously affect the quality of parts as well as hamper processing it in the equipment.

The drying of granules is an easy, low-cost process. For short-run production the material is spread in one inch layers in trays and placed in a small heated oven with generous venting to the atmosphere. A typical drying period may call for two hours at 200°F.

Do not attempt to plasticize a material until the correct drying conditions have been met. The Plastic Properties Chart (Appendix III) contains drying recommendations.

Dry Coloring

Many plastic parts today are made in color. However, it is impossible to stock all available materials in a wide range of colors. Even the material manufacturers themselves usually stock in natural color: that is, the color which the processed granules have before any pigments are added. Many standard color pigments are available commercially. To obtain small quantities of special or custom colored material, a small percentage of color pigment, usually in powder granular form, is mixed with the natural granules. A typical situation might call for 2% by weight of pigment to be thoroughly mixed with granules. Any sealable container will do as a mixing chamber. A few minutes of active shaking will produce a good color blend.

Pre-Heating

In the rare event that the plasticizing capacity of the molding machine constrains the rate of molding production, pre-heating may be useful. Since most thermoplastics are extremely sensitive to excessive or prolonged high heat, the material integrity can be protected by pre-heating the granules from room temperature to a point below that at which the granules become soft, before feeding the material into the plasticizing chamber of the machine. In this way, the amount of heat needed to complete the plasticizing is considerably reduced.

Plasticizing

The rate of plasticizing is an important consideration in a high-speed molding process. It becomes less significant and a minor problem in short-run applications; nevertheless, the method of evaluating the plasticizing rate can be helpful.Plasticizing typically depends on the size and temperature of the melt cylinder, the type and grade of material being used, the starting temperature of the material, and the rate at which material is fed into the melt cylinder.

Molding equipment is rated in its plasticizing capacity by the ability to melt styrene measured in pounds/hour. The mathematics of plasticizing in a production cycle are straightforward. For example, assume that the time required to plasticize polypropylene in a melt cylinder at 550°F is three minutes. If the melt cylinder capacity is four ounces, then the plasticizing rate of the equipment would be 1 1/3 ounces per minute. Production is then limited in continuous cycles to a rate not to exceed one shot of 1.3 ounce weight each minute, or .65 ounce every 30 seconds. Exceeding this rate will cause improperly plasticized material to be fed into the tool, and incomplete parts - or short shots - will result. By pre-heating the granules as previously mentioned the plasticizing rate can be increased.

Another variable condition which can change the rate of plasticizing is the melt cylinder temperature. For example, if a material has an ideal melt temperature of 400°F, the production rate can be increased by raising the cylinder temperature to 450°F. This change in melt cylinder temperature, to compensate for or to aid production rate increases, is normal in regular production cycles. It can also be a high risk venture. If, for example, the production rate was reduced and the melt cylinder temperature was not, the material in the cylinder would rapidly burn. Physical properties would be drastically reduced, colors would change, and in many cases, fumes or toxic gasses would be generated. Some materials would pass through the range of normal melt and "best flow" conditions and tend to re-solidify. It can be difficult to purge this degraded material from the equipment.

Every thermoplastic material has a temperature range outside of which it will not process as desired. Low temperatures will result in short-shots and require excessive injection pressures. Higher than ideal temperatures will produce gassing, discoloration, and impair physical properties.

For continuous good results, there is no better combination than an "average" melt cylinder temperature, properly dried and pre-heated granules, and a steady non-fluctuating production rate. Establishing the values for these variables is a simple yet critical task. The first step must be to obtain constant extrusion.

Extrusion

The material granules fed into the top of the heated melt cylinder are compacted by the ram piston. As they move downwards towards the nozzle, they become plasticized. Ideally, the melt cylinder should be refilled after each shot, granules being fed into the top as the heated material leaves the nozzle. The small diameter ribbon of plasticized material leaving the nozzle is known as extrusion.

In order that the extrusion be in a condition ideal for injection, it must be correctly heated, free of blisters or gassing, free of lumps which would indicate unmelted granules, and have a smooth glossy exterior. Use caution and wear protective gear when creating an extrusion. This molten material is under pressure and trapped air or gas may cause the material to pop and splatter when exiting the nozzle.

Before injecting a part of about one-half ounce, it is recommended practice to extrude a ribbon of this weight (or volume) from the nozzle. The extrusion gives immediate, visual evidence of the condition of the material prior to injection into a mold.

Injection

The act of injecting the heated material into a tool cavity usually requires considerable pressure. This pressure will vary dependent on the ease with which the material will flow, the size of the orifice through which it must pass, and the size or complexity of the cavity.

The variables to be considered in injection are the pressure and the rate of injection. Since correctly plasticized materials act typically as hydraulic fluids, for any given injection pressure the rate of flow will vary with the size of the orifice - or gate - between the nozzle and the cavity. If the gate area is doubled, the rate of flow will double. The cavity should be filled with material before the temperature drop caused by contact with the relatively cooler mold changes the material's flow characteristics. Material flowing into a tool will flow around both sides of any obstruction to meet on the reverse side of it. If the material temperature has dropped sufficiently, then the two fronts of material will not bond into each other and a weld-line develops in the part. Listed on page 101 are steps that can be followed to reduce or eliminate weak weld points (weld marks).

Naturally, each of these alternatives has both advantages and disadvantages. The material temperature can only be raised to the upper limit of the ideal range. A substantial increase in the mold temperature will reduce the rate of chilling and consequently slow down the production rate. Increases in injection pressure above that required to fill the mold will require additional clamping and part ejection pressure. An increase in gate size will leave a larger mark on the finished part. If the gate witness is not objectionable on the part, a gate correction should be the only necessary change to solve this problem.

Venting

As the material is injected under pressure into the tool, the air in the cavity must be allowed to escape to atmosphere. Any restriction to this escape of air will generate a back-pressure against injection causing a reduction in the rate of material flow and encouraging air entrapment within a part. It can also contribute to incomplete or short shots in the cavity.

It is customary to vent the mold at the parting line. That is, cut air vents from the cavity to the outside of the mold. A typica1 vent would be .003 inches deep and 1/4 inch wide. The vents should be located at the extreme corners of the cavity farthest from the gate. If during sampling one or more areas of a cavity tend to underfill -as indicated by short-shots or knit-lines - these areas should be vented. If a vent is too deep, a small amount of flash will tend to appear on the finished part. Over-venting is always preferable to under-venting since the flash is easily removed in a secondary operation. An unvented mold is an incomplete mold.

Sprue, Runner and Gate

After leaving the nozzle and before entering the cavity, the material must flow through one or more passages in the tool. The first of these, which leads directly from the nozzle is a tapered passage running in line with the melt cylinder towards the center of the mold. This is the sprue. Its function is to guide the material directly to the mold parting line. In low-cost tooling designed and built for prototyping or short-runs, the sprue may run directly from the nozzle to the cavity (see page 44). It may be tapered in either direction, so that the large end is open to the cavity, or so that the large end is adjacent to the nozzle and the small end connects to the cavity via a gate (see page 36). It is always tapered so that the material, once chilled, can be easily removed.

When the large end of the sprue is adjacent to the nozzle, it is normal to place a nozzle adapter between the mold top surface and the nozzle. The adapter has a conventional ball seat and a tapered and serrated short sprue. After chilling, the sprue may be removed from the mold by simply rotating the adapter which breaks off the sprue at the gate. A light tap on the bench top will eject the sprue from the adapter. Naturally, the hardened steel adapter also protects the top surface of the tool from damage or wear.

In tools where more than one cavity exists, or in some three plate tools, the material must be guided by runners from the lower end of the sprue to the cavities. It flows along, or parallel to, the tool parting line at right angles to the sprue. The runners should be as short as possible with a minimum of turns or corners. A good runner has a circular cross-section since this gives the lowest area of exposure to the cooling effects of the mold. A half-circle or flat, ribbon type runner is bad practice.

A gate is located between the runner and the cavity. This gate is a small orifice through which the material must flow to enter the cavity. The gate may be round or rectangular. Typically it will be as large as aesthetic considerations will allow. If the part to be molded has a wall thickness of .06 inches, the gate should be at least .06 x .06 inches square or an equivalent area if round (approximately .09 inches diameter). The gate should be as short as possible and never longer than its diameter or square size. Once again, half-circle or flat, rectangular gates are not desirable.

Chilling and Tool Temperature

The temperature at which a tool is maintained is always a compromise. A cool mold allows fast chilling of the material and higher production rates. However, if the material is cooled too quickly, poor surface finishes are obtained. Often a "rippled" finish or "orange-peel" effect will result on the part. Premature freezing of the gate can cause short-shots, knit-lines, and excessive sink. Material temperatures must be higher than ideal and higher injection pressures are often required to give an injection rate fast enough to fill the mold before the material chills.

An over-heated mold will cause extended chill times and resultant long cycles. The part when finally removed from the tool will still be hot: subsequent cooling outside of the cavity will cause distortion and reduce dimensional stability.

A fluctuating mold temperature will cause variations in part quality and size. Within the closed mold, shrinkage is restricted, and so parts leaving the mold at different temperatures will have different amounts of shrinkage.

As a cool mold is run, it tends to increase in temperature. However, with each successive shot, the rate of temperature rise is reduced until, after some period of time, the mold temperature will stabilize. Heat loss to atmosphere will balance heat gain from the material injected. With short-run tooling (usually aluminum) the temperature will stabilize well below that which will cause part distortion. Parts made on the tool before the stabilized temperature is reached will all be subject to size and quality variation.

To eliminate this problem, a pre-heat plate is recommended. This heater will quickly raise the mold to correct working temperature and will maintain this temperature throughout the day. (See the Plastics Properties Chart for recommended tool temperatures.)

Cycle Time

All thermoplastic materials are sensitive to heat. Because of this, the rate at which they can be plasticized is limited. Once plasticized, they may be maintained at this temperature for relatively short periods of time. Improperly plasticized materials cannot be successfully shot; over-heated materials will degrade and discolor.

As a result, the material should be kept moving through the machine at a relatively constant rate. It is therefore essential to establish a regular cycle of operation. The time interval between shots must allow for a thorough melt, and yet prevent burning. Naturally, the melt cylinder temperature can affect this time, as can the type of material. Typically for prototype and short-run molding, cycle times will vary from 30 seconds to 10 minutes, depending on the shot size, the amount of material to be plasticized between each shot, the material heat tolerance, and mold handling requirements. It is most important that, once established, the time interval between shots is held fairly constant. Use a clock or a timer to regulate the cycle.

Establish a cycle time which will allow a good shot each time and will maintain a desired mold temperature. If testing indicates that a material will plasticize at a rate of 4 ounces each 5 minutes in the machine, it is far better to set a production pace which will use only 3 ounces in 5 minutes. This way the material will always be thoroughly prepared. This is consistent with the principle of designing only up to about 80% of rated capacity. The apparent loss in capacity will actually pay off in reduction in scrap parts, and production of parts with consistently good dimensions and surface finish.

Ejection

When chilled - usually about 10 seconds after injection is complete - the part should be removed from the tool in such a way as to avoid any damage. A number of mechanical methods are used and each method has one factor in common. When the material cools, it shrinks away from the outer cavity and on to any internal insert or core. Hence, when the tool opens, it is possible to accurately predict on which side of the tool the part will be found. The ejector system must be designed to push the part off the core (see pages 38 & 84).

In conventional production tools, ejection is performed by a pattern of ejector pins in the mold base. These are attached to a moving plate which is forced upwards after the mold is opened (see page 96). In prototype or short-run tooling, a threaded core is often left loose and can be removed from the tool with the part molded around it, the part is then unscrewed from the core (see page 44). An ejection slip ring (see page 38) that is lever activated is another prototype technique for loosening or "stripping" parts off of a core.

Another well-proven method is to leave the sprue attached to the part and use this to pull the part from the mold (see page 66). If a reverse tapered sprue has been used, it is often possible to open the mold with the sprue still attached to the part and thus ensure that the part is withdrawn from the mold (see page 36).

To avoid any damage to the core or cavity in aluminum tooling, always use a plastic or wooden device to pry the part loose. When the sprue is attached to the part, the sprue can be pulled with pliers since it will be discarded.

CHAPTER TWO

TOOL DESIGN AND BUILDING

There are three distinct types of tooling in everyday use. They are identified by their method of usage, their intended life, and their cost.

HAND MOLDS

The hand mold is usually a low cost, simple tool which is placed in the injection press, clamped, filled, and then removed by hand to be opened on the work bench for removal of the molded part. Typically, production rates run to a maximum of 80 parts per hour. The tool consists of a few simple aluminum parts, each a functional part of the core or cavity. Tool life is usually measured in the hundreds of parts. The hand mold is used when part quantity requirements are low, and tool cost is more important than part running cost. In correctly made aluminum hand molds, the part quality and finish can be as good as from more expensive steel tooling.

SEMI-AUTOMATIC MOLDS

In this type of tooling, the upper or hot side of the mold is attached to the molding machine so that the nozzle and nozzle seat are adjacent. The sprue must be conventional in that it tapers to increase as it moves away from the nozzle. The base or core side of the mold is bolted directly to the machine table. Either a hand-operated or semi-automatic ejection system will aid in removing the part from its cavity. As the molding machine operates, the mold is clamped, filled, and opened. An operator is required to remove the parts and cycle the equipment. This type of tooling is used when part quantities tend toward the low thousands. The production rates would typically be 100 to 180 parts per hour depending upon the part size and the material being used. Once the equipment is set up, semi-skilled labor can be used to operate it. Tool materials would normally be aluminum with beryllium copper or steel inserts. Tool life is typically in the 10,000 shot range; this can be significantly extended by use of finishes such as hard anodizing or chrome plating, or use of steel cavities and cores.

This type of tooling is extensively used for short-run applications. Part costs are often a small fraction of the costs of fabricating by other methods and quality can be every bit as good as from full production tooling.

AUTOMATIC MOLDS

Since this type of tooling is used when production quantities are large and piece price is of supreme importance, discussion here will be limited. Tools are usually of heat treated steel, ejector systems are automatic, and mold heating and water chilling systems are added.

The cost of set-up and production of a few thousand parts can sometimes be less than the normal "set-up" charges for automatic production equipment.

TOOL QUALITY DESIGNATIONS

Tool quality can also vary, depending on the need for parts. The requirements for quantity, quality, dimensional accuracy, surface finish, and materials to be molded can drastically affect the required quality of the tooling to be built. In order to define tool quality simply, it is designated Class C, B, or A. These classifications are not intended strictly as an evaluation of the workmanship involved in tool building, but rather as an indication of expected tool life.

Most hand molds would be designated Class C. The design, use of materials in the tool, and total tool building time would reflect the need for economy due to small part quantity requirements. Rather than complicate the tool to develop very fine tolerances or intricate detail in the molded parts, the parts might be subjected to secondary machining operations after molding. It would obviously be better to drill and tap a #10-32 cross-hole in 50 parts, an hour's work at most, than to spend four hours making and fitting a rather delicate core to the tool. Every tooling decision must be made with economics in mind. The objective becomes to achieve the lowest possible total cost of producing the tool and the required molded parts. Since the hourly labor cost for operating the molding equipment will be less than the hourly costs of a machinist or toolmaker, the decision will usually be to reduce tool building time and to increase molding and post-mold finishing time.

A Class B tool would typically have more informal detail, be made from more expensive or harder-to-work materials, and be intended for semi-automatic use. The increased mold making labor would be justified by higher parts production, faster production rates and fewer secondary finishing operations. Tool cost is, however, still a significant consideration. Cross-holes in the part are produced by hand-pulled cores or loose inserts. Internal and external threaded cores or inserts would be hand-rotated. Tools may contain engraving, surface texturing, polished and plated sections. The mold may have more than one cavity. One Class B mold is an excellent, low cost short run production tool.

Class A tooling is produced with extended life, speed of operation, and part costs as prime considerations. It is used in short-run situations only as a stop-gap measure, usually for pilot or sample production. There is no such thing as low cost Class A tooling. To produce Class A tooling requires the services of experienced designers and toolmakers using the finest materials. There are many fine tool-making shops ready and willing to quote and build Class A tooling to your needs.

The object of this book is to help you to become proficient in the design and building of Class B and C tooling, using limited resources and knowledge. With proper engineering help and encouragement, anyone with a knowledge of drafting or machining can produce perfectly acceptable Class C tools. These tools will produce valuable and competitively priced molded parts in short-run quantities. Many companies have found that in an extremely short period of time, they were producing good Class B tooling. The examples illustrated in Chapter V are typical of the tooling produced by machinists previously inexperienced in toolmaking.

MAKING THE TOOLING DECISIONS

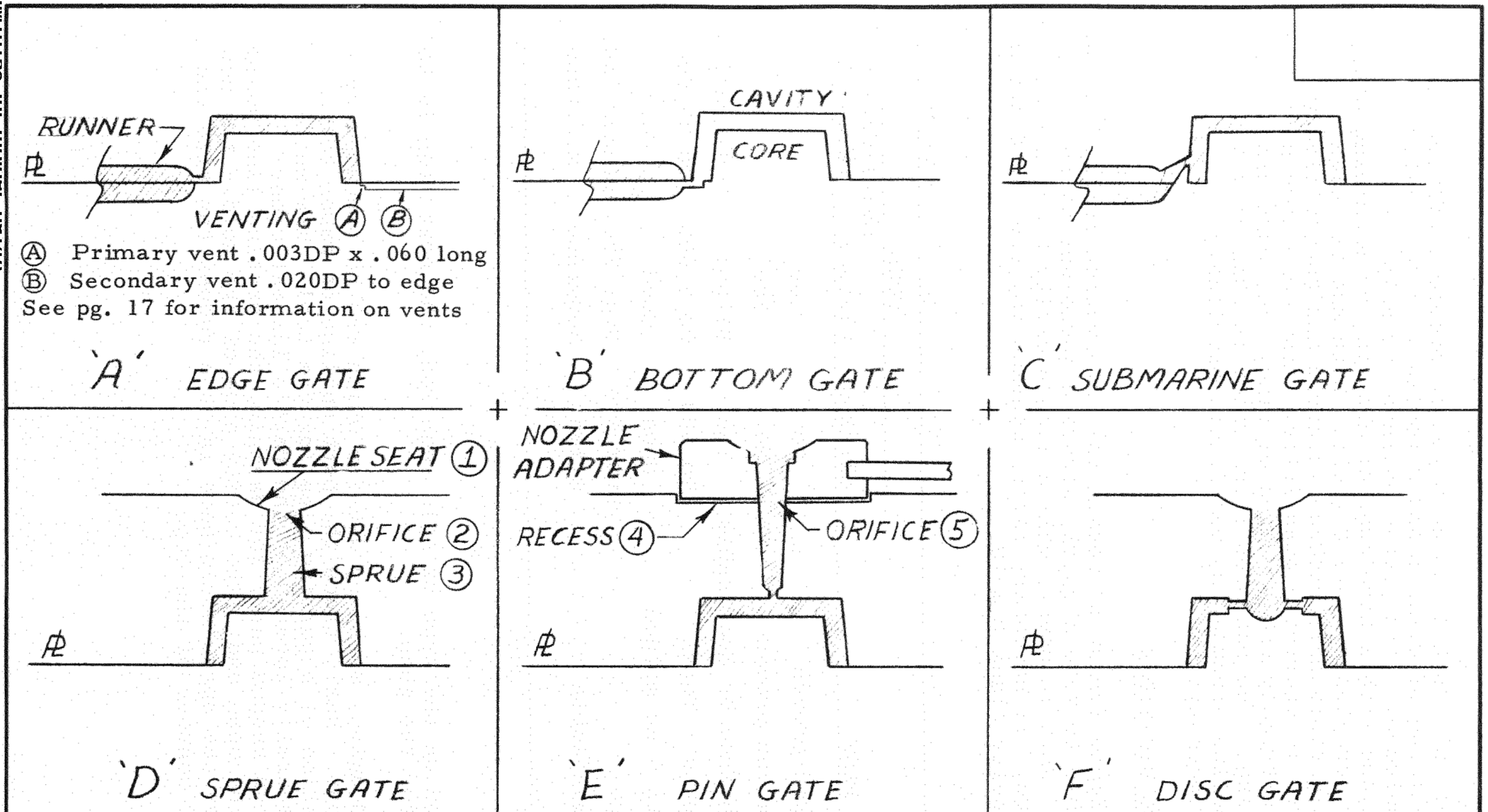
By using a simple format to notate the information relative to a part, it is possible to make a quick, accurate judgment of the required tooling. When in doubt, make a single cavity Class C tool and gain experience. If necessary later, upgrade the tool to Class B or make a multi-cavity Class B mold.

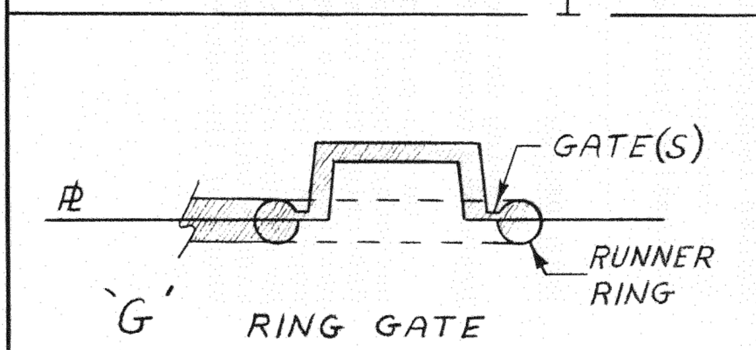
	TOOL		
PART	Class C	Class B	Multi-Cavity Class B
Part design firm and proven	No	Yes	Yes
Part material firm and proven	No	Yes	Yes
Post finishing required or acceptable	Yes	Slight	Minimal
Quantity requirements - Total	Too 500	To 5,000	To 25,000
Finish requirements - as molded	Open	Good	Good
Part cost allowance	Over 50¢	Over 25¢	Under 25¢
Toolmaking skill available	Slight	Moderate	Moderate

MOLD DESIGN

Having decided on the type of mold required, the next step is to prepare a design for the shop.

On the part drawing, mark any details which can be produced by secondary operations. Next, trace the part outline on paper to determine the location of a parting line for the tool. The main parting line is generally located at the part's largest cross-section. With a cup-shaped part, for example, the parting line would be at the rim of the open end. The cavity side of the mold - the hot side - which houses the sprue would have in it a recess the size and shape of the outer contour of the cup.





TYPICAL SPRUES, RUNNERS, GATES & VENTS

NOTES:

1. Nozzle Seat--1/2" radius x 1/8" deep
2. 7/32 Dia. for 'B' Nozzle & Anti-Drool Nozzle
   5/32 Dia. for 'A' Nozzle
3. Sprue Taper--3° to 5°
4. Nozzle Adapter Recess--1.520 Dia. x 1/32 deep
5. Mold Orifice for Nozzle Adapter .230" Dia.

The core side - the base - would have a protrusion on it the shape and size of the inside of the cup adjusted for anticipated material shrinkage. When placed together, the mold halves would be in contact all around the parting line and have a gap between the core and cavity equal to the wall thickness of the cup.

Refer to the illustrated examples of mold design in Chapter V. It is most probable that an idea illustrated, or a combination of ideas from one or more of these examples will provide you with a basic design to follow. The mold should be drawn so that each part is rendered to a simple machined block, manufacturable by an available machinist. To eliminate the necessity for many tight tolerance dimensions, show also an assembly picture, indicating only the basic outside sizes of the mold and the required fits (e.g. drive fit dowel or .001" oversize ream for a slip-fit or .005" clearance for a free fit).

Whenever possible, use standard sizes for the materials from which the mold is to be made. Quick reference to a material stock list will provide a wide assortment of materials and sizes. A mold must be designed so as to withstand pressure from clamping action of the machine and also pressure from within during injection. These pressures can be quite high. When designing aluminum molds, it is essential that a wall of at least 1 inch of metal remain outside the cavity. For a 1 inch diameter cavity, use at least a 3 inch diameter block. When milled cavities are of 1 inch depth, use at least a 2 inch plate. When semi-automatic tooling is to be built, use much larger material sizes so that the extra heat involved can be absorbed more evenly and be dissipated from the enlarged area. Additionally, the material has to be large enough to mount in the machine as well as accommodate ejector systems that may be built into the tool.

GATING

Two main factors influence gating: gate location and gate size. For symmetrically shaped parts such as, gears, discs, cams, and plates, the gate should be as near the center as possible. With a gear, for example, gate into the center hole or disc gate as shown on page 46. For sections which are cup-shaped, sprue gate as shown on page 12, or reverse sprue gate as on page 36. Avoid gating into a thin wall section. Gates should always allow material to flow into the heaviest section of a part. If necessary, make a heavy section on the part to gate into the machine away the excess material later. The function of a gate is to allow material as quickly as possible into the mold and produce even filling of the cavity with moderate injection pressure.

The material to be used can influence gating significantly. For vinyl, which flows freely when plasticized, a half-ounce part might typically require a .09" diameter gate. The same part from Noryl would need approximately twice the gate size.

* * *

To summarize the subject of tooling design, it is strongly recommended that these decisions be carefully planned around the following factors:

- ♦ Anticipated life of tool; quantity of parts required
- ♦ Selection of tool material
- ♦ Aesthetic requirements of part affecting selection of gating and parting line
- ♦ Critical dimensions of part; its relation to other parts in an assembly
- ♦ Material selection for part

TOOL BUILDING

There are a number of methods in use for manufacturing the parts of a mold. Besides various techniques for making the cores and cavities, there is a need to create gates, vents, sprues, and establish the proper fits, clearances, and drafts in the tool.

The core or insert, being a male form, is usually turned on a lathe or milled from plate or bar-stock material - or a combination of both. The cavity, however, being basically a recess, can present some problems. A round cavity is simple, and a rectangular cavity, providing it can have radiused corners, is easy to mill. However, a rectangular cavity with sharp corners all around is impossible to mill.

One method of producing a cavity with sharp corners is to machine mold inserts as shown on pages 42 and 90. Alternate, often more desirable methods of producing these sharp corners, as well as other complex configurations, include EDM'ing, hobbing and casting.

EDM (Electrical Discharge Machining)

EDM (Electrical Discharge Machining) operates on the principle of removing metal with electrical energy. An electrode having either a male or female shape designed to produce a corresponding cavity or core is positioned into the EDM machine. The electrode is then lowered into a work tank filled with dielectric fluid and to within a few thousandths of an inch of the work piece (mold plate). Electric arcs traveling across the gap then "cut" away the cavity material.

The electrodes are commonly made of EDM graphite, brass, or other conductive alloys such as copper, silver, and tungsten. The electrodes must be dimensioned to compensate for the overburn (spark gap) which occurs during the EDM process and also to allow for the plastic shrink factor. For very complex mold cavities or where several of the same electrode are required, electrodes can be purchased from companies which specialize in producing molded electrodes (usually copper tungsten) from models of the finished parts.

Any conductive material or metal - tough, hard, soft, brittle - can be EDM'd. Many conventional machining applications are now being replaced by EDM because it is more accurate, more reliable, or more economical. Also configurations that were thought to be impossible or very difficult to machine a few years ago are now being produced by EDM. Micro slots, burr-free holes, intricate dies, and contours make excellent EDM projects. The recessed contours and square corners as the example in page 96 is an ideal EDM application. For difficult shapes in carbide, exotic metals, hard or soft tool steel, aluminum, or other conductive materials, nothing matches the economics of electrical discharge machining.

Hobbing

To produce this cavity, it is necessary to produce a male form in steel with the shape of the desired cavity (see page 36). The hob was lathe turned to the required diameter, then milled to produce the external ribbing. The hob of oil hardening drill rod was then hardened by heating and quenching in oil. Next the cavity was prepared for hobbing by being lathe turned to the size of the hob at the bottom of the ribs. Finally, after being generously coated with white lead - an effective non-galling lubricant - the hob was forced into the cavity in a hydraulic press. It could equally well have been impacted into the cavity. It is possible to hob cavities in aluminum, kirksite, and soft low-carbon steel.

It is normal practice to hob a cavity to an over-depth condition. If a cavity must be finished to .500" depth, the hob should be forced into a depth of .530" and then the parting line of the cavity should be machined or surface ground down .030".

Casting

This same over-depth requirement exists when a cavity is to be cast. First, a model is made of the required part with the dimensions suitably increased to allow for shrinkage. From the model, a pair of epoxy or silicone rubber molds are cast. From these molds, plasters are cast and thoroughly dried by baking at 250°F for some hours. Finally, aluminum or kirksite is poured into the plasters and the resulting cores and cavities will be a fine detailed reproduction - in reverse - of the original model. This method of producing molds is used when the article to be reproduced is difficult to duplicate by machining as for example a sculpture or carving.

Please refer to the chapter on Epoxy Tooling for a complete and detailed description of low-cost casting techniques.

Fits and Clearances

When plastic is forced into a mold, considerable pressure is involved. The material will flow into all fine corners, edges, and cracks. Any space of .003" and greater will tend to become filled. If a clearance of .005" is allowed between a recess and the insert in it, the space will fill with material.

It is essential in producing a good mold to carefully size and fit all internal parts. For slide-fit dowels or ejector pins, use a reamer that is size plus .001" or .002".

Draft

When machining a cavity or core, always use a tapered mill cutter or lathe turn with the compound slide set at an angle. The cavity must decrease in size as it moves away from the parting line. A core must decrease in size from its base toward its highest point. The draft in a cavity must be at least 1/2° per side; the core should be drafted a minimum of 1° per side. Remember, the part will shrink away from the cavity and onto a core or insert (see Appendix I for Draft Angle factors).

Location of Mold Sections

In order that the desired accuracy be maintained in a part, it is necessary to locate the mold sections relative to each other. In lathe turned molds, location diameters can be machined during the making of the tool (see page 54). Molds made from square or rectangular plates are aligned by doweling the two matching pieces. See page 90 for an example of Leader Pins that locate the 'A' plate with the 'B' plate. The drilled and reamed holes should be a close fit to assure proper alignment and prevent any binding. Next, finish the side walls of the blocks to establish good edges that can be used for marking out the cavity. This preparation is a necessary step and will subsequently result in more accurate cavities and finished parts.

Any of the following operations will aid in relocating the mold halves: use different size dowel pins, stamp alignment marks, or mill flat one corner (see example on page 66).

Gates

The gate should be cut only after the cavity and core are completed. Start small and increase the size until no signs of knit-lines remain in the molded part. A large gate will produce a better quality part than a gate which is too small. Refer to page 12 for examples of various gate designs for different molding purposes.

Sprue

The size of the sprue at the nozzle end must be larger than the size of the nozzle orifice. Typically for a 3/16" diameter orifice nozzle (B size) the sprue orifice would be 7/32". The next most important item is the finish of the sprue hole itself. It must be smooth, free of chatter or scratches. Practice has shown that it is best to finish the hole with a straight flute hand reamer. Most local industrial tool suppliers offer tapered reamers in various sizes suitable for sprues.

Vents

One of the most important features of a tool is the vent system. Air must be allowed to escape as the material enters the cavity. Vents should be cut at the parting line in various locations around the cavity. See page 12 example 'A' and page 90 for vent design and dimensions. Occasionally it is necessary to vent a large cavity at locations other than the parting line. Core pins and inserts can be designed to provide this added venting.

If during sampling, it is found necessary to increase the gate size, it should also follow that the venting may have to be increased. The width and number of vents is determined by the size of the cavity and the rate of fill. Over-venting is rarely a problem since the resulting flash on the molded part can easily be trimmed; underventing is a common fault.

Tool Size

The size of the part will regulate to some extent the size of the finished tool. The mold surface at the parting line should be at least one inch larger all around than the part size. The plate thickness should be no less than one inch greater than the maximum cavity depth.

When mold halves are to be bolted together, use 1/2"-13 bolts. Small bolts tend to shear or have the threads strip. Bolts should be positioned symmetrically around the sprue, three for a round mold, four for a rectangular mold.

TOOLMAKING MATERIAL

Aluminum

The most widely used material for prototyping and short-run tooling is aluminum. It is readily available, lighter to handle than most other metals, and machines easily. It will take and hold a high polish, and the harder grades wear well. It is resistant to corrosion by most molding materials and the fumes that they may generate. Aluminum is not highly resistant to erosive or abrasive effects; however, it will easily take hard chrome plating or hard anodizing as surface finishes. These finishes are necessary when parts are needed in such materials as glass-filled nylons which are abrasive. Aluminum grades typically used for tooling are 2024T4 or 7075T6.

Beryllium

This is a copper-based alloy with a hard, low-friction surface. It will machine and polish well and is highly resistant to abrasion. Because of its high cost, it is used mainly for inserts or cores where aluminum does not have sufficient strength. It requires no heat treatment. Some molding compounds, especially those with chlorine content such as vinyl, may cause rapid corrosion to beryllium. Beryllium may also require special handling during machining due to fumes emissions.

Kirksite

This is a low-melting point, non-porous alloy, rather soft and free cutting. It will cast easily, capturing fine detail and so is usually used to produce molds of sculptured or detailed parts. It may be cast into well-dried plaster molds. The resulting castings are used as molds for plastic molding. Mold life may be extremely short, often well below 100 pieces; however, the mold can be easily made without intricate machining.

Steel

When strength, resistance to wear, and resistance to accidental damage are factors, steel is an obvious choice of mold making material. Tool steel is easy to machine, readily available, inexpensive, and can be moderately case-hardened at a minimal cost. Sprue bushings, cores, inserts, and thread forms can be made easily in tool steel and then case-hardened if necessary. To avoid the problems of heat treatment, tool steels or drill rod are highly recommended. These materials, although partially hardened, can be machined and polished and are favorites for sliding cores and loose inserts. Large hand operated molds should not be made out of steel due to the weight. A steel plate that is 5" x 6" x 2 1/4" weighs 20 pounds. A piece of aluminum the same size is only 7 pounds. In many applications it is better to have an aluminum mold base with steel inserts for ease of handling.

Other often used materials come in the form of ready-made components. They include hardened steel dowels used for location pins and cores, and ejector pins, which are available in many sizes, may be used as core pins.

MOLD FINISHING

Every mold, core, or cavity requires some finishing. First and foremost among the finishing operations is polishing. The idea here is to have a smooth surface as well as high lustre on the mold working surfaces. First, all machining marks must be rubbed away, and all undulations or ripples in the metal surface removed. Only then will polishing be effective. The tools used are as follows:

> For removing rough machining marks, lightly sand with emery or carborundum cloth. Next, rub with a rough polishing compound (approximately 280 grit) on balsa or hard wood sticks. Felt pads used in motorized hand tools can also be used with compound. For a finer finish, use a 600 grit compound finishing with a metal polish on cloth. Remember the molded part appearance will be no better than the surface finish of the mold cavity and core. Polishing may account for 10 to 15 percent of the total mold making time.

