

E. Alfredo Campo

The Complete Part Design Handbook

For Injection Molding of Thermoplastics

Hanser Publishers, Munich • Hanser Gardner Publications, Cincinnati

HANSER

The Author:

E. Alfredo Campo, 1213 Cerrito Perdido Lane , El Paso, TX 79912, USA

Distributed in the USA and in Canada by
Hanser Gardner Publications, Inc.
6915 Valley Avenue, Cincinnati, Ohio 45244-3029, USA
Fax: (513) 527-8801
Phone: (513) 527-8977 or 1-800-950-8977
www.hansergardner.com

Distributed in all other countries by
Carl Hanser Verlag
Postfach 86 04 20, 81631 München, Germany
Fax: +49 (89) 98 48 09
www.hanser.de

The use of general descriptive names, trademarks, etc., in this publication, even if the former are not especially identified, is not to be taken as a sign that such names, as understood by the Trade Marks and Merchandise Marks Act, may accordingly be used freely by anyone.

While the advice and information in this book are believed to be true and accurate at the date of going to press, neither the authors nor the editors nor the publisher can accept any legal responsibility for any errors or omissions that may be made. The publisher makes no warranty, express or implied, with respect to the material contained herein.

Library of Congress Cataloging-in-Publication Data

Campo, E. Alfredo.

The complete part design handbook : for injection molding of thermoplastics
/ E. Alfredo Campo.

p. cm.

Includes index.

ISBN-13: 978-1-56990-375-9

ISBN-10: 1-56990-375-1

1. Injection molding of plastics--Handbooks, manuals, etc. 2.

Thermoplastics--Design--Handbooks, manuals, etc. 3. Plastics--Handbooks, manuals, etc. I. Title.

TP1150.C36 2006

668.4'23--dc22

2006010219

Bibliografische Information Der Deutschen Bibliothek

Die Deutsche Bibliothek verzeichnet diese Publikation in der Deutschen

Nationalbibliografie; detaillierte bibliografische Daten sind im Internet über <<http://dnb.ddb.de>> abrufbar.

ISBN-10: 3-446-40309-4

ISBN-13: 978-3-446-40309-3

All rights reserved. No part of this book may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying or by any information storage and retrieval system, without permission in writing from the publisher.

© Carl Hanser Verlag, Munich 2006

Production Management: Oswald Immel

Typeset by Manuela Treindl, Laaber, Germany

Coverconcept: Marc Müller-Bremer, Rebranding, München, Germany

Coverdesign: MCP • Susanne Kraus GbR, Holzkirchen, Germany

Printed and bound by Druckhaus "Thomas Müntzer" GmbH, B. Langensalza, Germany

Dedication

To my wife Sandy, my son Jaime, and my daughter Michelle for the love and understanding they gave me during the six long years that it took me to produce all the original drawings, calculate examples and write this comprehensive handbook.

For my uncle, Captain Jaime Merchán and my aunt Ruth Reichman for the assistance provided while attending the University of Miami.

For my great grand uncle Rafael Ramirez who paid for my elementary and high school education after I lost my father at the age of five.

In honor of my beloved departed brother, Leoncio, and sister in-law Becky.

For my grand nephew, Alfredo Campo V. for winning the BMX world championship and the sportsman of the year in my native country Ecuador.

For Richard Scott, Bob Rackley and Bill Hawkins from the Du Pont Film Department for their leadership, teaching and guidance in how to be successful in the engineering technical research and development fields.

For Larry Gillespie, Director of Du Pont Engineering Polymers for providing me the opportunity to work in most of the technical positions in plastics, such as development of new compounds, economic project analysis, designs of international compounding facilities and laboratories in Japan, Mexico and Brazil.

For the Du Pont Film Department, Textile Fibers Department, Engineering Polymers Department, and Du Pont do Brazil that provided me several technical assignments in Plastics during thirty years of service to the company.

For all the Du Pont Plastics Customers that gave me the opportunity to improve their plastic products and optimize their injection molding manufacturing processes.

E. Alfredo Campo

Preface

This handbook was written for the injection molding product designer who has a limited knowledge of engineering polymers. It is a guide for the designer to decide which resin and design geometries to use for the design of plastic parts. It can also offer knowledgeable advice for resin and machine selection and processing parameters. Manufacturer and end user satisfaction is the ultimate goal.

This book is an indispensable, all inclusive, reference guide that can be used by any plastic product designer. There is no need to search through many books and catalogs for needed information. New illustrations, graphs and equations have been included to provide additional clarity for complex ideas. The equations have been verified to ensure correctness and not just copied from another source. Thousands of hours of research and cross referencing have gone into the completion of this work. In addition, more than 35 years of the “hands-on” experience of a plastics expert have been incorporated in this handbook.

The following topics are covered:

- Chapter 1 Plastic Materials Selection Guide:** Includes an introduction to plastic materials, the beginning of plastics, classification of polymer families. Each resin is discussed by its basic chemistry, properties, processing characteristics, advantages, disadvantages and limitations, typical applications and several product illustrations. Thermoplastic materials (35 generic families), thermoplastic elastomer materials (8 generic families), liquid injection molding of silicone, thermoset materials (16 generic families).
- Chapter 2 Engineering Product Design:** Starts with the introduction to structural product design principles, mechanical strength properties of thermoplastics. Centroid, section area, moment of inertia equations and tables. Beam deflection analysis methods. Structure analysis of beams, columns, flat circular plates, and torsion.
- Chapter 3 Structural Design for Thermoplastics:** Discusses the product wall thickness, structural rib design, sharp corners, bosses, threads, undercuts, integral life hinges, pin hinges. Encapsulation of inserts, types of metal inserts and anchorage, and electrical lead inserts.
- Chapter 4 Thermoplastic Gearing Design:** An introduction to and classification of gears. Standard spur, helical, bevel, and worm gears; properties required for thermoplastic molded spur gears, mounting gears on metal shafts, tolerances and mold shrinkage of gears. Plastic spur and helical gearing technology design, strength, horsepower rating, equations, tables, analysis examples and gear specification illustrations.
- Chapter 5 Plastic Journal Bearing Design:** An introduction to types of materials for journal bearings. Theory and design for lubrication. Design principles, performances, dimensions, clearances, molding effects, PV limits and surface finishing. Self-lubricated thermoplastic bearings. Equations, tables, and analysis examples.

- Chapter 6 Thermoplastic Spring Design:** Introduces cantilever beam spring design, applications, and analysis examples. Locating, fixing clip, flexible hinges, and torsional spring applications. Belleville spring washers' equations, tables, and analysis examples.
- Chapter 7 Thermoplastic Pressure Vessel Design:** Discusses thin- and thick-walled pressure vessels' basic principles, equations, tables, analysis examples, design guidelines, applications, and pressure vessel regulations.
- Chapter 8 Thermoplastic Assembly Methods:** Joining two or more components together: assembly method is selected based on product design geometry, size, end use requirements, thermoplastic material characteristics, automatic or manual assembly operation, and manufacturing costs. Each assembly method provides a description, process sequence, advantages and limitations, typical applications, equipment, product joint design, and its variations.
- Chapter 9 Thermoplastic Effects on Design:** Starts discussing the polymer melt behavior, reinforcement, degradation, moisture characteristics, mold shrinkage and critical properties. The molding process effects caused by molding cycle, melt/mold temperature, injection pressure and speed, etc. on product design dimensions, surface finishing, weld line strength and impact resistance and other molding problems.
- Chapter 10 Thermoplastic Injection Mold Design:** Provides an introduction of injection molds, classification and effects on product design. Types of steels, chemical composition, effects of alloying, heat treatment, properties and characteristics. Types of steels used for mold bases and mold components. Cavity surfaces finish procedures and specifications. Types of injection mold designs. Cold runners (two- and three-plate molds, interchangeable mold inserts and vertical insert encapsulation mold). Hot runner molds (internally and externally heated, insulated). Mold design system and other considerations, such as number of cavities, parting line, ejection, cooling, cold runner, gating, venting, cavity inserts sidewall strength, support pillars, molded parts tolerances, mold designer check list, general specifications for mold construction are covered.
- Chapter 11 Performance Testing of Thermoplastics:** It introduces various tests to which thermoplastic polymers are subjected, describes their properties (statistical analysis), such as mechanical, thermal, chemical resistance, rheometer melt viscosity, soldering heat resistance, electrical, flammability, smoke generation, weathering and micro-organism resistance. Test description, procedures, apparatus, test specimen and conditioning, and their significance are discussed here.
- Chapter 12 Thermoplastic Product Cost Analysis:** It discusses molding process variables and capital equipment cost. Three cost analysis methods are used to estimate the molded product user's price.

10 Injection Mold Design

This chapter provides practical information for planning the construction of a thermoplastic injection production mold. It discusses the function of the molds, categories of molds, mold bases, types of steels used in molds, common types of mold designs, the effects caused by the product design on the injection molding process, mold specifications, mold design check lists, and the basic mold design principles.

The thermoplastic injection molds may seem to be a very expensive piece of steel, with the cost of some molds higher than those for an injection molding machine. However in the long run, the mold represents only a very small fraction of the part cost. Good planning during the construction of the molds involves a lot of communication between the product designer, tooling engineer, process engineer, mold designer, mold builder, and resin supplier's technical representative.

The thermoplastic injection molding process obviously includes the use of the mold necessary to produce the end product. It is important to have some understanding of the product design, the thermoplastic resin, the injection molding process, and the technical information required to design and build the mold.

What is a thermoplastic injection mold? Simply stated, it is a collection of steel plates and other mold components which, when properly assembled and installed within the injection molding machine, is capable of producing the required molded products from a given thermoplastic material.

The majority of thermoplastic injection molds are made using high quality tool steel, designed and built by qualified mold makers, to exacting standards, capable of withstanding the high injection pressures and elevated temperatures of the process together with the usually fast cycles, which make injection molding an economically feasible manufacturing process.

Thermoplastic injection molds require a sprue bushing to provide entry of the viscous melt into the cavity. Because the mold must be mounted into an injection molding machine, provision must be made for proper clamping of the mold to the machine platens and for the ejection of the molded part.

Figure 10-1 shows a typical thermoplastic injection mold, together with the most common standard mold components.

10.1 Classification of Injection Molds

The plastic industry classifies thermoplastic injection molds in three general categories: prototype molds (25 to 1,000 parts), production molds (low volume from 1,000 to 10,000 parts) and high volume production molds (10,000 to 2,000,000 parts).

Prototype Molds

A prototype thermoplastic injection mold represents a preliminary step required in the development process of a new product. Prototype molding is used to investigate the molding characteristics of the resin, mold shrinkage, gating,

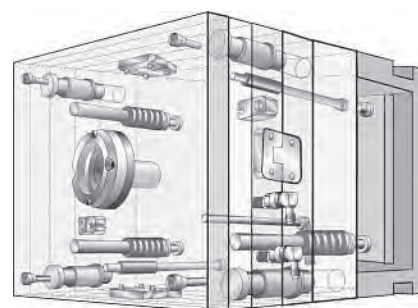


Figure 10-1 Typical mold for injection molding of thermoplastics

dimensional control of the molded part, molding process conditions, and molding cycle. The prototype molded parts are used for product quality control tests and occasionally can be used to meet initial market testing requirements. In simulating the production part, prototype molds become a relatively inexpensive learning device that the part designer can use to pinpoint and correct potential product design problems or material selection problems before expending on a production mold. A prototype mold may consist of an existing mold frame, interchangeable soft cavity inserts, simple cooling and ejection systems, manual insert loading and removal.

Production Molds

A production mold is built using a standard low cost mold base for the hardened tool steel cavities; this mold should be able to produce parts on demand at established production rates. The mold should allow easy repair and facilitate mold venting of the cavities to allow entrapped air and melt volatiles to escape during the molding cycle. The production mold must also have an automatic ejection system and a mold temperature control for consistent cooling of the thermoplastic melt to ensure minimum cycle time, lowest cost, and consistent quality.

High Volume Production Molds

These types of molds should have all the qualities of production tools, have multiple cavities and have fully interchangeable mold components. A high volume production mold should be designed to overcome any adverse outside force, with easy maintenance design. For example, how often do you start to take a mold apart only to find a multitude of inserts without numbering or position marking? How about pry slots? How about jack screw holes for cavity removal? These obstacles can be overcome through a good planning process for the design and construction of the mold, a customized prevention maintenance program, and a protective surface coating on the steel to prevent corrosion and erosion.

10.2 Effects of Product Design on the Injection Molding Process

The product designer must have knowledge of the part geometry, because it may create problems during the molding process; he must also be familiar with the properties of thermoplastic materials, the injection molding process in general, mold design, and the quality of mold construction and product design required to produce functional thermoplastic molded parts.

Producing quality thermoplastic parts requires converting the functional requirement of the application into a design. The product design's geometrical configurations should not only satisfy functionally, but it also has to meet the conditions required by mold design and construction and operation of the mold in order to produce quality parts and guarantee efficient molding operation. When designing injection molded thermoplastic parts, the product designer must also be aware of some part configurations that pose potential problems during the injection molding process. The part design requirements include uniform wall thickness, parting line location to balance the heat removal from both sides of the cavities, smooth internal corners, draft walls (to facilitate part

removal from the cavity), elimination of feather edges, elimination of fragile deep pockets (long thin cores), provide location for the gate, allow large permissible surface area for ejection, specify typical part dimension tolerances for plastics, and avoid the use of high-gloss surface finishing for the product.

10.2.1 Uniform Wall Thickness

Products that incorporate abrupt changes in wall thickness will create major mold design problems regarding the temperature control system of the mold. Abrupt changes in wall thickness make it difficult to maintain a uniform temperature throughout the mold cavities during the molding cycle. After the thermoplastic melt has been injected, variations in part wall thickness do not allow the walls to cool at consistent rates. Thick walls will shrink more than thin walls, causing part warpage, voids on thicker wall cross sections, poor dimensional control, long cycle times, poor surface finish, and structural defects.

10.2.2 Balance Geometrical Configuration

The positioning of the cavity should be balanced on both sides of the mold parting line. Both halves of the cavity should be subjected to the same volume of polymer melt for uniform cooling in the mold cavity. If one side of the cavity is injected with more melt than the other side, this side will become hotter. The hotter side of the cavity will have the tendency to stick on the deep hot spots, causing warpage, poor surface finish of the molded part, and long cycle times.

10.2.3 Smooth Internal Sharp Corners

Sharp corners create high stress concentrations on the thermoplastic part; they are also stress concentrators within the mold cavity. These sharp corner areas fail under high loads. Internal radii of at least 0.031 in should replace sharp corners in thermoplastic part design wherever possible. If a sharp corner is unavoidable, reduce the radii and polish this surface area; in addition, these mold cavity areas should be designed with removable inserts to facilitate ease of repair.

10.2.4 Draft Walls

Thermoplastic parts should be designed with positive draft walls. Minimum positive draft is required on all walls in the direction of mold opening or core pulling. Without draft, thermoplastic molded parts adhere to the mold cavity surface, causing drag marks and surface finishing defects. In many cases, the part will not be fully ejected so that the mold may close on it and cause damage. Lack of positive draft also increases cycle time and molding costs.

10.2.5 Feather Edges

Avoid the use of feather-shaped edges that require thin and fragile steel. Within the mold cavity, feather edges tend to break and chip, resulting in mold maintenance and downtime. Undetected broken and chipped feather edges will cause flashing problems as the thermoplastic melt fills into the mold vents. Feather edges become extremely hot and take longer to cool because cooling

water channels cannot be brought to the feather edge, thus increasing cycle time.

10.2.6 Proportional Boss Geometries

Avoid the use of long narrow cores. The height of the unsupported core should not exceed four times the core base thickness. During the molding process, the injection pressure will deflect long narrow cores, because they act as cantilever beams, causing parting line openings and possible early failure of the mold core insert. In critical cases, a structural analysis of the mold can be made based on expected forces and allowable deflection. Cores of greater height must be fully supported using core inserts to decrease the chance of failure and to ease repair.

10.2.7 Gate Type and Location

The gate is an important component in the injection molding process. The gate influences the type of mold needed for the application (two-plate, three-plate, hot runner, and automation). The location of the gate determines the mold shrinkage, the melt flow, part dimension, warpage, and weld line strength.

The gate functions as a thermo-valve between the runner and the cavity. The temperature is increased around the gate area by the melt injection speed, pressure, and temperature. The hot gate allows the melt to enter the cavity without shearing off the polymer; the gate cools off when the melt stops moving, closing the gate while the melt inside the cavity cools off under packing pressure.

10.2.8 Molded Product Ejection Surface Area

The mold ejection system automatically provides a uniform force to extract the molded product from the cavities. The ejection force breaks the vacuum between the internal surface wall of the part and the cavity core and ejects the parts from the mold. The ejection surface area of the product should be located in the direction of the moving half of the mold, where the ejector system is generally placed. The product designer should specify large ejection surface areas with heavy cross sections that are not critical for the functionality of the product.

The type of ejection system depends on the molded product's geometrical configuration and the permissible ejection surface area wall thickness, the stiffness, crystallinity rate, and melt temperature of the thermoplastic resin.

For example, punctuation holes or indentation defect marks on the external surface of a molded product may be produced by small diameter ejector pins pushing against a flexible and thin-walled cross section of a thermoplastic molded product during the ejection cycle.

10.2.9 Molded Product Tolerances

A realistic view of the cost of tolerances often helps avoid high molding costs without affecting the performance of the part. It may be unreasonable to specify close production tolerances on a part when it is designed to operate within a wide range of environmental conditions. Temperature-induced dimensional changes alone can be three to four times as great as the specified tolerances.

The tolerances for injection molded thermoplastic parts have been developed by the plastic molding industry. The purpose of these specifications is to assist the part designer in obtaining a quick and preliminary analysis for the different molding tolerance factors found in a generic injection molded thermoplastic part.

Understanding the limitations of this process and knowing how to control fine tolerances are the result of applying thermoplastic part design principles, working with molds and mold designers, being aware of the governing rules in polymer technology, and applying the latest technologies available for the injection molding process.

10.2.10 Surface Finish of Molded Product

Surface finish affects part quality, mold cost, mold cycle, and delivery time. Surface finishing is used to enhance surface clarity for appearance of the molded product. The standard steel finishes range from a number one (mirror finish) to a number six (grit blast finish). Any finish specifications on the part print must reference the molded product and not the mold itself. Specifying the mold surface finish does not necessarily produce the expected result on the finished molded product. Although a requirement for a part with a high-gloss finish requires a high-gloss finish on the mold cavity, other factors, such as resin, gating, melt and mold temperature, injection speed, and mold venting affect the surface finishing of the part. For extremely high-gloss finishing, the types of steel used in the cavities may need to be specified to ensure reasonable life of the polished cavity in production.

10.3 Effects of Mold Design on the Injection Molding Process

Mold design is an important consideration in the injection molding process, the following are general concepts for mold design, more detailed information is presented later in this chapter.

10.3.1 Runner System

Good runner design includes not only the correct geometry, size, and layout of the runner, but cooling, ejectability, and minimizing regrinds. A balanced runner system is required for filling all cavities at the same time; this minimizes cycle time and gives the best dimensional integrity to the molded product. Long and skinny or half-moon runners require higher injection pressures so the mold does not cool off prematurely causing incomplete parts. Long and thick runners increase the amount of regrinds, decreasing the efficiency of the molding process. Ejector pins should be provided at the intersection of cold runners to provide sufficient force to eject the runner. Runners should be placed on the ejector half of the mold, so that they can be pushed out by the ejector system.

10.3.2 Mold Cooling System

Mold cooling is one of the most important parameters for controlling dimensional integrity, physical properties, surface finishing, warpage, weld line strength,

and cycle time. A series of cooling channels along the length of a long cavity provides poor warpage control. All core pins should be cooled, especially if the projected length to core diameter ratio exceeds four. Hot core pins cause surface defects and longer molding cycles. A cooled pin is a more efficient system for extracting heat from the encapsulated surface area. Using water in contact with pins is far better than transferring heat across an air layer. Flexible resins need the ejector pins to be cooled for ejection. Temperature control of the sprue puller pin area reduces mold cycle time and ejection interruptions. High volume and turbulent flow of the cooling liquid is critical to maintain good temperature control in the mold. To control the corrosion inside the water lines stainless steel mold plates are used; Corrosion can also be controlled by plating the cooling channel or by the use of rust inhibitors in the water. The mold plates should be thick enough to provide room for the proper size of the cooling channels.

10.3.3 Ejector System

Uniform ejection is critical to control part warpage. Ejector pins, sleeves, rings, or plates must operate without obstructions. A guided ejector system allows ejector pins and cores to be precision aligned and will also bear the weight of the plates so that the pins do not wear and misalign. Early return systems should be provided as a safety feature. They drive all ejector pins to their seated positions before the mold closes, which could collide with pins that did not fully retract. A protector pin should be provided on all molds where the ejector pins or sleeves are under any slides. This locks the ejection plate in its retracted position to prevent collisions between ejector components and slides. For flexible, thin-walled, deep and box parts that are difficult to eject, special ejection systems should be used.

10.3.4 Mold Venting

Mold venting of cavities and runner systems to remove the entrapped air and polymer melt volatiles is a must. A mold without venting causes many processing problems.

10.3.5 Other Mold Devices

Other mold design devices can also increase the cost of the mold, as well as create difficulty in injection molding the thermoplastic product. One example is the mechanically activated side cores or slides. Although sometimes necessary, side cores are expensive mold components and will have high maintenance costs.

10.4 Design Considerations for Injection Molds

The main parameters for the design of a thermoplastic injection mold are: type, size, number of cavities, tolerances, runner layout, gating, venting, parting line, ejection system, surface finishing, steel hardness, and mold cooling among others. The geometrical design of an injection mold, the type of resin, the dimensional tolerances of the product, the part quality, and the volume of part production influence the selection of the steel for the mold because of considerations involving cavity forming and difficulties in heat treatment. Too great a difference in the wall thicknesses of mold cavity inserts necessitates greater care during heat

treatment, because the time to heat the thickest wall section uniformly may result in over heating the thinnest wall section. Quenching from high temperatures, cracking, or distortion may occur where a thin section adjoins a thick section.

Injection mold design is also important from the standpoint of service performance. If mold walls are made too thin, excessive elastic deflection and even cracking can result from service stresses. Overcoming excessive mold deflection that results in flashing problems, dimensional control problems, poor part surface finishing, and incomplete molded parts can only be accomplished by increasing the wall section size of the mold.

It is of extreme importance that any injection mold be designed and built by a qualified mold maker to the exacting standards demanded by the industry. Typical mold development procedures are as follows:

- The product design has been completed and approved for molding
- Mold planning process, where the product designer, tooling engineer, process engineer, mold designer, mold builder, and resin supplier's technical representative review the needs of the product and provide recommendations for the design and construction of the mold
- Development of a preliminary mold design proposal is the responsibility of the mold designer
- Review of the mold layout proposal by the product designer, tooling engineer, process engineer, mold designer, and purchasing, recommending mold changes if needed and providing authorization to finalize the mold design
- Completing the mold design details
- Reviewing the final mold design by the product designer, tooling engineer, process engineer, mold designer, and mold builder
- Construction of the mold
- First molding run evaluation to produce samples at the mold maker's shop, end user tool engineer and process engineer should be present.
- Inspection and documentation of all product samples whether they meet print dimensions (end user)
- Debugging of the mold if needed
- Approval of molded products and pre-production run (mold maker's shop)
- Surface treatment and polishing of the cavities and cores.

10.4.1 Preliminary Mold Design

Preliminary mold design begins from the product dimensional drawing. This represents the part geometry and required dimensions after the thermoplastic injection molding process, when the molded part has completed the mold shrinkage process and has been stabilized at room temperature for at least 24 hours. The preliminary mold layout requirements are the following:

- Mold base size, thickness of plates, and ejection traveling distance
- Placement, number, and size of cavity insert blocks

- Location, type, and size of water cooling systems
- Runner layout and type of gates
- Position and number of support pillars
- Ejector pins sizes and locations
- Parting lines layout
- Position of injection machine tie bars
- Injection machine stroke and shut height positions
- Automation requirements
- Hot runner mold feasibility

After a review of the preliminary mold layout proposal, authorization is given for detailed mold design.

10.4.2 Detailed Mold Design

The cavity details are designed in accordance with the part geometry and the thermoplastic injection mold design, based on good mold manufacturing practices. The critical final mold design requirements include: general mold layout, runner and gate system, cooling system, ejector system, cavity finishing and venting.

General Mold Layout

- The mold base specifications for optimizing the injection molding machine cost rates and cycle times
- Weak, delicate cavities that wear should be designed with a cavity block for strength and cooling using segmented cavity inserts so that mold maintenance costs can be optimized
- In a multi-cavity mold, the cavities should be positioned to balance the system, reducing the runner length.
- Allow between 0.003 and 0.005 in projected height beyond the cavity plates on each cavity half, so that good contact of the cavity halves can be made under clamping pressure.
- Eye bolt holes should be provided on both the top and back side of each half of the mold to ease mounting the mold into the injection molding machine
- The appropriate type steels should be used for each mold component based on the function, machinability, dimensional stability, tolerances, wear resistance, toughness, and strength required for the mold
- Mold components such as sprue bushing, core pins, inserts, and cavities should be keyed to prevent incorrect placement in the mold; movement of these components could cause problems in the process.
- Mold mounting/removal should be easy so that the mold setup time can be minimized
- Cavity inserts should be thick enough with smooth radii so that they are strong enough to survive during molding production

- One leader pin and one return pin on the ejector system should be offset to prevent a 180° reversal of the mold halves' orientation
- Mold plates should be strong enough and thick enough for the cavity insert block size and cooling system
- Sufficient support pillars should not interfere with the ejector pins. Location under the cavity is the primary position to avoid deflections of the cavity during the injection of the melt inside the cavity
- The ejector system should be designed so that travel is sufficient. Too much travel increases the cycle time, wear of the ejector pins and sleeves. Sufficient travel is such that the part can fall free from the mold
- The mold must be vented properly to allow entrapped air and melt volatiles to escape during injection. Otherwise, the steel of the cavity's deep pockets becomes corroded. The vent channels should be properly located to optimize venting without allowing flash. The dimensions of the vents are determined by the viscosity of the thermoplastic material used in the molding process
- The fixed half side of the mold (top clamping plate and "A" cavity plate) need to be thick enough to support leader pins and to provide room for the water cooling lines and other needs.

10.5 Types of Steels Required for Injection Molds

This section provides information on different steel compositions available to manufacture molds used to produce thermoplastic injection molded products. These steels constitute the most commonly used types of materials in the construction mold industry. Other materials such as beryllium-copper, cast aluminum alloy, forged aluminum alloy, cobalt-nickel alloy, and kirkcaldie are also used, but to a lesser extent.

Steels are the workhorse of materials used in molds. No other materials offer comparable versatility for product applications. Steels are produced in the greatest variety of forms and finishes, have strengths ranging from 30,000 psi to over 300,000 psi, and can withstand a range of temperatures from cryogenic up to 2,000 °F.

10.5.1 Major Steel Families

Because of the great range of steel types, properties and applications, steels are categorized into many families based on the chemical composition, heat treatment, surface finishing, critical properties (mechanical, thermal, corrosion resistance, electrical, etc.), typical processing characteristics, end use applications, and other factors. The major families of steel are the following:

Low Carbon Steels

SAE 1008, 1010, 1015, and 1025 are the lowest carbon steels selected when cold forming ability is the primary prerequisite. These steels have relatively low tensile strength values. Strength and hardness increase with carbon addition and/or with cold work, but a decrease in toughness or the ability to withstand

cold deformation is created. Low carbon steels are nearly pure iron in structure, they are readily welded, but do not machine freely, producing poor smooth finishes.

SAE 1016 to 1025 provide increased strength and hardness and reduced cold forming ability. Carburizing or case hardening is possible in some grades. Increase in carbon gives greater core hardness in thicker sections. Increase in manganese improves the hardening ability and also improves machining. SAE 1025 is used for larger sections or where greater core hardness is needed.

All of these steels may be readily welded or brazed. SAE 1020 is frequently used for welded tubing. These steels are used for forged parts; they usually machine better in the “as forged” condition without annealing, or after normalizing.

Medium Carbon Steels

The medium carbon steels, SAE 1030 to 1052, are selected for uses where higher mechanical properties are needed and frequent further hardening and strengthening by heat treatment or by cold work is done. The carbon and manganese levels selected increase the mechanical properties required in section thickness or in depth of hardening. The heat treatment preferred for any of the grades over 0.30% carbon allows selective hardening by induction or flame methods.

All of these groups of steels are used for forging, the section being governed by the section size and the physical properties desired after heat treatment. Medium carbon steels are popular for forging and general uses requiring greater strength than low carbon steels.

High Carbon Steels

Steels SAE 1055 to 1095 are the high carbon types, having more carbon than is required to achieve maximum as quenched hardness. They are used for applications where the higher carbon is needed to improve wear resistance, higher strength characteristics for cutting edges, for springs, and for special purposes. Selection of a particular grade is affected by the nature of the part, its end use, and the manufacturing methods available. Cold forming is not always suitable and most parts are heat treated before use.

Free Machining Carbon Steels

Low and medium carbon steels, in which sulfur, sulfur-phosphorus combinations, and/or lead are purposely added to improve machinability, are termed free-machining carbon steels. Their designations are SAE 1108 to 1151 for resulfurized grades and SAE 1211 to 1215 for rephosphorized and resulfurized grades. Leaded grades are indicated by the letter “L” with the number. Sulfur and phosphorus additions result in some sacrifice in cold forming ability, weldability, and forging ability. Lead additions have little effect on forming ability and forging ability, but impair weldability.

Carburizing Carbon Steels

This term is sometimes given to standard carbon steels, primarily low carbon grades, which are case hardened by various carburizing methods.

Hardening Carbon Steels

These heat treating grades are medium and high carbon steels, hardened by heat treatment before use. They also are referred to as water or oil hardening grades, depending on the quenching media used during heat treatment.

Carbon Spring Steels

This is a common term for medium and high carbon steels (SAE 1050 to 1095) used in spring applications. Annealed and pretempered strips and wires are the common conditions and forms. These steels also are used for music and piano wire, rope wire, and saw blades.

Low Temperature Carbon Steels

These are low carbon (0.20 to 0.30%), high manganese (0.70 to 1.60%), silicon (0.15 to 0.60%) steels produced to a fine grained structure with uniform carbide dispersion achieved through careful composition control and heat treatment. The steels feature relatively high strength and toughness combinations with ductility transition temperatures as low as 130 °F.

High Strength Low Alloy Steels

These are low carbon, manganese steels containing alloying elements, such as chromium, columbium, copper, molybdenum, nickel, titanium, vanadium, and zirconium. Their mechanical properties and corrosion resistance are superior to the more widely used structural carbon steels. These steels usually are used without heat treatment although annealing, normalizing, and stress relieving is required.

Carburizing Alloy Steels

These steels, which include all the low carbon standard alloy steels, are widely used for carburized parts. Low alloy grades (e.g., AISI 4023, 4118, 5015) are used for parts requiring better core properties than are obtainable from carburizing grades of carbon steels. The higher alloy grades (e.g., AISI 3120, 4320, 5120, 8620) are used for better case and core properties. Steel selection among the higher alloy grades depends primarily on hardening ability required to obtain the desired set of properties for the specific conditions of size and heat requirements.

Alloy steels, direct hardening grades quenched and tempered to specific strength and toughness levels, include most of the standard AISI and SAE alloy steels (13XX, 31XX, 40XX, 41XX, 43XX, 46XX, 50XX, 51XX, 5XXX, 61XX, 86XX, 87XX, 92XX, 98XX); they are by far the most widely used alloy steels.

They are also classified by SAE by the carbon content and as low, medium, and high hardening ability grades. Other designations include water or oil hardening types, or simply quenched and tempered steels.

H-Alloy Steels

They are standard direct hardening alloy steels that meet specific hardening ability limits as determined by end quench tests. The steels carry the letter “H” following their conventional numerical designations (e.g., AISI 3120H, 4340H, 8740H).

Boron Alloy Steels

Boron steels are direct hardening alloy steels containing very small amounts of boron to intensify hardening ability. They are also H-alloy steels and carry the letter “B” after the first two digits of their designations (e.g., AISI 10B18, 50B46, 94B17).

Nitriding Steels

These are low and medium carbon/chromium, chromium/molybdenum, or chromium/aluminum steels that are case hardened by nitriding. They are referred to as low carbon, quenched and tempered, constructional alloy steels to distinguish them from the higher carbon quenched and tempered constructional alloy steels. They are noted for high strength, toughness, and weldability and for greater corrosion resistance than structural carbon steels. Their carbon content ranges from 0.10 to 0.20%, the typical alloying elements are chromium, molybdenum, nickel, copper, vanadium, titanium, boron, and zirconium.

Low Temperature Alloy Steels

These are primarily low carbon, nickel steels of relatively high strength and very high toughness at temperatures of 75 to 320 °F. The three most common grades have a maximum carbon content of 0.13 or 0.20% and nominal nickel contents of 2.25, 3.50 and 9%.

Tool Steels

Tool steels are medium and high carbon alloy steels noted primarily for their high hardness, abrasion resistance, and resistance to softening at elevated temperatures. Low carbon and relatively low alloy grades also are available. Tool steels are further classified as prehardened, carburizing, water, oil, and air hardening grades, hot and cold work grades, high speed grades, shock resisting grades, and by other designations. Despite their name, their primary use is for injection mold components and for non-tooling applications.

Electrical Steels

Electrical steels are alloy steels containing from 0.50 to 5.0% silicon and featuring relatively high permeability, high electrical resistance, and low hysteresis loss. They also are classified as grain-oriented and non-oriented types with the latter type further classified as low, medium, and high silicon grades. AISI designates the steels by the prefix letter “M” (for magnetic material) followed by a number that originally was approximately equal to 10 times the core loss in watts/seconds (for 29 gage sheet at 15 kilogausses and 60 cycles).

Stainless Steels

These are alloy steels that have superior corrosion resistance than carbon and conventional alloy steels as a result of moderate to high additions of chromium. Most grades also are noted for heat and oxidation resistance and some stainless steels have very high strength. These materials can be grouped into six major types: austenitic, martensitic, ferritic, age-hardenable austenitic, age-hardenable semi-austenitic, and age-hardenable martensitic.

- Austenitic stainless steels 200 and 300 series designations are non-magnetic, non-hardenable by heat treatment

- Ferritic stainless steels 400 series are magnetic and non-hardenable by heat treatment.
- Martensitic stainless steels 400 and 500 series designations are magnetic, can be hardened by quenching and tempering.
- Age-hardenable austenitic stainless steels (e.g., A-286) are strengthened by heating at moderate temperatures.
- Semi-austenitic stainless steels (e.g., 17-7PH) are austenitic in the annealed condition and martensitic in the hardened condition.
- Age-hardenable martensitic stainless steels (e.g., 17-4PH) are strengthened by aging reactions during tempering.

High Temperature Steels

Steels for high temperature service can be put into two categories: those primarily used for heat or oxidation resistance and those used for structural requirements. The most common heat resistant steels are the austenitic stainless steels that can be used up to about 2,000 °F. However, numerous other steels, such as the martensitic stainless steels, aluminized, and chromized carbon steels also fall into this category.

High temperature (700 to 1,200 °F) structural steels include low alloy martensitic steels, Cr-Mo-V medium alloy air hardening steels, martensitic stainless steels, precipitation hardening stainless steels, and Cr-Ni-Mo alloy steels.

Both categories of high temperature steels are classified under the AISI 600 series of high temperature and high strength alloys.

Ultra High Strength Steels

These are the highest strength steels produced. The designation is somewhat arbitrary, but generally refers to steels having yield strengths above 160,000 psi.

The major types of ultra high strength steels are: medium carbon, low alloy, quenched, and tempered steels such as AISI 4130 and AISI 4340, 5Cr-Mo-V medium alloy air hardening steels, martensitic stainless steels, cold rolled austenitic stainless steels, precipitation hardening stainless steels, 12 and 18% nickel maraging steels, and 9Ni-4Co quenched and tempered steels.

10.6 Steels for Thermoplastic Injection Molds

The selection of a thermoplastic injection mold material depends on a number of factors, including such considerations as product design (geometry, part dimensional control, and tolerances), application end use requirements, thermoplastic material, number of mold cavities, expected production life of the mold, injection molding process conditions, and mold design and construction. The particular usefulness of steels is based on the unique combination of properties achievable through alloying (chemical composition) and uniform heat treatment process. Specific properties of steels that can be modified to achieve desirable characteristics are machinability, polishing ability, surface hardness and depth hardening, wear resistance, corrosion resistance, dimensional stability, thermal conductivity, mechanical strength, toughness, and cost. No other class of mold material is as versatile as steel.

The main factors to be considered in selecting the optimum steels for making a thermoplastic injection mold are:

- Product design geometry, tolerances, end use requirements
- Molded product manufacturing costs
- Type of thermoplastic (properties and process characteristics)
- Injection molding process conditions
- Number of molded parts to be produced with the mold
- Surface finish of the molded parts
- Number of cavities and mold size
- Mold cooling and venting design
- Type and size of runner layout (cold, balanced, or hot runner)
- Geometry, location, and number of gates (two plate, three plate mold, or hot runner drops)
- Cavity forming methods (machined, hobbing, EDM) composed of several assembled inserts or a single unit
- Heat treatment method.

10.6.1 General Steel Selection Procedures

The following recommended tools are used to assist in the selection of optimum steels for thermoplastic injection mold components:

- Material selection guide publication by the American Society for Metals (ASM) Metals Handbook, Volume 1, “Materials for Plastic Molds”
- Technical information provided in this section, “Types of Steels Required for Injection Molds”
- Comparative tested properties provided by steel suppliers design handbooks of candidate steel compositions
- Mold designers’ and mold manufacturers’ recommendations of the types of steels with proven performance in similar injection mold applications and the years of experience working with these materials.

The principal kinds of steel used in making thermoplastic injection molds are selected by their chemical composition, nitriding, prehardened, carbonizing, oil hardening, air hardening, and stainless properties. The principal methods used for forming cavities in steel molds are conventional machining, hobbing, and electrical discharge machining (EDM). Heat treatment is part of the mold making process, unless a prehardened type of steel is selected. Finishing of the mold surface cavity is usually accomplished by grinding and polishing.

It is expected that the information provided in this section will help in selecting a suitable mold steel and mold making process for economic production of a thermoplastic injection molded part. Such decisions are generally considered to be the responsibility of mold designers, although they may receive assistance from others such as mold makers, product designers, process engineers, tool engineers, and resin supplier technical engineers.

10.6.2 Properties and Characteristics of Tool Steels

Uniformity of mechanical properties in the annealed as well as heat treated condition is considered important for tool steels. This requires attainment of a uniform micro structure throughout the mold component section after annealing and after heat treating, or, in a case hardened zone, after carbonizing and heat treating. Good machinability results from a uniform micro structure together with a relatively low hardness level in the annealed condition. There is a direct correlation between annealed hardness and hobbability. Working with soft steels, hobbability improves dramatically as hardness decreases. A uniform micro structure also facilitates machining steels in the prehardened condition. Low distortion in heat treatment is dependent on uniformity of the annealed micro structure, chemical composition, slow heating, and a controlled quenching rate. Good polishing ability is dependent on steel quality, hardness, and the type and uniformity of micro structure of the hardened cavity surface.

Abrasion resistance, including the ability to retain a satisfactory polished surface under service conditions, depends on hardness level and micro structure. The presence of fine excess alloy carbides in a high carbon martensitic matrix results in high abrasion resistance and good polishing ability. Higher corrosion resistance is attained with stainless mold steels or by chrome or nickel plating of the nonstainless steels.

Commercially available high quality tool steel, alloy steels, and stainless are used in the construction of thermoplastic injection molds. Extensive precautions are generally taken in melting, processing, and inspection to prevent the occurrence of defective conditions. For example, using ultrasonic quality control inspections to detect defects is routine practice in the steel making process.

10.6.3 Effects of Alloying Elements on Tool Steel Properties

The steels used for thermoplastic injection molds range in composition from the relatively low alloy oil hardening types containing less than 0.50% total alloying elements through the medium alloy air hardening types up to the most highly alloyed steels that contain up to 21.00% of alloying elements by weight.

Alloying elements are added to steels to achieve certain desirable properties or characteristics that would otherwise be unattainable. Some alloying elements, either alone or in combination, enable heat treatments to be carried out which alter the micro structure providing the desirable tool steel properties.

The addition of more than one alloying element to a steel often produces a synergistic effect. Thus, the combined effects of two or more alloying elements may be greater than the sum of the individual effects of each alloying element.

10.6.4 Chemical Composition of Steels Used for Molds

Typical chemical compositions of commonly used steels used to fabricate the mold components are given in Table 10-1.

Table 10-1 Chemical Composition of Steels Used for Molds

Steel Type	C	Mn	Si	Cr	Mo	V	P	S	Ni	Al
SAE-1015	0.15	0.30	—	—	—	—	0.04	0.05	—	—
SAE-1018	0.18	0.60	—	—	—	—	0.04	0.05	—	—
SAE-1020	0.20	0.90	0.20	—	—	—	0.04	0.05	—	—
SAE-1030	0.30	0.75	0.25	—	—	—	0.04	0.05	—	—
AISI-4130	0.30	0.48	0.20	0.95	0.20	—	0.15	0.15	—	—
AISI-4140	0.43	0.75	0.15	0.80	0.15	—	0.15	0.15	—	—
AISI-4150	0.50	0.90	0.30	0.95	0.20	—	—	—	—	—
SAE-6145	0.45	0.70	0.20	0.80	—	0.15	0.04	0.04	—	—
SAE-A2	1.00	0.70	0.30	5.00	1.00	—	—	—	—	—
SAE-A4	1.00	2.00	0.35	1.00	1.00	—	—	—	—	—
SAE-A6	0.70	2.00	0.30	1.00	1.25	—	—	—	—	—
SAE-D2	1.50	0.30	0.30	12.00	1.00	1.00	—	—	—	—
SAE-H13	0.35	0.40	1.10	5.00	1.50	1.10	—	—	—	—
SAE-O2	0.90	1.60	0.25	—	—	—	—	—	—	—
SAE-P2	0.07	0.30	0.15	2.00	0.20	—	—	—	0.50	—
SAE-P6	0.10	0.50	0.25	1.50	—	—	—	—	3.50	—
SAE-P20	0.35	0.90	0.50	1.70	0.40	—	—	—	—	—
SAE-P21	0.20	0.30	0.30	0.25	—	0.20	—	—	0.40	1.20
SAE-S7	0.50	0.70	0.25	3.25	1.40	—	—	—	—	—
Stainless 410	0.15	1.00	1.00	12.25	—	—	—	—	—	—
Stainless 420	0.15	1.00	1.00	13.00	—	—	—	—	—	—
Stainless 440C	1.05	1.00	1.00	17.00	0.75	—	—	—	—	—

10.6.5 Effects of Alloying on Tool Steels

The contribution of alloying elements on the mechanical properties and the characteristics of tool steels can be summarized as follows:

Carbon (C)

Carbon is the most influential element in controlling hardness, depth of hardening, and strength. Raising the carbon content by different amounts will increase the hardness, depth of hardness, strength, and abrasion resistance achievable after heat treatment and reduce ductility and toughness. Carbon combines with the carbide forming elements (Fe, Mn, Si, Cr, Mo, W, V, P, S, Ni and Al) to produce hard carbide particles that contribute significantly to wear resistance. The amount of carbon in tool steels is specified for attaining certain properties (such as in the water hardening category, where higher carbon content may be chosen to improve wear resistance, although to the detriment of toughness) or, in the alloyed types of tool steels, in conformance with the other constituents for producing well balanced metallurgical and performance properties.

Manganese (Mn)

The principal function of manganese is to combine with free sulfur to form discrete sulfide inclusions and thus to improve hot working ability. Manganese is also a deoxidizing agent. In smaller amounts (0.60%), manganese is added for reducing brittleness and to improve forging ability. Larger amounts of manganese improve hardening ability, permitting oil quenching for non-alloyed carbon steels, thus reducing deformation and toughness.

Molybdenum (Mo)

Molybdenum is a strong promoter of certain metallurgical properties of alloy steels, such as deep hardening and toughness and is used for this purpose in most of the thermoplastic injection mold steels. Molybdenum contributes to secondary hardening on tempering when added in amounts from 0.20% to 1.50% or higher. Secondary hardening enables exceptionally high tempering temperatures to be used to obtain a given hardness and strength level. The higher tempering temperatures increase elevated temperature stability and strength and result in more complete relief of residual stresses for greater dimensional stability. It is used often in larger amounts in certain high speed tool steels to replace tungsten, primarily for economic reasons, often with nearly equivalent results.

Vanadium (V)

This element contributes to the refinement of the carbide structure, improving the forging ability of alloy tool steels. Vanadium has a very strong tendency to form a hard carbide, which improves both the hardness and the wear properties of tool steels; however, a large amount of vanadium carbide makes grinding very difficult. Vanadium is a relatively expensive alloying element and a strong carbide former that is usually added to control grain size and to increase wear resistance. Vanadium combined with carbon produces one of the hardest alloy steels.

Aluminum (Al)

Aluminum combines with nickel to form an intermetallic alloy steel where its hardness is controlled by the cooling rate. Aluminum is used to produce tool steels such as SAE-P21 (thermoplastic injection mold steel). Aluminum and manganese are the principal elements contributing to hardening ability so that full heat treated hardness can be obtained by air cooling. Distortion during heat treatment is lessened as the required quenching rate is reduced.

Silicon (Si)

Silicon is used in most tool steels in quantities from 0.15% to 1.10%. The principal function of silicon is as a deoxidizing agent during melting and to improve the hot forming properties of the steel. In combination with certain alloying elements, the silicon content is raised for increasing the strength and toughness of steels used for tools which have to sustain shock loads. Silicon increases hardening ability slightly and, in higher quantities, retards tempering reactions, thus allowing the use of higher tempering and operating temperatures.

Chromium (Cr)

Chromium is a carbide forming element with a dual function. When present as carbides, all or some may dissolve on hardening and reprecipitate on tempering, it contributes strongly to hardening ability, abrasion resistance, and toughness. When additional chromium, more than what can be utilized by the carbon to

form carbides, is added, this chromium remains in solution and contributes to corrosion resistance. However, high chromium levels raise the hardening temperature of the tool steel, causing hardening deformation problems. A high percentage of chromium also affects the grinding ability of the tool steel.

Nickel (Ni)

Nickel in combination with other alloying elements is used to improve the toughness and the wear resistance of tool steels. Nickel is added to increase the hardening ability of alloy steels. In SAE-P21 tool steel, nickel combines with aluminum to form an intermetallic compound on aging to increase hardness and strength. Large amounts of nickel are needed to ensure the formation of martensite without carbon. Because of its tendency to promote higher annealed hardness and lower machinability, nickel improves aging, providing strength and hardness.

Tungsten (W)

Tungsten is one of the important alloying elements of tool steels. Tungsten improves the hot hardness, or the resistance to the softening effect at elevated temperature; tungsten forms hard, abrasion resistant carbides, thus improving the wear resistant properties of the tool steels.

Cobalt (Co)

A cobalt alloying element is used in applications where an increase in hot hardness is needed for the tool steels. Substantial addition of cobalt, however, raises the critical quenching temperature of the tool steel with a tendency to increase the decarburization of the surface and the reduction in toughness.

10.6.6 Effects of Heat Treatment on Tool Steel Properties

Heat treatment of thermoplastic injection molds is carried out to achieve satisfactory combinations of abrasion resistance, strength, and toughness in both carburizing and deep hardening types of mold steels. When a mold is to be heat treated, precautions should be taken to prevent surface carburization or decarburization as well as distortion and cracking. The thermoplastic injection mold cavities need to be heat treated by methods that introduce residual stresses; such a condition may result in distortion during subsequent heat treatment. Therefore, a stress relief treatment at about 1,200 °F is recommended if considerable machining has been done. To allow for distortion on subsequent heat treatment, thermoplastic injection mold components are generally rough machined within 0.125 to 0.25 in of final dimensions before stress relieving. By a “trueing up” machining operation after the stress relief treatment, more reliable allowances for distortion can be made to limit the amount of grinding necessary after heat treatment.

The typical methods for heat treatment of thermoplastic injection mold components are the following:

Flame Hardening

This is probably the oldest method used to increase wear resistance of the steel materials. AISI 4140 steel is usually purchased by screw manufacturers in the heat treated condition. Normally, this condition is 28–32 Rc, which provides

good mechanical strength of approx. 100,000 psi tensile yield. The steel is still readily machinable in this condition. The mold component can be further flame hardened to approx. 48–55 Rc. Of course, this cannot be done with low carbon steels.

Approximately 0.40% carbon content is needed to achieve this result on a practical basis. The process employs an open gas/oxygen flame followed by rapid quenching. The usual depth of hardness is approx. 0.125 in, but the hardness tapers off as you go deeper from the outside.

Induction Hardening

This process gives the same result as flame hardening, but uses induction heat created by magnetic flux reversals rather than a flame.

Nitriding

A hard outside case can be obtained by subjecting the mold components to a high nitrogen atmosphere (ammonia gas) at elevated temperatures of approx. 950 °F. This comparatively low temperature causes minimal distortion but provides a very high case hardness of 60–70 Rc. The depth of case ranges from 0.020 to 0.024 in. This thin case diminishes in hardness from the outside, causing the mold components to wear rapidly once any significant wear has occurred. Allowance for 0.0005 to 0.001 in of growth must be considered when designing mold components before the nitriding process. The high hardness is caused by the formation of metallic nitrides. A proper nitriding steel, such as Crucible Nitriding 135 or Ryerson Nitralloy 135M should be used to develop maximum hardness from the process. These steels are similar to AISI 4140 in their chemistry, but they have 0.95% to 1.30% aluminum added to form very hard aluminum nitrides. Alloy steels, such as AISI 4140 can be nitrided, but they exhibit a slightly lower hardness with a little increased depth of case. Nitriding is usually done over the entire mold component. This improves wear resistance to abrasion by fiber glass or abrasive mineral reinforced thermoplastic resins.

Ion nitriding is a process superior to gas nitriding, which is described above. Ion nitriding is more expensive, but causes less distortion because of lower processing temperatures. The hardness, depth of case, and wear properties are very similar in either process.

Precipitation Hardening

Precipitation hardening is a low temperature process used to harden certain grades of stainless steels. 17-4PH stainless steel is an example of this type, where the PH stands for precipitation hardening. These grades are usually supplied in condition “A” (solution treated), which is similar to being annealed. The stainless steel component is then machined and hard-surfaced before precipitation hardening.

Heat Treatment Process

Rapid heating of a thermoplastic injection mold component to the austenitizing temperature and inadequate support of the mold component in the furnace can result in unacceptable distortion. Bolting the mold component to a support can prevent sagging. A support may also be necessary to prevent distortion of a large mold component during its removal from a furnace before quenching. Since the greatest shape distortion generally occurs during quenching, the objective should be to cool all sections of the mold component at the same rate. A mold

component with uniform wall thicknesses on all sides will cool more uniformly than a mold component with both thick and thin walled cross sections. It is desirable to use a quenching rate that is not much faster than necessary to achieve the desired degree of hardening. Lower distortion can be achieved by air hardening or precipitation hardening of the steel.

In carrying out heat treating operations, such as stress relieving and austenitizing, the holding time at temperature is determined by the maximum cross section wall thickness of the mold component. Tempering temperatures should be selected to achieve a satisfactory combination of hardness and toughness. Shape distortions on tempering can be minimized by heating the mold component slowly to the tempering temperature starting with a cold furnace, tempering as required by the desired properties and cross section wall thickness, followed by furnace cooling.

The use of a protective furnace atmosphere, a vacuum furnace, or salt bath is generally necessary to prevent excessive oxidation and carburization or decarburization during austenitizing that could adversely affect the surface finish and surface properties. Deterioration and nonuniformity may also result from failure to clean all the oil or grease from a mold component surface before hardening by using carburizing compound for case hardening or by not removing any copper that may be on the surface.

10.6.7 Prehardened Tool Steels

The most commonly used prehardened steels for making thermoplastic injection mold components are SAE P20, P21 and other medium carbon low alloy grades of suitable quality. Prehardened SAE H13 tool steel is available in rounds from a limited number of suppliers. Stainless steel 420 is also available in the prehardened condition. The use of prehardened steels eliminates the possibility of cracking, distortion, scaling, and pitting that can occur during heat treatment of a mold component that has been machined in the annealed condition. Prehardened steels with hardnesses in the range of 28 to 32 Rc are used mainly for injection mold plates. The upper limit of this hardness range corresponds to about the upper limit of adequate machinability using conventional metal cutting tools. A stress relief at 100 °F below the tempering temperature of these steels is usually carried out after rough machining the mold cavity to avoid excessive residual stress that might result in distortion or possible cracking during subsequent grinding.

Prehardened SAE P20 Tool Steel

Prehardened SAE P20 tool steel is a 0.35% carbon, 1.70% chromium, 0.40% molybdenum, 0.90% manganese, and 0.50% silicon tool steel available in the heat treated condition at 28–32 Rc as a result of prior hardening and tempering. This steel has excellent dimensional stability, good machinability, polishing ability and toughness, but fair wear resistance. Eliminating a second machining operation normally required by steels that must be heat treated is considered suitable for various sizes of molding machines and types of injection molds. To obtain greater surface strength, carburizing is followed by conventional hardening and tempering treatments to obtain adequate hardness and strength.

Prehardened SAE P21 Tool Steel

Prehardened SAE P21 is mainly used for injection molds; its chemical composition is 0.20% carbon, 0.40% nickel, 0.25% chromium, 0.02% vanadium, 0.30%

manganese, 0.30% silicon, and 1.20% aluminum; available in the precipitation hardened condition at 32–39 Rc as a result of prior solution treating and aging. This steel exhibits high strength, excellent dimensional stability, good machinability, fair toughness and wear resistance. Cavities and cores are cut directly from this steel. Uniformity of hardness and strength can be attained in a relatively large section, provided cooling is at a sufficient rate from the solution treating temperature to prevent softening as a result of aging.

Prehardened SAE H13 Tool Steel

Prehardened SAE H13 steel is a 0.35% carbon, 0.40% manganese, 5.00% chromium, 1.50% molybdenum, 1.10% silicon, and 1.10% vanadium tool steel available in the heat treated condition at 30–36 Rc as a result of prior hardening and tempering (65–74 Rc). This steel has optimum dimensional stability, excellent toughness and wear resistance. This tool steel is primarily used for protection from abrasion, for injection molds requiring high thermal shock resistance, and good polishing ability. A drawback to this tool steel is the difficulty of machining or rebuilding by hard-surfaced welding. It is also difficult to straighten by conventional methods.

A free machining grade of SAE H13 tool steel is produced by use of carefully controlled and evenly dispersed sulfide additions in the melting operation. Such a steel can be prehardened to 50–52 Rc and still possess adequate machinability. This has the advantage of higher strength although the polishing ability may be adversely affected by the increased sulfide content.

Prehardened AISI 4130 Alloy Steel

Medium carbon alloy steel AISI 4130 prehardened is a 0.30% carbon, 0.95% chromium, 0.48% manganese, 0.20% silicon, and 0.20% molybdenum alloy steel available in the heat treated condition at 28–32 Rc. The hardening ability is low and it is quenched in water. This steel has optimum dimensional stability and machinability, excellent toughness and adequate wear resistance. This steel is advantageous for injection molds because of high toughness, moderate strength, and high thermal shock resistance.

Prehardened AISI 4140 Alloy Steel

Medium carbon alloy steel AISI 4140 prehardened is a 0.40% carbon, 0.80% chromium, 0.75% manganese, 0.15% silicon, and 0.15% molybdenum alloy steel available in the heat treated condition at 28–32 Rc and stress relieved conditions. It has good strength and can be flame hardened or rebuild by hard-surfaced welding.

The hardening ability is medium and it is quenched in oil. This steel has optimum dimensional stability and machinability, good toughness, strength and wear resistance.

It is important to make sure that special, easy to machine grades are not selected. These grades usually contain either lead or are resulfurized. This makes the mold component impossible to rebuild by hard-surfaced welding due to extreme porosity and stress cracks.

Prehardened AISI 4150 Alloy Steel

Medium carbon alloy steel AISI 4150 prehardened is a 0.50% carbon, 0.95% chromium, 0.90% manganese, 0.30% silicon, and 0.20% molybdenum alloy steel available in the heat treated condition at 30–36 Rc. The hardening ability is high

and it is quenched in oil. This steel has high strength, very good dimensional stability and machinability, good wear resistance and moderate toughness.

AISI 4340 Alloy Steel

This alloy steel is similar to AISI 4140, but includes nickel as an alloying element plus a greater percentage of molybdenum. This provides a slightly higher strength, but the major difference is the greater penetration of heat treatment providing superior mechanical properties at the core of the mold component.

Nitralloy 135M Alloy Steel

A number of steel manufacturers produce a product similar or identical to Nitralloy 135M, but this is the name best known. Nitralloy is similar to AISI 4140, but is slightly lower in physical properties. The major difference is the inclusion of a small portion of aluminum.

CPM-10V Tool Steel

This material is a totally new concept for tool steels based on particle metallurgy. CPM stands for "Crucible Particle Metallurgy". This process enables the incorporation of very high amounts of carbon, chromium, and vanadium not possible by conventional methods used to manufacture steels. CPM-10V steel is intermediate between tool steels and cemented carbides, yet it has excellent strength. It also has much higher toughness than conventional tool steels, if heat treated properly. It provides superb wear resistance in fiber glass reinforced thermoplastic processing components such as injection screws, check valves, barrels, nozzles, mold cavity inserts, and gate inserts.

CPM-10V large bar stock is very expensive, but in smaller sizes (2.50 in) is fairly competitive. The material is probably the best choice for mold components subjected to more than 10% glass fibers or abrasive mineral reinforced thermoplastic resins, especially if these polymers have high melt flow rates.

10.6.8 Carburizing Tool Steels

The two most commonly used carburizing steels for making thermoplastic injection molds are SAE P2 and SAE P6. Hobbled or machined injection molds made from these steels are case hardened by a carburizing treatment followed by hardening and tempering treatments. The aim of case hardening is to attain an adequate combination of abrasion resistance and toughness for a particular application. Carburizing steels are used for injection molds when a higher hardness is required than can be achieved with prehardened steels.

Selection of the carburized case depth depends on the service conditions to which the mold component will be subjected. The deeper and harder the case hardened zone, the greater the resistance to abrasion. The stronger and tougher the core zone, the shallower is the required case depth. However, a large cavity usually requires a greater case depth along with a tougher core than a small cavity. A greater case depth is also required if a mold is subjected to a high injection molding pressure or if a cavity is shallow compared to other dimensions.

Carburizing SAE P2 Tool Steel

SAE P2 steel is a 0.07% carbon, 2.00% chromium, 0.50% nickel, 0.15% silicon, and 0.30% molybdenum tool steel available in the annealed condition. This steel has fair dimensional stability, good toughness, and very good wear resistance.

After a mold cavity is formed from SAE P2 tool steel by hobbing, the mold cavity is carburized to a case depth of 0.05 to 0.06 in and heat treated to a case hardness of 60–64 Rc with an internal core hardness of 14–18 Rc. Due to the high strength and abrasion resistance at the surface, carburized SAE P2 tool steel's main advantages are that it can be easily cold hobbled in the annealed condition and it can subsequently be polished to a high luster in the carburized condition.

Carburizing SAE P6 Tool Steel

SAE P6 steel is a 0.10% carbon, 3.50% nickel, 1.50% chromium, 0.25% silicon, and 0.50% molybdenum tool steel available in the annealed condition. This steel has fair dimensional stability, good toughness and machinability, and very good wear resistance. After a mold cavity is formed from SAE P6 tool steel by machine cutting, the mold cavity is carburized and heat treated to a case hardness of 58–61 Rc with an internal core hardness of 26–28 Rc. SAE P6 tool steel is comparable to SAE P2 with regard to surface strength, abrasion resistance, and polishing ability. Due to the higher internal core hardness, SAE P6 mold cavities are capable of withstanding higher injection pressures than SAE P2 mold cavities.

10.6.9 Oil and Air Hardening Tool Steels

The most commonly used oil and air hardened tool steels for making thermoplastic injection mold components are SAE O2, S7, H13, A2, A4, A6, and D2. These tool steels offer suitable combinations of high abrasion resistance, high strength, and high polishing ability required for thermoplastic injection mold components. To achieve a mirror finish on the cavity surfaces it is necessary to attain a hardness greater than 54 Rc. Because of its lower hardness in the annealed condition, machinability is superior to that of the prehardened steels.

Oil Hardening SAE O2 Tool Steel

SAE O2 tool steel is a high carbon oil hardened tool steel available in the spheroidized/annealed condition. SAE O2 chemical composition is a 0.90% carbon, 1.60% manganese, and 0.25% silicon tool steel. It is capable of attaining surface hardness up to 63 Rc by heat treatment. This steel has optimum machinability, adequate dimensional stability, toughness, and wear resistance.

Air Hardening SAE S7 Tool Steel

SAE S7 tool steel is an air hardening 0.50% carbon, 0.70% manganese, 0.25% silicon, 3.25% chromium, and 1.40% molybdenum shock resisting tool steel that is available in the spheroidized/annealed condition. The suggested hardness range of 52–58 Rc can be attained by air hardening and tempering. This steel has excellent toughness, very good machinability, dimensional stability, and wear resistance. SAE S7 tool steel is suitable for injection mold components when high strength, toughness and thermal shock resistance are required.

Air Hardening SAE A2, A4 and A6 Tool Steels

These are commercially medium alloy air hardened tool steels in the spheroidized/annealed condition. Hardness in the range of 58–60 Rc can be attained by heat treatment. This hardness range is suitable for injection molds.

SAE A2 steel is a 1.00% carbon, 0.70% manganese, 0.30% silicon, 5.00% chromium, and 1.00% molybdenum tool steel that has the highest abrasion resistance of the group.

SAE A4 steel is a 1.00% carbon, 2.00% manganese, 0.35% silicon, 1.00% chromium, and 1.00% molybdenum produced as a free machining grade tool steel. SAE A4 tool steel has the advantage of requiring a lower hardening temperature than SAE A2.

SAE A6 is a 0.70% carbon, 2.00% manganese, 0.30% silicon, 1.00% chromium, and 1.25% molybdenum produced as a free machining grade tool steel. SAE A6 tool steel has the higher toughness of the group and the advantage of requiring a lower hardening temperature than SAE A2.

Air Hardening SAE D2 Tool Steel

SAE D2 tool steel is a 1.50% carbon, 0.30% manganese, 12.00% chromium, 1.00% molybdenum, 0.30% silicon, and 1.00% vanadium air hardened tool steel available in the spheroidized/annealed condition. A hardness of 58–60 Rc can be achieved by heat treatment. This tool steel has excellent dimensional stability and wear resistance, but poor toughness. SAE D2 tool steel is the most abrasion resistant of the deep hardening steels listed in Table 10-1. It has been primarily used for applications requiring abrasion and corrosion resistance. A drawback to this tool steel is the difficulty of machining or rebuilding by hard-surfaced welding; it is difficult to straighten by conventional methods.