Protective coatings such as chrome or nickel plating, hard anodizing, or case hardening should only be applied after thorough polishing. Likewise, surface texturing, for example, and blasting to remove lustre, or photo-etching to produce a pattern, must be applied over a smooth surface to achieve the desired quality.

CHAPTER THREE

PART DESIGN AND MATERIAL SELECTION

When designing a part for injection molding, the method of toolmaking becomes an important consideration. It is the designer's responsibility to consider the alternatives and produce a design which is moldable using conventionally made toolings.

First, break down a part into simple components. Two or three simple pieces are often easier to tool than one complex part. The parts can be assembled easily together using snap-fits, solvent bonding, ultrasonic welding, as well as conventional use of rivets, eyelets, screws, etc. When quantities are not high, the assembly labor will be more than justified by the savings in tooling cost.

The most significant rules in part designing are keep the wall sections constant, and use adequate draft on all vertical surfaces. Indicate on the part drawing the location of the parting line. Show clearly the direction and angle of draft on the inside and outside faces. If a situation calls for "no draft" on a part, have extra material on the part and mold it with draft. Use secondary operations to remove the excess material and generate the parallel surfaces.

Wall thicknesses on molded parts are typically from .03" to .100". Excessive pressures are needed to push material into cavities with less than a .03" wall. Sections over .200" will tend to cool too slowly and stresses and sink marks will result.

It is bad practice to have extremes of thickness or localized heavy sections on a plastic part. Differences in cooling and chilling rates will invariably cause warpage in the finished item. When a heavy section must exist, use it as filling point for the cavity. Gate into a heavy section if possible; try never to fill a heavy section from a thin section.

The best possible source of information for the plastic parts designer is to study ready-made parts in the office, stores, and home. Examine the parting lines, drafts, gates, ejectors, surface finish, and try to judge the material. Probably the most critical decisions facing the designer are establishing acceptable tolerances and selecting the material. To a great extent one will affect the other, and both can significantly affect costs.

TOLERANCES

The most common error made by designers using plastics is to call out tolerances normally found in metalworking. The skilled toolmaker can cut cavities and cores in metal as accurately as any other fine machinist; but as in other industries, close tolerances cost money. Once the mold is made, the parts coming from it will not all necessarily be the same size.

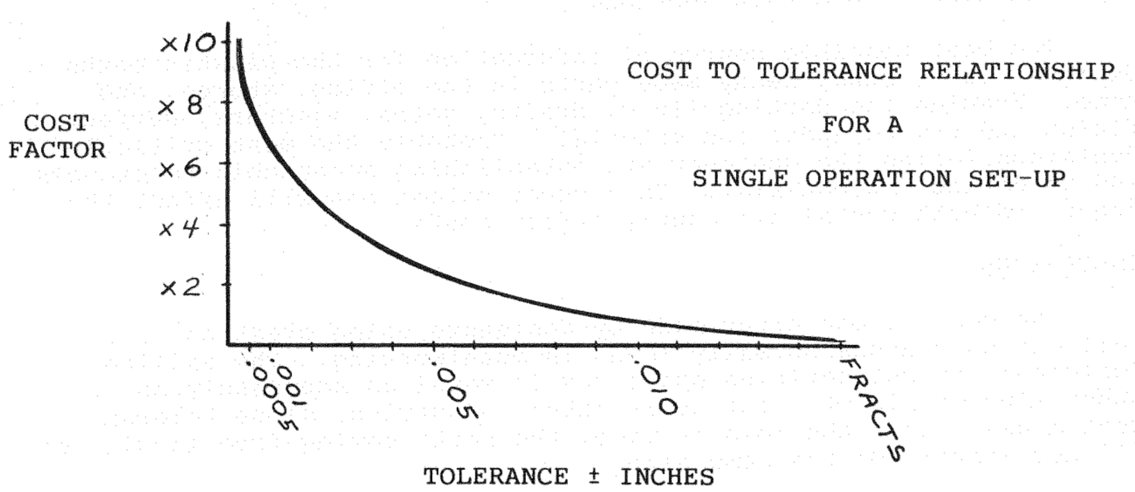
All plastic materials will shrink when cooled from a plasticized semi-fluid to a chilled part. The shrinkage will depend on the mold temperature and the material temperature change.

If a cavity is cut exactly four inches long at 70°F, then is heated during the molding process to 200°F , it will increase in length and so will parts molded in it. If the mold expands .005" in length, the minimum part variation will be greater than this. If, in the same mold, a part is shot at 500°F and another part is molded from material plasticized at 600°F, the parts will vary in length due to shrinkage factors. Indeed, parts will actually shrink on a three dimensional basis thus creating even greater potential for tolerance variation.

A third factor is the variation in material shrinkage even when the plasticizing and mold temperature can be held constant. These changes can be the result of ambient temperature or humidity fluctuations, or of changes in injection pressure and speed.

When a machinist is to cut a cavity, he must estimate how much oversize he must machine to allow for shrinkage. This shrink allowance can vary from .002 to .004 inch per inch for materials such as polystyrene up to .020 to .025 inch per inch for different grades of acetals. It is not reasonable to demand a +.000 -.005 tolerance on an inch since this leaves no latitude at all for the toolmaker or molder. Close tolerances can be achieved in plastic but only if mold dimensions can be adjusted after the mold is sampled. This way actual shrinkages are established and the process variations are held to tight limits. (See Appendix II for machining shrinkage factors.)

Many studies have been run to establish the relationships between tolerance and cost. The results of one such study are graphically displayed below. Plastics are almost always used to reduce manufacturing costs in a product. Only careful designing and critical study of actual tolerance requirements can maximize these savings.



THERMOPLASTIC MATERIALS

The physical properties of injection moldable materials vary from the soft flexibility of gum rubber to the brittleness of glass; temperature resistance varies from complete softening and dissolving in hot water to an ability to withstand a brief exposure to flame. Costs vary from a few cents to several dollars a pound. (See Appendix III for physical properties and cost per pound.) The decision as to which material to use for an application need not be a difficult one since over ninety percent of thermoplastic parts used are made from no more than a dozen basic materials. Often a part can be successfully made from any one of a number of available materials. Listed below are the most commonly used thermoplastics with a brief description of their most notable characteristics and uses.

A.B.S.

A.B.S. is a copolymer - a mixture of acrylic (which gives it a high lustre and hard surface), butadiene (a rubber which adds impact strength), and styrene (a rigid, low-cost base). Telephones, typewriters, instrument cases, and household appliances are made from A.B.S. It colors well, resists staining, and solvent bonds easily. Low cost.

Acetal

Parts which must maintain a springiness such as latches and snap catches are usually made of acetal since it strives to retain its molded shape. A low co-efficient of friction, good wear resistance, and rigidity make it excellent for bearings and gears. It has excellent resistance to chemicals. Medium cost.

Acrylic

This material has good optical clarity and is available in brilliant, transparent colors that are stable against discoloration. Its glossy surface and high-impact strength is resistant to outdoor weathering. These characteristics plus excellent electrical and heat properties make this material widely used for light control lenses. Medium cost.

Nylon

As any ardent fisherman will know, nylon is a tough material with a high resistance to abrasion. It tends to be self-lubricating and is used extensively for bearings, gears, and wear surfaces. Nylon is resistant to common solvents. Molding is made more difficult by the material's narrow processing temperature range and its extremely high moisture absorption rate. Medium cost.

Phenylene Oxide

This is a high-temperature resistant material used for electrical components such as switch housings and junction boxes. It is hard, wears well, and will solvent bond. High cost.

Polycarbonate

This material is extremely tough. Impact resistance is its real asset although clarity and surface lustre make it a favorite for appliances such as electric drill housings and handles for kitchen appliances. Windows and lenses are often molded of polycarbonate particularly for non-breaking uses. Polycarbonate will solvent-bond readily. High cost.

Polyester

Polyester is dimensionally stable and has low moisture absorption. It has excellent resistance to a broad range of chemicals such as gasoline, alcohols, and dilute acids and bases. Many parts requiring impact strength, smooth surfaces, and good wear properties are made of polyester. Medium cost.

Polyethylene

Many food containers used in the home are polyethylene. So are many toys and utensils. The material will float on water, has a waxy feel, scratches easily, and resists most solvents, chemicals, and detergents. It molds easily at low temperature, has high dielectric strength. Low cost.

Polypropylene

This is the lowest density common plastic. It floats easily, can be sterilized by boiling, has a high lustre surface, molds easily, and is fast replacing other plastics in the electronics industry as a component material. It will not absorb water, and it resists most chemicals and solvents. It is flexible, but has poor wear resistance. Hinges molded in polypropylene will frequently exceed one million flexings without failure. Low cost.

Polysulfone

Polysulfone is strong, rigid, and has a very high heat-deflection temperature along with excellent electrical properties. It is toxicologically inert, resistant to stain, taste, odor, and a wide variety of chemical compounds found in food and medical environments. High cost.

Styrene

Styrene varies from a crystal clear, brittle material used for cosmetic containers and transparent boxes for low-cost jewelry items to flexible, high impact colored plastic used for toys and low cost containers. It has low resistance to chemicals and heat, will solvent bond easily. Low cost.

Urethane

The wear resistance of this material is often many times that of rubber compounds, and it resists oils and many chemicals. Usually rubbery in feel and flexibility, its uses include noise-free, long-wearing gears and drive belts, electronic encapsulations, shoe heels and impact or shock pads. High cost.

Vinyl (PVC - Polyvinyl Chloride)

Most electrical wire coverings and plugs are made of vinyl. It is the most widely used encapsulation material for injected electronic components. It molds easily at low temperatures and will reproduce fine mold detail. Most vinyls are heavily plasticized by the addition of oil-like materials. It has poor temperature resistance and if over-heated, it discolors, can disintegrate, and emit chlorine gas. Vinyl can be solvent-bonded but has low chemical resistance. Low cost.

FILLERS AND MODIFIED MATERIALS

In order to improve or change the physical properties of thermoplastics, other non-thermoplastic materials are added to the granules by the materials manufacturers. Some of the most common fillers and their values are discussed briefly below:

Glass

Fibers of glass, similar to glass cloth, are mixed with many materials such as nylon, phenylene oxide, and polypropylene to increase the material strength. Percentages are from 10 to 40% of the filler. Both impact and heat resistance are greatly increased. Moldability is often improved by the glass because it acts as a good heat carrier, speeds plasticizing, and aids flow into thinner sections. Gate sizes may need to be increased especially with the higher fill percentages due to higher viscosity.

Glass beads are also being used as a bulk filler and cost reducer in some more expensive materials. Surface hardness is increased and shrink rates are considerably reduced. The glass particle sizes are extremely small so gate sizes are not affected. Moldabilty is often improved - especially with nylons.

Lubricants

For the production of parts subjected to high wear or abrasion, lubricants such as powdered teflon or molybdenum disulfide (MOS) are added in small quantities. Percentages would typically be from 2 to 5%: wear resistance is increased, and since the lubricant is impregnated into the part, external lubrication is often avoided.

Carbon Fibers

Thermoplastics that are fortified with carbon fibers have a strength and modulus value similar to die-cast alloys. The strength, lubricity and conductivity of carbon-filled materials has opened doors for plastic applications not previously possible.

Plasticizers

Some materials, such as vinyl, are naturally hard and brittle. To make such materials flexible and soft, oil-like chemicals are added in quantities from 5 to 30%. A normal plasticizer is D.O.P. di(2-ethylhexyl) phthalate which will attack some thermoplastic materials. Styrene should never be used in close contact with vinyls containing plasticizers unless considerable product testing is first done to evaluate any possible problems.

Other fillers commonly used include chalk, wood flour, titanium dioxide, ceramics, and metallic powders. The material suppliers' specification sheets will give full details on fillers and material properties.

* * *

Materials such as polypropylene and polyethylene are known as homopolymers since each is a pure polymer. When these materials are mixed by the materials manufacturer during processing, the resulting blend is known as copolymer. This term denotes only that the material is a chemical blend and has physical properties resulting from both parent materials.

The material manufacturer can modify his materials' physical properties by changes within his process. Such changes are known as cross-linking. This phrase "cross-linked" will be noted in the manufacturers' specification sheets and is usually associated with considerable increases in the physical strength of a material. Most polymers consist of molecular chains (long series of chemical groups joined end to end). These groups tend to align during processing like short lengths of chain piled in a box. The strength of the material is related to the length and strengths of these chains. Cross-linking, as the name implies, involves adding extra links between each chain and its neighbors. This process has gained importance and specification sheets should be studied for resulting changes in physical and molding conditions.

Physical properties and molding conditions of many commonly used materials are listed in Appendix III. This appendix is only a guide. Space prevents a complete, comprehensive list of materials. It is highly recommended to refer to the material manufacturer's information data sheet before processing any thermoplastic. Any person who works with plastic materials should be aware of any cautions, hazards, or special handling procedures that may be required.

CHAPTER FOUR:
EPOXY TOOLING

No study of low-cost tooling could be considered complete without a long, careful look at the advantages offered by poured epoxies. The cavity, and often the core of a mold, can be produced by pouring and heat curing a high temperature epoxy around a pattern or prototype part. When the pattern is removed, every detail, even finger marks on the pattern, will be faithfully reproduced.

ADVANTAGES

The advantages of epoxy tooling include the short time required to produce an extremely accurate and detailed mold, the low cost of materials, and the very significant savings in toolmaking labor. After some practice, good molds can be made by persons totally unskilled in normal moldmaking procedures. The equipment used - the mixing bowl, stirring stick, and simple oven - are far more readily available in the laboratory than the lathe, mill, and hobbing press.

LIMITATIONS

The limitations of epoxy tooling are involved with the need for an accurate pattern or prototype part to duplicate, and the life of the tool in terms of its ability to produce duplicate parts. The epoxy materials, although hard and rigid to the touch, are not as strong as aluminum or kirksite. The mold wear from abrasion is greater. Epoxies are generally strong in compression but poor in tensile or shear load conditions. It is necessary then to back-up the epoxy with a metal box and to avoid thin, unsupported members within a cavity or core.

An additional limitation involves the rate at which epoxy can transmit and dissipate the heat created in the process of injection molding. Simple two part epoxies are, in fact, good insulators. To overcome this problem, epoxies used in preparation of molds are heavily filled with powdered metal, usually aluminum. These fillers improve heat transmission and increase the strength and surface hardness of the materials. It is not uncommon, however to have long mold closed/injection cycles (often two to three times longer than those with conventional metals) due to the slow heat transfer rates.

TYPES

Mold making epoxies are of the two-part type. Part "A" is a metal-powder filled resin. Part "B", the hardener, is either a liquid or a powder. These are mixed thoroughly together usually in a ratio of three parts "A" to one part "B". Once mixed, the material has a workable life of about a half-hour before starting to gel. Three companies make epoxy materials used to produce injection mold cores and cavities. They are The Devcon Corporation, Furane Products, and Ren Plastics (a division of Ciba-Geigy Corporation).

All of these materials require heat curing. Once cured, the epoxies will withstand injection molding of thermoplastic materials at 600°F or less.

ECONOMICS

The costs of epoxies used for moldmaking are in the range of $18 to $20 per pound. The volume of one pound is approximately 16 cubic inches of mixture. When using a metal back-up shoe around the cast resin, it is seldom necessary to use more than half an inch thickness of epoxy. This means that when making a core and cavity for a part 3" x 3", the total epoxy used would be approximately 1/2 pound or $10 maximum cost.

The shoes or back-up boxes can be reused. Once a mold has completed its required production life, the epoxy can be removed and the shoes are then available for housing a new mold (or re-pouring the old one should it be damaged or wear out).

Experience has shown that the economic savings involved with epoxy tooling housed in reusable shoes can be as great as 70% of the cost of conventional tooling. These costs do not include the availability of metal working equipment required for conventional tool building.

MANUFACTURING OF EPOXY MOLDS

Design

There are two basic types of epoxy tools used for injection molding. These are the simple short-life hand mold (see page 92), and the Class "B" short-run tool, cast directly in the A & B plates of a conventional mold base (see page 94).

A simple drawing should be prepared for the mold, indicating the size of the pocket and the position of the parting line. A good practice is to draw the outline of the pattern first using a red pencil. Then draw the mold around the pattern using the parting line and the sprue positions to locate the mold plates. Two views, a plan and a side view, will be necessary to show the details and establish the pocket size. A good rule of thumb is to allow one inch of aluminum or a half inch steel plate around the epoxy as a back-up.

Pattern Making

The pattern can be made of any material which will withstand the first stage processing temperatures of approximately 200°F without deforming or giving off gas. Metal, plastic, and wood are most commonly used. Metal patterns should be free of bumps, machine marks, or rough surfaces. Plastic patterns should be of sufficient strength to allow for removal from the solidified epoxy after the first heat stage of curing (pre-cure). Wood patterns, which are easily constructed, are the most critical in terms of finishing requirements. All porosity must be eliminated and the wood grain smoothed, or the poured epoxy will adhere firmly to the pattern. It has been found from experience that the shellac-based varnishes, often used for wood pattern finishing in the foundry industry, are not suitable for use with epoxies. They tend to soften, become sticky, and often give off gas during pre-cure baking. The liquid polyethylene finishes and the vinyl emulsion wood glues, when brushed or rubbed well into the wood and thoroughly dried are more acceptable.

The final preparation of the pattern is a most important phase of epoxy moldmaking. First, rub or spray a hard wax-type parting agent on the pattern and allow this to dry. Then buff well and repeat the process. When you are absolutely sure that the pattern has been thoroughly coated with parting agent, wax it once more! Remember, epoxies are among the best known adhesives and will bond readily to most surfaces. They will not bond to a surface which is well waxed or well protected with silicone. Use one or both of these parting agents generously.

For critical dimensional areas, the pattern should be made larger than the required finished product by the amount dictated by the "shrink" of the thermoplastic material to be injected into the finished mold. If a part dimension must be one inch long when molded in ABS with a shrink rate of .005 inch per inch, then the pattern must be made 1.005 inches long. The shrinkage of the epoxy moldmaking material is very low. In sections of no more than a half-inch thick, it can be disregarded entirely.

Mounting the Pattern

Before the epoxy can be poured, the pattern must be properly mounted. When it has a flat surface at the parting line, such as a flat disc or cup-shaped item, the pattern can be easily secured to a flat surface such as a sheet of glass or metal using a layer of double-backed tape. The entire surface must now be waxed and buffed and can be lightly sprayed with silicone parting agent. Position over this the shoe or box and secure in place with weights or clamps.

When a part has detail on both sides of the parting line (a sphere or a cylinder), it must be mounted into a layer of soft clay with the lower portion submerged into the clay so that the other surface of the clay is at the parting line (imagine a marble pressed into a layer of clay until half submerged). The top surface of the clay should then be leveled and smoothed. Next, oven-dry the clay with the pattern in place. Plaster of paris may be used in place of clay and should be also thoroughly dried.

Next, thoroughly seal and wax the clay or plaster as described for wood preparation. Finally position the box or shoe over the pattern and secure in place.

The complete assembly should then be put on a heater plate or preferably in an oven and raised gently to 200°F.

Mixing the Epoxy

It is essential for the preparation of good epoxy mixes that two rules be strictly observed. First estimate the amount of epoxy required using the known fact that one pound equals 16 cubic inches. Then MEASURE CAREFULLY the CORRECT amount of hardener, stirring well with a tool. The mixture must now be THOROUGHLY MIXED by stirring slowly by hand for at least five minutes. Then return the mix to the oven.

Machine mixing is not advisable because it folds air into the mix, and air entrapment is the biggest cause of epoxy mold failure. Bubbles on or near the mold surface will rupture under molding pressure and cause parts to hang in the mold.

While the mix is reheating and bubbles are riding to the top, the heated mold half can be removed from the oven and positioned for pouring. Remember that as epoxy is poured into the molding box, air must be allowed to escape. It is good practice to put the mold and plate assembly on a heater plate and tilt it slightly so that the epoxy can be poured at the low end of the mold. The level of the liquid epoxy thus rises slowly in the mold avoiding air entrapment. When full, the mold can be returned to the oven for pre-cure. If possible, leave the filled mold in the oven at 200°F overnight.

When fully pre-cured, remove the flat plate from the parting line of the mold. Leave the pattern in the mold or replace it into the molded cavity. Once again, thoroughly wax, buff, rewax, and spray the mold parting line and underside of the pattern. Place the lower mold shoe in position using the dowels to locate it accurately. Bolt or clamp the mold halves together.

Mix, heat, and pour in the lower mold shoe as per the upper half of the mold and pre-cure in the oven. Next remove the pattern and complete the mold cure as recommended in the epoxy manufacturer's instructions.

Mold Finishing

In many cases, the only machining required to complete the mold is the drilling and the hand reaming of the sprue passage. As for all metal molds, the sprue must be tapered. Wherever possible, use a straight flute pin reamer.

Runners should be cut where necessary using a ball end mill cutter. There should be no need to polish an epoxy mold if the pattern was correctly finished; however in areas where the epoxy is machined a good polish can be obtained with the use of a lightly abrasive metal polish applied with a soft wood polishing stick or a cotton-tipped swab. After polishing, apply a coating of hard wax and buff well.

If an abrasive plastic material such as a "glass filled" is to be injected into the mold, it is possible to Teflon® coat the core and cavity. First remove all traces of wax from the epoxy using a solvent or a degreaser such as carbon tetrachloride; then spray or paint on a thin coating of Teflon "S" and air dry. Next bake the mold at 350°F for two hours to harden the Teflon.

(Note: Teflon is a registered tradename of the DuPont Company. Teflon is a PTFE compound. PTFE stands for polytetrafluoroethylene which is a class of fluoroplastic. PTFE is sold under other tradenames such as Halon by Ausimont, USA; Fluon by ICI Americas; and Hostaflon by Hoechst Celanese.)

Metal core pins or inserts should be added to the mold after the final oven cure. DO NOT mold metal inserts, cores, or ejector pins into the epoxy. These will all complicate the moldmaking and tend to cause stresses and air entrapment. Add these details as is done with conventional all-metal tooling.

The remaining item is venting, which is extremely important in epoxy tooling, particularly since injection pressures should be kept as low as possible. Venting techniques described in the metal tooling chapter apply equally to epoxy tooling. Finally, it is recommended that a good mold release agent be sprayed into the mold cavity and core. This is preferable to a layer of well buffed hard wax.

RUNNING OF EPOXY MOLDS

In order to prepare and extend the life of epoxy tooling, care should be taken during its use to prevent accidental damage. First, gently warm the mold to its running temperature as required to suit the material to be injected. Clamp the mold firmly but slowly (without sudden impact). Inject the material at a moderate or slow rate without undue use of high pressures.

Because of slower-than-metal heat transfer rates, the chill times after the completion of injection and before mold opening should be extended by approximately a factor of two. A part which in all metal tooling would chill in 15 seconds should be allowed a full 30 seconds to chill (possibly longer). Adjust this "holding time", including maintaining the pressure of the injection ram, until acceptable parts and cycles are attained. If the temperature of the mold continues to rise, reduce the frequency of injecting parts into it. After a part is ejected, wait a short time before reusing the mold. Class B tools can also be cooled by circulating cold water through holes drilled in the shoes or mold plates surrounding the epoxy.

MOLD LIFE

The complexity of the part being shot, the care taken in moldmaking, handling and use, and the type of material being processed all affect the life of an epoxy tool. One hundred parts is a normal expected life of a tool maintaining close tolerances and using high temperature materials.

Some epoxy molds can produce at least a thousand parts if carefully used with non-abrasive plastic materials. There are instances on record of epoxy tools producing over 50,000 shots. If the pattern is retained after completing the mold, a worn or damaged mold can be remade using the shoes or mold plates a second or third time. Often only one side of a mold need be rebuilt. Simply place the pattern into the good half of the mold and recast the worn section.

MOLD REPAIR

Repairing an epoxy mold is usually a simple task. Machine or scrape away an area around the damage. Drill a hole through the mold to the back if necessary. Carefully wax a molded part or the original pattern and press it firmly into the mold. Heat to remove the pattern, and complete the cure as recommended. Rewax the mold and continue with production.

In summary, epoxy tooling has been found to be an excellent prototype alternate technique - particularly when dealing with complex shaped parts. In the absence of significant machine shop capability, it may be the best economical technique for producing useful and quality prototype parts. In some cases, it is the mold building technique of choice for making low volume production runs. Epoxy tooling is a significant viable avenue in the search for practical prototype and low production methods.

CHAPTER FIVE

CASE EXAMPLES OF SHORT-RUN MOLDING REQUIREMENTS

The following case examples are intended to illustrate tool design, construction, material selection, and the molding process. In most examples, short-run molding is compared with an alternative method of fabrication and costs of both approaches are estimated. These examples are based on actual case histories: however, where necessary, costs and lead times have been approximated. Also adjustment of hourly rates for inflation and local conditions may be required in some examples especially as the cost justifications have not been "rescaled" since their actual occurrence. As an average, the estimated cost savings at any point in time are in excess of 65 percent of the alternate method of fabrication, and the savings in lead times are equally significant. You will note that for many of these examples, the only possible manufacturing process was injection molding.

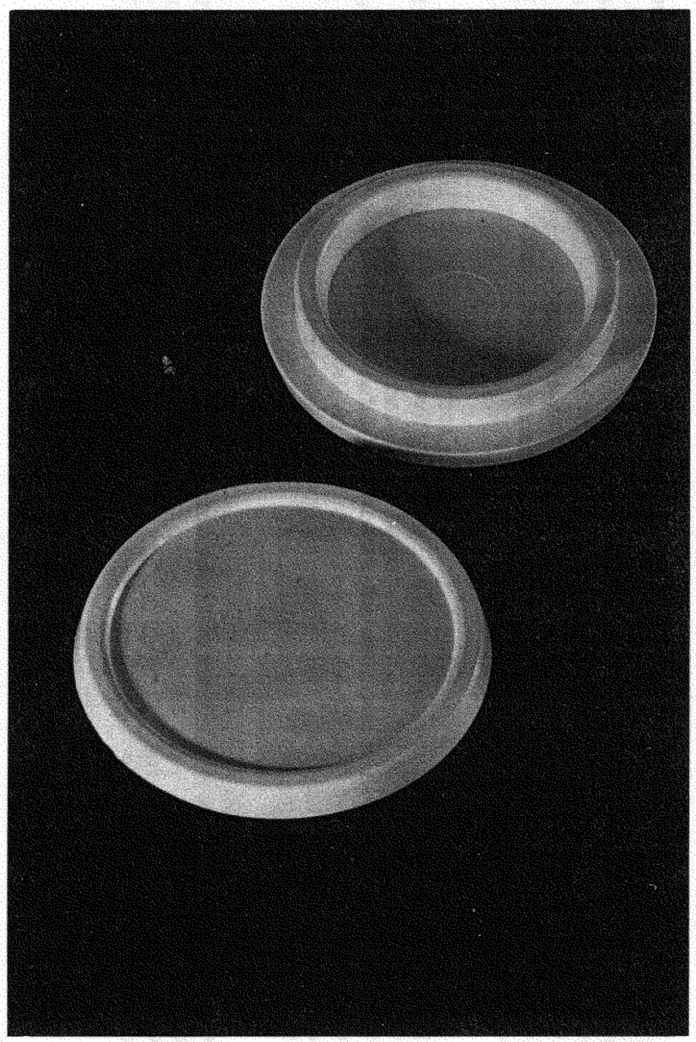
The selection has been grouped as follows:

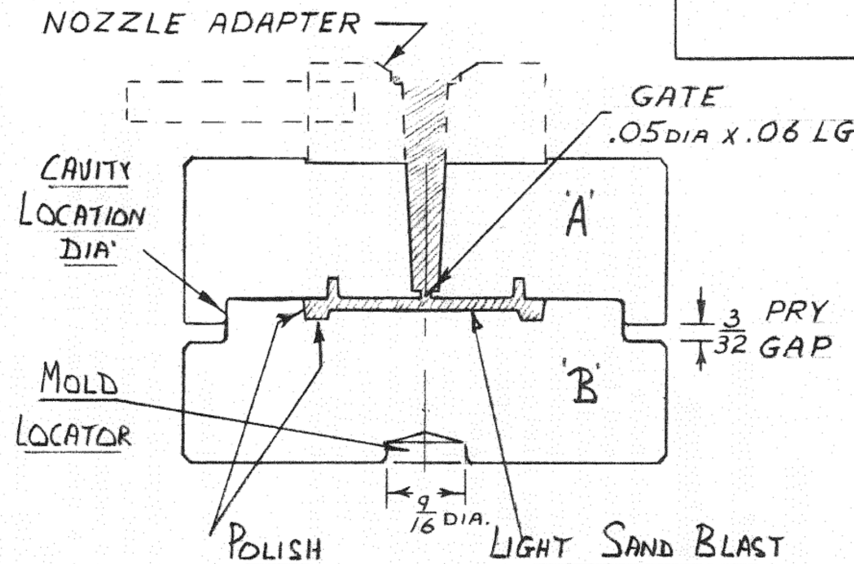
A Series	-	2 plate tools emphasizing lathe work
B Series	-	3 & 4 plate tools emphasizing lathe work
C Series	-	2 plate tools emphasizing mill work
D Series	-	multi-cavity & semi-automatic tools
EP Series	-	epoxy tooling
EDM Series	-	tools using EDM technique

Emphasis has been placed on hand molds and semi-automatic tools of the Class B designation. Some repetitive items such as mold mounting holes and slots, hidden lines and other common details have been deleted from some sketches in order to "highlight" other mold design features. Many of these could have been built as Class C tools, the parts finished with secondary operations, and the total economics of the job improved.

The indirect advantages of this short-run injection molding approach include: reduction of parts inventory as interim production requirement for parts can be completed and the tool returned to storage; flexibility of production scheduling due to low lead times for first samples and quick set-up for subsequent production runs; savings of purchasing and accounting overheads; and full control by production management over quality assurance.

If your applications are primarily in R&D and product development, the in-house injection molding approach enables you to maintain better control over proprietary information, promotes strong feedback between design and molding results, as well as rapid project turnaround at lower tooling costs.





(a) SHOOT THRU' NOZZLE ADAPTOR W/'B' BALL NOZZLE.

(b) REMOVE 'B' PLATE - ROTATE ADAPTOR TO BREAK GATE.

(c) LIFT OUT PART & TRIM.