10.6.10 Stainless Steels

Stainless steel types 410, 420 and 440C are martensitic stainless steels commonly used for thermoplastic injection molds. Their high corrosion resistance is of advantage in molding relatively corrosive thermoplastics or molding under relatively corrosive atmospheric conditions.

Stainless Steel 410

Stainless steel 410 is a 0.15% carbon, 1.00% manganese, 1.00% silicon, and 12.25% chromium steel available in the annealed condition. Suitable heat treatment can produce a hardness of 38–41 Rc. This stainless steel has excellent polishing ability eliminating the need for chrome plating. The toughness is excellent, dimensional stability and wear resistance properties are good, but the machinability is fair. It has slightly better corrosion resistance than stainless steel 420.

Stainless Steel 420

Stainless steel 420 is a 0.15% carbon, 1.00% manganese, 1.00% silicon, and 13.00% chromium steel available in the annealed condition. By heat treatment, a hardness of 50–54 Rc is attained, which results in higher strength and abrasion resistance than stainless steel 410. This stainless steel has excellent polishing ability, eliminating the need for chrome plating. The toughness, dimensional stability, and wear resistance properties are good, but the machinability is fair.

Stainless Steel 440C

Stainless steel 440C is a 1.05% carbon, 1.00% manganese, 1.00% silicon, 17.00% chromium, and 0.75% molybdenum steel available in the annealed condition. By heat treatment, a hardness of 56–61 Rc can be attained, which results in higher strength and abrasion resistance than stainless steel 410 and 420. This stainless steel has excellent polishing ability, eliminating the need for chrome plating. The toughness is poor, dimensional stability and wear resistance properties are very good, but the machinability is fair.

10.6.11 Steels Used in Thermoplastic Injection Mold Components

Prehardened, carburizing, and deep hardening steels are all used for general purpose thermoplastic injection molds because of their service record, cost, and ready availability. Prehardened steels of moderate strength and wear resistance are widely used, but as operating conditions become more severe, high alloy tool steels become more common. Carburized tool steels are used for high flow, fiber glass, and abrasive mineral reinforced injection molding thermoplastics. Heat resistant steels are used for injection molding thermoplastics requiring high mold temperatures. Stainless steels, as well as chrome and nickel plated prehardened or deep hardened steels, are used for injection molding corrosive thermoplastic resins.

The types of steels used in the construction of the mold components can be divided into two groups, typical commercial mold base combinations of steels as shown in Figure 10-2 and the steels used for the remaining mold components.

The hardness of a mold cavity is usually in the range between 50 and 60 Rc and it may be chromium- or nickel-plated with plating layers usually under 0.002 to 0.005 in thick. Care must be taken when chromium-plating. The plated components must be treated to prevent hydrogen embrittlement where the hydrogen gas generated in plating can cause loss of steel toughness.

The steels listed in Table 10-2 can be used for making thermoplastic injection mold components. The selection of steels depends on the mold component's operating functions, type of thermoplastic resin to be molded, number of parts to be produced, and the method of forming the mold cavity.

There are other materials used in mold components, depending on the application:

Aluminum is used for prototype molds. The types used are 6061 and 7075 series, with tensile strengths over 75,000 psi and relatively good scratch resistance.

Copper, aluminum, or beryllium copper alloys are used in molds, they have high thermal conductivity and can be hardened to 45 Rc. Their problems are poor machinability; they are expensive and have low strength and stiffness, so core deflection can be a problem.

Bronze is used in some cases for wear resistance in mold mechanisms and occasionally for cavity inserts.

Steel "A": SAE 1015, 1018, 1020, 1030.

Low carbon hot rolled steel with good tensile strength properties, machines easily, allowing faster and smoother cuts.

Steel "B": AISI 4130, 4140, 4150.

Preheat treated medium carbon alloy steels (28–36 Rc), high strength, machinability, and toughness, with good wear resistance properties.

Steel "C": SAE P20, P21.

Medium hardening ability preheat treated tool steels (28–39 Rc), excellent dimensional stability and surface finishing, high strength, good machinability and toughness, but low wear resistance properties.

Steel "D": SAE A4, A6, O2, S7, H13, P6, P20.

Preheat treated medium carbon tool steels, chrome and nickel plated, excellent dimensional stability and surface finishing, high strength.

Steel "E": Stainless Steel 410, 420, 440C

High corrosion resistance by heat treatment to 38–54 Rc, high strength, surface finishing, and wear resistance, but hard to machine

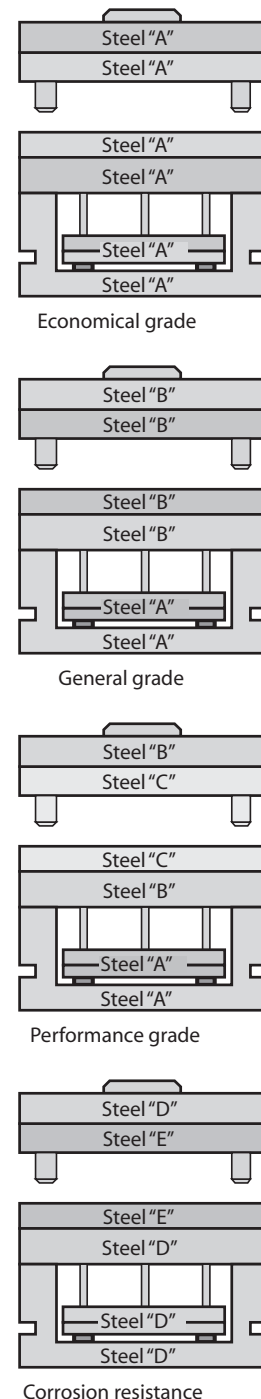


Figure 10-2 Typical commercial mold base combinations of steels

Table 10-2 Steels Used in Thermoplastic Injection Mold Components

Mold components	Types of steels used
Locating ring	SAE 6521; 6524 (28–32 Rc)
Extension nozzle bushing	AISI 4140 (28–32 Rc)
Sprue bushing	SAE 6145; Ampco 940/stainless steel
Sprue puller pin	Nitrided H13 (65–74 Rc)
Ejector pin	Nitrided H13 (65–74 Rc); CM 50 (60–65 Rc)
Ejector sleeve	Nitrided H13 (65–74 Rc)
Ejector plate	SAE 1020 (30–35 Rc)
Ejector retainer	SAE 1020 (30–35 Rc)
Leader pin	SAE 1018; 8020 (30–35 Rc)
Return pin	Nitrided H13 (65–74 Rc)
Ejector stripper plate	Prehardened P20; H13; S7; A2; A4; A6
Cavity block	Prehardened or precipitation hardening P20; P21; H13; S7; A2; A4; A6; P2; P6; Stainless steel 420 (28–50 Rc)
Cavity insert	Prehardened or precipitation hardening H13; D2; S7; A2; A4; A6; P2; P6; stainless steel 420; 440C (50–64 Rc)
Gate insert	D2; CPM 10V; CPM 9V (50–60 Rc)
Core block	Prehardened or precipitation hardening P20; P21; H13; S7; A2; A4; A6; P2; P6; stainless steel 420 (28–50 Rc)
Core insert	Prehardened or precipitation hardening H13; S7; P2; stainless steel 420, 440C; beryllium copper; Ampco 940
Slide	Carburized or nitrided P20; nitrided P21; O2; A2; A6; P2
Wear plate	SAE A2; A6; P2 bronze-plated
Angle pin	Nitrided H13 (65–74 Rc)
Knockout rod	SAE 1020 (30–35 Rc)
Support pillar	SAE 1040 (28–32 Rc)
Stop pin	SAE 1040 (28–32 Rc)
Parting line interlock (male)	SAE 8620 (50–55 Rc)
Parting line interlock (female)	SAE 8620 (55–60 Rc)
Insulator sheet	Asbestos-free glass reinforced polymer composite

Zinc alloys are too soft and not strong enough for production molds, unless the injection pressure and melt temperature are very low, and the molded product does not require good dimensional tolerances. “Kirksite” is used for prototype molds, but “Kirksite” has higher thermal conductivity than steel and will produce molded parts with different shrinkage than parts from a production steel mold.

Electro-deposited cavities are used for prototype molds and for small specialized cavities used in production steel molds.

10.7 Mold Cavity Surface Finishing

The cavity surface finishing, runner and gating system, mold temperature, the type of thermoplastic resin, and the process molding conditions will determine the quality surface finish that will be obtained on the molded thermoplastic parts. The type of tool steel, its quality, structure, and heat treatment process can also affect the polishing ability of the cavity and the surface finish obtained. Several new technical innovations or improvements by various leading steel mills are producing better quality tool steels as a result of alloying compositions, modern manufacturing processes, and metallurgical quality control.

The product design surface finish specifications may require the use of a specific type of thermoplastic resin. The product surface finish must be studied in great detail before proceeding with the design and manufacture of the mold. The difference between a number three cavity surface finishing (good mold finish) and a number one surface finishing of optical quality (mirror finish) can double the price of the mold. When the mold design is more complicated, the tool steel for the mold cavity is more expensive, and the heat treating method is more difficult; machining, grinding, and polishing represent the effort of many man hours reflecting the price increase.

Processing with a mirror finish mold requires a better quality thermoplastic resin that is more expensive. The material handling efficiency for the resin is low, the resin must be dry before molding and the use of reground material (runners and rejected parts) is not allowed. The molding process efficiency is also low with long molding cycles and high rejection rates, which require extra quality control support. High maintenance costs, spare parts for inventory, and mold cavity insert replacements, requiring special equipment for repairs, which are also time consuming. The life expectancy of such a mold is typically low because of complications and a new spare mold will probably be needed for the molding process.

The surface finish characteristics of the thermoplastic resins are important parameters for the injection molding process. Unreinforced amorphous resins have better surface finish characteristics than glass- or mineral-reinforced semi-crystalline resins. For example, injection molded products (lenses, tail lights, or dial faces) from transparent materials such as acrylic and PC will require the highest quality finish. Other materials, such as low density polyethylene or high impact polystyrene, which have a natural shine, do not reproduce the high level finish that is generated with a diamond polished surface (No.1 surface finish).

The Society of the Plastics Industry and the Society of Plastics Engineers developed mold steel surface finishing standards, which provide an excellent visual comparison. The standards consist of a group of six pieces of steel, each finished to a different level and used as a comparison guide in mold finishing.

Finish	Process parameters
Number 1	8,000 Grit (0.0 to 3.0 diamond compound micro range)
Number 2	1,200 Grit (up to 15.0 diamond compound micro range)
Number 3	320 Grit, abrasive cloth
Number 4	280 Grit, abrasive stone
Number 5	240 Grit, dry blast (5.0 in distance at 100.0 psi pressure)
Number 6	24 Grit, dry blast (3.0 in distance at 100.0 psi pressure)

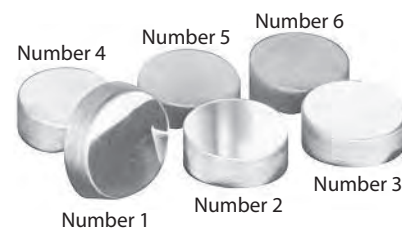


Figure 10-3 SPI-SPE mold steel surface finishing comparison kit

The grades range from number six, which is a 24 grit dry blast finish, to number one, which reflects a diamond polish with an 8,000 grit compound.

The official SPI-SPE mold steel surface finish standard comparison kit and the descriptions are shown in Figure 10-3.

There are other, more sophisticated, methods of measuring surface characteristics. One such method is the use of an instrument called a profilometer. It measures the surface “roughness” of the metal. The stylus of this instrument is moved over the surface of the steel; the peaks and valleys of the surface finish are fed into the analyzer and the surface roughness is charted in terms of micro inches. By definition, a micro inch is a millionth of an inch (0.000001 in). The surface of any material is not truly “smooth” when examined under the microscope, but rather generally covered with ridges and scratches. The profilometer measures the depth of the scratches and the height of the ridges and reports them as maximum peak-to-valley height, along with the average deviation from the mean surface. The average deviation from the mean surface is called the “Root Mean Square” and is known as RMS. A micro inch is the most often used value for specifying surface finishes. Mold steel surface finishing comparison standards have established the mold steel surface finish assigned number versus the RMS relative value in micro inches. Table 10-3 shows the equivalent RMS values for the SPI-SPE mold steel surface finishes.

Table 10-3 Equivalent RMS Values of Standard Mold Steel Surface Finish

Mold finish	RMS value range	Mold finish	RMS value range
Number 1	0.50 to 1.00	Number 4	12.00 to 15.00
Number 2	1.00 to 2.00	Number 5	26.00 to 32.00
Number 3	7.00 to 7.50	Number 6	160.00 to 190.00

Table 10.4 provides a reference for the finishing process required to produce the desired RMS surface finish classification.

Table 10-4 RMS Surface Classifications Based on the Finishing Process Used

RMS surface range	Finishing process parameters
0.00 to 10.00 RMS	Fine abrasive hand polish, coarse to fine diamond polish, ultra-fine hand-rubbed diamond finish.
10.00 to 20.00 RMS	Fine cylindrical grind, smooth ream, fine surface grind, medium grind hand polish.
20.00 to 40.00 RMS	Fine machine grind, medium surface grind, rough abrasive polish, ream.
40.00 to 100.00 RMS	High grade machine finish, coarse surface grind, smooth cylindrical grind, smooth hand file.
100.00 to 250.00 RMS	Medium machine cuts, coarse surface grind, smooth disc finish grind, medium file.
250.00 to 500.00 RMS	Heavy machine cuts, rough filing, rough disc grinding, sand cast surfaces.

10.7.1 Mold Surface Finishing Process Procedures

The various surface textures for mold cavity and core surface finishing levels required in the thermoplastics injection molding industry are more demanding than the conventional molds built for other plastic processes, such as blow molding or thermoforming. The mold cavity surfaces must be prepared to accept the final polish, whether it is a 320 grit cloth finish or a number one (1) diamond polish mirror finish. The polished surface must be free of waves, ripples, distortions, pits, scratches, or orange peel.

Machining

Machining is the first step in the manufacturing process of a mold component. It determines the tools to be used, the surface condition obtained during the machining operation, and the methods of polishing and finishing. When the mold components are machined according to the size and geometry of the mold design that has already accounted for the mold shrinkage of the thermoplastic part, the cavity dimensions are left with about 0.001 to 0.003 in of stock for subsequent finishing operations.

When the mold cavity and core inserts are to be heat treated, the details are machined to within 0.0003 to 0.0005 in of the final surface finish.

Mold components that come off the duplicating machine will have the characteristic ridges and valleys resulting from this particular type of machining. The ridges on either side of the valley must be leveled to expose an even surface for the final finishing. The metal removal operation can be aided by the use of a metal dye painted over the surface to serve as a guide in metal removal. It is often advisable to machine cavities between 0.002 and 0.003 in deeper than necessary and remove the proper amount from the top surface, after the mold has been tested in the injection molding machine.

Electrical Discharge Machining (EDM)

Electrical erosion is a metal removal process using a master electrode that is electrically conductive. Copper alloys are generally used to make the master electrode because hardness is not a requirement. Cast zinc and machined graphite are also used in some instances. The principle of spark erosion is used by the mold making industry. The gap between the master and the cavity insert is quite uniform and small. As the master descends, small intense sparks are generated wherever the gap is reduced. Erosion occurs on both master and cavity inserts, the master's negative polarity erodes only at 1/4 to 1/10 of the speed at which the cavity insert erodes at positive polarity. The cavity insert may be hardened before the EDM process begins so that distortion due to heat treatment is eliminated. The dielectric fluid must circulate continuously to remove the minute particles that are formed between the master and the cavity insert. Electrical erosion is slow compared to mechanical cutting of soft steels, but for certain conditions, such as narrow deep slots, it offers great advantages. To utilize the EDM process, the mold maker roughs out stock by machining wherever possible and then sends the cavity insert to the EDM processor.

Grinding

To remove ridges and rough cut marks, a hand or flexible shaft grinder is fitted with a grinding wheel or abrasive disc. Extreme care must be exercised to prevent the grinder from following the ridges and removing more material than necessary.

Grinding strokes should follow the crest of the ridges until the mean surface is reasonably constant. A more recently developed tool, the portable belt sander, can also be used in this “roughing” phase of the operation. Discs or abrasive belts have the advantage of being able to span a greater area, but care must be taken because they can also cut very quickly. Grits in the range between 60 and 320 are used. Filing provides ample control and keeps the surface quite even. The grade of file used will be determined by the amount of metal to be removed.

Rough grinding, or coarse filing, can leave the metal surface full of tears and waves. Finer wheels, drums, and files may be used to improve the surface texture and bring the cavity or core closer to print dimensions.

After all the metal has been removed to make the part to print (plus the allowance for finishing stock), the surface should be examined to determine if it is ready for stoning.

The surfaces of heat treated thermoplastic injection molds are ground before polishing. Improper grinding can generate very high temperatures and precautions should be taken to avoid the following defective surface conditions:

- Softening from over tempering
- Embrittlement due to rehardening
- Distortion and unfavorable residual stress
- Cracking

Avoiding these problems requires grinding with a proper wheel, at its recommended rotational speed, at an adequately low rate of metal removal, and with an adequate coolant. If residual stresses imposed by grinding are not relieved, pitting during subsequent polishing may result. Adequate relief of residual grinding stresses can usually be accomplished by heating to about 100 °F below the tempering temperature.

Stoning

Choosing the initial grit of stone depends on the degree of finish left by the machining, grinding, or filing operation. Machining or duplicating usually results in a coarser finish than grinding; therefore, a coarser grit stone would normally be used. Preliminary stoning may be done with a 240 grit stone to remove final dips, depressions, waves, or other imperfections to obtain a flat or properly contoured surface. If defects are not too great, a 320 grit stone will be sufficient. The stone should be moved back and forth (medium pressure applied), over the surface in a 90° direction from the direction made by the last operation. Before they are used, the stones should be soaked in a contaminant-free oil base lubricant.

The recommended stoning procedure for a cavity surface is as follows:

- Stone with 240 grit
- Stone at 90° to previous scratches with 320 grit until the previous scratches are removed
- Stone at 90° to previous scratches with 400 grit until these scratches are removed
- Stone at 90° to previous scratches with 900 grit

Polishing is the process of producing a series of overlapping “scratches” that get progressively finer. To accomplish this, it is important that for each finer grade stone used, the angle (direction) is changed relative to the marks made by the preceding coarser stone.

After each grit finish is completed, the entire workplace must be thoroughly washed with clean stoning oil and wiped with a clean tissue to remove all particles of the grit remaining on the surface. This is necessary to ensure that none of the particles of the coarser grit will be picked up at a later time by a finer grit stone, causing deeper scratches.

Here are a few hints pertaining to the use of polishing stones:

- Do not use a stone that is too coarse
- Always dress the polishing stone with a grinding wheel or coarse paper to provide the maximum contact with the work surface
- Use care when dressing the polishing stone
- Use sufficient stoning oil to prevent the stone from loading
- Hold the polishing stone firmly for directional control, but press only hard enough to make the stone cut
- Make sure the stone marks from previous grit size are all crossed out
- Change stoning direction with each successive grit
- Clean the work area thoroughly between each change of grit
- Keep each grit of polishing stone in a separate stoning oil can
- Exercise utmost care when stoning at an edge (parting line).

If the mold cavities and cores need heat treatment, the question often arises as to how far the polishing should proceed before the part is heat treated.

After the heat treating process, the knock out holes (or any other holes in the work area) must be covered or filled to avoid “dishing” (rounding off sharp edges) as the finishing continues. A 240 or 320 grit stone will remove the scale developed during the heat treatment. Make sure to “cross” stone the last marks until they are removed. Then proceed to finer stones, such as 500, 600, or 900 grit.

Surface finishes from stoning are often sufficient after using a 600 to 900 grit stone. While the mold cavity surface exhibits a soft, “matte” finish, it will be smooth and flat enough to allow easy ejection of the thermoplastic molded parts.

Luster or shine can be developed on this type of surface with some 500 to 600 grit paper that will remove the fine stone marks. Other methods for developing the final “shine” on the surface include the use of diamond compound with brushes and felt bobs.

Orange peel, the lightly dimpled finish that often appears during the final buffing operations of polishing, can be caused by several conditions. If the cavity insert steel has been overheated during heat treatment, the steel structure is changed resulting in an inconsistent surface hardness. This permits the softer surfaces to be deformed or abraded away more quickly than the harder surfaces. Consequently, care should be taken during heat treatment to ensure uniform surface hardness. If polishing pressure is too high, this softer area can be torn from the surface,

leaving pits. By looking at the pits through a magnifying glass, a little “tail” at the edge of the pit following the direction of polishing can be seen.

The following factors may contribute to the formation of orange peel and pits in polished mold cavity inserts:

- Low mold cavity surface hardness due either to an incorrect tool steel composition or improper heat treatment
- Presence of excessive nonmetallic particles
- Presence of internal porosity, particularly at the center regions of mold cavity inserts of relatively large cross section
- Retained austenite on the mold cavity surface usually caused by overheating during hardening or carburizing. Retained austenite has lower strength than martensite; therefore, it may cause deformation and breakage
- Clustering of carbides causes overheating, nonuniform hardness, and low strength areas and the removal of the carbide particles can result in scratches during the carburizing process.

If orange peel or pitting occurs, it may be possible to repair the mold cavity surface by first removing the defective condition by hand polishing with a fine stone, stress relieving at 100 °F below the tempering temperature, and hand polishing with diamond paste using light pressure and appropriate passes.

If orange peel or pitting is caused by excessive retained austenite, this condition can usually be alleviated by transforming the retained austenite using additional tempering treatments. Using a second additional tempering treatment allows tempering of the brittle martensite formed during the first tempering treatment. Another method of reducing the retained austenite content is to cool the mold cavity insert to a temperature between 100° and 150 °F. A subsequent tempering treatment of the brittle martensite formed during this cold treatment is advisable.

Over-carburized or pack-hardened steel can also produce similar defects. In these instances, the orange peel is caused by a similar inconsistent hardness of the surface, but the pits are the result of very hard particles of iron carbide being pulled out of the surface. Most of these pits will have no tails. Orange peel can also appear on the surface of properly heat treated steel that has been finished with powered, mechanical devices. In this case, the surface defect is caused by excessive pressure of the polishing implement (brush, felt, etc.) against the steel, or by the over-polishing of the mold cavity surface.

The harder the tool steel, the better the “polish” that can be achieved. Since orange peel occurs when the tool steel is stressed above its yield point (the yield point of steel can be increased by hardening or nitrating), it follows that the softer tool steels are more prone to orange peel. Remember, over polishing and polishing with too much pressure are the causes of pitting and orange peel.

Diamond Polishing

Polishing of thermoplastic injection mold cavity surfaces with abrasives is carried out either by some type of rotating machine or by hand. Machine polishing is more economical, but avoiding over-polishing caused by excessive pressure and/or speed is more difficult. Hand polishing usually involves preliminary work with a series of silicon carbide stones (240 to 900 grit) and a finishing operation with

diamond pastes from 15.0 to 2.0 micron. There is less chance of over-polishing if light pressures are used. Severe buffing or the use of loaded stones may result in severe residual stress and pitting.

The final diamond polish is the last step in the process of polishing; if one of the previous stoning steps has not been done properly, the final smooth luster surface will not be satisfactory. If mistakes have been made earlier during the finishing process, they will certainly show up on the final surfaces.

Stone finishing the mold cavity surfaces involves the preparation of the tool steel surface by producing a series of crisscrossing “scratches” that will get finer and finer. Coarse grits remove more metal and cut deeper. As the surface gets smoother and the surface begins to develop, it means that the peaks are being lowered to the depths of the valleys. In other words, the RMS measurement of the surface irregularity is becoming more minute.

Since there is a practical limit to the size of the “scratch” that can be achieved with conventional abrasives, very small grit sizes of diamond must be used in order to obtain an optical quality polish or a Number One (1) finish.

There is also another fundamental difference between the finishing process that has been done up to now and the final diamond polish. The stoning process prepares the surface for the final polish or the honing operation, in which the stone has the abrasive ability to cut the metal. On the other hand, diamond polishing is more a lapping operation. Fine diamond compounds remove very little metal while developing the final luster of the finish.

There are more than a dozen grades of diamond compounds ranging from 120 for fast cutting to 14,000 for super finishing, which are used to polish the mold cavity surfaces.

The starting point of diamond polishing will depend, to some degree, on the sequence of stones that have been used to prepare the mold cavity surface.

Beginning with a coarser diamond grade, a small amount of the compound is applied directly to the surface being worked. Then, by means of a bristle, brass, or steel brush, the compound is swirled over the surface using a rotary tool at slow speed. A speed of 500 rpm for roughing and 5,000 to 10,000 rpm maximum for final polishing is a good general rule. Using light to moderate pressure, care must be taken to keep the brush flat to the mold cavity surface to avoid cutting deep swirl marks. A “crisscrossing” action should be employed when using diamond compounds. The compound in use will become darker, indicating that metal is being removed and mixed with the compound.

The surface should be brushed until all that is visible are fine swirly marks left by the brush’s rotary action. There should be no stoning marks visible at all.

The next step, the removal of the swirly marks left by the bristle brush, is accomplished with a felt product. Felt bobs are available in various degrees of hardness, preassembled in a shanked nylon holder. Mounted in a rotary tool and using light to moderate pressure, the surface is polished with diamond compound until all that is visible are felt swirls.

The cavity surface should be thoroughly cleaned to remove all residual particles of the previous grade, before applying a finer diamond compound.

The final step in polishing with diamond compound is a hand operation. Depending on the various configurations of the mold cavity surface, fine tissue paper, felt sticks or cotton swabs may be used with an ultra fine grade of compound to arrive at the final high gloss luster.

A better polishing surface is found in the fiber direction than across the direction of the fiber. If the fiber direction can be made to correspond to the largest cavity surface, less difficulty from pitting during polishing will be encountered.

For finishing a stainless steel cavity surface, it is important to use polishing and buffing compounds without iron oxide. If iron oxide particles are impregnated into the surface of a stainless steel, rust spots and surface pits will develop.

The following diamond polishing guidelines should be used:

- Apply a small amount of diamond compound at first, then add more if required
- Do not mix the diamond compound grades
- If the diamond compound gets dry or hard, add a clean diamond thinner or lubricant
- If the first grade of diamond compound used does not remove marks from the last stoning operation, **STOP!** These marks must be removed with a coarser grade of compound or you will end up with highly polished, shiny scratches
- Clean mold cavity surfaces thoroughly before progressing to a finer grade of diamond compound
- Do not use more than one grade of diamond compound on the same brush, felt, or lap
- Be sure that each step completely removes marks left from the previous step
- Never use an abrasive stick or a piece of abrasive cloth because the grit could become embedded in the lap and cause serious scratches on the mold cavity surface.

10.8 Thermoplastic Injection Mold Bases

The majority of the thermoplastic injection mold designs falls within the two-plate mold categories. However, several other mold designs are also used by the plastics industry. These molds are known as two-plate, interchangeable cavity inserts, vertical encapsulation, lost core, three-plate, hot runner, insulated runnerless, and stacked types of molds.

10.8.1 Standard Mold Base Components

Because of the similarities in the two-plate mold construction, it is desirable to have some standard mold base available to permit the thermoplastic injection molds to be produced in quantity, with a short delivery time, thereby reducing manufacturing costs. Logically, it is advantageous for the mold makers to purchase a mold base quickly at reasonable cost, rather than expending engineering mold design time, steel selection, construction, and assembly of the mold base components. The mold base and component manufacturers have developed tool technologies for the new engineering thermoplastic polymers, producing a range of standard high quality products that are available, ready to use in the designs and manufacture of thermoplastic injection molds.

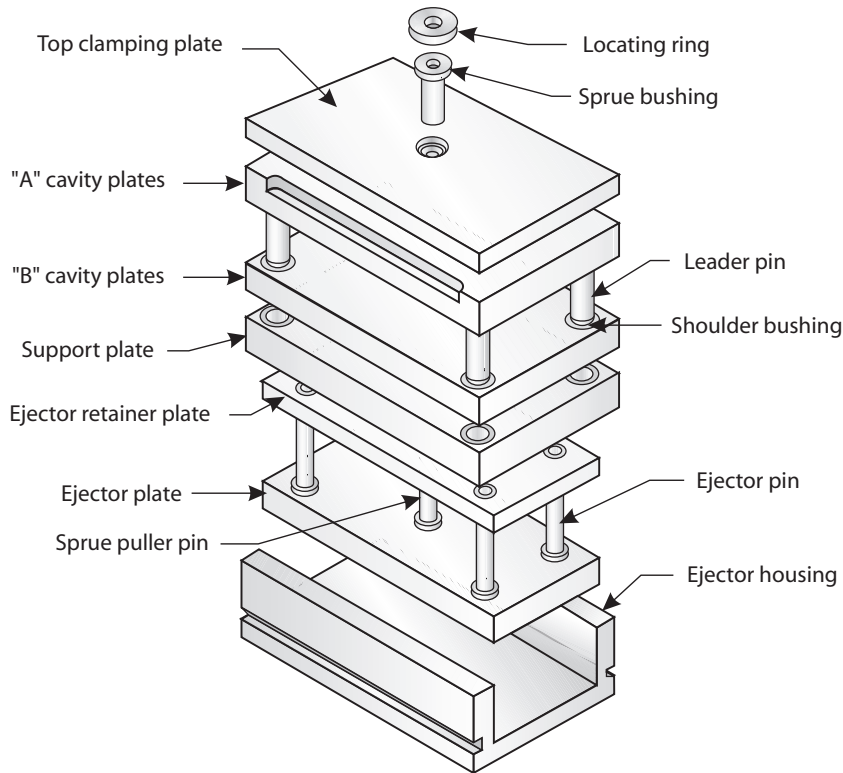


Figure 10-4 Standard two-plate mold base components

A thermoplastic injection mold base may be defined as an assembly of mold components that conforms to an accepted structural shape and size. The proper type of mold base is specified by the number of plates, premachining of the plates (if desired), steel selection, dimensions, ejection travel distance, size and type of locating ring and sprue bushing required by the injection molding machine. The remaining mold base components are suitably attached together and a guidance system incorporated. Of course, the mold base does not include runners, cavities, cooling, venting, hardness, finishing etc.; these aspects of mold manufacturing must be left to a specialized mold maker.

The thermoplastic injection two-plate mold has been adopted as the standard mold base by mold component manufacturers because this particular mold construction is the most widely used design in industrial practice.

Mold bases in a wide range of sizes made to suit a variety of purposes are produced by a number of manufacturers. These standard mold bases need only to be machined to make the mold components for a particular part. It is necessary to know the terminology and function of the components that make up the mold base. Figure 10-4 shows the location and nomenclature of the basic thermoplastic injection two-plate mold base.

10.8.2 Functions of the Mold Base Components

The most important features of standard thermoplastic injection mold bases are reviewed here to provide the reader with a working knowledge of the basic components and their functions.

Top Clamping Plate

The top clamping plate supports the “A” cavity plate, locating ring, and sprue bushing. In some mold designs, these two plates may be combined into a single, thicker plate, which serves both functions. It holds the stationary half of the mold to the stationary platen of the injection molding machine.

Locating Ring

The locating ring is that portion of the mold that is fitted into the stationary platen of the injection machine. Its intended purpose is to properly situate the mold in relation to the injection nozzle of the machine. Most standard injection machines use different diameters with a variety of design features to accommodate the platen entry for the nozzle. The correct sizes of locating rings are provided in the molds to fit a particular type of injection machine. Where relatively thin-walled parts are being molded and injection pressures may be high, the locating ring may be required to retain the sprue bushing within the mold, so the nozzle and sprue bushing are aligned.

Sprue Bushing

The sprue bushing seals off the melt from the injection nozzle conveying the melt through a conical shaped internal channel and forcing it into the sprue puller, runner, gate, and cavity confines of the mold itself. The sprue bushing has a tapered internal round channel that can vary in size as needed. In proper mold design, the sprue is made as short as possible, consistent with a given part design, thereby reducing the injection pressure drop in the runner system. Both, the locating ring and the sprue bushing, are usually supported by the top clamping plate, which is used to support the stationary half of the mold.

“A” Cavity Plate

The “A” cavity plate contains and supports the cavity or cavities or the core insert, sprue bushing, and the runners for the parts to be molded. In some cases, the cavity may be cut directly into the solid steel plate, while in others the cavities can be constructed separately and inserted into pockets within the cavity plate. The “A” cavity plate is part of the stationary section of the mold half; this plate is where the leader pins are mounted.

“B” Cavity Plate

The “B” cavity plate contains or supports the other half of the cavity or a core section of the molded part and also contains the leader pin bushings. The plane between these two plates is the normal parting line of the mold, which separates the two halves of the tool. The “B” cavity plate is the top plate of the movable section of the mold half. It is used to hold the sprue puller and ejector pins as well as the core inserts, or the cavity inserts.

Support Plate

The “B” cavity plate is mounted on top of the support plate. The support plate is used to provide strength to the cavities to avoid deflection during melt injection inside the cavities.

Ejector Housing

The ejector housing parallel blocks are added to provide the height required for the movement of the ejector system. The base plate of the ejector housing is used

for clamping the moving half of the mold to the moving platen of the machine. The ejector housing is a single unit for the ejection system. The injection mold base is manufactured with the parallels (vertical supports) welded to the bottom clamping plate.

Bottom Clamping Plate

A bottom clamping plate secures the movable half of the mold to the movable platen of the injection molding machine. If the mold is exceptionally large, the ejector system may require additional support, provided by the insertion of support pillars that bear the load between the bottom support plate and bottom clamping plate.

Ejector Retainer Plate

Mounted on top of the ejector plate, this plate retains the ejector head pins, ejector return pins, and sprue puller pin through counter bored holes.

Ejector Plate

The ejector plate is bolted together with the ejector retainer plate to form a unit. It acts as a back support plate for the ejector pins, return pins, and the knockout bar. These pins pass through drilled holes in the “B” cavity plate, insert cavity, and support plate.

Stop Pins

The stop pins are mounted on top of the bottom clamping plate; they are used as stops for the ejector housing when the ejector system returns as the mold closes.

Support Pillars

The support pillars are round bars placed between the support plate and the bottom clamping plate; they have the same height as the parallels. Bolted to the bottom clamping plate, they are used as additional support to avoid deflection of the “B” cavity plate.

Sprue Puller Pin

Pin located below the main runner, directly under the large diameter of the sprue channel. It is used to pull the solid sprue out of the bushing automatically when the mold opens and the molded parts and runner system are ejected.

Ejector Pins

The ejector pins enter the cavity to make contact with the molded part.

Return Pins

The return pins contact the stationary cavity plate and prompt the movement of the ejector plates back to the normal position prior to the next injection shot (not identified in Figure 10-4).

Leader Pins

The leader pins, used to align the plates on the closing of the mold, are hardened and ground steel pins mounted into one of the mold halves. One of the leader pins is offset so that the mold halves can only be closed when the leader pins are in the correct relative position.

Shoulder Bushings

Hardened and ground steel bushings are mounted into the other half of the mold, in-line with the leader pins. They serve as bearing surfaces for the leader pins.

10.8.3 Types of Standard Mold Bases

Figure 10-5 shows two basic styles of mold bases; one has a rectangular cross section (this is the most common form), the other is round.

The rectangular cross section mold bases are commercially available in various standard configurations as shown in Figure 10-6.

It is important for the mold designer to appreciate that the majority of mold designs can be broadly classified within one of these mold base types. For example, a slide-type of mold can be described as a two-plate mold with the addition of a slide assembly.

The mold designer may have to accept a compromise in mold size; mold base types, and the range of sizes; for certain molds, some modification to the mold base is necessary to accommodate a specific feature. Considered overall, however, the advantages of using standard mold bases outweigh the disadvantages and for a large number of mold designs, the standard mold base can be beneficially used by the mold designer, mold maker, and end user. Nevertheless, on occasion it is necessary to have a “made-to-measure mold” so a special feature may be incorporated or a specific mold cooling system be adopted, or simply because a standard mold base of a suitable shape or size is not available.



Figure 10-5 Rectangular and round standard mold bases

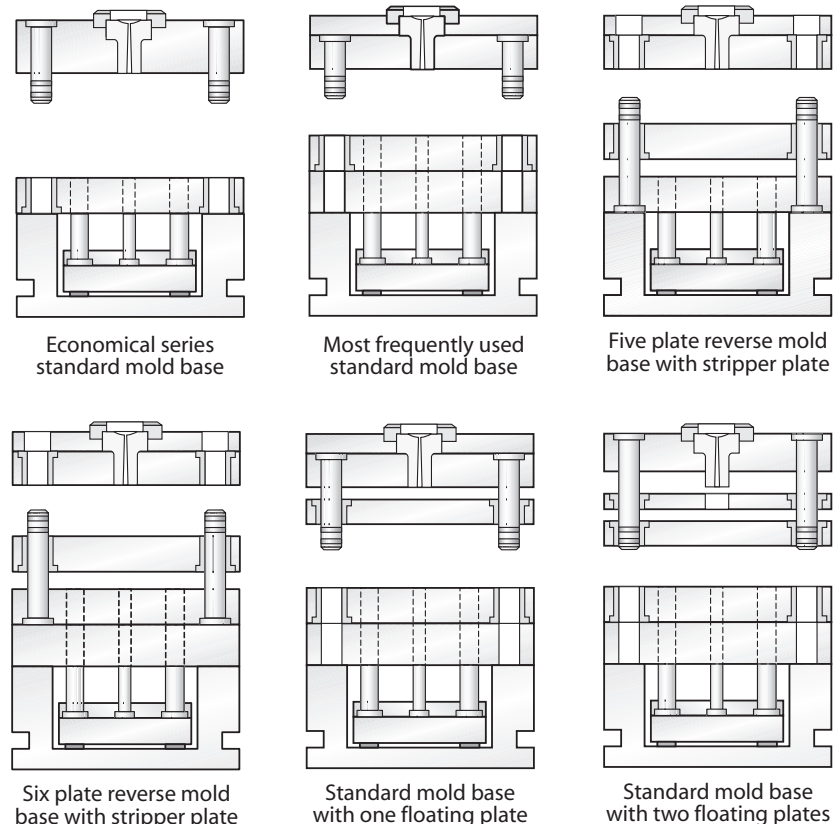


Figure 10-6 Types of standard commercial mold bases (rectangular)

Advantages of Standard Mold Bases

- Design drawing for individual unit sizes, reduces mold design time
- Less steel needs to be carried in stock, investment is reduced
- Buying and stock controls are simplified
- Mold base price is known, estimating the cost of the mold is easier
- Waiting time for steel blanks, etc., is avoided
- Shaping, planing, and drilling of steel plates and blocks is avoided
- Machining, grinding, hardening, fitting of pins and sleeves is avoided
- The ejector plate is already in place
- The individual mold plates are screwed and doweled together
- Machining time and labor costs are reduced
- Design work on the insert cavities can usually begin immediately
- Mold base components are standard; if a component is damaged during manufacture or in production, the part can be quickly replaced
- Mold maintenance and down time is reduced
- Mold delivery time is reduced
- Efficient mold maker's engineering and use of production labor, provide mold cost reductions, satisfied customers and higher profits.

Disadvantages of Standard Mold Bases

- Number of mold base sizes available is limited
- Large size mold bases are limited
- Maximum depth of mold base plates is also relatively small
- The ejector system travel may be larger than is actually required
- Cooling channels in a desired mold location are difficult to obtain because of conflict with other mold base components
- Extra number of support pillars positioned relatively far apart are needed to avoid deflection of the mold "B" cavity plate
- The support plate cannot be unscrewed independently to expose the ejector assembly; the mold's moving half must be disassembled
- Limited number of special tool steel plates for the standard mold bases.

10.9 Types of Thermoplastic Injection Molds

Designs for injection molds differ depending on the product design geometry, end use applications, product tolerances (type of gate and location), product life, allowable product cost, molding production requirements, efficiency (maximum economy), amount of mechanization, factors that will dictate the size of the mold and the type of thermoplastic material being molded.

The most common types of injection mold designs are the following:

10.9.1 Two-Plate Molds

Figure 10-7 shows a so called two-plate mold that certainly has more than two plates. It is the common name for a thermoplastic injection mold with a single parting line. The parting line of a mold can best be defined as that surface where both halves of the mold separate to permit the injection molded parts and runners to be ejected from the mold. One side of the cavity is mounted in the "A" cavity plate with the locating ring and sprue bushing assembled into the stationary half of the mold.

The "B" cavity plate is part of the moving half of the mold and contains the cores, the runner systems, and the ejector system, that are operated by the knockout bar that is attached to the machine actuator and the ejector plate assembly, moving the ejector pins and sprue puller pin forward to eject the molded parts and the runner system when the mold opens.

The molten thermoplastic is fed directly from the machine nozzle through the sprue bushing that is connected to the runner and edge gates, filling the cavities with melt. Then the mold cavities cool off, forming the final solid product (small containers).

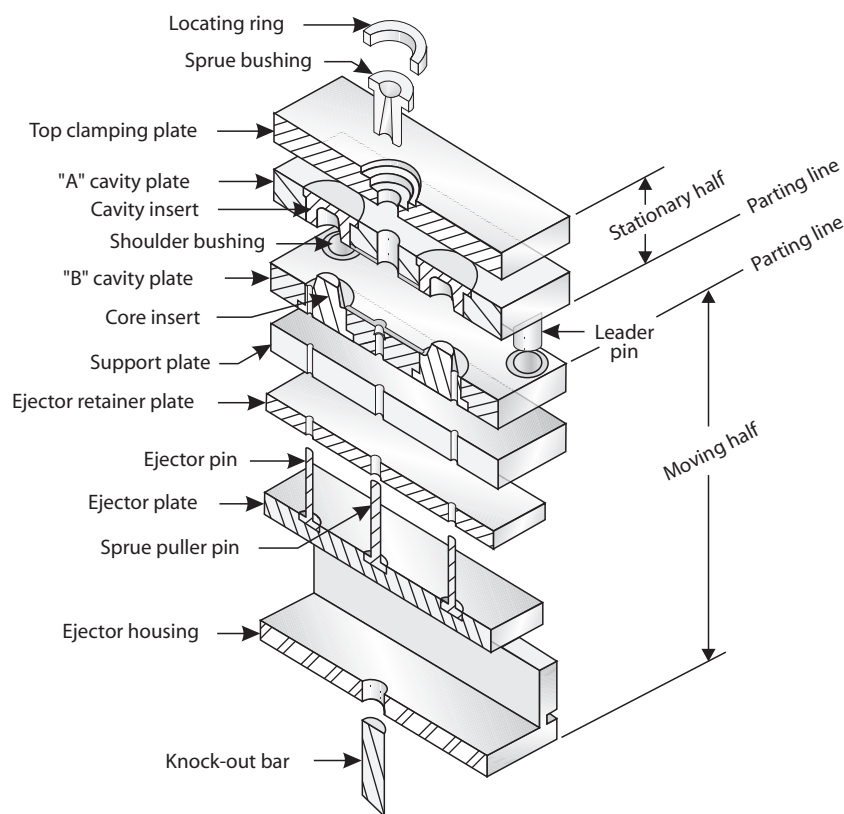


Figure 10-7 Two-plate mold and components (cross section)

10.9.2 Round Mate® Interchangeable Insert Molds

The Round Mate® System consists of a circular master frame with easily removable interchangeable round inserts, as shown in Figure 10-8. For high injection molding production operations, or to produce different parts at the same time, the use of multi-cavity inserts and a hot runner system is available in all sizes. The Round Mate® System provides the flexibility and ease of operation to change inserts in the injection molding machine in just ten minutes. This Round Mate® System saves needless hours of mold making time. Using the universal Round Mate® Frame, standard machining and chase work are done once.

For new injection molded parts, only the individual, interchangeable inserts are needed at a reduced cost of a complete, traditional mold.

The Armoloy plated, AISI 4140 tool steel master frame is completely rust and corrosion resistant. The design of the inserts allows the cooling system to be wiped free of all mineral deposits and corrosion. Maintenance is reduced to a minimum.

Cavities and runner systems can be cut directly into the solid insert. Inserts are available in a choice of tool steels and can be heat treated or plated. SAE P20 (prehardened) and SAE H13 (heat treatable) are in stock for immediate shipment. Other tool steels, such as stainless steel 420, SAE S7, SAE A6, are available as special order. Master frames and inserts are available in 2.50, 4.00, 6.00 and 8.00 in diameters.

The Round Mate® system has several advantages:

- Fast standard mold delivery time
- Reduced mold costs. Design and labor time are reduced to a minimum because the master frame and mold insert blanks are standard. Normal chase work is already completed. The Round Mate® system has a unique water cooling jacket, cavity pockets, guided ejection system, and many other features already incorporated.
- Fast insert changeover. Simple hex-key wrenches are used to change the mold insert in the injection molding machine in less than ten minutes.
- Improved part quality. The benefits of a round shape include more uniform clamping pressure in the mold, balanced heat transfer and cooling, completely self-centering, more efficient runner layout, better part quality, and faster molding cycles.

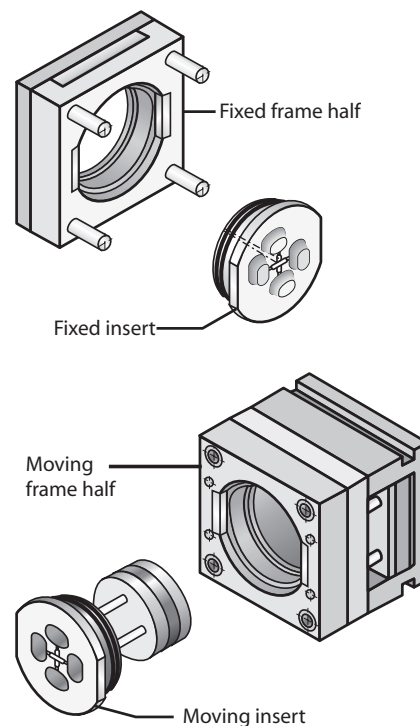


Figure 10-8 Round Mate® frame and interchangeable cavity inserts

10.9.3 Master Unit Die Interchangeable Insert Molds

With the advent of the standard mold base, Master Unit Die Products, Inc. has developed a system for the quick replacement of inserts, which are easily interchangeable within a single frame for all injection molding machine applications. This approach was certainly a major breakthrough in reducing mold costs for the plastics industry. By combining a standard mold frame with interchangeable mold inserts, this concept not only cuts mold building costs and reduces mold delivery time, it also increases productivity by reducing the injection molding machine down time significantly.

Master Unit Die Products, Inc. offers frames in four basic styles and companion inserts in two basic styles. These frames and their companion inserts are in use throughout the world. A typical MUD mold is shown in Figure 10-9.

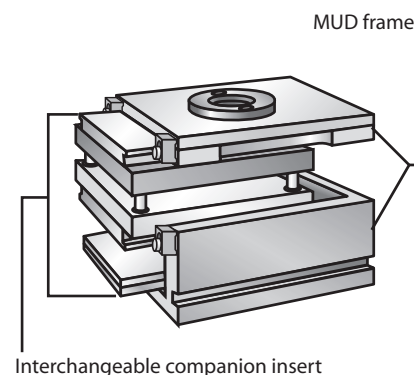


Figure 10-9 MUD mold frame and interchangeable cavity inserts

MUD systems find applications in different areas:

- *Prototype parts.* The MUD system is ideal for prototype parts. Fast mold design is achieved by not using the entire mold assembly. Prototype part mold costs can be reduced to an absolute minimum.
- *Short production runs.* The almost instant interchangeability of MUD inserts is a big advantage when a variety of parts are scheduled for production. Production time can be gained during changeovers.

MUD systems offer several advantages:

- *Lower mold costs.* The concept makes the inserts easily interchangeable within a single frame. Mold costs are reduced because only the insert is replaced, not an entire standard mold base
- *Quicker delivery.* The simplicity of the system means less time is required for mold fabrication. Standard blank inserts are usually available from stock within ten days
- *Faster setup.* The Master Unit Die frame remains in the injection molding machine with the cooling lines operational when inserts are interchanged. Initial production setup is simply a matter of sliding and clamping the insert in position (leader pins and bushings in the core and cavity plates assure alignment)
- *Instant changeover.* Inserts are easily removed and interchanged by loosening four clamps, disconnecting the cooling lines, removing one insert and sliding in another, then simply reversing the process, the complete change taking normally less than ten minutes
- *Minimum purging.* The injection molding machine's plastifying unit is less likely to overheat, requiring purging, because of instant changeover of the MUD inserts (material and production savings).
- *Easier maintenance and repair.* The MUD inserts are easier to remove and reinstall, they are lighter, smaller, easier to handle and store, which is a big plus when maintenance or repair is required
- *Greater flexibility.* Most Master Unit Die frames will accommodate two or more inserts. When single molding applications are scheduled, a blank insert can be installed in the other section of the frame.
- *Maximum versatility.* Inserts are available in "T" and standard styles. Mold design latitude is almost unlimited because inserts can be engineered for parts requiring stripper plates, sleeve ejection, single or double cam action, hydraulic, mechanical or pneumatic powered cylinders, and any feature desired including three or four plate molds.

10.9.4 Three-Plate Mold Cold Runner System

The three-plate mold design is used where a center pin point gate is required on a multi-cavity mold or when the molded part geometry requires a stripper plate ejection.

This three-plate mold system operates as follows: The mold is opened at the parting line and the sprue is pulled immediately by the sprue puller pin. The entire three-plate runner system thereby moves back with the moving cavity

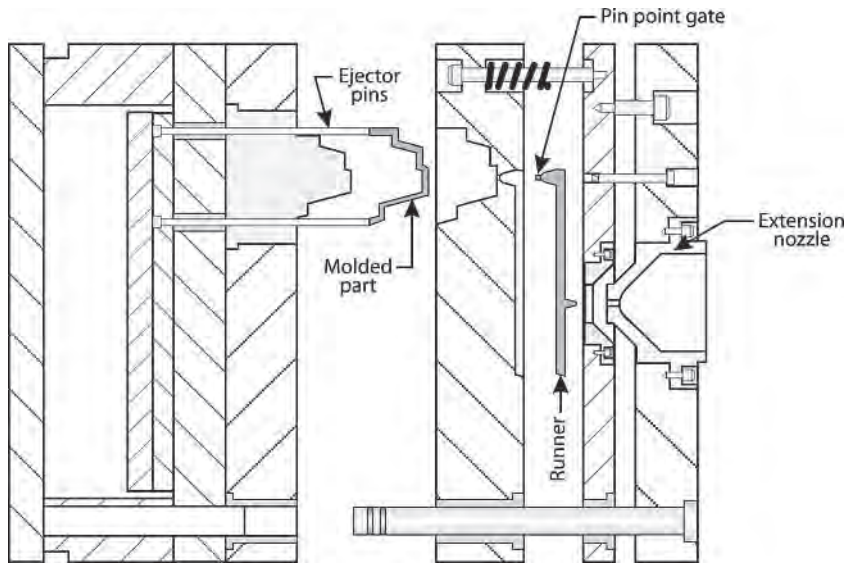


Figure 10-10 Three-plate mold cold runner system

plate, the runner is withdrawn from the sucker pin by the stripper plate. When the floating distance of the sucker pin has been taken up, the three-plate runner system with the pin point gate breaks off from the cavity as a result of the pull-off force produced by the sucker pin. The continuous movement of the “A” cavity plate causes the sucker pin to shear off from the three-plate runner and frees itself from the cavities ready for ejection. When the mold is closed, the plates are progressively returned to their original positions. An illustration of a three-plate mold system, cold runner, and a pin point gate is shown in Figure 10-10.

10.9.5 Vertical Insert Mold for Thermoplastic Encapsulations

The vertical mold clamp and horizontal injection system for encapsulating inserts was developed to produce electrical and electronic thermoplastic products such as electric cord plugs, transformers, motor housings, automotive speed and temperature sensors, etc. Figure 10-11 shows a thermoplastic encapsulated automotive ABS sensor and a fuel injector produced by this molding process.

The properties of special thermoplastic injection molding polymers are ideal for encapsulating various types of inserts by using vertical clamp and horizontal injection molds. These polymers have excellent electrical properties, high end use temperatures, and good chemical resistance. Some of these thermoplastic resins are also “wire friendly” for encapsulating delicate electrical wires and leads.

Figure 10-12 shows a typical vertical clamp and horizontal injection mold for encapsulating wire wound bobbins and magnet inserts for the production of electronic ABS sensors.

This type of mold may look similar to a two-plate mold, but it operates completely differently. The upper half of the mold (top clamping and “A” cavity plates) is mounted to the moving vertical platen of the machine, but it does not have a sprue bushing and a locating ring. The lower half of the mold is mounted to the fixed platen, shuttle platen, or rotating table of the machine, depending on

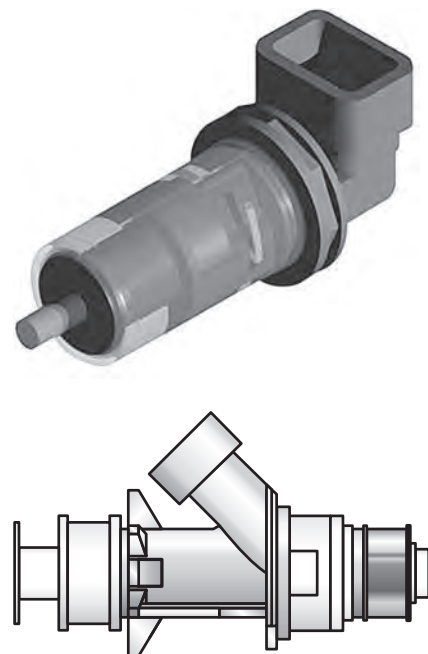


Figure 10-11 Thermoplastic encapsulated ABS sensor and fuel injector

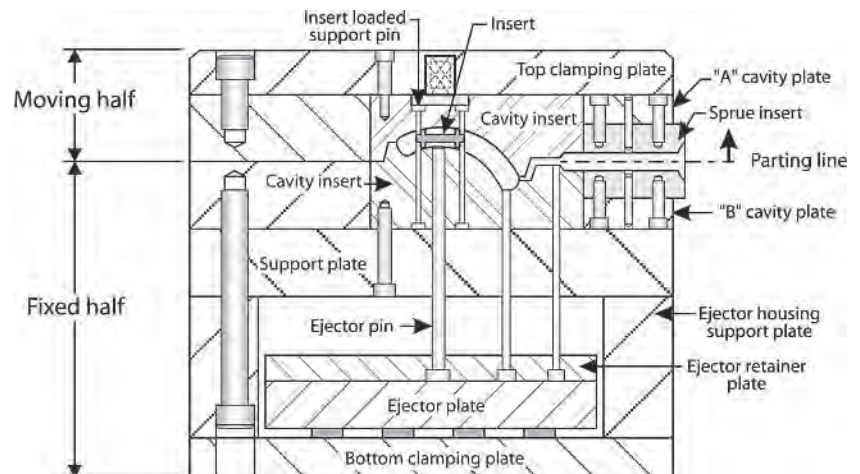


Figure 10-12 Vertical clamp, horizontal injection unit

the application. The ejection system is located in this lower half and connected to the knock-out bar that is operated by a hydraulic actuator. The injection unit is in-line with the mold parting line; the melt is injected into the split sprue bushing (mounted in each half of the mold) that is connected directly to the main cold runner and sub runners, distributing the melt through the gates to the cavities.

The inserts are loaded horizontally in the lower cavities, on top of the support pins. When the mold is closed, the spring loaded upper support pins lock the inserts in position, preventing the inserts from moving during the melt injection in the cavities. The melt shrinks around the inserts forming a good mechanical bond between the inserts and the thermoplastic during the mold cavity cooling. The molded parts and the runner system remain in the lower half of the mold, when the mold opens, the ejector pins push the molded parts out from the lower cavities and the operator removes the parts and loads the inserts to start a new molding cycle.

10.9.6 Hot Runner Molding Systems

In a hot runner system the melt directly fills the cavity from which the solidified thermoplastic part is ejected when the mold opens. The melt is conducted through heated channels through the runnerless manifold where the melt is proportionally subdivided, depending on the number of cavities and injected to the cavities through the temperature controlled gates or drops. Hot runner molding can result in as much as a 50% cost savings, part quality is improved, cycle time is faster, and since parts are finished when they leave the mold, it is not necessary to remove sprues or runners. Hot runner molding systems eliminate the use of cold runners, the melt is maintained at a constant temperature until it reaches the cavities, providing a uniform melt viscosity and better injection pressure control reducing the melt shot capacity. The hot runner molding systems are more expensive, they require an additional temperature controller, higher maintenance, and operational training.

Figure 10-13 shows a typical hot runner mold cross section illustrating the operational sequences of this molding process.

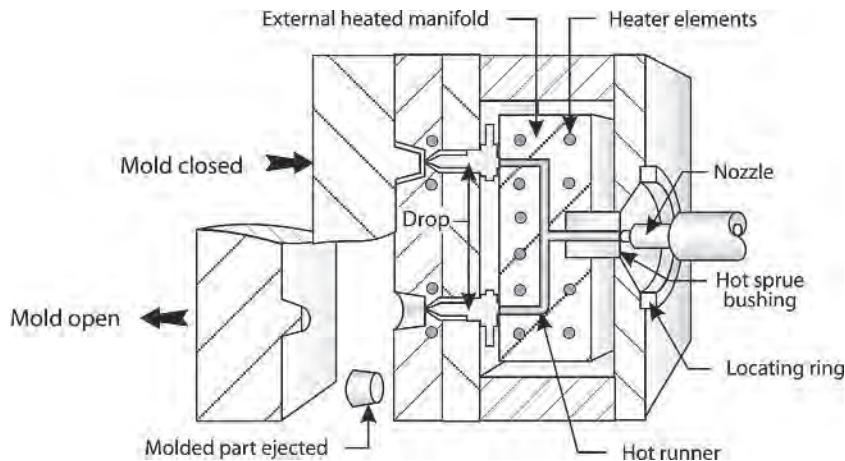


Figure 10-13 Typical hot runner mold system (cross section)

10.9.7 Hot Runner Mold Temperature Control Systems

Second only to gating, the most common source of problems in hot runner molds is temperature control throughout the system. A high degree of thermal uniformity in the manifold and from drop to drop is essential to avoid degradation of resin and to obtain uniform filling, packing, shrinkage, appearance, etc. in all cavities. Each drop has a different temperature profile. The temperature setting for drops may be substantially higher than in the manifold. Manifold temperature should not exceed the process melt temperature. However, lower manifold temperatures will minimize minor streamlining flaws with thermally sensitive resins. If residence time in the manifold is short, a low manifold temperature will have little effect on the temperature of the flowing melt.

Some of the devices used to maintain uniform temperature control are the following:

Cartridge Heaters

The runnerless manifold temperature is controlled with cartridge heaters. The temperature is dependent on the cartridge heater design, wattage, location, support, and installation tolerances of heaters, thermocouples, thermal conductivity of metals used and independent zone controls. Serious problems may result when a single cartridge heater fails, causing the controller to increase heat from other cartridges in the same zone. Temperature uniformity will be upset by the development of hot or cold areas, increasing the overall manifold temperature. Loose fitting cartridges also tend to burn out more frequently.

Cartridge heaters are also used as internal heaters inside the manifold runner channels or as torpedoes for drops. Different installation tolerances will cause temperature variations because the thermocouple in the cartridge heater is insensitive to fit problems.

Tubular Heaters

Tubular heaters, similar to those used to heat the oven of an electric range, are also used to heat hot runner manifolds. They are pressed into grooves on both

sides of the manifold plate. They may be shaped and configured to follow the melt flow paths around the connecting area of each drop. They also provide a more uniform heat input along the length of the heater and are more resistant to burnout because fewer individual heaters are used.

Cast-in Heaters

Thermally conductive metals such as copper and copper alloys are often used in hot sprue bushings, portions of manifolds, and drops to improve heat distribution and uniformity. Permanent heater elements are often cast into these metals to improve durability. Although failure is much less frequent, they are more expensive.

Heat Pipes

Kona Corporation has patents on the use of heat pipes to obtain uniform temperatures in manifolds and drops. These devices not only conduct heat rapidly from one area to another, but they tend to equalize temperature along their path by releasing more heat to colder areas. This is a natural result of conducting heat by vaporizing and condensing a liquid in the heat pipe. The colder the area, the more liquid will condense and the more heat will be released. Heat pipes require external heater bands located only at the end of the bushing. They also make it possible to reduce manifold temperature below that needed to keep gates operating, which is an important consideration for heat sensitive resins.

Bands and Coil Heaters

These are most often used in sprue bushing, pipe connected manifolds and drops. Coil heaters lend themselves to the necessary wattage distribution to avoid overheating the center section of a bushing in an effort to provide enough heat to keep the gate open. Placement of band heaters close to the gate area and use of conductive metal in drops or sleeves can also control the temperature distribution.

Torpedo Heaters

They are used in sprue bushings, drops, and internally heated manifolds. These torpedo heaters are helpful in distributing wattage effectively in drops and internally heated manifolds. Manufacturers of torpedo heaters have made significant progress in overcoming the inherent lack of temperature uniformity of heated manifolds or torpedoes supported at one or both ends in cold steel mold plates.

10.9.8 Hot Runner Mold Gates (Drops)

The most critical hot runner mold design parameter is the gate. The thermoplastic melt must be kept in a fluid condition up to the point of separation. The gate area must freeze rapidly and without serious flaws. Control of gate freezing along with gate vestige, part quality, and appearance are not only functions of temperature control, but of the specific gate design details needed to avoid serious molding problems. A small insulating ring of titanium (very low heat conducting metal) between the bushing and cavity is used to maintain the temperature differential at the gate.

The types of gates used with the hot runner molds are the following:

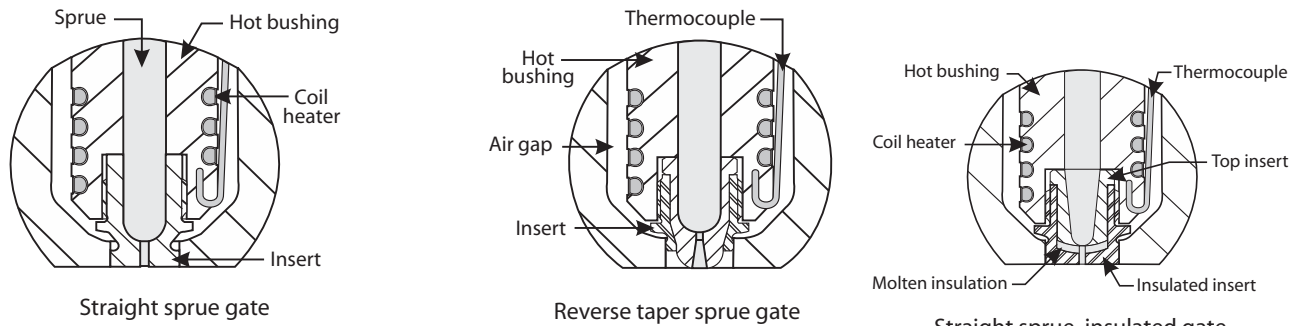


Figure 10-14 Two sprue gates (drops), straight and reverse taper

Straight Sprue Gate

An open gate is a simple, restricted opening to the cavity. It is usually difficult to keep this gate from freezing with semi-crystalline and high melt temperature resins, which leave a long vestige. A straight sprue gate is shown in Figure 10-14, left illustration.