2 PLATE MOLD - LATHE ONLY -		
SCALE: 1:1 DATE:	APPROVED BY	DRAWN BY R.M.
MOLD MAT'L: 3" O.D. C.R.S. 1015 OR ALUM' 2024-T4		
COVER-INDICATOR LIGHT	MAT'L: COLORED, PROPRIONATE	DRAWING NUMBER A-1

PART REQUIREMENT

DESCRIPTION: Cover/Indicator NUMBER: A-1

MATERIAL: Translucent - color tinted plastic

QUANTITY: 1,000 parts

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchased item, lead time 6 weeks 3 colors - varying color ratios Minimum purchase - 1,000 parts x 55¢ each	$550

SHORT RUN MOLDING METHOD:

Tool cost - 4 hours x $10/hr. + material	$ 55
Molding cost - 12 hours x $5/hr. + material	65
Lead time - 1 week	$120
Estimated savings: 5 weeks and	$430

ENGINEERING DETAIL

MATERIAL SELECTION: "Cellulose" acetate, butyrate or propionate

Reason: Optically good, translucent colors available, high-impact strength, flexible (for press fit to panel), dimensionally stable

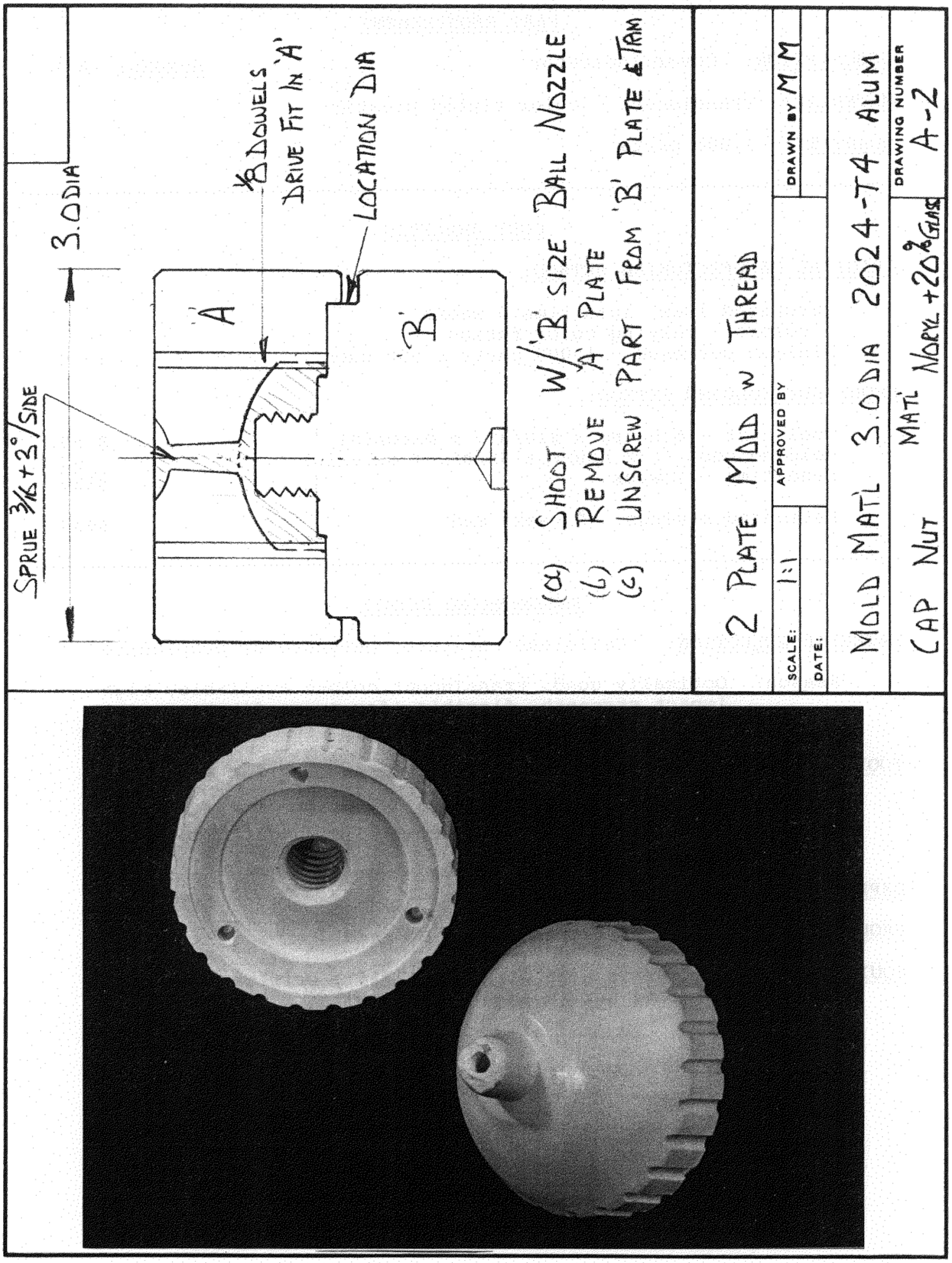
TOOL: "A" plate part cavity with small sprue gate for aesthetic reasons

"B" plate polished and sandblasted to give diffused light effect

EXPECTED TOOL LIFE: 5,000 parts

PRODUCTION RATE: 80 parts per hour - by hand

EQUIPMENT: Morgan-Press G-55 with "B" (3/16") orifice ball nose nozzle and nozzle adapter



SPRUE 3/16 + 3°/SIDE
3.0 DIA
1/8 DOWELS
DRIVE FIT IN 'A'
LOCATION DIA
'A'
'B'
(a) SHOOT W/'B' SIZE BALL NOZZLE
(b) REMOVE 'A' PLATE
(c) UNSCREW PART FROM 'B' PLATE & TRIM
2 PLATE MOLD W THREAD
SCALE: 1:1
DATE:
APPROVED BY
DRAWN BY M M
MOLD MAT'L 3.0 DIA 2024-T4 ALUM
CAP NUT
MAT'L NORYL +20% GLASS
DRAWING NUMBER
A-2

PART REQUIREMENT

DESCRIPTION: Cap Nut NUMBER: A-2

MATERIAL: Glass-filled Phenylene Oxide (Noryl*)

QUANTITY: 100 parts

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Machined from acetal (Delrin**) bar stock	
Cost to lathe and mill - 30 hours x $10/hr	$300

SHORT RUN MOLDING METHOD:

Tool cost - 6 hours x $10/hr + material	$ 65
Molding cost - 2 hours x $5/hr + material	12
Lead time - 2 days	$ 77
Estimated savings:	$223

ENGINEERING DETAIL

MATERIAL SELECTION: 20% Glass-filled Phenylene Oxide

Reason: High strength, low cold flow, light weight

TOOL: "A" plate is cavity and sprue gate

"B" plate is core and base

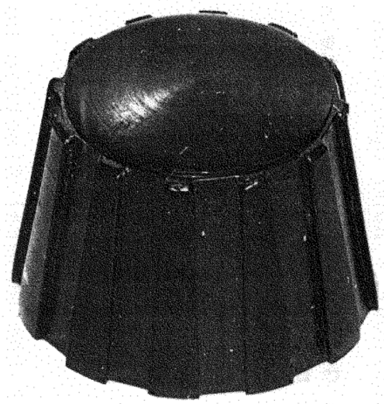
EXPECTED TOOL LIFE: 500 parts

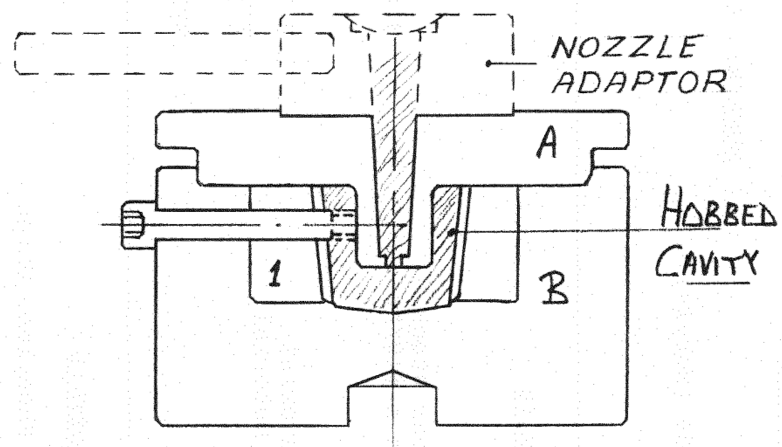
PRODUCTION RATE: 60 parts/hour - hand operation

EQUIPMENT: Morgan-Press G-55 with heater plate and "B" (3/16") orifice ball nozzle

*Noryl is a trade name of the General Electric Co.

**Delrin is a trade name of the DuPont co.





(a) SHOOT THRU' NOZZLE ADAPTOR W/ 'B' BALL NOZZLE
(b) UNSCREW & REMOVE ALLEN BOLT & GATE
(c) REMOVE 'B' PLATE - TWIST KNOB OFF 'A' PLATE.

2 PLATE MOLD - LATHE & HOB		
SCALE: 1:1	APPROVED BY	DRAWN BY R M
DATE:		
MOLD MATL. 2024 T4 ALUM HOB- OIL HD'N DRILL ROD		
TIMER KNOB W/SET SCREW HOLE MATL 'NORYL		DRAWING NUMBER A-3

PART REQUIREMENT

DESCRIPTION: Timer Knob with set screw NUMBER: A-3

MATERIAL: Noryl

QUANTITY: 200 parts

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Turn and mill from aluminum bar - anodize black	
Cost - 200 x $3.75 each	$750

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr + material	$135
Molding cost - 7 hours x $5/hr + material	40
Lead time - 3 days	$175
Estimated savings: 2 weeks and	$575

ENGINEERING DETAIL

MATERIAL SELECTION: Noryl, SE-100, black

Reason: High strength, low cold flow, high heat resistance, good surface finish

TOOL: "A" plate is core and reverse tapered sprue with gate.

"B" plate is base and locator for insert.

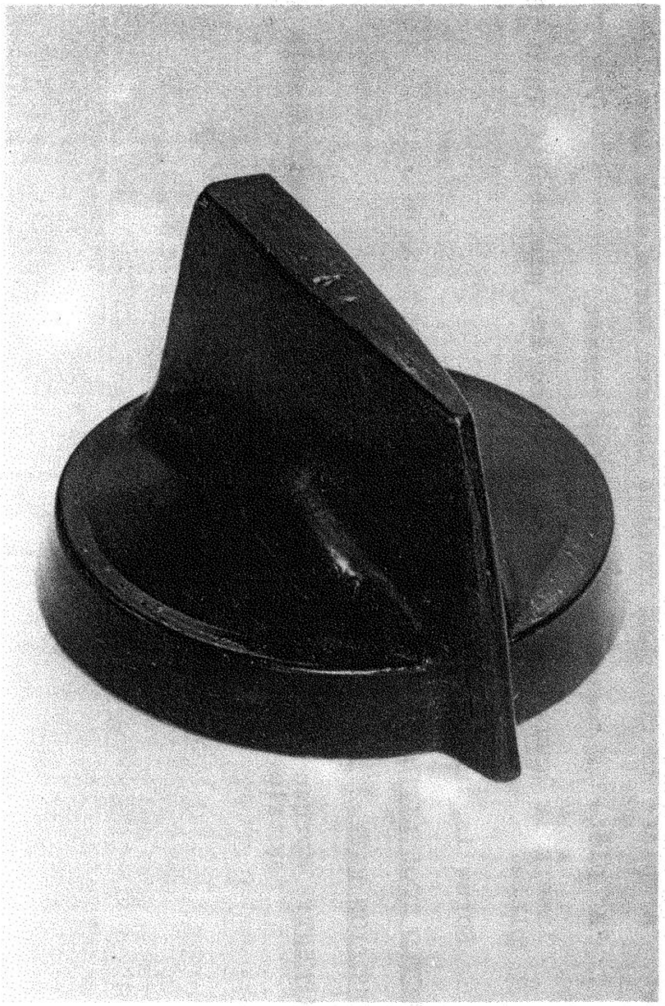
Insert is hobbed after lathe turning and pieces fit into "B" plate.

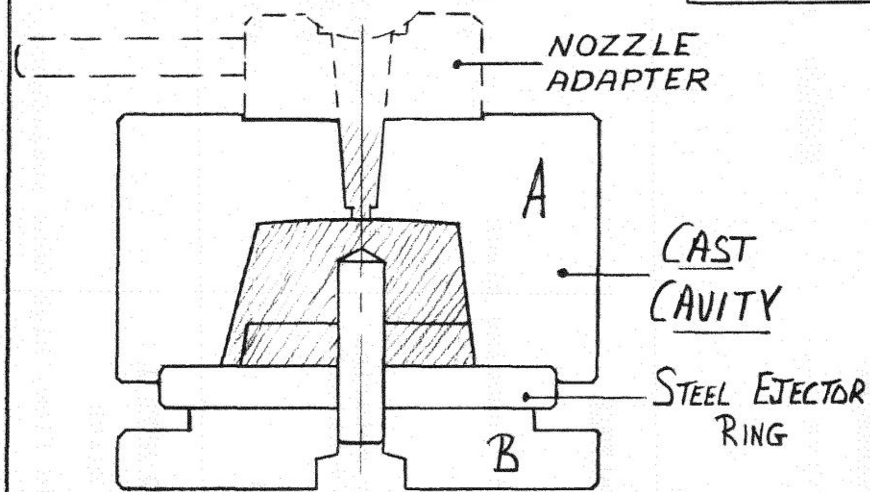
Screw is standard allen cap.

EXPECTED TOOL LIFE: 500 pieces

PRODUCTION RATE: 30 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-55 with heater plate, "B" ball nozzle and nozzle adapter





(a) SHOOT THRU' NOZZLE ADAPTOR USING SHUT-OFF NOZZLE

(b) ROTATE ADAPTOR TO BREAK GATE.

(c) REMOVE 'A' PLATE.

(d) RAISE EJECTOR RING TO REMOVE PART. TRIM.

2 PLATE MOLD - CAST CAVITY - STEEL EJECTOR RING		
SCALE: 1:1 DATE:	APPROVED BY	DRAWN BY PM
CAV'- 356 ALUM' CASTING - BASE: 2024-T4		
KNOB-CONTROL VALVE	MAT'L P. PROP.	DRAWING NUMBER A - 4

PART REQUIREMENT

DESCRIPTION: Knob, Control Valve (Custom) NUMBER: A-4

MATERIAL: Polypropylene - black

QUANTITY: 50 pieces

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchase a standard knob and modify to fit square shaft	
Cost - 50 x $5 (Estimated)	
Lead time - 3 weeks	$250

SHORT RUN MOLDING METHOD:

Tool cost - 10 hours x $10/hr + material	$125
Molding cost - 2 hours x $5/hr + material	12
Lead time - 3 days	$137
Estimated savings: 2 weeks and	$113

ENGINEERING DETAIL

MATERIAL SELECTION: Polypropylene - black

Reason: High-lustre finish, will press onto shaft without cracking, low cost and easy to mold

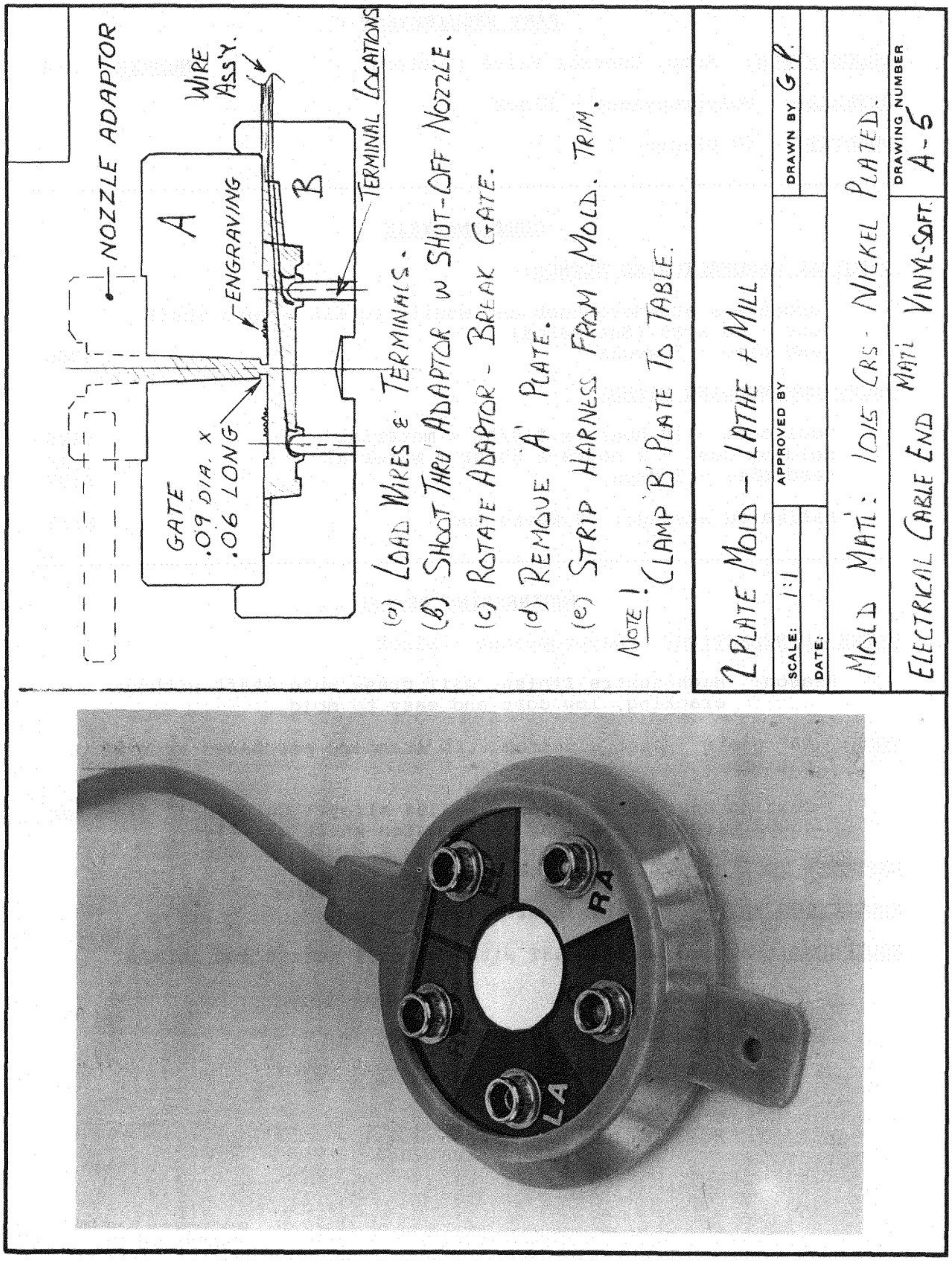
TOOL: "A" plate - cast aluminum with standard purchased knob as pattern

Casting cost is about $15 in #356 alloy. Core is $\frac{1}{4}$" diameter dowel with flats ground on to match shaft of valve.

EXPECTED TOOL LIFE: 500 pieces

PRODUCTION RATE: 40 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-55T with "B" ball nozzle and nozzle adapter


NOZZLE ADAPTOR
WIRE ASS'Y.
TERMINAL LOCATIONS
ENGRAVING
A
B
GATE
.09 DIA. X
.06 LONG
(a) LOAD WIRES & TERMINALS.
(b) SHOOT THRU ADAPTOR W SHUT-OFF NOZZLE.
(c) ROTATE ADAPTOR - BREAK GATE.
(d) REMOVE 'A' PLATE.
(e) STRIP HARNESS FROM MOLD. TRIM.
NOTE! CLAMP 'B' PLATE TO TABLE.
2 PLATE MOLD - LATHE + MILL
SCALE: 1:1
APPROVED BY
DRAWN BY G.P.
DATE:
MOLD MAT'L: 1015 CRS - NICKEL PLATED.
ELECTRICAL CABLE END
MAT'L VINYL-SOFT.
DRAWING NUMBER A-5
RA
LA

PART REQUIREMENT

DESCRIPTION: Electrical Cable End - 5 part wire NUMBER: A-5

MATERIAL: Soft Vinyl - grey

QUANTITY: 5,000 pieces/month

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchase from custom molder - no bidders

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr + material $135
Molding cost - 60 parts/hr x $5/hr - 8¢ each
Piece cost - 9¢ each

One operator x 60 shots/hr produces 2,400 parts a week

ENGINEERING DETAIL

MATERIAL SELECTION: Vinyl - soft, grey

Reason: Material is flexible, good electrical resistance, easy to mold, low cost, good surface finish

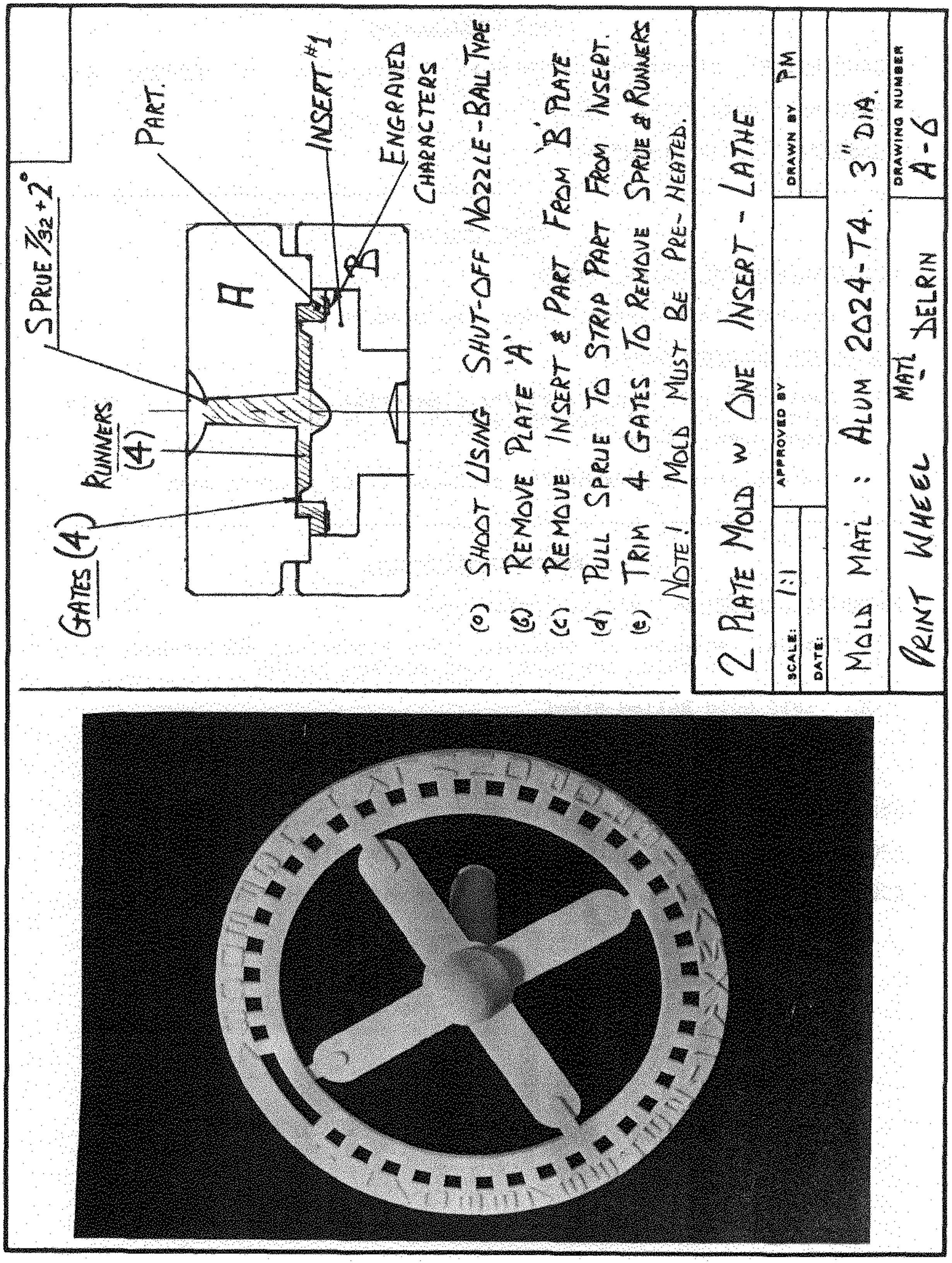
TOOL: 1015 Cold Rolled Steel

All steel - nickel plated for protection against corrosion from vinyl

EXPECTED TOOL LIFE: 25,000 pieces

PRODUCTION RATE: 60 parts per hour - hand cycle

EQUIPMENT: Morgan-Press G-55T with shut-off nozzle and nozzle adapter


GATES (4)
RUNNERS (4)
SPRUE 7/32 + 2°
PART.
INSERT #1
ENGRAVED CHARACTERS
A
B
(a) SHOOT USING SHUT-OFF NOZZLE - BALL TYPE
(b) REMOVE PLATE 'A'
(c) REMOVE INSERT & PART FROM 'B' PLATE
(d) PULL SPRUE TO STRIP PART FROM INSERT.
(e) TRIM 4 GATES TO REMOVE SPRUE & RUNNERS
NOTE! MOLD MUST BE PRE-HEATED.
2 PLATE MOLD W ONE INSERT - LATHE
SCALE: 1:1
APPROVED BY
DRAWN BY PM
DATE:
MOLD MAT'L : ALUM 2024-T4. 3" DIA.
PRINT WHEEL
MAT'L DELRIN
DRAWING NUMBER
A-6

PART REQUIREMENT

DESCRIPTION: Print Wheel NUMBER: A-6

MATERIAL: Delrin

QUANTITY: 25 prototype parts only

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchased turned and engraved aluminum rings	
Turning 25 x $3/each	$ 75
Engraving - 25 x $7.50 each (36 characters)	190
Lead time - 3 - 4 weeks	$ 265

SHORT RUN MOLDING METHOD:

Tool cost - 6 hours x $10/hr + $25 engraving	$ 85
Molding cost - 2 hours x $5/hr (set up and run)	10
Lead time - 2 days to first part	$ 90
Estimated savings: 3 weeks and	$ 170

ENGINEERING DETAIL

MATERIAL SELECTION: Delrin (Acetal)

Reason: Provides a hard surface, high strength, good surface detail

TOOL: All aluminum bar stock

"A" plate is sprue seat, sprue and half cavity.

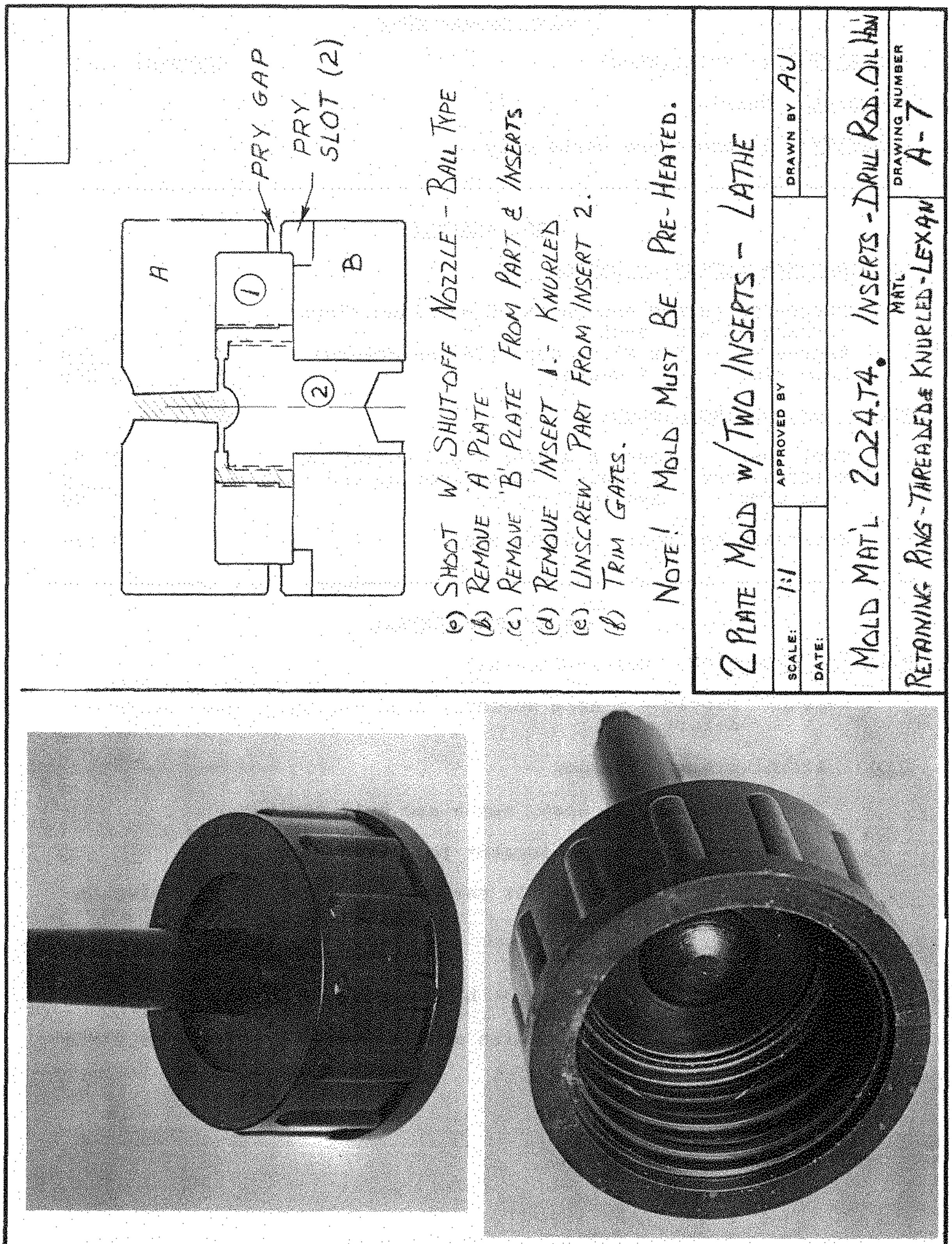
"B" plate is base and locator for insert #1.

Insert #1 is core and half cavity with reversed engraving on face.

EXPECTED TOOL LIFE: 100 pieces

PRODUCTION RATE: 50 pieces/hour - hand cycle

EQUIPMENT: Morgan-Press G-55T with shut-off nozzle and heater plate


PRY GAP
PRY SLOT (2)
A
B
1
2
(a) SHOOT W. SHUT-OFF NOZZLE - BALL TYPE
(b) REMOVE 'A' PLATE
(c) REMOVE 'B' PLATE FROM PART & INSERTS
(d) REMOVE INSERT 1.- KNURLED
(e) UNSCREW PART FROM INSERT 2.
(f) TRIM GATES.
NOTE! MOLD MUST BE PRE-HEATED.
2 PLATE MOLD w/TWO INSERTS - LATHE
SCALE: 1:1
DATE:
APPROVED BY
DRAWN BY AJ
MOLD MAT'L 2024.T4. INSERTS-DRILL ROD. OIL HDN
RETAINING RING-THREADED & KNURLED - MAT'L LEXAN
DRAWING NUMBER A-7

PART REQUIREMENT

DESCRIPTION: Retaining Ring - threaded and knurled NUMBER: A-7

MATERIAL: Lexan* - black

QUANTITY: 1,000 parts a year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchased screw machine parts from aluminum stock	
Lead time - 5 weeks	
1,000 parts x $1.25 each, plus $50.00 set up (single quantity discount purchase)	$1,300

SHORT RUN MOLDING METHOD:

Tool cost - 10 hours x $10/hr + material	$ 110
Molding cost - 50 hours x $5/hr + material	160
Lead time - Parts to be run as needed	$ 270
Estimated savings: 4 weeks and	$1,030

ENGINEERING DETAIL

MATERIAL SELECTION: Lexan - black (Polycarbonate)

Reason: Extreme strength and impact resistance, low cold flow, good physical appearance

TOOL: Aluminum 2024-T4 or better such as 7075-T6

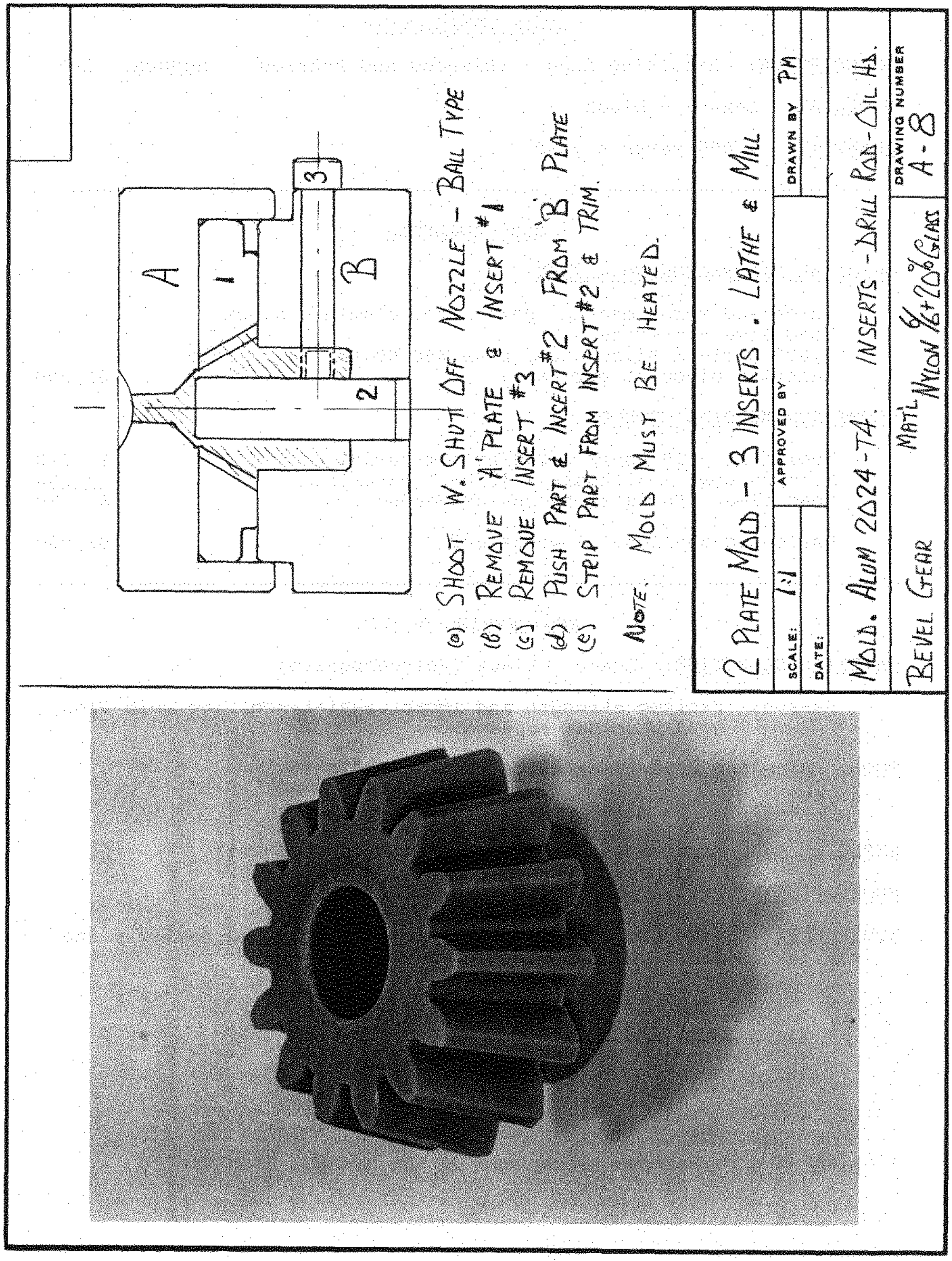
Inserts - O1 drill rod

EXPECTED TOOL LIFE: 5,000 pieces (with steel inserts)

PRODUCTION RATE: 30 pieces/hour - hand cycle

EQUIPMENT: Morgan-press G-55 with shut-off nozzle and heater plate

* Lexan is a registered trade name of the General Electric Co.


A
1
B
2
3
(a) SHOOT W. SHUT OFF NOZZLE - BALL TYPE
(b) REMOVE 'A' PLATE & INSERT #1
(c) REMOVE INSERT #3
(d) PUSH PART & INSERT #2 FROM 'B' PLATE
(e) STRIP PART FROM INSERT #2 & TRIM.
NOTE: MOLD MUST BE HEATED.
2 PLATE MOLD - 3 INSERTS . LATHE & MILL
SCALE: 1:1
DATE:
APPROVED BY
DRAWN BY PM
MOLD. ALUM 2024-T4. INSERTS - DRILL ROD - OIL H.
MAT'L NYLON 6/6 + 20% GLASS
BEVEL GEAR
DRAWING NUMBER A-8

PART REQUIREMENT

DESCRIPTION: Bevel Gear - Low Speed - Medium Load

NUMBER: A-8

MATERIAL: 20% Glass-filled Nylon 6/6

QUANTITY: 75 parts per year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchase steel pinion: drill and tab hub Cost - 75 x 4.95 plus labor	$450

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr + material	$125
Molding cost - 4 hours x $5/hr + material	25
Parts to be run as needed	$150
Estimated savings:	$300

ENGINEERING DETAIL

MATERIAL SELECTION: 20% Glass-filled Nylon 6/6

Reason: High impact, high load carrying factor, easy to mold, low shrinkage

TOOL: "A" plate is sprue seat with locator for insert #1.

Insert #1 is internal gear ring hobbed from metal gear.

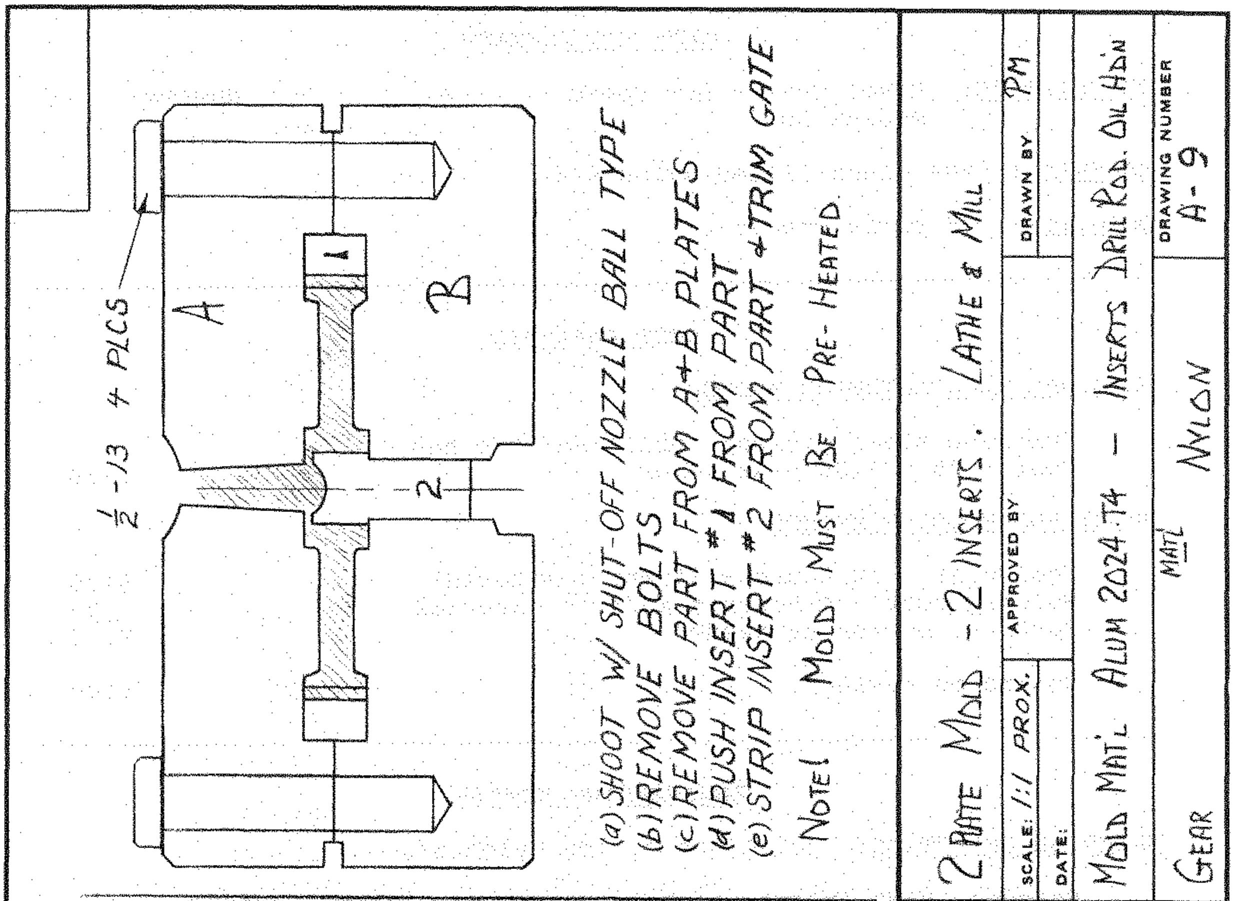
"B" plate is base-part cavity.

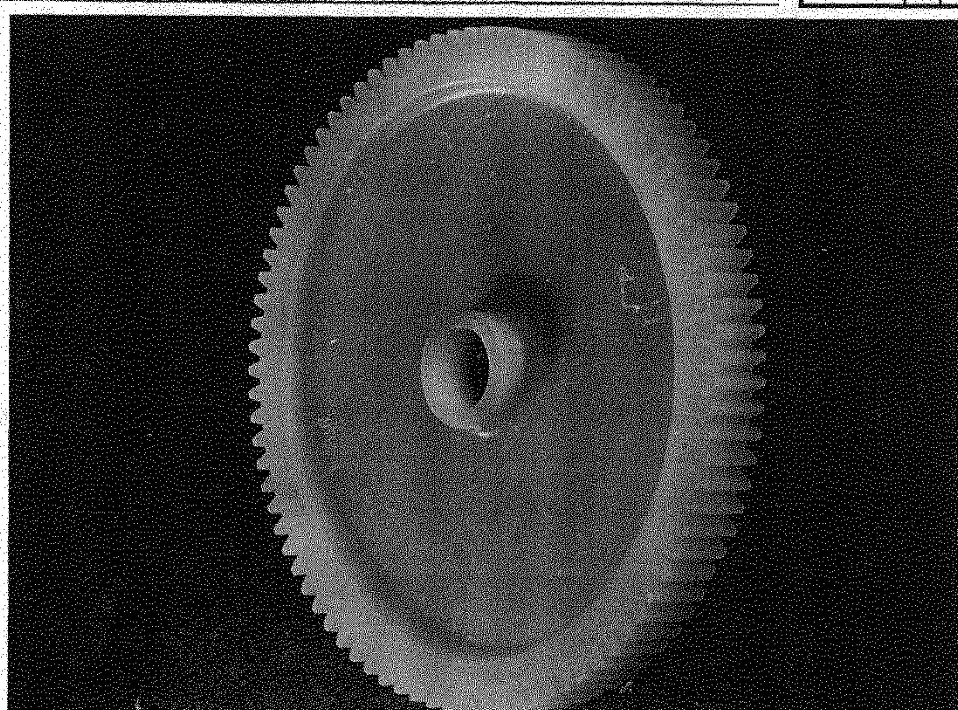
Insert #2 is ½" diameter hard dowel.

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 20 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-55 with shut-off nozzle and heater plate


1/2 -13 4 PLCS
A
B
1
2
(a) SHOOT W/ SHUT-OFF NOZZLE BALL TYPE
(b) REMOVE BOLTS
(c) REMOVE PART FROM A+B PLATES
(d) PUSH INSERT #1 FROM PART
(e) STRIP INSERT #2 FROM PART + TRIM GATE
NOTE! MOLD MUST BE PRE-HEATED.
2 PLATE MOLD - 2 INSERTS. LATHE & MILL
SCALE: 1/1 PROX.
DATE:
APPROVED BY
DRAWN BY PM
MOLD MAT'L ALUM 2024 T4 - INSERTS DRILL ROD. OIL H'D'N
GEAR
MAT'L NYLON
DRAWING NUMBER A-9



PART REQUIREMENT

DESCRIPTION: Gear - light duty, timer head NUMBER: A-9

MATERIAL: Nylon 6/6

QUANTITY: 200 pieces a year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchase standard metal gears and modify Cost - 200 x $7.50	$1,500

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr + material	$ 150
Molding cost - 10 hours x $5/hr + material	75
Parts to be run as needed	$ 225
Estimated savings:	$1,275

ENGINEERING DETAIL

MATERIAL SELECTION: Nylon 6/6

Reason: Rigid, self-lubricating, quiet running, easy to mold

TOOL: "A" plate is upper cavity and sprue seat.

"B" plate is lower cavity and base.

Insert #1 is internal gear ring to form teeth.

Insert #2 is core - to form hub.

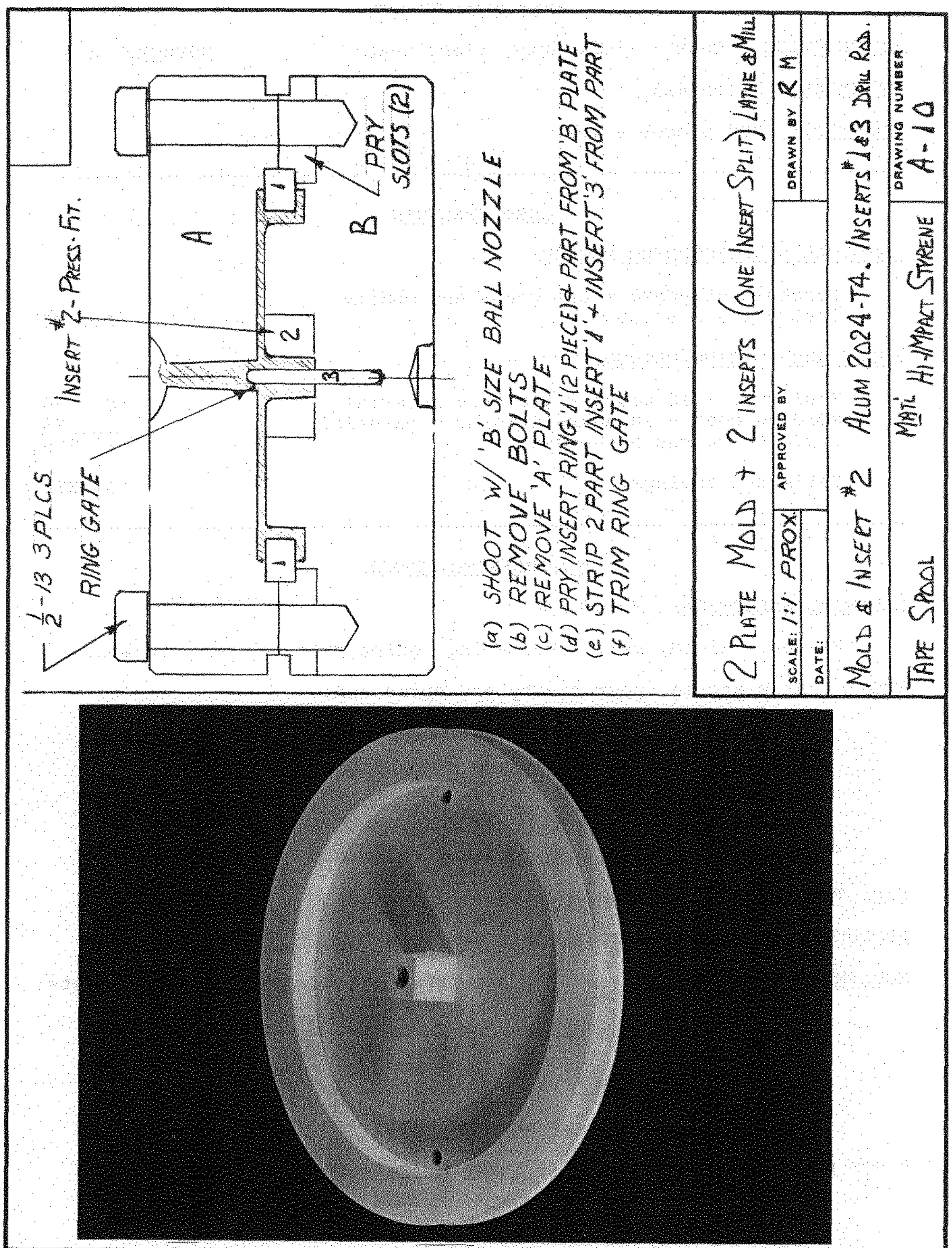
* Mold halves are bolted together (Gear O.D. = 3 5/8").

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 20 parts/hour - hand run

EQUIPMENT: Morgan-Press G-100T plus shut-off nozzle and heater plate

* See page 105 Relationship of Clamp to Injection Pressure



1/2-13 3 PLCS
RING GATE
INSERT #2 - PRESS-FIT.
A
B
PRY SLOTS (2)
1
2
3
(a) SHOOT W/ 'B' SIZE BALL NOZZLE
(b) REMOVE BOLTS
(c) REMOVE 'A' PLATE
(d) PRY INSERT RING '1' (2 PIECE) + PART FROM 'B' PLATE
(e) STRIP 2 PART INSERT '1' + INSERT '3' FROM PART
(f) TRIM RING GATE
2 PLATE MOLD + 2 INSERTS (ONE INSERT SPLIT) LATHE & MILL
SCALE: 1:1 PROX
APPROVED BY
DRAWN BY R M
DATE:
MOLD & INSERT #2 ALUM 2024-T4. INSERTS #1 & 3 DRILL ROD.
TAPE SPOOL
MAT'L HI-IMPACT STYRENE
DRAWING NUMBER A-10

PART REQUIREMENT

DESCRIPTION: Tape spool - high speed indexing tape deck - special custom

NUMBER: A-10

MATERIAL: High impact styrene - white

QUANTITY: 500 pieces total x 50 a month

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Inserts in steel production base in large machine	
Tool costs - inserts/set and mounting	$ 350
Molding cost - 5 hours x $12.50/hr + $35 set up	97
Total cost	$ 447
Lead time - 6 weeks	

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr	$ 120
Molding cost - 17 hours x $5/hr + material	95
Lead time - 3 days to first part	$ 215
Estimated savings: 5 weeks and	$ 232

ENGINEERING DETAIL

MATERIAL SELECTION: Styrene - high impact, white

Reason: Rigid, light weight, white color to match customer product need, good finish

TOOL: "A" plate is upper cavity, sprue gate and nozzle seat.

"B" plate is lower cavity and base.

Insert #1 is split ring.

Insert #2 has rectangular hole to produce drive boss.

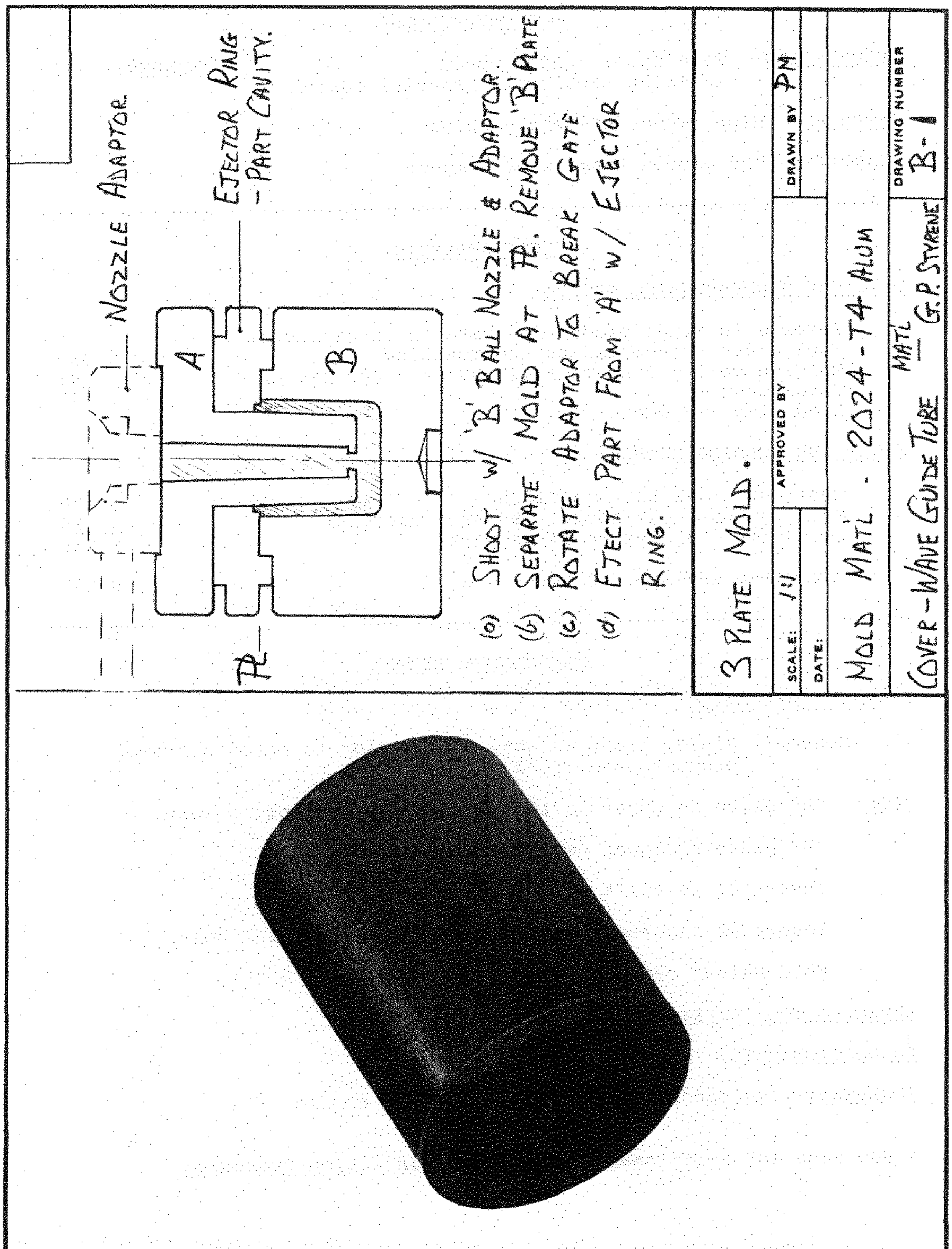
* Mold halves are bolted together (part O.D. = 3").

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 30/hour - by hand

EQUIPMENT: Morgan-Press G-55 with "B" ball nozzle

* See Page 105 Relationship of Clamp to Injection Pressures


NOZZLE ADAPTOR
EJECTOR RING
- PART CAVITY.
A
B
PL
(a) SHOOT W/ 'B' BALL NOZZLE & ADAPTOR.
(b) SEPARATE MOLD AT PL. REMOVE 'B' PLATE
(c) ROTATE ADAPTOR TO BREAK GATE
(d) EJECT PART FROM 'A' W/ EJECTOR RING.
3 PLATE MOLD.
SCALE: 1:1
DATE:
APPROVED BY
DRAWN BY PM
MOLD MATL · 2024-T4 ALUM
COVER-WAVE GUIDE TUBE
MATL G.P. STYRENE
DRAWING NUMBER B-1

PART REQUIREMENT

DESCRIPTION: Cover Tube - plaque presentation model — NUMBER: B-1

MATERIAL: General Purpose Styrene (platable)

QUANTITY: 200 pieces

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Machine from aluminum bar stock, polish and lacquer	
Cost - 200 pieces	
Lead time - 2 weeks	$ 400

SHORT RUN MOLDING METHOD:

Tool cost - 6 hours x $10/hr	$ 60
Molding cost - 4 hours x $5/hr + material	22
Vacuum metalize 200 pieces	45
Lead time - 1 week	$ 127
Estimated savings: 1 week and	$ 273

ENGINEERING DETAIL

MATERIAL SELECTION: G. P. Styrene

Reason: Good smooth surface finish, to be vacuum metalized, rigid, low cost

TOOL: "A" plate is cone and reversed sprue gate.

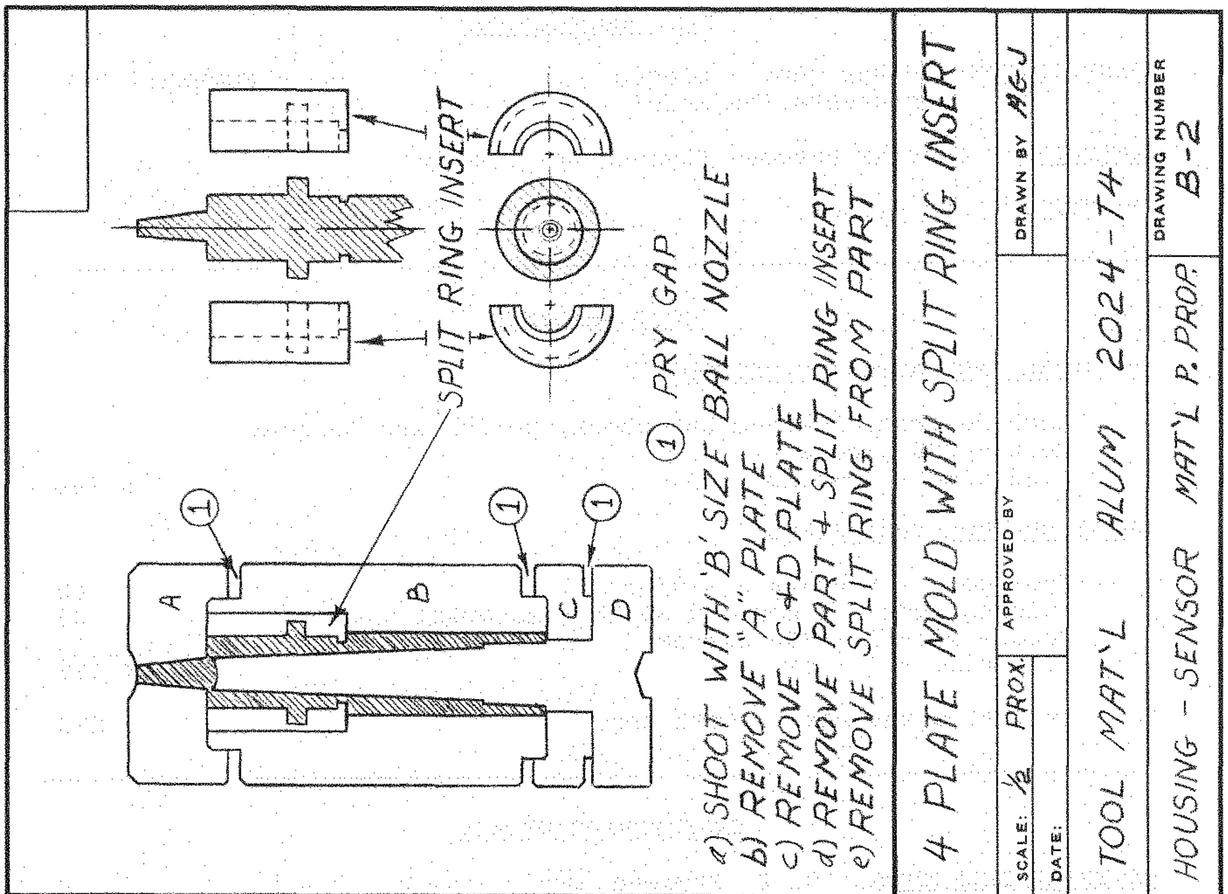
"B" plate is base and cavity.

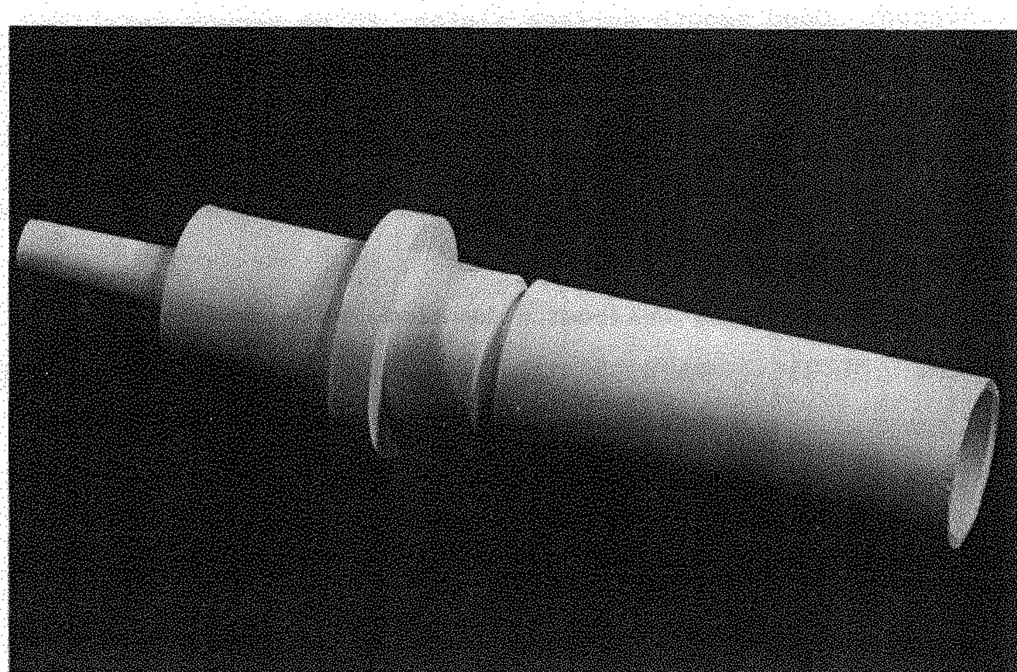
Ejector ring added to aid ejection from core without damage.

EXPECTED TOOL LIFE: 500 pieces

PRODUCTION RATE: 50 pieces/hour - by hand

EQUIPMENT: Morgan-Press G-55 with "B" ball nozzle and nozzle adapter


A
B
C
D
1
SPLIT RING INSERT
SPLIT RING INSERT
1 PRY GAP
a) SHOOT WITH 'B' SIZE BALL NOZZLE
b) REMOVE "A" PLATE
c) REMOVE C+D PLATE
d) REMOVE PART + SPLIT RING INSERT
e) REMOVE SPLIT RING FROM PART
4 PLATE MOLD WITH SPLIT RING INSERT
SCALE: 1/2 PROX
APPROVED BY
DRAWN BY MG-J
DATE:
TOOL MAT'L ALUM 2024-T4
HOUSING - SENSOR MAT'L P. PROP.
DRAWING NUMBER B-2



PART REQUIREMENT

DESCRIPTION: Housing - Corrosion Sensor NUMBER: B-2

MATERIAL: Polypropylene

QUANTITY: 200/year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

The sensor was cast in epoxy. This did not properly seal the electrical components, thus causing underwater failures. The process cycle was long and the rejection rate was high.

SHORT RUN MOLDING METHOD:

Tool cost - lathe turning (16 hrs x $20/hr) + material	$ 340
Molding cost - 8 hours x $8/hr + material	70
Lead time - 1 week	$ 410

ENGINEERING DETAIL

MATERIAL SELECTION: Polypropylene

Reason: Easy to mold and very low moisture absorption rate

TOOL: 3" Rd Aluminum 2024-T4.

"A" plate is sprue seat.

"B" plate is cavity and locator for split ring insert.

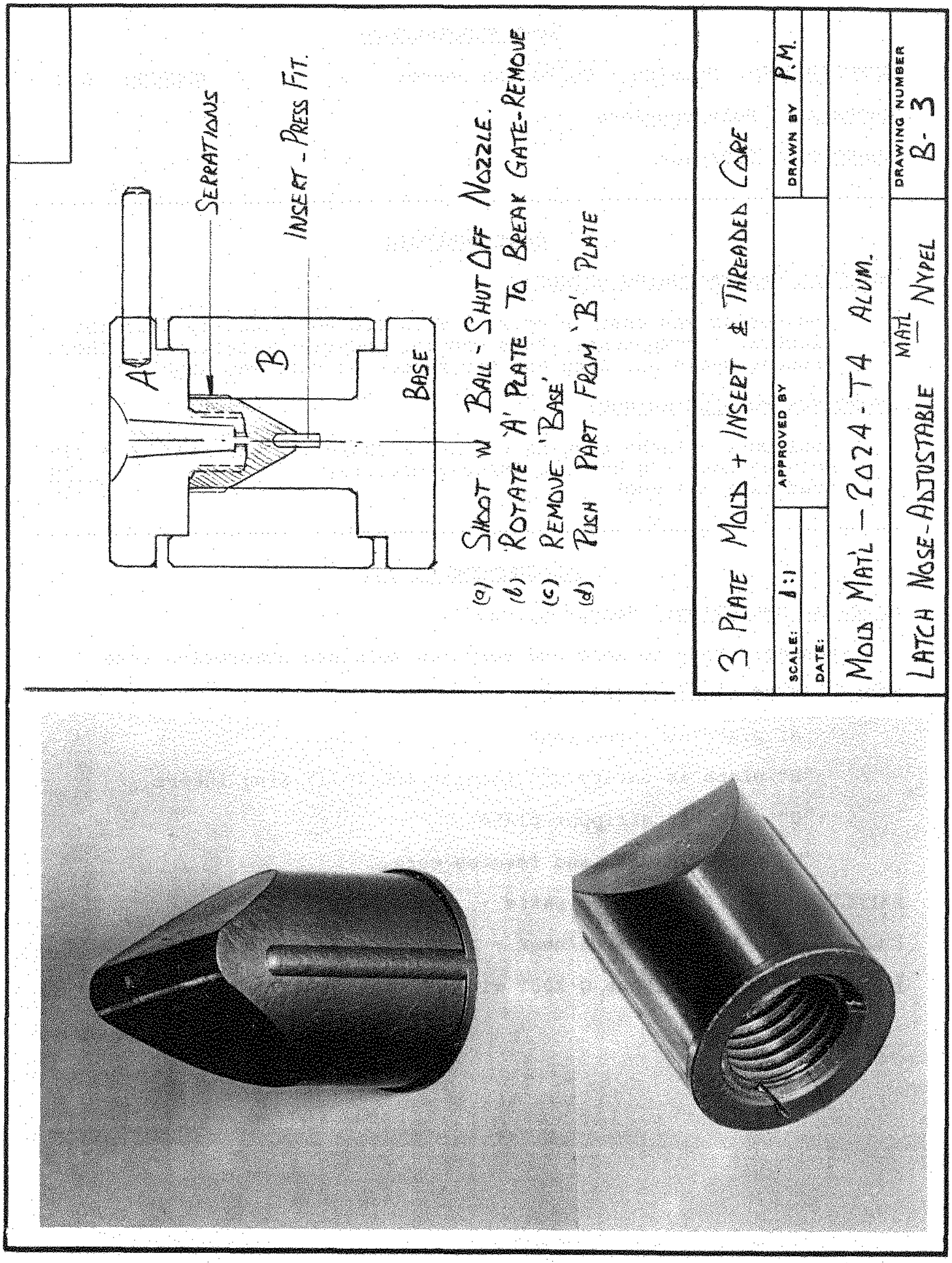
"C" plate is stripper plate.

"D" plate is base and reverse core.

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 30 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-100T with "B" ball nozzle



SERRATIONS
INSERT - PRESS FIT.
A
B
BASE
(a) SHOOT W BALL - SHUT OFF NOZZLE.
(b) ROTATE 'A' PLATE TO BREAK GATE-REMOVE
(c) REMOVE 'BASE'
(d) PUSH PART FROM 'B' PLATE
3 PLATE MOLD + INSERT & THREADED CORE
SCALE:
DATE:
APPROVED BY
DRAWN BY P.M.
MOLD MAT'L — 2024-T4 ALUM.
LATCH NOSE-ADJUSTABLE
MAT'L — NYPEL
DRAWING NUMBER B-3

PART REQUIREMENT

DESCRIPTION: Latch Nose - adjustable NUMBER: B-3

MATERIAL: Nylon - metal filled

QUANTITY: 100 pieces - prototype only

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

1 cavity inserts in existing steel mold base. Run on large production machine.

Tooling cost	$ 350
Molding cost - 1 hour x $12/hr + set-up x $35	47
Lead time - 4 weeks	$ 397

SHORT RUN MOLDING METHOD:

Tool cost - 10 hours x $10/hr + material 6" x 2" dia.	$ 105
Molding cost - 4 hours x $5/hr + material	$ 22
Lead time - 3 days	$ 127
Estimated savings: 3 weeks and	$ 270

ENGINEERING DETAIL

MATERIAL SELECTION: Nylon - metal filled, bronze color

Reason: Match customer products, long-wearing, very high impact strength, self-lubricating

TOOL: "A" plate is threaded core and sprue gate (gate is internal - not visible).

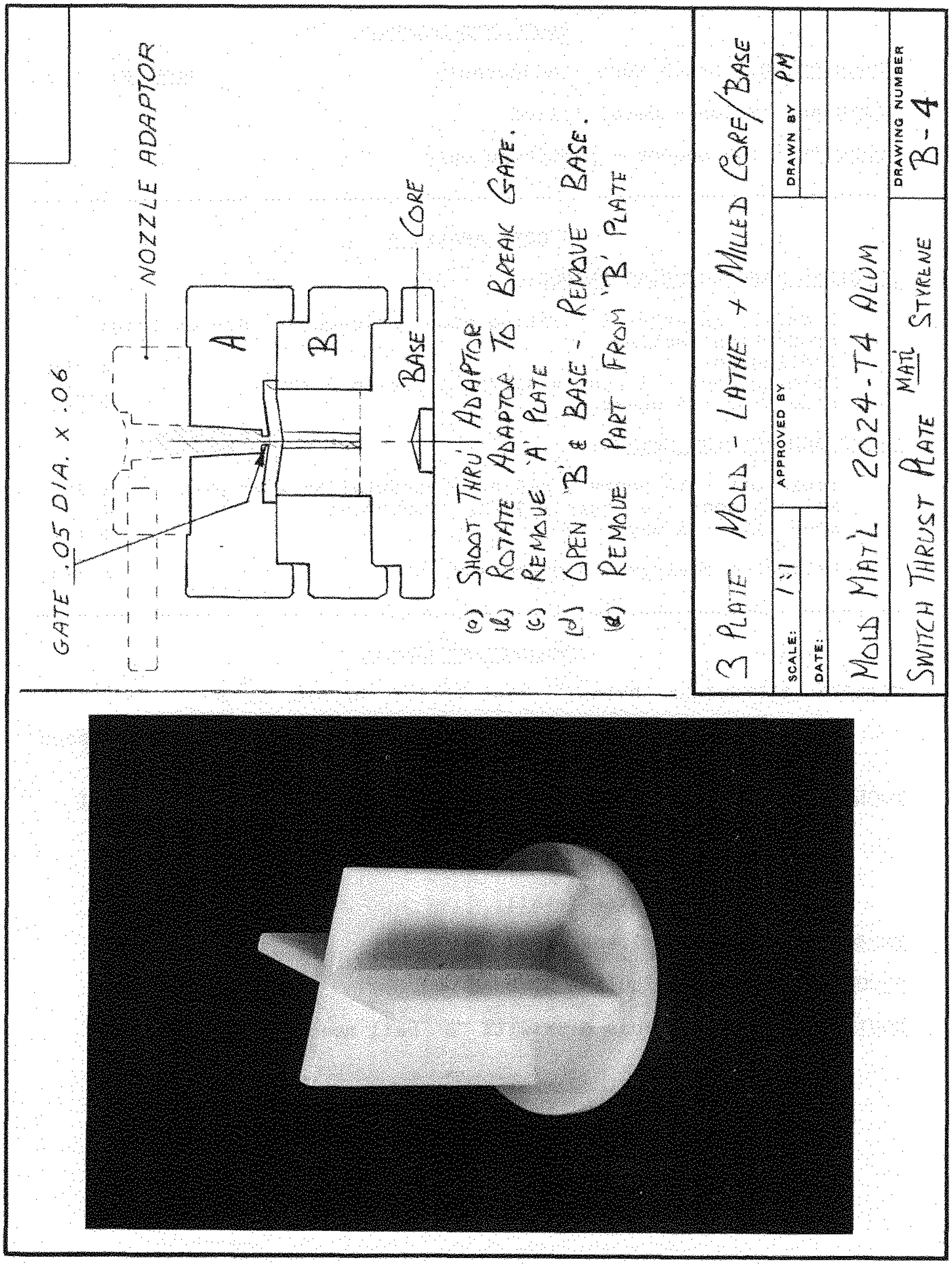
"B" plate is cavity wall.