Reverse Taper Sprue Gate

The reverse taper gate is a modification of the straight sprue gate. A reverse taper sprue gate land between 0.25 and 0.75 in moves the separation point away from the cold cavity and into the bushing where it can be controlled better, making this gate more suitable for engineering polymers. The resulting long gate vestige on the part is a serious disadvantage. A reverse taper sprue gate is shown in Figure 10-14, right illustration.

Straight Sprue Molten Insulated Gate

Figure 10-15 shows these types of gates; they are straight sprue gates with molten insulated traps for melt surrounding the gate close to the cavity. These traps are intended to provide an impediment to heat transfer away from the gate. Much of the thermoplastic trapped in this insulated melt may freeze, but portions closest to the gate may be hot, which constitutes a serious hold-up spot for heat sensitive resins.

Tit Edge Gate

The concept of a hot bushing drop feeding two or four side tit edge gates is an attractive way to simplify runnerless molds for small parts. Current designs do not appear to be well suited to control gate freezing and separation for semi-crystalline and high temperature resins. Tit edge gates tend to shear flush with the gate opening and leave a frozen stub in the gate, which either freezes solidly preventing subsequent shots or is injected into the next shot as a cold slug. Figure 10-16 shows these types of gates.

Hot Tip Spreader Insert Gate

The hot tip spreader is a small, conical insert in the gate end of a hot bushing drop made of copper alloys to aid heat transfer. The spreader's internal tip contacts the inside diameter of the hot bushing and draws heat from the melt and bushing heater. Several streamlined passages around the cone allow melt to pass into the cavity. The purpose of this spreader is to deliver heat to the gate separation point (prevent freezing) and to establish the exact location of the separation point.

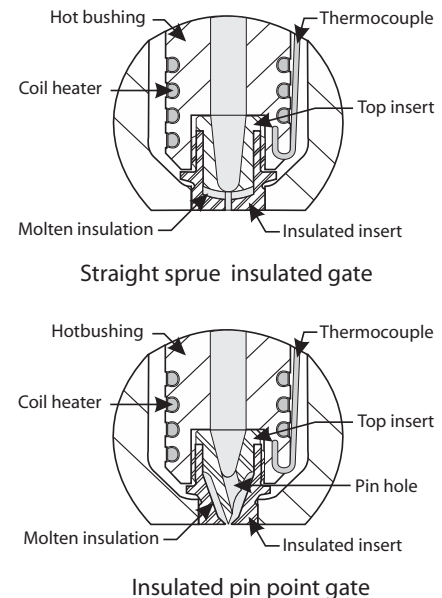


Figure 10-15 Two insulated gates, straight sprue and pin point

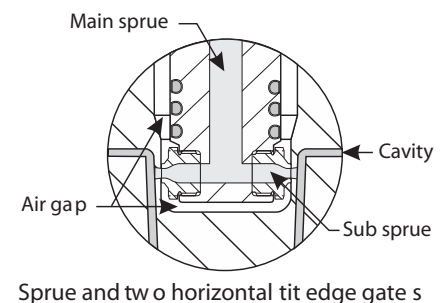


Figure 10-16 Hot runner drops, main and sub sprues tit edge gates

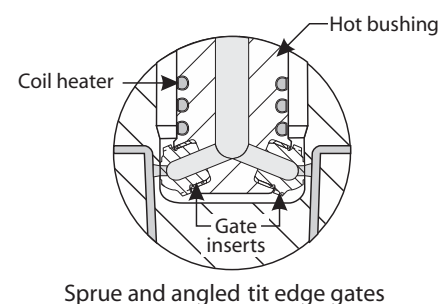


Figure 10-16 Hot runner drops, main and sub sprues tit edge gates

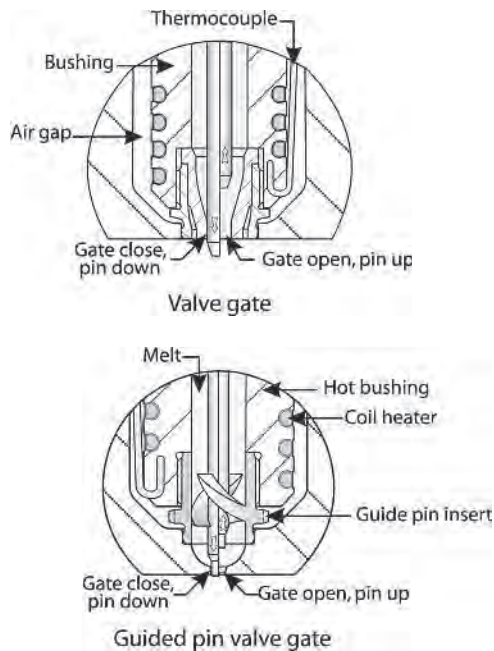


Figure 10-18 Hot runner drops, reciprocating pin valve gates (Courtesy: Mold Master)

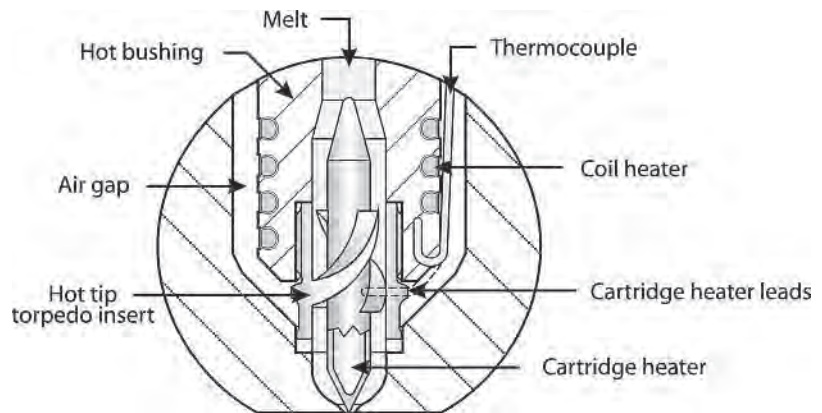


Figure 10-17 Hot runner drop, hot tip spreader gate

The external spreader tip is very close to the cavity surface and it tends to limit gate vestige. Figure 10-17 shows this type of gate.

Reciprocating Pin Valve Gate

A valve gate is an open gate with a reciprocating valve pin. The valve is opened and closed during each cycle by means of a hydraulic, pneumatic, or spring mechanism. Valve pin gates provide maximum control of gate vestige and cosmetics, but they add complexity and introduce serious wear problems for glass or mineral reinforced thermoplastic resins. Valve gates also tend to push cold slugs into the gate area of the part, causing lower impact resistance or appearance problems. Figure 10-18 shows a pin valve gate (top illustration) and a guided pin valve gate (bottom illustration).

Hot Tip Fixed and Valve Torpedo Gates

These types of gate are established, when an internally heated hot tip torpedo is used in the drop. The hot tip torpedo approaches or enters the gate area. The reciprocating hot tip torpedo works as a valve pin controlling melt flow at the gate. The torpedo is forced to retract from the gate by the injection force of the melt on the torpedo tip (gate open), just before the melt is injected. When injection is complete, Belleville springs above the torpedo flange support, which were compressed during injection, cause the torpedo to close the gate. Figure 10-19 shows three types of hot tip torpedo gates.

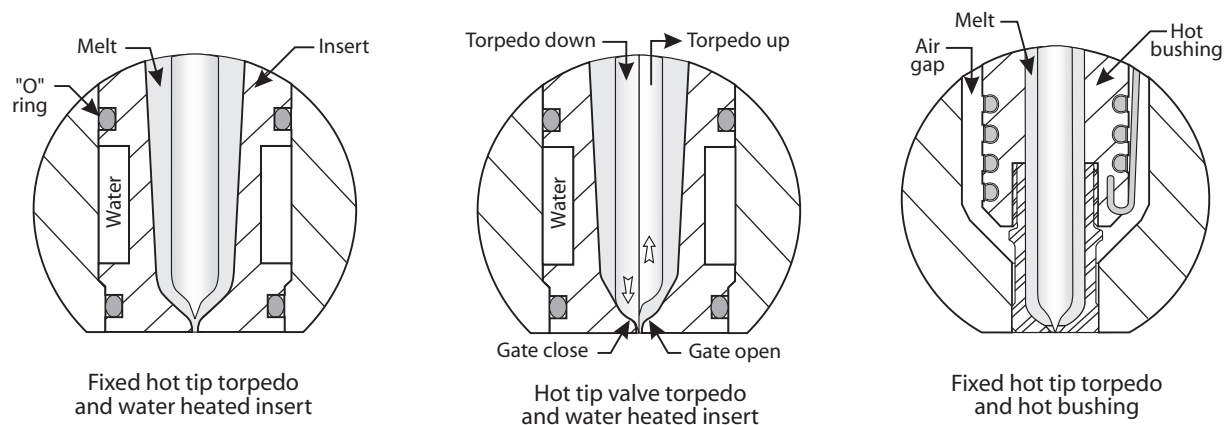


Figure 10-19 Hot runner drops, hot tip fixed and valve torpedo gates

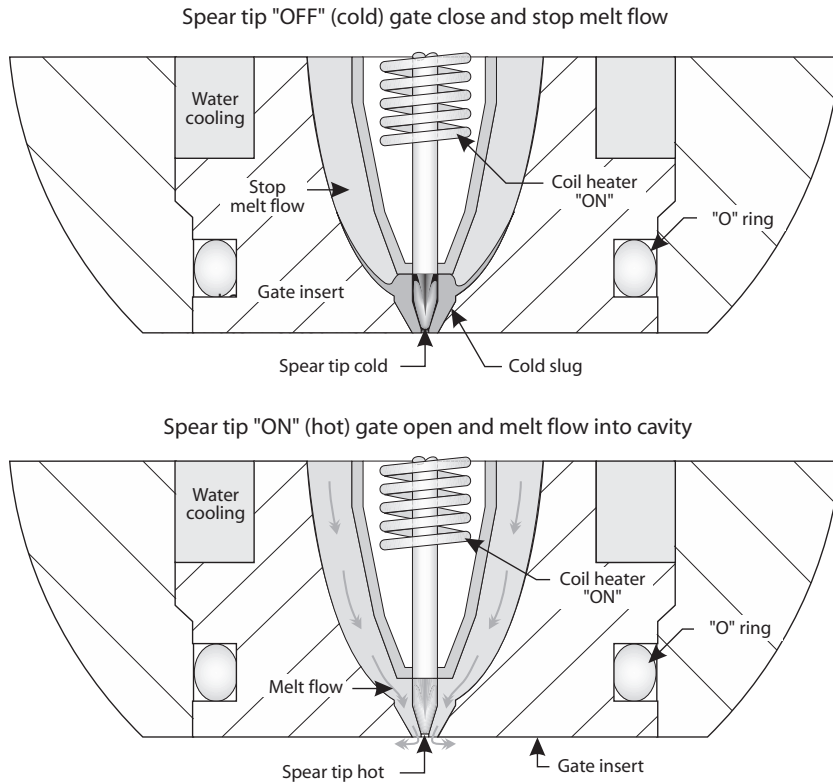


Figure 10-20 Hot spear tip thermo-valve gate
 (A) spear tip "OFF" (cold) gate closed and melt flow stops;
 (B) spear tip thermo-valve gate "ON" (hot) gate opens and melt flows into cavity

Hot Spear Tip Thermo-Valve Gate

This type of thermo-valve gate is a fixed hot tip torpedo with dual temperature control probe heaters. The main heater in the probe keeps the thermoplastic melt flowing through the whole length of the drop and a very small tip heater raises the tip temperature very rapidly. The tip heater cycles during every injection molding cycle, permitting the melt to flow (high tip temperature) during injection and stopping the melt flow or freezing (low tip temperature) during the remainder of the cycle. This thermo-valve gate is a patent of the Spear Systems, Inc. Figure 10-20A shows the spear tip "OFF" or gate closed and Figure 10-20B shows the spear tip "ON" or gate open.

10.9.9 Types of Hot Runner Molding Systems

There are three different methods used to control the molten thermoplastic temperature inside the hot runner molding systems:

- Internally heated hot runner molding systems use torpedo cartridge heaters with thermocouples that are mounted inside the runners, leaving a clearance area between the outside diameter of the torpedo and the inside diameter of the runner to convey the melt
- Externally heated hot runner molding systems use different types of heat sources to control the manifold melt temperature surrounding the runner channels

- Insulated hot runner molding systems take advantage of the heat insulation properties of the thermoplastic melt by allowing relatively thick layers of thermally insulating thermoplastic to freeze around a large outside diameter of an insulated runner, allowing the hot thermoplastic melt to flow through the center of the insulated runner.

The balancing of hot runners is an important mold design requirement, as well as for cold runners. A balanced runner distributes the melt to the cavities at the same time, packing all the cavities with the same injection pressure, to obtain uniform dimensional control in all the molded parts.

Where there is a substantial difference in temperature between the hot manifold and the cold mold plates, provisions must be made to adjust for the differential coefficient of linear thermal expansion between these two components. Supports, spacers, and some bushings may be surface mounted on plates so that they can slide during heating and cooling, although this can lead to problems in sealing the manifold and drop melt pressure in the channels. The mold design can accommodate movement during expansion or contraction, but the length must be adequate to allow this motion. Drops containing torpedoes can accommodate expansion, if the torpedo is heated first and cooled last so that the torpedo is not bound in thermoplastic, and can float during expansion and contraction. This, however, does provide an opportunity for error by the operator and possible damage to the mold.

Insulated hot runner molding systems will not tolerate significant molding cycle interruptions without freezing of the melt flow path or insulated runner, which then requires removal of the insulated runner and restarting the mold. A delay time of one to two minutes can cause freezing of the melt path depending on resin melt temperature, mold temperature, melt throughput, and other factors. Although runner removal and restarting are relatively simple, it takes between five to ten minutes. Recurring interruptions will affect productivity and become a problem.

The removal of the insulated runner can be a process advantage; it eliminates the need for the difficult purging process required during start-up or color change; in addition, insulated runners are always clean and empty for start-up. Small particles of dirt or trapped metal that plug small gates are also easily removed by changing the insulated runner. In a hot runner mold, with externally or internally heated manifolds, the restarting and cleaning process requires a lot of maintenance work and operational down time. It requires removal and disassembly of the hot runner molds, requiring heaters, seal replacements, and cleaning using fluidized bed equipment, or sending the unit to the vendor for service.

10.9.9.1 Internally Heated Hot Runner Systems

The thermoplastic melt is injected through a sprue bushing heated with an internal torpedo. The system consists of bores drilled into the internally heated manifold. A torpedo is inserted in each runner and the drops are centered with end caps.

The melt flowing in the runners, around the torpedo's surface, is kept molten by a thermocouple and cartridge heater inside the torpedo. Heating the material "from the inside out" is very efficient, since it allows heater loads to be much less than externally heated hot runner manifolds. The outer layer of the melt stream solidifies to form an insulating layer of solid thermoplastic that further

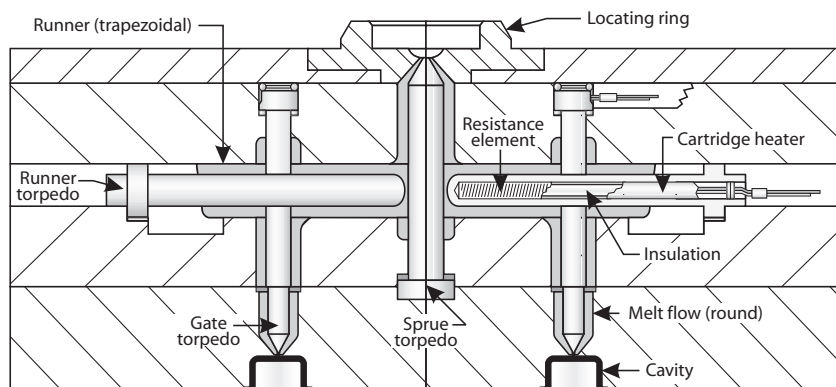


Figure 10-21 “H” manifold, internally heated hot runner

reduces energy requirements and permits the manifold surface temperatures to be maintained between 100 and 110 °F lower than the torpedo.

Often, internally heated hot runner designs have a single main runner that intersects a gate torpedo to direct the melt flow to the gates. The main and secondary runners use a common “H” manifold layout pattern providing a balanced melt flow between the main and the intersecting secondary runners. A variety of internally heated manifold runner layouts are possible for carrying molten thermoplastic to the cavities.

Either round or trapezoidal runner cross sections are used to define the manifold runner parting line. The use of integral cartridge heaters in all torpedoes also improves heat transfer throughout the system.

The hot runner internally heated system temperature control inside the melt channels (sprue, runners, and drops) is inherently nonuniform, because the torpedo must be adequately supported in the colder mold plates at one or both ends. The torpedo cartridge heaters tend to run hotter as the distance from the supported end increases. Streamlining the internal runner layout is not possible, when the large main runner diameters and internal torpedoes intercept the sprue, sub runners and drops that are located perpendicularly, the interception from the path is offset, causing hold-up spots in the conveying channels and the necessity for the thermoplastic melt to flow around these obstacles.

Cleaning the internally heated hot runner system requires disassembly of the mold, removing the runner with the torpedoes, and reassembling the mold using new torpedoes and pressure seals, because they are difficult to remove from the runner without damage. Figure 10-21 shows a cross section view of an internally heated hot runner system.

High temperature engineering thermoplastic polymers and heat sensitive resins are not recommended for use with the internally heated hot runner systems. However, for commodity thermoplastic resins processed at low temperatures that are not heat sensitive (PE and PP), this system is ideal for high production applications. The cost of the internally heated hot runner systems is lower than the externally heated hot runner systems.

10.9.9.2 Externally Heated Hot Runner Systems

Each externally heated hot runner manifold design has a balanced runner configuration, free of hold-up spots and smooth surface finish to maximize part

production. The hot sprue bushing satisfies various melt flow requirements, having a replaceable seal ring that provides a leak proof seal between the hot sprue bushing and the manifold.

The externally heated hot runner manifolds use several temperature control systems, such as cartridge heaters, cast-in heaters, heat pipes, heater bands, coil heaters, and torpedo heaters to ensure full heat transfer and faster start-up. They are designed to support high injection pressures (20,000 psi) and temperatures, with adjustments for the differential coefficient of linear thermal expansion between the hot manifold and the cold cavity plate that houses the drops. Supports are used to avoid deflection with spacers for thermal insulation. A structural joint seal design is used for assembling the hot sprue bushing, main and sub manifolds, and drops.

When selecting a nozzle, it is important to take into consideration the part design complexity, the wall thickness, flow length, and the shear rate of the resin. The nozzle should satisfy a range of shot weights based on the internal channel length and diameter, and the viscosity of the material to be molded.

Several types of gates (drops) are used for the externally heated hot runner systems, depending on the gate vestige, color changes, resin, shot size, and overall performance required by the application. The drops are mounted in the cavity plate clearance pocket, using an air gap for thermal insulation of the drops and the thermal expansion of the hot manifold.

The externally heated hot runner systems require that the heated components, manifolds, and drops be insulated from the cooler parts of the mold. Some type of separation between the manifold and the mold plate pocket is needed to prevent complete freezing or to maintain adequate flow through the melt path diameter.

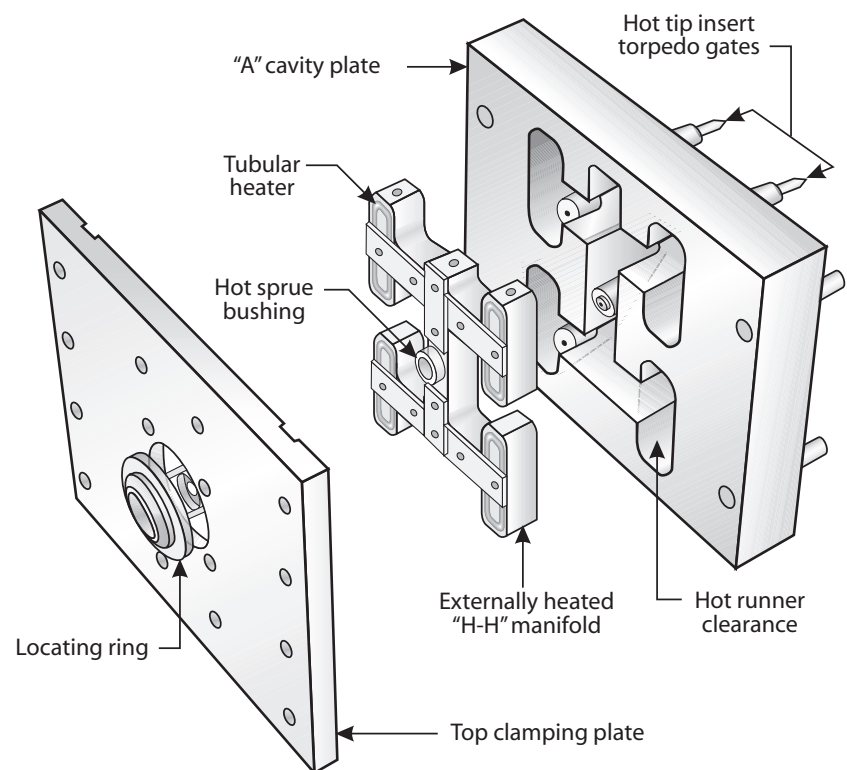


Figure 10-22 Externally heated, "H-H" hot manifold with mold plates

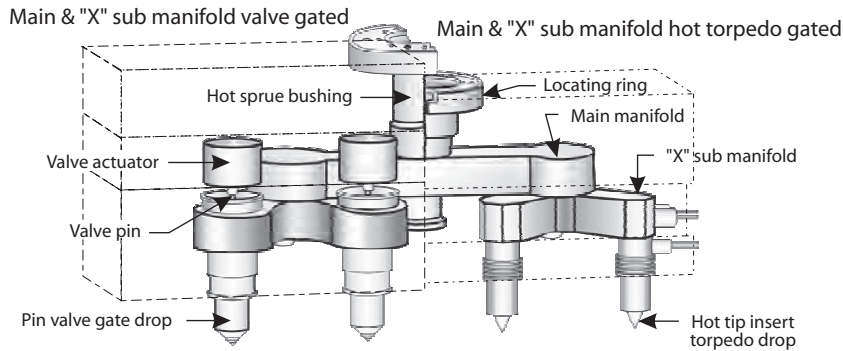


Figure 10-23 Two integrated cast-in externally heated hot manifolds

The externally heated hot runner manifolds, sprue bushing, and drops must be designed carefully to provide a streamlined flow path. All these components should be manufactured from corrosion resistant, high heat conductive, high temperature resistant, and high strength metal alloys.

The externally heated hot runner systems are more expensive, require more training and maintenance, are difficult to clean, and need more time to heat the polymer inside the flow channels for restarting the molding process after a long interruption.

Figure 10-22 shows a typical externally heated hot runner tubular system using an “H-H” manifold with hot tip insert torpedo gates.

Figure 10-23 shows two integrated cast-in, externally heated hot runner manifolds (main and “X” sub manifold). The left illustration has reciprocated pin valve drops and the right illustration has hot tip insert torpedo drops.

10.9.9.3 Insulated Hot Runner Molding Systems

The insulated hot runner molding system was developed at Du Pont’s Plastics Technical Center in Geneva, Switzerland more than 25 years ago. The insulated hot runner mold is not really a commercial system. Du Pont has arrangements with special suppliers for the three hot tip torpedo designs and the additional system components. However, Du Pont technical assistance is required for the selection of the insulated hot runner molding system, mold design recommendations, detailed drawings of the specific system components and specifications. Information about the closest supplier of system components, mold start-up and troubleshooting assistance, insulated hot runner molding system technical training and other services are offered only to Du Pont customers.

The insulated hot runner mold system is a particularly good choice when there will be frequent color changes or for other reasons such as high quality requirements that make thorough purging of a hot manifold very troublesome.

Insulated hot runner molds are more difficult to start and operate than three-plate molds, but are much easier than other hot runner molds.

An insulated hot runner mold is a cross between a hot runner mold and a three-plate mold. The insulated runnerless mold system is relatively inexpensive and ideal for heat sensitive thermoplastic polymers. Most of the higher melting point engineering polymers are a little more difficult to process because of the greater differential temperatures between melt and mold.

The thermoplastic melt remains fluid enough in the center of the insulated runner to permit repeated injection, even if the runner inserts and the mold plates are colder. The solidified melt on the surface of the insulated runner provides exceptional thermal insulation for the inner fluid core.

The insulated runner is nearly round and has either approx. 0.90 in diameter for the standard systems or 0.50 in diameter for the mini-insulated hot runner molds. The runner half in each insulated runner insert mounted in each mold plate is tapered and cut a little less deep than the radius to ease runner ejection.

The insulated hot runner mold systems benefit from thermal insulation properties of the thermoplastic polymers. All runner systems require some thermal isolation to aid in start-up, to prolong freeze-off during brief cycle interruptions, and especially to keep flow paths sufficiently large to avoid an excessive injection pressure drop through the insulated runner. Thermal isolation is achieved by using two separate insert halves to form the large insulated runner and by reducing the metal to metal contact with adjacent cavity plates. Reduction of contact is achieved by “waffling” the contacting insert surfaces and mold plate cavities or by inserting a thin (0.005 in thick) sheet of Nomex® aramid paper between these two base surfaces. In addition, coolant lines around the sprue bushing and cartridge heaters without thermocouples are temperature controlled by a common unit. The cartridge heaters should be mounted at each side, in the same direction and close to the half runner. Both runner insert halves should have the same amount of heaters for a thermal balance of the insulated runners. The runner inserts and sprue bushing operating temperatures should be less than the melt temperature and higher than the mold cavity temperature.

Figure 10-24 shows a cross section view of the Du Pont reciprocating insulated hot runner molding system. The other Du Pont systems operate the same way, except that the hot tip torpedoes have different configurations and are fixed, requiring the torpedo gate height to be adjusted by moving the spacers on top of the hot tip torpedo, to satisfy the application requirements.

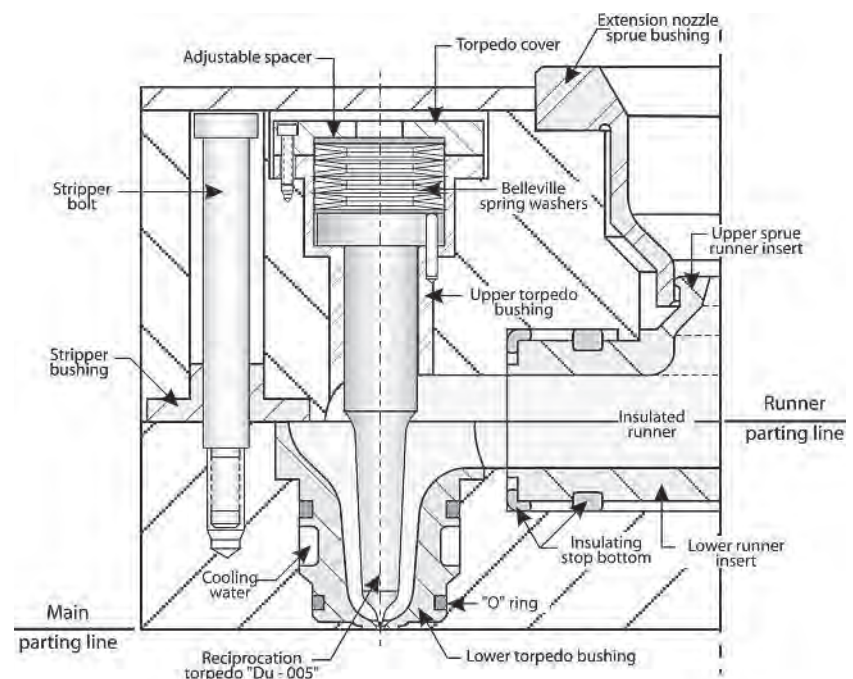


Figure 10-24 Du Pont insulated hot runner reciprocating system

The temperature of the gate area is controlled by circulating mold coolant around the lower torpedo insert bushing using “O” rings to avoid freezing the hot gates. This bushing is also useful for replacing worn or damaged gates without major reconstruction of cavities. If unusually low mold coolant temperatures are to be used, the requirements for maintaining an adequate flow path through the insulated runner may become more stringent. Higher throughput and/or better heating and isolation of the insulated runner may be necessary.

To maintain an adequate open flow path through the insulated runner, a minimum of 8 g of the melt must flow through each insulated runner and drops every minute. This can be accomplished by a combination of short molding cycles and small parts or longer cycles and larger parts.

Many insulated runnerless mold systems are not directly gated into the part but instead into small sub runners. This is done for a variety of reasons:

- To cluster small parts and increase melt throughput
- To hold large numbers of cavities with fewer drops
- To use multiple gates in a single part to control roundness or to achieve adequate flow in long, thin, or complicated parts
- To gate through sub runners with tunnel gates to eliminate gate vestiges or stringing.

Long insulated runners are not recommended because the melt flow decreases with distance and causes bending of hot tip torpedoes during ejection of the insulated runner because of shrinkage. The maximum flow distance from the machine nozzle to the farthest drop should not exceed 15.00 in.

During molding cycle interruptions or at the end of a shift, the entire cross section of the insulated runner will solidify in time and provisions must be made to ensure easy removal of the solidified insulated runner system. This is accomplished most simply by closing the mold, removing the four latches from the insulated runner parting line of the mold that normally remains closed and replacing the four latches to lock the main cavity parting line. Then, the mold is opened at the insulated runner parting line, the runner is pulled off the hot tip torpedoes by the stripping washer force activated by stripper bolts under the extended runner lips, dropping the insulated runner automatically. Without this automatic feature, damage to hot tip torpedoes, parting line, etc. during manual insulated runner removal is almost certain.

Four parting line latches (two at each side) are usually sufficient, because these latches hold either parting line closed, during injection of the molded part or the insulated runner. The molding machine clamping force is needed to resist injection force. Because the two parting lines of an insulated hot runner mold are stacked, the clamping force required is determined by the projected area of the cavities or of the insulated runner, whichever is greater, not by the sum. One must never attempt to purge through an insulated hot runner mold with these external latches, because the mold will be damaged and personnel can be injured. There is no need for purging through an insulated hot runner mold; runner removal is easier and more effective.

Figure 10-25 shows the Du Pont insulated hot runner operational sequence; the top two illustrations show the normal injection molding process, where the mold cavity is injected and the molded part is ejected. The bottom two illustrations show the automatic ejection of the insulated runner.

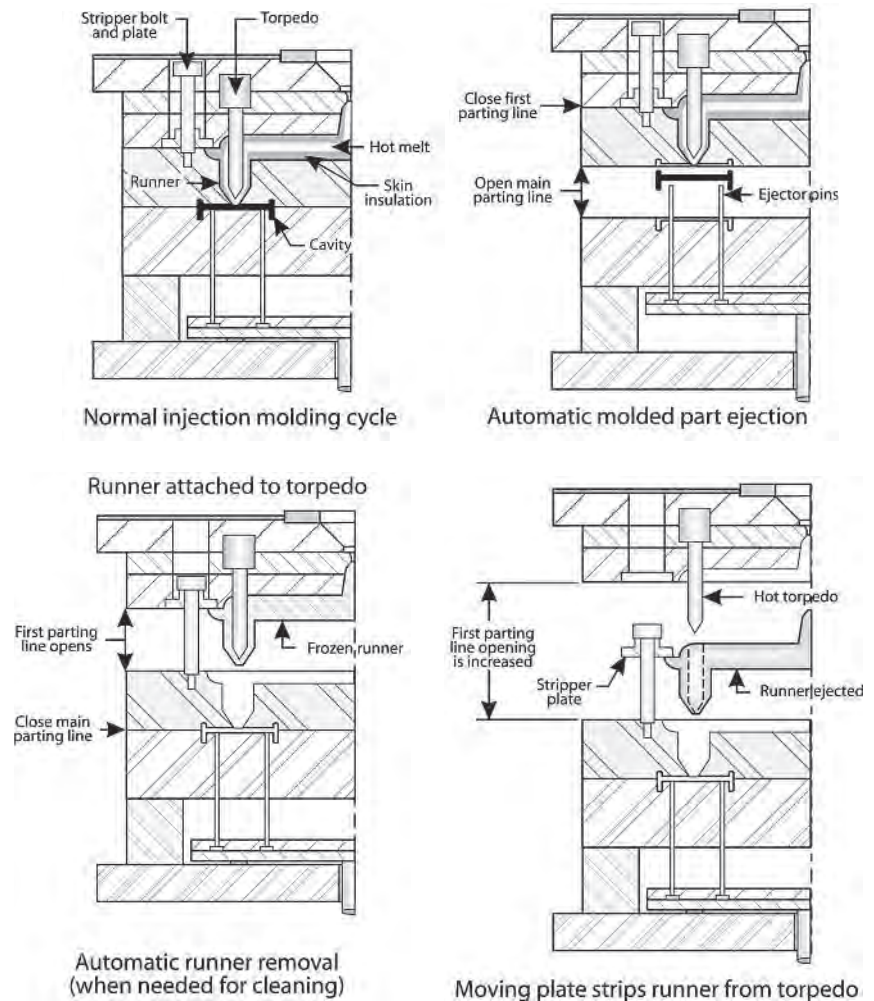


Figure 10-25 Du Pont insulated runner mold operational sequence

The insulated hot runner molding systems employ three hot tip torpedo design shapes. The three hot tip torpedoes are built to meet Du Pont specifications. They contain a short special design cartridge heater at the bottom with a ceramic insulator above this to protect the wires and a hold-down spring above the insulator. The thermocouple is in the cartridge heater and both are pressed inside the torpedo tip insert to control the heat toward the tip. The torpedo tips inserts/heaters are mounted (hot interference fit) with the torpedo shells, the insert tips are diamond polished, wear resistant and thermally conductive.

Standard Fixed Hot Tip Torpedo "DU-085"

The standard fixed hot tip torpedo is for general purpose applications and is most commonly used for engineering polymers. The hot tip torpedo is fixed in position during the molding process. The control of gate vestiges and freezing is dependent on the tip temperature and the tip location in the gate area. The exact tip position can be adjusted for different resins, molding cycles, and to correct gate vestiges or freezing problems. This is done by interchanging the four spacers of slightly different wall thicknesses; one spacer is above the hot tip torpedo top support flange and three other spacers below. This adjustment will be made during original development and start-up of a new mold and will

rarely have to be changed afterwards. Figure 10-26 shows the standard fixed torpedo “DU-085”.

Reciprocating Standard Hot Tip Torpedo “DU-005”

The difference between the fixed and the reciprocating hot tip torpedoes is their geometry. The reciprocating hot tip torpedo works as a mechanical valve to control the melt flow at the gate. The reciprocating hot tip torpedo is forced to retract from the gate seat by the injection force of the melt on the tip of the torpedo (gate open), just before the melt is injected. When injection is complete, Belleville spring washers above the support flange, which were compressed during injection, push the hot tip torpedo down against the gate seat, closing the gate.

The reciprocating hot tip torpedo stroke, although it is only 0.06 in, may cause the cartridge heater or even more delicate thermocouple wire breakage.

Figure 10-27 shows the standard reciprocating hot tip torpedo “DU-005”.

Fixed Mini Hot Tip Torpedo “DU-200”

The fixed mini insulated hot runner molding system satisfies the need for a runnerless mold to fit in small injection molding machines used in the production of very small parts and to permit closer mold layouts of these small cavities. The small molding machines are often incapable of holding larger molds with the size and stack height required by the standard large hot tip torpedo systems. Since the fixed mini insulated hot runner molding system depends on the same limitations imposed by the insulated runner concept, the insulated runner diameter for this system is smaller. The test results of the fixed mini insulated hot runner molding system have shown that much lower melt throughput than 8.00 g/min is viable.

The fixed mini hot tip torpedo has a larger diameter upper portion and a tapered lower portion design to improve the durability of the unit. The fixed mini hot tip torpedo operates with a 120 V cartridge heater and a thermocouple for closed loop control.

Figure 10-28 shows the fixed mini hot tip torpedo “DU-200”.

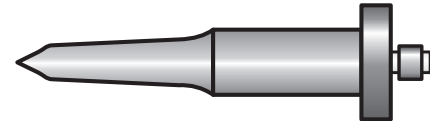


Figure 10-26 Standard fixed hot tip torpedo “DU-085”

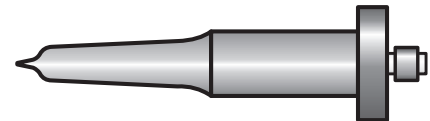


Figure 10-27 Standard reciprocating hot tip torpedo “DU-005”

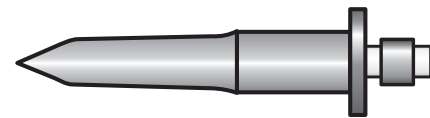


Figure 10-28 Fixed mini hot tip torpedo “DU-200”

10.9.10 Thermoplastic Stack Injection Molds

Thermoplastic stack injection molds are similar to standard molds, but they use cavities on more than one level or on both sides of the mold parting lines. These types of molds require higher clamp forces, usually 15 to 20% more, but they can increase molding output by 80 to 100%.

Applications of stack molding include shallow containers and lids, threaded and tamper proof bottle caps, disposable tableware, and deep draw containers.

The stack mold was originally developed in the plastic compression molding industry. Admiral Distributors, Inc. of Chicago, Illinois in 1955 employed two identical sets of compression molds, one above the other, used for the production of phenolic radio phonograph lids. This allowed two pieces to be molded from a conventional one-cavity press, with the same molding cycle and pressure as required for molding single parts.

Thermoplastic stack injection molds have been used since the mid 1960s. However, it was not until the late 1970s and the early 1980s that the stack molds

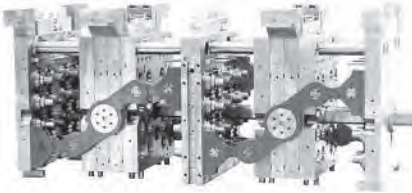


Figure 10-29 Stack mold, four level or tandem mold (Courtesy: Husky Corp.)

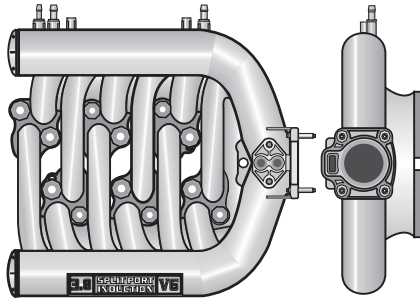


Figure 10-30 Lost core thermoplastic V-6 intake manifold (Courtesy: Ford)

gained in popularity. Stork Plastics Machinery of Hengelo, Holland and Husky Corp. of Canada developed a unique thermoplastic stack injection molding machine and companion stack molds, used to mold double layers of the parts automatically, using the clamp force available.

The new thermoplastic stack injection molding process incorporates the specialized tooling required into the machine. This includes hot runner systems, valve gating, and rack and pinion gears for mold segment separation during clamp opening and closing. These thermoplastic stack injection molding process innovations are shown in Figure 10-29. The stack mold and this special machine is a turn-key molding system built by Husky Corp. in Canada.

10.9.11 Lost Core Thermoplastic Injection Molds

These molds are used in the manufacturing of large, multifunctional, and hollow thermoplastic injection molded components as shown in Figure 10-30.

This new technology represents a breakthrough for design engineers who have long searched for a method to produce such parts, meeting quality, dimensional, and repeatability demands in a high volume production environment.

“Lost Core Molding System”

A metallic core is the latest advancement in the metal alloy casting process. This technology is a proven foundry process and it has found acceptance in the plastic industry as a method for injection molding hollow thermoplastic components. The basic principal is to use a metal alloy to form a core insert that provides the internal shape (see Figure 10-31, left) while the mold surface cavity forms the external geometry of the thermoplastic part. Once the encapsulated component is formed (see Figure 10-31, middle), the inner core insert is melted-out, leaving the finished hollow component (see Figure 10-31, right).

The lost core molding system combines the use of low melting point metal alloys to produce the metallic alloy cores, with the thermoplastic injection molding process using the metal cores as an insert. Once the thermoplastic component has been encapsulated around the metal core, the metal is melted-out via the fully integrated oil bath with a near total retrieval of the low melting point alloy for future reuse.



Figure 10-31 Lost core insert and thermoplastic intake manifold (Courtesy: Electrovert)

The metallic core process uses several metal alloys suited for the type of thermoplastic resin selected and application requirements. For engineering thermoplastic resins, a 42% tin and 52% bismuth alloy is used, having a melting temperature of 281 °F. The metallic cores can withstand the injection speeds, pressures, and temperatures of the thermoplastic injection molding process.

The encapsulated assembly (thermoplastic component and metallic core) is placed in an inclined position inside a hot oil bath for two to three hours (melt-out stage). The core is efficiently extracted from inside the part without harming the polymer's physical properties, or causing warpage of the thermoplastic part.

Many components have been commercialized using the lost core molding process. These include intake manifolds, water pump housings, power steering fluid reservoirs, thermostat housings, fuel rail lines, and other automotive components. All these applications are physically viable and economical alternatives to vibration welded assemblies of two thermoplastic injection molded shells or metal systems (sand casting molds and machining).

Lost Core Molding Design

The lost core manufacturing procedure consists of three main processes:

- Metallic alloy core forming
- Injection molding to encapsulate the metallic core insert
- Metallic alloy core melt-out

The design of the lost core injection molding system consists of metal alloy core forming equipment and a vertical clamp type of injection molding machine as shown in Figure 10-32 (higher equipment cost) or a standard horizontal (lower equipment cost) type injection molding machine. The metal core is inserted horizontally from the back side using a loading robot. The encapsulated molded part with the metal core is horizontally removed from the working side of the molding machine using a second robot. Both parts are hung by the robot on a monorail to be conveyed to the hot oil tank where the metallic core is melted-out.

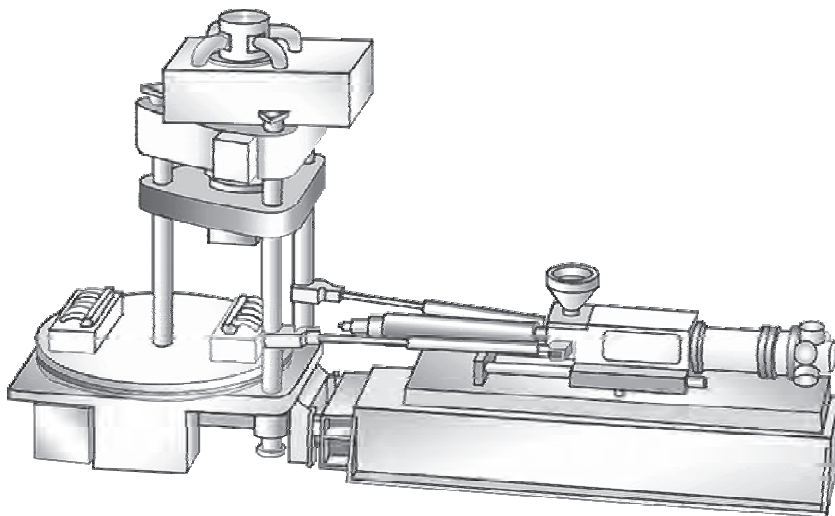


Figure 10-32 Lost core intake manifold injection molding machine

Two weight scales on both sides of the injection molding machine (for the metal core and the encapsulated molded part) are used to detect and discharge faulty products automatically.

Molding Machine Clamping System

Besides the high strength, high cycle double-toggle mold clamping mechanisms, the metal core insertion and part removal operation is incorporated during mold opening and closing to shorten the molding cycle. The mold clamping force overcomes the polymer melt injection pressure; however, the mold size is much bigger than the mold cavity projected area, because of the complex design. Sometimes the mold weight may reach as much as 15,000 pounds.

Rigid supports for the machine's moving platen are needed to avoid damaging the mold parting line surfaces, because the clamping force plus the weight of the mold could over stress, deforming the machine moving platen. Therefore, the toggle clamp type system is more advantageous than a direct hydraulic or a hydro-mechanical clamp system.

An injection molding machine with a large tie-bar clearance is needed. A tie bar puller is used as an effective countermeasure, allowing the use of smaller size molding machines. The most important function of the clamping system is to deliver stable and precise mold opening/stopping accuracy even at maximum speed, without affecting the hydraulic oil temperature or overloading the clamping system.

Injection Unit

For intake manifold applications, using a large shot size (product weight about 4.00 to 6.00 lbs), the injection capacity should be 1.5 to 2.0 times larger than the product. A precise injection process control with a stable high speed filling and instantaneous switch-over from fill to packing conditions is important.

Barrel, plastifying screw, free flow check valve, and nozzle of wear resistant steels are recommended when processing nylon 6/6 resin reinforced with 30% to 35% fiber glass. In addition, a high compression injection screw with a compression ratio between 3.50 and 4.00 should be used. A screw length-to-outside diameter ratio between 20 and 24 is needed.

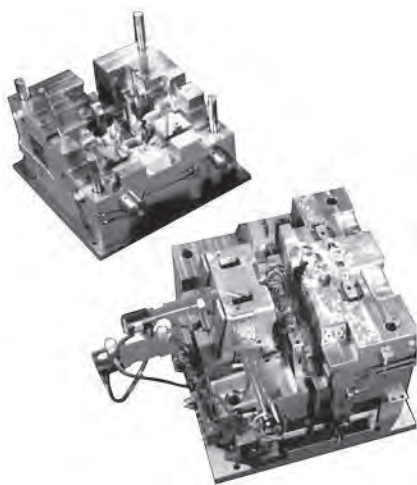


Figure 10-33 Lost core thermoplastic intake manifold mold (Courtesy: Ford)

Lost Core Thermoplastic Injection Molds

A typical lost core thermoplastic injection mold used for the production of nylon intake manifolds (automotive four cylinder engines), is shown in Figure 10-33.

These types of molds are very complicated to design, build, and operate because of the product design complexity and the critical and important functions required in the lost core thermoplastic injection molding process.

The following mold design considerations are required in this process:

- The low melting temperature metallic alloy cores have less strength than common steel materials; they are easily moved or bent by the hot melt of the polymer. The metal core is fixed inside the mold with a locking pin, but if the pin reinforcement force is less than the thrust force generated by the melt injection pressure, the metal core will shift and the over molded part wall thickness will be nonuniform. Both ends of the metal core need to be

supported, but the thrust force generated by the injection pressure should be a compressive stress.

- Nonuniform mold cavity filling during injection causes metal core deformation and nonuniform wall thickness of the parts due to the injection pressure. Type, dimensions, number, and location of the gates, melt and mold temperatures, injection speed and pressure distributions should be estimated by using a “Melt Flow Computer Analysis Program”, making corrections if needed, such as adding gates on the opposite side or increasing the number of gates to balance the melt injection forces that can deform the metal core insert.
- The hydraulic actuators for the mold slides should be bigger with longer slide strokes, because this causes the core operating time to be increased, the simultaneous movement of the mold closing/opening operation is effective in shortening the cycle time.
- Provisions should be made in the mold, so that the space needed for the pick-up can be secured without interfering with the robots and the mold during core insertion and part removal.
- After insertion, the metal core is fixed by a locking pin through the mold hydraulic slides, so when the stopping accuracy varies, the locking pin cannot be inserted.
- The hydraulic ejector does not need a long stroke to push out a normally inserted metal core, but if the ejector pin does not properly return, the subsequent insertion cannot be done.

The lost core thermoplastic molding process provides several advantages, among them the flexibility in designing and manufacturing hollow thermoplastic components of varied internal geometries in one piece.

The lost core molding process provides several design advantages, for example, the intake manifold increases engine horsepower by increasing air inlet speed through the channels (smooth surface finish), lower air inlet temperature (thermoplastic insulation), and multi-functional design

The lost core molding process is applicable for several engineering thermoplastic resins. It is able to produce a textured surface in the internal walls of the thermoplastic components via a metallic core insert. It also produces good dimensional control of the thermoplastic components and metallic core inserts. It has proven to be economically viable in the production of one-piece thermoplastic components. This is especially true in medium to large volume production runs. The lost core molding process allows for the manufacturing of components that were not previously considered for injection molding as one piece thermoplastic parts, especially where the “value-added factor” of the component is enhanced. The process produces uniformly structured metallic cores with good surface finish, free from voids. Post machining of the core, before the injection molding process, is not normally required.

The metal alloy used to manufacture the metallic cores is recycled nearly 100% using an integrated metal recovery system. The metal alloy cores are certified as having no health hazards. Since the alloy is reusable, there are no environmental disposal problems.

10.10 Number of Mold Cavities

The mold cost increases with the number of cavities, while the molding process costs or machine use time decreases. The optimum number of cavities is the interception between the mold cost and the molding process as a function of the number of cavities. When the number of cavities is higher than the optimum number, the mold cost and the molding process costs affect the efficiency of the injection molding process.

The cost of manufacturing of an injection molded product consists of three major elements: the cost of the thermoplastic materials, the cost of the molding process, and the cost of the mold. The thermoplastic material costs for a molded product are generally independent of processing variables. The remaining two costs, molding process and mold costs, are directly affected by many parameters, including the number of cavities per mold. Since the number of cavities used affects the cost of a molded product, considering the economics resulting from the number of cavities chosen is important. Too often, the cost of manufacturing is an uneconomical incentive when the mold cost is too high, resulting from too many cavities, or high molding process costs, resulting from too few cavities.

10.10.1 Cavity Number Limitations

Often, using the economic optimum number of cavities may be impossible to implement. In these cases, the maximum number of cavities should be considered. Judgment resulting from mold fabrication experience can never be replaced by theoretical calculations. Incorporating the optimum number of cavities into a mold may be impractical due to such things as:

- Complexity of runner, gating, or venting systems
- Overall dimensions of mold base or tie-bar spacing
- Projected area of cavities for clamping capacities
- Core pulling or cam action mechanisms
- Tolerances required on finished parts

10.10.2 Number of Mold Cavities Equation

The number of cavities equation is designed to balance the mold investment against the molded product cost. The number of suggested cavities is based on an empirical rule. The yearly molding production rate of parts is divided into an ordering frequency of about every 50 days. The 50 days molding production requirements should be fabricated in 200 hours using a three shift molding operation.

Example 10-1

For example, if the yearly molding production requirement is 500,000 molded parts, the 50 days frequency requirement will be 100,000 parts. The estimated cycle is 60 shots per hour or 60 molded parts per cavity per hour.

The number of cavities is:

$$\begin{aligned}
 \text{Number of cavities} &= \frac{\text{Molded parts needed to produce in 50 days}}{(200 \text{ hours}) \times (\text{molded parts/cavity/hour})} \\
 &= \frac{100,000 \text{ molded parts needed in 50 days}}{200 \text{ hours} \times 60 \text{ parts per cavity per hour}} \\
 &= 8.33 \text{ or eight (8) cavities}
 \end{aligned}$$

This equation has been checked and has proven to be an accurate method to calculate the number of mold cavities. The important criteria for specifying a suitable tool steel, surface hardness, and finishing for the cavities is the severity of usage and the duration of product activity. Severe usage is defined as a mold running for about half of the available time, whereas running for 10% of the available time is viewed as mild usage.

10.11 Mold Parting Line

The parting lines of a mold are those portions of both mold plates, next to the cavities, which butt together to form a seal and prevent the loss of thermoplastic material from the mold cavities.

We can classify the parting line as either flat or non-flat. The non-flat types include stepped, profiled, and angled parting lines.

The flat parting line is the simplest to manufacture and maintain. It can be surface ground and is easily bedded down in the mold cavity surface.

To bed down a pair of mold plates is the process of marrying the two mold surfaces together. This is accomplished by blue marking one surface, momentarily bringing the two plates together and subsequently removing any high spots that will be apparent on the other surface. The plates are said to be bedded down when an even film of blue marking is transferred from one plate to the other.

10.11.1 Flat Mold Parting Line

The parting line depends entirely on the shape of the component. The cavity for the rectangular molded part in Figure 10-34 shows that the mold cavity can be die-sunk into one mold plate. The position of the parting surface will therefore be at the top of the molded part, and the parting line is a flat surface. For appearance, this is the ideal arrangement as the parting line is not noticeable, unless flashing molding problems develop in the injection molding process.

The flat parting line must be selected so that the molded part can be removed from the mold. Figure 10-35 shows a flat rectangular cavity that incorporates a double beveled edge. The parting line for this component cannot be on its top surface, because this will create an undercut in the mold. The only suitable choice for the parting line is at the center of the double bevel. This parting line selection allows for half of the required form to be die-sunk into each of the two mold halves.

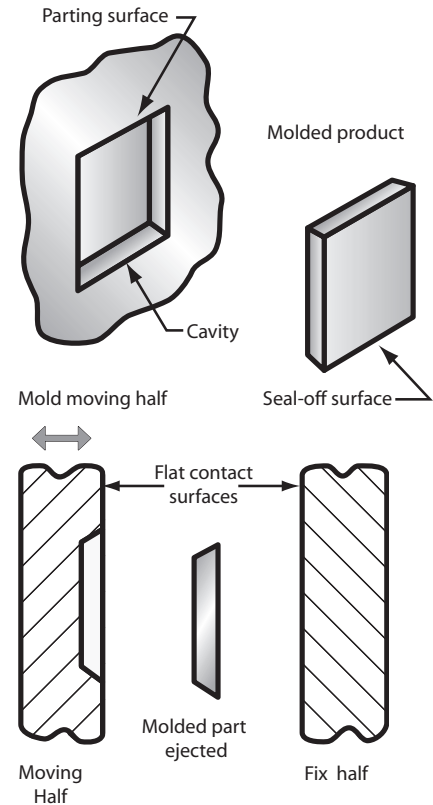


Figure 10-34 Flat mold parting line and cavity location

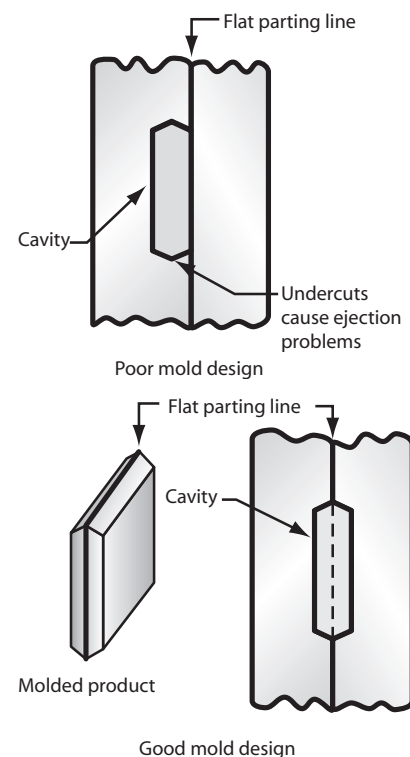


Figure 10-35 Flat parting line selection affects mold performance

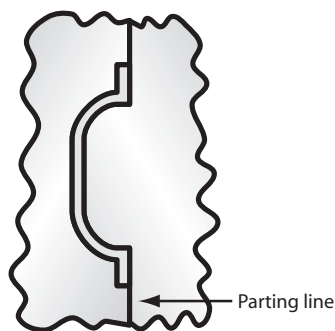
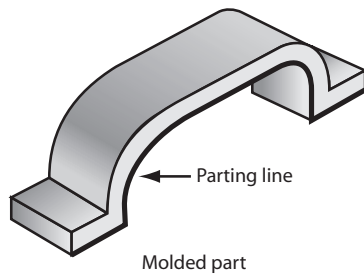


Figure 10-37 Profiled mold parting line

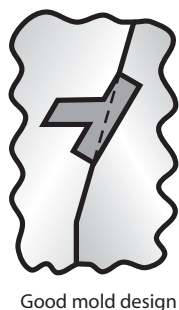
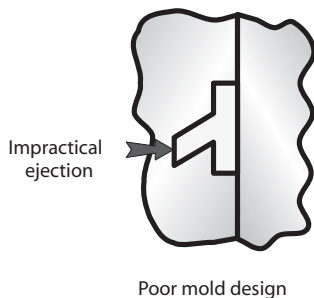
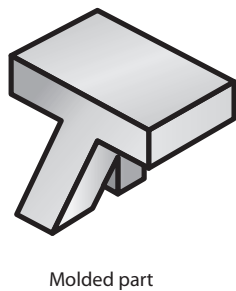


Figure 10-38 Angled mold parting line

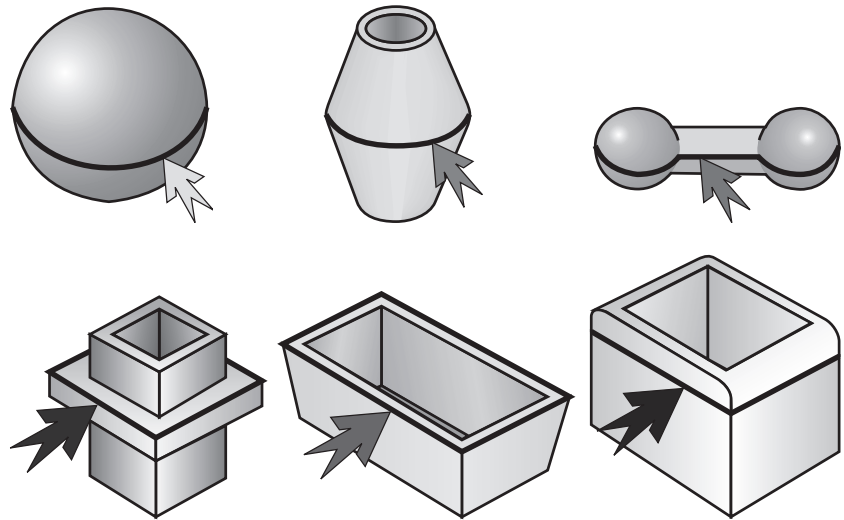


Figure 10-36 Typical flat parting line for different molded shapes

A number of molded shapes that permit a flat parting surface to be adopted are shown in Figure 10-36. The arrows show the parting line on the molded parts.

10.11.2 Non-Flat Mold Parting Line

Many molded parts require a parting line that lies on an angular or curved surface. In these cases, the parting line must be stepped, profiled, or angled.

Profiled Mold Parting Line

A profiled parting line illustration is shown in Figure 10-37. The molded part is shown in the top illustration. It should be noted that, while in cross section the molding form is constant, the general form (side view) incorporates curves. As the edge of the component is square with the face (apart from cavity draft), the entire cavity geometry can be sunk into one mold cavity plate. Thus, the general form of the parting surface will follow the inside surface of the mold cavity.

Angled Mold Parting Line

The designer is frequently confronted with a component that, while regular in form, cannot be ejected from the mold cavity if a flat parting line is adopted.

Figure 10-38 shows the molded part on the left, the ejection problem in the middle, and the recommended angled parting line in the bottom illustration.

Complex Mold Parting Line

The complex parting lines consider other component's edge forms (that is, either square, double-beveled, or incorporating a radius) where the edge form is not constant. This often leads to quite complex parting line surfaces. Figure 10-39 shows a simple product design geometry using a complex parting line.

To determine the parting line, a number of cross sections are drawn through the part to be molded and the maximum dimension of each is defined when viewed in the draw direction. The parting line will pass around all these points

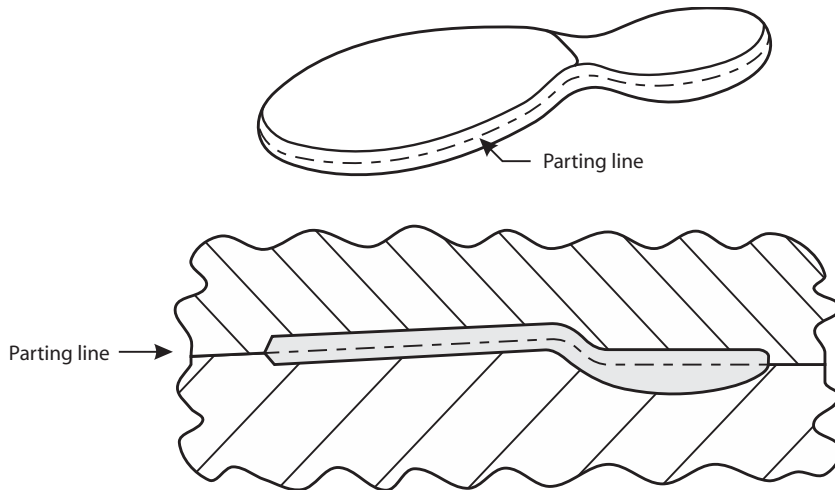


Figure 10-39 Complex mold parting line

of maximum dimension. Once the parting line has been determined, the mold's parting line surface can be drawn. This parting surface does not vary in the transverse direction and, if wanted, it can be machined completely across the mold's surface. This surface is confined to the area next to the cavity to reduce the time spent in bedding-down on the undulating surface to a minimum.

Local Stepped Mold Parting Line

On certain components it may be found that the change from one point of maximum dimension to another is quite abrupt. In these cases, the parting line is stepping from one plane to the other, preferably at an angle. Permitting the mold maker certain latitude in blending the cavity form in the step region to ensure that local undercuts do not occur is usually necessary.

Incorporating a step or profiled parting surface to compensate for one or two small irregularities in an otherwise regular form is frequently necessary. Normally, this is best achieved by localizing the change in parting surface to permit most of the surfaces to be kept flat.

Consider a more complex example, as shown in a box shaped component with a lug on one of the side walls in Figure 10-40. This component can be molded in two ways: either by locally stepping the parting line surface or by designing a mold of the side cavity type. The simpler of the two methods is to step the parting surface locally and should be the method of choice when applicable. The molded part, cavity top view, core bottom view, and the part cross section through the mold are shown in Figure 10-40. The lug is above the general parting surface, which requires a projection from the core side to raise the level locally to the core parting surface. A complementary recess is die-sunk into the cavity plate to accommodate this projection. The projection is bedded into the recess on a slight taper. An angle (ϕ) of between 2° and 5° per side is suitable. While straight sides are simpler to produce and to bed down, flashing problems can develop quickly because of wear occurring between the two sliding members.

The recess in the cavity plate is locally deepened to form the lower part of the cavity for the lug. The top part is formed by the top surface of the core parting line.