Base is latch nose detail.

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 30/hour - by hand

EQUIPMENT: Morgan-Press G-55 with "B" ball nozzle


GATE .05 DIA. x .06
NOZZLE ADAPTOR
A
B
BASE
CORE
(a) SHOOT THRU' ADAPTOR
(b) ROTATE ADAPTOR TO BREAK GATE.
(c) REMOVE 'A' PLATE
(d) OPEN 'B' & BASE - REMOVE BASE.
(e) REMOVE PART FROM 'B' PLATE
3 PLATE MOLD - LATHE + MILLED CORE/BASE
SCALE: 1:1
DATE:
APPROVED BY
DRAWN BY PM
MOLD MAT'L 2024-T4 ALUM
SWITCH THRUST PLATE
MAT'L STYRENE
DRAWING NUMBER
B-4

PART REQUIREMENT

DESCRIPTION: Thrust Plate - low voltage switch

NUMBER: B-4

MATERIAL: Styrene - medium impact

QUANTITY: 200/year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Machine from bar stock	
Labor - lathe and mill from bar stock - 12 min/ea. 40 hrs x $10/hr	$ 400
Material - 15 feet of bar x $1.25/ft	20
Lead time - 3 weeks	$ 420

SHORT RUN MOLDING METHOD:

Tool cost - 8 hours x $10/hr + material (6" x 3" diameter aluminum bar stock)	$ 85
Molding cost - 4 hours x $5/hr + material	22
Lead time - 3 days	$ 107
Estimated savings: 2 weeks and	$ 313

ENGINEERING DETAIL

MATERIAL SELECTION: Styrene - medium impact

Reason: Electrically non-conductive, dimensionally stable, rigid, low cost, good impact resistance

TOOL: Three plates with lathe-turned locations

"A" plate is part cavity plus sprue gate for aesthetic reasons.

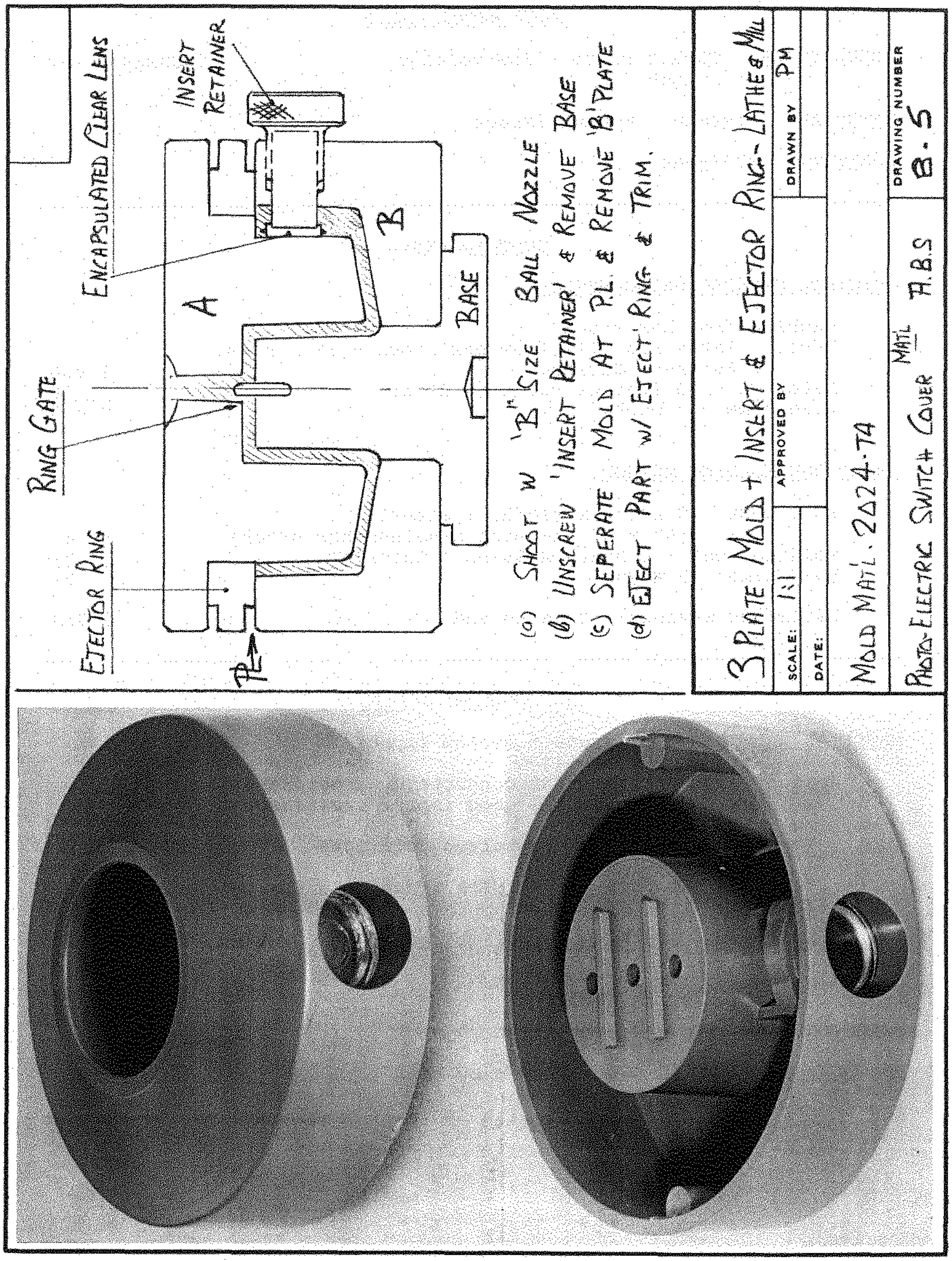
"B" plate is part cavity and stripper plate.

Base is part cavity.

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 60 parts/hour - by hand

EQUIPMENT: Morgan-Press G-55 with "B" ball nozzle and nozzle adapter



Ejector Ring
Ring Gate
Encapsulated Clear Lens
Insert Retainer
A
B
Base
P.L.
(a) Shoot w 'B' Size Ball Nozzle
(b) Unscrew 'Insert Retainer' & Remove Base
(c) Seperate Mold at P.L. & Remove 'B' Plate
(d) Eject Part w/ 'Eject' Ring & Trim.
3 Plate Mold + Insert & Ejector Ring - Lathe & Mill
SCALE: 1:1
APPROVED BY
DRAWN BY PM
DATE:
Mold Mat'l. 2024-T4
Photo-Electric Switch Cover
Mat'l A.B.S
DRAWING NUMBER B.5

PART REQUIREMENT

DESCRIPTION: Cover - Photo Electric Switch   NUMBER: B-5

MATERIAL: A.B.S.

QUANTITY: 25 prototype parts and sales samples for customer approval and evaluation of ultrasonic welding

---

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Bypass prototyping stage - go directly to production	
Tooling delays sales samples	
Lead time- 16 weeks	
Costs - Multi-cavity steel mold	$7,400
High probability of design change following initial production	

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr + material	$ 130
Molding cost - 2 hrs x $5/hr + material	12
Total cost, samples - engineering, sales customers	$ 142
Lead time - 3 days	

NOTE: Prototypes were granted provisional UL approval.
Estimated savings: 15 weeks of time, value of design evaluation before high tooling dollar investment, opportunity to test molding of lens into part and ultrasonic welding parameters

---

ENGINEERING DETAIL

MATERIAL SELECTION: A.B.S.

Reason: Parts to be ultrasonically welded on assembly. Outdoor usage requires parts to be ultraviolet resistant and weatherproof with good electrical resistance and low cost.

TOOL: "A" plate is core and sprue gate.

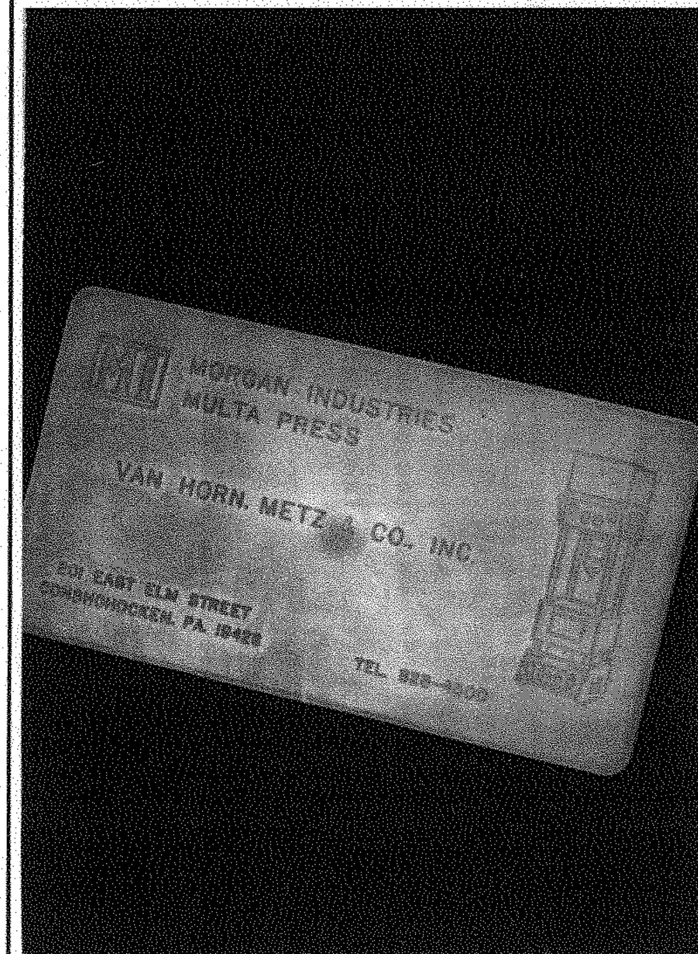
"B" plate is cavity and insert retainer.

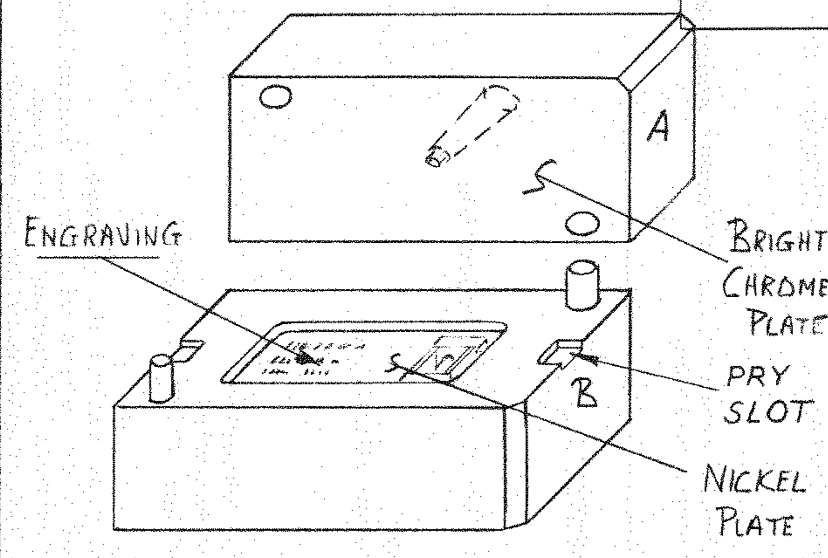
Base is reverse core.

EXPECTED TOOL LIFE: 100 pieces, due to aluminum thread wear

PRODUCTION RATE: 12 parts/hour

EQUIPMENT: Morgan-Press G-100T with "B" ball nozzle





(a) SHOOT THRU NOZZLE ADAPTOR
(b) OPEN MOLD - STRIP PART FROM 'B'
(c) ROTATE ADAPTOR TO BREAK GATE
(d) TRIM CARD.

2 PLATE MOLD - MILLED CAVITY + ENGRAVING

SCALE: ISO
DATE:
APPROVED BY
DRAWN BY RM

MOLD MATL ALUM 2024-T4 - 1½ Plate

BUSINESS CARD - MATL STYRENE-COLORS
DRAWING NUMBER C-1

PART REQUIREMENT

DESCRIPTION: Engraved Chip/Business Card NUMBER: C-1

MATERIAL: Any conventional thermoplastic - any color

QUANTITY: Estimated 5,000 pieces

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

This is a demonstrator tool for customer education and evaluation of materials and color mixes. No previous method existed. Tool used to demonstrate machine "in plant" by distributors.

SHORT RUN MOLDING METHOD:

Tool cost - Milling 6 hours x $10/hr + material	$ 65
Engraving - $40 (Outside Vendor)	40
Chrome and Nickel Plating (Outside Vendor)	30
	$ 135

ENGINEERING DETAIL

MATERIAL SELECTION:

Mold designed and gated to run any conventional material (pre-colored or dry-colored)

TOOL: Aluminum 2024-T4

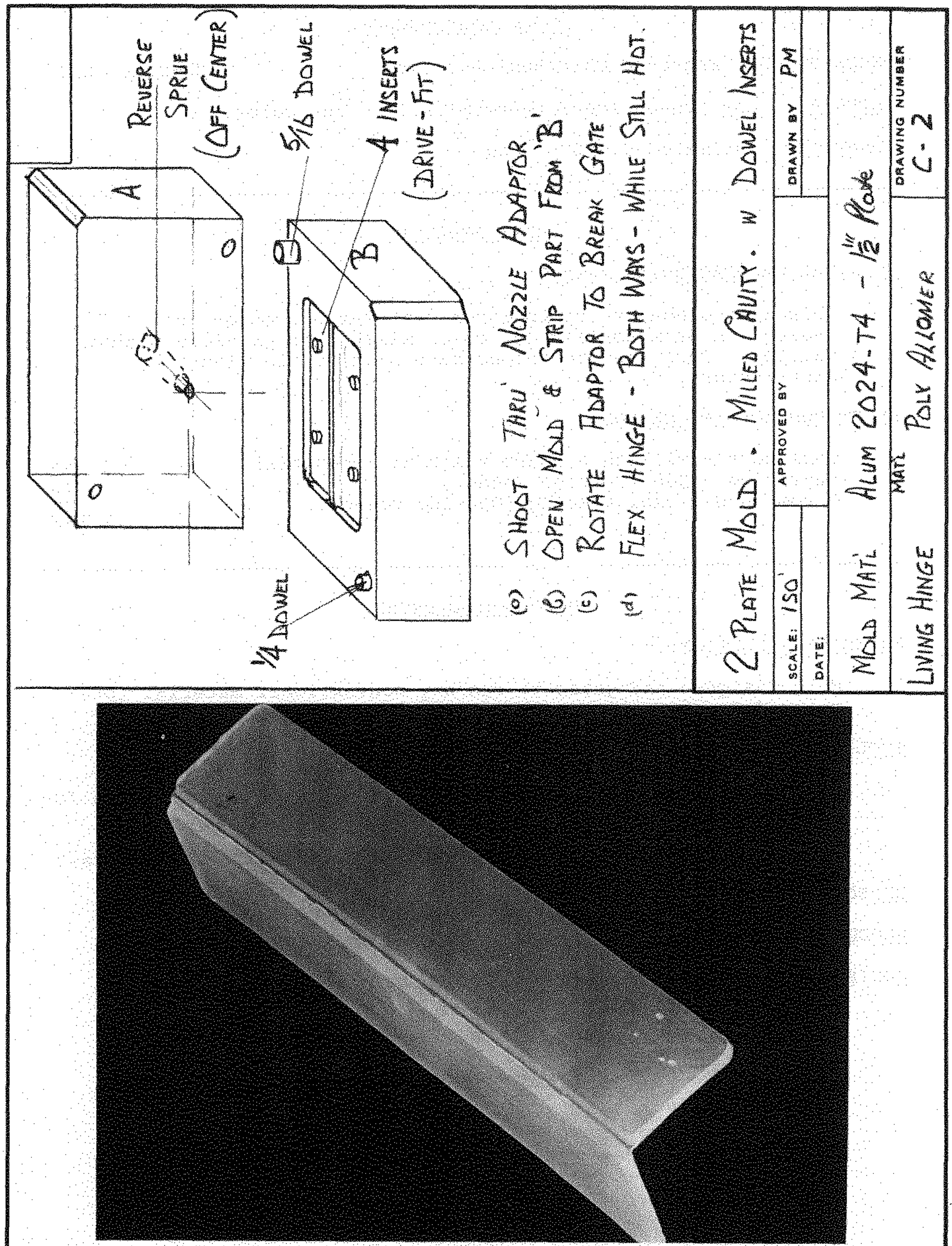
Chrome-plated and polished "A" plate for high gloss finish for color evaluation

Cavity nickel plated for improved surface finish and contrast

EXPECTED TOOL LIFE: 5,000 parts

PRODUCTION RATE: 60 pieces/hour depending on material

EQUIPMENT: Any Morgan-Press model with ball nozzle and nozzle adapter


REVERSE SPRUE (OFF CENTER)
A
5/16 DOWEL
4 INSERTS (DRIVE-FIT)
B
1/4 DOWEL
(a) SHOOT THRU NOZZLE ADAPTOR
(b) OPEN MOLD & STRIP PART FROM 'B'
(c) ROTATE ADAPTOR TO BREAK GATE
(d) FLEX HINGE - BOTH WAYS - WHILE STILL HOT.
2 PLATE MOLD . MILLED CAVITY . W DOWEL INSERTS
SCALE: ISO
DATE:
APPROVED BY
DRAWN BY PM
MOLD MAT'L ALUM 2024-T4 - 1 1/2" Plate
LIVING HINGE
MAT'L POLY ALLOMER
DRAWING NUMBER C-2

PART REQUIREMENT

DESCRIPTION: Living Hinge NUMBER: C-2

MATERIAL: Polyallomer, carbon black filler (in different concentrations)

QUANTITY: 25 pieces, for sub-zero temperature hinge evaluation

---

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Part must be injected molded	
Estimated tooling cost	$ 480
Set up and running - five color mixes	120
Lead time - 6 weeks	$ 600

SHORT RUN MOLDING METHOD:

Tool cost - milling of blocks and cavity (12 hours x $10/hr) + material	$ 140
Molding cost - 5 batch mixes and purging (5 hours x $5/hr) + material	30
Lead time - 3 days	$ 170
Estimated savings: 5 weeks and	$ 430

ENGINEERING DETAIL

MATERIAL SELECTION: Polyallomer

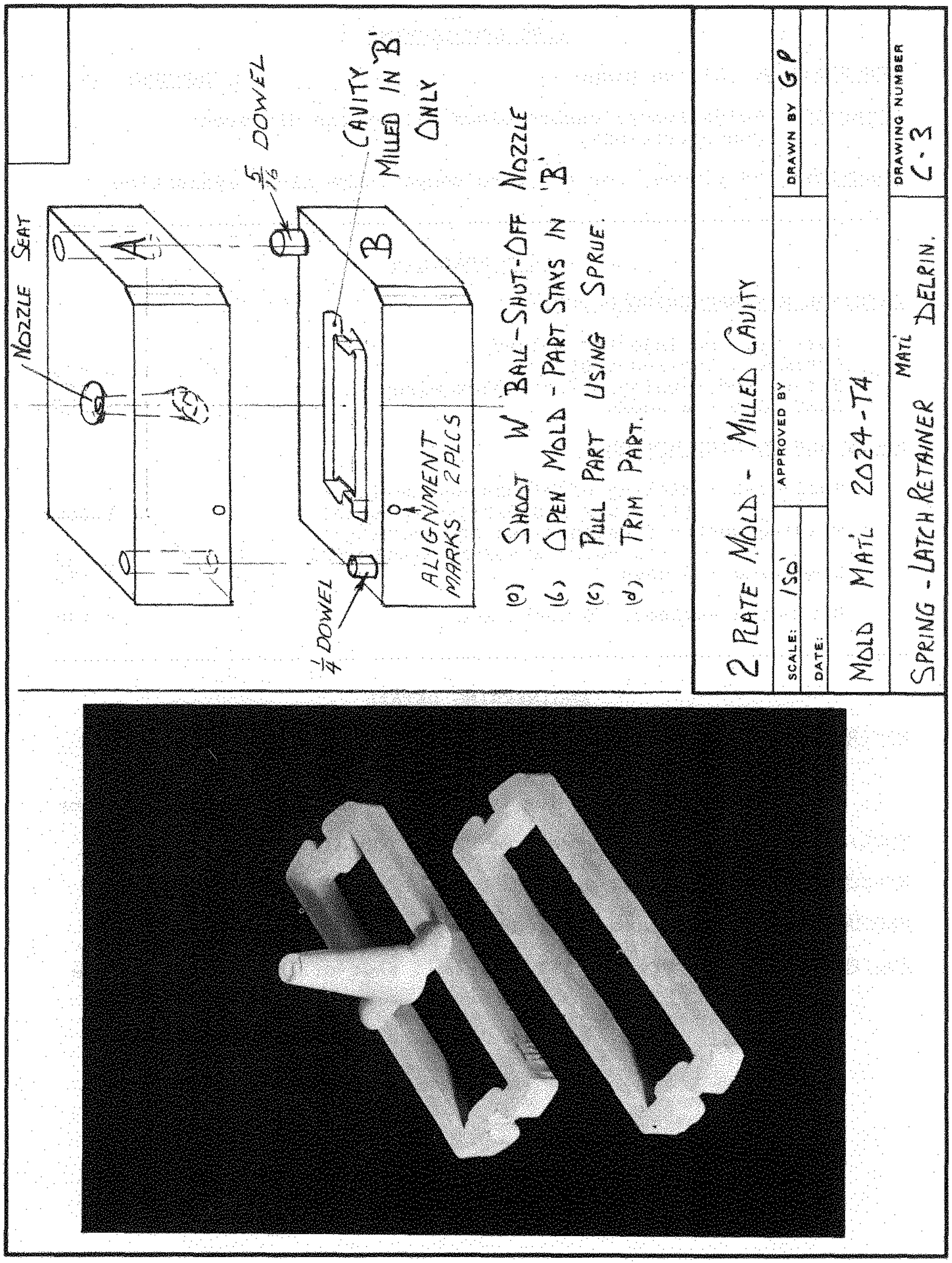
Reason: A good hinge material, flows easily through small section, good low temperature physical characteristics

TOOL: Aluminum 2024-T4, steel core pins (dowels)

EXPECTED TOOL LIFE: 500 pieces

PRODUCTION RATE: 5 parts/hour due to filler mixing and changeover

EQUIPMENT: Morgan-Press G-55T with ball shut-off nozzle and nozzle adapter


NOZZLE SEAT
NOZZLE
A
5/16 DOWEL
CAVITY MILLED IN 'B' ONLY
B
1/4 DOWEL
ALIGNMENT MARKS 2PLCS
(a) SHOOT W' BALL-SHUT-OFF NOZZLE
(b) OPEN MOLD - PART STAYS IN 'B'
(c) PULL PART USING SPRUE.
(d) TRIM PART.
2 PLATE MOLD - MILLED CAVITY
SCALE: ISO
DATE:
APPROVED BY
DRAWN BY GP
MOLD MAT'L 2024-T4
SPRING - LATCH RETAINER
MAT'L DELRIN.
DRAWING NUMBER C-3

PART REQUIREMENT

DESCRIPTION: Spring NUMBER: C-3

MATERIAL: Delrin 500, natural color

QUANTITY: 200 pieces

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Alternate method, machine from Delrin plate	
Estimated mfg. cost (due to tight tolerances)	$ 800
Lead time - 2 to 3 weeks	

SHORT RUN MOLDING METHOD:

Tool cost - milling of cavity & block (8 hours x $10/hr) + material	$ 90
Molding cost - 4 hours x $5/hr + material	25
Lead time - 2 days	$ 115
Estimated savings: 2 weeks and	$ 685

ENGINEERING DETAIL

MATERIAL SELECTION: Delrin 500 natural

Reason: Excellent spring material, molds easily into hot mold

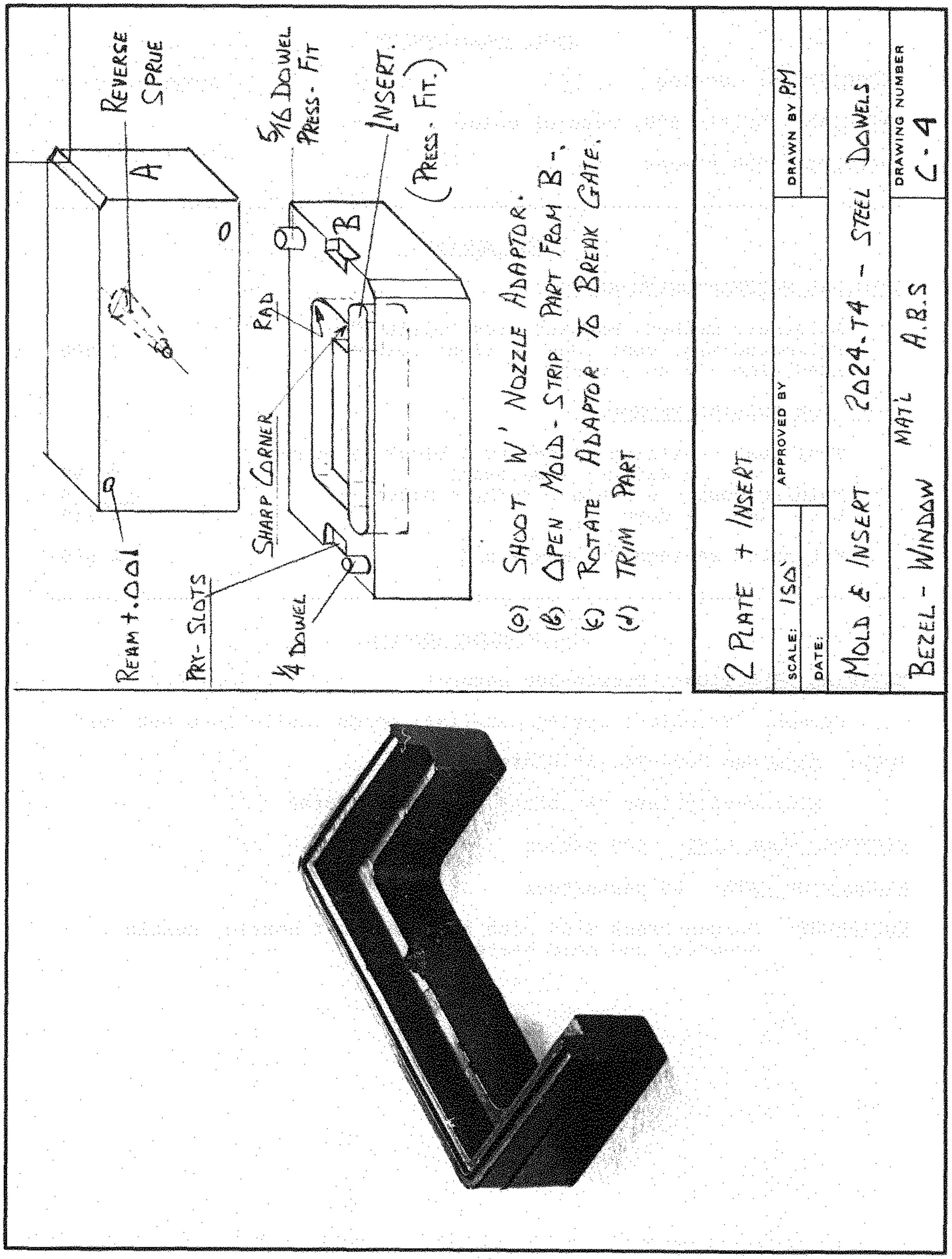
TOOL: Aluminum 2024-T4, 1" plate

Mill cavity into "B" plate - no loose cores

EXPECTED TOOL LIFE: 500 pieces

PRODUCTION RATE: 60 parts/hour

EQUIPMENT: Morgan-Press G-55 with ball shut-off nozzle, nozzle adapter, and mold heater plate


REVERSE SPRUE
A
5/16 DOWEL PRESS-FIT
INSERT.
(PRESS-FIT.)
B
RAD
SHARP CORNER
REAM +.001
PRY-SLOTS
1/4 DOWEL
(a) SHOOT W' NOZZLE ADAPTOR.
(b) OPEN MOLD-STRIP PART FROM B-.
(c) ROTATE ADAPTOR TO BREAK GATE.
(d) TRIM PART
2 PLATE + INSERT
SCALE: ISO
DATE:
APPROVED BY
DRAWN BY PM
MOLD & INSERT 2024.T4 - STEEL DOWELS
BEZEL - WINDOW
MAT'L A.B.S
DRAWING NUMBER
C-4

PART REQUIREMENT

DESCRIPTION: Bezel NUMBER: C-4

MATERIAL: A.B.S. - black

QUANTITY: 200 (100 sets - 2 pieces to a set)

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Part must be injection molded - has a .008 high bead on after-surface for ultrasonic welding to mating assembly. Estimated cost of conventional tooling and parts.	$ 550
Lead time - 6 weeks	

SHORT RUN MOLDING METHOD:

Tool cost - mill cavity and insert (15 hours x $10/hr) + material	$ 160
Molding cost - 4 hours x $5/hr + material	25
Lead time - 1 week	$ 185
Estimated savings: 5 weeks and	$ 365

ENGINEERING DETAIL

MATERIAL SELECTION: A.B.S.