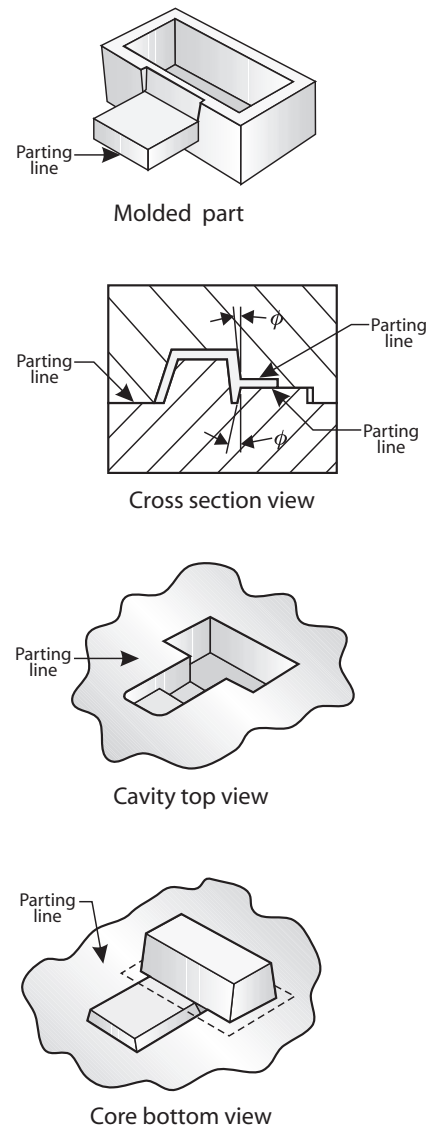
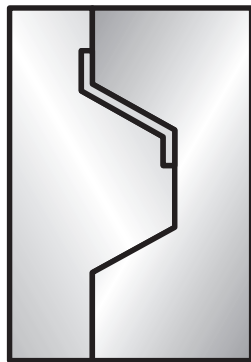
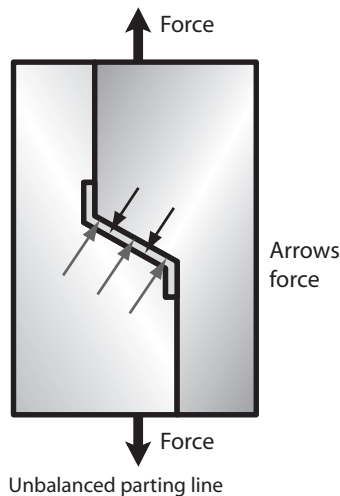


Figure 10-40 Local stepped mold parting line

10.11.3 Balancing of Mold Parting Line Surfaces

When the parting surface is not flat, there is the question of unbalanced forces to consider in certain instances. This is best shown in a mold with stepped parting line surfaces as in Figure 10-41. The thermoplastic melt, when under pressure within the cavities, will exert a force that will tend to open the mold in the lateral direction. If this happens, some flashing may occur on the angled face. The movement between the two mold halves will be resisted by the support pillars; but even so, because of the large forces involved, balancing the mold by reversing the step is desirable so that the parting surface continues across the mold as a mirror image of the section that includes the cavity. Specifying an even number of cavities is often convenient (2, 4, 8, 16, 32 and 64) when considering this type of mold, as cavities positioned on opposite sides of the mold's center line serve to balance the mold.

When balancing is not practical because of the size, very sturdy support pillars must be incorporated behind the support plate of the mold.



Balanced parting line

Figure 10-41 Balancing of mold cavity parting line surface forces

10.12 Mold Ejection Systems

The injection molded parts are subjected to size reduction during the cooling cycle, caused by the mold shrinkage characteristics of the thermoplastic material. If parts with accurate dimensions are needed, allowances must be made for this shrinkage when establishing the dimensions for the cavities. The molded parts are ejected from the mold cavities after they have cooled down. Special mold manufacturing procedures are required for the ejection systems automatically to remove the runner system and the molded parts from the mold.

All mold surfaces that come in contact with the thermoplastic melt and the draft walls in the direction of the draw should be carefully polished. Lubrication of the mold surfaces that contact the thermoplastic melt is not recommended. There is a danger that scratches and cracks will develop when the mold is opened or the parts ejected, particularly when molding fiber glass reinforced resins. The mold cavities should have a mirror finish by grinding and polishing the areas in the direction of ejection to eliminate any scratches and indentations. Wherever possible, the surface should be lapped, since even microscopic scratches and indentations are filled with thermoplastic melt under high injection pressure, preventing smooth ejection.

Ejection equipment is provided on the injection machine behind the moving platen for automatic actuation of an ejector system. Because of this, the mold's ejector system will be most effectively operated if it is placed in the moving half of the mold.

Only flat parts with a pronounced draft toward the direction of ejection can be ejected without special ejector pins on the parts themselves. However, this requires the runner system, the gate, the sprue, and the molded part to have sufficient strength to eject the molded parts from the center by means of the sprue puller ejector pin.

10.12.1 Ejector Plate Assembly

The ejector plate assembly is the part of the mold to which the ejector components are attached. The ejector assembly is contained in a pocket, directly behind the support plate. The assembly consists of an ejector plate, a retaining plate, and a knock-out bar. One end of the knock-out bar is threaded and then screwed into the ejector plate. In this design, the knock-out bar functions not only as an actuating member but also for guiding the assembly. The parallel portion of the knock-out bar passes through a bushing fitted in the bottom holding plate of the mold.

10.12.2 Ejector Plate

The purpose of this member is to transmit the ejector force from the actuating system of the machine to the molded part via an ejector component.

The force required to eject the molded parts is high, particularly with those molded parts that are deep and incorporate little draft. Most ejector plates should be thick enough to resist deflecting to any significant extent. Deflection tends to occur at the beginning of the ejector stroke when there is maximum adhesion between the molded part and the core. The deflection of any beam is inversely proportional to the cube of its depth and, therefore, a relatively small increase in plate thickness will decrease deflection of the plate.

If an ejector plate does deflect to any extent, side forces are applied to the ejector components that result in increased wear in the mold plate holes or ejector sleeves and bent ejector pins.

During the injection part of the cycle, with certain pin and sleeve type ejector systems, the melt pressure acts directly on the ejector component. H-13 tool steel (65–74 Rc hardened) is suitable for this application.

10.12.3 Retaining Plate

The retaining plate is securely attached to the ejector plate by screws. Its purpose is to retain the ejector pins. The thickness of the plate is governed by the depth of the heads of the ejector pins it retains.

For small molds the retaining plate is made to the same general dimensions as the ejector plate. For larger molds, however, holding one or most of the ejector pins is convenient from the mold making viewpoint to incorporate local retaining plates.

10.12.4 Ejector Sleeves

Ejector sleeves are used in cases where containers and tubular parts cannot be ejected by means of a stripper plate. To prevent the ejector sleeves from making scratches on the polished mold surfaces, which would interfere with smooth ejection, the external and internal diameters are made 0.004 to 0.007 in smaller or larger than the respective mold surfaces. During the ejection process, they slide by without making contact with the interior and exterior. Since, for the most part, the ejector sleeves are thin-walled tubes, which are difficult to harden without deforming them, it is advisable to obtain ejector sleeves made of high strength tool steel (to 65–74 Rc). They should be hardened and tempered before finishing.

10.12.5 Types of Mold Ejection Systems

When the melt cools off inside the mold cavities, it contracts by an amount depending on the material being processed. For a molded part that has no internal form, for example a rectangular shape, the molded part will shrink away from the cavity external walls, around the internal cores, thereby permitting a simple ejection technique to be adopted (for example, a valve jet of air). However, when the mold cavity has an internal form, the thermoplastic melt, as it cools, will shrink onto the core and a positive type of ejection system is necessary.

The mold designer has several ejection systems from which to choose, but, usually, the choice will be restricted depending upon the shape of the molded part.

The basic mold ejection systems are the following:

- Ejection pin
- Ejection sleeve
- Ejection blade
- Air poppet valve ejection
- Mechanic valve ejection
- Stripper plate or ring ejection

Mold Ejector Pin System

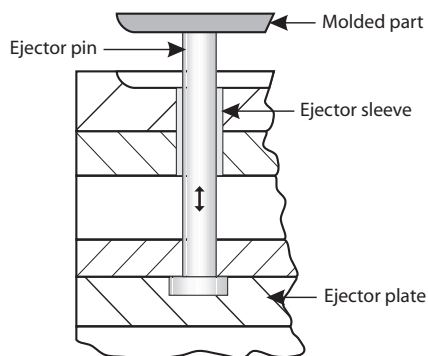


Figure 10-42 Mold ejector pin system

The mold ejector pin system is the most common type of ejection because it is generally the simplest to incorporate in a mold design. With this mold ejection system, the molded product is ejected by the application of a force by a circular steel rod called an ejector pin.

Figure 10-42 shows the principle of operation. The ejector pin is in the moving half of the mold and it is held back by the return pins (not shown). In operation, the ejector plate assembly, to which the ejector pin is attached, is moved forward relative to the mold cavity plate, causing the ejector pin to push out (eject) the molded part from the mold cavity.

The working diameter of the ejector pin must be a good slide fit in its mating hole in the mold plate. If it is not, the thermoplastic melt will creep through the clearance and a mass of material will progressively build up behind the mold plate.

The rear part (head) of the ejector pin is fitted into a suitable hole that is bored and counter bored in the ejector retaining plate. The rear face of the ejector pin head is backed up by the ejector plate. This arrangement must allow the ejector pin to float.

Mold Ejection Blade System

The main purpose of the mold ejection blade system is the ejection of very slender parts, such as ribs and other projections, which cannot satisfactorily be ejected by the standard mold ejector pin system.

A blade is basically a rectangular ejector pin. While the blade ejector can be machined from a solid rod, it is usually fabricated by inserting a blade of steel into a slot machined into a standard type of ejector pin. The blade may be pinned or it may be brazed. The advantage of the two part construction is that the blade can easily be replaced should it become damaged.

The blade ejector element is fitted to the ejector assembly in an identical manner to the one described for the standard ejector pin. Figure 10-43 shows a cross sectional view through the mold that incorporates the ejection blade system.

The rectangular ejection blades are fitted in shaped holes in the mold “B” cavity plate. If this slot is to be machined into a solid mold plate, it presents a costly end-milling operation. Easing the machining of this aperture by using a commercial ejector blade subassembly is desirable. Therefore, a blade ejector can be used on the edge of a molded part. A slot is machined completely through the shoulder portion. Then the core insert is fitted into a suitable recess in the mold cavity plate forming a rectangular aperture, into which the ejector blade slides.

Mechanical Ejection Valve System

The valve ejector system is basically a large diameter poppet ejector pin. Valve type ejection is normally used for the ejection of relatively large components in situations where using standard parting surface pins is impractical. It is also used as an alternative to stripper plate ejection in certain situations.

The valve type ejector system is designed to apply the ejector force to the inside surface of the cavities as shown in Figure 10-44. We have previously noted (for pin ejection) that this is an undesirable practice because of the possibility of distorting the molded parts during ejection. However, with the mechanical valve ejection, the risk of the molded parts being distorted is reduced, because the poppet has a large effective ejection area.

The ejector element incorporates a valve type head (poppet) that is seated in a nest machined into the cores as shown in Figure 10-44. To ensure a leak free joint, a small parallel portion is provided at the major diameter of the valve. This is a sliding fit in a complementary parallel recess in the cores.

Because the ejector element forms a relatively large part of the cavity, a large force will be applied to the element by the melt tending to move it rearwards. The head must therefore be carefully bedded into its nest in the core insert block.

In operation, when the mold opens, the shank of the valve ejector element is moved by the knock-out bar of the injection machine and the molded parts are ejected. Immediately the mold closing stroke commences the return pins causing the valve to be returned to its seating. For fully automatic operation it is normal practice to use a jet of air to blow the molded part off the valve as there is a tendency in certain instances for the molded parts to stick to the ejection valves.

For a multi-cavity mold, an ejector assembly and an ejector grid must be used. The valve element is attached directly to the ejector plate by screws. A gap of 0.125 in minimum should be incorporated behind the ejector plate to ensure that both valves are seated in their respective nests. The return pins, not shown, are incorporated to return the ejector plate and valve elements to their rear positions as the mold is closed.

Air Poppet Valve Ejection System

With this ejection method the ejector force is provided by compressed air, which is introduced directly onto the lower cavity face via a small air ejector valve. For this method to operate efficiently, the adhesion between the cavity wall and the core must be broken locally to permit the compressed air to be introduced. This is achieved by causing the valve ejector to move forward slightly (by air pressure).

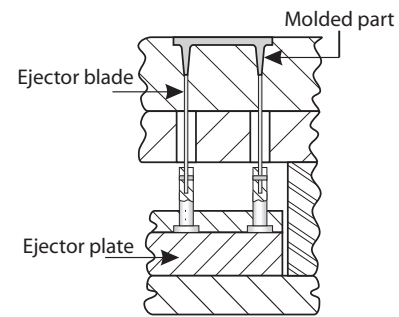


Figure 10-43 Mold ejection blade system

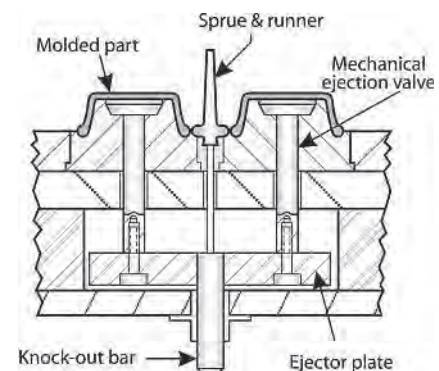


Figure 10-44 Multi-cavity mechanical ejection valve system

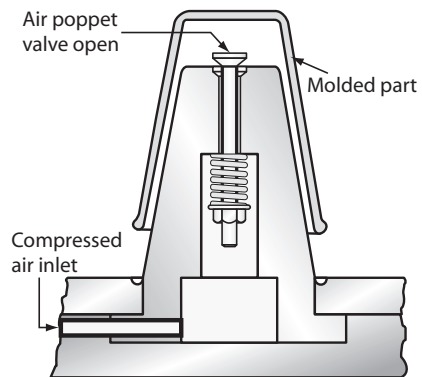


Figure 10-45 Air poppet valve ejection system

The effective ejector force depends on the pressure of the compressed air and the area on which it acts. Therefore, the larger the area of the component to be ejected, the greater the ejector force. There is, however, one drawback in that, while the compressed air finds an escape route, the effective ejector force is rapidly diminished. This method of ejection is particularly suitable for box-type components, where the side walls act as a seal during a major part of the ejection stroke, preventing the escape of the compressed air.

Figure 10-45 shows an air poppet valve ejector element. The head nestles in a complementary shaped recess in the core insert, while the shank of the element is a sliding fit in a hole next to the nest. To provide a passage for the compressed air, small slots are machined longitudinally on the periphery of the hole.

The element is held onto its seating by means of a compression spring. This spring is held in position by a nut. To prevent the loss of compressed air from the rear side of the core insert, a sealing plate and associated “O” ring are incorporated. A port connects the air ejector valve to the compressed air supply.

The compressed air can be introduced either manually or automatically. With manual control, the machine operator controls the two way valve at any suitable point in the cycle. With automatic control, a cam is fitted onto the machine bed so that at the required point in the opening stroke, the valve spool is “ON” and compressed air flows into the mold. Immediately, the roller (attached to the spool) runs off the cam at either end and the compressed air supply is cut off.

The operational sequence of the air poppet ejection valve system is as follows: when the control valve is opened, compressed air passes into a chamber. A force is applied to the air valve ejector that overcomes the force exerted by the compression spring and the valve ejector leaves its seating. At this point, the adhesion between the molded part and the core joint surfaces is broken in the area next to the air valve ejector. Compressed air is now free to flow from the chamber through the slots, through the open valve and into the space between the molded part and the core, ejecting the molded part.

Air Poppet Valve Ejection System Advantages

- No ejector grid or ejector assembly is required; lower mold costs
- Can be fitted in either mold half. It is a suitable method of ejecting box-type components that are to be center gated on the inside
- The ejection can be performed anytime during the mold opening
- The air valve ejector element acts as a vacuum breaker between the molded part and the core surfaces. This system should work in coordination with other ejection systems to complete the ejection.

Stripper Plate Ejection System

In very deep, hollow parts, which shrink very firmly onto the cores, individual ejector pins are no longer adequate, because the individual ejector pins are likely to pierce the bottom of a deep container. Such cases require ejection by means of a stripper plate. The stripper plate covers the entire rim of the container and pushes the molded part off the core as the ejector rod actuates the ejection mechanism. Locating this stripper plate on the stationary half of the mold to help in loosening certain parts when the core has to be located on the stationary half of the mold is also possible.

Figure 10-46 shows the principle of operation used in this stripper plate ejection technique. The stripper plate is mounted between the cavity plate and the core cavity plate. The aperture in the stripper plate is a sliding fit on the core.

When the mold starts to open, the stripper plate moves back with the core plate. The molded part sticks to the core because of the shrinkage forced during the cooling cycle. Once the molded part is clear of the cavity, the movement of the stripper plate is initiated, while the core plate continues the rearward movement. The molded part is then withdrawn from the core by the stripper plate and the molded part ejected. The mold is then closed in preparation for the next molding cycle.

The stripper plate depth must be relatively thick; the depth dimension must be sufficient to resist the bending forces applied to the stripper plate as the molded part is ejected.

The lengths of the support pillars are generally longer than what is specified for a corresponding two-plate mold. The reason is that the support pillars must support the stripper plate over the complete ejection stroke and also the projected mold moving half assembly when the mold is opened. The leader pin's dimensional tolerances must be precise enough to enter the mold's fixed half shoulder bushings during mold closing time, and to ensure precision alignment between the cavity and the core.

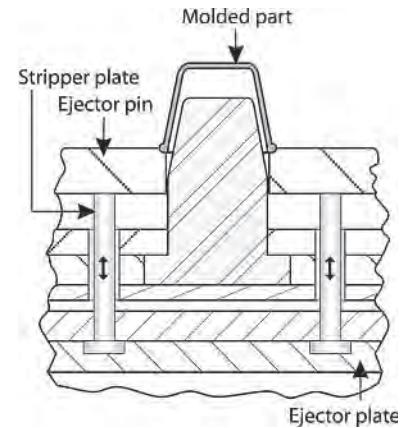


Figure 10-46 Stripper plate ejection system

10.13 Injection Mold Cooling

One fundamental principle of the thermoplastic injection molding process is that hot melt enters the mold cavity, where it cools rapidly to a temperature at which it solidifies sufficiently to retain the shape of the cavity. While the melt flows more freely in a mold cavity, a longer cooling period is required before the solidified molded parts can be ejected. On the other hand, while the melt solidifies quickly in a cold mold, it may not reach the extremities of the cavity. A compromise between the two extremes must be made to obtain the optimum molding cycle.

The thermoplastic injection molding process embodies the technological ability, innovation, and efficiency required for injection molding a product, while maximizing the amount of profit.

The thermoplastic injection molding process efficiency is affected by the mold cooling design. The difference in productivity between a correct and incorrect mold cooling design can represent an increase of 20 to 40% in the molding process costs.

The term mold cooling means lowering the temperature of the thermoplastic melt in the cavity to form a molded product. When molds require heat for proper operation they are being practically cooled as the temperature of the mold is lower than the thermoplastic melt temperature. The heat transfer flows from a high temperature source to a contacting element of lower temperature.

The main source of heat removal or mold cooling is obtained by an adequate circulation temperature control and volume control of the cooling fluid. If an adequate amount of properly treated water is available at any temperature and volume required by the molding process, the mold will be properly cooled.

Heat removal depends on temperature, pressure, viscosity, thermal diffusivity, and thermal conductivity. The heat transfer calculations are based on steady state or equilibrium conditions. In the thermoplastic injection molding process, the temperature, pressure, and viscosity are constantly changing as the melt flows and cools in the mold cavity. The complexities of the shapes of the molded products are beyond analytical determination. The molecular weight, molecular structure, and distribution of the thermoplastic melt are not constant.

In spite of all these variables, several mold cooling computer analysis programs have been developed by making many heat transfer assumptions to simulate the mold cooling process and the thermal behavior of the thermoplastic melt. These mold cooling programs are approximations and have some technical value in the development of new products and for a novice engineer who is learning thermoplastic injection mold cooling technology.

During the cavity filling stage, the hottest material will be near the entry point, i.e., the gate, and the coolest material will be at the point farthest from the entry. The temperature of the coolant fluid, however, increases as it passes through the mold. Therefore, to achieve an even cooling rate over the molding surface, it is necessary to place the incoming coolant fluid next to the mold cavity surfaces. Ultimately, adopting the idealized approach is not always practical and the mold designer must use a fair amount of common sense when laying out coolant circuits to avoid unnecessarily expensive molds.

The layout of a circuit is often complicated by the fact that the cooling channels must not be drilled too close to any other hole in the same mold plate. The mold plate has several holes or recesses, to fit ejector pins, support pillars, guide bushings, sprue bushing, cavity, and core inserts, etc.

To obtain the best possible position for a mold cooling circuit, it is good practice to lay the circuit in at the earliest opportunity in the mold design. The other mold components such as ejector pins, bushings, vents, etc., can then be positioned accordingly.

10.13.1 Mold Temperature Control

The thermoplastic injection molding process requires the rapid removal of heat from the mold cavities, so that the molded parts can be removed from the mold in the shortest time and in a condition that the parts meet the quality control requirements.

The mold temperature control system includes the mold, the cooling channels, the different mold cooling systems used in the mold design, the type of fluid with adequate capacity for its circulation, and the method of temperature control.

Predicting the best temperature conditions for a given mold and thermoplastic material is neither possible nor necessary. In many applications there will be several different temperatures maintained for different mold components to meet the quality control specifications. These process settings should be evaluated to select the best balance between economics and part quality.

The mold cavity surface temperature is measured at the beginning of the molding cycle, for example 140 °F. When the hot thermoplastic melt is injected into the mold cavity, the mold temperature increases to 160 °F. At the end of each molding cycle, the mold temperature decreases to 140 °F. A constant mold temperature means that the amount of heat removed per shot by the total cooling system is the same as the amount of heat provided by the hot thermoplastic melt. Mold

temperature fluctuations will affect the dimensional control of the molded part, warpage, flashing, surface finishing, and reduce physical properties.

The mold cavity reaches its maximum temperature very quickly and the mold cooling stage of the cycle is used to reduce this temperature to the base operating temperature. This rate depends on the temperature differences, the area of the cooling surfaces in the mold and in the cooling channels, the transfer rates of the heat from the thermoplastic melt to the thermoplastic/metal interface through the metal and through the metal/water interface. The heat removed by the mold and radiated into the operating molding area is relatively constant.

It is important that a mold cooling system produces a uniform temperature over the entire surface of the mold cavity at the same level as the cooling channels. It will remove heat from the thermoplastic melt at the highest possible rate consistent with the quality and properties required in the molded part.

10.13.2 Factors Affecting Mold Cooling

The mold temperature is affected by several thermoplastic injection molding process factors related to mold cooling:

- Thermoplastic material (process melt temperature, crystallization rates, modulus of elasticity)
- Part wall thickness, size, complexity, dimensional control, and finish
- Shot weight, process automation, cooling time
- Mold base, cavity, and core material
- Size and shape of the mold, cavity, and core
- Efficiency of mold cooling systems
- Size and location of cooling channels
- Velocity, capacity, pressure drop, and temperature of cooling fluid
- Operating ambient conditions (temperature, moisture, and air flow)

10.13.3 Effects Caused by Elevated Mold Temperature

If the mold temperature is too high, the molding cycle time increases, reducing the efficiency of the injection molding process. The thermoplastic melt remains in the fluid stage longer with higher mold temperatures, causing the melt to flash through the parting line vents. Higher thermoplastic melt flow rates cause burn marks on the molded parts and steel corrosion of the cavities. Even if the melt does not burn, the inadequate venting may seriously reduce the physical properties of the molded part at the weld lines.

With a hot mold, the gate will remain open longer, permitting more melt to be packed into the cavity. The excessive ejection forces may lead to molded-in stresses and failure of the molded part during ejection or in service. This extra injection pressure results in forcing the thermoplastic melt more tightly against the cavity surface, requiring higher ejection forces that may lead to ejection problems. When ejecting a hot molded part, the ejector pins may cause ejection marks, increasing the stress or causing perforation of the part

surface. Over-packing the cavities will increase the deformation (deflection) of the mold steel plate that may cause shifting, dimensional control, and other ejection problems.

When hot thermoplastic melt hits the cold mold cavity surface, the melt freezes against the cavity wall, orienting the molecular structure in the direction of the flow. Orientation leads to molded-in stress of the molded parts; they have a higher tensile strength in the direction of the flow, because they have more carbon-to-carbon molecular linkages than in the direction perpendicular to the melt flow. The carbon-to-carbon linkage forces are stronger than the weaker electrostatic bonds that are predominantly perpendicular to the direction of the melt flow.

The sprue performance is also affected by the temperature of the mold (too hot). The sprue requires a longer cooling time to become solid, a hot sprue will break or stick, causing cycle interruptions. Too high a mold temperature causes excessive shrinkage, sink marks, and voids. If one side of the mold overheats, the thermal expansion of that side may cause the leader pins to seize in the bushings; this condition can pull the mold off from the molding machine platen.

10.13.4 Effects Caused by Too Low a Mold Temperature

Cold molds cause processing problems, such as incomplete molded parts, poor surface finishing, oversize parts, uneven density distribution causing severe molded-in stresses that may lead to immediate or premature failure.

A cold mold causes the melt to freeze before it fills the cavity giving an incomplete shot. If it does fill, there may not be enough melt in the cavity causing voids, sink marks, splay marks, poor weld lines, poor mold shrinkage control, and reduction of the physical properties of the part causing brittleness, poor temperature resistance, and poor dimensional stability at elevated temperatures.

These are just some major problems caused by the mold temperature conditions being too cold. These are all compounded if the cooling system lacks the capacity or instrumentation to maintain a consistent operation.

10.13.5 Mold Heat Transfer Methods

There are three methods for transferring heat in the injection molding process: radiation, convection, and conduction. The hot thermoplastic melt from the plastifying unit nozzle is transferred into the mold sprue bushing. The heat from the thermoplastic melt moves by convection through the polymer until it reaches the cavity surface of the mold. The heat is then conducted through the mold cavities to the mold cooling system and the cooling channels. The heat is then transferred to the cooling fluid. A substantial amount of the heat reaches outside the mold surface, where the heat is lost by radiation.

Of the heat added by the plastifying unit to the thermoplastic melt in the injection molding process, approximately 60% is removed by the mold; however, some heat is left in the molded parts after they are removed from the mold cavities after ejection. Approximately 35% of the heat is removed by radiation from the mold and 25% is removed by the cooling fluid that controls the mold cavity surface temperature. The percentages vary, depending on the thermoplastic material, melt process temperature, mold temperature control efficiency, molding cycle, molded product wall thickness, application requirements, size and complexity of the product design.

10.13.5.1 Cooling Channel Diameter

Injection molds with a cooling channel circuit require seals in the connections to prevent leakage of the cooling fluid. The presence of cooling fluid on the surface of the mold cavities will cause defects in the surface and reduction of the physical properties of the molded part. Pipe threads are used to connect the cooling channels with the other components and this is an effective and economical type of seal. The cooling channel diameter is related to the size of the pipe fitting inside diameter; the point of a change in cross section is largely problem free. With inserted cores having drilled cooling channels, such sealing problems are avoided with threaded pipes. Figure 10-47 shows a cross section assembly of a pipe fitting/mold plate and a table for the cooling channel diameters per nominal pipe sizes.

Nominal pipe size (in)	Tap drill used (in)	Channel dia. "D" (in)	Surface area "A" (ft ²)
1/8	5/16	0.3125	0.982
1/4	7/16	0.4375	1.374
3/8	9/16	0.5625	1.767
1/2	11/16	0.6875	2.160

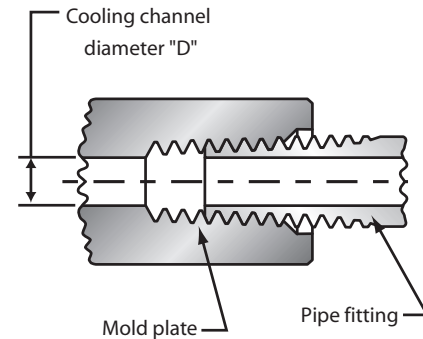


Figure 10-47 Cooling channel related to nominal pipe size

10.13.5.2 Mold Cooling Channel Location

The rate of heat removal from the mold will vary directly with the thermal conductivity of the material used in the construction of the mold components. Table 10-5 states the values of various materials used for mold construction. From this table, we can see that beryllium copper will remove heat approximately six times faster than tool steel and eight times faster than stainless steel.

Table 10-5 Thermal Conductivity of Mold Materials

Mold material	Description	Thermal conductivity (BTU/h/ft ² /°F/in)
1020 Alloy steel	Hot rolled steel	20.00
4135 Alloy steel	Medium carbon steel	24.70
P-20 Steel	Tool steel	20.00
S-7 Steel	Alloy tool steel	21.00
H-13 Steel	Alloy tool steel	16.30
420 Alloy steel	Stainless steel	14.40
440 Alloy steel	Stainless steel	14.00
Kirksite	Cast low Carbon steel	62.00
Brass	Brass casting	70.00
Moldmax® copper alloy	Beryllium copper	60.00
Ampco 940 copper alloy	Ampco copper	125.00
Protherm® copper alloy	Beryllium copper	145.00
2024 Aluminum alloy	Aluminum	70.00
6061 Aluminum alloy	Aluminum	96.00
7010 Aluminum alloy	Aluminum	92.00
7075 Aluminum alloy	Aluminum	75.00

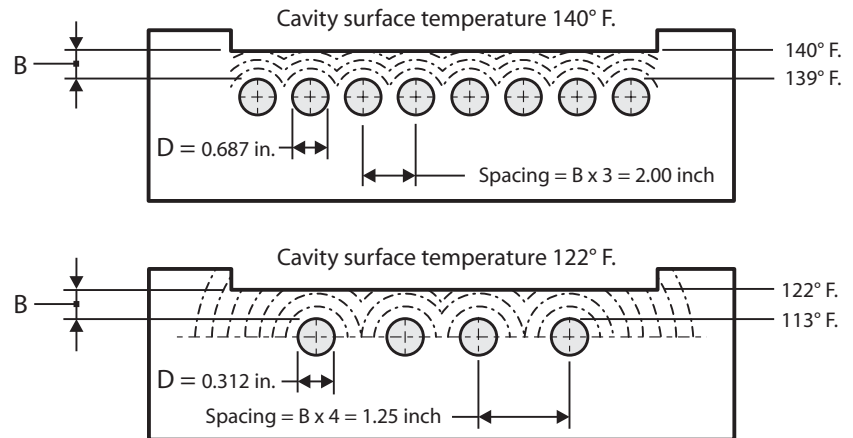


Figure 10-48 Surface temperature profile per channel diameter, location, spacing, and cooling fluid temperature

10.13.5.3 Cavity Surface Temperature Profiles

Figure 10-48 shows the surface temperature profiles, based on the cooling channel diameters and spacing. The minimum cooling channel location in reference to the cavity surface for maximum cooling to maintain the integrity of the mold varies with the modulus of elasticity of the cavity insert material.

For example, if the cavity insert is made from tool steel, the edge distance from the cavity surface to the tangent of the cooling channel diameter (B) should be equivalent to the cooling channel diameter (D). If the cavity plate is made from beryllium copper, this distance (B) should be 1.50 diameters. The reason is that the modulus of elasticity of tool steel is 30,000,000 psi, while that of beryllium copper is only 20,000,000 psi. Therefore, the distance from the cooling channel to the cavity surface is 50% more for beryllium copper than for tool steel. This reduces the effective cooling advantage of beryllium copper over tool steel to a factor of two rather than three. The closer the cooling channel is to the thermoplastic melt the higher the rate of heat removal.

Drilling long cooling channels requires accurate machining. If the hole wanders too close to the surface edge of the cavity, the plate may collapse under molding pressures. If the spacing between the cooling channels and the cavity surface varies, so will the cooling rate.

The very low thermal conductivity of the thermoplastic materials prevents the rapid transfer of heat from inside the molten thermoplastic wall to the external surface where it can move by conduction through the metal of the mold into the cooling fluid.

10.13.5.4 Spacing Between Cooling Channels

The smaller the spacing, the more uniform the temperature in the mold cavity and the higher the temperature increases at the surface of the cavity during injection. It is recommended that the spacing between the cooling channels be between three and four cooling channel diameters (D). If, however, the design requires a larger spacing between cooling channels, the spacing (B) from the cavity surface and the channel diameter (D) must also increase. With increasing wall thickness, more heat must be removed from the mold, requiring an increase in the cooling channel diameters (D).

10.13.5.5 Mold Heat Removal Analysis

Some factors that affect the rate and amount of heat transfer are the thermoplastic material, size, geometry, and wall thickness of the molded parts, the melt flow rate, melt viscosity, specific heat, thermal conductivity, density, molding process, and mold temperature operating conditions.

How much heat is transferred in the mold is calculated by using a complex set of equations to obtain the values for the Reynolds number, Prandtl number, Nusselt laminar flow number, Nusselt turbulent flow number, heat transfer coefficient, and heat transfer rate.

Reynolds Number

Osborne Reynolds discovered some basic principles of flow in the 18th century. He took a container with a source of inlet water that kept it always full. A pot of dye was fed externally into a tube. Water could be bled from this container at a controlled rate through an outlet valve. When the flow rate in the tube was low, the dye assumed a straight path parallel to the tube. Reynolds called this flow rate condition laminar flow. As the flow increased, mixing of the dye began to occur. When flow became very rapid there was a complete mixing of the dye in the tube. This he called turbulent flow. He derived a formula to describe this phenomenon for a circular tube using a dimensionless number now known as the Reynolds number.

In a laminar flow, the fluid will glide along the walls, unlikely to collide with the walls or other molecules or to distract the flow from the direct straight line path. In a turbulent flow, the fluid molecules are moving with much higher speed and slam into the walls, giving better heat transfer characteristics. It is desirable to have a Reynolds number higher than 3,500 as the minimum starting point to control the temperature of the mold.

For example, the turbulent flow rate of cooling water at 50 °F has similar cooling characteristics to the laminar flow rate of water at 32 °F.

Reynolds Number Equation

$$Re = \frac{v \times D \times \rho}{12 \times \mu_F} \quad (10-1)$$

Prandtl Number Equation

$$Pr = \frac{3,600 \times c_p \times \mu_F}{K} \quad (10-2)$$

Nusselt Laminar Flow Number Equation

$$Nu = 1.86 \left(Re \times Pr \times \frac{D}{12 \times L} \right)^{0.33} \left(\frac{\mu_C}{\mu_F} \right)^{0.14} \quad (10-3)$$

Nusselt Turbulent Flow Number Equation

$$Nu = 0.116 \times Pr^{0.33} (Re^{0.66} - 125) \left[1 + \left(\frac{D}{12 \times L} \right)^{0.66} \right] \left(\frac{\mu_C}{\mu_F} \right)^{0.14} \quad (10-4)$$

Heat Transfer Coefficient Equation

$$U = \frac{\text{Nu} \times k \times 12}{D} \quad (10-5)$$

The heat transfer rate is equal to the heat transfer coefficient, multiplied by the exposed surface cooling area, multiplied by the difference in temperature between the mold cavity and the coolant fluid.

Heat Flow Rate Equation

$$Q = U \times A \times L \times (T_C - T_F) \quad (10-6)$$

Re = Reynolds number

Pr = Prandtl number

Nu = Nusselt flow number

L = Cooling channel length (ft)

v = Cooling fluid velocity (ft/s)

D = Cooling channel diameter (in)

ρ = Cooling fluid density (lb/ft³)

μ_F = Cooling fluid viscosity (lb-s/in²)

μ_C = Cooling fluid viscosity at cavity temperature ((lb-s/in²)

k = Cooling fluid thermal conductivity (BTU/h/ft²/°F/in)

c_p = Cooling fluid specific heat (BTU-lb-°F)

T_C = Surface cavity temperature (°F)

T_F = Cooling fluid temperature (°F)

A = Exposed surface cooling area (ft²) = $(\pi \times D) / 12$

U = Heat transfer coefficient (BTU/ft²-h-°F)

Q = Heat flow rate (BTU/h)

Table 10-6 Properties of Circulating and Cleaning Cooling Fluid

	50% EG 50% Water 32 °F	Water 50 °F	Water 85 °F	Water 100 °F	50% EG 50% Water 150 °F	Water 150 °F
Viscosity μ (lb-s/in ²)	0.00554	0.000878	0.000515	0.000456	0.001	0.000287
Density ρ (lb/ft ³)	66.45999	62.3799	62.1677	62.000		
Thermal conductivity K (BTU/h/ft ² /°F/in)	0.240	0.333	0.357	0.363		
Specific heat C_p (BTU/ft ² -h-°F)	0.760	1.0490	0.9980	0.9980		
Prandtl number Pr	63.16	9.97	5.18	4.15		

EG = Inhibited ethylene glycol fluid

From the heat transfer rate (Eq. 10-6) the conditions to increase the rate can be deducted:

- Increase the difference in temperature between the mold cavity surface and the mean cooling fluid
- Improve the efficiency of the heat transfer coefficient
- Increase the exposed surface cooling area.

There are two ways to increase the difference in temperature between the thermoplastic melt and the cooling fluid. One way would be to add more heat to the thermoplastic melt, this approach is self defeating, increasing the molding cycle time. Lowering the mold cooling fluid temperature may cause molding problems.

Increasing the exposed surface cooling area channels and making them as large as possible, is very beneficial. The limitations are that large-diameter channels will reduce the cooling fluid velocity and decrease the Reynolds number. Using a greater number of small diameter channels will increase the pressure drop in the cooling fluid system or lead to a reduction of fluid that can be passed through the cooling system.

Example 10-2

Determine the Reynolds number and the heat flow rate of an injection mold required to operate with a mold cavity temperature of 150 °F using water at 50 °F as cooling fluid at a flow velocity of 0.50 (ft/s), through a cooling channel 1 ft long with a diameter of 0.4375 in behind the cavity insert.

Reynolds Number

$$\begin{aligned} \text{Re} &= \frac{v \times D \times \rho}{12 \times \mu_F} \\ &= \frac{0.5 \times 0.4375 \times 62.3799}{12 \times 0.000878} = 1,295.14 \text{ (Laminar Flow)} \end{aligned} \quad (10-1)$$

Prandlt Number

$$\begin{aligned} \text{Pr} &= \frac{3,600 \times c_p \times \mu_F}{k} \\ &= \frac{3,600 \times 1.049 \times 0.000878}{0.333} = 9.956 \end{aligned} \quad (10-2)$$

Nusselt Laminar Flow Number

$$\begin{aligned} \text{Nu} &= 1.86 \left(\text{Re} \times \text{Pr} \times \frac{D}{12 \times L} \right)^{0.33} \left(\frac{\mu_C}{\mu_F} \right)^{0.14} \\ &= 1.86 \left(1295 \times 9.956 \times \frac{0.4375}{12 \times 1.0} \right)^{0.33} \left(\frac{0.000278}{0.000878} \right)^{0.14} = 16.920 \end{aligned} \quad (10-3)$$

Heat Transfer Coefficient

$$\begin{aligned}
 U &= \frac{\text{Nu} \times K \times 12}{D} \\
 &= \frac{16,920 \times 0.333 \times 12}{0.4375} = 145.54 \text{ (BTU/ft}^2\text{-h-}^\circ\text{F)} \quad (10-5)
 \end{aligned}$$

Heat Flow Rate

$$Q = U \times A \times L \times (T_C - T_F) \quad (10-6)$$

$$A = \frac{\pi \times D}{12} = \frac{3.1416 \times 0.4375}{12} = 0.114 \text{ (ft}^2\text{)}$$

$$Q = 145.54 \times 0.114 \times 1.0 \times (150 - 50) = 1,659.156 \text{ (BTU/h)}$$

Example 10-3

Determine the Reynolds number and the heat flow rate of an injection mold required to operate at a mold cavity temperature of 150 °F using water at 50 °F as cooling fluid at a flow velocity of 2.00 (ft/s), through a cooling channel 1 ft long with a diameter of 0.4375 in behind the cavity insert.

Reynolds Number

$$\begin{aligned}
 \text{Re} &= \frac{v \times D \times \rho}{12 \times \mu_F} \\
 &= \frac{2.0 \times 0.4375 \times 62.3799}{12 \times 0.000878} = 5,180.56 \text{ (Turbulent Flow)} \quad (10-7)
 \end{aligned}$$

Prandlt Number

$$\begin{aligned}
 \text{Pr} &= \frac{3,600 \times c_p \times \mu_F}{k} \\
 &= \frac{3,600 \times 1.049 \times 0.000878}{0.333} = 9.956 \quad (10-8)
 \end{aligned}$$

Nusselt Turbulent Flow Number

$$\begin{aligned}
 \text{Nu} &= 0.116 \times \text{Pr}^{0.33} (\text{Re}^{0.66} - 125) \left[1 + \left(\frac{D}{12 \times L} \right)^{0.66} \right] \left(\frac{\mu_C}{\mu_F} \right)^{0.14} \\
 &= 0.116 \times 9.956^{0.33} (5,180.56^{0.66} - 125) \\
 &\quad \times \left[1 + \left(\frac{0.4375}{12 \times 1.0} \right)^{0.66} \right] \left(\frac{0.000278}{0.000878} \right)^{0.14} \\
 &= 56.487 \quad (10-9)
 \end{aligned}$$

Heat Transfer Coefficient

$$\begin{aligned}
 U &= \frac{\text{Nu} \times k \times 12}{D} \\
 &= \frac{56.487 \times 0.333 \times 12}{0.4375} = 515.936 \text{ (BTU/ft}^2\text{-h-}^\circ\text{F)} \quad (10-10)
 \end{aligned}$$

Heat Flow Rate

$$Q = U \times A \times L \times (T_C - T_F) \quad (10-11)$$

$$A = \frac{\pi \times D}{12} = \frac{3.1416 \times 0.4375}{12} = 0.114 \text{ (ft}^2\text{)}$$

$$Q = 515.936 \times 0.114 \times 1.0 \times (150 - 50) = 5,881.67 \text{ (BTU/h)}$$

Example 10-4

Determine the Reynolds number and the heat flow rate of an injection mold required to operate at a mold cavity temperature of 150 °F using 50% additive and 50% water at 32 °F as cooling fluid at a flow velocity of 2.00 (ft/s), through a cooling channel 1 ft long with a diameter of 0.4375 in behind the cavity.

Reynolds Number

$$\begin{aligned}
 \text{Re} &= \frac{v \times D \times \rho}{12 \times \mu_F} \\
 &= \frac{2.0 \times 0.4375 \times 66.4599}{12 \times 0.005540} = 874.73 \text{ (Laminar Flow)} \quad (10-12)
 \end{aligned}$$

Prandtl Number

$$\begin{aligned}
 \text{Pr} &= \frac{3,600 \times c_p \times \mu_F}{k} \\
 &= \frac{3,600 \times 0.760 \times 0.005540}{0.240} = 63.156 \quad (10-13)
 \end{aligned}$$

Nusselt Laminar Flow Number

$$\begin{aligned}
 \text{Nu} &= 1.86 \left(\text{Re} \times \text{Pr} \times \frac{D}{12 \times L} \right)^{0.33} \left(\frac{\mu_C}{\mu_F} \right)^{0.14} \\
 &= 1.86 \left(874.73 \times 63.156 \times \frac{0.4375}{12 \times 1.0} \right)^{0.33} \left(\frac{0.001}{0.00554} \right)^{0.14} \quad (10-14) \\
 &= 28.860
 \end{aligned}$$

Heat Transfer Coefficient

$$\begin{aligned}
 U &= \frac{\text{Nu} \times K \times 12}{D} \\
 &= \frac{28.860 \times 0.240 \times 12}{0.4375} = 189.981 \text{ (BTU/ft}^2\text{-h-}^\circ\text{F)} \quad (10-15)
 \end{aligned}$$

Heat Flow Rate

$$Q = U \times A \times L \times (T_C - T_F) \quad (10-16)$$

$$A = \frac{\pi \times D}{12} = \frac{3.1416 \times 0.4375}{12} = 0.114 \text{ (ft}^2\text{)}$$

$$Q = 189.981 \times 0.114 \times 1.0 \times (150 - 32) = 2,555.62 \text{ (BTU/h)}$$

Example 10-5

Determine the Reynolds number and the heat flow rate of an injection mold required to operate at a mold cavity temperature of 150 °F using water at 50 °F as cooling fluid at a flow velocity of 2.00 (ft/s), through a 4.00 ft long cooling channel with a diameter of 0.4375 in and behind the cavity.

Reynolds Number

$$\begin{aligned}
 \text{Re} &= \frac{v \times D \times \rho}{12 \times \mu_F} \\
 &= \frac{2.0 \times 0.4375 \times 62.3799}{12 \times 0.000878} = 5,180.56 \text{ (Turbulent Flow)} \quad (10-17)
 \end{aligned}$$

Prandtl Number

$$\begin{aligned}
 \text{Pr} &= \frac{3,600 \times c_p \times \mu_F}{k} \\
 &= \frac{3,600 \times 1.049 \times 0.000878}{0.333} = 9.956 \quad (10-18)
 \end{aligned}$$

Nusselt Turbulent Flow Number

$$\begin{aligned}
 \text{Nu} &= 0.116 \times \text{Pr}^{0.33} (\text{Re}^{0.66} - 125) \left[1 + \left(\frac{D}{12 \times L} \right)^{0.66} \right] \left(\frac{\mu_C}{\mu_F} \right)^{0.14} \\
 &= 0.116 \times 9.956^{0.33} (5,180.56^{0.66} - 125) \\
 &\quad \cdot \left[1 + \left(\frac{0.4375}{12 \times 4.0} \right)^{0.66} \right] \left(\frac{0.000278}{0.000878} \right)^{0.14} \\
 &= 53.160 \quad (10-19)
 \end{aligned}$$

Heat Transfer Coefficient

$$\begin{aligned}
 U &= \frac{\text{Nu} \times K \times 12}{D} \\
 &= \frac{53.160 \times 0.333 \times 12}{0.4375} = 485.548 \text{ (BTU/ft}^2\text{-h-}^\circ\text{F)}
 \end{aligned}
 \tag{10-20}$$

Heat Flow Rate

$$Q = U \times A \times L \times (T_C - T_F) \tag{10-21}$$

$$A = \frac{\pi \times D}{12} = \frac{3.1416 \times 0.4375}{12} = 0.114 \text{ (ft}^2\text{)}$$

$$Q = 485.548 \times 0.114 \times 1.0 \times (150 - 50) = 22,140.98 \text{ (BTU/h)}$$

10.13.5.6 Heat Transfer Barriers

Clean circulating fluid, mold cooling system, and cooling channel conditions are very important for mold temperature control. To avoid internal corrosion in the mold cooling channels, it is highly recommended to coat the channels with electroless nickel or an equivalent coating before the mold goes into production. Being able to clean the mold cooling channels is also essential. Blind holes should be eliminated. When possible, the mold cooling channels should be drilled to the end of the mold plates and plugged. One way of cleaning them is with copper gauze or a wire brush on a wooden mandrel attached to a hand drill. Another method is using circulating mold cooling channel cleaning equipment with a cleaning agent designed to remove degraded circulating fluid. High temperature heat transfer fluids and ethylene glycol that contain deposits of carbon, scale, sludge, and most foreign matter found in cooling and heating circuits can be removed during maintenance of the mold.

Air gaps between the assembled components of the mold cause a tremendous loss in heat transfer. No matter how well a cavity or core insert is fitted into a mold cavity plate, there will be an air gap. Connecting the cooling channels into the cavities and core inserts wherever possible is essential.

The use of insulation sheets on the external surfaces of the top and bottom clamping plates of the mold improve the mold cooling efficiency by preventing heat dissipation through the injection molding machine platens.

10.13.5.7 Injection Mold Temperature Controllers

Closed loop injection mold temperature controllers are used to control the velocity of the heat exchange fluid. These closed circulation systems are available in various models, depending on the application (temperature, capacity, number of cooling units, type of cooling fluid, etc.). Heat is supplied when required by electric heaters. Cooling is supplied by the heat exchange of refrigerated or tower water through copper coils in the controller.

This system has two advantages. It provides fast coolant circulating rates and a high Reynolds number (3,500 or higher), equivalent to a turbulent flow. It also

permits easy treatment of distilled water to prevent scale from forming in the cooling channels of the mold, keeping the cooling channels and the cooling system clean. However, since because cooling tanks are made of stainless steel they should have manifolds to permit different flow rates to different parts of the mold. Each manifold should have its own pressure gauge and thermometer, eliminating a major cause for loss of efficiency in the cooling system.

Connecting water lines is time consuming. It may account for as much as half of the mold set-up time. This often leads to shortcuts by the set-up crew who do not always understand the necessity for the best possible mold cooling.

Determining the best mold temperature conditions means switching water lines and seeing the results. To ease set-up and experimenting with the mold, quick disconnect fittings and high temperature/pressure hoses are used to connect the mold to the cooling system. The male quick disconnect plugs are installed on the mold either protruding or countersunk and can be left in the mold. The quick disconnect female or socket has a ball check in it so that when it is disconnected it does not leak.

There should be no restriction in the flow because the inside diameter of the hoses or of the quick disconnect fittings are smaller than the inside diameter of the system. It is preferable that the hose diameters be slightly larger. Manifolds at the source of water for intake and return should be available.

The larger the temperature differences between the thermoplastic melt to be cooled and its cooling fluid, the quicker the rate of heat removal.

10.13.5.8 Mold Cooling Fluid

The most important properties of the cooling fluid are the heat transfer coefficient and the flow viscosity. Salt water is one of the best cooling fluids, but causes steel corrosion inside the cooling channels.

A common and practical closed loop cooling fluid is a blend of distilled water with undiluted ethylene glycol additive with corrosion inhibitors, specifically formulated for use in injection mold cooling. This special additive has a synergistic package of industrial inhibitors that protect metal components by forming an extremely thin, bonded chemical coating. The inhibitors also keep the PH slightly on the base (non-acid) side, offering further corrosion protection.

A blend of 25% additive and 75% distilled water for mold cavity temperatures between 28 °F and 197 °F and 50% additive and 50% distilled water for mold cavity temperatures between 0 °F and 204 °F is recommended. Do not use more than 65% of the circulating additive, because it reduces the heat transfer properties, making the cooling fluid more viscous, thus reducing the flow velocity.

This circulating fluid additive is a better additive than pure ethylene glycol, which is as corrosive as water and almost neutral in PH. When exposed to air in a circulator tank, it quickly becomes acidic and even more corrosive. It is also better than an automotive antifreeze additive, which also becomes acidic very quickly. While based on ethylene glycol, antifreeze has silica-based inhibitors that depend on agitation to scrub away corrosion. After two years, the inhibitors form a silica gel that prevents heat transfer and the entire cooling fluid with antifreeze must be replaced.

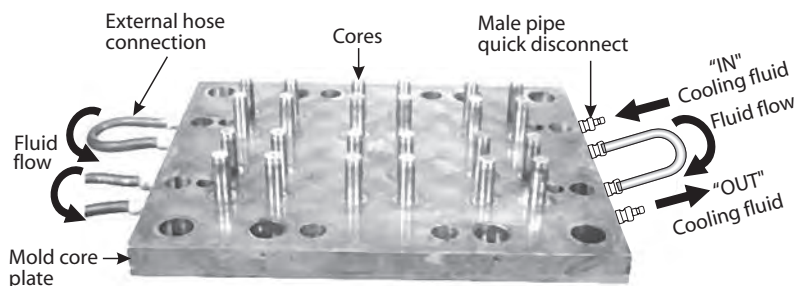


Figure 10-49 Mold core plate parallel cooling with hose connections

10.13.5.9 Mold Plate Cooling Systems

The temperature of a mold plate is controlled by circulating cooling fluid through cooling channels drilled in the mold plate. The cooling channels are normally interconnected to form a circuit. The cooling circuit may be formed by one or more levels, depending on the depth of the mold cavity plate.

The stationary cavity inserts and the movable core plates should always be cooled separately, using two separate mold cooling controllers to adjust each cooling fluid flow rate and temperature.

Parallel Flow Mold Cooling

The method of drilling parallel cooling channels across the mold plate is not recommended. The cooling fluid temperature increases proportionally to the traveling distance through the mold cooling channels. This leads to temperature variations between the mold cavities, affecting the mold shrinkage of the molded parts, increasing the risk of significant dimensional differences between the cavities.

To improve the mold cooling efficiency, a uniform turbulent flow around the cavities and the gate areas is recommended, where spiral or curved cooling channels may be located close to the gate.

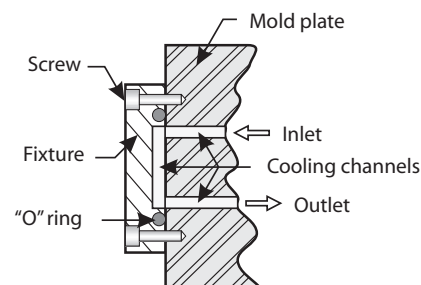
Mold Plate Parallel Circuit Cooling

Parallel circuit cooling using external hose connections is an economical method for cooling mold plates. Figure 10-49 shows this cooling method. The temperature across the mold plate is nonuniform, but the cooling fluid entrance area is the coldest section of the plate. The cooling fluid removes heat from the hot cores, increasing its temperature as it travels through the parallel cooling channels. The cores placed close to the cooling channel entrance have lower temperatures than the cores located at the cooling circuit discharge end.

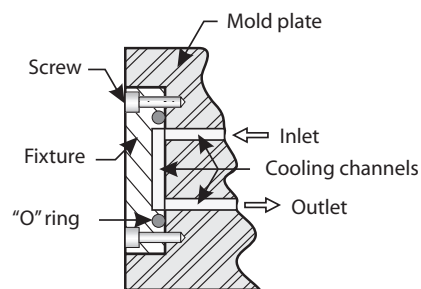
10.13.5.10 Mold Plate Cooling Channel Connections

The simplest approach for cooling a mold plate having a small, shallow cavity is to drill two cooling channels, one on either side parallel to the cavity, connected at one end by means of a mechanical fixture, using an internal cross drilling or a milled slot in conjunction with a connecting mold plate.

Figure 10-50 shows the two design concepts for connecting the parallel cooling channels, using a mechanical fixture with an internal channel milled in the base wall (mold plate), providing a continuous flow path for the coolant. These mechanical connecting fixtures are mounted onto the side of the mold plate and



Surface mount cooling channel connecting fixture



Sunk mount cooling channel connecting fixture

Figure 10-50 Mold plate cooling channels connecting fixture designs

secured by screws. A gasket is incorporated to prevent leakage of the cooling fluid.

The top illustration shows the mechanical connecting fixture externally mounted onto the mold plate. This design is cheaper, but it suffers from the disadvantage that the connecting fixture may be disturbed during the mold setting operation and cooling fluid leakage may occur.

The bottom illustration shows the mechanical connecting fixture mounted in a pocket, flush with the side surface of the mold plate. This alternative design is recommended for connecting the parallel cooling channels without interferences.

10.13.5.11 Mold Cavity Plate Cooling Systems

Figure 10-51 shows a simple mold cavity layout using three methods for cooling the mold cavity plate.

The top illustration shows the mold cavity plate connected with three external hoses, starting with a top level going to the bottom and coming back to the top again. This cooling method is not recommended, because the cavity temperature is nonuniform, causing uneven cooling.

The lower left illustration shows the mold cavity plate being cooled by the cooling fluid. The drilled channels are interconnected and plugged, going in and around three sides of the mold cavity plate. This method will also give uneven cooling as the left hand side of the cavity will be warmer because there is no cooling channel to remove the heat.

The lower right illustration shows the proper mold cavity plate cooling method. The cooling fluid goes in, flows around all sides of the cavity and out. Pipe plugs that can be removed are used for easy cleaning of the cooling channels.

Figure 10-52 shows a mold cavity plate with four cavities and a single cooling circuit formed by drilling cooling channels, using plugs to seal the end of the channels and placing a diverting plug to control the cooling flow direction.

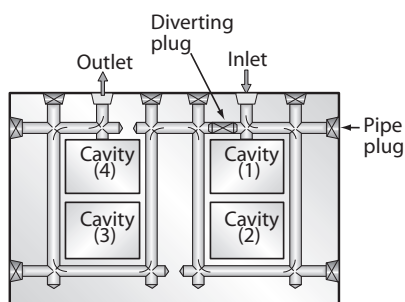


Figure 10-52 Four-cavity mold plate cooling channel pattern design

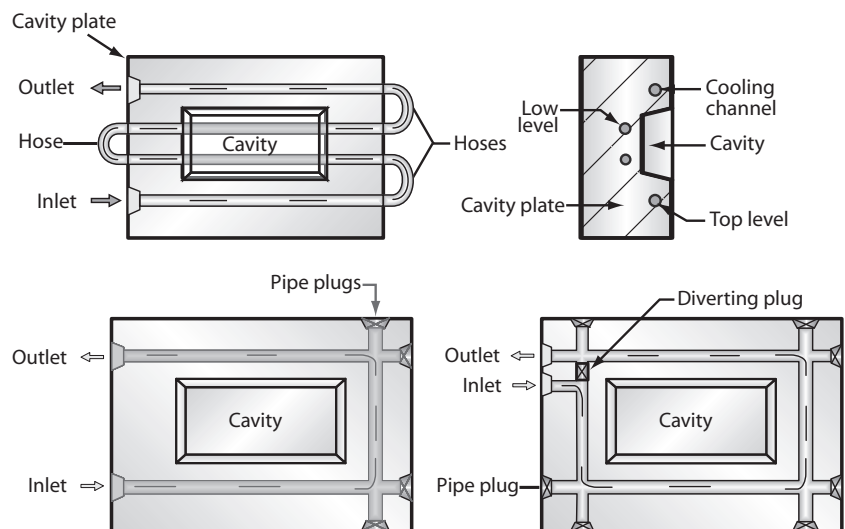


Figure 10-51 Mold cavity plate cooling channel connection systems

This mold cooling system does not provide an even temperature control for the cavities. The sides of the cavities that are next to each other are hotter than the other sides, because the heat cannot be removed from these areas. Additionally, the average temperature in each cavity is different. Cavity (4) is the coldest, while cavity (1) has the highest temperature, because of the effects of a single cooling circuit and the mold layout. This mold plate cooling method is not recommended for critical molding applications.

10.13.6 Mold Cavity Insert Cooling

Mold cavity insert cooling consists of spiral grooves cut to cool the outside surface of the cavity insert. Figure 10-53 shows a cavity insert cross section assembly and the grooves cut into the cooling plate. This cooling method provides the best performance and uniform cooling temperature of the cavity surface. This method is used when drilling inside the cavity is difficult or when added cooling is needed. The cooling fluid circulates around the cooling plate spiral groove, starting from the middle and exits at the end of the spiral groove or cavity outside diameter. The area exposed to the cooling fluid (formed by the grooves) is in direct contact with the cavity base wall to control the uniformity of the cavity surface temperature. It is best to use an "O" ring rather than other types of gaskets, because the "O" ring allows a metal-to-metal seal. This makes holding the cavity insert in place easier.

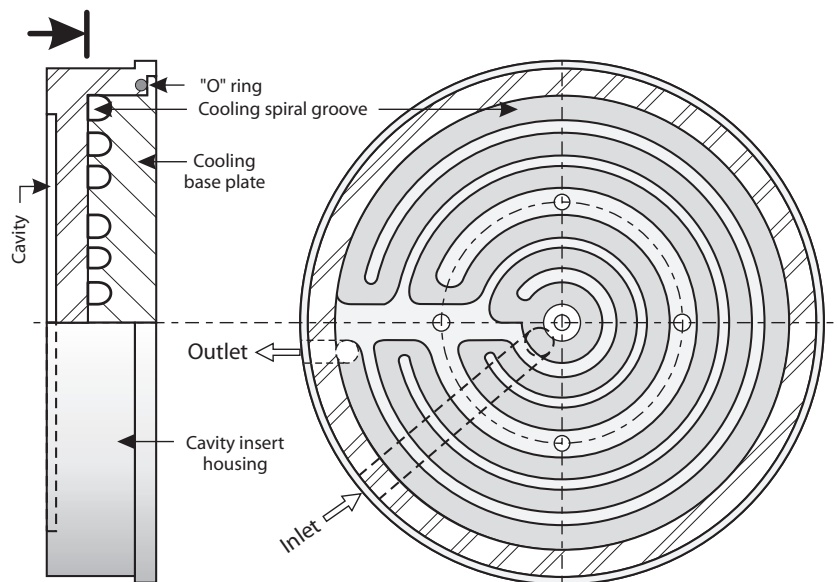


Figure 10-53 Round cavity insert and spiral groove cooling base plate

10.13.6.1 Circular Cavity Insert Cooling

Figure 10-54 shows cavity insert cooling by circulating cooling fluid around the insert. The cooling fluid is prevented from leaking by using "O" rings. The bottom illustration shows an incorrect method of applying the "O" rings. When the cavity insert is mounted into the mold cavity plate, the chances of tearing or damaging the "O" rings are high. This installation problem does not show until the mold is in the machine. The top illustration shows the preferred method of placing the "O" rings in compression; this method makes it very difficult to damage the "O" rings.

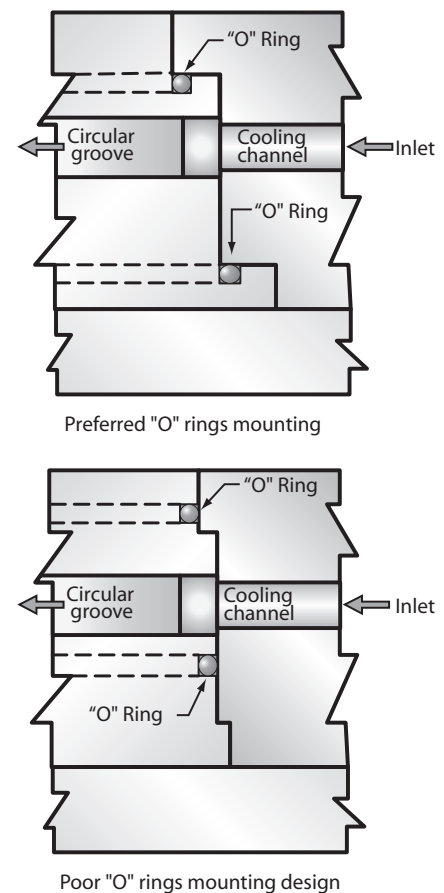
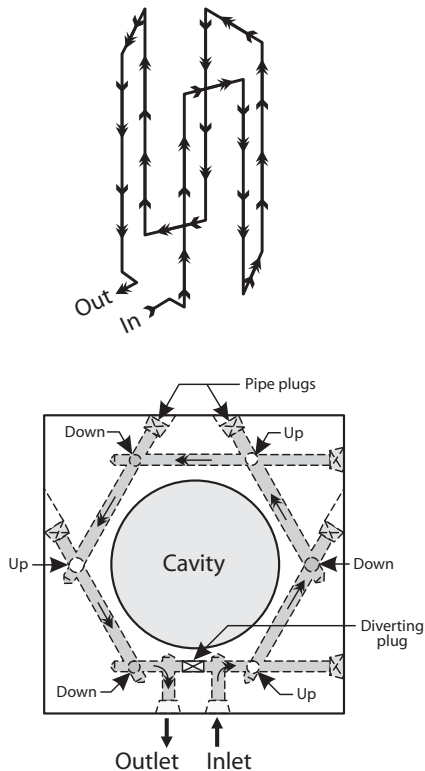
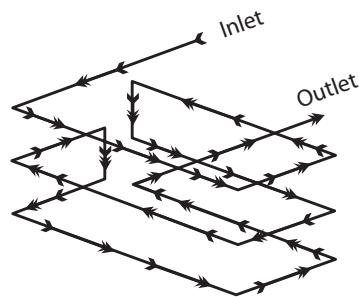


Figure 10-54 Circular cavity insert cooling, "O" ring placement

Schematic pattern cooling flow

**Figure 10-55** Round, deep cavity insert cooling channel system

Schematic pattern cooling flow

Figure 10-56 Deep cavity insert with three levels of channel cooling**10.13.6.2 Round Deep Cavity Insert Cooling**

Figure 10-55 shows a large round and deep cavity insert cooling system using cooling channel flow patterns along the path of the deep cavity. This method offers the maximum amount of cooling and a uniform temperature for the cavity.