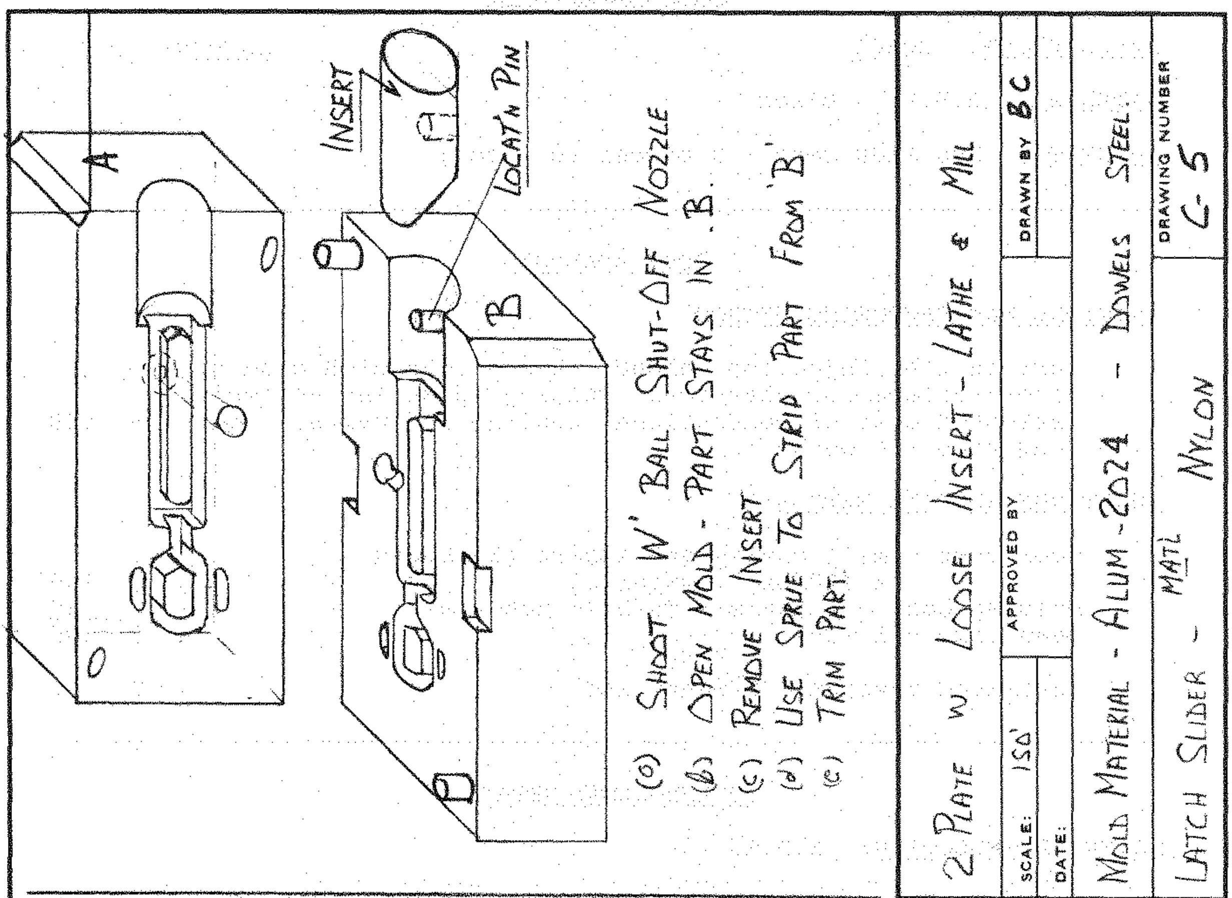
Reason: Has good appearance, good weldability and high strength

TOOL: Aluminum 2024-T4 - insert same

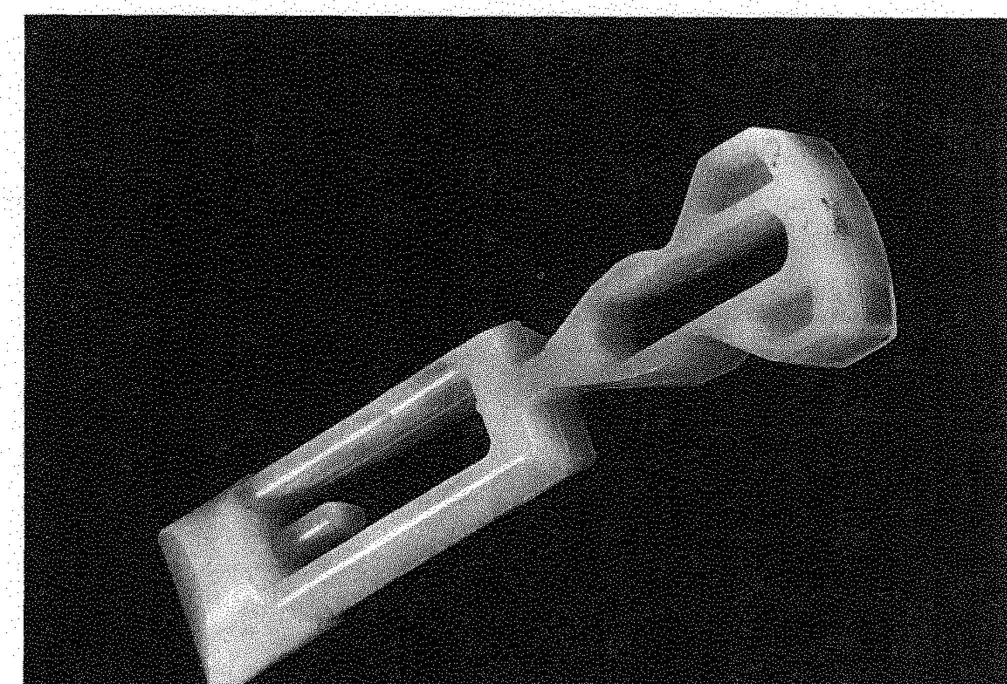
EXPECTED TOOL LIFE: 1,000 pieces

PRODUCTION RATE: 50/hour

EQUIPMENT: Morgan-Press G-55 with ball nozzle, nozzle adapter, pre-heat plate



A
B
INSERT
LOCAT'N PIN
(a) SHOOT W' BALL SHUT-OFF NOZZLE
(b) OPEN MOLD - PART STAYS IN 'B'.
(c) REMOVE INSERT
(d) USE SPRUE TO STRIP PART FROM 'B'
(e) TRIM PART.
2 PLATE W LOOSE INSERT - LATHE & MILL
SCALE:
DATE:
APPROVED BY
DRAWN BY BC
MOLD MATERIAL - ALUM-2024 - DOWELS STEEL
LATCH SLIDER - MAT'L NYLON
DRAWING NUMBER
C-5



PART REQUIREMENT

DESCRIPTION: Latch — NUMBER: C-5

MATERIAL: Nylon, bronze powder filled

QUANTITY: 100 parts - sales samples

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Parts must be molded	
Material not available in bar form	
Estimated cost of conventional tooling	$ 800
Set up and running	35
Lead time - 6 weeks	$ 835

SHORT RUN MOLDING METHOD:

Tool cost - milling, lathe turning (36 hours x $10/hr) + material	$ 380
Molding cost - 2 hours x $5/hr + material	15
	$ 395
Estimated savings: 5 weeks and	$ 440

ENGINEERING DETAIL

MATERIAL SELECTION: Nylon, bronze filled

Reason: High impact strength, low friction, looks like conventional bronze sintered part

TOOL: Aluminum 2024-T4

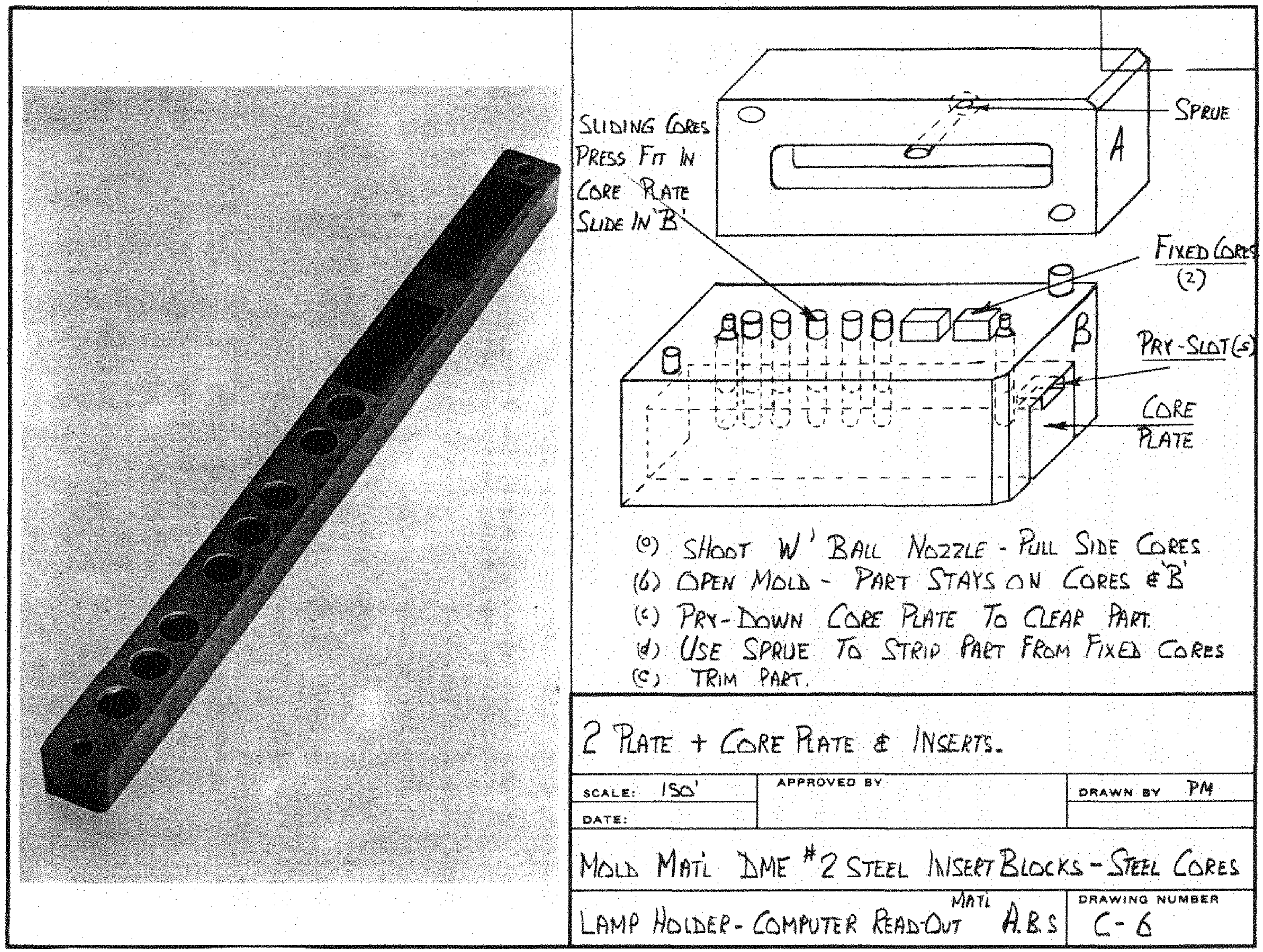
Cavity cut in A & B plates

Off-set sprue, side gate

EXPECTED TOOL LIFE: 1,000 pieces

PRODUCTION RATE: 50/hour - hand cycle

EQUIPMENT: Morgan-Press G-100T with ball shut-off nozzle, nozzle adapter, and mold heater plate


SLIDING CORES PRESS FIT IN CORE PLATE SLIDE IN 'B'
SPRUE
A
FIXED CORES (2)
B
PRY-SLOT(S)
CORE PLATE
(a) SHOOT W' BALL NOZZLE - PULL SIDE CORES
(b) OPEN MOLD - PART STAYS ON CORES & 'B'
(c) PRY-DOWN CORE PLATE TO CLEAR PART.
(d) USE SPRUE TO STRIP PART FROM FIXED CORES
(e) TRIM PART.
2 PLATE + CORE PLATE & INSERTS.
SCALE: ISO'
APPROVED BY
DRAWN BY PM
DATE:
MOLD MAT'L DME #2 STEEL INSERT BLOCKS - STEEL CORES
LAMP HOLDER - COMPUTER READ-OUT
MAT'L A.B.S
DRAWING NUMBER C-6

PART REQUIREMENT

DESCRIPTION: Lamp Holder NUMBER: C-6

MATERIAL: A.B.S., black

QUANTITY: 1,200 parts

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Alternate method - machine from black acrylic plate	
Quoted cost	$1,400
Lead time - 5 weeks	

SHORT RUN MOLDING METHOD:

Tool cost - 40 hours x $10/hr + material	$ 430
Molding cost - 40 hours x $5/hr + material	240
Lead time - 3 weeks	$ 670
Estimated savings: 2 weeks and	$ 730

ENGINEERING DETAIL

MATERIAL SELECTION: A.B.S., black

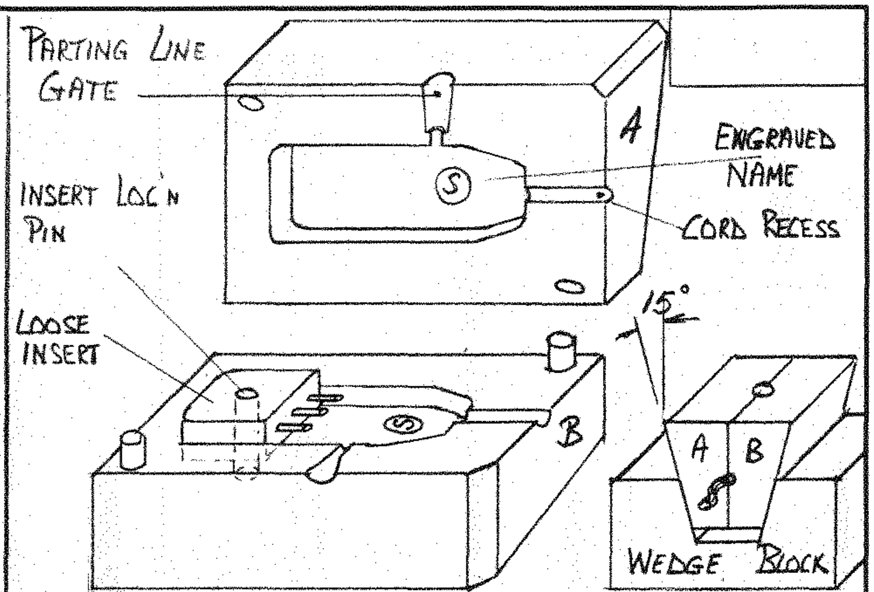
Reason: Easy molding, good finish, good dielectric characteristics, rigid

TOOL: Aluminum 2024-T4, steel core pins, and cross pins

EXPECTED TOOL LIFE: 2,500 parts

PRODUCTION RATE: 30/hour - hand cycle

EQUIPMENT: Morgan-Press G-100T with ball nozzle, pre-heat plate and nozzle adapter



(a) LOAD WIRES TO PINS ON INSERT - LOCATE CORD
(b) PUT MOLD IN WEDGE BLOCK - SHOOT W' FLAT NOZZLE
(c) OPEN MOLD - REMOVE PLUG & INSERT
(d) STRIP INSERT FROM PLUG & TRIM GATE
(e) RE-LOAD INSERT

2 PLATE W' INSERT & WEDGE BLOCK

SCALE: ISO	APPROVED BY	DRAWN BY P.M.
DATE:		

MOLD MAT'L ALUM. 2024 .T4. WEDGE BLK STEEL.

5 PIN 'FEMALE' PLUG	MAT'L VINYL	DRAWING NUMBER C-7

PART REQUIREMENT

DESCRIPTION: Three-pin Female Plug NUMBER: C-7

MATERIAL: Vinyl

QUANTITY: 10,000 assemblies

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Parts molded on production equipment in 5,000 part runs	
Tool cost	$1,600
Part price - 20¢ each	$2,000
Lead time - 8 weeks	$3,600

SHORT RUN MOLDING METHOD:

Tool cost - 30 hours x $10/hr + engraving + material	$ 380
Molding cost - 250 hours x $5/hr + material	1,330
Lead time - first part 2 weeks	$1,710
Estimated savings: 4-6 weeks and	$1,890

ENGINEERING DETAIL

MATERIAL SELECTION: Vinyl, soft grey

Reason: Matches wire coating in hardness and color

TOOL: Aluminum 2024-T4

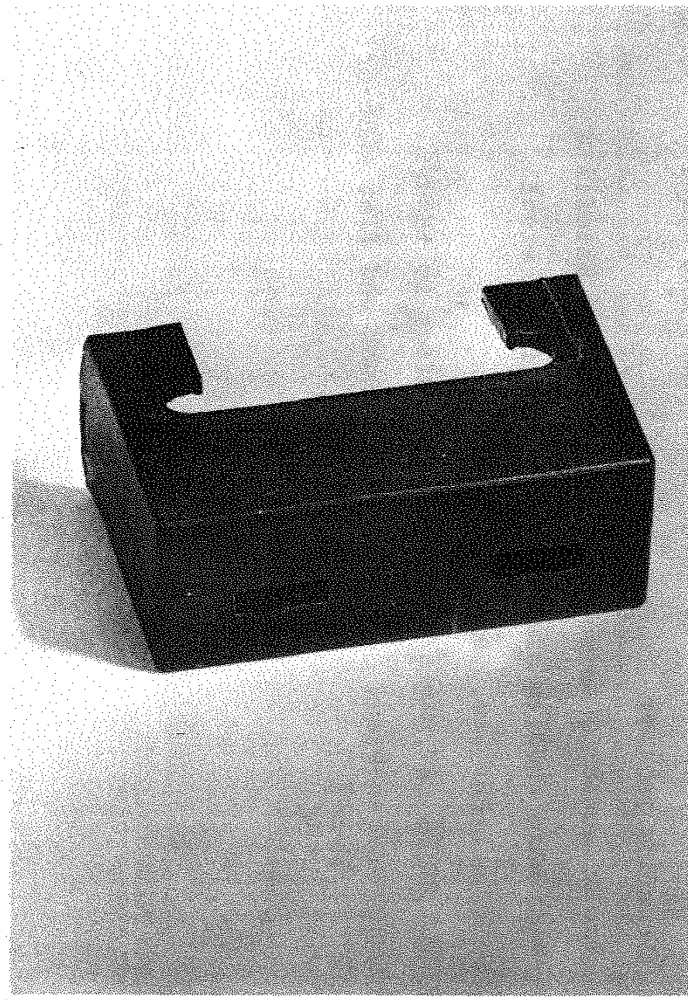
Not affected by corrosive nature of vinyl materials

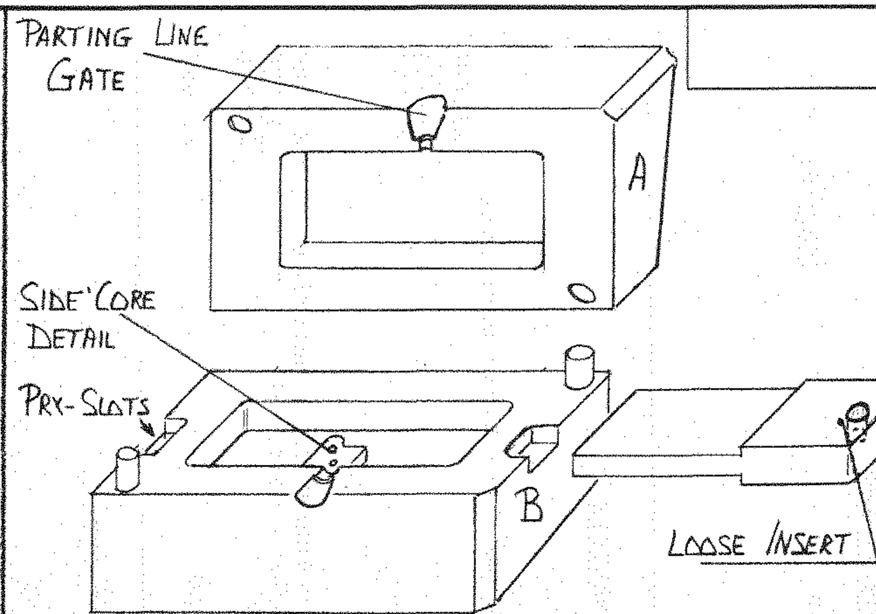
Must hard anodize aluminum or nickel plate steel

EXPECTED TOOL LIFE: 10,000 parts

PRODUCTION RATE: 40 parts/hour hand load wires to insert

EQUIPMENT: Morgan-Press G-55 with wedge block and flat shut-off nozzle





(a) LOAD INSERT INTO 'B' CAVITY
(b) PUT MOLD INTO WEDGE BLOCK - SHOOT W/ FLAT NOZ
(c) OPEN MOLD - REMOVE PART & INSERT.
(d) STRIP INSERT FROM PART - TRIM

2 PLATE & WEDGE BLOCK W LOOSE INSERT.

SCALE: 1.50'	APPROVED BY	DRAWN BY GP
DATE:		

TOOL MAT'L - 2024-T4 ALUM - INSERT - TOOL STEEL

TERMINAL BLOCK COVER — MAT'L POLYSULFONE

DRAWING NUMBER C-8

PART REQUIREMENT

DESCRIPTION: Terminal Block NUMBER: C-8

MATERIAL: Polysulfone

QUANTITY: 1,000 parts

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Parts must be injection or compression molded	
Estimated cost of conventional tooling	$ 675
Set up, running and materials	210
Lead time - 5 weeks	$ 885

SHORT RUN MOLDING METHOD:

Tool cost - 20 hours x $10/hr + material	$ 240
Molding cost - 50 hours x $5/hr + material	350
Lead time - 2 weeks	$ 590
Estimated savings: 3 weeks and	$ 295

ENGINEERING DETAIL

MATERIAL SELECTION: Polysulfone, black

Reason: High temperature, high voltage resistance, and high strength

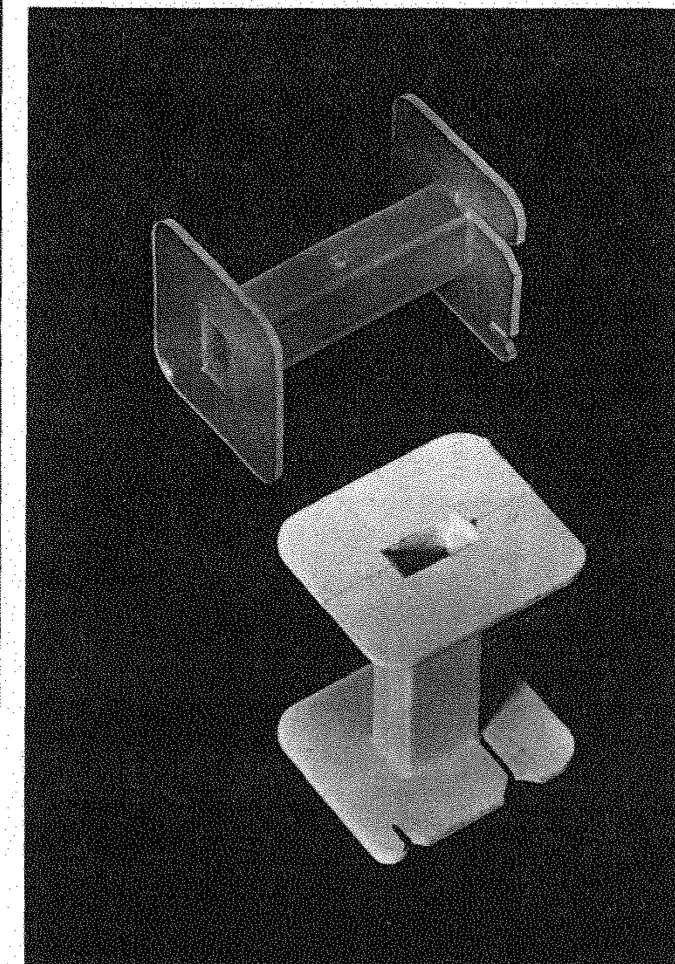
TOOL: Aluminum 2024-T4

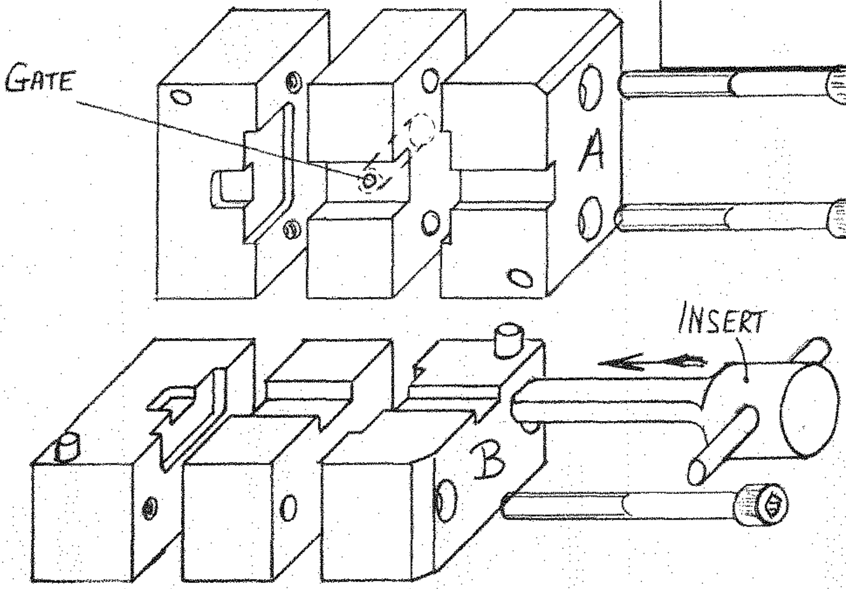
Tool steel side core and insert

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 20 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-100T with wedge block, flat "B" nozzle and heater plate





(a) BOLT 'A' & 'B' PLATE ASSYS TOGETHER (4 SCREWS)
(b) POSITION INSERT - SHOOT W' ADAPTOR & BALL NOZZLE
(c) SLACKEN 4 SCREWS 1/4 TURN - OPEN MOLD
(d) PART & INSERT STAY IN 'A' - . BREAK GATE w ADAPTOR
(e) REMOVE PART & INSERT - STRIP INSERT
(f) TRIM GATE

2 PLATE (SPLIT BLOCK) w INSERT. MILL'

SCALE: ISO'	APPROVED BY	DRAWN BY
DATE:		

MOLD MAT'L ALUM 2024-T4. INSERT. TOOL STEEL

BOBBIN . COIL RELAY. MAT'L NYLON. DRAWING NUMBER C-9

PART REQUIREMENT

DESCRIPTION: Bobbin NUMBER: C-9

MATERIAL: Nylon 6/6

QUANTITY: 250 - prototype use only

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Insert in existing mold base, hand pulled core	$ 600
Set-up and run parts	65
Lead time - 4 weeks	$ 665

SHORT RUN MOLDING METHOD:

Tool cost - 32 hours x $10/hr + material	$ 360
Molding cost - 13 hours x $5/hr + material	75
Lead time - 1 week	$ 435
Estimated savings: 3 weeks and	$ 230

ENGINEERING DETAIL

MATERIAL SELECTION: Nylon 6/6

Reason: High impact, good electrical resistance, flows into thin sections

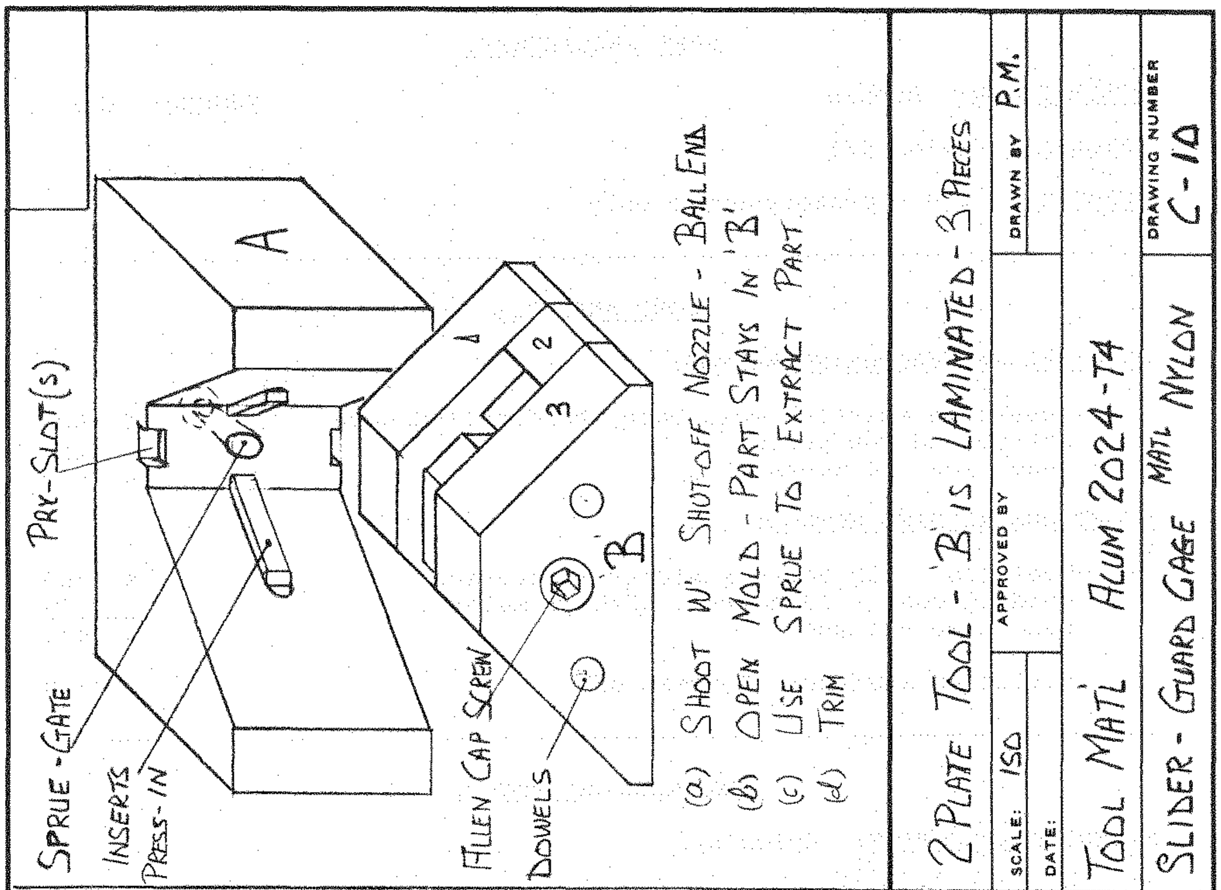
TOOL: Aluminum 2024-T4

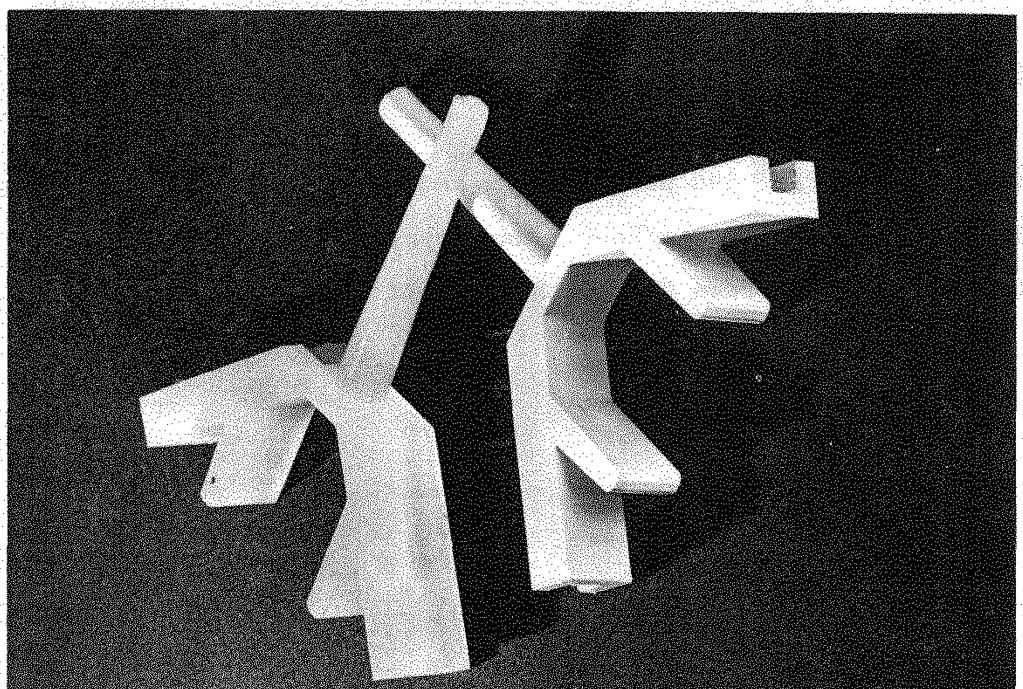
Tool steel rod insert

EXPECTED TOOL LIFE: 500 parts (bolts and threads controlling factor)

PRODUCTION RATE: 20 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-55 with shut-off nozzle, nozzle adapter and heater plate


PRY-SLOT(S)
A
1
2
3
'B'
SPRUE-GATE
INSERTS
PRESS-IN
ALLEN CAP SCREW
DOWELS
(a) SHOOT W/ SHOT-OFF NOZZLE - BALL END
(b) OPEN MOLD - PART STAYS IN 'B'
(c) USE SPRUE TO EXTRACT PART
(d) TRIM
2 PLATE TOOL - 'B' IS LAMINATED - 3 PIECES
SCALE: ISO
DATE:
APPROVED BY
DRAWN BY P.M.
TOOL MAT'L ALUM 2024-T4
SLIDER - GUARD GAGE MAT'L NYLON
DRAWING NUMBER C-10



PART REQUIREMENT

DESCRIPTION: Slider NUMBER: C-10

MATERIAL: Nylon

QUANTITY: 400 a year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Machine from bar stock - 400 x $2.00 each	$ 800

SHORT RUN MOLDING METHOD:

Tool cost - 24 hours x $10/hr + material	$ 250
Molding cost - 10 hours x $5/hr + material	60
	$ 310
Estimated savings:	$ 490

ENGINEERING DETAIL

MATERIAL SELECTION: Nylon

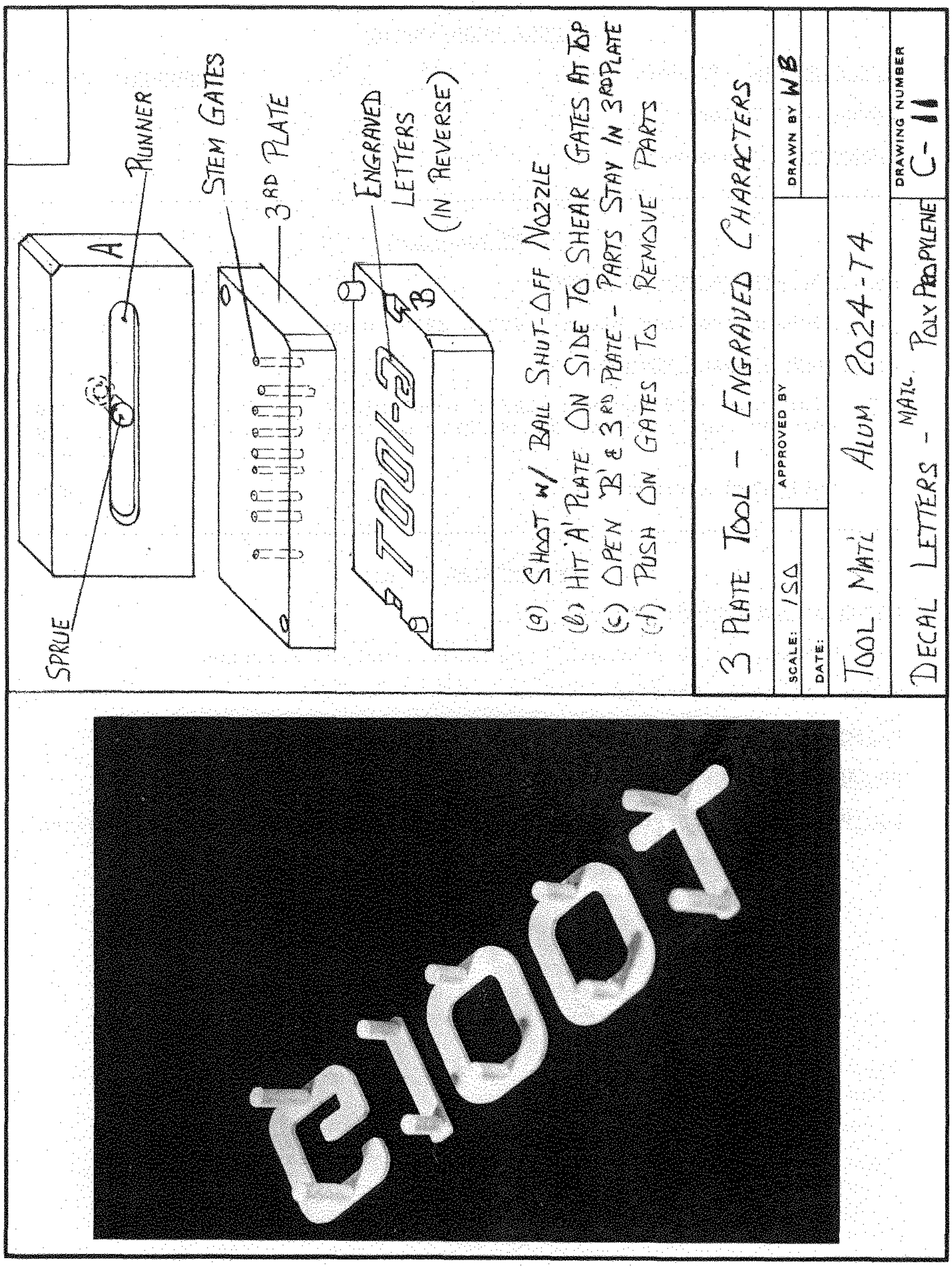
Reason: High strength, low coefficient of friction

TOOL: Aluminum 2024-T4

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 50 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-100T with shut-off nozzle and heater plate


SPRUE
RUNNER
A
STEM GATES
3RD PLATE
ENGRAVED LETTERS (IN REVERSE)
B
TOOL-2
(a) SHOOT w/ BALL SHUT-OFF NOZZLE
(b) HIT 'A' PLATE ON SIDE TO SHEAR GATES AT TOP
(c) OPEN 'B' & 3RD PLATE - PARTS STAY IN 3RD PLATE
(d) PUSH ON GATES TO REMOVE PARTS
3 PLATE TOOL - ENGRAVED CHARACTERS
SCALE: 1SQ
DATE:
APPROVED BY
DRAWN BY WB
TOOL MAT'L ALUM 2024-T4
DECAL LETTERS - MAT'L POLY PROPYLENE
DRAWING NUMBER C-11

PART REQUIREMENT

DESCRIPTION: Letters NUMBER: C-11

MATERIAL: Polypropylene

QUANTITY: 100 a year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Buy printed decal and attach to unit (100 x $3.50)	$ 350

SHORT RUN MOLDING METHOD:

Tool cost - 12 hours x $10/hr + material	$ 130
Molding cost - 2 hours x $5/hr + material	15
	$ 145
Estimated savings:	$ 205

ENGINEERING DETAIL

MATERIAL SELECTION: Polypropylene

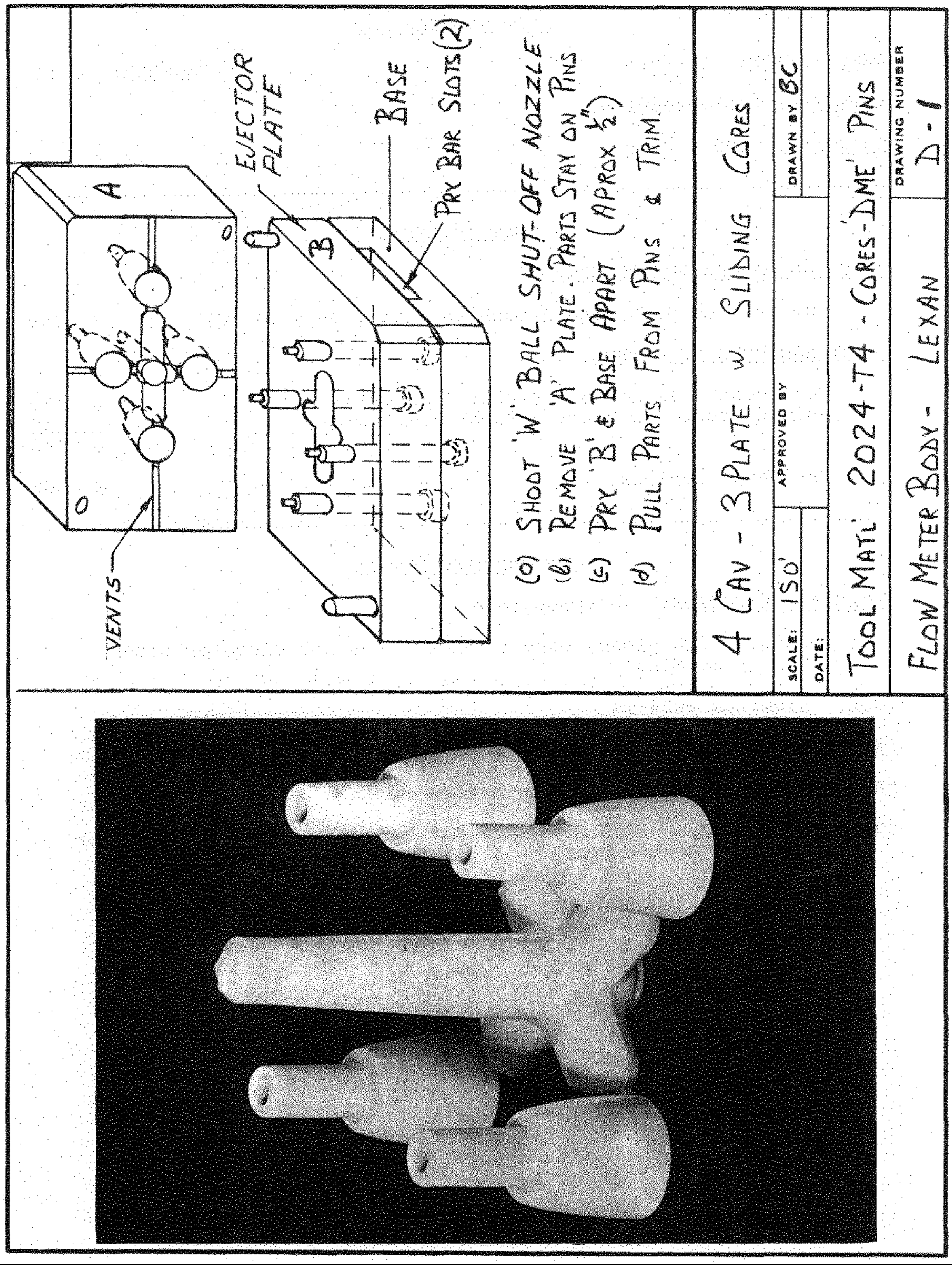
Reason: High gloss, easy to mold, does not discolor after molding

TOOL: Aluminum 2024-T4

EXPECTED TOOL LIFE: 1,000 parts

PRODUCTION RATE: 50 parts/hour - hand cycle

EQUIPMENT: Morgan-Press G-55 with "A" ball nozzle, nozzle adapter and heater plate


VENTS
EJECTOR PLATE
BASE
PRY BAR SLOTS (2)
A
B
(a) SHOOT 'W' BALL SHUT-OFF NOZZLE
(b) REMOVE 'A' PLATE. PARTS STAY ON PINS
(c) PRY 'B' & BASE APART (APROX ½")
(d) PULL PARTS FROM PINS & TRIM.
4 CAV - 3 PLATE W SLIDING CORES
SCALE:
DATE:
APPROVED BY
DRAWN BY BC
TOOL MAT'L 2024-T4 - CORES-'DME' PINS
DRAWING NUMBER
D-1
FLOW METER BODY - LEXAN

PART REQUIREMENT

DESCRIPTION: Flow Meter Body NUMBER: D-1

MATERIAL: Lexan

QUANTITY: 5,000 a month maximum - 60,000 total

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchased item (part of assembly) Estimated cost - 10¢/pair	$3,000

SHORT RUN MOLDING METHOD:

Tool cost - 60 hours x $10/hr + material	$ 650
Molding cost - 15,000 shots x 120/hr; 125 hours x $5 + material	700
	$1,350
Estimated savings:	$1,650

ENGINEERING DETAIL

MATERIAL SELECTION: Lexan

Reason: High temperature resistance, very high impact strength

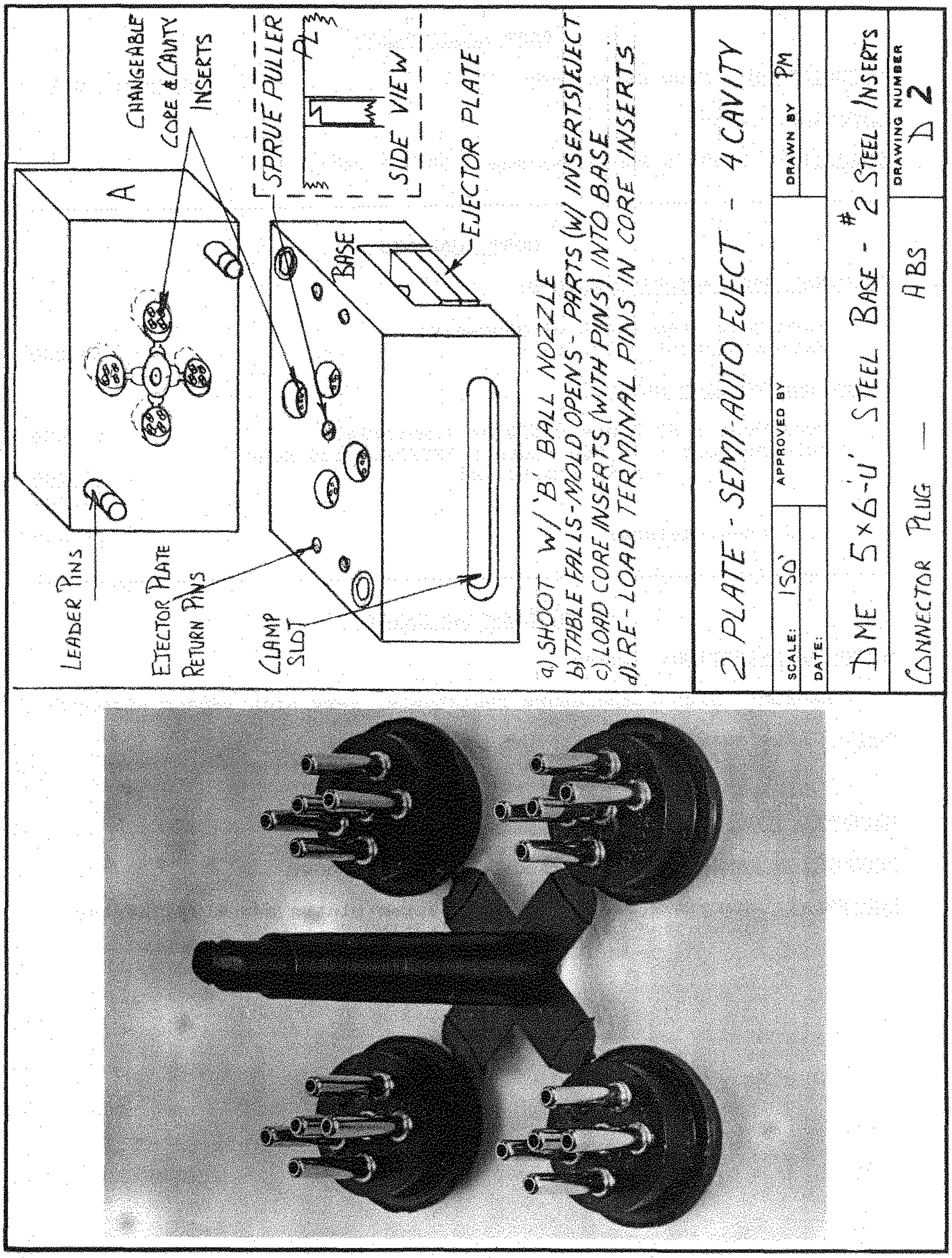
TOOL: Aluminum 2024-T4 with DME pins as cores

Four cavity mold, semi-automatic

EXPECTED TOOL LIFE: 100,000 parts

PRODUCTION RATE: 120 shots/hour = 480 parts/hour

EQUIPMENT: Morgan-Press G-100T with upper platen assembly, heater plate and ball shut-off nozzle


LEADER PINS
EJECTOR PLATE RETURN PINS
CLAMP SLOT
A
CHANGEABLE CORE & CAVITY INSERTS
SPRUE PULLER
PL
SIDE VIEW
BASE
EJECTOR PLATE
a) SHOOT W/ 'B' BALL NOZZLE
b) TABLE FALLS - MOLD OPENS - PARTS (W/ INSERTS) EJECT
c) LOAD CORE INSERTS (WITH PINS) INTO BASE
d) RE-LOAD TERMINAL PINS IN CORE INSERTS
2 PLATE - SEMI-AUTO EJECT - 4 CAVITY
SCALE:
APPROVED BY
DRAWN BY PM
DATE:
DME 5×6-'U' STEEL BASE - #2 STEEL INSERTS
CONNECTOR PLUG —
ABS
DRAWING NUMBER
D 2

PART REQUIREMENT

DESCRIPTION: Connector Plug NUMBER: D-2

MATERIAL: A.B.S

QUANTITY: 10,000/week maximum - 250,000 estimated total

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchased items - 5¢ each	$12,500

SHORT RUN MOLDING METHOD:

Tool cost - 80 hrs x $10/hr + material	$ 1,050
Molding cost - 320 parts/hour (2 operators) = $10 / 320 = 3¢ each	7,500
	$ 8,550
Estimated savings:	$ 3,950

ENGINEERING DETAIL

MATERIAL SELECTION: A.B.S.

Reason: Good flow, good appearance, good electrical resistance

TOOL: DME, steel base 5 x 6 - "U"

Tool steel inserts (extra set for loading pins)

Base plate is toe clamped to table platen

Knock out rods activate ejector plate

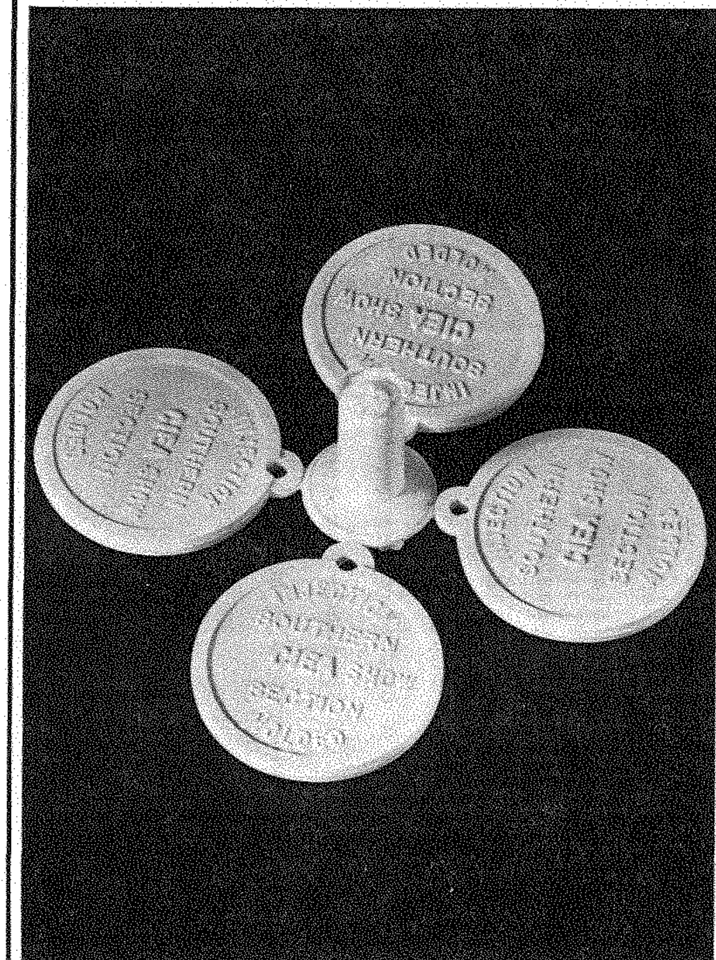
EXPECTED TOOL LIFE: 500,000 parts

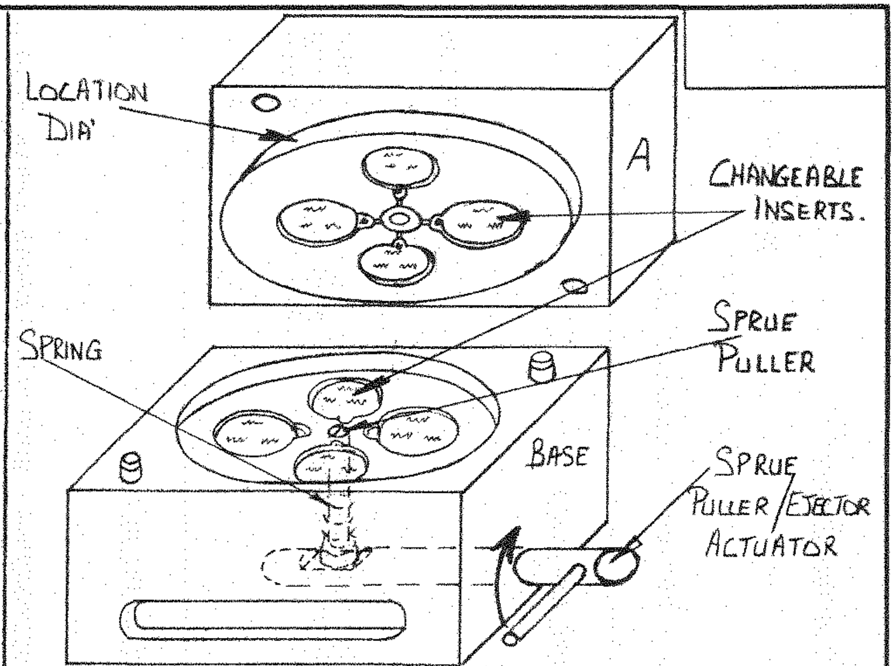
PRODUCTION RATE: 80 shots/hour

1 operator for equipment

1 operator for loading pin inserts

EQUIPMENT: Morgan-Press G-100T with upper platen assembly and "B" ball nozzle





(a) SHOOT W/ BALL SHUT-OFF NOZZLE
(b) TABLE FALLS & OPENS MOLD - PARTS STAY IN BASE
(c) ROTATE LEVER - SPRUE IS FREED - PARTS EJECTED.

2 PLATE SEMI-AUTOMATIC - 4 CAVITY		
SCALE: 1SO' / DATE:	APPROVED BY	DRAWN BY RM
ALUM' 2024-T4 - 5/8 DRILL ROD ACTUATOR.		
KEY RING TAG POLYPROPYLENE		DRAWING NUMBER D-3

PART REQUIREMENT

DESCRIPTION: Key Ring Tag NUMBER: D-3

MATERIAL: Polypropylene

QUANTITY: 100,000 parts

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Not applicable

SHORT RUN MOLDING METHOD:

Tool cost - 44 hours x $10/hr + material $ 475
Molding cost - 180 shots/hr = 720 parts/hr x $5/hr

Estimated part cost 1.2 cents

ENGINEERING DETAIL

MATERIAL SELECTION: Polypropylene

Reason: Molds at average temperature of 450°F, flows easily, is noncorrosive, can be cooled and reheated in barrel without degrading

TOOL: Aluminum 2024-T4 with aluminum inserts (8) which are quickly changeable

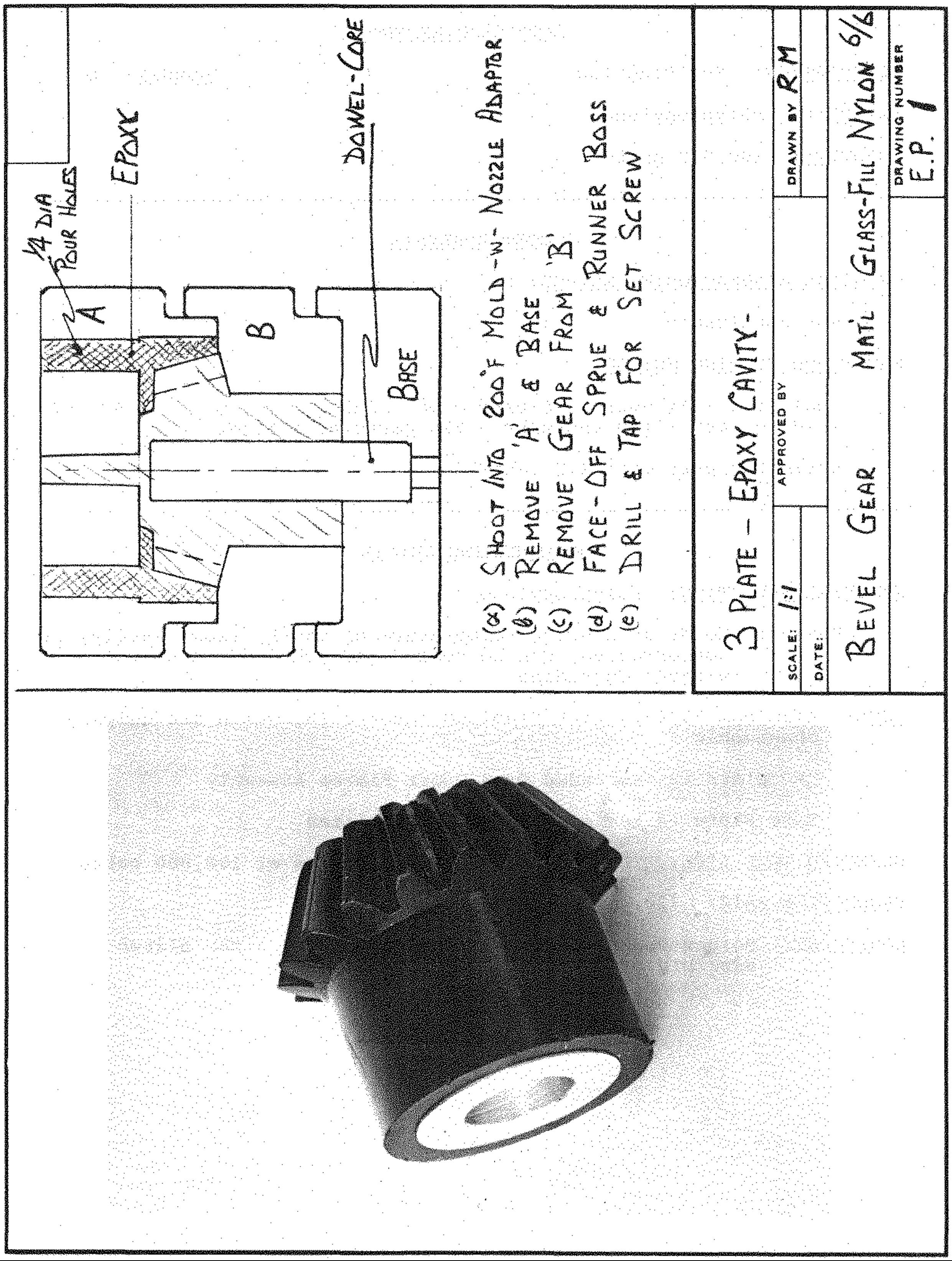
"A" plate is suspended from Upper Platen Assembly

Base Plate is toe clamped to table platen

EXPECTED TOOL LIFE: This mold produced in excess of 100,000 parts

PRODUCTION RATE: 720 parts/hour

EQUIPMENT: Morgan-Press G-100T with heater plate, upper platen assembly and anti-drool ball nozzle


1/4 DIA POUR HOLES
EPOXY
DOWEL-CORE
A
B
BASE
(a) SHOOT INTO 200°F MOLD-W-NOZZLE ADAPTOR
(b) REMOVE 'A' & BASE
(c) REMOVE GEAR FROM 'B'
(d) FACE-OFF SPRUE & RUNNER BOSS
(e) DRILL & TAP FOR SET SCREW
3 PLATE – EPOXY CAVITY-
SCALE: 1:1
DATE:
APPROVED BY
DRAWN BY R M
BEVEL GEAR
MAT'L GLASS-FILL NYLON 6/6
DRAWING NUMBER E.P. 1

PART REQUIREMENT

DESCRIPTION: Bevel Gear NUMBER: EP-1

MATERIAL: 30% Glass filled Nylon 6/6

QUANTITY: 100 pieces a year

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Purchase standard steel gear and modify hub size	
Cost - 100 pieces	$ 710
Lead time - 3 weeks	

SHORT RUN MOLDING METHOD:

Tool cost - mold base machining - 4 hrs x $10/hr	$ 40
Epoxy Cavity - use existing gear as pattern - 4 hrs	48
Molding cost - 5 hours x $5/hr + material	35
Lead time - 1 week	$ 123
Estimated savings: 2 weeks and	$ 587

ENGINEERING DETAIL

MATERIAL SELECTION: Glass filled nylon 30%

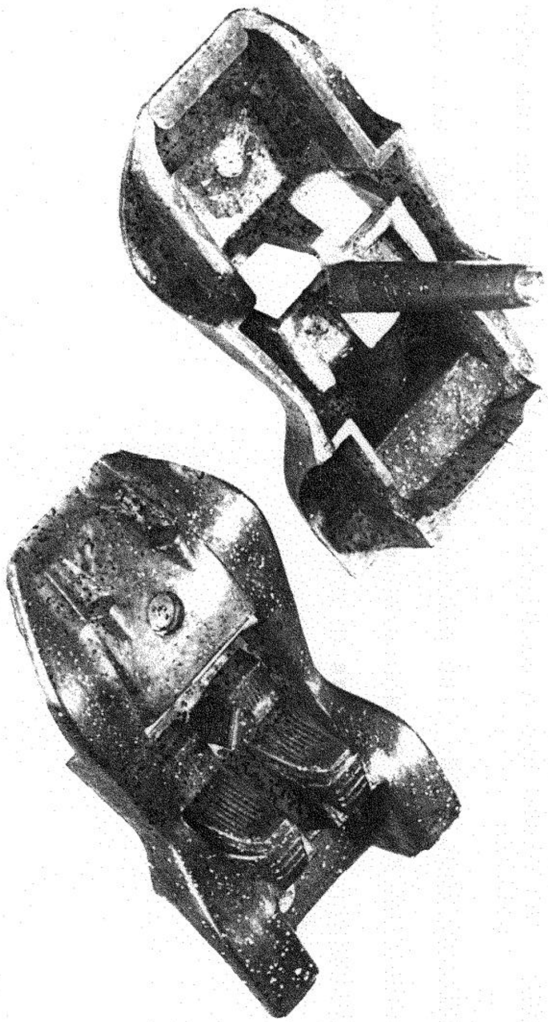
Reason: Low speed, moderate torque application, intermittent service

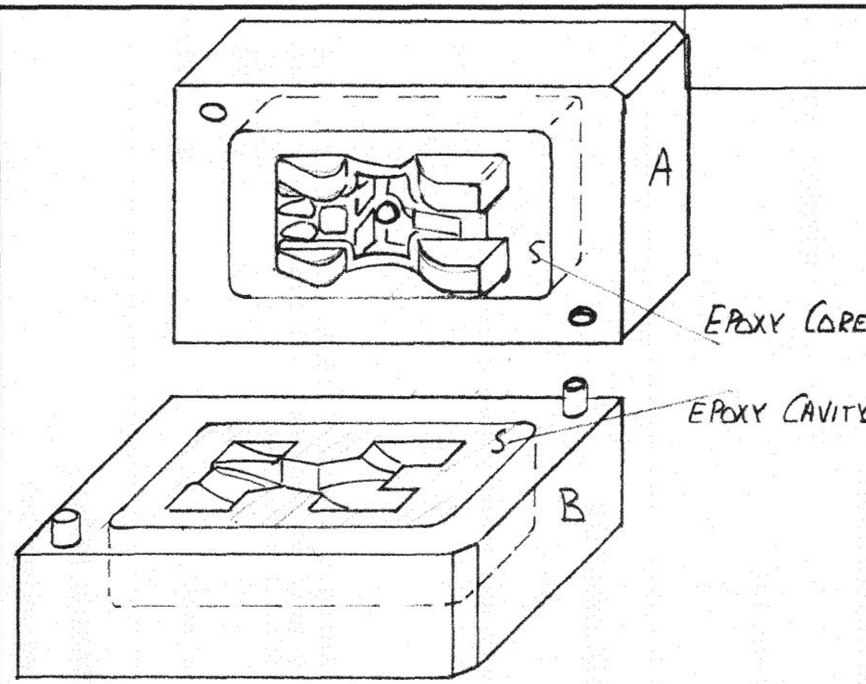
TOOL: "A" Plate has a machined cavity 1/4" larger than the gear pattern

EXPECTED TOOL LIFE: 100 pieces minimum - can be recast in 4 hours

PRODUCTION RATE: 20 shots/hour - by hand

EQUIPMENT: Morgan-Press G-55T with nozzle adaptor, "B" nozzle and heater plate





(a) SHOOT USING NOZZLE ADAPTOR
(b) OPEN MOLD - PART STAYS ON CORE "A"
(c) ROTATE NOZZLE ADAPTOR - BREAK SPRUE
(d) TAP SPRUE TO REMOVE PART FROM CORE

2 PLATE - EPOXY CORE & CAVITY		
SCALE: ISO'	APPROVED BY	DRAWN BY PM
DATE: 2 JUNE 71		
DUNE BUGGY - SCALE MODEL		
		DRAWING NUMBER E.P. - 2

PART REQUIREMENT

DESCRIPTION: Model for sales use - Dune Buggy NUMBER: EP-2

MATERIAL: Tinsel filled - dry colored, polypropylene

QUANTITY: 250 parts total (50 each - 5 colors)

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Wood pattern (4 x full size) - cast plasters male and female, pantograph mold halves	
Estimated cost, tooling only	$ 1,300
Lead time - 4 weeks	

SHORT RUN MOLDING METHOD:

Plastic model (full size), 2 cavity pockets in aluminum blocks	
Epoxy core and cavity	
Tool cost - 36 hrs x $10/hr + material	$ 390
Lead time - 2 weeks	
Estimated savings: 2 weeks and	$ 910

ENGINEERING DETAIL

MATERIAL SELECTION: Polypropylene (natural, dry color and tinsel flakes)

Reason: Low cost, translucent, easy to mold

TOOL: "A" Plate is aluminum pocketed 1/4" deep

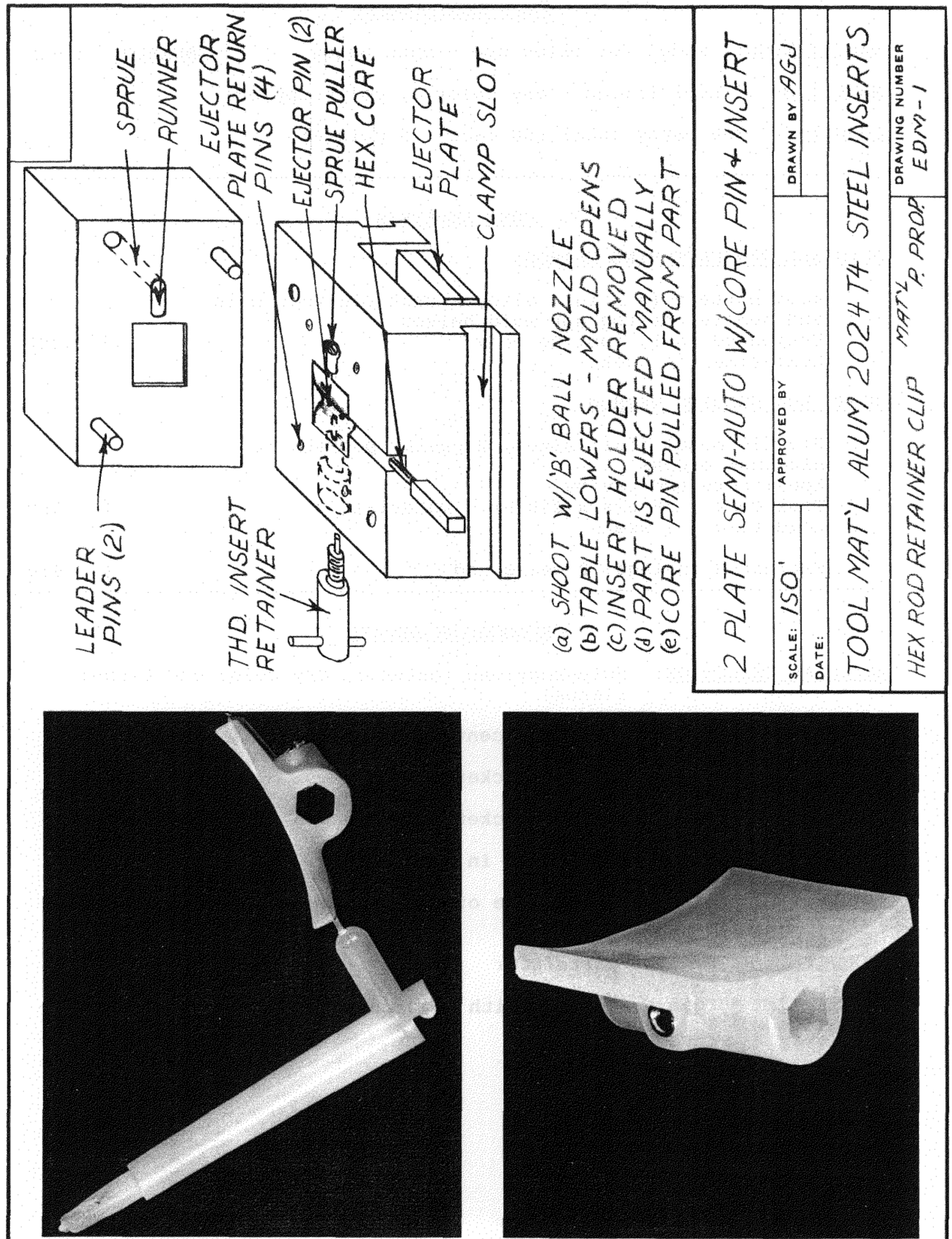
"B" plate is aluminum pocketed 1/2" deep

Cavity and core are cast in Devcon "C"

EXPECTED TOOL LIFE: Actual life of mold exceeded 1,200 parts during test

PRODUCTION RATE: 100 parts/hour

EQUIPMENT: Morgan-Press G-55 with nozzle adapter


SPRUE
RUNNER
EJECTOR PLATE RETURN PINS (4)
EJECTOR PIN (2)
SPRUE PULLER
HEX CORE
EJECTOR PLATE
CLAMP SLOT
LEADER PINS (2)
THD. INSERT RETAINER
(a) SHOOT W/'B' BALL NOZZLE
(b) TABLE LOWERS - MOLD OPENS
(c) INSERT HOLDER REMOVED
(d) PART IS EJECTED MANUALLY
(e) CORE PIN PULLED FROM PART
2 PLATE SEMI-AUTO W/CORE PIN + INSERT
SCALE: ISO'
APPROVED BY
DRAWN BY AGJ
DATE:
TOOL MAT'L ALUM 2024 T4 STEEL INSERTS
HEX ROD RETAINER CLIP
MAT'L P. PROP.
DRAWING NUMBER
EDM-1

PART REQUIREMENT

DESCRIPTION: Hex Rod Retainer Clip NUMBER: EDM-1

MATERIAL: Polypropylene

QUANTITY: Needed 100 pieces for market testing and then 2500/year

---

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Project not economically feasible if parts are machined in plastic or metal

SHORT RUN MOLDING METHOD:

Tool cost - Machine electrode from EDM grade graphite;
make mold base and EDM cavities
60 hrs x $24/hr + material $ 1,600
Molding cost - 2 hours x $12/hr

Estimated part cost over 3 year period is 15¢ each

ENGINEERING DETAIL

MATERIAL SELECTION: Polypropylene

Reason: Easily molded an mating parts or finished assembly are polypropylene

TOOL: Aluminum 7075-T6 with tool steel inserts and core pins

"A" Plate is suspended from Upper Platen Assembly

Base plate is toe clamped to table platen

Ejector plate is manually operated

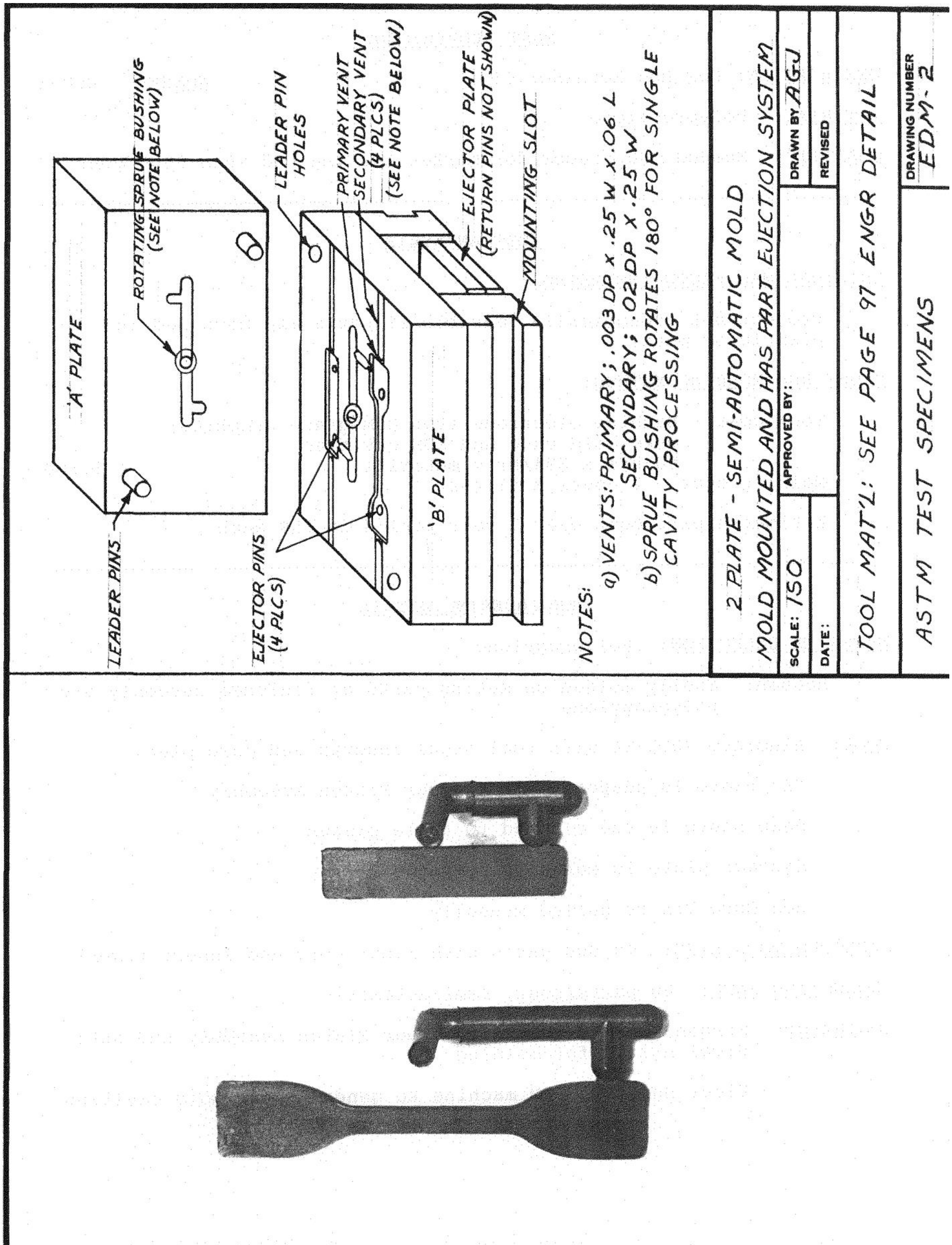
Hex Core Pin is pulled manually

EXPECTED TOOL LIFE: 10,000 parts with minor core and insert rework

PRODUCTION RATE: 80 parts/hour, semi-automatic

EQUIPMENT: Morgan-Press G-100T with Upper Platen Assembly and anti-drool nozzle for molding

Elcut Super 25 EDM machine to generate the mold cavities


'A' PLATE
ROTATING SPRUE BUSHING (SEE NOTE BELOW)
LEADER PINS
LEADER PIN HOLES
PRIMARY VENT
SECONDARY VENT (4 PLCS) (SEE NOTE BELOW)
EJECTOR PLATE (RETURN PINS NOT SHOWN)
MOUNTING SLOT
'B' PLATE
EJECTOR PINS (4 PLCS)
NOTES:
a) VENTS: PRIMARY; .003 DP x .25 W x .06 L
SECONDARY; .02 DP X .25 W
b) SPRUE BUSHING ROTATES 180° FOR SINGLE CAVITY PROCESSING
2 PLATE - SEMI-AUTOMATIC MOLD
MOLD MOUNTED AND HAS PART EJECTION SYSTEM
SCALE: ISO
APPROVED BY:
DRAWN BY AGJ
DATE:
REVISED
TOOL MAT'L: SEE PAGE 97 ENGR DETAIL
ASTM TEST SPECIMENS
DRAWING NUMBER EDM-2

PART REQUIREMENT

DESCRIPTION: Tensile and Impact Test Specimens NUMBER: EDM-2

MATERIAL: Any conventional thermoplastic

QUANTITY: Varies depending on data requirements

COST ANALYSIS

ORIGINAL MANUFACTURING METHOD:

Not applicable

SHORT RUN MOLDING METHOD:

Tool cost - 40 hrs x $30/hr + material $ 1,800
Molding cost - 30 parts/hr x $6.00 + material
Estimated part cost 25¢ each

ENGINEERING DETAIL

APPLICATIONS:

Tensile and Impact Test Specimens are used throughout the plastic industry for material quality and process control tests. Tensile and Impact test data are used in the development and improvement of thermoplastic materials.