10.13.6.3 Rectangular Deep Cavity Insert Cooling

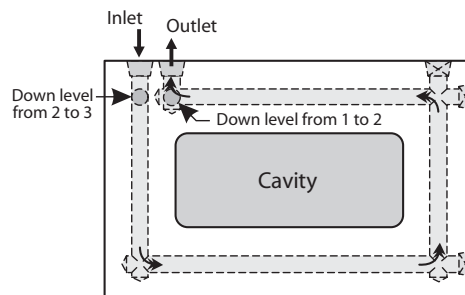
Figure 10-56 shows a large rectangular and deep cavity insert cooling system using a flow pattern with three levels of cooling channels along the path of the deep cavity. This cooling method also provides the maximum amount of cooling with a uniform temperature control for the cavity.

10.13.6.4 Core Insert Cooling Systems

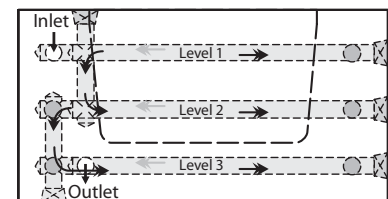
During the thermoplastic injection molding process, when the hot melt is injected into the mold cavity, it shrinks onto the cores, forming a vacuum between the melt and core surfaces, providing for direct heat transfer. The elimination of air between these joints causes the melt to transfer heat at an accelerated rate, controlling the temperature of the core. Meanwhile, the melt shrinks away from the cavity surfaces, forming an air gap or thermal insulation, in which the heat is conducted poorly, reducing heat transfer to the cavity. Accordingly, the greatest amount of heat is transmitted to the core. To offset this, the core temperature should be set lower than the cavity to compensate for the heat transfer differences between these components.

For shallow depth cores (less than 1.00 in), parallel cooling channels drilled into the core plate (single level) can be used; the cooling channels being situated beneath the core as shown in Figure 10-49. For deeper cores, however, the single level circuit is not sufficient to permit the coolant to transfer heat away from the core surface fast enough. Special arrangement must be made to permit the circulation of coolant inside the core insert.

If not enough heat is transmitted to the body of the mold and if the core is not cooled adequately by an appropriately designed channel, the temperature in the core increases as the molding production continues, causing a thermal overload. The consequences are increase in cycle time until production is stopped, and for cores with a square or rectangular cross section, the danger of side wall distortion.



Top view



Side view

Efficient core insert cooling systems normally involve passing a coolant fluid directly through channels or holes incorporated within the body of the cavity inserts. The cooling design is usually dependent on the size and shape of the insert. Only by circulating the coolant fluid deeply inside the core insert can efficient transfer of heat from the core surface be achieved. There are many alternative arrangements for cooling deep core inserts and most of these designs will be covered in the following section.

Cooling System for Small Cores (0.125 in Diameter)

Cooling of long cores with small diameters or width that are used to form blind holes, through holes, grooves, and slots is sometimes very problematical and requires appropriate design measurements of the mold.

Air may be used to cool cores with diameters or widths of 0.125 in and above. The air cools the core either from the outside during the ejection time, e.g., through appropriately placed openings in a stripper ring, or flows through a channel in the core. The disadvantage is the poor cooling efficiency.

Cores provided with a central channel are better because of the efficient cooling they provide. The air may flow during the entire time the mold is open (time for opening and closing the machine and for ejecting the part). If, however, this time is not adequate to remove the heat, air cooling may be used during the entire cycle time with an appropriate mold design (discharge of the air through holes in the stationary half of the mold).

This cooling system requires through holes or slots in the part to be molded, depending on the air pressure used (high pressure tends to produce whistling).

Significant temperature fluctuations of these core cooling systems occur at the core during a molding cycle. The ideal situation for precision injection molding with largely constant specified mold temperatures cannot be realized here. It should, however, be pointed out very clearly that these cooling systems do provide opportunities for removing the heat from thin cores reliably.

Figure 10-57 shows an air jet mold core cooling system.

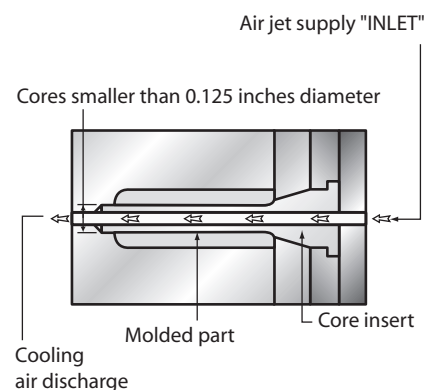


Figure 10-57 Air jet pressure core insert cooling system

Cooling Systems for Cores Greater than 0.250 in Diameter

In this case, the good thermal conductivity of copper or its alloys are used. A copper pin is press-fitted into a blind hole centrally positioned in the core. This copper pin protrudes into a cooling channel and allows the conduction of heat from the core to the cooling medium. An increase in diameter of the copper pin fit inside the core improves the heat conduction and heat transfer to the cooling fluid because of the copper pin's greater surface area.

Figure 10-58 shows a solid copper rod mold core cooling system. Figure 10-59 shows a 0.250 in diameter mold core bubbler cooling system. The core is channeled down the center (blind hole) and a straight bubbler with an outside diameter smaller than that of the channel is introduced. The temperature control medium (cooling fluid) is fed to the tip of the core through the tube and controls the core temperature during the return flow. In single cavity molds, this is likely to be the hottest part of the core insert as it is directly opposite the sprue.

While being the cheapest of the deep cooling methods to incorporate, the deep bubbler cooling design suffers from two major disadvantages:

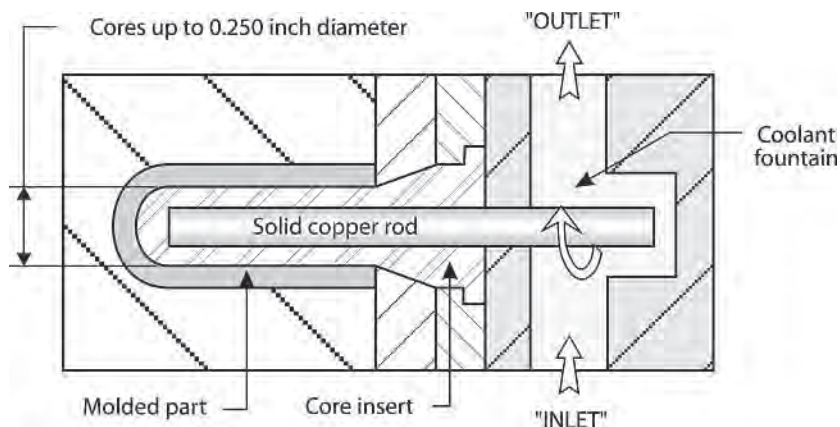


Figure 10-58 Solid copper rod core insert cooling system

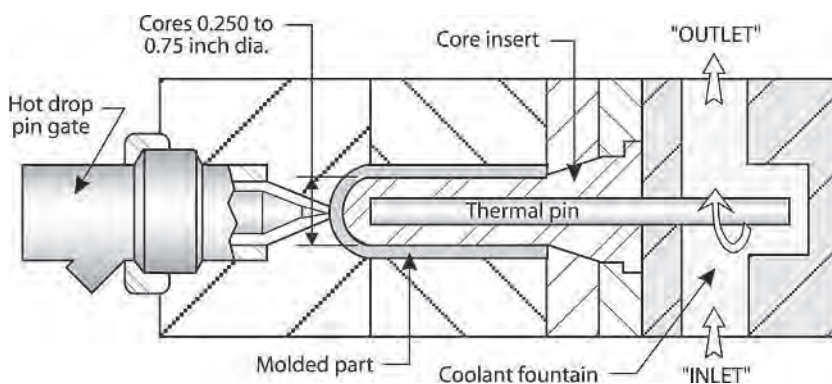


Figure 10-59 Thermal pin core insert cooling system

- The flow rate of the cooling fluid drops markedly as it enters the core insert. This means that the required turbulent flow is not achieved in this region and that the transfer of heat to the coolant is less effective.
- It is possible for an air pocket to be formed at the top of the bubbler. An uneven temperature profile, with associated molding problems, will result. It is essential that this port is always situated at the highest point of the chamber when the mold is mounted on the injection machine. It is important that molds incorporating a bubbler cooling design be engraved with information about which way the mold should be mounted on the machine.

The design can be successfully used for molds for which the production rates are not important and where mold costs must be kept to a minimum.

A more recent development is the use of thermal pins (heat pipes) instead of copper pins. Heat transport in these heat pipes is based on the phase changes (liquid/vapor and vice versa) of an encapsulated medium.

Figure 10-59 shows a thermal pin mold core cooling system.

Cascade Water Junction Cooling System

Cascade water junctions are mold temperature control systems commercially available on the market. They are compact in design, can be installed and removed in "one step", and they have 360° seals for accurate predetermination of port locations.

For cascade type mold cooling applications, the cascade water junctions provide mold cooling design flexibility and ease of operation. Their compact design makes them ideal for cooling inserted cores or spot cooling in hard to reach areas of the molds. They can be rotated a full 360° without affecting their positive seal and are easily connected and disconnected even when installed internally.

Final location of the ports on the body of the water junction can be accurately predetermined, thus assuring proper lateral alignment with pipe clearance holes. Water lines may be connected to the same side or opposing sides of the water junction. A slot on the water junction shows a body end port position and can be turned with a screwdriver to align the ports with pipe clearance holes.

The brass tube has the rigidity to maintain uniform spacing inside the water channel and is threaded into the body for firm support.

Cascade water junctions are ideal for cooling molds where cooling channels through the mold plate are not possible due to interference. The nipple type water junction provides low cost rigid installation. The 2.00 in long pipe nipple can be replaced with a longer pipe nipple to suit the application.

The socket type water junction is more easily connected and disconnected when the mold is assembled. The socket is equipped with internal Viton® seals.

Figure 10-60 shows two DME cascade water junction cooling systems.

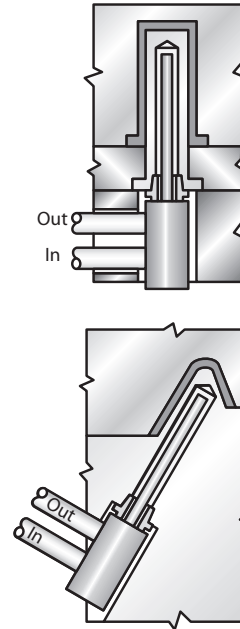


Figure 10-60 Mold cascade water junction cooling system

Cooling Systems for Cores Greater than 0.375 in Diameter

In cores with a diameter of 0.375 in and above, straight or spiral baffle cooling systems are used. Figure 10-61 shows a straight brass baffle core cooling system. Figure 10-62 shows a spiral brass plug baffle core cooling system.

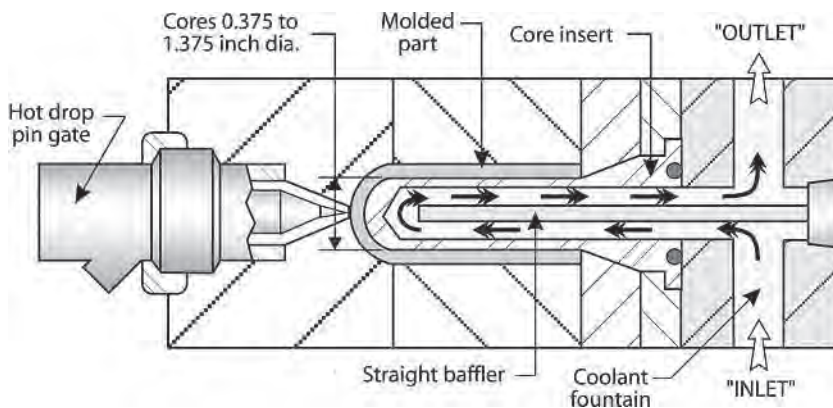


Figure 10-61 Straight brass plug baffle core insert cooling system

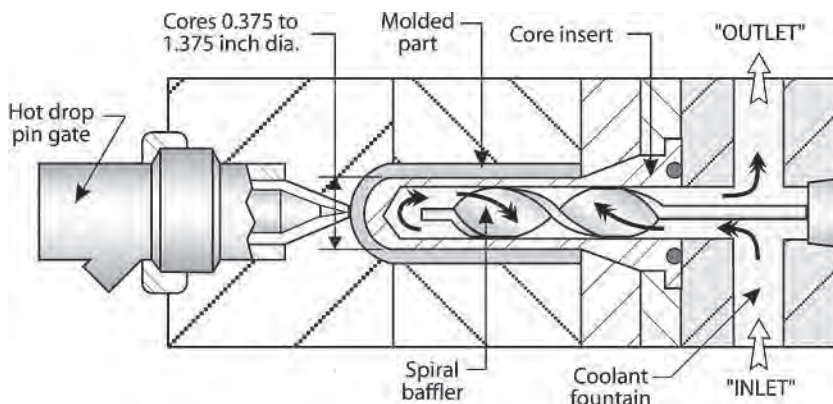


Figure 10-62 Spiral brass plug baffle core insert cooling system

The core is channeled down the center (blind hole) and a straight or spiral brass plug baffle with an outside diameter smaller than the channel is inserted. The coolant is fed to the bottom of the core, and the channel is split by the plug baffle, entering the front half and controlling the core temperature during the return flow. The result is a nonuniform temperature distribution in the core. The spiral brass plug baffle system is more efficient than a straight brass plug baffle system.

Cooling System for Cores Greater than 0.750 in Diameter

The core inserts greater than 0.750 in diameter permit the use of an outer core and an inner core with a drilled channel at the center interconnected at the top with a machined internal spiral groove. The inlet and outlet channels are drilled into the inner core, as shown by the dotted lines in Figure 10-63. The spiral creates turbulence in the flowing temperature control medium and thus improves its efficiency. The temperature control medium flows to the tip of the core in one half of the channel and returns through the internal spiral groove that is connected to the outlet channel. The core insert is fitted into a solid support and "O" rings are used to prevent leakage.

This is a very efficient design that ensures that the cooling fluid follows a precise path and no "dead cooling spots" are possible. The inner core forms a positive support to the insert outer core or shell; therefore, the wall thickness of this shell can be small. A more rapid transfer of heat from the melt is achieved as the cooling fluid passes relatively close to the mold cavity surface.

The advantage of this design is that it does not weaken the core to any major extent. Such cooling systems may be justified for large production of high quality and precision injection molding applications.

Figure 10-63 shows an internal mold core spiral cooling channel system.

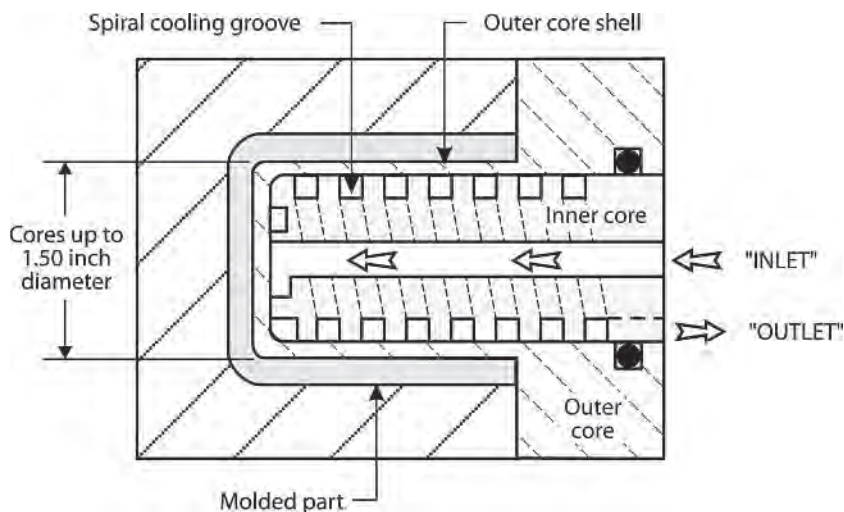


Figure 10-63 Core insert internal spiral cooling groove system

Core Cooling System for Large Double Walled Parts

With cores greater than 1.50 in diameter for an injection molded end cup with double circular walls, a single core cooling system cannot satisfy the requirements of the application. To achieve the desired requirements, two separate core cooling systems must be employed. This design basically consists of a bubbler circuit

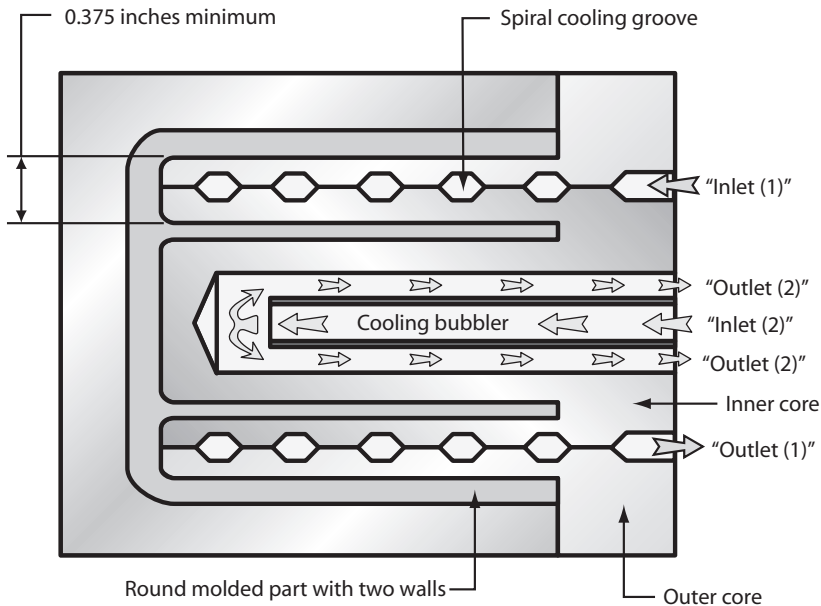


Figure 10-64 Double cooling core insert for round two walled parts

placed inside the inner core and a spiral circuit machined between the inner and outer cores. This type of core cooling satisfies the requirement for parallel flow cooling. Unfortunately, in practice this double cooling system is both difficult and expensive to produce.

Figure 10-64 shows a core insert with double internal cooling systems used for injection molding a rounded end cup with two parallel walls.

Core Valve Ejector Cooling System

Bubbler cooling systems are used to remove heat from a valve ejector mounted as an integral component of the core. The head of the valve ejector is found on the top surface of the core, opposite to the gate. The valve ejector normally forms a relatively large part of the core insert top surface. Facilitating the dissipation of heat from this component is desirable.

The stem of the valve ejector is bored to hold a bubbler system unit. The connectors are coupled to the supply and return lines via flexible hoses to allow for the ejector valve movement. The cooling fluid passes, via the inlet, down the center of the pipe, and back to the outlet via the outside of the pipe. This is the simplest method of cooling the valve type ejector, particularly if a commercial water junction unit is used. If passing the coolant through the head of the valve ejector is necessary, a more complex circuit is needed.

The support feature is provided by a central column that can be integrated with the inner core, as a separate cooling circuit. Obviously, this is needed if the depth of the bubbler requires a column that is relatively long.

The design has two primary objectives:

- To support the central region of the core against possible deflection.
- To have a solid central region of the core insert to permit the valve ejector to be incorporated. An additional “O” ring must be incorporated to avoid cooling fluid leakage through the core assembly.

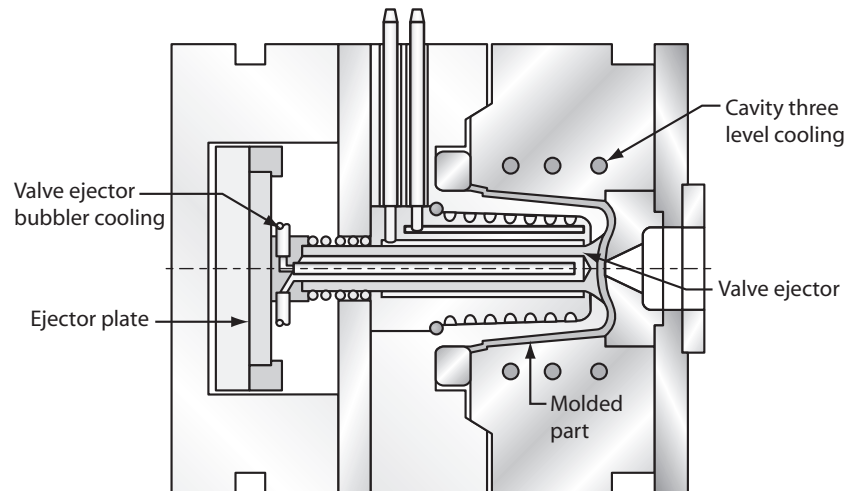


Figure 10-65 Mold cavity, core insert, and valve ejector cooling

The disadvantage of the bubbler cooling system is the flow rate drop as it enters the annulus with possible air pockets being formed. To overcome this cooling problem, it is recommended to incorporate a large diameter ejector valve and bubbler system, having its own coolant circuit in the mold design. This will ensure that an efficient coolant circulation system is used at the critical area where it is required; that is, at the hottest location of the core insert, namely the ejector valve head surface where the gate is located.

Figure 10-65 shows a cross section view of a valve ejector bubbler cooling system, core insert, and cavity cooling.

Mold Cooling Summary

These basic principles lead to very simple procedures for designing efficient mold cooling systems.

- Place the cooling channels at a proper distance from the mold cavity surface
- The minimum cooling channel center distance to be three to four diameters in length
- Do not use small diameters for cooling channels, specify 0.4375 in diameter minimum
- Cool directly in the cavity and core inserts
- Cool auxiliary mold plates if required
- Drill through cooling channels and use pipe plugs
- Be sure that the Reynolds flow will occur throughout the mold. The Reynolds number should not be less than 3,500
- Treat the cooling channels with electroless nickel or equivalent. If not possible, clean them regularly
- The cooling fluid should be a blend of distilled water and special corrosion resistant additive (25% minimum to 65% maximum) so that no scale deposits are found in the cooling channels

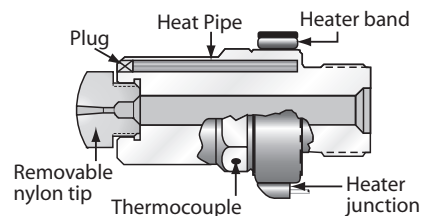
- Use a portable digital thermometer with a surface and needle adapter to measure the cavity surface and process melt temperature during molding without interruptions to the process
- Hook up the cooling fluid on the mold so that it must fill all the cooling channels before it over-flows. Adjust the cooling fluid temperature and flow rate on the mold cooling system
- Vary the cooling flow rate, the mold temperature, parallel or series hookups, and direct or counter flow
- Take samples under each condition, test them and make the best compromise between economics and quality
- Record this information for future runs.

10.14 Injection Molding Machine Nozzle

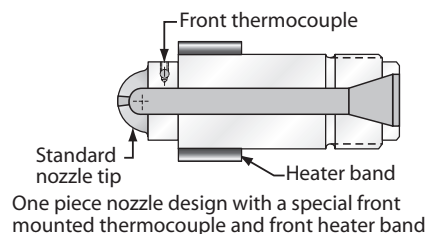
The nozzle that screws into the barrel end cap is a tubular shaped component of various lengths and inside diameters, long enough to reach the mold. It is intended to bridge from the injection unit to the mold, providing passage for the melt through the stationary platen to the mold's sprue. It commonly consists of a one-piece unit or removable separate nozzle tip, which screws into the main body of the nozzle. The main nozzle body extends from the barrel end cap to the sprue, using an independent heater band and a thermocouple at the rear end. The nozzle tip provides the final transition from the larger diameter of the nozzle body to the sprue bushing inlet.

With a cold sprue, the outlet orifice of the nozzle tip is slightly smaller than the inlet orifice of the sprue. This ensures that the frozen sprue can be ejected. A removable nozzle tip is desirable for changing molds with different sprues and molding different plastic materials. The orifice of the nozzle tip can also be designed for various materials and processes. However, these standard nozzles are not designed to be temperature-control devices.

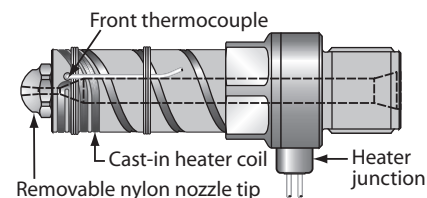
Figure 10-66 shows three nozzle designs. The Kona nozzle uses four heat pipes mounted around the entire streamlined flow channel of the nozzle. The heat is drawn from the barrel and its own heater band, using a thermocouple for uniform heat distribution of the nozzle. The one piece nozzle design has a front-mounted thermocouple and heater band. The Mold Master injection nozzle has a front mounted thermocouple and strategically distributed heaters that have been cast around the nozzle body providing precise and uniform internal temperature control through the entire flow channel, eliminating internal hold up spots.



Kona nozzle with heat pipes, heater and thermocouple



One piece nozzle design with a special front mounted thermocouple and front heater band



Mold master injection nozzle with a fused heater coil strategically distributed with front thermocouple

Figure 10-66 Injection nozzle designs with better temperature controls

10.14.1 Mold Cold Runner System

The cold runner system transfers the thermoplastic melt from the injection nozzle to the cavities. It consists of sprue, sprue puller, runner, cold slug pockets, and gates. All these components will be discussed in detail in this chapter.

10.14.1.1 Cold Runner Sprue

The sprue transfers polymer melt and heat from the molding machine nozzle to either a runner system or directly to a cavity. A cold sprue is formed inside a sprue bushing, which must withstand repeated impacts from the injection nozzle each time it engages the sprue bushing. The sprue bushing must have good wear and corrosion resistance, a good heat transfer coefficient, surface finish, and heat resistance. Most sprue bushings are made from SAE 6145 hardened tool steel.

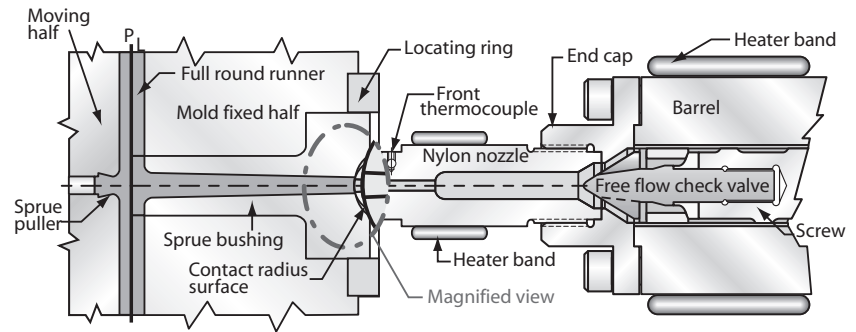


Figure 10-67 Contact radius surfaces between injection nozzle tip and sprue bushing

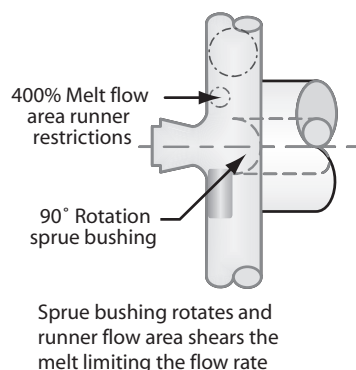
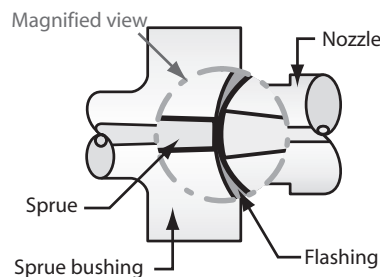
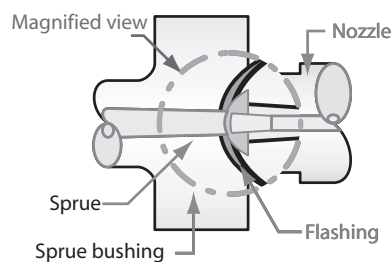


Figure 10-68 Interface problems between injection nozzle and standard sprue bushing

The contact radius surface between the injection nozzle and the sprue bushing helps with alignment of the flow channels between these components. If there were any misalignment, an undercut would be created and inhibit the sprue from being pulled from the bushing. The radius of the sprue bushing should be slightly larger than the injection nozzle tip to ensure sufficient sealing without flashing. However, if either contact radius surface of these components is in poor condition, flashing will retain the sprue and manual removal will be necessary. The sprue bushing "O" diameter should be a minimum of 0.031 in larger than the injection nozzle tip orifice diameter to avoid molding cycle interruptions to free the sprue from the bushing. When the mold opens it is essential that the sprue breaks away from the injection nozzle and is pulled completely from the sprue bushing. With single-cavity molds, the sprue feeds directly into the base of the molded part and the sprue is pulled while the molded part is ejected from the cavity.

Figure 10-67 shows the interface between the contact radius surfaces of the sprue bushing and one piece injection nozzle design.

Figure 10-68 shows three common problems between these components. In the top illustration a lack of interface contact radius surfaces between the sprue bushing and the injection nozzle tip causes flashing problems. The middle illustration shows a larger diameter for the nozzle contact orifice than the sprue "O" diameter, which causes sprue undercuts and flashing that also contribute to ejection problems of the sprue.

The main runner is machined in the base of the sprue bushing, linking the sprue to the runner system. There is rarely any positive anti-rotation provided on the sprue bushing to ensure runner alignment. This commonly results in these two sections of the runner becoming misaligned, causing melt flow restriction during the production run, as shown in the bottom illustration. A retaining dowel pin located in the shoulder of the sprue bushing ensures runner alignment.

The interface between the sprue bushing and the nozzle must not be deformed as flashing or ejection problems might occur.

The temperature controls or thermal interfaces between the injection nozzle and sprue bushing contact surface radius is one of the major problem areas in the injection molding process. The machine's injection unit plasticizes and delivers the polymer melt at the correct process viscosity, temperature, and injection pressure to the injection nozzle to fill the mold. Standard sprue bushings and nozzles have difficulties maintaining a uniform melt viscosity, if only one heater band in the middle and a thermocouple at rear of the nozzle are used.

The temperature of the injection nozzle tip is the most critical and difficult to control. If either the nozzle or the sprue bushing is not thermally insulated, heat is dissipated from the nozzle tip to the atmosphere when idle, or through the sprue bushing, while the sprue bushing becomes colder because the heat is absorbed by the mold base and machine fixed platen.

Injection nozzles with removable tips (different heat transfer coefficient) have orifices of different diameters, designs, and volumes. Each section requires various heat levels, which one heat source cannot deliver.

The lack of positive contact surface seals, alignment and temperature control between these components is an injection molding process problem. The injection nozzle provides a force to seal with the sprue bushing. This interaction causes heat control problems and cooling of the melt as it enters the mold.

Following are some guidelines when designing or specifying a sprue:

- The sprue must not freeze before the runner system and cavities. This is necessary to complete the molding process screw forward time.
- The sprue must be ejected easily and reliably.
- The sprue interfaces with the injection machine nozzle must not have any flash to avoid sprue ejection problems.
- At the base of the main runner, in line with the sprue, a sprue puller pocket should be provided to act as a cold slug well.
- Sprue dimensions should be calculated based on the sprue length and total shot size.
- The outlet sprue diameter should be at least 0.062 in larger than the molded part wall thickness, depending on the sprue length, shot size, polymer crystallinity, and cold runner system total pressure drop.
- The sprue bushing "O" diameter should be at least 0.031 in larger than the outlet orifice of the injection nozzle.
- The sprue bushing inside diameter must be tapered to 2.38° per side or 0.50 in/ft. The tapered surface reamed and hardened for the sprue to be pulled out of the orifice when the mold is opened.
- Sprue bushing radii should be modified, slightly increasing the outside edges to reduce the contact radius with the nozzle face for a better alignment and seal surface.
- The sprue bushing should have a dowel pin positioned to prevent rotation and misalignment with the main runner.
- Reduce the sprue length to a minimum to control the molding cycle.

Standard Sprue Bushings

Standard sprue bushings are most commonly used in the construction of thermoplastic injection molds. They are commercially available in various designs and sizes. These standard sprue bushings are used for simple 2- and 3-plate mold designs and for injection molding commodity thermoplastics of low or medium crystallinity rates, good thermal stability, and low melt process temperatures.

Figure 10-69 shows a typical standard commercially available sprue bushing.

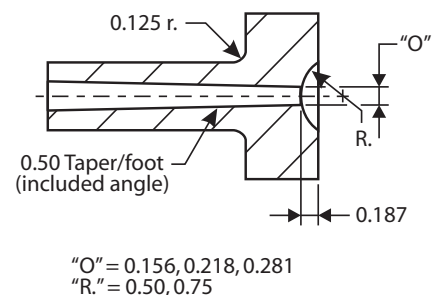


Figure 10-69 Standard commercially available sprue bushing

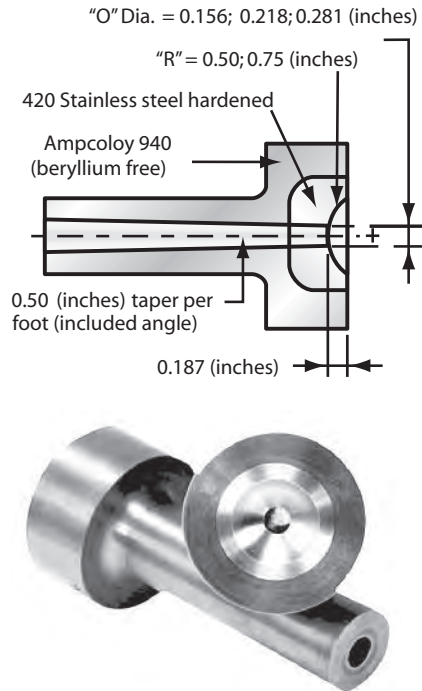


Figure 10-70 Performance alloy sprue bushing to improve temperature control

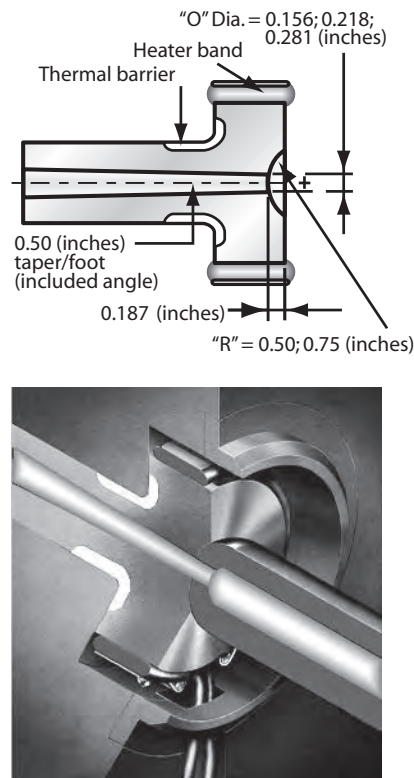


Figure 10-71 Tranzisprue, a sprue bushing with temperature control

Performance Alloy Sprue Bushings

Figure 10-70 shows a new type of sprue bushing commercially available in designs and dimensions found in standard sprue bushings, but with better temperature control and performance required for more critical molding applications. These sprue bushings are made of Ampcoloy 940 (high thermal conductivity alloy) to improve heat transfer; however, this alloy is very soft. To improve the toughness required at a contact surface radius, an insert made of 420 stainless steel hardened is fused inside the alloy body.

TranziSprue™

Engineering thermoplastics for high melt process temperatures with high melting point temperatures, fast crystallization rates, requiring high mold temperatures, and using fast molding cycles will cause the sprue bushings to show poor performance.

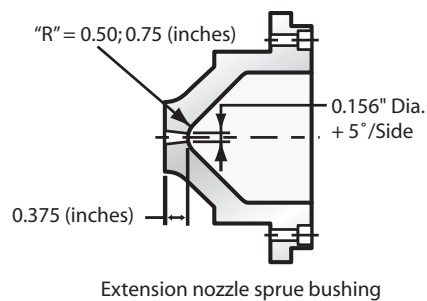
A new type of sprue bushing with the same standard dimensions, reasonably priced, has shown excellent performance when molding engineering polymers. Figure 10-71 shows the Tranzisprue™ design. It uses a heater band and thermocouple to control the contact surface radius temperature and insulates the sprue from the mold base plate by using thermal barriers.

Extension Nozzle Sprue Bushings

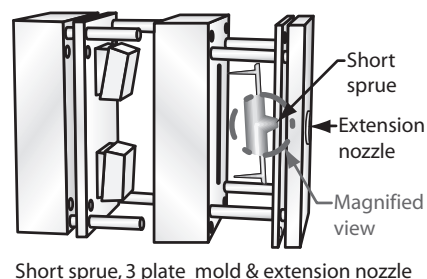
Extension nozzle sprue bushings are used to reduce the length of the sprue (economical reasons), improve the cold runner system melt injection efficiency, reduce the cold runner system pressure drop, and lower the injection pressure required for the molding process.

Variations of the extension nozzle sprue bushings are used in horizontal 2-plate molds feeding directly to a single cavity or into a runner feeding several cavities in 3-plate molds and in vertical inserts encapsulation molds.

Figure 10-72, top shows a commercially available extension nozzle sprue bushing, the bottom illustration shows an extension nozzle sprue bushing mounted in a 3-plate mold, reducing the length of the sprue.



Extension nozzle sprue bushing



Short sprue, 3 plate mold & extension nozzle

Figure 10-72 Extension nozzle sprue bushing and short sprue mold

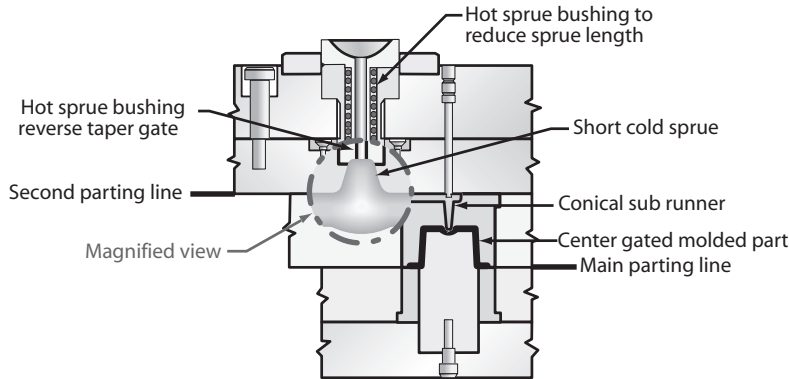


Figure 10-73 Short sprue in a 3-plate mold, using a hot sprue bushing

Hot Sprue Bushing

A hot sprue bushing is an alternative to an extension nozzle sprue bushing. The hot sprue bushing directs the melt from the injection molding machine nozzle directly into a single cavity or into a cold runner feeding either multiple cavities or multiple gates feeding a single cavity.

In applications where a long sprue and large diameter control the molding cycle (prolonged cycle time), the hot sprue bushing provides an excellent alternative. The hot sprue bushing with a good gate design (reversed taper) has few of the drawbacks of a hot runner system. They are relatively inexpensive, easy to install, operate, and maintain. They require only one additional temperature control.

Figure 10-73, shows a hot sprue bushing application used to reduce the length of a cold sprue in a 3-plate mold.

How to Size the Sprue

Knowing the shot weight passing through the sprue and the length of the sprue bushing, the sprue size is calculated using Eq. 10-22.

$$D_{\text{Avg.}} = \frac{W^{0.5} \times L^{0.25}}{16} = \frac{\sqrt{W} \times \sqrt[4]{L}}{16} \quad (10-22)$$

Where:

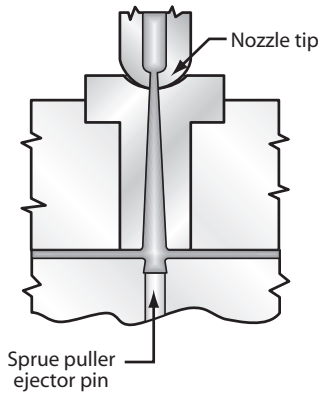
- D_{Avg} = Average diameter of taper sprue (in)
= Average inlet/outlet sprue diameters to the nearest 0.06 in
- W = Total shot weight (oz)
- L = Sprue length (in)

Example 10-6

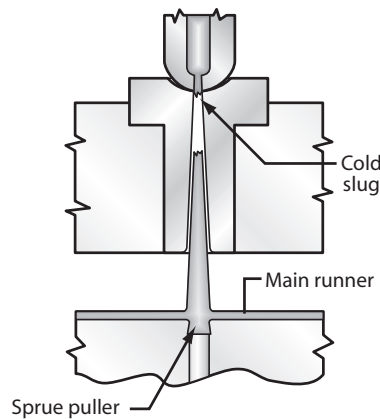
Determine the thermoplastic sprue diameters for a total shot weight of 4.0 oz and a sprue length of 4.00 in.

$$D_{\text{Avg.}} = \frac{4^{0.5} \times 4^{0.25}}{16} = \frac{2 \times 1.414}{16} = \frac{2.828}{16} \approx 0.187 \text{ in Dia.}$$

Taper sprue diameters: $D_{\text{inlet}} = 0.125 \text{ in}$, $D_{\text{outlet}} = 0.25 \text{ in}$



Mold closed, nozzle injects melt inside sprue bushing, sprue puller pocket and runner



Mold opens and breaks the sprue from the nozzle tip by action of sprue puller undercut

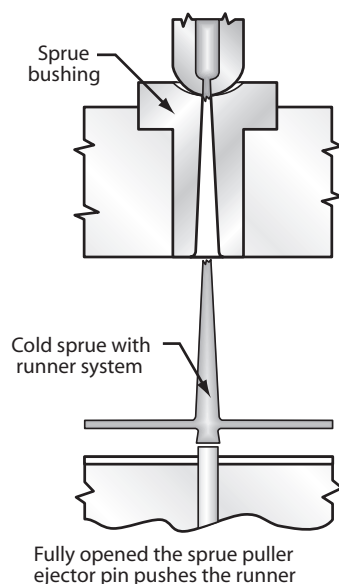


Figure 10-74 Ejection sequence of the cold sprue and runner system

10.14.1.2 Sprue Puller

The cold slug well or sprue puller pocket is one of the most effective traps for the cold slugs or colder melt front. When the melt stops flowing, the heat transfer is reduced, the tip of the nozzle becomes colder and a cold slug is formed inside the injection nozzle's front orifice. Its function is to stop the flow of the melt front at the nozzle each time the mold opens and the cold sprue breaks from the nozzle tip.

During mold opening, the cold sprue is pulled from the sprue bushing by an undercut in the sprue puller pocket. The sprue puller is an automatic ejection device that functions in coordination with the sprue puller ejector pin. When the ejector pin moves forward, it pushes the cold slug well, forcing the sprue out of the undercut allowing the sprue and runner to fall freely.

Figure 10-74 illustrates the sprue puller operational sequence.

The size of the sprue puller is often the determining factor that controls the length of the molding cycle. A big sprue puller requires a longer cooling time to cool off and become rigid to eject the sprue from the sprue bushing without breaking. If the sprue puller's large volume (mold cooling is not provided in this area) is not frozen sufficiently at the time of mold opening, it will break and the sprue will remain lodged in the sprue bushing. To avoid this, the molding process cycle must be longer to allow additional cooling time for the sprue puller.

When designing cold runner molds, the ejection of the sprue and the sprue puller runner system should be analyzed to make the best selection to ensure that the runner system does not stick to the fixed half of the mold, causing interruptions in the molding process. The sprue pullers are found directly on the opposite side of the sprue entry and on the lower side of the main runner. The thermoplastics melt from the nozzle flows through the sprue bushing, the front cold slug fills the sprue puller pocket, and upon solidifying, provides sufficient adhesion to pull the sprue away from the nozzle and the sprue bushing as the mold is opened.

There are many techniques and designs of sprue pullers; however, only the recommended sprue puller designs that have shown excellent performance and reliability in the field will be discussed here.

Figure 10-75 shows three small and efficient sprue puller designs recommended for controlling cooling problems and to reduce the injection molding process cycle time.

Illustration (a) shows two sizes of reverse taper sprue pullers (based on mold size). It is the most common design used for molding both high and low melt temperature materials, unreinforced, impact modified, fiber glass, and mineral reinforced resins.

Illustration (b) shows the "Z" type sprue puller. It is used in similar applications as the reverse taper. It is not recommended with brittle materials. The internal radius in the ejector pin should be smooth and polished. The ejector pin should not rotate to avoid hang-ups of the runner and the ejector pin. The ejector pin has a short operational life (breakage at the thin cross section).

Illustration (c) shows two sizes of annular ring pullers (based on mold size); this design is recommended only for unreinforced resins.

Designing the optimum runner layout with exact dimensions will maximize savings of both raw materials and energy consumption during the injection

molding process. At the same time, runner size is constrained by the amount of pressure drop and the injection capacity of the machine. This mold design section presents some simple procedures to follow in minimizing runner size, maximizing productivity, and molded part quality.

Cold runner systems consume material and energy. Since molding a runner system obviously costs money, it only makes sense to minimize the amount of unusable material molded into the cold runner system. Even though the cold runner will probably be reground and recycled, it is important to keep its weight and size to the absolute minimum because some thermoplastic polymers tend to degrade during repetitive processing.

It is necessary to provide a flow-way in the injection mold to transfer the melt from the injection machine nozzle to each mold cavity. This flow-way is referred to as the cold runner system. Normally the runner comprises a sprue, runner, and gate. These terms apply equally to the molded parts that are removed from the runner and gates in the process of extracting the molded shot.

A typical full shot for four cavities and runner is shown in Figure 10-76. The thermoplastic melt passes through the sprue, main runner, branch runners, and gate before entering the cavity. It is desirable to keep the traveling distance of the material to a minimum to reduce pressure and heat losses. Therefore, careful consideration must be given to the mold cavity layout.

The purpose of the runner cold slug pockets is to catch the melt that has chilled at the front of the nozzle.

The runner is a channel machined into the mold cavity plate surface (parting line) to connect the sprue with the entrance (gate) to the cavity.

There are two key functions of a runner that always must be thought out carefully during the mold design stage. The first obvious function of the cold runner is to convey melt into individual mold cavities. In this respect, the runner works like a fluid transfer line and the principles of flow govern the design of the runner. The second, often less obvious function of the cold runner system, is a mechanical one. For instance, in a three-plate mold, sucker pins must be designed so that the runner is retained on the front plate and is stripped off the sucker pins as the mold opens. Design for mechanical operation of the mold should also allow the runners to fall free if automatic operation is desired.

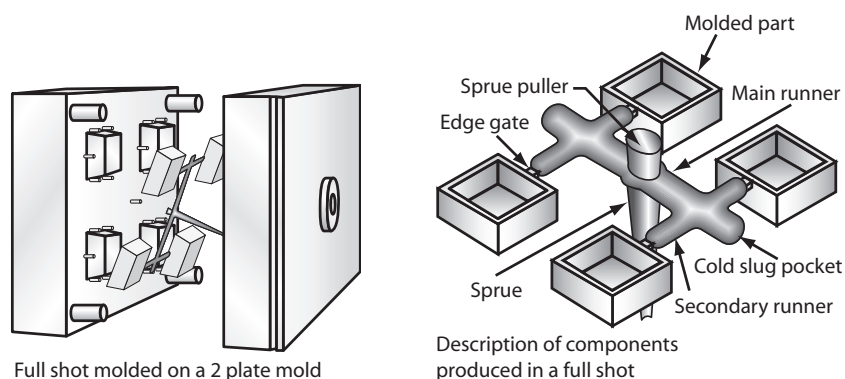
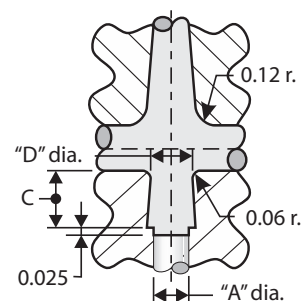
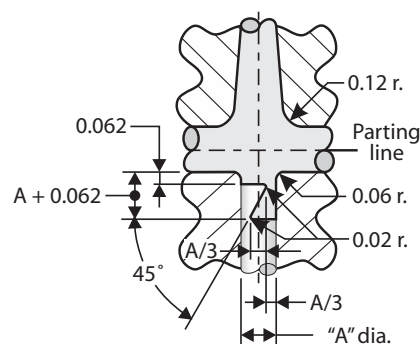


Figure 10-76 Illustration of molded components in a complete shot

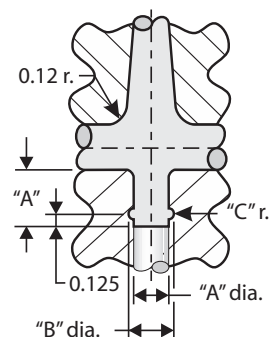


A	B	C	D
0.250	0.276	0.250	0.250
0.375	0.414	0.375	0.375

Reverse taper



"Z" type sprue puller



A	B	C
0.250	0.258	0.004
0.375	0.386	0.006

Annular ring

Figure 10-75 Recommended sprue puller designs

If additional handling of a cold runner system is anticipated, such as breaking the gate from the molded parts, the runner diameter connected to the gate should be large enough to provide the rigidity to conform to the degating requirements. In such a case, sizing a runner to the absolute minimum flow requirement, otherwise highly desirable, might be shortsighted from a material handling point of view. These mechanical functions of the cold runner system are important and can often determine the practicality of automatic operation or how well a mold functions.

The surface walls of the runner channel must be smooth to prevent any restriction to flow. Also, as the runner has to be removed with the molded part, there must be no machine marks left which would tend to retain the runner in the mold plate channels. The runner channels should be polished and hardened in the flow direction of the melt.

Important considerations for designing the cold runner system:

- Cold runner system layout
- Cold runner cross section geometry
- Cold runner dimensions.

Cold Runner System Layout

The layout of the runner system will depend on the following factors:

- The number of cavities
- The geometry of the molded parts
- The type of mold (i.e., two-plate or three-plate mold)
- The type, geometry, and size of the gates.

Cold Runner Cross Section Geometries

The cold runner cross section geometries used in mold designs are usually one of four configurations, as shown in Figure 10-77 (a full round, a parabolic, a trapezoidal, and a half round cold runner).

While the half round runners were prevalent in the earlier days of injection molding, they are rarely encountered today. The half round runner design is the most inefficient of the four designs in delivering a melt, because its ratio of pressure loss to runner unit volume is very high.

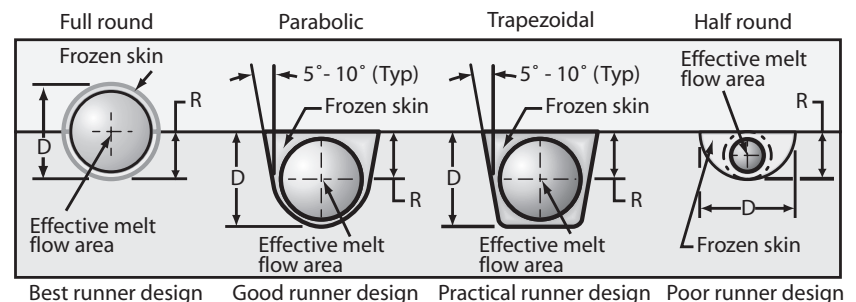


Figure 10-77 Typical cold runner cross section geometries

The full round, parabolic, and trapezoidal designs are the most efficient. Each offers distinct advantages for specific situations. The parabolic and trapezoidal runners are machined only on one cavity plate, the side where the ejector system is located and where the runner must be retained as the mold opens. It is then stripped away during part ejection. The trapezoidal runner is commonly used with three-plate molds. The full round cold runner can be machined into both cavity plate surfaces, where the parting line is located and there are no mechanical obstacles. A full round runner offers the lowest pressure loss per unit volume of all designs.

The criteria for efficient cold runner design are that the runner should provide a maximum cross section area from the standpoint of pressure transfer and minimum contact on the periphery from the standpoint of heat transfer.

As the thermoplastic melt progresses through the mold's cold runner system, the melt adjacent to the cold mold surface rapidly decreases in temperature, forming an external solidified skin. The melt that follows will pass through the center of this solidified material and because of the low thermal conductivity that most thermoplastics possess, the solidified skin acts as insulation and keeps the temperature of the melt's central core hot, allowing the internal melt to flow.

The cross section area of the runner must be sufficient to permit the melt to pass through to fill the cavity and for packing pressure to be applied for shrinkage control of the molded part before the runner freezes. The cross section area of the runner should not be able to control the injection cycle.

The main objection to the full round runner is that this runner is formed from two semicircular channels machined separately in each of the mold plates. It is essential that these channels are accurately matched to prevent the development of an undesirable and inefficient runner system. Accurate matching of the channels means that the mold cost for a mold containing round runners will be higher than for one containing trapezoidal runners.

Cold Runner Dimensions

Considerations required for specifying the cold runner dimensions:

- The wall thickness and volume of the molded part
- The distance of the cavity from the main runner or sprue
- The mold cooling system for the runner and gates
- Type of cold runner cross section design
- The thermoplastic melt flow rate
- The thermoplastic viscosity and shear rate characteristics

The runner length should always be kept to a minimum to reduce pressure losses and the cold runner system should be balanced. Runner balancing means that the distance, the volume, and heat transfer characteristics should be identical for each channel. When the thermoplastic melt travels from the sprue to each cavity, it ensures that all the cavities will fill uniformly and without interruptions.

Figure 10-78 shows the difference between balanced and unbalanced cold runner systems.

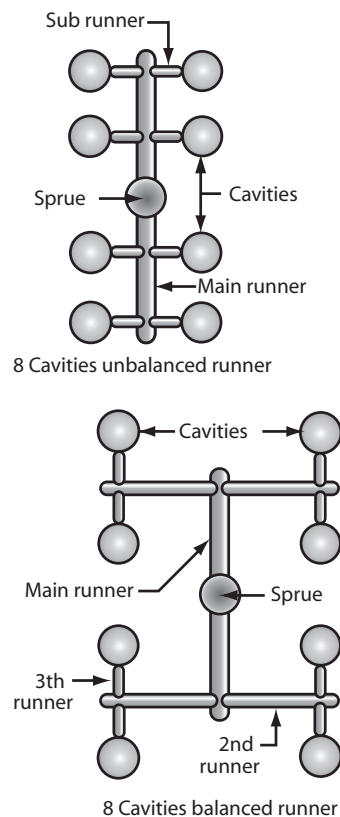
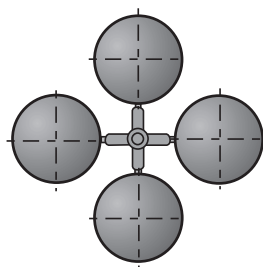
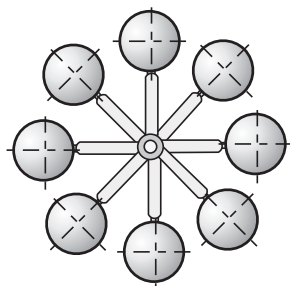


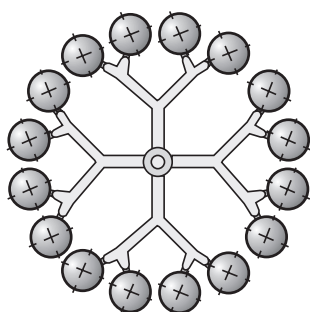
Figure 10-78 Difference between balanced and unbalanced runners



4 Cavities balanced runner



8 Cavities balanced runner



16 Cavities balanced runner

Figure 10-79 Three different balanced runner configurations

Figure 10-79 shows three different balanced runner configurations that provide uniform injection pressure to all cavities.

There have been a number of misconceptions about proper cold runner design. In the past, many injection molders and tool builders felt that the larger the runner, the faster the melt would be conveyed to the cavity. They also believed that the lowest possible pressure loss through the runner's system to the cavity would be the most desirable. There are now techniques for computing the minimum runner size required to convey the melt at the proper rate and pressure loss to achieve optimum molding quality.

10.14.1.3 How to Design a Cold Runner System

The first step is to examine the mold layout and assign the cavity spacing. Always, the distance between cavities should be as short as possible. The second step is to figure out the gate locations, which depend on part design and performance criteria. In striving for a balanced cavity layout, most mold designers prefer 2, 4, 8, 16, 32, or 64 cavities. Sometimes, having balanced cavities is not possible, especially when the goal is to put as many cavities into the mold as possible to reduce molding costs.

If good general design principles have been used for determining runner length and cavity spacing, finding the most efficient runner size using a relatively simple set of calculations is now possible.

Computing Runner Diameters

These computations are based on a key rheological property of the thermoplastic resin to be molded. This property is the material's shear rate vs. its melt viscosity at several commonly encountered melt temperatures for that material. Usually, this information is available from the resin supplier and is frequently displayed in molding manuals for individual materials.

Since no single calculation will do the job, starting with a reasonable runner size estimate is necessary, based on a balanced melt distribution through the runner. In fact, we must select these diameters based on flow and equal volume distribution to all cavities. The melt flow must be proportionally distributed as shown in Table 10-7:

Table 10-7 Balanced Cold Runner Melt Flow Distribution

Sprue	100% Flow
Main runner	50% Flow
2nd Runner	25% Flow
3th Runner	12.5% Flow
4th Runner	6.25% Flow
5th Runner	3.12% Flow

Table 10-8 was developed considering the balanced cold runner configuration for 64 cavities as shown in Figure 10-80, equal flow distribution, and the most popular cutter sizes used by the mold makers.

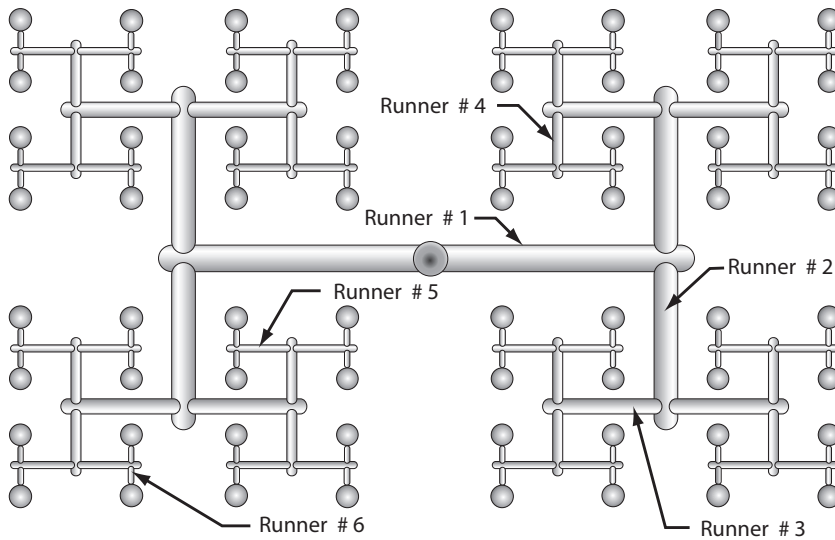


Figure 10-80 64-cavity balanced runner (reference for Table 10-8)

Table 10-8 Equal Flow Runner Diameter (in)

2 Cavities	4 Cavities	8 Cavities	16 Cavities	32 Cavities	64 Cavities
Runner #1	Runner #2	Runner #3	Runner #4	Runner #5	Runner #6
0.375 Dia.	0.312 Dia.	0.281 Dia.	0.250 Dia.	0.187 Dia.	0.156 Dia.
0.312 Dia.	0.281 Dia.	0.218 Dia.	0.187 Dia.	0.156 Dia.	0.131 Dia.
0.281 Dia.	0.250 Dia.	0.218 Dia.	0.187 Dia.	0.156 Dia.	0.125 Dia.
0.250 Dia.	0.218 Dia.	0.187 Dia.	0.156 Dia.	0.125 Dia.	0.109 Dia.
0.218 Dia.	0.187 Dia.	0.156 Dia.	0.125 Dia.	0.125 Dia.	0.093 Dia.
0.187 Dia.	0.156 Dia.	0.131 Dia.	0.125 Dia.	0.093 Dia.	0.078 Dia.
0.156 Dia.	0.131 Dia.	0.125 Dia.	0.093 Dia.	0.078 Dia.	
0.131 Dia.	0.125 Dia.	0.093 Dia.			
0.125 Dia.	0.125 Dia.	0.093 Dia.			
0.093 Dia.					

Gate Runner Larger than Part Wall Thickness

The last runner connected to the gate and cavity should be 0.040 in larger than the wall thickness of the molded part. Figure 10-81 shows the cross section of the molded part, edge gate, and the last runner diameter in relation to the part wall thickness.

With the calculated diameter from Table 10-8, select the column for the number of cavities, select the row closest to the diameter required for the application, and use the other recommended runner diameters for the preliminary pressure drop calculations. A minimum runner diameter of 0.078 in should be used to prevent premature runner freeze-off and high shear rate conditions.

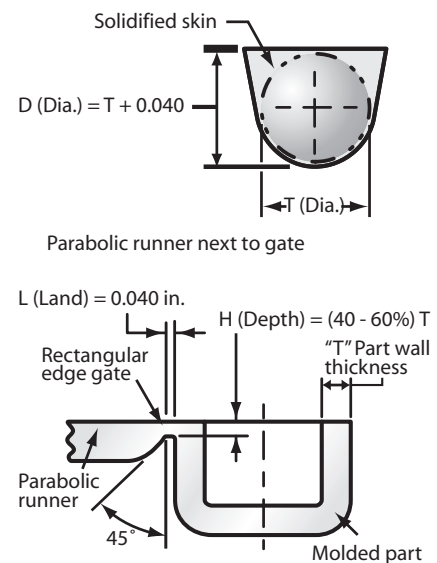


Figure 10-81 Runner diameter next to gate larger than part wall thickness

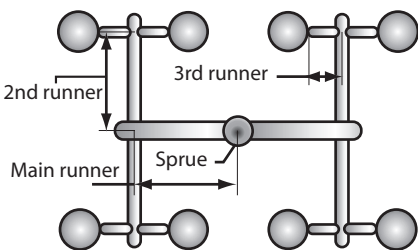


Figure 10-82 Eight-cavity balanced cold runner analysis

Example 10-7

Calculate the optimum runner size of an eight-cavity mold as shown in Figure 10-82, the molded part is made of unreinforced nylon 6/6, with a specific gravity of 1.00, weighing 15.0 grams; an injection speed of 3.0 seconds is used.

Using a tertiary runner diameter of 0.125 in to satisfy the molded part wall thickness, based on Table 10-8 for eight cavities and a tertiary runner diameter of 0.125 in, a secondary runner diameter of 0.131 in, and a main runner diameter of 0.156 in are needed.

With the lengths of the runners as shown in the table below, we can now compute the total volume of the melt that must be conveyed through each runner segment. Since this is a balanced mold, we need to calculate only half of the mold.

Table 10-9 Runner Dimension Analysis

	Length (in)	Original diameter (in)	Modified diameter (in)
Main runner	5.00	0.156	0.140
2nd Runner	3.00	0.131	0.131
3rd Runner	1.00	0.125	0.125
Total runner weight (g)		7.37	6.78
Runner pressure drop (psi)		5,785	6,036

Since the molded parts weigh 15.0 g each, we must convey through each half of the primary runner the weight of four parts plus the weight of the runner’s system itself. Here, we must convey 60.0 g (4 × 15) of material plus the volume of the runner’s system through the left side of the mold.

In this example, we assume that we are using a material with a specific gravity of 1.0. If the volume of half the runner’s system is “X”, the primary runner must carry 60 + “X” g of material within 3.0 s, giving us a flow rate (Q) of (60 + “X”) / 3.0 in³/s.

Because the melt viscosity depends on shear rate and temperature, the only remaining data we need are the melt temperature and the viscosity vs. shear rate graph of unreinforced nylon 6/6. Using the graph shown in Figure 10-83, we select the melt viscosity of the material at the calculated shear rate.

With the above information, we can now determine the shear stress on the resin in the runner’s system: shear stress equals melt viscosity times shear rate. Finally, we can compute the pressure drop through the primary runner; pressure loss being the key factor in determining an optimum runner diameter. This procedure can be used to calculate pressure drops occurring in the secondary and tertiary runners as well.

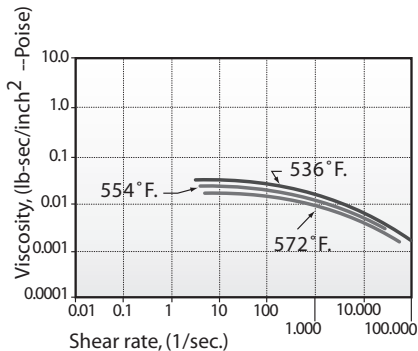


Figure 10-83 Viscosity vs. shear rate for unreinforced nylon 6/6 (Courtesy: Du Pont)

How to Calculate the Pressure Drop in the Runner

Engineering a cold runner system requires an understanding of the pressure drop of the polymer melt as it passes through a channel. This pressure drop is controlled primarily by the volumetric flow rate or injection speed, the melt viscosity, temperature, and the channel dimensions. While it is possible to reduce the melt viscosity by increasing the melt temperature, thus reducing the pressure drop, most injection molding resins have an ideal melt temperature that provides fast cycles and optimum part quality.

A well engineered cold runner should be designed by assuming an ideal melt temperature. This temperature can be found in the resin supplier's molding manual.

The other assumption that must be made initially is how much pressure drop can be tolerated. If the injection molding machine can deliver 20,000 psi pressure, the runner system should be designed so that the pressure required is somewhat less than the machine's capacity. A runner pressure drop below 10,000 psi should be used for difficult parts to fill (thin and long walled parts); a pressure of 5,000 psi is usually adequate to fill and pack out most cavities. This means, that the runner's system can be designed for a 10,000 psi pressure drop.

Example 10-8

The sample calculations of an eight-cavity, balanced runner layout, with a main runner of 5.0 in length and a diameter of 0.156 in, a secondary runner with 3.0 in length and a diameter of 0.131 in, a tertiary runner with 1.0 in length and a diameter of 0.125 in. We assume that all runners are full round, a resin specific weight of 1.0, part weight of 15.0 g, and an injection time of 3.0 s.

For eight cavities together, the total volume is 120 g or 7.32 in³.

$$\begin{aligned} \text{Volume (in}^3\text{)} &= \frac{\text{Weight (g)}}{\text{Specific Gravity} \times 16.38} = \frac{120}{1.0 \times 16.38} \\ &= 7.32 \text{ in}^3 \end{aligned} \quad (10-23)$$

$$\text{Runner Volume} = V = \pi r^2 L \quad (10-24)$$

Where:

r = Runner radius (in)

L = Runner length (in)

Thus,

$$\text{Primary runner volume: } V_1 = 3.1416 \times 0.078^2 \times 10 = 0.191 \text{ in}^3$$

$$\text{Secondary runner volume: } V_2 = 3.1416 \times 0.0655^2 \times 12 = 0.161 \text{ in}^3$$

$$\text{Tertiary runner volume: } V_3 = 3.1416 \times 0.0625^2 \times 8 = 0.098 \text{ in}^3$$

$$\text{Total volume (runner + parts)} = 7.32 + 0.191 + 0.161 + 0.098 = 7.77 \text{ in}^3$$

Since the flow splits at the intersection of the sprue and primary runner into two identical halves of the runner system, we need only calculate the pressure loss through one half of the mold. The volume of the melt, therefore, that must be conducted through the primary runner in this half of the system is 3.88 in³.

$$\text{The volumetric flow rate} = Q = \frac{\text{Volume}}{\text{Fill Time}} = \frac{3.88}{3.00} = 1.29 \text{ in}^3/\text{s} \quad (10-24)$$

$$\text{The shear rate} = S_r = \frac{4 \times Q}{\pi \times r^3} \quad (10-25)$$

$$\gamma = \frac{4 \times 1.29}{3.1416 \times 0.078^3} = 3,463 \text{ (1/s)}$$

The melt viscosity at this shear rate and at the specified melt temperature must be read from the unreinforced nylon 6/6 graph (Figure 10-83). For this hypothetical example, the apparent melt viscosity, $\mu = 0.0070 \text{ lb}\cdot\text{s}/\text{in}^2$.

$$\text{The shear stress} = \tau = \mu \times \gamma_r \quad (10-26)$$

$$\tau = 0.0070 \times 3,463 = 24.248 \text{ psi}$$

$$\text{The pressure drop} = P = \frac{\tau \times 2 L}{r} \quad (10-27)$$

$$P = \frac{24.24 \times 2 \times 5}{0.078} = 3,107 \text{ psi}$$

Now, the next runner segment must be considered. The total volumetric flow through each secondary runner is 3.88 in^3 minus the volume in the primary runner:

$$(3.88 - 0.095) / 2 = 1.89 \text{ in}^3 \text{ (remember that the flow splits in half again at the secondary runner).}$$

The volumetric flow rates in each secondary segment
 $= 1.89 / 3 = 0.63 \text{ in}^3/\text{s}$.

Thus,

$$\gamma = \frac{4 \times 0.63}{3.1416 \times 0.065^3} = 2,930 \text{ (1/s)}$$

The melt viscosity at that shear rate is $0.0080 \text{ lb}\cdot\text{s}/\text{in}^2$. Therefore,

$$\tau = 0.0080 \times 2930 = 23.44 \text{ psi}$$

$$P = \frac{23.44 \times 2 \times 3}{0.065} = 2,160 \text{ psi}$$

Volumetric flow through each tertiary runner can be calculated by subtracting the volumes of primary and secondary runners, or simply by adding the total tertiary runner volume and the total part volume and dividing by eight cavities:

$$\frac{0.096 + 7.32}{8} = 0.927 \text{ in}^3$$

The volumetric flow rate is $0.927 / 3 = 0.309 \text{ in}^3/\text{s}$.

$$\gamma = \frac{4 \times 0.309}{3.1416 \times 0.062^3} = 1,611 \text{ (1/s)}$$

The viscosity corresponding to this shear rate is $0.010 \text{ lb}\cdot\text{s}/\text{in}^2$, and:

$$\tau = 0.010 \times 1,611 = 16.11 \text{ psi}$$

$$P = \frac{16.11 \times 2 \times 1}{0.062} = 515.2 \text{ psi}$$

The total pressure loss from the sprue to each gate is the sum of the pressure losses through each segment:

$$\text{Pressure drop (total)} = 3,107 + 2,163 + 515.2 = 5,785 \text{ psi}$$

These preliminary calculations show that the total pressure drop can be increased by using smaller cold runner diameters to hold a pressure drop below 10,000 psi, and reducing the total runner weight.

By repeating the calculations for progressively smaller runner diameters until we reach the targeted pressure drop, we eventually come up with the ideal runner diameters, as shown in Table 10-9. In this cold runner analysis example (calculations omitted), the tertiary runner diameter cannot be reduced below 0.125 in, because of the restrictions of the molded part wall thickness. The secondary runner diameter should remain the same, because the pressure drop does not present a problem. However, the main or primary runner can be reduced from 0.156 to 0.140 in, causing the total pressure drop of the runner to increase from 5,785 to 6,036 psi (insignificant), while reducing the total runner weight from 7.37 to 6.78 g.

10.14.2 Determining the Injection Pressure Needed

How much pressure drop can be tolerated by the time the melt reaches the cavity while making sure that there is adequate pressure left to pack out the part and achieve consistent dimensions? One assumption that seems to hold for most injection molds is that there should be at least 5,000 psi available to fill a cavity. Since there are very few absolute rules in injection molding, this 5,000 psi pressure recommendation should be considered only as a “rule of thumb”. It is not recommended to run the molding machine at its maximum pressure capability. In our example, assuming an available pressure of 15,000 psi is a better selection. Because we have allowed 5,000 psi to fill the mold, the runner system must be designed with a 10,000 psi pressure drop. With the calculations in the previous section we see that the pressure drop is 5,785 psi. This result also reveals that our initial estimate of runner diameter using the values from Table 10-9 were much too small for the length of the runners. We must go back and recalculate, based on larger runner diameters.

We may start by using the pressure drop equation to recalculate pressure loss for a new primary runner diameter. To simplify the process, keep the shear-stress value constant and substitute a larger runner diameter value until we obtain a lower pressure drop. It is then necessary to go back to the equation for shear rate and recalculate to find the actual pressure drop for the new runner diameter.

As before, it may be necessary to recalculate new values for the secondary and tertiary runner diameters as well. The calculations should be repeated by assuming larger and larger runner diameters, until the targeted pressure loss is achieved. With some practice, this can usually be done manually in several hours or with the help of a runner pressure drop software program and a computer.

The same runner system with larger runner diameters calculated gives an overall pressure drop of 5,000 psi. The result is that now we have a runner system engineered to perform the required task of conveying the melt at sufficient velocity and low enough pressure losses to achieve the required part quality, while providing for the lowest volume of material in the runner itself.

In conclusion, the above example is intended to show that properly sizing a runner system is a matter of simple arithmetic and trial and error. This computation might take several hours, but it might save many more hours during the start-up plus considerable downtime and expense if the mold does not have to be sent back to the tool builder for changes in the cold runner's system.

10.14.3 Cold Runner Flow Tab

The use of a flow tab as shown in Figure 10-84 is helpful in maintaining consistent production. The flow tab is made by machining a groove, longer than the expected flow, about 0.25 in wide and 0.020 in thick that extends from a runner cold slug pocket. It is marked at 0.125 in intervals to show the extent of flow into the tab. A venting at the end of the tab should be provided to avoid any restriction to flow. The operator can check periodically to determine if the tab length changes, showing a change of molding conditions and part size.

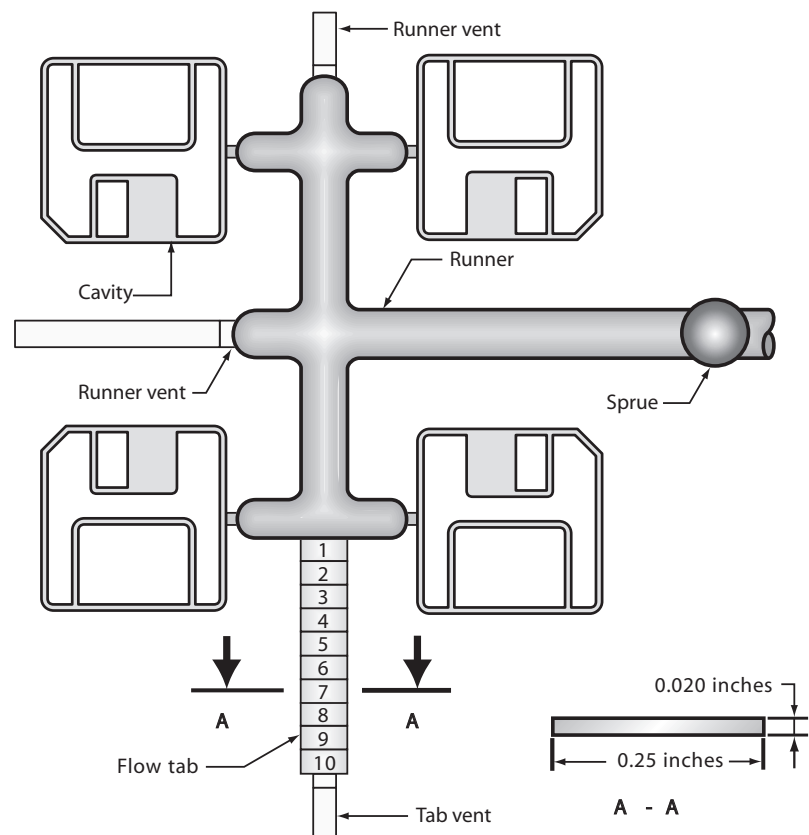


Figure 10-84 Mold cold runner flow tab

10.15 Mold Cavity Gating

The gate operates as a thermo valve; the valve opens allowing the hot melt to flow at high injection speed and pressure to fill the cavity and the valve closes when the melt stops flowing. Designing the gate as small as possible to reduce its visibility on the molded product is always desirable. However, a small gate causes the most severe restriction to flow into the cavity. Because of the especially high shear rates of the melt as it passes through the reduced gate area, the melt temperature increases and a substantial amount of the potential energy represented by the pressure on the material is converted to heat by friction. The effect on the material is drastic and, for shear-sensitive materials, there is substantial degradation in the molecular weight of the material as it passes through the gate. If a thermoplastic reinforced resin is compounded with fiber glass or mineral additives, the severe flow patterns generated at the gate will break up the reinforcing materials and convert fiber to powder with a substantial loss in the reinforcing properties of the resin.

The whole idea of gating is that the runners must form a good hydraulic system to allow sufficient quantities of hot melt to reach and enter the gates with minimum heat loss. This allows the maximum injection pressure possible to be transmitted to the mold cavity without interruption of melt flow through the gate. When the injection screw/check valve stops its forward travel, pressing against the melt channels, material in the small gates will freeze quickly, making it essential that all necessary material flow through in one uninterrupted movement. Because the gates do freeze after the cavities are filled and packed under pressure, no additional material can be forced into the cavities. However, with large gates, forcing in additional material after the thermoplastic melt has partially set may result in a laminated condition around the gate as the hot melt moves over the relatively colder material that is already in the cavity.

The gate size is important for the following reasons:

- The gate freezes soon after the cavity is filled allowing the injection screw/check valve to pack the cavity and control the quality of the molded product during the cooling cycle
- The correct type of gate allows for simple gate separation from the molded product and automatic molding process
- After de-gating, only a small witness mark remains on the molded part
- Uniform quality control of multi-cavity molds can be achieved
- The correct screw forward time (fill and pack) with the gate open allows higher density, reduces mold shrinkage, and provides better dimensional control, increasing the molding process efficiency.

The optimum type, geometry, dimension, and location of the gate are determined by the following factors:

- The viscosity and shear rate characteristics of the resin to be molded
- The shot size or volume of material to be injected
- The melt and mold processing temperatures
- The crystallinity rate or time required to freeze the melt in the mold cavity
- The size, complexity, and wall thickness of the molded part

- Molded product requirements (flatness, roundness, tolerances, surface finish, toughness, strength)
- Type of injection molding process (manual, automatic, runnerless, two- or three-plate, vertical inserts, lost cores, air assist).

10.15.1 Types of Mold Cavity Gates

To obtain the optimum injection molding process conditions, the type of gate must be carefully chosen. Gates commonly used are sprue gate, edge gate, tab gate, fan gate, diaphragm gate, external ring gate, internal ring gate, spider gate, film gate, pin point gate, tunnel gate, and runnerless gate systems.

Sprue Gate

When the molded part is injected directly from a sprue, the feed section is called a sprue gate. The main disadvantage with this type of gate is that it leaves a large gate mark on the molded part, requires manual sprue removal, and single cavity molds. The size of the mark depends on the diameter at the small end of the sprue, the sprue angle, and the sprue length. The gate mark can be reduced by keeping the dimensions to a minimum.

The sprue small diameter should be 0.031 in larger than the nozzle exit inside diameter to avoid sprue ejection problems and molding cycle interruptions. An extension nozzle sprue bushing (see Figure 10-72, top illustration) can often be used to an advantage, because it enters a recess in the mold and cuts down the overall sprue length. A sprue gate is illustrated in Figure 10-85.

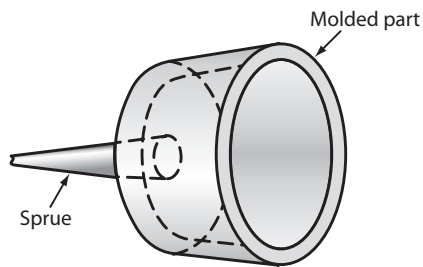


Figure 10-85 Sprue gate

Rectangular Edge Gate

The edge gate is a general purpose gate with the simplest geometry; it has a rectangular channel machined in one mold plate to connect the runner to the cavity.

The rectangular edge gate offers the following advantages:

- The cross sectional geometry is simple and cheap to machine
- Close accuracy in the gate dimensions can be achieved
- The gate dimensions can be easily and quickly modified.

The filling rate of the cavity can be controlled relatively independently of the gate freezing time. One disadvantage of this type of gate is that after gate removal, a witness mark is left on a visible surface of the molded part. This is more noticeable with certain materials when the molded parts are broken from the gates.

Edge Gate Critical Dimensions

Because of the rectangular shape, the dimensions of the gate are given by width (W), depth (H), and land length (L) as shown in Figure 10-86. The pressure drop across the gate is approximately proportional to the land length and should be kept as small as possible to be consistent with the strength of the steel that remains between runner and cavity. A land of 0.040 in is recommended.

The minimum depth of the gate controls the time for which the gate remains open. This gate open time must be sufficient for the melt to reach the extremities of the cavity. Providing the wall section of the component has been correctly

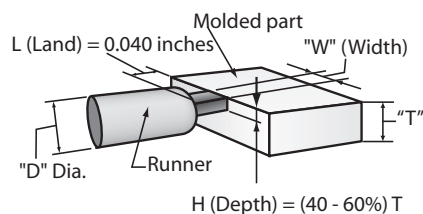


Figure 10-86 Rectangular edge gate

chosen with regard to the maximum length of flow required, it is reasonable to expect a relationship between the gate depth dimension, the wall thickness of the molded part, and the crystallinity of the thermoplastic resin. In practice, a gate depth ranging from 40% to 60% of the molded part wall thickness is recommended.

The cross sectional area of the gate ($H \times W$) controls the rate at which the thermoplastic melt enters the cavity. Gate width (W) should be between the depth (H) and the runner (D) diameter, depending on the volume of the melt that must flow through the gate.

Overlap Edge Gate

This gate is a variation of the rectangular edge gate and is used to feed certain types of molded parts. Figure 10-87 shows the overlap edge gate is machined into the plain mold plate so that it bridges the gap between the end of the runner and the end wall of the cavity.

This gate may be used for all the common molding materials except rigid PVC. This gate, being attached to the molding surface, does require more careful removal and finishing than for edge gates. The size of the gate can follow the general pattern suggested for the rectangular edge gate with the same limitations.

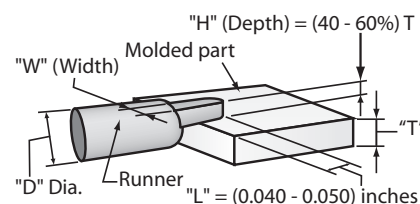


Figure 10-87 Overlap edge gate

Fan Edge Gate

The corresponding dimensions of the fan gate are not constant, the width increases from the runner diameter or from a perpendicular sub runner as shown in Figure 10-88, while the taper depth decreases to maintain a constant cross sectional area throughout the length of the fan component of the gate.

The width of the gate at the cavity is relatively wide and, because of this, a large volume of material can be injected in a short cycle time. This type of gate can be used advantageously for large area, thin-walled molded parts. The fan shape appears to spread the flow of the melt as it enters the cavity and a more uniform filling is obtained with fewer flow lines and surface blemishes.

The gate may be used for all common molding materials, except rigid PVC. The relevant gate sizes that must be chosen are the land length (L), the gate width (W), and the gate depth (H). The land length needs to be slightly longer than

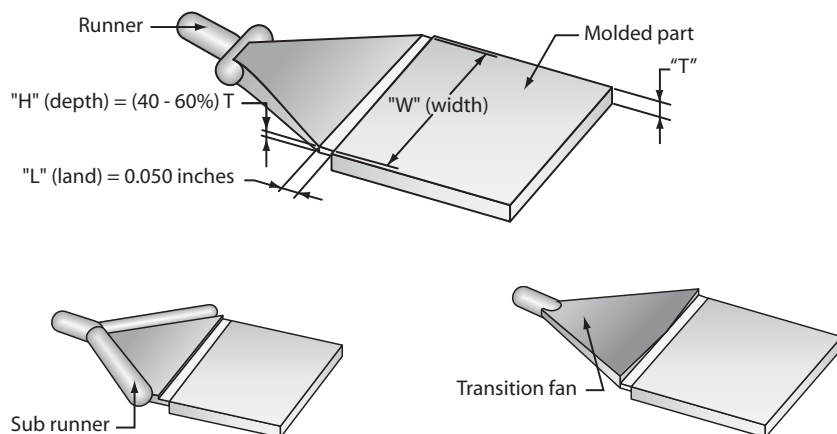


Figure 10-88 Various types of fan edge gates

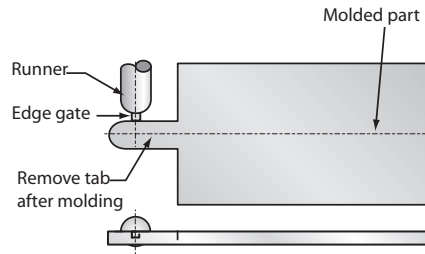


Figure 10-89 Tab edge gate

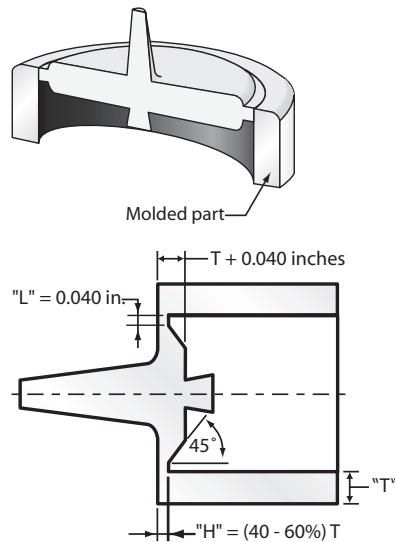


Figure 10-90 Sprue diaphragm gate

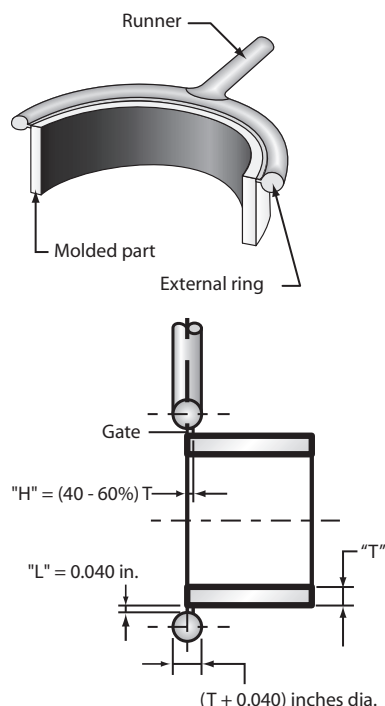


Figure 10-91 External ring gate

for the rectangular edge gate, a suggested size of 0.050 in is ideal to trim the gate using a knife while the molded part is hot. The width used is often much bigger to obtain the full benefits of the fan gate design.

The main disadvantage of the design is that a large witness mark is left on the molded part that must subsequently be trimmed and finished. Designing the gate relatively narrow is therefore advantageous.

To maintain a uniform pressure distribution inside the trapezoidal configuration of the fan gate the depth must be blended, stream-lined, progressively increased, and tapered down accordingly.

The effective fan gate depth (H) between the transition sub runner and the cavity progressively increases from a minimum at the center line to a maximum at the outer gate width (W). To compensate for the pressure differential caused by a uniform fan gate depth, it is common practice to increase the depth of a wide fan gate at either side to provide for more even melt flow through the fan gate.

Tab Edge Gate

A side tab perpendicular to the runner and edge gate is added to a molded part, as shown in Figure 10-89. The tab and gate are removed after mold ejection.

This technique prevents the undesirable jetting problem by changing the melt flow direction, forcing the melt to fill uniformly in a smooth steady flow depending on the shape of the cavity. The tab gate leaves large witness marks. This gate was developed for high viscosity amorphous resins, such as acrylic.

Sprue Diaphragm Gate

This gate is used for single-cavity tubular shaped injection molded parts. It may also be used for multi-cavity tubular parts using hot runner molds.

Diaphragm gates are ideal for molding round products requiring precision dimensional control (TIR, roundness) and optimum strength without weld lines.

The sprue leads into a circular distribution disc 0.040 in thicker than the wall thickness (T) and smaller than the part inside diameter, see Figure 10-90. This disc functions as a runner, allowing the melt to flow radially for constant filling of the continuous circular gate. The center disc is sometimes tapered from the center toward the gate to save material, provide thinner wall thickness, efficient mold cooling, and faster cycles. Sprue and gate are removed using a secondary post molding operation.

External Ring Gate

The function of this gate is similar in performance to the diaphragm gate, but it is recommended only for unreinforced, low melt temperature, low crystallization rate thermoplastic resins. This type of gate is used for tubular type molded parts when more than one cavity is required in a simple two-plate mold. The gate provides for feeding all around the external periphery of the cavity and permits the use of a conventional runner system to connect the cavities.

A trapezoidal runner is normally used because this type of molding would be ejected using a stripper plate. The gate is like a concentric film between the runner and the cavity. The dimensions of this gate are similar to the diaphragm gate.

Figure 10-91 illustrates an external ring gate.

Internal Ring Gate

The internal ring gate functions similarly to the external ring gate. This gate is used for molding small tubular molded parts usually made of unreinforced or reinforced thermoplastic resins. The sprue is connected to the internal ring via two or more sub runners, feeding all around the internal periphery of the cavity through a continuous and concentric circular film gate.

Figure 10-92 shows a typical internal ring gate and its dimensions.

Spider Gate

The spider gate functions similarly to the diaphragm gate but does not provide the same results. This gate is used for tubular molded parts. The spider gate uses two or more internal spider tunnel or rectangular edge gates to feed the internal periphery of the cavity (equal spaces) and permits the use of a sprue system (single cavity) or a hot runner mold to feed several cavities. Figure 10-93 shows a sprue connected to three spider tunnel gates and a sprue connected to four spider edge gates.

Film Edge Gate

The film gate may be considered a long rectangular edge gate. Film gates are used for injection molding large, thin-walled, good surface finishing thermoplastic components to help in the production of warp-free products.

Film gates are useful for those materials that exhibit differential mold shrinkage, for which reduced central feeding of the melt is impractical.

The gate normally extends across the complete width of the molded part, although a smaller width may be used initially. The film gate width should be increased and the molding process parameters optimized until the product is satisfactory and meets the specified requirements.

The film gate is similar in principle to the diaphragm, external, and internal ring gates that provide a uniform and continuous wide entrance for the melt flow to fill and pack the cavity without molding defects and operational problems. The film gate depth may be less than for a corresponding rectangular edge gate if the melt flow rate, the crystallinity rate, the shear rate, and viscosity do not exceed the boundaries and molding characteristics of the polymer.

A parallel runner and a film gate are provided at the side face of the cavity to feed the melt evenly. A slender steel wall exists in the mold, between the runner and the cavity; a minimum film gate land of 0.050 in is suggested. A film edge gate is illustrated in Figure 10-94.

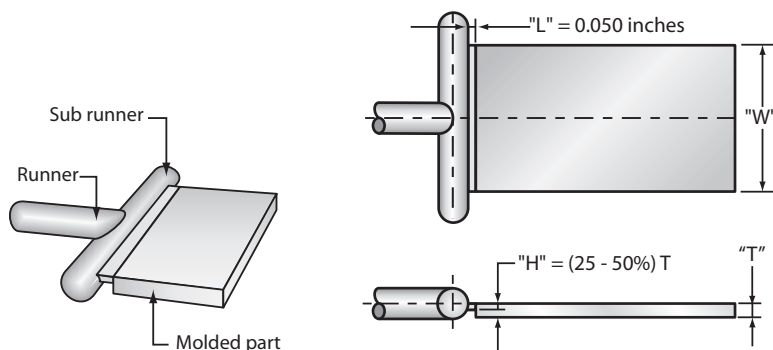


Figure 10-94 Film edge gate

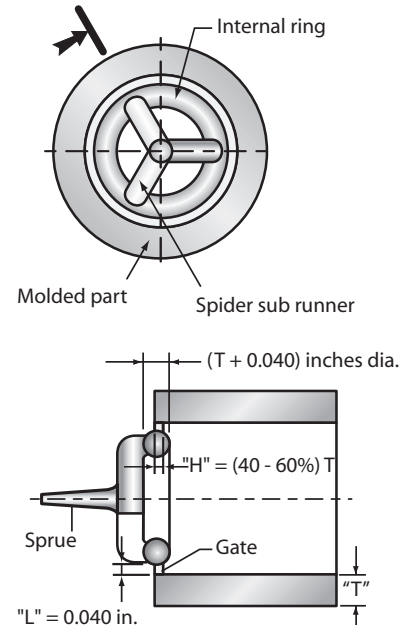
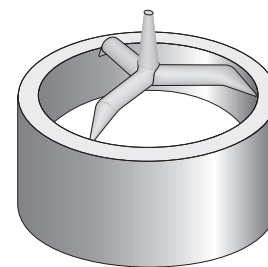


Figure 10-92 Sprue, three sub runners with internal ring gate



Tunnel spider gate

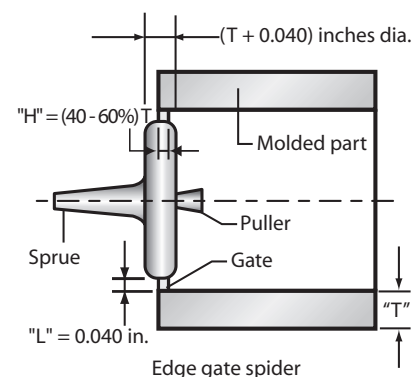


Figure 10-93 Three-spider tunnel gate and four-spider edge gate

Pin Point Gate Cold Runner System

The pin point gate is a cold runner system that offers many molding process advantages, such as precision dimensional control of molded circular gears. This system causes little surface imperfections to the molded product. The circular gate marks are relatively small and insignificant for many applications.

This system is used as an alternative method to gating into the center of a component. This gate system is often used for single or multiple pin point gating. It is desirable for resins that exhibit high differential mold shrinkage characteristics.

The pin point gate connects the cavity to the conical sub runner, to a parabolic or trapezoidal runner, and to a sprue.

A three-plate mold with three or four equally spaced pin point gates in each cavity has been used successfully. An extra plate is added behind the cavity plate to permit a runner system to be behind the mold cavities.

This system is composed of a sprue, a sprue puller, parabolic or trapezoidal runners, ejection shoulders at the end of the runner, conical sub runners, taper/round pin point gate, and sucker pins used to separate the molded part from the pin gate to eject the runner system automatically from the mold by using the same sucker pins.

Because of the pin point gate geometry, it may be difficult to control the melt flow rate, melt process temperature, shear rate, and gate freezing time independently. A compromise between these melt behavior characteristics is needed to find out which one of the thermoplastic resin's properties are the most important for molding conditions to improve the process efficiency.

Figure 10-95 details a pin point gate.

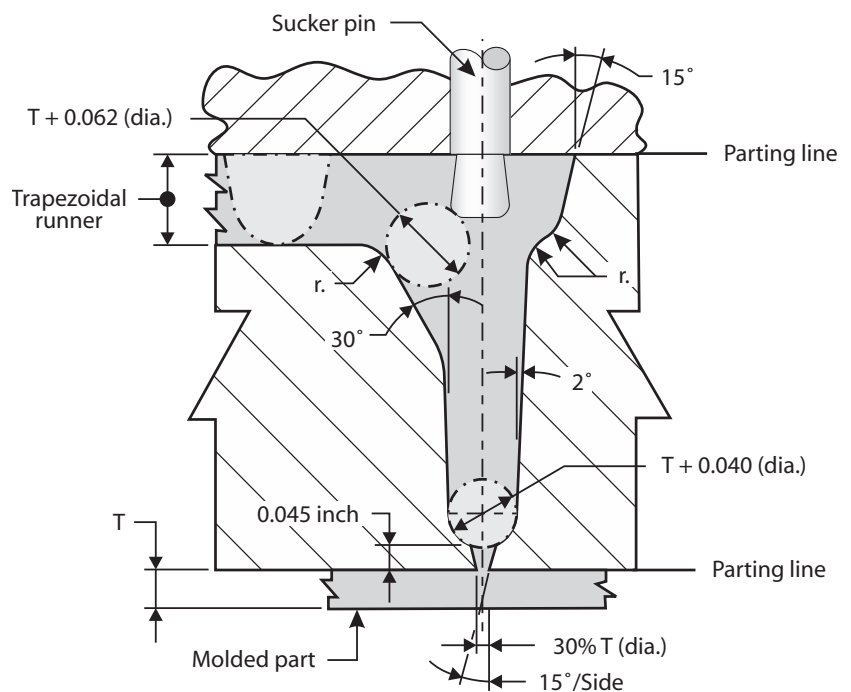


Figure 10-95 Pin point gate runner system for a three-plate mold

Round Edge Gate

The round edge gate is a matching semicircular channel machined in both halves of the mold, between the runner and the cavity. The round edge gate is recommended only for the injection molding process of melt processing rubber compounds (TPE), of molded parts with a minimum wall thickness of 0.150 in, and when dimensional control is not critical for the application.

The round edge gate is an advantage for melt processing of rubber (TPE). The melt flow rate is affected by the shearing force applied to the polymer melt during the injection molding process.

The round edge gate is an efficient melt shearing device. By increasing the shear rate, causing the melt process temperature to increase by injecting the melt under high injection pressure and fast injection velocity, increasing the melt at higher flow rates. The high melt process temperature and fast melt flow rates used to fill the cavities causes the round gate area to increase the temperature, forcing the round gate to remain open during the time required to complete the screw forward cycle to allow packing pressure to be applied to the cavity, thus increasing the density and reducing the mold shrinkage of the molded parts.

Standard Tunnel Gates

The standard tunnel gate is a circular or oval gate that submerges and feeds into the cavity below the parting surface of the mold. Feeding into the side of the cavity has several advantages:

- No cavity matching problems using a two-plate mold.
- The oval cross section controls the filling rate (cavity); the conical shape and the length (conical sub runner) control the freezing time of the gate.
- The gate is sheared off from the cavity automatically during ejection.

The runner stops at a steel safety margin distance from the edge of the cavity. A conical sub runner is machined at an angle between 30° and 45° and is stopped short of the cavity wall by a distance (gate land length). The gate is then machined or EDM (burned) at the same angle and tapered at 15° per side to join the conical sub runner and pin gate to the cavity.

The molded parts and the runner systems are removed separately from the mold; this means that a separate runner ejection is advantageous, particularly as a certain amount of deformation of the runner is necessary to shear off and remove the cold runner system from the mold. The sequence of a tunnel gate operation is illustrated in Figure 10-97.

The standard tunnel gates are found in two varieties: short tunnel and long tunnel gates. Generally, the short tunnel is preferred. When a long tunnel is used, the angle between the part and the tunnel is important. The angle should not exceed 30° to ensure gate shearing. The long tunnel gate should never be used with high crystalline resins or molded parts requiring precision dimensional control.

The steel safety margin must be at least 0.078 in or greater. The short tunnel angle between the center line of the conical sub runner and the cavity wall is normally between 30° and 45°. The dimensions for the oval short tunnel gate cross section at the cavity side wall should be between 30% and 40% of the molded part wall thickness (T), based on the melt flow rates, crystallinity rates, the modulus of elasticity, how much fiber glass or mineral reinforcement is used, and the melt viscosity of the thermoplastic resin.

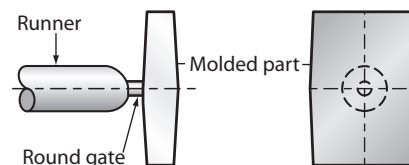


Figure 10-96 Round edge gate

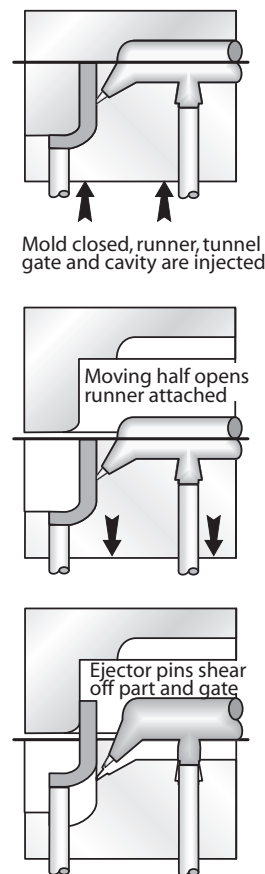
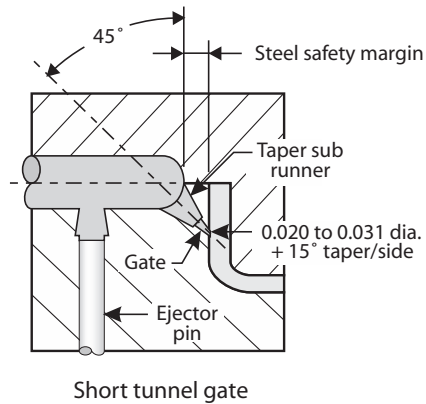
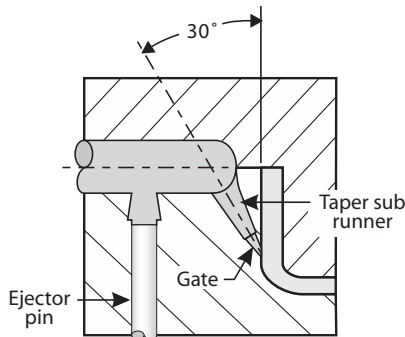


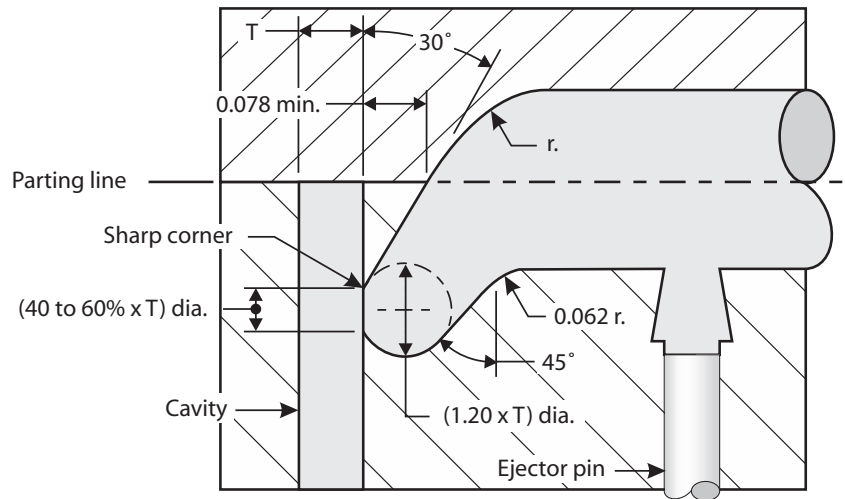
Figure 10-97 Tunnel gate, sequence of ejection



Short tunnel gate



Long tunnel gate (not recommended)

Figure 10-98 Standard tunnel gates**Figure 10-99** Du Pont type "C" tunnel gate

Du Pont Type "C" Tunnel Gate

This type of tunnel gate was developed by Du Pont in response to the need for molding resins with high crystallization rates and to produce close dimensional tolerance molded parts and aid automatic ejection. This tunnel gate cannot be machined like standard tunnel gates. The tunnel type "C" orifice is fabricated by first building an electrode and then burning the tunnel as specified in Figure 10-99.

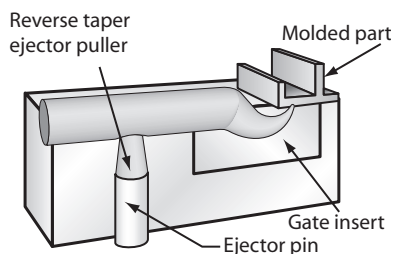
The round gate cross section diameter is between 25% and 35% of the part wall thickness. The melt volume diameter behind the tunnel gate pocket should be 20% larger than the part wall thickness (T). This pocket is necessary to bring more melt heat and keep the tunnel gate open for mold shrinkage control, without producing high shear forces, gate surface marks, or ejection problems.

Banana or Cashew Gate

Figure 10-100 shows a banana or cashew gate as a variation of a tunnel gate. Other than the standard tunnel gate, it can provide gating into the lower base of the molded component. Its primary limitation is that its curved shape forces the material in the gate to undergo considerable distortion during ejection. This requires the polymeric material to exhibit good ductility at the time of ejection. If a non-ductile material is used, its ductility can be increased by increasing the cross-section of the body of the gate to ensure that it is at an elevated temperature at the time of ejection. This method works best with amorphous materials that have a broad solidification temperature.

This type of gate can be purchased as a specially made, powder injection molded gate insert. It has a relief along the side walls, which will help thermally insulate the gate from the cooled mold. The warmer gate will remain more flexible and thereby facilitate ejection.

The dimensions for the gate openings should be 30 to 70% of the wall thickness. Parts formed with smaller gate openings will most likely have poor packing control and cause part shrinkage.

**Figure 10-100** Runner with banana or cashew gate

10.15.2 Different Types of Hot Runner Gates

The most critical part of all runnerless molds is the gate. The thermoplastic melt must be kept in a fluid state up to the point of separation, while the area just beyond the gate in the cavity must freeze rapidly and without serious flaws. Most runnerless injection molding problems occur in the gate areas.

Runnerless Mold Hot Tip Torpedo Insert Gate

The hot tip torpedo insert gate is a small conical spreader at the gate, which aids heat transfer to the melt and limits the formation of a vestige in the gate area. It contacts the inside diameter of the hot sprue bushing and draws heat from the torpedo cartridge heater. Several streamlined passages around the cone allow the passage of melt into the cavity. The purpose of this heated insert is to deliver heat to the gate separation point and to establish the exact location of the separation point.

Hot Runner Mold Valve Pin Gate

A valve gate is an open gate with a reciprocating valve pin. The valve is opened and closed during each cycle by means of a hydraulic, pneumatic, or spring mechanism. Valve gates provide maximum control of gate vestiges and cosmetics, but add complexity and introduce serious wear problems with fiber glass and/or mineral reinforced resins. Valve gates also tend to push cold slugs into the gate area of the molded part and these cold slugs may cause impact failure or surface appearance problems.

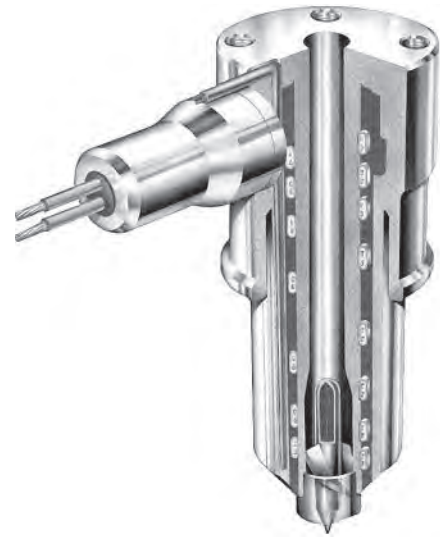


Figure 10-101 Runnerless mold hot tip torpedo insert gate (Courtesy: Mold Master)

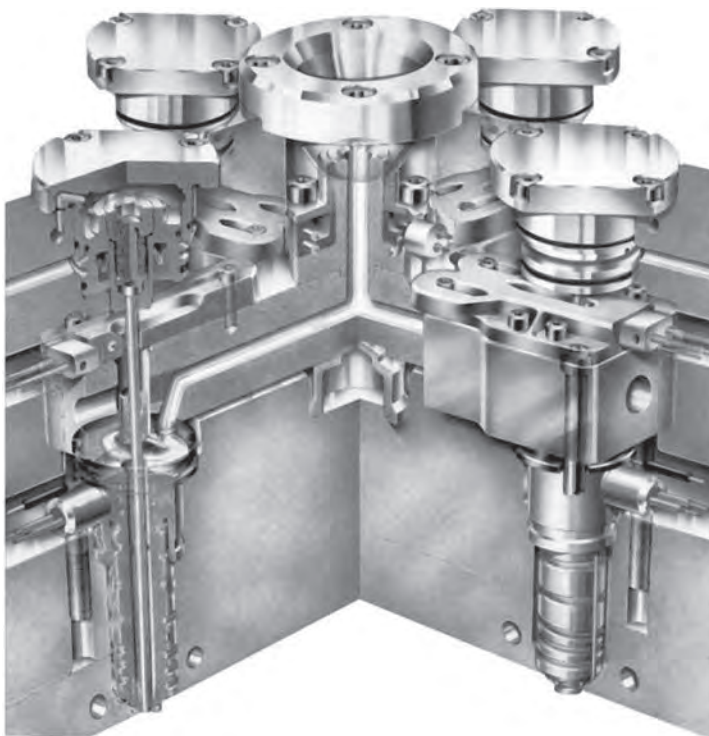
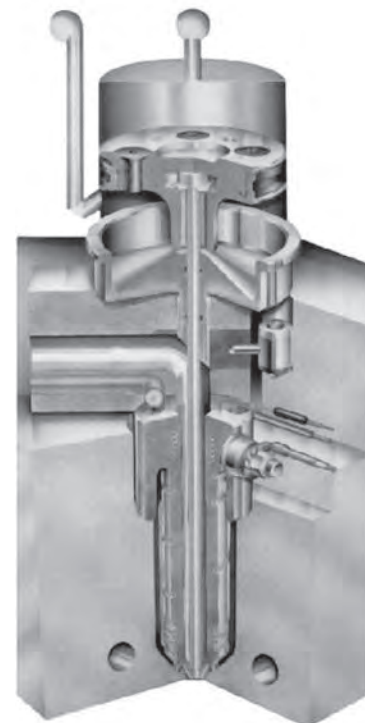
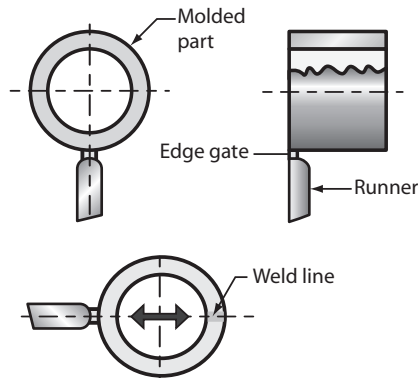
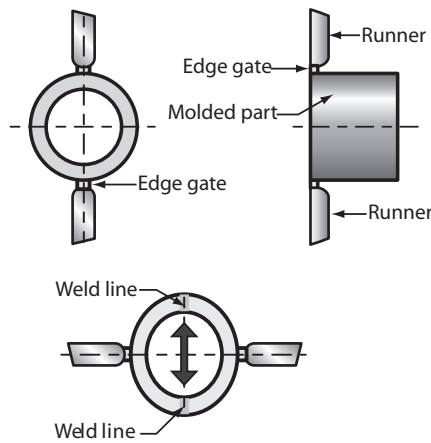


Figure 10-102 Two types of hot runner mold valve pin gates

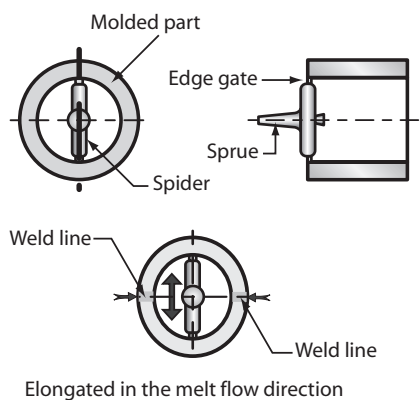




Elongated in the melt flow direction

Figure 10-103 Molding effects caused by a single external edge gate

Elongated perpendicular to the gates

Figure 10-104 Effects caused by two external runners and edge gates**Figure 10-105** Effects caused by two internal spider runners and gates

10.16 Gate Molding Effects

Ideally, the gate should be positioned to allow an even flow of the melt into the cavity, so that it fills uniformly and the advancing melt front spreads out and reaches the various cavity extremities simultaneously.

The location and the type of gates used in the molds affect the geometry and dimensional size of the molded part. These dimensional variations become more difficult to control with molding materials with high mold shrinkage characteristics.

The differential values based on the flow orientation and the types of reinforcements used to compound the polymers also affects dimensional control. The injection molding processing parameters and mold design also play an important role in controlling the final dimensions of the molded parts.

To illustrate the molding problems caused by the gate location, the following mold gate design cases have been prepared. To injection mold a thermoplastic bushing using a single edge gate, the melt is oriented parallel to the gate, and high pressure is applied at the opposite side of the gate. The round bushing becomes oval in the direction of the flow and a weld line is formed.

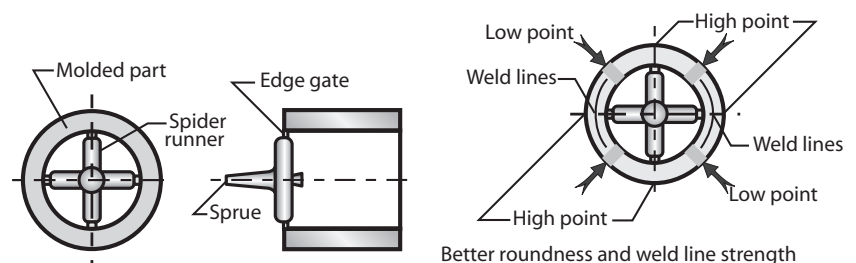
Figure 10-103 shows this molding effect caused by a single edge gate.

If two edge gates placed at an 180° angle are used to mold a thermoplastic bushing; the molding effects are similar to the previous case. The injection pressure from both gates causes a dual high force 90° away from the gates. The molded bushing becomes oval or elongated (less than in the previous case), with two stronger weld lines. Figure 10-104, illustrates the molding problems caused by the edge gates and the locations.

If two spider runners with edge gates are used to mold a bushing, the effect is opposite to the previous case. The injection pressure from both gates causes a dual high force in front of the gates. The bushing becomes oval, with two stronger weld lines 90° from the gates. Figure 10-105 shows the effects caused by the gates.

If a four spider sub runner gate is used to mold a bushing, the effects are similar to the previous cases. The pressure from the gates causes four high forces in front of the gates. The molded bushing becomes wavy with four high points at the gates and low points between the gates, but the weld lines are stronger and the roundness much better. Figure 10-106, illustrates these molding effects on the part.

To obtain optimum round dimensions of the injection molded thermoplastic bushing, the use of a sprue gate is recommended.

**Figure 10-106** Molding effects caused by four internal spiders and gates

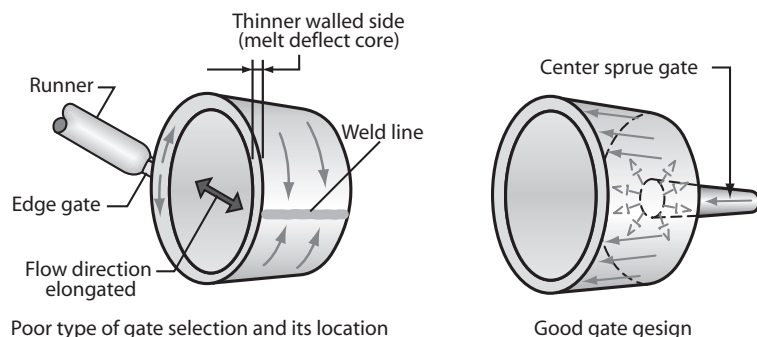


Figure 10-107 Molding effects caused by the type of gate and location

Such an ideal position for the gate is possible in certain geometries of circular cross sections, for example, a cup in which the melt is injected through a sprue gate at the center of the base. Figure 10-107, illustrates the molding effects caused by the type of gates and location on a molded cup.

Another strong reason for central gating components, such as a pen cap, is that side edge gating may cause deflection of the core. Edge gating provides a more rapid melt flow down one side of the cavity, resulting in a differential pressure that can move the core out of position. This creates a thinner wall in the cavity on the opposite side of the edge gate, adding another weakness to the weld line and incomplete molded pen cap or short shot.

Central gating at the base of the pen cap cavity can overcome these molding problems; however, this solution may lead to complications in a multi-cavity three-plate mold or a hot runner mold. Figure 10-108 shows where gate type changes and relocation were necessary to eliminate the molding problems.

For rectangular molded parts, there is no ideal gate position; even in this case, the central position gate is often preferable, unless the parts are thin-walled, particularly when the material used can exhibit differential shrinkage causing distortion.

When the edge gate is used, and most molded parts are edge-gated for reasons of mold economics, it should be positioned so that the melt flow immediately meets a steel cavity wall. When a rectangular cavity that is long and narrow is edge-gated at the center of the long end of the bar, the melt entering at high velocity and pressure produces a jetting mark. The melt front quickly sets on contact with the cool mold cavity walls. More melt then enters the cavity and flows around the original jetted melt front.

The resulting flow lines are often visible on the finished molded bar.

The molded bar processing problems can be overcome by using an overlap gate or an edge gate located at the top side of the cavity as shown in Figure 10-109. The flow of material from this type of gate or optimum edge gate location is forced to affect the side wall of the cavity causing the melt to form an advancing front that progressively fills the cavity, displacing the air trapped inside the cavity. This molding situation keeps the molded bar free of jetting marks on the surface.

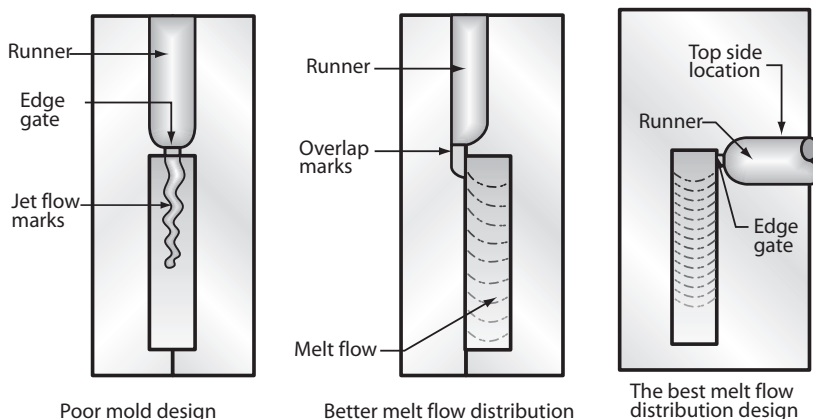


Figure 10-109 Molding effects caused by type of gate and location

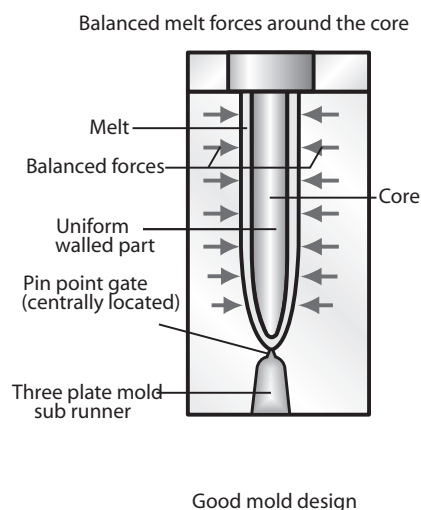
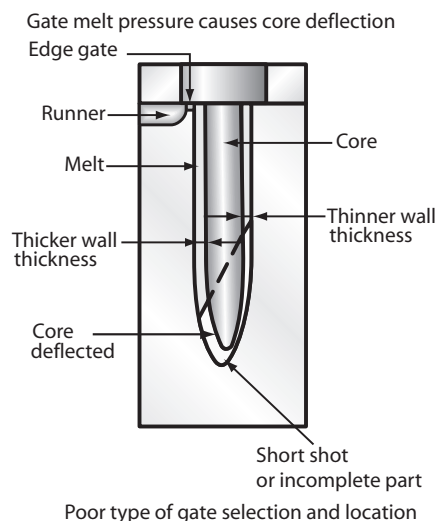
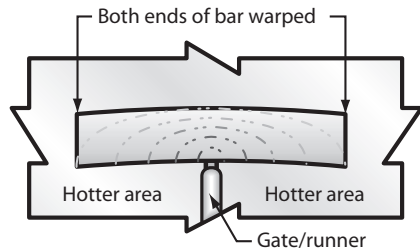
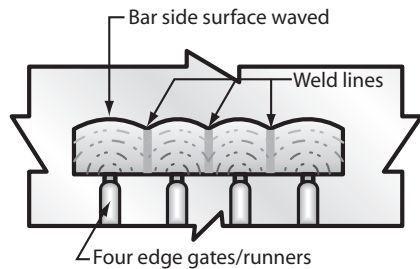


Figure 10-108 Molding effects caused by gate type and location



Poor mold design



Poor mold design

Figure 10-110 Effects caused by type, number, and location of gates

The edge gate located in the middle of the rectangular bar will cause bending problems, resulting from the high injection pressure of the gate at the middle far side of the cavity and the differential mold temperature between both sides of the cavity (see Figure 10-110, top illustration). If multi-edge gates are used on one side of the cavity (see Figure 10-110, bottom illustration), side surface waviness across the gates and weld line marks between the gates are formed.

10.17 Mold Venting Systems

Injection molding of thermoplastics requires the material to be melted by frictional heat and then to be injected into a closed mold cavity at high pressure and velocity. As it enters the cavity, the melt must displace the gasses trapped inside the cavity at mold closing time. If these gasses are not expelled from the cavity, they could suddenly compress between 2 and 5% of the original volume, causing additional high pressure inside the cavity. This high pressure increases the temperatures well above those suitable for the injection molding process. These elevated temperatures cause local ignition (dieseling) of the thermoplastic melt and burning of the molded products. What is more dramatic are the mold deposits, the corrosion in the mold cavity tool steel caused by the ignition sparks (dieseling effect) and the degraded polymer gasses.

The openings through which these gasses escape are known as vents; the ideal vent would be one that will allow gasses to expel freely from the cavity while completely blocking the escape of molten polymer, which would cause flashing.

Vents can take many forms and be strategically located at various areas of the mold. The type and location of the vent are determined by several factors, such as part geometry or product design, part end use requirements, mold design, injection molding machine, maintenance, process characteristics of the polymer, quality control, molding conditions, process efficiency, productivity, and part cost.

When the mold has been completed without considering the venting requirements, incorporating the best type of venting in the mold may no longer be possible.

Inadequate mold venting is a serious problem causing defective molded parts with flashing, burned marks, weak weld lines, short shots, gas voids, sink marks, surface finishing flow marks, and warpage (nonuniform shrinkage), creating quality control problems in the injection molding process.

Lack of proper venting will cause excessive use of injection pressure for the molding process, which will cause a high degree of internal stresses. Vents not only provide an effective means of displacing the trapped air in a cavity by the thermoplastic melt, but also permit the escape of gasses caused by the degraded polymer.

Improper mold vent dimensions used for a type of polymer or worn vents entrap gasses within the cavity that cannot be expelled before the injected melt solidifies.

Relief edge vents have to be small enough to prevent the polymer melt from entering the venting channels. A cavity can be considered sufficiently vented when the polymer melt is injected in the cavity at high speeds without any sign of burn marks on the molded part or corrosion spots in the cavity tool steel surfaces.

In a mold with several gates or core pins, the polymer melt inside the front of the cavity is divided at each gate and around the core pins. These different melt flows eventually converge and may therefore encapsulate the entrapped gasses causing weld line problems, gas voids, and burn marks. Weld lines often contain entrapped gasses causing weakness at the weld lines, especially if the melt fronts are colder after traveling a long distance inside the cavity. Venting at this location may be found necessary to improve the weld line strength, produce better surface finishing, and faster molding cycles. The venting problem is further improved by adding ring groove vents in the core pins and ejector pins. Venting the sprue puller, all the runner cold slug pockets, including runner ejector pins, also needs to be considered.

When molding thick section parts, venting the mold using a variety of techniques may be desirable. Venting the mold is strictly an art, perhaps a lost one. Mold venting design is a function dependent on the geometry and end use requirements of the part, types of resin, mold design, molding machine, process conditions, and quality control specifications.

10.17.1 Product Design for Venting

Much of the trouble can be avoided if venting problems are anticipated during the basic design of the part and the mold. The geometry of the part to be molded should be studied to decide the best location for the parting line, ejection system, and gating to achieve the best mold venting. In some applications, altering the design of the part to provide adequate venting may be necessary. Design modifications need not necessarily be extensive or affect the part requirements. For example, a slight increase or decrease in thickness of a section of the part may be sufficient to create preferential flow conditions for placing a desired vent.

Several product design geometries can lead to venting problems:

- A thermoplastic injection molded product designed with thin-walled sections connected and surrounded by thick sections should be avoided. Figure 10-111 shows that the last place to fill the cavity using an edge gate would be the thin circular section in the center of the cavity. The melt inside the cavity starts filling the heavy sections next to the gate. When the melt front faces obstacles or restrictions such as thin walls, the melt temperature decreases and the melt flow rate is lower. The injection pressure forces the colder melt to fill the small openings or thin walled sections where the trapped air from the cavity has been moved. The trapped air is heated and compressed by the melt causing short shots and burn marks. If such geometries are needed, special parting face vents should be implemented through the cavity insert contact surfaces.
- Blind, deep holes in the cavity should be avoided. Figure 10-112 shows this type of product design problem. When a center shaft cannot be eliminated, the shaft depth should be reduced and vented.
- Avoid any geometry that could cause a preferential filling along the parting line or around core pins, where the melt flow is divided and the unvented section of the cavity is last to fill. Figure 10-113 shows how preferential filling will occur around the thicker upper cup lip portion of the part. Premature sealing of the parting line vents will result. This mold will be extremely difficult to fill if proper venting is not installed at the lower thin wall section of the cavity.