TOOL: Tool Steel mold base and standard mold components

"A" Plate is suspended from Upper Platen Assembly

Base Plate is toe clamped to table platen

Knock out rods activate ejector plate

Sprue bushing rotates to shoot one cavity at a time

Tensile cavity machined by CNC program or EDM electrode

EXPECTED TOOL LIFE: 100,000 shots or more

PRODUCTION RATE: 30/hour - semi-automatic

EQUIPMENT: Morgan-Press G-55T with Upper Platen Assembly, Heater Plate and Shut-Off Nozzle

Elcut Super 25 EDM machine to generate the tensile specimen cavity

CHAPTER SIX

MATERIAL HANDLING

Each family of thermoplastic materials has its own processing characteristics. To switch from one compound to another may require more than merely readjusting temperatures and pressures. Proper drying (essential for most thermoplastics) and other handling requirements must also be observed. Some polymers not properly processed present hazards to the work environment. Always begin by following the instructions and suggestions of the manufacturer of the material.

When experimenting with a new material or an unfamiliar grade, use an open orifice "B" nozzle first. Establish a good extrusion and note the temperatures. This requires that the material be allowed to drool freely. Once these conditions have been established, a shut-off nozzle can be used to eliminate drooling.

In general, material such as vinyl, urethane, polypropylene, nylon, and acetals which drool rapidly, will require low injection pressures (2,000 - 4,000 psi) providing that the gates are adequate in size and the mold temperatures are correct. Materials which drool more slowly such as ABS, phenylene oxide, and polycarbonate will tend to need slightly higher injection pressures (4,000 - 6,000 psi) and larger gates. Heavily filled materials may not drool at all and will require pressure from the ram to extrude from the nozzle. Pressures of 7,000-9,000 psi may be needed to produce good parts. In addition, at each pressure level, the ram speed should be adjusted as needed. If pressures significantly higher than indicated here are required, it is most probable that the runners and gates are too small and mold venting is inadequate. Excess injection pressure can actually pre-stress molded parts and cause them to be inherently defective. Eliminating mold design deficiencies will ensure better molding results at the correct temperatures and pressures. Typical problems encountered when molding are as follows:

NYLON

The mold will not fill correctly.

The temperature range at which nylon will flow correctly is rather narrow. A nylon which will drool readily at 580°F may be hard or impossible to move at 570°F. At a sustained 600°F, the material will gas, blister, discolor, and flow only as a froth. It is essential with nylon to establish the correct extrusion temperature and to maintain it accurately. Shoot parts at a constant rate, and do not attempt to correct short shots by overheating the material. When in contact with the mold, the nozzle temperature will drop rapidly and the nylon in it will chill instantly; so clamp and inject as quickly as possible into a good hot mold.

After shooting, the nozzle will almost invariably be frozen. Wait until it starts to drool again before attempting to shoot another part. If the time required to unfreeze the nozzle is more than two minutes, the nozzle temperature could be raised by 10°F. With the shut-off nozzle, wait until the frozen plug drops free before reshooting.

ABS

Surface color is blemished.

This material has a wide temperature molding range. The problems usually encountered are with surface finish, especially with the dark colors. When the material is plasticized rapidly, the temperature of the extrusion will vary as colder spots, from the center of the cylinder, mix with material nearer the cylinder walls. When injected, these cold spots tend to show up as streaks on the surface.

Always thoroughly dry ABS and pre-heat it for an hour before molding to raise its temperature evenly. In this way even the centers of the granules are heated and the chance of cold spotting will be reduced.

Overheated ABS will also discolor, most noticeably in the lighter colors. When grossly overheated or retained at high heat longer than necessary, the ABS will tend to resolidify and both surface gloss and physical strengths will suffer. ABS compounds tend to gas when heated, molds should be well vented.

ACETALS

Parts are undersize and surface finish is wrinkled.

Acetals have the highest shrinkage rates of all thermoplastic materials. To minimize the effects of shrinkage, parts should be shot as fast as possible, through gates as large as possible into molds heated to 250°F. The injection pressure should be maintained until the gates have chilled, usually three to five seconds after the fill is complete. Acetals are lubricated by the addition of chemicals and only light quantities of mold release are needed on cores or inserts.

Acetal polymers must be handled carefully to prevent accidents. For example, the formaldehyde generated by heating acetals is harmful to eyes and lungs. Good room ventilation is recommended when molding these compounds. This material tends to gas freely; therefore molds should be extensively vented or back pressure will slow the filling. Again thorough drying of this hygroscopic material just prior to its use will substantially minimize gassing and enhance ease of processing.

* * *

The chart on the opposite page summarizes most trouble shooting measures for injection molding.

TROUBLE SHOOTING FOR INJECTION MOLDING

Short Shots or Surface Wrinkles -

- Increase ram speed
- Increase injection pressure
- Increase mold temperature
- Increase ram pressure time
- Increase runners or gate size
- Increase mold venting
- Increase time between shots

Sinks -

- Increase ram pressure time
- Increase ram speed
- Enlarge gate
- Reduce material temperature
- Increase mold venting

Discoloring -

- Decrease material temperature
- Reduce time between shots
- Thoroughly dry material
- Thoroughly preheat material

Gassing -

- Decrease material temperature
- Thoroughly dry material
- Preheat material thoroughly

Weld Marks -

- Increase mold venting
- Increase material temperature
- Increase gate size
- Move gate or use multiple gate
- Increase injection speed and pressure
- Add overflow puddle

Warping of Parts -

- Reduce mold temperature
- Increase ram pressure time
- Increase time between shots
- Reduce material temperature
- Check part design for section variations

Part too Small -

- Increase injection time
- Increase injection pressure
- Reduce material temperature
- Enlarge gate

Part too Large -

- Reduce injection pressure
- Decrease injection time

Dimension Variation Shot to Shot -

- Establish and maintain cycle time
- Keep mold temperature constant
- Maintain constant material temperature
- Maintain constant injection pressure
- Increase mold venting

Surface Streaking -

- Raise mold temperature
- Thoroughly preheat material
- Thoroughly dry material
- Reduce injection rate

Relationship of Clamp to Injection Pressures

The clamp force holding the mold closed must exceed the total internal pressure to be generated within the mold cavity or the mold will open (flash) during injection.

FOR EXAMPLE: assume an injection pressure of 5,000 psi, a mold with 3 square inches of cavity area and a viscosity factor* of .7: the internal pressure = 5,000 psi x 3 square inches x .7 = 10,500 pounds, and thus the selected clamp force must exceed 5 1/4 tons.

If the internal cavity pressure of a specific mold design should exceed the 8 tons or 20 tons clamp force applied by the Morgan-Press, the mold halves can be bolted together. (See pages 48 & 50).

A good rule is to start with minimum calculated clamp force and low injection pressure and increase the injection pressure and/or ram speed after each cycle until the mold is properly filled. When proper injection settings are attained, the clamp force should be set at the lowest tonnage necessary to keep the mold shut. This minimizes mold and equipment fatigue.

APPENDIX I

DRAFT ANGLE TABLE

(ANGLE PER SIDE)

DEPTH	½°	1°	2°	3°	4°	5°	6°	7°	8°	9°	10°	11°	12°	15°	20°	25°	30°	DEPTH
1/32	.0003	.0005	.001	.0016	.002	.0027	.003	.0038	.004	.005	.0055	.006	.0066	.008	.011	.014	.018	1/32
1/16	.0005	.0011	.002	.0033	.004	.0055	.007	.0077	.009	.010	.011	.012	.013	.017	.023	.029	.036	1/16
3/32	.0008	.0017	.003	.0049	.006	.008	.010	.0115	.013	.015	.0165	.018	.020	.025	.034	.044	.054	3/32
1/8	.0011	.0022	.004	.0066	.009	.0109	.013	.015	.018	.020	.022	.024	.027	.033	.045	.058	.072	1/8
5/32	.0014	.0028	.005	.008	.011	.014	.016	.019	.022	.025	.027	.030	.033	.042	.057	.073	.089	5/32
3/16	.0016	.0033	.006	.0098	.013	.016	.019	.023	.027	.030	.033	.036	.040	.050	.068	.087	.108	3/16
7/32	.0019	.0039	.008	.011	.015	.019	.023	.027	.031	.035	.039	.042	.047	.059	.080	.102	.126	7/32
1/4	.0022	.0043	.009	.013	.018	.022	.026	.031	.035	.040	.044	.049	.053	.067	.091	.117	.144	1/4
9/32	.0025	.005	.010	.014	.020	.024	.030	.034	.040	.045	.049	.055	.060	.075	.102	.131	.162	9/32
5/16	.0027	.0055	.011	.016	.022	.027	.033	.038	.044	.050	.055	.061	.066	.084	.114	.146	.180	5/16
11/32	.003	.006	.012	.018	.024	.030	.036	.042	.049	.055	.061	.067	.073	.092	.125	.160	.197	11/32
3/8	.0033	.0066	.013	.020	.026	.033	.039	.046	.053	.060	.066	.073	.080	.100	.136	.175	.217	3/8
13/32	.0035	.007	.014	.021	.028	.035	.043	.050	.057	.064	.071	.079	.086	.108	.148	.189	.234	13/32
7/16	.0038	.0077	.015	.023	.031	.038	.046	.054	.062	.069	.077	.085	.093	.117	.159	.204	.253	7/16
15/32	.0041	.008	.016	.024	.033	.041	.049	.058	.066	.074	.083	.091	.100	.126	.171	.219	.270	15/32
1/2	.0044	.0088	.018	.026	.035	.044	.053	.061	.071	.079	.088	.097	.106	.134	.182	.233	.289	1/2
17/32	.0046	.009	.019	.028	.037	.046	.056	.065	.075	.084	.093	.103	.113	.142	.193	.247	.306	17/32
9/16	.0049	.0099	.020	.030	.039	.049	.059	.069	.079	.088	.099	.109	.120	.151	.205	.262	.325	9/16
19/32	.0052	.010	.021	.031	.042	.052	.062	.073	.084	.094	.105	.115	.127	.159	.216	.277	.343	19/32
5/8	.0055	.011	.022	.033	.044	.055	.066	.077	.088	.100	.110	.120	.133	.167	.227	.291	.361	5/8
21/32	.0057	.011	.023	.035	.046	.057	.069	.082	.092	.104	.115	.127	.140	.176	.239	.306	.379	21/32
11/16	.006	.012	.024	.036	.048	.060	.072	.085	.096	.109	.121	.133	.147	.183	.250	.321	.397	11/16
23/32	.0063	.013	.025	.038	.050	.063	.075	.088	.101	.114	.126	.139	.153	.193	.261	.335	.415	23/32
3/4	.0065	.013	.027	.039	.053	.066	.079	.092	.106	.119	.132	.146	.159	.201	.273	.350	.433	3/4
25/32	.0068	.014	.028	.040	.055	.068	.081	.096	.110	.124	.137	.152	.166	.209	.284	.364	.451	25/32
13/16	.0071	.014	.029	.043	.057	.071	.085	.100	.115	.129	.143	.158	.173	.218	.296	.379	.469	13/16
27/32	.0074	.015	.030	.045	.059	.074	.089	.104	.119	.134	.149	.163	.180	.226	.307	.393	.487	27/32
7/8	.0076	.015	.031	.046	.061	.077	.092	.107	.123	.139	.154	.171	.186	.234	.318	.408	.505	7/8
29/32	.0079	.016	.032	.048	.063	.079	.095	.111	.128	.144	.159	.176	.193	.243	.329	.422	.523	29/32
15/16	.0082	.017	.033	.050	.066	.082	.098	.115	.132	.149	.165	.182	.200	.251	.341	.437	.541	15/16
31/32	.0085	.017	.034	.051	.068	.084	.101	.119	.137	.153	.170	.188	.206	.260	.353	.452	.559	31/32
1	.0087	.0175	.035	.052	.070	.087	.105	.123	.141	.158	.176	.194	.213	.268	.364	.466	.577	1
	½°	1°	2°	3°	4°	5°	6°	7°	8°	9°	10°	11°	12°	15°	20°	25°	30°	

APPENDIX II

SHRINKAGE FACTOR TABLE

INCHES PER INCH

FRACT.	DECIMAL EQUIV.	.003	.004	.005	.006	.007	.008	.009	.010	.011	.012	.013	.014	.015	.016	.017	.018	.019	.020	.021	.022	.023	.024	.025	FRACT.
1/32	.03125	.0001	.0001	.0002	.0002	.0002	.0003	.0003	.0003	.0003	.0004	.0004	.0004	.0005	.0005	.0005	.0006	.0006	.0006	.0007	.0007	.0007	.0008	.0008	1/32
1/16	.0625	.0002	.0003	.0003	.0004	.0004	.0005	.0006	.0006	.0007	.0008	.0008	.0009	.0009	.0010	.0011	.0011	.0012	.0013	.0013	.0014	.0014	.0015	.0016	1/16
3/32	.09375	.0003	.0004	.0005	.0006	.0007	.0008	.0008	.0009	.0010	.0011	.0012	.0013	.0014	.0015	.0016	.0017	.0018	.0019	.0020	.0021	.0022	.0023	.0023	3/32
1/8	.125	.0004	.0005	.0006	.0008	.0009	.0010	.0011	.0013	.0014	.0015	.0016	.0018	.0019	.0020	.0021	.0023	.0024	.0025	.0026	.0028	.0029	.0030	.0031	1/8
5/32	.15625	.0005	.0006	.0008	.0009	.0011	.0012	.0014	.0016	.0017	.0019	.0020	.0022	.0023	.0025	.0027	.0028	.0030	.0031	.0033	.0034	.0036	.0038	.0039	5/32
3/16	.1875	.0006	.0008	.0009	.0011	.0013	.0015	.0017	.0019	.0021	.0023	.0024	.0026	.0028	.0030	.0032	.0034	.0036	.0038	.0039	.0041	.0043	.0045	.0047	3/16
7/32	.21875	.0007	.0009	.0011	.0013	.0015	.0018	.0020	.0022	.0024	.0026	.0028	.0031	.0033	.0035	.0037	.0039	.0042	.0044	.0046	.0048	.0050	.0053	.0055	7/32
1/4	.250	.0008	.0010	.0013	.0015	.0018	.0020	.0023	.0025	.0028	.0030	.0033	.0035	.0038	.0040	.0043	.0045	.0048	.0050	.0053	.0055	.0058	.0060	.0063	1/4
9/32	.28125	.0008	.0011	.0014	.0017	.0020	.0023	.0025	.0028	.0031	.0034	.0037	.0039	.0042	.0045	.0048	.0051	.0053	.0056	.0059	.0062	.0065	.0068	.0070	9/32
5/16	.3125	.0009	.0013	.0016	.0019	.0022	.0025	.0028	.0031	.0034	.0038	.0041	.0044	.0047	.0050	.0053	.0056	.0059	.0063	.0066	.0069	.0072	.0075	.0078	5/16
11/32	.34375	.0010	.0014	.0017	.0021	.0024	.0028	.0031	.0034	.0038	.0041	.0045	.0048	.0052	.0055	.0058	.0062	.0065	.0069	.0072	.0076	.0079	.0083	.0086	11/32
3/8	.375	.0011	.0015	.0018	.0023	.0026	.0030	.0034	.0038	.0041	.0045	.0049	.0053	.0056	.0060	.0064	.0068	.0071	.0075	.0079	.0083	.0086	.0090	.0094	3/8
13/32	.40625	.0012	.0016	.0020	.0024	.0028	.0033	.0037	.0041	.0045	.0049	.0053	.0057	.0061	.0065	.0069	.0073	.0077	.0081	.0085	.0089	.0093	.0098	.0102	13/32
7/16	.4375	.0013	.0018	.0022	.0026	.0031	.0035	.0039	.0044	.0048	.0053	.0057	.0061	.0066	.0070	.0074	.0079	.0083	.0088	.0092	.0096	.0101	.0105	.0109	7/16
15/32	.46875	.0014	.0019	.0023	.0028	.0033	.0038	.0042	.0047	.0052	.0056	.0061	.0066	.0070	.0075	.0080	.0084	.0089	.0094	.0098	.0103	.0108	.0113	.0117	15/32
1/2	.500	.0015	.0020	.0025	.0030	.0035	.0040	.0045	.0050	.0055	.0060	.0065	.0070	.0075	.0080	.0085	.0090	.0095	.0100	.0105	.0110	.0115	.0120	.0125	1/2
17/32	.53125	.0016	.0021	.0026	.0032	.0037	.0043	.0048	.0053	.0058	.0064	.0069	.0074	.0080	.0085	.0090	.0096	.0101	.0106	.0112	.0117	.0122	.0128	.0133	17/32
9/16	.5625	.0017	.0023	.0028	.0034	.0033	.0045	.0051	.0056	.0062	.0068	.0073	.0079	.0084	.0090	.0096	.0101	.0107	.0113	.0118	.0124	.0129	.0135	.0141	9/16
19/32	.59375	.0018	.0024	.0029	.0036	.0042	.0048	.0053	.0059	.0065	.0071	.0077	.0083	.0089	.0095	.0100	.0107	.0113	.0119	.0125	.0131	.0137	.0143	.0148	19/32
5/8	.625	.0019	.0025	.0031	.0038	.0044	.0050	.0056	.0063	.0069	.0075	.0081	.0088	.0094	.0100	.0106	.0113	.0119	.0125	.0131	.0138	.0144	.0150	.0156	5/8
21/32	.65625	.0020	.0026	.0033	.0039	.0046	.0053	.0059	.0066	.0072	.0079	.0085	.0092	.0098	.0105	.0112	.0118	.0125	.0131	.0138	.0144	.0151	.0158	.0164	21/32
11/16	.6875	.0021	.0028	.0034	.0041	.0048	.0055	.0062	.0069	.0076	.0083	.0089	.0096	.0103	.0110	.0117	.0124	.0131	.0138	.0144	.0151	.0158	.0165	.0172	11/16
23/32	.71875	.0022	.0029	.0036	.0043	.0050	.0058	.0065	.0072	.0079	.0086	.0093	.0101	.0108	.0115	.0122	.0129	.0137	.0144	.0151	.0158	.0165	.0173	.0180	23/32
3/4	.750	.0023	.0030	.0038	.0045	.0053	.0060	.0068	.0075	.0083	.0090	.0098	.0105	.0113	.0120	.0128	.0135	.0143	.0150	.0158	.0165	.0173	.0180	.0188	3/4
25/32	.78125	.0023	.0031	.0039	.0047	.0055	.0063	.0070	.0078	.0086	.0094	.0101	.0109	.0117	.0125	.0133	.0141	.0148	.0156	.0164	.0172	.0180	.0188	.0195	25/32
13/16	.8125	.0024	.0033	.0040	.0049	.0057	.0065	.0073	.0081	.0089	.0098	.0106	.0114	.0122	.0130	.0138	.0146	.0154	.0163	.0171	.0179	.0187	.0195	.0203	13/16
27/32	.84375	.0025	.0034	.0042	.0051	.0059	.0068	.0076	.0084	.0093	.0101	.0110	.0118	.0127	.0135	.0143	.0152	.0160	.0169	.0177	.0186	.0194	.0203	.0211	27/32
7/8	.875	.0026	.0035	.0044	.0053	.0061	.0070	.0079	.0088	.0096	.0105	.0113	.0123	.0131	.0140	.0149	.0158	.0166	.0175	.0184	.0193	.0201	.0210	.0219	7/8
29/32	.90625	.0027	.0036	.0045	.0054	.0063	.0073	.0082	.0091	.0100	.0109	.0119	.0127	.0136	.0145	.0154	.0163	.0172	.0181	.0190	.0199	.0208	.0218	.0227	29/32
15/16	.9375	.0028	.0038	.0047	.0056	.0066	.0075	.0084	.0094	.0103	.0113	.0122	.0131	.0141	.0150	.0159	.0169	.0178	.0188	.0197	.0206	.0216	.0225	.0234	15/16
31/32	.96875	.0029	.0039	.0048	.0058	.0068	.0078	.0087	.0097	.0107	.0116	.0126	.0136	.0145	.0155	.0165	.0174	.0184	.0194	.0203	.0213	.0223	.0233	.0242	31/32
1"	1.000	.0030	.0040	.0050	.0060	.0070	.0080	.0090	.0100	.0110	.0120	.0130	.0140	.0150	.0160	.0170	.0180	.0190	.0200	.0210	.0220	.0230	.0240	.0250	1"

PLASTIC PROPERTIES CHART

Material	Molding Quality	Inj. Mold Temps	Mold Temp.	Machine Quality	Linear Shrinkage In./In.	Specific Gravity (Density)	Cont. Temp. Resist.	Burn Rate
ABS(Acrylonitrile-Butadiene-Styrene	G-E	450-520	180	G-E	.005-.008	1.01-1.21	140-230	Slow
Acetals	Exc.	400-450	250	F-E	.013-.028	1.41-1.61	185-220	Slow
Acrylics	Exc.	375-500	150	G-E	.001-.008	1.11-1.20	140-200	Slow
Ethyl Cellulose	Exc.	400-500	180	Good	.005-.009	1.09-1.17	115-185	Slow
Cellulose Acetate	Exc.	375-490	150	Exc.	.003-.008	1.22-1.34	140-220	Slow
Cellulose Acetate Butyrate	Exc.	400-480	180	Exc.	.003-.006	1.15-1.22	140-220	Slow
Fluoroplastics	Exc.	550-760	250	Exc.	.01-.06	1.75-2.20	300-550	None
Nylon	Exc.	480-650	200	Exc.	.002-.015	1.01-1.37	175-300	*
Phenylene Oxides	Exc.	525-650	250	Exc.	.001-.006	1.06-1.36	175-265	Self-ext.
Poly-allomer	Exc.	430-485	120	Good	.01-.02	.896-.899	-	Slow
Polysulfone	Exc.	600-700	250	Exc.	.007	1.24	300-400	*
Poly-butylene	Good	290-380	120	Poor	.003 .026 aged	.910-.915	225	-
Polycarbonates	G-E	480-650	250	F-E	.001-.009	1.10-1.52	220-275	*
Polyesters	Good	500-540	180	Exc.	.018	1.37-1.38	175-250	Slow
Polyethylenes	Exc.	250-500	110	G-E	.007-.050	.910-1.45	180-275	V-Slow
Polypropylenes	Exc.	400-550	120	Good	.009-.025	.890-.906	190-300	Slow
Polystyrenes	Exc.	425-600	120	F-G	.001-.006	1.04-1.33	140-200	Slow
Urethanes	G-E	320-480	120	F-E	.009-.030	1.11-1.25	190	*
Vinyl Polymers & Copolymers	G-E	300-425	100	Good	.001-.050	1.05-1.72	120-200	*

All Temperatures are degrees F.

* - Slow to Extinguishing

PLASTIC PROPERTIES CHART

Bonding Agents	Moisture Absorp. Rate	Drying Reqmt.	Clarity Natural Material	Use of Shut-off Nozzle	Trade Name	& Manufacturer	Cost/lb. (50 lb. Quantity)
Mc, MEK	Med.	180 2 hrs.	Tl to Op	Option	Cycolac Lustran	Marbon Chem. Monsanto	1.25 - 3.00
None	Low	190 4 hrs.	Op	Must	Delrin Celcon	DuPont Celanese Plas.	3.00 - 5.25
Mc, MEK	Med.	180 2 hrs.	Tp to Op	Option	Implex	Rohm & Haas	2.50
MEK	High	150 24 hrs.	Tp to Op	Option	Tenite	Eastman Chem.	2.50 - 3.50
MEK	V-High	180 24 hrs.	Tp to Op	Option	Tenite Celanese	Eastman Chem. Celanese Plas.	3.00 - 4.00
Ketones	V-High	180 24 hrs.	Tp to Op	Must	Tenite	Eastman Chem.	3.00 - 4.00
None	V-Low	None	Tp to Tl	Not Rec	Teflon	DuPont	13.00 - 18.00
Phenol	High	200 4 hrs.	Tl to Op	Must	Zytel Plaskon	DuPont Allied Chem.	3.00 - 5.50
Aliphatics	Low	200 2 hrs.	Op	Option	Noryl	G.E. Company	2.15 - 3.52
None	V-Low	None	Tp to Tl	Must	Tenite	Eastman Chem.	3.50 - 5.00
Mc	Low	275 4 hrs.	Tp	Option	Udel	Union Carbide	5.00 - 7.00
- -	V-Low	None	Tl	Option		Shell	2.00 - 3.00
Ethylene Dichloride	Low	200 4 hrs.	Tp to Op	Must	Lexan Merlon	G.E. Company Mobay Chem.	2.80 - 4.50
Halogen	V-Low	200 2 hrs.	Op	Option	Celanex Valox	Celanese Plas. G.E. Company	2.75 - 4.50
None	V-Low	None	Tp to Op	Option	Enjay Marlex	Enjay Chem. Phillips Pet.	.65 - .75
None	V-Low	None	Tp to Op	Option	Pro-Fax	Hercules	.75 - 1.40
Mc, MEK	Med.	180 2 hrs.	Tp to Op	Option	Lustrex Styron	Monsanto Dow Chemical	.60 - 2.50
None	Med.	180 2 hrs.	Tp to Op Colors	Must	Roylar Texin	Uniroyal Chem. Mobay Chem.	4.00 - 6.00
T.H.F.	High	150 4 hrs.	Tp to Op	Must	Geon Vyram	Goodrich B.F. Monsanto	3.00 - 4.25

Mc - Methylene Chloride
MEK - Methyl Ethyl Ketone

Tl - Translucent
Tp - Transparent
Op - Opaque

PARTIAL CUSTOMER LIST BY PRINCIPAL APPLICATION

A. MODEL SHOP/PROTOTYPING/SAMPLING

Abbott Laboratories
Admiral Midwest Mfg. Div.
Ametek Div. U.S. Gauge
Amicon Corp
ARRK Creative Network
Aquamatic, Inc.
Bailey Fisher & Porter
Barber Colman Co.
Batesville Casket
Baxter Health Care
Beckman Instruments
Behr of America
Bell Sports
Calcomp
C. A. Norgren Co.
Carpenter Technology Co.
Chesebrough Ponds
Compression Inc.
Davol Inc
Dentsply International Corp.
Diebold Inc.
Dresser Mfg.
Eaton Corp.
EFE Services
Eli Lilly & Co.
Emerson Electric
General Electric
General Surgical
Glaxo Wellcome
Gould Electronics
GTE
Hasbro Inc.
Hewlett Packard
Hobbs Div. Stewart Warner
Homelite Div. Textron
Hosmer Dorrance Corp.
IBM Research
ITT Aerospace Controls
JPL, Cal Tech
Kirchner MTD
Lee Company
Leviton Mfg.
Littlefuse, Inc.
Merck & Co.
3M Co.
Motorola
Panduit Corp.
Paragon Electric
Parker Brothers
Physio Control Corp.
Puritan Equipment, Inc.
Reebok International
Revlon
Roadmaster Corp.
Ron-Vik Inc.
Siemens Corp.
Somnus Medical Technology
Teledyne Brown Engineering
Texas Instruments (2)
U.S. Air Force
U.S. Navy
United Technologies Electro
Valcor Engineering
Whirlpool Corp.
Xomed

B. CABLE/CONNECTOR/ENCAPSULATION/INSERT MOLDING PRODUCTION

AEM Corp.
Autotronic Controls Corp.
Gates & Sons, Inc.
Geco-Prakla
Geophysical Cable Corp.
Houston-Sigma Technology
Hydrolab Corp.
International Transducer Corp.
ITT Cannon Electric
Litton Resources
Lockheed Martin
M & H Electric
Mark Products
Merit Industries
Oyo Geospace Canada
Radarsonics, Inc.
Sentry Medical Products
U.S. Navy

C. GENERAL SHORT-RUN PRODUCTION

Amphenol Fiber Optics
Amistar Corp.
Aquatec Water Systems
Bard Implants Div.
Bausch & Lomb
Boston Medical
Boston Scientific
Camcar Textron
Cardiotronics
Carolina Medical Electronics
Chattanooga Corp.
Commercial Pattern
Courtauld's Aerospace
Dynacept
Eagle-Picher, Inc.
Endo Vascular Technologies
Fastec Inc
FemRX
Fleetguard
General Equipment
Gibson Product Development
Hach Plastics
Hough Manufacturing Corp.
Hygenic Corp.
Imagyn Medical
Kontes Glass Co.
Kussmaul Electronics
Laser Alignment
Lisk Company
Liston Scientific
Litton Poly-Scientific
Lone Peak Engineering
Luke Air Force Base
Medtronics, Inc.
Millipore Corp.
Nightsun Performance Lighting
Puget Sound Naval Shipyard
Robins Air Force Base
Salter Labs
Scimed Life Systems
Schick Technologies
Seybert Castings
Uniloc Div. Rosemont
Watlow Gordon
Xerox

D. EDUCATIONAL

Arizona State University
Calhoun St. Community College
Cooper Union
Cornell University
Grand Valley State Univ.
Illinois State Univ.
Michigan Technological Univ.
The Ohio State Univeristy
Pennsylvania State University
Rochester Inst. of Tech.
Univ. of Akron
Univ. of Connecticut

E. PROCESS CONTROL/QUALITY CONTROL/MATERIAL DEVELOPMENT

Aristech Chemical
Bayer Plastics
Bluewater Plastics
Calsonics
FMC Corp.
Huntsman Polymers
Network Resins
Norton Company
Novacor Chemicals
Planet Polymers
Rapid Industrial Plastics

NOTE: MENTION OF MORGAN-PRESS USERS' NAMES DOES NOT CONSTITUTE THEIR ENDORSEMENT OF THE PRODUCT.