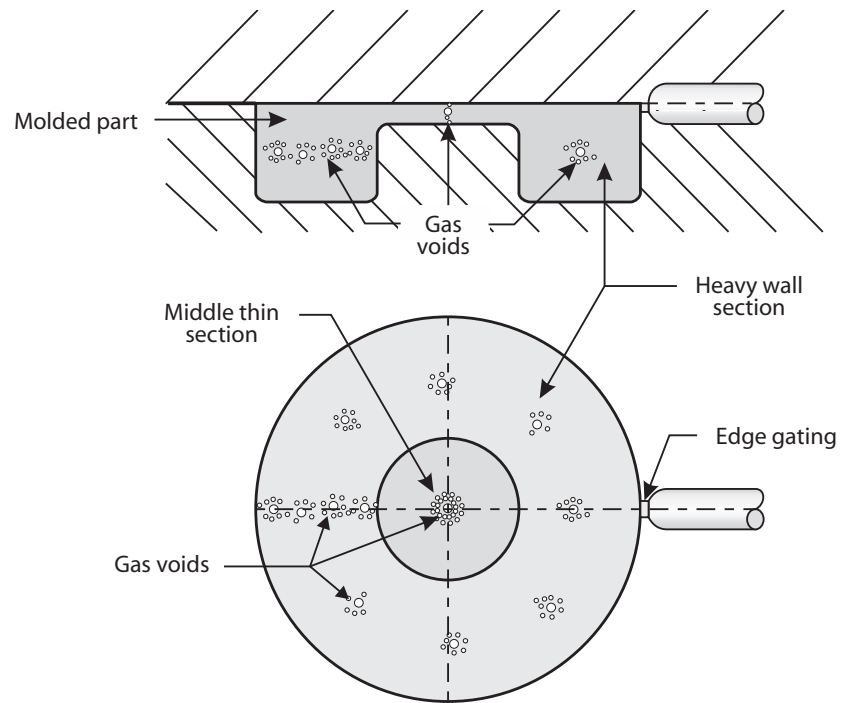


Figure 10-111 Venting problems, outer heavy/middle thin wall section

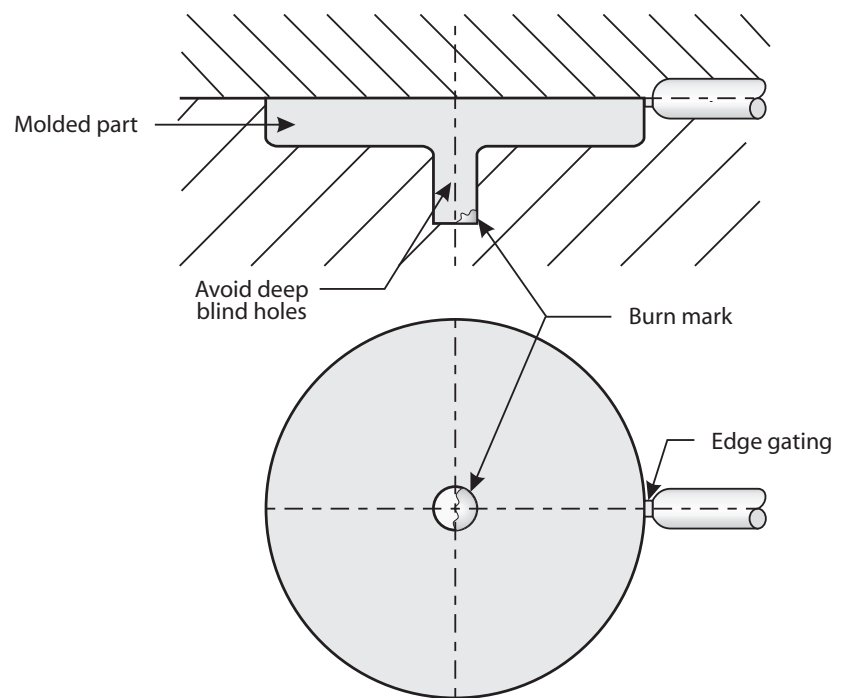


Figure 10-112 Venting problems with a cavity/center deep blind hole

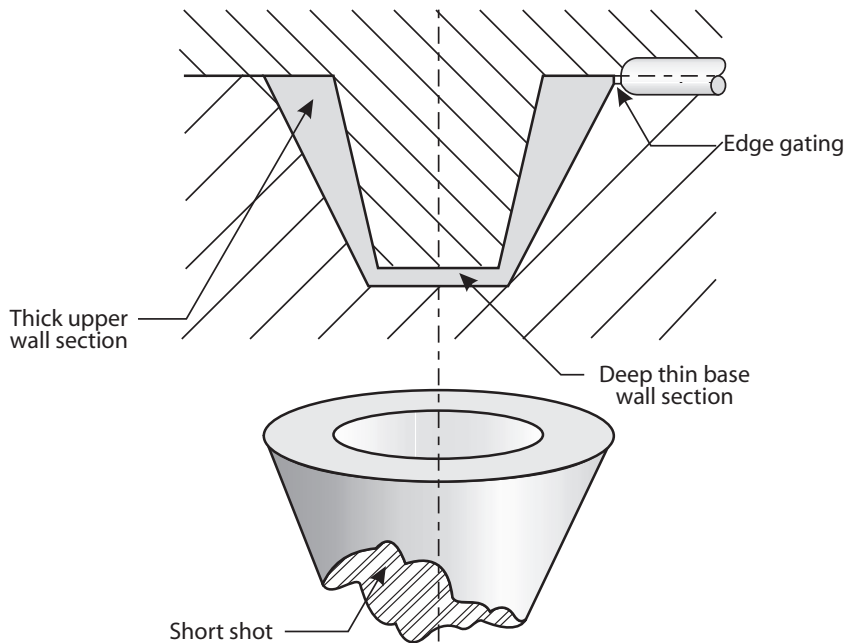


Figure 10-113 Venting problems of thick upper/thin base taper cone

10.17.2 Venting Characteristics of Thermoplastic Polymers

The molding resin, the product geometry and end use requirements, and the molding process often influence the extent to which mold venting is needed. There are two characteristics of thermoplastics that lead to venting problems:

- Thermoplastic resins that release large quantities of vapor when heated require more mold venting than materials that do not.
- Thermoplastic resins that have sharp melting points, fast melt flow rates, and high nucleation. For a fast injection, the mold must be well vented. The melt flow rate influences the need for mold venting. A high melt viscosity or low melt flow rate PE will require less mold venting than a more fluid and fast setting polymer such as nylon.

10.17.3 Mold Deposit Problems

Mold deposit is a hazy surface on the mold cavity; if the mold deposits are allowed to accumulate, they can blemish the surface of a molded part or reduce its glossy appearance, occasionally causing sticking problems. In severe cases, mold deposits can build up sufficiently to change critical molded part dimensions.

Small core pins used in mold cavities generally function at higher mold temperatures than the rest of the mold cavity, thus being susceptible to mold deposits. Mold deposits can narrow an orifice in a cavity, as in an aerosol molded part, or worse, seal it, causing part rejection problems. Reducing the mold deposit problem requires cleaning off the cavity or changing the molding resin to new low mold deposit thermoplastic compounds.

Cleaning of Mold Deposits from the Cavity

Where molding conditions cannot be changed to reduce mold deposits, a scheduled cleaning can prevent these deposits from being baked onto the mold cavities, making the mold deposit removal easier.

- Vapor mold deposit removal has been successful by using aerosol cleaners like “Slide”, resin remover “The Stripper”, mold cleaner “Chem Trend”, the heat from a propane torch, or a hot air gun and chrome cleaners, and heavy duty cleaning compounds such as trisodium phosphates. If the mold deposit has accumulated for long periods, ultrasonic cleaning with trisodium phosphate or liquid detergents can be used
- Additive deposits can also be removed with benzyl alcohol or a 50/50 mix of benzyl with isopropyl alcohol.

10.17.4 How to Avoid Venting Problems

Flashing can also be caused by excessively fast closing of the injection machine moving platen and mold, trapping the cavities/runners’ air in dead/deep end sections in the mold cavities. This problem can be reduced by slowing the closing of the mold by controlling the clamping speed.

- Set the return stop position control of the moving platen at a minimum clamp open position to eject the molded parts with a safe working clearance
- Set the transition speed position control of the moving platen 0.50 in from the mold fixed half
- Set the last speed position control of the moving platen between 0.062 and 0.125 in before closing the mold
- Set the platen to high speed and low pressure starting at the mold open position on the control placed 0.50 in from the mold fixed half
- Set the platen to low speed and low pressure for the travel distance between 0.062 and 0.50 in
- Set the platen to very low speed and high clamping pressure during the final travel.

At this stage, the mold is closed by high clamping force and is ready for injection, where the mold vents allow the trapped air and polymer volatile gasses to escape into the atmosphere. However, improper venting can cause flashing or poor weld lines on the molded part. These are usually a sign of late venting, after the polymer has started to become solid. When venting a mold, a slight increase in the shot size is necessary as material may be dragged out of the cavity by the partial opening of the mold. As a rule, if there is no puff of steam or vapor escaping the mold, then the mold vents have not expelled the volatile trapped gasses from the cavities.

Another cause of flashing is the excessive clamping force applied to the mold cavity inserts. This high force closes the parting line cavity vents and seals off the escape of the gasses causing a partial opening of the mold.

Any molding condition that will result in increasing the rate at which the mold cavity will be filled will increase the need for better mold venting.

These are the molding conditions affecting high shear rates:

- Higher injection pressure
- Faster injection speed
- Higher polymer melt temperature
- Higher melt residence time inside the plastifying unit
- Larger injection molding machine melting capacity
- Higher mold temperature
- High compression plastifying screws
- Higher nozzle temperature
- Molding machine hold-up spots in the barrel, screw, check valve, adapter, and nozzle

10.17.5 Planning Mold Venting

A typical product development problem results when the mold venting is neglected during the mold design stage by a forgetful mold designer or during its construction by a hurried toolmaker. A mold with inadequate or nonexistent venting will frequently be the cause of poor part surface appearance and/or performance, high reject rates, and lowered productivity. The project anticipated as a profitable injection molding venture, starts affecting customer production requirements and consuming huge amounts of profit.

Mold venting design should be studied and carried out during initial design and construction of the mold. Location of the mold parting line, the type, dimensions, number and location of the gates, including the best venting system, needs to be addressed. Preliminary mold design layouts offer greater possibilities to install proper vents at correct locations to eliminate trapped gasses inside the mold cavities. This work should be completed before accepting the mold, because during production there is not enough time to take the mold out of the injection molding machine to incorporate the correct venting. Unless the molded parts are rejected for flashing, burn marks, weak weld lines, short shots, poor surface finishing or voids, convincing anyone involved in the molding process that missing vents in a mold can be causing these kind of molding problems is often difficult. An experienced molder knows that gasses trapped inside the cavities are the cause of these problems, but because production demands often allow only a makeshift form of mold venting (a portable grinder, masking tape, reduced clamping force, lower injection pressure), the molder is usually ignored.

One method used to verify that the burning problems on the molded parts are caused by trapped gasses inside the cavities is called the “dieseling effect”. The trapped gasses are compressed very quickly by high pressures and high temperatures, causing the gasses to ignite (sparks). To determine if the mold has venting problems, use a natural color resin for testing the mold, spray an aerosol lubricant such as “WD-40” onto the mold cavity surface, then close the mold and inject the melt using high speeds. The ejected molded part should be blackened in the areas where the gasses and “WD-40” spray were trapped causing an explosion and burning of the molded part.

If an injection mold is poorly vented, molded parts are likely to suffer from weak weld or knit lines, because the trapped gas keeps the melt flow fronts from fusing together when the melt is split. Occasional short shots, internal gas voids or sink marks, burn marks, and surface flow marks can occur. Progressively, these problems become more frequent and severe because improper vents tend to clog with condensed gasses from the polymer.

What is not apparent is the mold corrosion problem that may occur in the cavity's deep pockets or on the surrounding surfaces of the mold base plates or ejector pins. While most polymers' melts may be totally non-corrosive, high melt temperatures, high injection pressures, and fast injection speed cause degradation of the plastic at the point where air and gasses are trapped in the cavities with each shot, causing an internal explosion (dieseling) and, in time, pitting corrosion may result in the tool steel of the cavities.

10.17.6 Mold Venting Process Problems

Inadequate venting or a mold without vents causes many injection molding problems:

- Flashing problems are the most common defects in the injection molding process. Parting line flashing is the result of a fast clamping speed while closing the mold, an excessive injection speed and pressure, high melt and mold temperatures, long residence time of the melt in the plastifying unit, wet material or high moisture absorption of the resin, excessive amounts of reground and contaminated resins. The flashing problem is caused by the high temperature and pressure required by the melt to overcome resistance to flow created by compressed gasses. The flashing problem will not be eliminated by decreasing the mold temperature, because of the fast crystallization of the melt. Venting the mold correctly to remove the trapped gasses is necessary, using the recommended melt temperature, injection speed, and mold temperature. These molding conditions will allow an efficient melt quality, uniform crystallization, faster cycles, and quality molded parts that comply with production requirements.

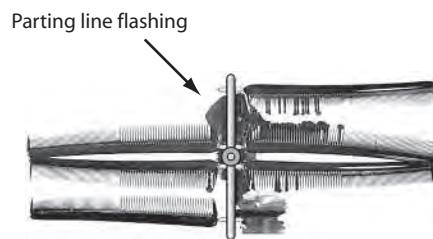


Figure 10-114 Poor mold venting causes flashing molding problems



Figure 10-115 Poor mold venting causes incomplete molded parts

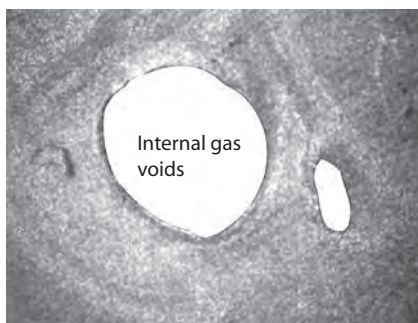


Figure 10-116 Micro structural analysis showing internal gas voids

- Figure 10-114 shows a typical parting line flashing molding problem.
- Short shots are a result of the inability to fill the cavities completely. The main causes of this problem are poor venting or molds without vents. The check valve may not hold pressure, small nozzle orifice, small gate size, low melt/mold temperatures, low injection speed and/or pressure and a thin walled part with long cross sections are also causes.
- Figure 10-115 shows a typical example of a short shot molding problem.
- Internal gas voids are found in thicker wall sections and/or in thin sections surrounded by thick wall sections of a molded part. Figure 10-116 shows a micro structure analysis of a gas void problem found in a molded part. Sink marks and internal gas voids (porosity) in a molded part are caused by a thick walled section molded from a high mold shrinkage polymer at a high mold temperature. This causes the surface of the molded part to crystallize or to become solid (skins), while the melt inside the frozen surface wall is still in a liquid soft stage. Voids are also caused by high melt temperature, a short screw forward time or lack of packing the cavity, check valve leakage, small nozzle orifice, and small gate size.

- Poor weld lines can be developed behind core pins or obstructions that split the melt stream, causing two moving melt fronts to come together during the filling of the mold cavity. Ideally, the melt should fill the mold rapidly enough so that the melt front interfaces are hot enough to cause a strong weld line. Good venting in front of the weld line, proper gate size and location, fast injection speed, and resins without lubricants will insure a strong weld line. Avoid the use of mold release spray in the mold cavity surface; this lubricant is a main contributor to the formation of weak weld lines. When the cavity is inadequately vented, the cavity filling can be slowed if the interfaces cool off excessively, resulting in a weak weld line.

Figure 10-117 shows two micro structure analyses of poor weld line problems found in two thermoplastic molded parts.

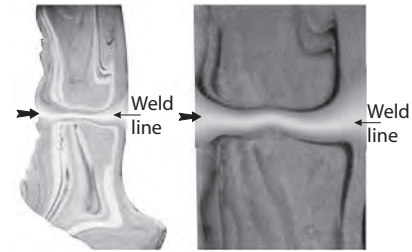


Figure 10-117 Micro structural analyses showing the weld lines

- Burn marks are molding defects of molded parts caused by highly compressed hot gasses that could not escape from the mold cavity fast enough. The molded parts are burned at a location that was the last place to fill the cavity, where the pocket did not have sufficient venting. The compressed trapped gasses increase the mold temperature (diesel effect) causing the burning of the thermoplastic. High melt and mold temperatures, excessive injection speed and pressure are also contributors to this burning problem.

Figure 10-118 shows a burn mark problem found in a thermoplastic molded part.

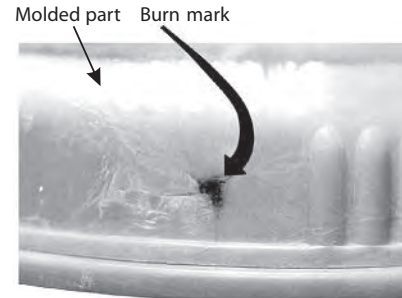


Figure 10-118 Burn mark problem of a thermoplastic molded part

- Corrosion of the metal used to construct the mold cavity insert can be caused by the degraded, decomposed thermoplastic melt produced when the trapped gasses within the mold cavity are highly compressed. This generates temperatures far above the thermal stability of the thermoplastic causing an ignition (sparks) every molding cycle, corroding the mold cavity tool steel.

Figure 10-119 shows a corrosion problem found in a mold core insert.

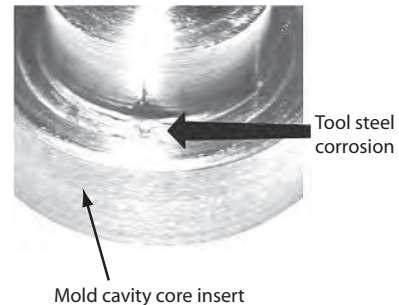


Figure 10-119 Core insert corrosion caused by poor mold venting

- Poor surface or rough finishing on a thermoplastic molded part is caused by the inability to fill the mold cavity quickly. The trapped gas slows down the melt flow speed rate, cooling off the melt because of poor venting or no venting. The process conditions such as low screw rpm, a low screw back pressure, check valve leakage, small nozzle orifice, small gate size, low melt/mold temperatures, and low injection speed/pressure may also cause poor surface finish of the molded parts.

Figure 10-120 shows surface defect problems found in two thermoplastic molded bushings.



Figure 10-120 Two molded bushings showing poor surface finishing

- Mechanical, electrical, and chemical resistance properties can be decreased by poor mold venting. Locally burned areas, gas voids or weak weld lines caused by inadequate mold venting reduce the tensile strength, impact resistance, and dimensional tolerances of thermoplastic molded parts.
- Long molding cycles caused by higher melt/mold temperatures, extended cooling time, long melt residence time, and hold-up spots also cause mold deposits.
- Mold deposits in various compositions accumulate on the mold cavity surfaces. There are different types of mold deposits: they can be clear or opaque and can range from white to dark, waxy to hard. Product geometry, type of resin, mold design, and molding process play an important role in controlling the mold deposit problems.

Figure 10-121 shows a mold deposit problem found in a molded cup.

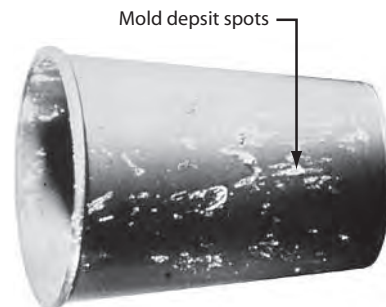


Figure 10-121 Molded cup showing mold deposit problems

Vapor Deposits in the Cavity

The most common deposit develops as vapors from low molecular weight resins react with the trapped gasses in the cavity. For example, when vapors from formaldehyde polymers (acetal resins) are trapped in a mold cavity, deposits usually form at the last place the cavity was filled in a poorly vented mold. The faster the molten resin enters the mold cavity, the higher its mold temperature and the higher the injection pressure, the higher the shear forces applied to melt; these factors contribute to the mold deposit problem.

To reduce the velocity for compressing the trapped gasses in the mold cavity, the following molding conditions are suggested:

- Slow the injection speed
- Reduce melt residence (hold-up) time inside the plastifying unit
- Reduce the melt temperature
- Reduce mold cavities' hot spot temperatures
- Improve the nozzle temperature control
- Improve the design of the mold venting system
- Eliminate molding machine hold-up spots in the barrel, screw, check valve, adapter, and nozzle.

Additive Deposits in the Cavity

Additive deposits are caused when an ingredient in a specific thermoplastic resin formulation separates during processing and forms a film on the mold cavity surface. Additive deposits usually appear at the gate or other flow obstructed areas.

Thermoplastic resins containing plasticizers, stabilizers, lubricants, toughening agents, incompatible pigment concentrates in the carrier master batch in combination with high shear forces are the causes for the separation of additives.

10.17.7 Mold Venting Design

Several areas of the mold can be used to expel trapped gasses. The mold venting efficiency will depend on many factors, such as product geometry or part design, part end use requirements, mold design, injection molding machine, maintenance, process characteristics of the thermoplastic polymer, quality control, molding conditions, process efficiency, productivity, and molded parts manufacturing costs. Mold venting efficiency also depends on the types and number of vents used, locations of the vents, and physical dimensions of the vents used in the mold cavities. Also a factor is the type, geometry, dimensions, number, and locations of the gates; the type and size with a balanced runner layout system properly vented around all cold slug pockets and runner ejection pins. The type of equipment used for grinding or machining the vents, craftsmanship, and careful construction of the mold is also important.

There are several venting methods available for thermoplastic injection mold applications, including parting line cavity venting in line or perpendicular to the ring groove vent. There are two types of cavity insert parting face venting, one horizontal and the other vertical. Both are perpendicular to the ring groove vent.

A stripper plate with ring groove vents is found either at the horizontal parting face position or a taper parting face position (one thirty o'clock).

Ejector rings are also vented like the stripper plate; also used are vented core and ejector pins modified with ring groove vents, vented ejector blades, ejection assist poppet air valves modified for venting the body using a ring groove vent, sintered porous vent, magic seal negative coolant vent, and mold vacuum vent. Optimum venting methods are selected after a complete study of molded part end use requirements, mold venting design methods, cost of the mold, and additional auxiliary molding equipment needed.

The vents should be ground and polished in those areas where the relief edge vent sections will trap air or gas. The air in the mold is at 330 to 380 °F and will get much hotter when compressed by the fast injection speed of the melt, which is hot enough to oxidize and erode the tool steel from the mold cavity and core inserts. There have been instances where enough heat was developed with high injection pressures, fast injection speed and a mold without vents to cause molded parts to burst into flame upon ejection from the mold cavities.

Cavities used for the molding of parts of simple geometry, as for example, bars, discs, or plaques of uniform thickness, do not cause any venting problems. However, by venting thin-walled parts, the trapped gasses are expelled through the parting line cavity vents, parting face cavity insert vents, ejector pins, and runner's cold slug vents.

Deep draw items such as containers, boxes, etc., if center-gated, will not cause venting difficulties. Occasionally, it may not be possible or desirable to center-gate a deep draw item. When such parts are side- or edge-gated and the ejector pin marks are also undesirable, the problem of mold venting is not easily overcome, especially when the mold is already constructed.

Even a well vented injection mold will not last forever and as aging of the mold occurs, some adjustments will need to be made.

An injection mold cannot have too much venting. Molds are most commonly vented at the parting line of the cavity. Unfortunately, this may not be the location where the gasses are trapped. However, several other venting techniques are used to remove the trapped gasses in deep blind pockets.

10.17.7.1 Parting Line Cavity Venting

Parting line cavity venting allows the trapped gasses from the cavities to escape without causing any melt flow restrictions. The thermoplastic melt can fill and pack the cavities well, providing a good surface finishing, strong weld lines, maximum mechanical and electrical properties, fast molding cycles, and optimum molding quality.

Parting line flashing is caused by the internal gas pressure being hotter and greater than the molding machine clamping force, pushing the moving half of the mold apart, releasing the compressed hot trapped gasses through this small opening with the melt causing parting line flashing problems. A well designed mold venting system conveys the trapped air through the runners and gates entering inside the cavities after closing the mold. The trapped air and the polymer volatiles are expelled outside the mold by the melt front high injection speed and pressure.

Parting line venting is accomplished by grinding or machining the mold cavity inserts contact area that forms the mold parting line. The cavity inserts should

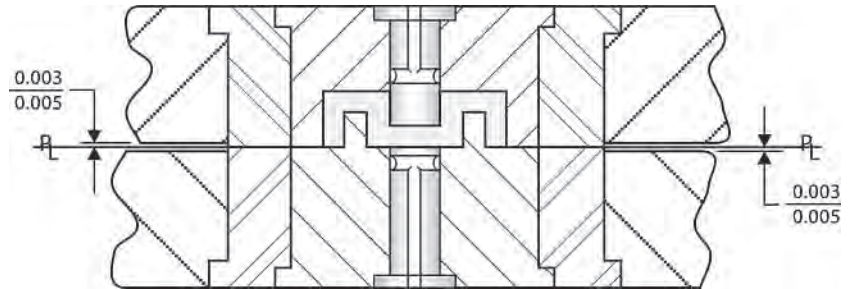


Figure 10-122 Mold base plates and cavity insert mount clearances

be mounted between 0.003 and 0.005 in above the level of the mold base cavity plate. This insert contact area is used to create an effective seal around the cavity, making sure the parting line venting functions without any problems and reducing the chance of flashing the cavity.

Figure 10-122 shows mold base plates and cavity insert mount clearances.

A parting line cavity relief edge vent is a precision engineered opening machined or ground starting at the parting line cavity outside wall until reaching the escape or bridge vent channel.

The cavity relief edge vent consists of the following components:

- *Vent land length.* The length of the vent is 0.040 in. This dimension has been used for venting the molds creating a proper pressure drop to allow the air to convey while stopping the melt flow. This dimension has shown good venting results for many years.
- *Vent land depth.* The depth of the vent is directly related to the polymer melt viscosity and flow rate. A high melt viscosity with slow flow rate requires a deeper vent land channel, than a low melt viscosity with a high flow rate (see Table 10-10).
- *Vent width.* The width of the vent varies with the cavity/runner volume and the number of vents used (0.093 to 0.187 in).

There are two types of parting line cavity vents found parallel to the cavity plane surface; one vent is in line with the escape vent channel, while the other vent is perpendicular to the ring groove vent as shown in Figures 10-123 and 10-124.

A parting line cavity vent (in line) is recommended for the following injection molding applications:

- Irregular geometries of the cavities projected areas
- Single or multiple edge gates
- Melt flow filling and packing the cavity from the edge gate wall to the cross section last to fill; the melt flows inside the cavity like parabolic waves, with the maximum stream velocity in the center and zero velocity at the cavity sidewalls
- Melt flow fills the thicker cross sections next to the gate first and the restricted or thin cross sections last
- Trapped gasses are conveyed directly to the outside of the mold through the parting line cavity escape vent channels.

Parting line cavity vents perpendicular to the ring groove vent are recommended for the following mold applications:

- Symmetrical round cavities projected areas with uniform wall thickness
- Center gates (diaphragm, spider gates)
- Three-plate mold with pin point gates equally spaced concentrically around the center of the cavity
- Cavity uniform melt flow filling
- Number of relief edge vents and bridge groove vents equally spaced and perpendicularly connected to the ring groove vent
- Ring groove vents around the parting line cavity insert border area
- Escape vent channels from the ring groove vent to outside the mold
- Compression strength for sealing the cavities inserts contact area minus venting area is less than 20% of the yield strength of the metal used for the cavity inserts.

Parting line cavity venting design details required for both types of vents and for various types of polymers are shown in Table 10-10 for common types of resins and in Figures 10-123, 10-124.

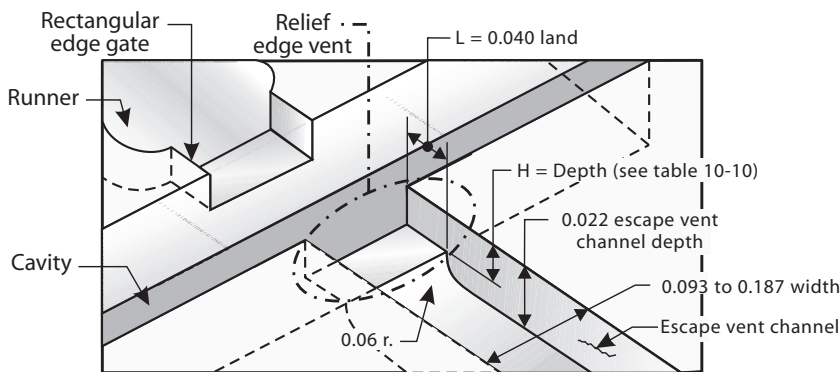


Figure 10-123 In-line parting line cavity venting (in)

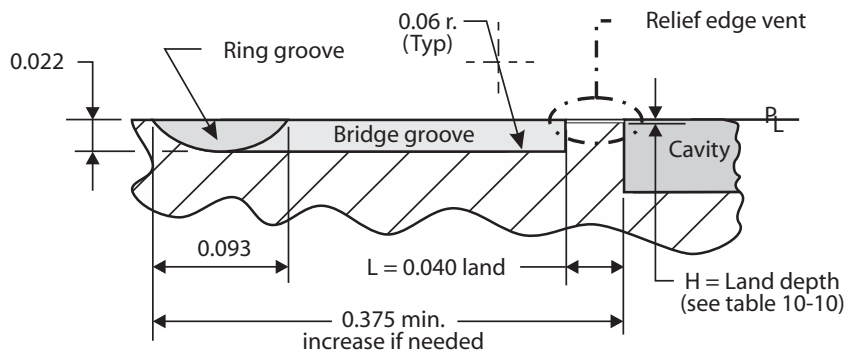
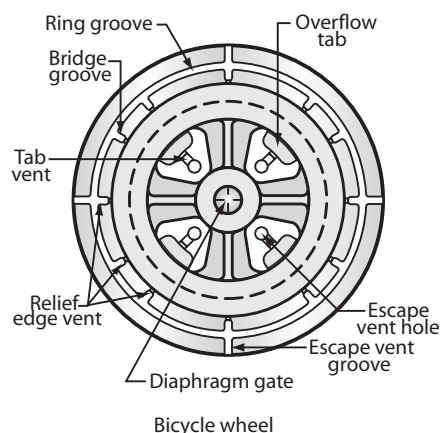
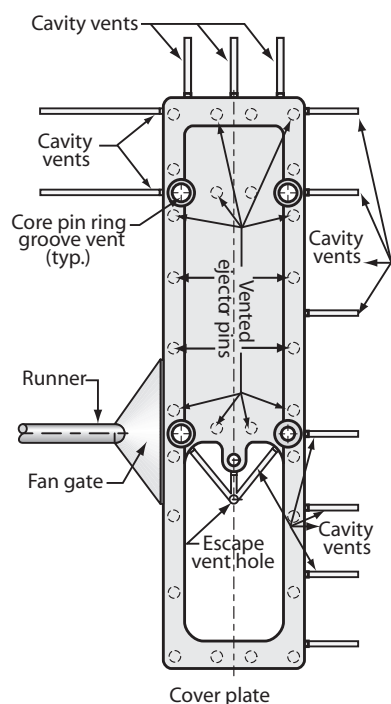
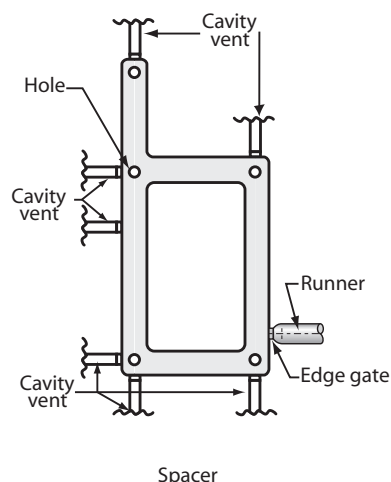


Figure 10-124 Parting line cavity venting perpendicular to the ring groove vent (in)

**Table 10-10** Cavity Relief Edge Vent Depth for Common Types of Resins

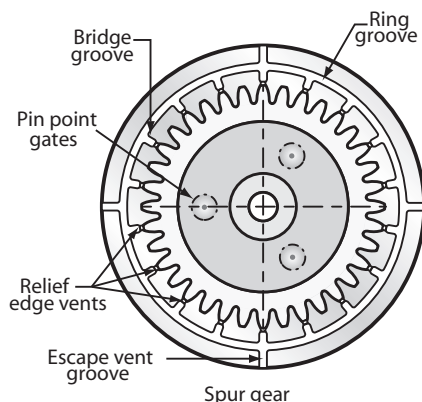
Resin	Depth "H" (in)
Acetal	0.001–0.002
ABS	0.002–0.003
Acrylic	0.002–0.003
Nylon unfilled	0.0005–0.001
Nylon glass reinforced	0.0007–0.0015
Nylon mineral reinforced	0.001–0.002
LCP glass reinforced	0.0003–0.0007
PC	0.0015–0.003
PE	0.001–0.002
PP	0.001–0.002
PS	0.001–0.002
PPO	0.001–0.003
PSU	0.001–0.002
PET glass reinforced	0.001–0.0015
PVC rigid	0.001–0.003
PVC flexible	0.001–0.002

Figure 10-125 shows four mold layout illustrations, where both types of parting line cavity venting systems are used in these mold venting designs.

Mold Cavity Insert Parting Face Venting

Several insert components are used to form a complete vented cavity insert used in mold construction. This method is used when certain forms of a molded product require minor cavity changes to mold another product. Replacement of damaged or broken components of the cavity insert may require replacing only the critical components of the cavity without having to replace the complete cavity, or it can be done when installing vents in the cavities, because machining it as an integral component of the mold cavity is difficult.

These cavity insert components are fabricated separately from special steels or the same material as the cavity, and the inserts are installed in the proper position and become part of the mold cavity insert unit. Often, they are called integral cavity or core inserts. They are also used in the formation of narrow slots to produce vented support ribs on the molded part. One advantage of constructing the cavity with one or several inserts separated from the mold base plate is the cavity insert joint surface walls can be used for the installation of vents to improve the venting system of deep and complicated cavity geometries.

**Figure 10-125** Parting line cavity venting systems

The metal-to-metal contact area (parting line) outside the cavities between the two insert cavities mounted in each half of the mold base must not be reduced excessively by the venting channels. If the cavity inserts are raised above the mold base surface, the insert contact area outside the cavities should be large enough to keep the compressive stress less than 20% of the yield strength of the tool steel used for the mold cavity inserts.

Figure 10-126 shows four cavity insert parting face vent designs using various insert components to form a complete vented cavity insert.

The illustration a) shows a replaceable engraving cavity insert. The vertical parting faces between both inserts can be used to install vents at the deep surface of the cavity and escape vent channels ending at the bore. The side of the bolt hole could be modified to machine escape vent channels.

The illustration b) shows a cavity for a deep and thin ribbed part made by using three additional insert components. These parting face cavity inserts at the bottom rib walls can be used to install vents and escape vent channels.

The illustration c) shows a cavity insert with a bolted replaceable side insert and a bottom insert. The cavity inserts can be used to install parting face vents and escape vent channels.

The illustration d) shows a cavity insert made by using two inserts. The contact area between these inserts could be used at the bottom of the cavity to install parting face cavity insert vents and escape vent channels.

There are three types of parting face cavity insert vents, based on the plane of contact between the inserts: horizontal, vertical, and vertical tapered with respect to the cavity plane surface. The escape vent channel should be connected in line with the parting face cavity insert vents. Figure 10-127 shows the horizontal parting face cavity insert vent details. Figure 10-128 shows the vertical (or vertical tapered) parting face cavity insert vent details.

The vertical tapered parting face insert vent is used for venting through the internal taper hole (matching the core insert shut-off dimensions) of an ejector ring or a stripper plate.

Figure 10-129 shows two different molds and their cavity insert venting systems; these two applications incorporate various venting designs, such as parting face cavity insert venting, parting line cavity venting, and core and ejector pin groove venting systems.

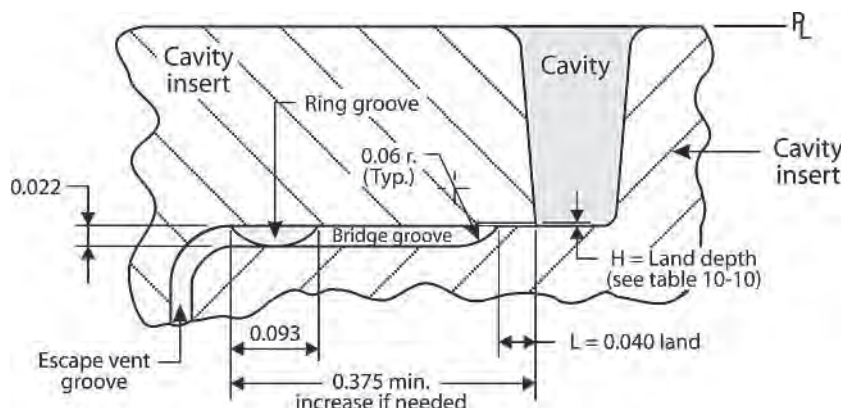


Figure 10-127 Horizontal parting face insert cavity venting (in)

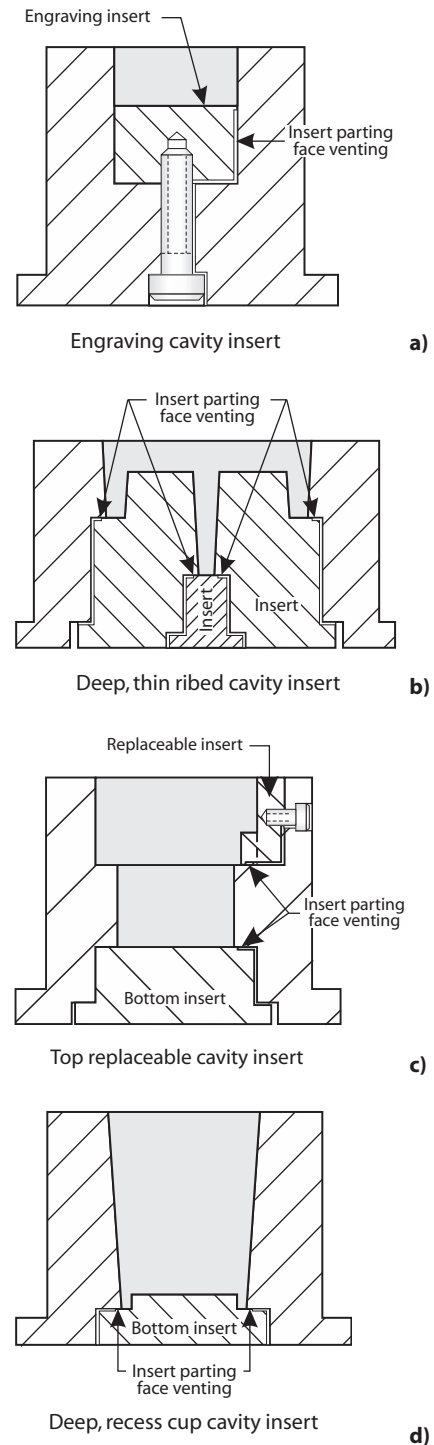


Figure 10-126 Different mold cavity insert parting face venting systems

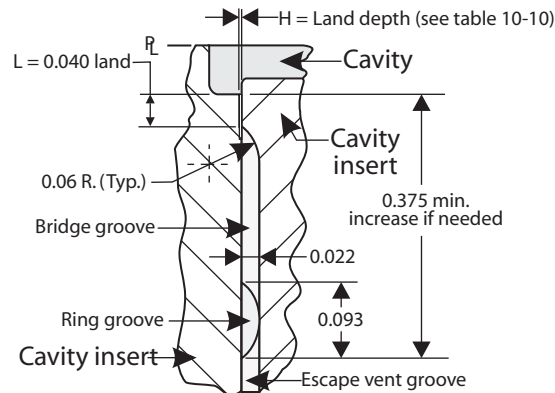
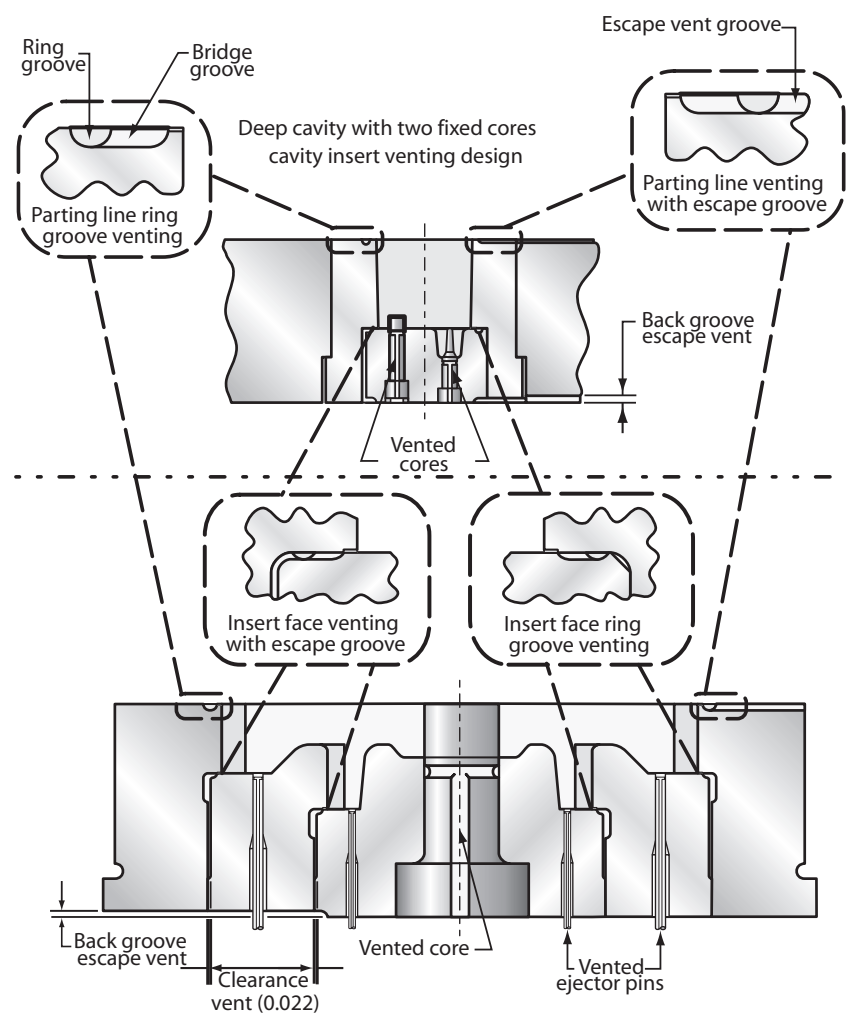


Figure 10-128 Vertical parting face insert cavity venting (in)



Spur gear with two sets of teeth cavity insert venting design

Figure 10-129 Two mold illustrations of cavity insert venting systems

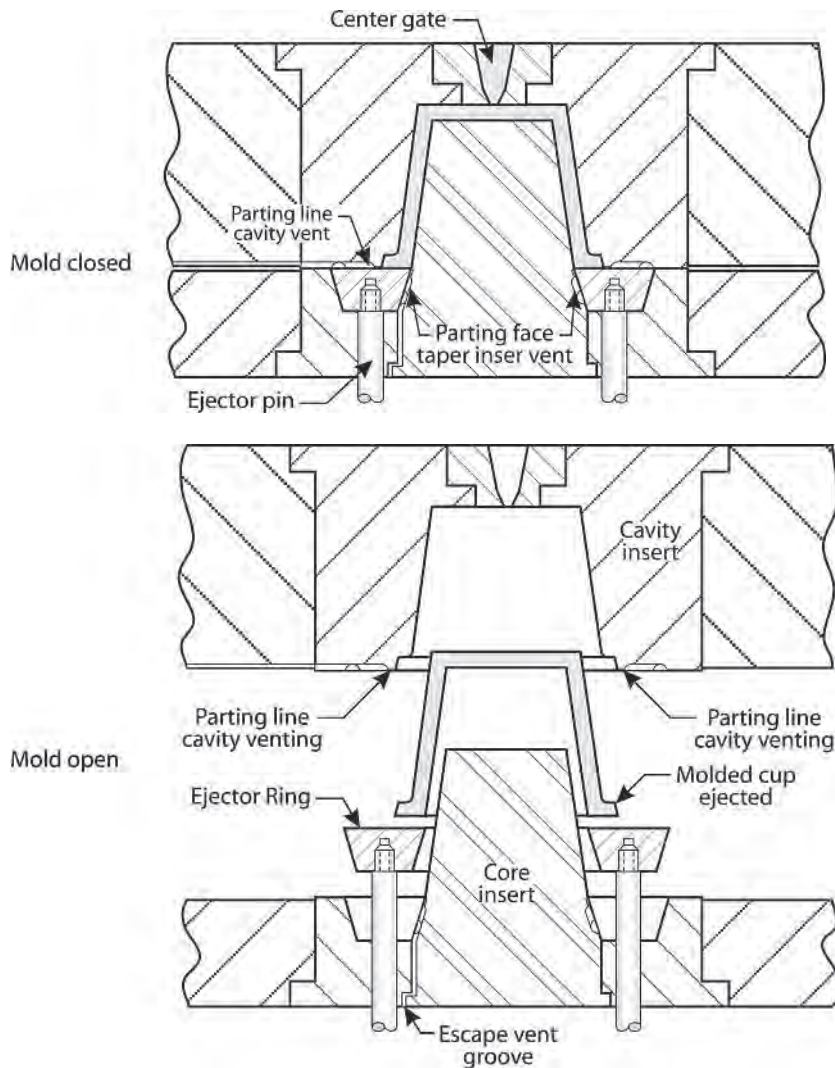


Figure 10-130 Ejector ring and parting line cavity venting system

Ejector Ring Venting System

The ejector ring venting principle is similar to the vertical parting face insert or the stripper plate venting systems. To avoid the use of a large stripper plate, each cavity is provided with an individual vented ejector ring. This system is used to eject deep and thin-walled containers from the cavities. Figure 10-130 shows the parting line cavity venting, combined with a vertical taper parting face insert venting between the ejector ring internal taper wall and the core insert lower external taper wall.

Core and Ejector Pin Ring Groove Venting

The ring groove venting is the most widely used venting technique for round cores and ejector pins. This venting system is similar to the relief edge vent used to vent the cavities. The land (L) is 0.040 in, the depth (H) is the clearance between the ejector sleeve inside diameter and the pin outside diameter, the width is the circumference length of the pin. Round pins, such as stationary core pins and ejector pins, can be modified by machining a ring groove connected with a longitudinal flat on the shaft and head surface of the pin.

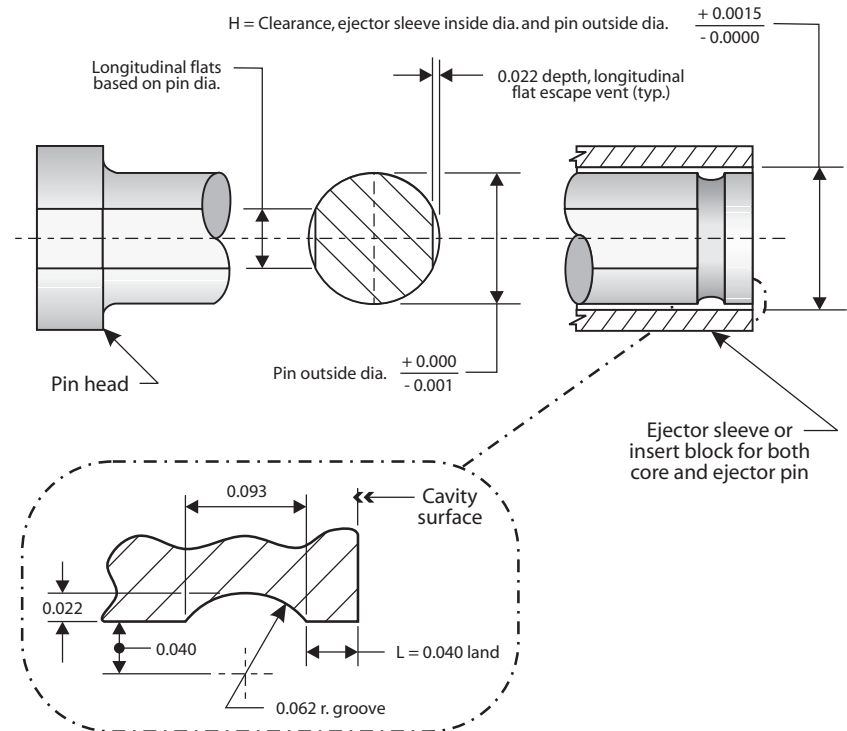


Figure 10-131 Core or ejector pins ring groove venting details (in)

Figure 10-131 shows the design details for stationary core pins and ejector pins. This method allows venting the mold cavity in hard to reach areas away from the parting line. The ring groove vented pin is self cleaning and effective for removing the trapped gasses from the cavity without causing flashing problems.

Figure 10-132 shows a core or ejector pin ring groove vent projection view. For ejector pins ring groove vents, the length of the longitudinal flats should only clear the base of the mold support plate to discharge the trapped gasses away from the mold base.

The ejector pins are precision made of thermal shock resistant H-13 or CM-50 tool steel. The heads are hot forged for uniform grain flow and high tensile strength, then annealed and finished for easy machining the ring groove venting.

A final finishing operation (including ring groove venting) is performed on the full length of the ejector pin to reduce wear and prolong pin life.

The higher core hardness makes the ejector pins ideal for use in high melt/mold temperature injection molding applications.

The following are recommendations for the selection of ejector pins and to improve mold performance for an efficient molding operation:

- Avoid the use of small diameter ejector pins in high volume and critical injection molding applications. Small diameter and long pins wear off quickly and have the tendency to break during molding
- Core hardness of 50–55 Rc reduces nicking, dishing, and bending
- Unchippable surface treatment of 65–74 Rc reduces flashing problems
- Annealed heads (30–35 Rc) and good finishing permit easy machining
- Final surface finish (5–10 RMS) reduces wear and prolongs pin life.

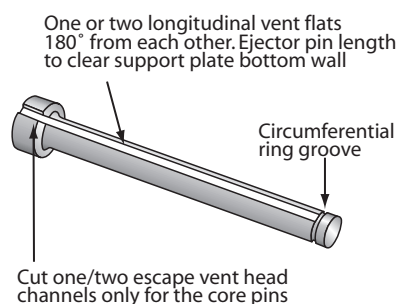


Figure 10-132 Core or ejector pin ring groove vent projection view

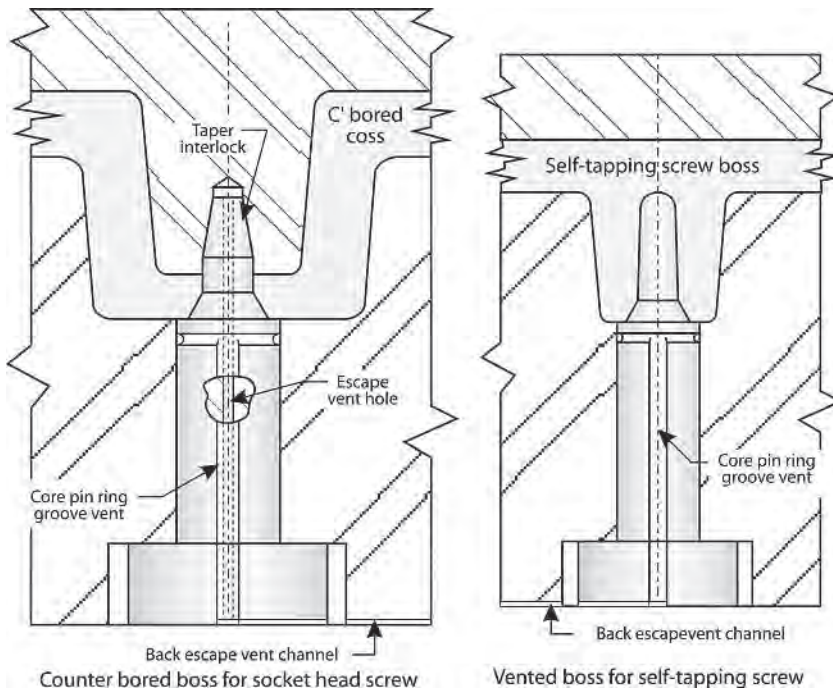


Figure 10-133 Two screw bosses using cores with ring groove vents

Figure 10-133, left illustration, shows how to vent a counterbore hole cavity using an interlock core with ring groove venting; the right illustration shows how to vent a boss (for self-tapping screws) cavity using a ring groove vented core. The vented cores improve the molded bosses' weld line strength, sizes, finishing, and they are completely filled (good density) without burn marks.

Rectangular Ejector Blade Venting

Narrow rectangular ejector blades are used to push out ribs and thin sections of a molded part from the cavity. Ejector blades are thin components that do not have sufficient area to install vents around their sides. The blade fits into the sleeve ejector and is mounted into the ejector plate. As the mold ejector unit is activated, the ejector blade pushes the molded part off the cavity.

One method for venting through these thin ejector blades is to taper grind two flats starting from the top corners, as shown in Figure 10-134.

This venting technique has not been well implemented or proven in thermoplastic injection mold applications. This method could be acceptable for thermoset compression molds, because the transfer melt flow, fill pressure, and mold temperatures are relatively lower than the thermoplastic injection molding process.

Air Poppet Valve/Stripper Plate Cavity Venting

The air poppet valves are designed to help with the vacuum problem often encountered during injection molding of deep draw and thin-walled parts. The air poppet valve prevents mold damage caused by product ejection problems. Air flow (timed to coincide with the ejection cycle) opens or pushes the poppet valve to break the vacuum between the core's deep surface and the inside wall of the molded product, starting the ejection process. To complete the part ejection

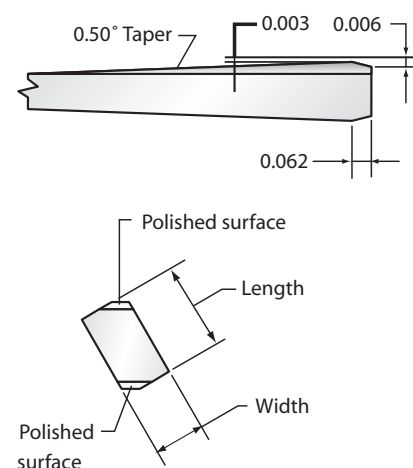


Figure 10-134 Vented rectangular taper flats ejector blade (in)

automatically, the poppet valve should work in coordination with other ejection systems, such as ejector pins, ejector rings, or stripper plates. The air poppet valve should never be used as the sole means of part ejection, or as an alternative to replace other standard ejection methods.

The precision ground poppet valve body outside diameter can be modified by machining a ring groove and escape vent channels to provide venting at the deep section of the cavity, while the poppet valve seat prevents the melt from entering the valve during injection. Each poppet valve is matched to the body to assure reliable performance.

Air Poppet Valve Operating Conditions

The poppet valve air pressure and machine ejection should be activated simultaneously to relieve a vacuum build up in the cavity during part ejection. The air flow to the poppet valve must be fully released to the atmosphere after each cycle to insure that the poppet valve closes. Thermoplastics melt injected into a partially open poppet valve could cause damage to the valve and/or the mold.

Stripper plate ejection is generally used where ejector pin marks would be objectionable on the molded product and where a maximum ejection surface is required. Tapered parting face vented stripper plates are used on single and multiple cavity molds to eject round or rectangular parts from the male core

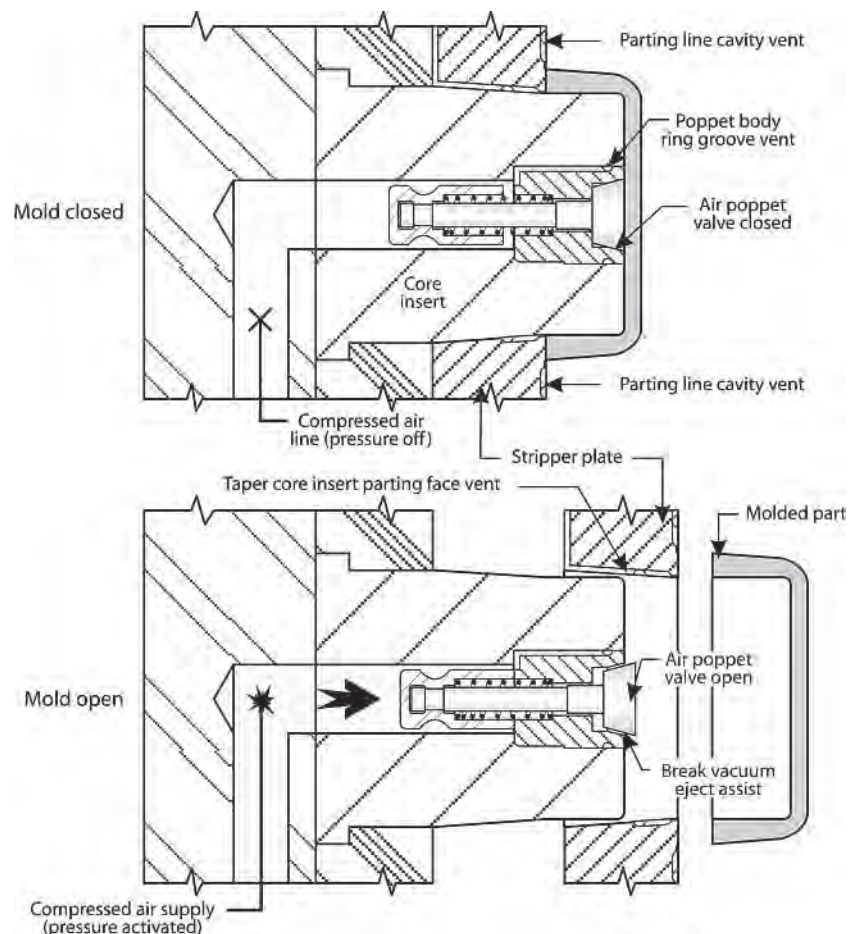


Figure 10-135 Air poppet valve/stripper plate cavity venting system

cavity inserts. An angle of 5° is machined in the stripper plate parting face; the vent is located between the stripper plate internal tapered wall and the insert core lower walls.

Figure 10-135 shows how to vent the air poppet valve and the stripper plate to avoid venting problems, automatically ejecting the molded parts without defects or interruptions in the molding cycle.

UniLifter® Internal Undercut Ejection Venting

Standard UniLifter® components (undercut releasing system) simplify mold design and construction for release of molded products with undercuts. A variety of sizes of standardized components are used in many mold applications. The smooth travel of U-Coupling in T-Gib eliminates heel binding. The radiused dovetail design lets the core blade seat automatically at the required angle.

The UniLifter® ejector blades are made from H-13 tool steel with a hardness of 38–42 Rc, designed for conventional machining and grinding of the vents at the upper surfaces below the lower cavity surface.

The type of venting used for this system is determined by the cross section of the ejector. For rectangular blades, thickness and width is supplied surface ground with a maximum of 0.010 in additional stock over nominal for fitting into the insert pockets and/or to fit a nominal size product. For round core blades, diameters are supplied with a tolerance of (+0.000/–0.001) in for fitting in a bored hole or bushing.

The following are mold design guidelines for the UniLifter® system:

- *General installation.* It is recommended that the UniLifter® be installed with T-Gib mounted on top of the ejector plate
- *Angles.* Designs using angles between 5° and 10° will typically yield the best results
- *Lifter guides.* UniLifter® guides are recommended for designs with angles of 15° or whenever less than half the core blade is bored in the core insert
- *Guided ejection.* It is recommended that guided ejection be used in all mold designs
- *Core blade venting.* Vents should be ground in all permissible upper surfaces below the lower cavity surface. Venting should be added after establishing the final core and guide dimensions. Vent dimensions for different types of polymers are provided in the cavity parting line or ejector pin ring groove venting detail drawings shown in Figures 10-123, 10-124, 10-127, 10-128, and 10-131.
- *Fit and finish.* Recommended clearance for a core blade ranges between 0.001 and 0.0015 in where permissible. Additional performance can be obtained by applying a hard surface coating after the machining operation is completed (Ti N coating, chrome plating)
- *Locking angles.* Locking angles may be designed in the mold if required to provide a locking surface to balance the cavity projected area force caused by the injection molding pressure.

Figure 10-136 shows a mold venting application using the UniLifter® internal undercut ejection venting system.

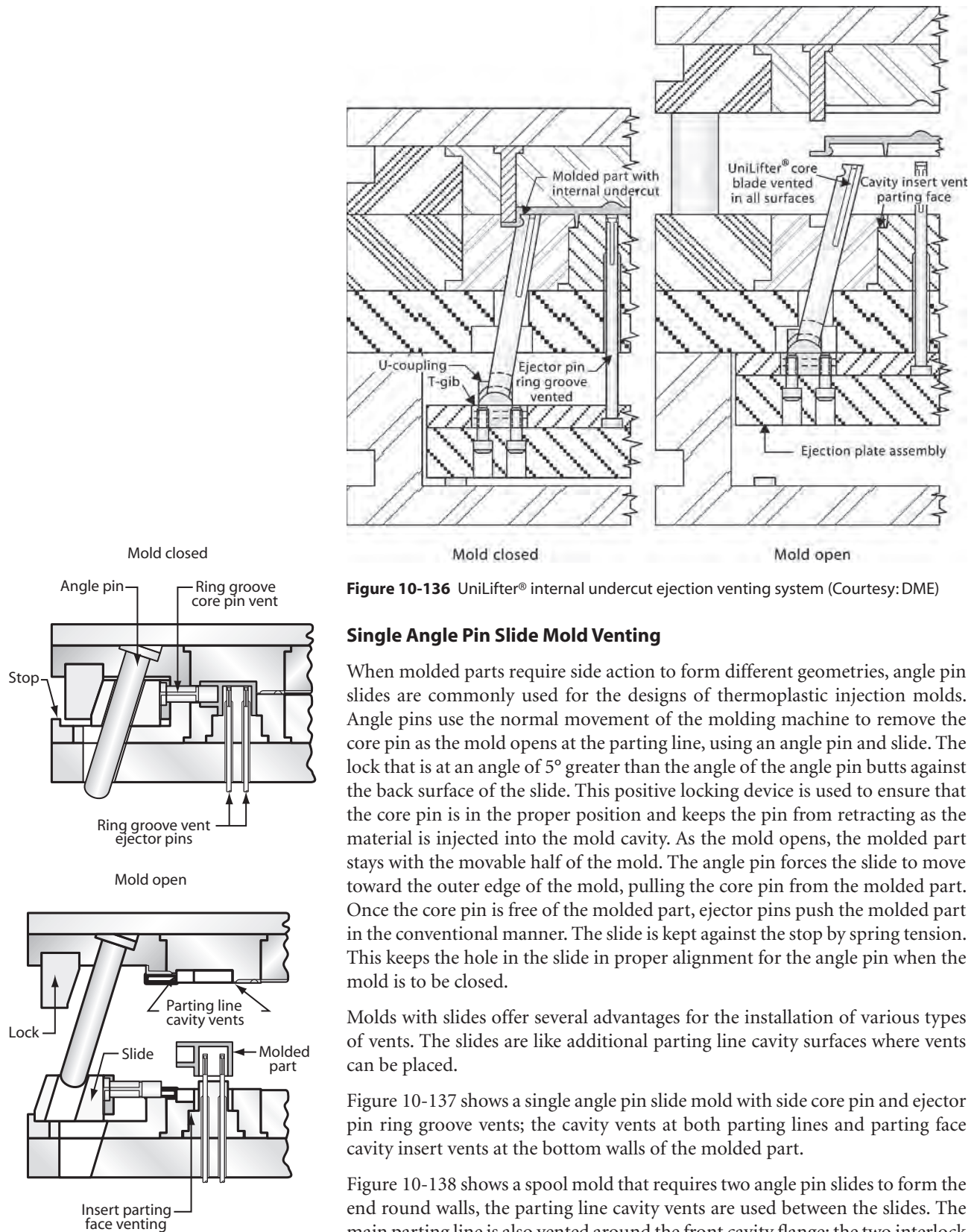


Figure 10-136 UniLifter® internal undercut ejection venting system (Courtesy: DME)

Single Angle Pin Slide Mold Venting

When molded parts require side action to form different geometries, angle pin slides are commonly used for the designs of thermoplastic injection molds. Angle pins use the normal movement of the molding machine to remove the core pin as the mold opens at the parting line, using an angle pin and slide. The lock that is at an angle of 5° greater than the angle of the angle pin butts against the back surface of the slide. This positive locking device is used to ensure that the core pin is in the proper position and keeps the pin from retracting as the material is injected into the mold cavity. As the mold opens, the molded part stays with the movable half of the mold. The angle pin forces the slide to move toward the outer edge of the mold, pulling the core pin from the molded part. Once the core pin is free of the molded part, ejector pins push the molded part in the conventional manner. The slide is kept against the stop by spring tension. This keeps the hole in the slide in proper alignment for the angle pin when the mold is to be closed.

Molds with slides offer several advantages for the installation of various types of vents. The slides are like additional parting line cavity surfaces where vents can be placed.

Figure 10-137 shows a single angle pin slide mold with side core pin and ejector pin ring groove vents; the cavity vents at both parting lines and parting face cavity insert vents at the bottom walls of the molded part.

Figure 10-138 shows a spool mold that requires two angle pin slides to form the end round walls, the parting line cavity vents are used between the slides. The main parting line is also vented around the front cavity flange; the two interlock core pins are vented through the ring groove vent channels. Parting face venting is used between the slide base and the mold cavity plate surfaces.

Figure 10-137 Single angle pin slide mold venting system

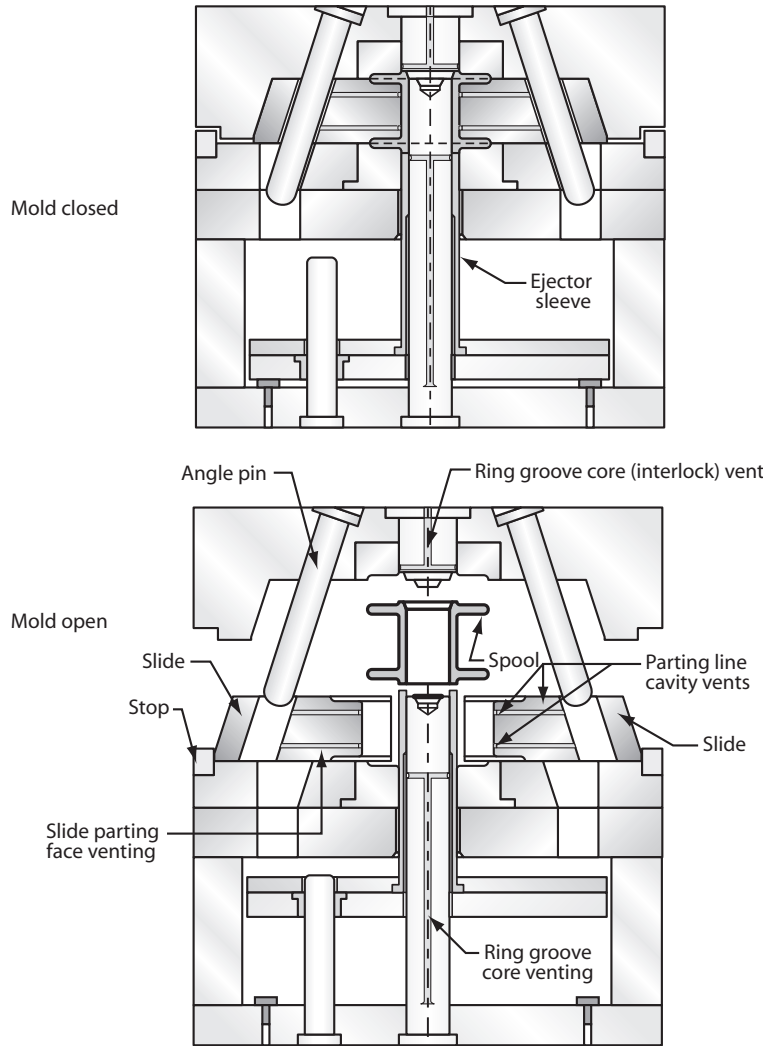


Figure 10-138 Double angle pin slides mold venting system

Cold Runner Venting System

The runner system begins at the machine nozzle tip radius with the mold sprue bushing, where the polymer melt front solidifies forming a skin around the forward orifice of the nozzle during the molding process. The quantity of solidified polymer melt front can be reduced by the proper design and temperature control of the nozzle tip. Operating the nozzle tip front inside orifice as a thermo valve controls the sprue small diameter extension length inside the nozzle.

When injecting the mold cavity at a high rate of speed, a runner system without vents can be responsible for unbalanced filling of the mold cavity causing burns and corrosion of the cavity metal. Since the volume of melt or shot size moving through the mold is in part dependent on the melt resistance to flow, runners without vents will slow down the mold cavity filling rate. Venting the runners and sucker pin puller of a three-plate mold is as important as a two-plate mold.

The first cold slug pocket of the runner system is the sprue puller pin with ring groove vent channels. If an opening for the cold slug cannot be found at the sprue puller pocket, the runner system should be designed with additional safety

protections. The sprue and all runners should have cold slug pockets, ejector pin ring groove vents, and runner vents. These items are used to catch the melt front cold slug and to discharge the gasses inside the mold runner system to the atmosphere. Without these items, the polymer melt front cold slugs may plug small runners, gates, or the cavities and runner vents.

Venting the runner system is much more important than the sprue because all air left in the runner system must be exhausted through the cavity vents.

Raising the cavity inserts between 0.003 and 0.005 in above the level of the mold base cavity plates is standard practice with some mold makers to obtain a better seal around the cavity and reduce flash.

A sprue puller is the least important area, because it is effective only in exhausting some air in the sprue. It is important for very long sprue bushings feeding small runners and low volume cavities.

The following guidelines help avoid venting problems:

- Install a generous pocket extension length between 0.25 and 0.50 in at the end of all runners.
- Machine parting line vents from the end of each runner cold slug pocket to the outer edge of the mold. The size of these secondary runner cold slug pockets and vented ejector pins should be the same as the cross section area of the runner diameter.
- The mold runner parting line vents must not allow flashing.
- The area of the runner vent (land length and land depth) must be large enough to prevent a rise in gas pressure in the mold cavity.
- Install vented ejector pins in the runner channels and gate pullers.
- Roughly grind the mold base cavity plates.
- The land of the vents should be polished and have the same hardness as the cavities. In addition, the cavities and land of the vents could be chrome plated after the mold has been tested and approved for production.
- The runner's escape vent channel's depth is used to convey the gasses outside to the atmosphere. The channel depth starts at the end of the runner vent land length to the external edge of the mold.

Table 10-11 Runner Relief Edge Vent Depth for Common Types of Resins

Land depth "H" (in)	Resin
0.0015–0.0025	Acetal
0.0025–0.0035	ABS
0.0025–0.0035	Acrylic
0.0010–0.0015	Nylon Unfilled
0.0015–0.0020	Nylon Glass R.
0.0015–0.0025	Nylon Mineral R.
0.0010–0.0015	LCP Glass R.
0.0020–0.0035	PC
0.0015–0.0025	PE
0.0015–0.0025	PP
0.0015–0.0025	PS
0.0015–0.0035	PPO
0.0015–0.0025	PSU
0.0015–0.0020	PET Glass R.
0.0015–0.0035	PVC Ridged
0.0015–0.0025	PVC Flexible

Figure 10-139 shows the runner parting line venting design and Table 10-10 provides the values for the runner relief edge vent depth for common types of resins.

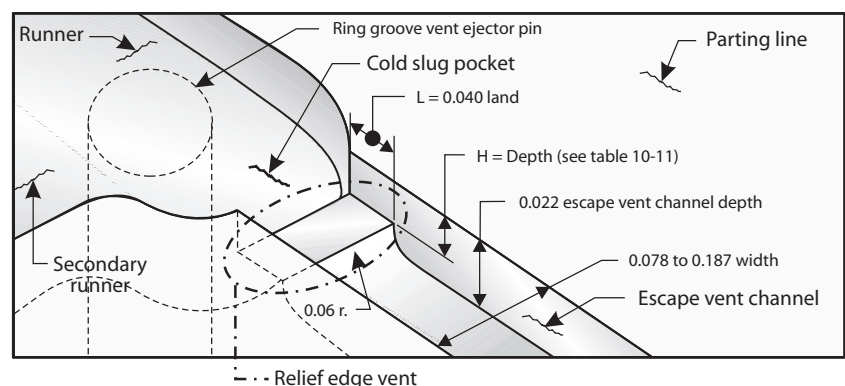


Figure 10-139 Runner cold slug pocket parting line venting (in)

Figure 10-140 shows various types of ring groove vents for sprue puller pins, runner/gate ejector pin vents, and three-plate mold runner venting systems.

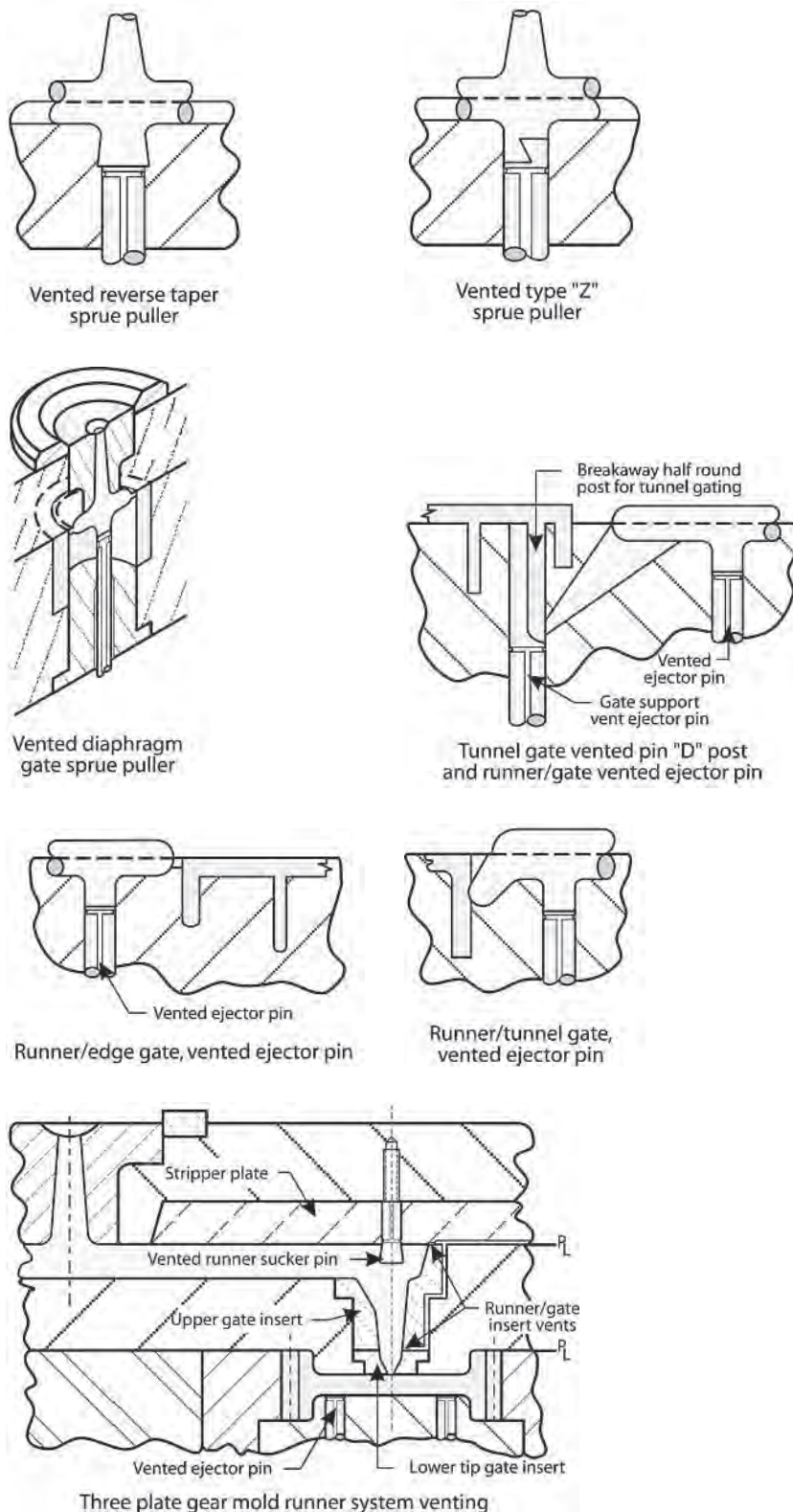


Figure 10-140 Additional cold runner system locations for venting

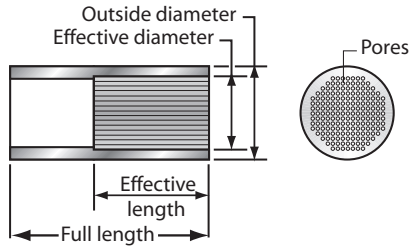


Figure 10-141 Sintered porous insert plug mold venting (Courtesy: DME)

10.17.8 Mold Venting Using Sintered Porous Insert Plugs

Sintered vents are unique venting plugs composed of enough straight, parallel, and uniform pores made by a powdered metallurgy process. The pores allow trapped air or gas to escape from the mold cavity during the injection molding process, reducing the occurrence of defective parts. However, based on vent pore diameter and thermoplastic polymer melt characteristics, the pores usually fill with mold deposits or other substances quickly, becoming inoperative and requiring maintenance. Sintered vent plugs should not be used in injection molding applications requiring mold cavity temperatures above 180 °F.

Figure 10-141 shows a sintered porous self-contained standardized vent plug insert (DME) used for venting thermoplastic injection molds.

The porous vent plugs are characterized by

- Number of pores
- Pore diameter based on resin viscosity
- Porosity % of effective diameter
- Effective diameter
- Effective length

Advantages of Sintered Vent Plugs

- Venting the air from the mold or removing the trapped gasses by using sintered vent plugs reduces the occurrence of molding process problems such as short shots, flashing, and burn marks of molded parts
- Self-contained standardized vent plugs save engineering time in mold design, installation, and maintenance
- A variety of standard size sintered vent plugs is commercially available
- Fast and easy replacement or cleaning of sintered vent plugs from the production molds improves the process efficiency and productivity
- Standard size sintered vent plugs have been tested in the field to ensure product reliability

10.17.8.1 Types of Sintered Vent Plugs for Different Polymers

Sintered vent plugs with a 0.03 mm vent pore diameter should be used with high melt flow rate polymers, such as nylon, high melt flow acetal, PBT, PET, polyethylene, or polypropylene. Sintered vent plugs with a 0.05 mm vent pore diameter are recommended for medium melt flow rate polymers, such as polycarbonate, PPO, PS or ABS. When injection molding high molecular weight, high viscosity polymers (low melt flow rates), such as HMWPE, high viscosity acetal, high impact resistance nylon, acrylic, or TPE, select the sintered vent plugs with 0.10 mm vent pore diameter.

Stainless steel sintered vent plugs are recommended for polymers that have heat sensitive process characteristics, short melt residence time, and high processing temperatures.

Melt degradation causes a chemical reaction discharging gaseous fumes. The melt sticking to the mold cavity walls is corrosive to the steels used for the construction of the molding machine and to the tool steels selected for the fabrication of the molds. Especially damaging are such polymers as PVC, acetal contaminated with

PVC, PC flame retardant, or PTFE. Stainless steel sintered vent inserts are also recommended for flame retardant thermoplastic compounds or thermoplastics using unstable thermo colorants.

10.17.8.2 Installation Procedures

- The press-fit is 0.01 to 0.02 mm for outside diameters of 10 mm or less, and 0.015 to 0.035 mm for outside diameters more than 10 mm.
- Use a plastic or wooden hammer for installation. Do not tap the pore surface of the sintered vent plug with a metallic or hard tool. The use of hard tools will result in clogging or chipping of the sintered vent plugs.
- Do not grind, machine, or cut the pores of the sintered vent plugs.

Figure 10-142 shows a mold design application using a sintered vent plug to remove the trapped gasses from the mold cavity.

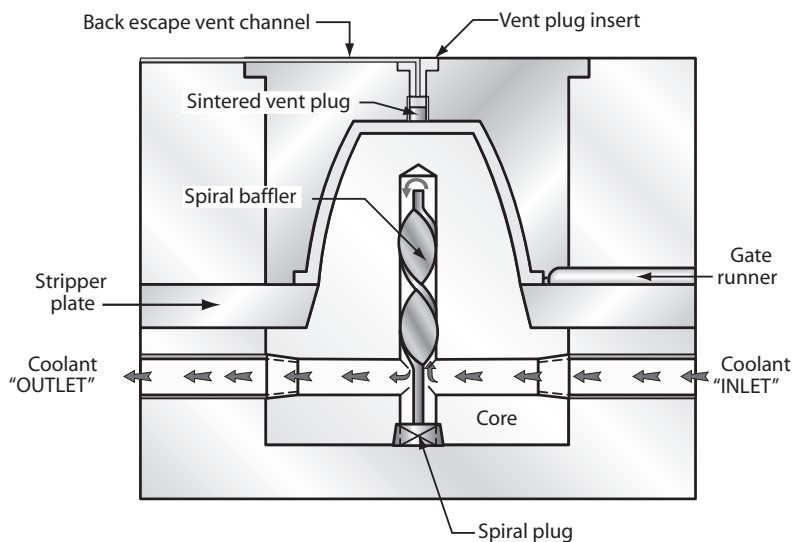


Figure 10-142 Sintered vent plug insert mold venting application

10.17.9 Logic Seal (Negative Coolant Pressure) Mold Venting

This type of venting makes use of the negative pressure found in the mold temperature controller (vacuum circulator) causing a vacuum (suction force) to remove the trapped gasses from the mold cavity. This mold venting system, also known as “Leak Stopper”, was developed and manufactured by Logic Seal.

Inadequate mold venting poses a major molding problem when the gasses are trapped inside the cavities. The mold must eliminate the air trapped in the cavity after the mold closes, and also the gasses generated by the thermoplastic melt itself. As the melt enters the runner and cavity, the melt must displace these gasses pushing them ahead of the melt front. If these gasses are compressed they may exit through the mold parting line, cores, and inserts causing flashing problems.

Designing molds using negative water cooling pressure to vent the mold has several advantages: cycle time improvements, faster injection speed, lower melt/mold temperatures, better injection molding processing, and reduction of rejected molded parts.

The introduction of new thermoplastic engineering resins has made mold venting more difficult. As they are processed, these new compounds produce more gasses than the old resins, because they contain increased levels of fillers, reinforcements, stabilizers, plasticizers, impact modifiers, pigments, and other additives, all of which can add to the amount of compressed trapped gasses that cause mold venting problems.

10.17.9.1 Logic Seal Venting Through Sintered Insert Plugs

A solution for improving the mold venting problems is to increase the mold venting area and to control the mold temperature required by the thermoplastic polymer.

The Logic Seal venting technique uses sintered vent plugs that allow the cavity's trapped gasses to pass through the sintered vent plugs, but not the thermoplastic melt. The negative coolant pressure regulates the mold temperature and cools the sintered porous plug directly without causing coolant leakage inside the cavity. If the sintered metal inserts are not cooled, they overheat causing the sintered vent pores to fill with mold deposits or other substances and quickly become inoperative.

Figure 10-143 shows a typical mold application using sintered vent inserts for removing cavity gasses. Mold venting using negative coolant pressure and sintered vent inserts means a deep draw cavity can be center-gated at its base, allowing the top to retain a perfect finish. Several sintered metal inserts are pressed into the vertical wall of the core insert, forming the inside wall of the molded part.

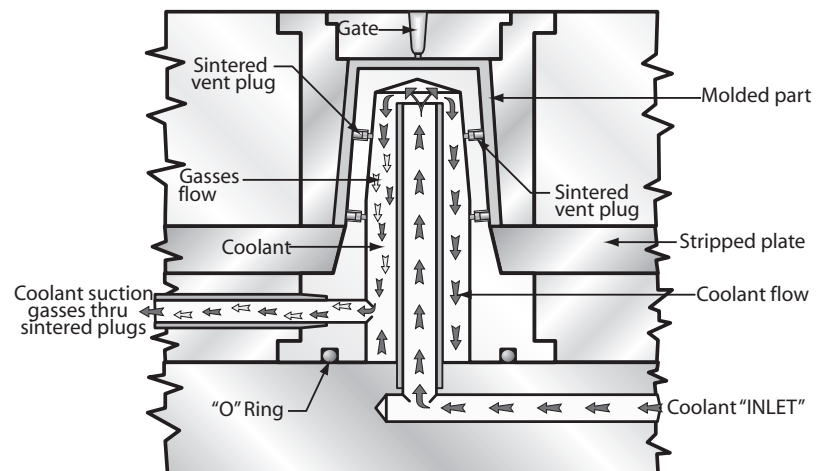


Figure 10-143 Logic seal sintered plug mold venting application

These sintered metal insert vent plugs are connected with a discharge orifice to the negative pressure waterline of the core, bringing coolant right up to the sintered vent plugs, always keeping them at the same or close to the coolant temperature. The sintered vents do not overheat or become plugged.

10.17.9.2 Logic Seal Cavity Parting Line Cores and Ejector Pin Venting

Another mold venting method uses vented cores and ejector pins (with ring groove vent channels). These pins are mounted directly in the middle of the negative pressure waterline. This method has several advantages: because the

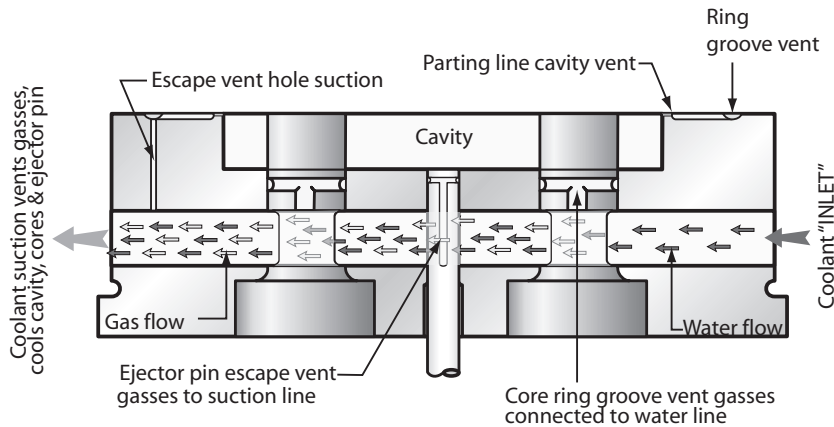


Figure 10-144 Logic seal cavity parting line, core and pin venting

vented pins run directly through the coolant, the pins will not overheat. Standard pins without direct contact with coolant generally have higher temperatures than the cavity surface temperature. The core and ejector pin ring groove vents will not become plugged with mold deposit or other substances reducing the mold venting performance.

Another important advantage is that the tool makers are not constrained by the placement of pins, which can be located for maximum venting and ejection effectiveness. The cooling water negative pressure flows around the pins providing for maximum cooling and the vacuum suction to expel the trapped gasses from the runner system and mold cavity into the cooling water line. Tool makers, already familiar with the conventional technique of venting cores and ejector pins by machining ring groove vent channels to convey the gasses to the atmosphere, may feel more comfortable with this method before applying the mold venting technique through sintered vent plugs.

Figure 10-144 shows a typical mold application using cavity parting line vents, vented cores, and ejector pins for removing the trapped gasses with negative coolant pressure through mold vent channels.

10.17.10 Mold Cavity Vacuum Venting System

Mold vacuum venting systems are widely used for thermoplastic injection molds. This system eliminates the trapped gasses from the mold along with volatiles that may be generated by the thermoplastic melt during the molding process. This method has several advantages: a properly constructed and working system will allow the fill of dead end sections of the mold cavity since there will be no gasses to impede the thermoplastic melt flow. Poor weld lines and molded-in stress concentrations are also reduced as there is no excessive heat build up by the dieseling effect of compressed gasses and there is no need to wait for the gasses to escape from the mold cavities.

The molds can be run at higher temperatures; as the thermoplastic melt is injected faster into the cavities, there is more frictional heat developed at the gate, decreasing melt viscosity, which also promotes a faster filling time. While the mold vacuum venting system will not show impressive gains on all injection molding applications, this technique has proven to facilitate faster injection cycles, to eliminate molding reject parts, and given better quality control.

The process is quite simple and inexpensive when molders have vacuum lines available to each molding machine. Portable vacuum systems are used when the unit is moved from machine to machine as required. A commercial mold vacuum venting unit is self-contained, with all controls and gauges mounted in a common panel for immediate installation and operation. Installation requires mounting the unit on the injection molding machine or on a separate stand for portability. It is recommended to connect the mold vacuum venting to the mold using a Teflon® steel braided hose when mold temperatures exceed 325 °F and to mount the two limit switches on the mold, platens, or rear rails of the injection molding machine that automatically begin the vacuum and blow back cycles.

A vacuum gauge should be mounted as close to the mold as possible next to the molding machine to measure the vacuum. Leaks in flexible hoses or connections may give a faulty picture of what is happening in the mold, if the gauge is on the vacuum reservoir. A flexible hose connects to a trap mounted between the mold and the reservoir to keep the thermoplastic melt volatiles out of the vacuum pump oil. The vacuum reservoir is required because most vacuum pumps do not have the capacity to obtain the necessary pressure drop in the time it takes to fill the mold. Mounting the sealing grooves and vacuum lines on the stationary half of the mold is advisable.

The sequence of operation is to close the mold and when the reciprocating screw/check valve stops plastifying the resin and moving it forward for injection, to start the vacuum system by a limit switch or a manually operated valve. When the screw forward time (filling and packing the cavity) is completed, the vacuum is turned off. With injection, the system is activated when the screw starts forward. The vacuum pump runs constantly to maintain the vacuum in the reservoir.

10.17.10.1 Required Mold Modifications

Minor mold modifications are required to use the mold vacuum venting system. A mold parting line venting installation consists of an “O” ring and a ring groove vent channel around the cavities. The “O” ring is required to seal the mold and prevent ambient air from entering the cavities, which, in turn, helps to create the vacuum. The gasket material should be Viton®, silicone rubber, or other materials that will withstand the mold temperature and pressure without melting or crushing. The gasket material should be soft enough to conform to the other half of the mold and provide a seal, but at the same time not hold the mold open. Between the sealing ring and the mold cavity is another ring groove channel tied into the bridge groove channels and the cavity relief edge vents.

The most common method is to use a vacuum line, a ring groove vent channel, bridge groove channels, cavity relief edge vents, and “O” ring. This simply involves cutting a groove around the mold cavity face 0.105 in deep with a 0.135 to 0.140 in end mill to install a 0.125 in diameter “O” ring (the larger the diameter the better its performance) and a ring groove vent connecting the cavity relief edge vents. The ring groove vent channel should also be as deep and wide as possible. A 0.093 in wide by 0.022 in deep machining with a 0.125 in diameter end mill is recommended.

To keep the “O” ring in place in a parting line venting application, it can be glued in the mold cavity insert by using RTV (room temperature vulcanizing) silicone. A Neoprene® “O” ring can withstand temperatures up to 160 °F; a Viton® “O” ring can withstand temperatures up to 420 °F.

Figure 10-145 shows the mold vacuum venting cross section and top view of a mold lower half parting line cavity vacuum venting.

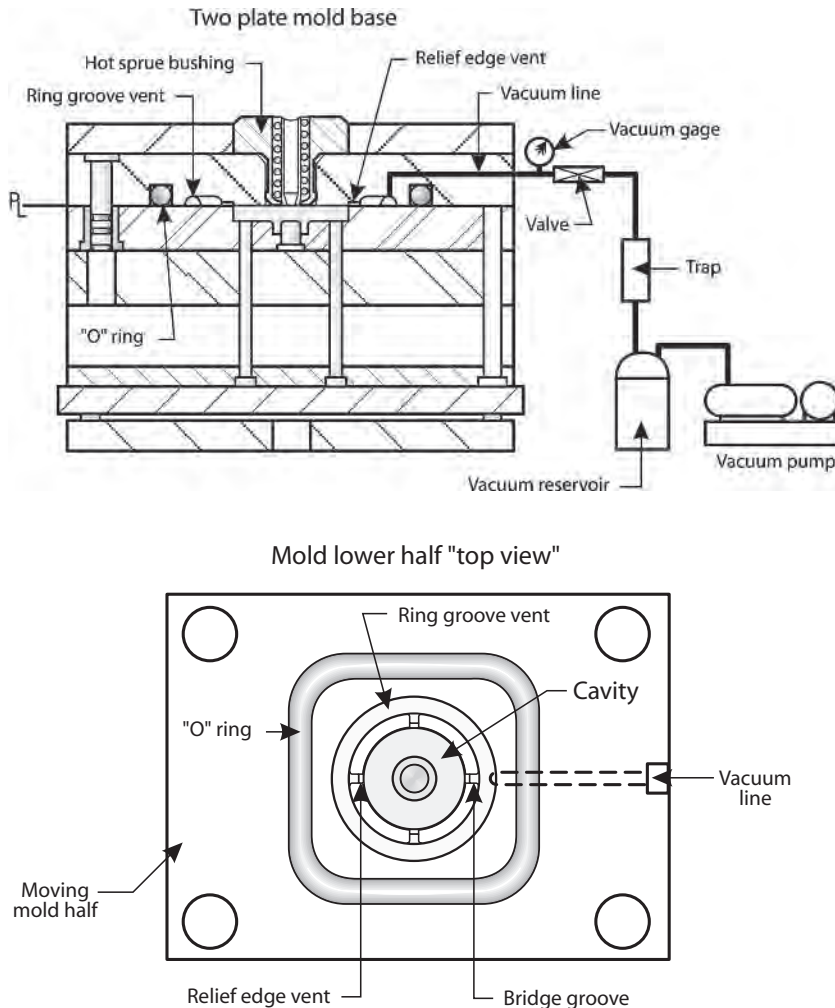


Figure 10-145 Mold vacuum parting line venting system

For a thermoplastic injection molded product with deep thin ribs, a second mold cavity insert is used to form the rib and to provide a contact area between the insert walls to machine desirable parting face cavity insert vents. This vent should be connected to the mold vacuum unit close to the bottom of the cavity insert.

It is recommended that both vertical walls of the mold cavity insert be sealed by using Viton® "O" rings or be coated with a fine coat of silicone grease or RTV silicone to prevent vacuum leakage.

Figure 10-146 shows how the mold vacuum venting technique is accomplished by using the parting line cavity vents; the mold parting face cavity insert vents are connected to the mold vacuum venting unit.

Another mold vacuum venting method is the use of a modified ejector pin machined with ring groove vent channels connected to the mold unit vacuum line.

This mold vacuum venting technique requires that the vented pin be sealed at the support plate of the mold to prevent vacuum leakage. A Viton® "O" ring seal is recommended due to its durability and heat resistance.

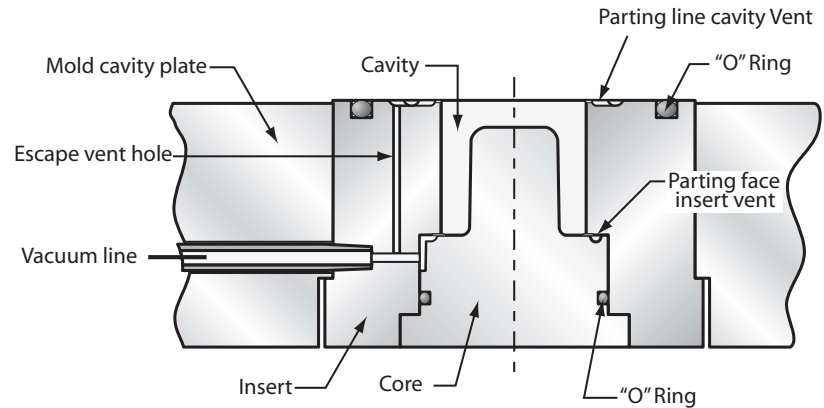


Figure 10-146 Parting face insert vacuum venting system application

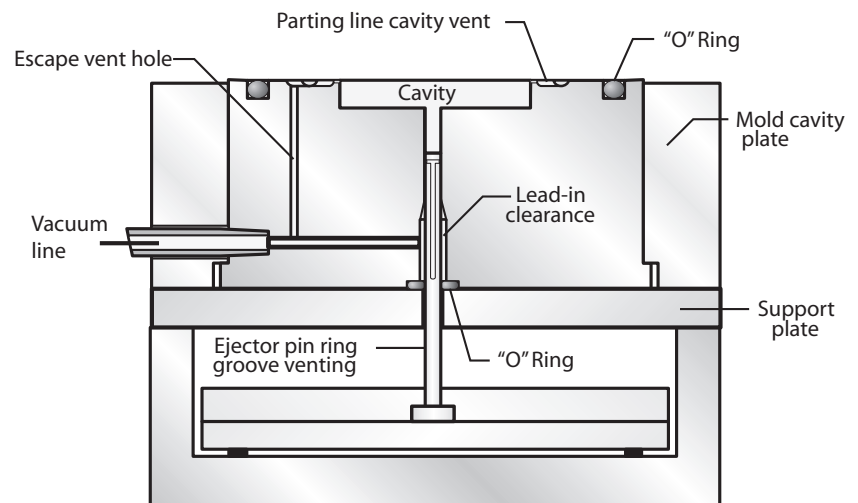


Figure 10-147 Mold vacuum venting a cavity blind hole by ejector pin

Figure 10-147 shows a mold cavity with a deep blind hole in the middle of the cavity. Venting this part is accomplished by having a vented ejector pin with ring groove vent channels connected to the mold vacuum venting unit.

10.17.10.2 Evacuation Time Analysis

To figure out if the evacuation or vacuum cycle interferes with the injection molding cycle time, it is necessary to evaluate the following conditions:

- Screw forward time required to fill the mold cavity
- Total shot weight, including mold cavities, runners, gates, and sprue
- Specific gravity of the thermoplastic polymer.

The analysis is based on evacuating 1.00 in³ of mold cavity volume and inlet air pressure of 60.00 psi at 5.5 cfm (maximum efficiency). The mold vacuum unit can draw 26.00 in of mercury in 0.131 s. These characteristics are obtained with a sealed mold by using a hose of 0.375 in inside diameter with a length not to exceed 60.00 in.

Example 10-9

Calculate the evacuation time for processing 33% fiber glass reinforced nylon based on the following conditions:

- Screw forward time required to fill mold cavities is 2.00 s
- Total shot weight is 250.00 g
- Specific gravity of the resin is 1.38.

$$\text{Volume} = \frac{\text{Shot Weight (g)}}{\text{Specific Gravity} \times 16.38} = \frac{250}{1.38 \times 16.38} = 11.00 \text{ in}^3 \quad (10-8)$$

To calculate the evacuation time, use the vacuum draw versus evacuation venting time graph in Figure 10-148.

Consider drawing a vacuum of 24.00 in of mercury, the evacuation time is calculated by multiplying the total shot volume 11.00 in^3 by the 0.039 s to draw a vacuum of 24.00 in of mercury (see Figure 10-135), it would take $(11.00 \times 0.039) = 0.43 \text{ s}$. If 26.00 in of mercury vacuum is required, the evacuation time would increase to $(11.00 \times 0.131) = 1.45 \text{ s}$.

The evacuation time (0.43 and 1.45 s) at both levels of the vacuum is less than the screw forward time to fill the cavities in 2.00 s; therefore, the mold vacuum venting unit will not interfere with or increase the molding cycle time.

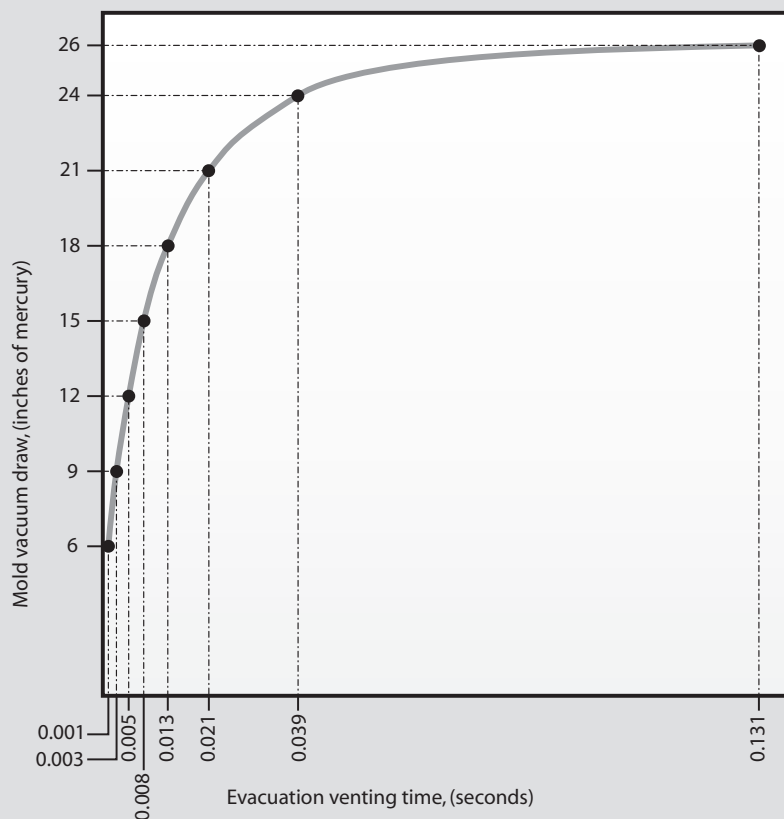


Figure 10-148 Vacuum draw vs. evacuation venting time

10.18 Mold Cavity Insert Contact Area Strength

The parting line of a mold cavity insert is subjected to the repeated impact loads during mold closing. This type of action can cause changes in dimensions, molding ejection problems, and flashing at the parting line.

To find out the clamping force required for a mold that is going to be used in the injection molding process, multiply the total parting line projected area (including the runner and venting system) by a clamp force factor of 5.0 t/in^2 .

To calculate the maximum clamping force of an injection molding machine based on the strength of the cavity insert contact area, the use of two cavity insert parameters is required: The cavity insert contact surface area, minus the mold venting areas at the parting line and the cavity insert strength to support the maximum clamping force without deflecting and/or compressing the cavity insert sidewalls. However, because of the operating conditions to which cavity inserts are subjected (repeated impact action, high clamping forces, high injection pressures, fast changes in temperatures, wear, corrosion, close tolerances, long service life), a high safety factor is used. It is recommended to use only 20% of the yield strength of the steel used for the construction of the cavity insert.

In the following the strength of the cavity insert contact area is determined:

- Determine the clamping force required for injection molding
- Select only 20% of the yield strength for the cavity insert steel
- Calculate the cavity and runner system projected contact area
- Calculate the mold insert surface contact area at the parting line

Example 10-10

Determine the clamping forces required based on the injection molding conditions and on the strength of the cavity insert contact area. Figure 10-149 shows the cavity insert dimensions. Assume the cavity insert is center-gated and is made of P-20 tool steel with a yield strength of 125,000 psi.

The total venting area, projected cavity area, and the surface contact area of the cavity insert are calculated as follows:

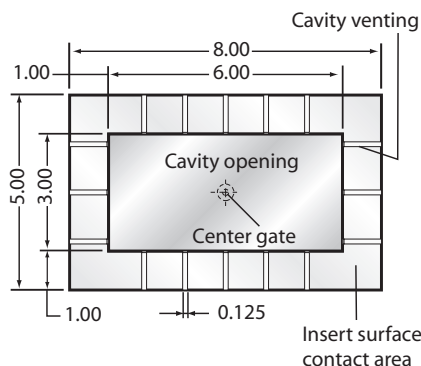


Figure 10-149 Mold cavity insert, "top view" (in)

$$\begin{aligned} \text{Total venting area} &= 16 \text{ (cavities)} \times 0.125 \text{ (width)} \\ &\quad \times 1.0 \text{ (length)} \\ &= 2.0 \text{ in}^2 \end{aligned}$$

$$\begin{aligned} \text{Projected cavity area} &= 3.0 \text{ (width)} \times 6.0 \text{ (length)} + 2.0 \text{ (vent)} \\ &= 20.0 \text{ in}^2 \end{aligned}$$

$$\begin{aligned} \text{Insert surface contact area} &= 8.0 \text{ (length)} \times 5.0 \text{ (width)} \\ &\quad - 18.0 \text{ (cavity contact area)} \\ &\quad - 2.0 \text{ (vent)} \\ &= 20.00 \text{ in}^2 \end{aligned}$$

The clamping force required for holding the cavity and the maximum load recommended for the cavity insert contact area is calculated as follows:

$$\begin{aligned} \text{Clamp force (molding)} &= 20.0 \text{ (cavity projected area)} \times 5.0 \text{ t/in}^2 \\ &= 100 \text{ t} \end{aligned}$$

Work stress (steel)	= 125,000 (yield strength) × 0.20 (safety factor) = 25,000 psi
Clamp force (insert strength)	= 25,000 (stress) × 20 (contact area) = 500,000 lbs. = 500,000/2,204 (lbs./t) = 226.86 t.

10.18.1 Cavity Insert Sidewall Strength

Because the cavity insert is subjected to high internal pressures during the injection of the melt, calculation of the cavity inserts sidewall thickness is very important in keeping the cavity insert deflections below a specified maximum requirement. While the compression stress of the parting line contact surface area in the mold must remain within safe limits, it is the actual physical deflection of the cavity insert sidewalls of the mold that is very important in the operation of the mold. If these requirements are maintained within the specified limits, generally the stress values will also be satisfactory.

The formulas used for calculating the cavity insert sidewalls are suitable for square or rectangular geometries only where the cavity insert length exceeds the cavity depth. As the depth approaches the length or exceeds it, its effect as a cantilever beam should be taken into account when calculating the total deflection.

The following assumptions are made to calculate the strength around the cavity:

- An insert sidewall is a fixed beam with a uniform distributed load.
- An insert sidewall is a uniform internally loaded picture frame insert.
- Each cavity insert sidewall is considered a freely supported beam with a uniform load applied to a rectangular mold cavity plate.
- The effects of mold base steel strength, clamping force, the mold support plate thickness and the number of support pillars are ignored.
- The applied load for combined sidewall thickness of cavity plate and insert is 20,000 psi, and for the cavity insert sidewall only 10,000 psi.

Figure 10-150 shows two methods for mounting cavity inserts in the base mold cavity plates. One is the picture frame in which the opening is cut through the base mold plate; the other is the blind pocket in which a recess is machined to receive the insert cavity. The picture frame insert design is selected due to the high cost of accurate machining of a blind deep pocket insert.

The deflection should not allow the clearance between the components to increase to such a degree that flashing can occur at the mold parting line.

As the thermoplastic melt cools and shrinks, the cavity injection pressure is reduced and the sides of the mold cavity inserts are not loaded, therefore they return to their original straightness. If the cavity insert walls are bent, this causes ejection problems, the molded part becomes trapped between the steel insert cavity faces of the male and female halves of the mold. The greater the mold cavity insert deflection, the greater the difficulty in opening the halves of the mold.

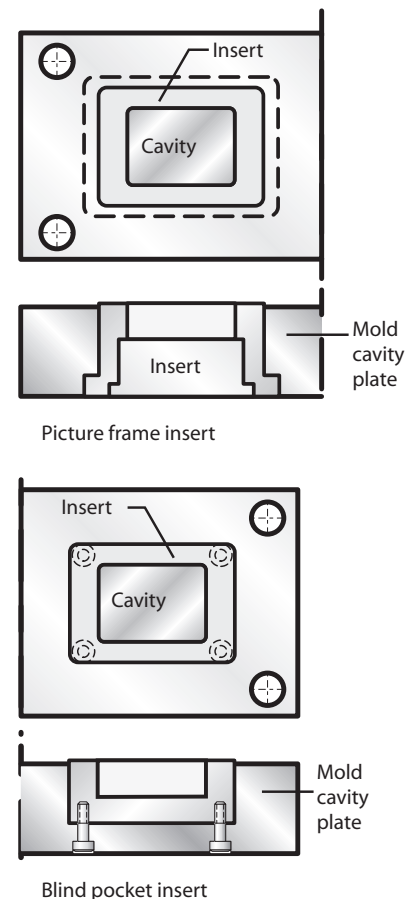


Figure 10-150 Mold cavity insert types and mounting

10.18.2 Methods to Calculate the Strength of Cavity Insert Sidewall

The first method to calculate the strength of the cavity insert sidewall is based on the standard fixed beam deflection equation:

$$\delta = \frac{W \times L^4}{384 \times E \times I} \quad (10-28)$$

Where:

δ (in) = Deflection, insert sidewalls

W (lb.) = Load inside cavity

= P (psi) Pressure of cavity $\times H$ (in) Depth of a cavity

L (in) = Length of a cavity

E (psi) = Modulus of elasticity (H-13 is 30×10^6 psi);

I (in⁴) = Moment of inertia of cavity insert sidewall

= $(H \times t^3) / 12$

H (in) = Depth of a cavity

t (in) = Thickness of cavity insert sidewall.

The necessary wall thickness (t) to permit a specified maximum deflection can therefore be obtained by transposing Eq. 10-28:

$$t = \left(\frac{W \times L^4}{32 \times \delta \times H} \right)^{1/3} \quad (10-29)$$

This calculation is first made for the long side of the cavity insert. For the short sides, a similar calculation can be made, or the same wall thickness determined for the long side is used, as this wall thickness will give an even smaller deflection.

Using Eq. 10-15, the minimum sidewall thickness (t) for a given deflection is obtained. In practice, however, the sidewalls of a mold cavity insert do not completely fulfill the basic requirements of end fixing, on which the fixed beam formula is based, and this method can lead to much higher deflections in service. If this formula is used, it should be restricted to molds in which the two mold halves fit in each other so that the moving male half of the mold prevents the opening of the mold by acting as an interlock.

The second method is to consider the sidewall of the insert cavity as a freely supported beam, according to the basic Eq. 10-30:

$$\delta = \frac{5 \times W \times L^4}{384 \times E \times I} \quad (10-30)$$

Then the sidewall thickness for a given deflection (δ) is calculated by using the following Eq. 10-31:

$$t = \left(\frac{5 \times W \times L^4}{32 \times \delta \times H} \right)^{1/3} \quad (10-31)$$

Equation 10-31 is commonly used, giving the greatest sidewall thickness for a given deflection compared to previous methods. In this respect, it provides the

greatest margin of safety, and in many molds a greater sidewall thickness above that strictly necessary is often of little consequence.

Perhaps, a more rational approach is offered by considering the mold cavity insert as a “picture frame” insert, considering each sidewall of the cavity insert independently as a uniformly loaded beam, with fixed ends and freely supported ends respectively. However, this does not hold true, because the ends are neither fixed nor free; nor does any insert sidewall deflect independently of the others, i.e., the effects of runner pressure drop and the corners must be taken into account.

The third method considers each sidewall of the cavity insert as a freely supported uniformly loaded rectangular plate. To fulfill these conditions of support, the base of the cavity insert must be either solidly or rigidly registered into the side, while the open end of the insert cavity must be completely and firmly tied across by substantial registers in the moving half of the mold. Then, from a formula derived by Timoshenko, the approximate maximum deflections for the cavity insert sidewall are found by using Eq. 10-32:

$$\delta = \frac{K \times P \times H^4}{E \times t^3} \quad (10-32)$$

Where:

K = a constant having the values as shown in Figure 10-151,
for the ratio $L / (H \text{ or } F)$

L = longest length of the cavity insert sidewall

H = depth of the cavity insert

F = mold “B” cavity plate thickness

P = injection pressure inside the cavity

E = modulus of elasticity of the cavity insert

t = cavity insert sidewall thickness.

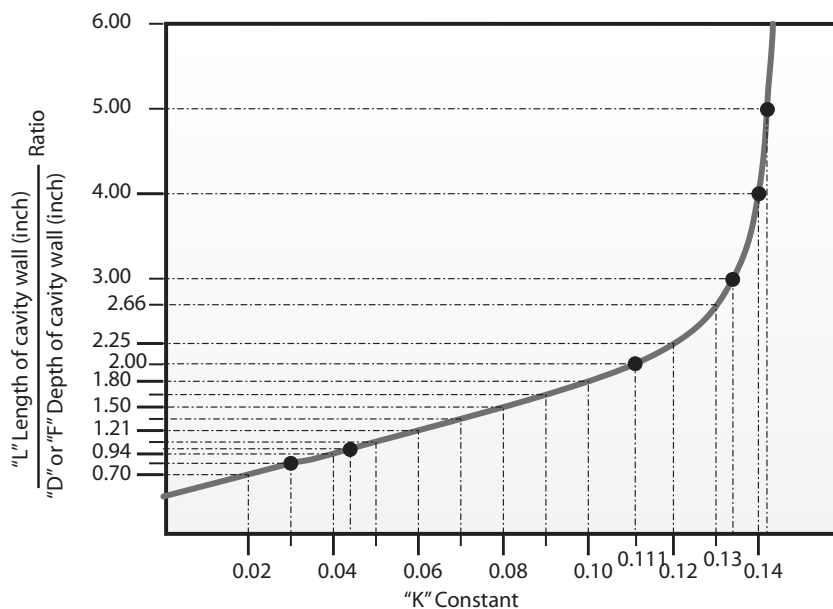


Figure 10-151 Cavity insert (length/depth) ratio vs. constant “K”

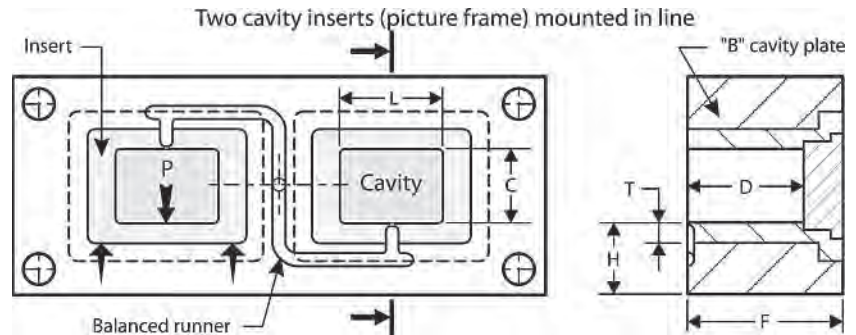


Figure 10-152 Two cavity inserts, picture frame mounted in cavity plate

Figure 10-152 shows how to calculate the cavity insert sidewall and the combined sidewall thicknesses of the mold "B" cavity plate and the cavity insert.

Example 10-11

Two cavity inserts made of H-13 tool steel are mounted in line in two individual mold cavity plate openings made of P-20 tool steel as shown in Figure 10-142. Calculate the cavity insert sidewall thickness (t) that meets the requirements of a maximum deflection (δ) of 0.001 in, when 10,000 psi injection pressure (P) is applied inside the cavity.

The cavity insert sidewall thickness (T) is calculated by transposing the Eq. 10-32 in a function of T :

Where:

K = Constant per Figure 10-141, where L / H ($8.0 / 3.0 = 2.66$) = 0.13

E (H-13 flexural modulus) = 30×10^6 lb./in²

$$T = \sqrt[3]{\frac{K \times P \times H^4}{E \times \delta}} = \sqrt[3]{\frac{0.13 \times 10,000 \times 3^4}{30 \times 10^6 \times 0.001}} = 1.52 \text{ in}$$

Example 10-12

Calculate the combined sidewall thickness (N) for the previous example for a maximum deflection (δ) of 0.0001 in, at 20,000 psi injection pressure (P).

By using the previous equation, the sidewall thickness (N) can be calculated with:

K = Constant per Figure 10-141, where L / F ($8.0 / 4.0 = 2.00$) = 0.111

E (P-20 flexural modulus) = 25.56×10^6 lb./in²

$$N = \sqrt[3]{\frac{K \times P \times F^4}{E \times \delta}} = \sqrt[3]{\frac{0.111 \times 20,000 \times 4^4}{25.56 \times 10^6 \times 0.0001}} = 6.06 \text{ in}$$

The cavity plate width (J) = $(2 \times N) + C = (2 \times 6.06) + 3.50 = 15.62$ in or standard mold base cavity plate width = 15.875 in

Example 10-13

Let us assume that a sectional mold is being built by cutting one large opening in the mold base cavity plate for mounting the two cavity inserts. An insert spacer is mounted in the middle opening between the two cavity insert internal sidewalls and a smaller runner injects in the middle of both cavities in the same direction, as shown in Figure 10-153. Calculate the combined cavity plate sidewall deflections, one at location (a) where 20,000 psi injection pressure is applied inside the cavity and maximum deflection (middle) of the mold base cavity plate.

A beam load diagram representing the cavity plate's large opening conditions is developed to find the corresponding deflection equations used in this example.

$$E \text{ (P-20 flexural modulus)} = 25.56 \times 10^6 \text{ psi}$$

$$I \text{ (Moment of inertia)} = (F \times N^3) / 12 = (4.00 \times 6.06^3) / 12 = 74.18 \text{ in}^4$$

$$W = L \times F \times P = 8 \times 4 \times 20,000 = 640,000 \text{ lb.}$$

$$a = (L + t) / 2 = (8 + 1.52) / 2 = 4.76 \text{ in}$$

$$b = 2 \times L + S + 3 \times t = 2 \times 8 + 3 + 3 \times 1.52 = 23.56 \text{ in}$$

Deflection at "a":

$$\begin{aligned} \delta_a &= \frac{W \times a^2}{49 \times E \times I} (3 \times b - 4 \times a) \\ &= \frac{640,000 \times 4.76^2}{49 \times 25.56 \times 10^6 \times 74.18} (3 \times 23.56 - 4 \times 4.76) \\ &= 0.0080 \text{ in (80 times the recommended deflection).} \end{aligned}$$

Deflection at the middle:

$$\begin{aligned} \delta_M &= \frac{W \times a}{196 \times E \times I} (3 \times b^2 - 4 \times a^2) \\ &= \frac{640,000 \times 4.76}{196 \times 25.56 \times 10^6 \times 74.18} (3 \times 23.56^2 - 4 \times 4.76^2) \\ &= 0.0130 \text{ in} \end{aligned}$$

The deflection at the point of the load (a) is 80 times and at the middle 130 times greater than for the two openings for mounting the cavity inserts shown in Example 10-13. The maximum deflection of 0.0130 in of the mold cavity plate sidewall thickness would permit some thermoplastic melt to flash, the mold parting line opening would become progressively worse, and associated problems would increase. Thermoplastic polymers that cannot flow in such narrow spaces interfere with the mold opening, because an interference of 0.0080 in per side is a sizable amount for the ejecting force to overcome.

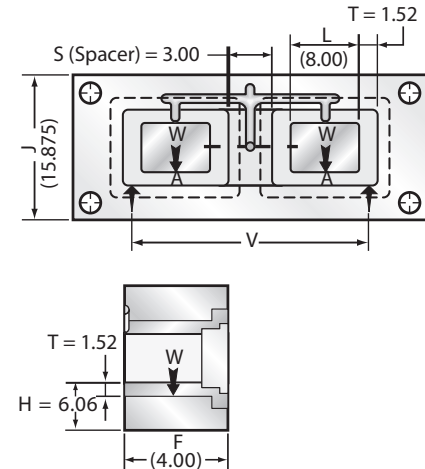
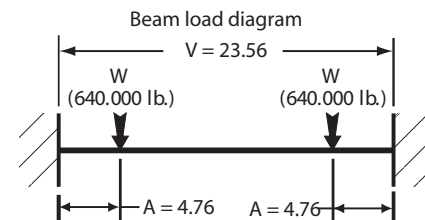


Figure 10-153 One hole cavity plate, two inserts mounted in line



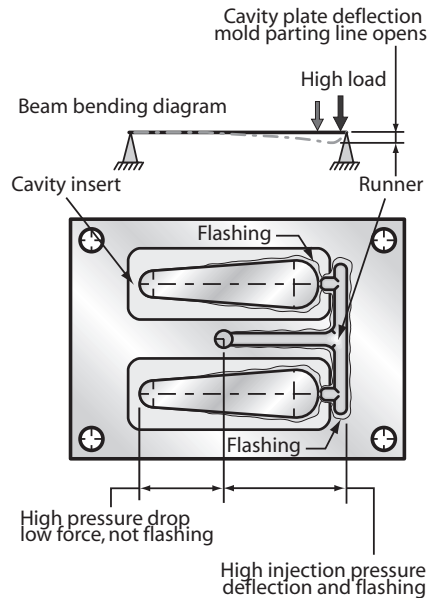


Figure 10-154 Mold layout out of balance with parallel cavities

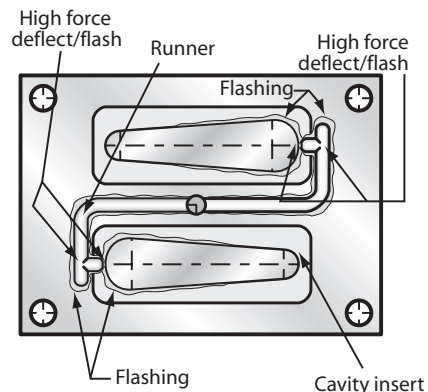


Figure 10-155 Balanced mold layout, cavity rotated, long runners

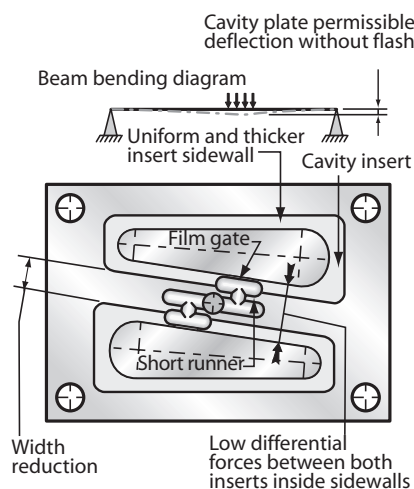


Figure 10-156 Symmetrical balanced mold with short runner and film gates

There have been cases where the deflection of the plate was large enough to cause a sizable increase in wall thickness of the molded products. When the cavity injection pressure is reduced, the mold cavity plate steel springs back to the original position with a force equal to the one that caused it to deflect. The pressure from deflected steel on the increased wall thickness of the molded part could be so great that mold opening is impossible. Only by completely disassembling the mold into its basic components while the mold is hanging in the molding machine would it be possible to separate both halves of the mold. After clearing out the thermoplastic melt, the mold would need to be reassembled and reestablished into operation. Such problems can be avoided by calculating the required cavity insert and mold cavity plate sidewall thicknesses and strength correctly.

10.19 Mold Layout Case Studies

The following mold layout illustrations show three schemes for positioning the cavity inserts and the cold runner system in the mold cavity plate. The last two mold layouts were modified to reduce the out-of-balance forces that cause the parting line flashing. Various other mold layout schemes (hot runner) can be set up to achieve the same objectives as the last mold layout illustration.

Although some small out-of-balance force may be tolerated in practice, it is essential that the greatest care is exercised to make sure these forces do not become excessive, otherwise, flashing can result. Correcting this problem can only be accomplished by changing the molding parameters, employing less than optimum melt and mold temperatures, and reducing injection speeds and pressures.

Figure 10-154 shows an out-of-balance mold layout that causes flashing problems at the right side of the mold. The mold is considerably out of balance because of the mold layout position of the maximum projected area portions of each long cavity and runner. The parts were molded of glass reinforced PET (fast melt flow resin), using high process and mold temperatures, fast injection speeds, and the maximum clamping capacity to fill the cavities.

Figure 10-155 shows a mold layout improvement by rotating the lower cavity insert 180° to balance the forces at both sides of the mold. This modified mold layout requires a longer runner and small edge gates in the same areas as the previous layout. It also requires high injection pressures for the long runners and restricting gates to fill the thin and long wall cavity. Almost half the injection pressure is lost through the runners and gates. The remaining injection pressure is used to fill the cavity. The highest injection pressure occurs at the entrance of the cavity where the projected area is the largest, causing forces greater than the clamping force available, pushing the moving half or opening the parting line of the mold, causing flashing problems at each side of the cavities and runners.

Figure 10-156 shows a mold layout scheme for symmetrical balancing and reduction of cavity pressure, using short runners, bigger gates (film or fan type of gate) positioned at the same location in both cavities. This is the preferred mold layout to balancing the cavity melt flow length for a uniform melt flow of the cavities. The injection pressure conditions will be symmetrical and in balance for both cavities, while the overall size of the mold could be reduced allowing the smaller mold to use a lower capacity injection molding machine, thus reducing costs of the molded parts.

The optimum mold lay out provides the following benefits:

- Optimum mold venting performance
- Faster and uniform molding cycles
- Reduction of injection pressure, melt, and mold temperatures
- Reduction of cavity deflections without flashing
- Thinner cavity insert sidewalls
- Reduction of overall mold size
- Runner length, diameter, and pressure drop reductions
- Larger gate size closer to sprue to reduce flow path and filling the cavity uniformly using lower injection speed and pressure
- Balanced melt flow in the cavities (part dimensional control)
- Reduction of rejected molded parts

10.20 Mold Support Pillars

The mold cavity plate surface should be between 0.003 and 0.005 in lower than the cavity insert surface to provide a parting line contact surface area in both mold halves. Figure 10-157 shows the benefits of support pillars; they greatly increase the capacity of the mold for supporting the projected area of the cavities and runner system. By providing additional stiffness to the support plate, they prevent deflection of the mold cavity inserts.

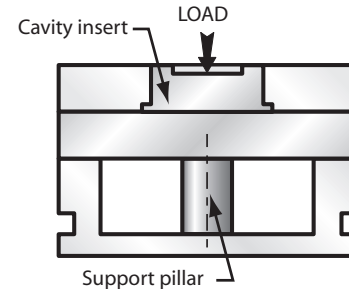
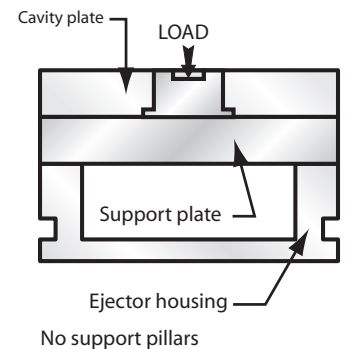
For example, a 11.875×15 in mold base with a maximum permissible load on the support plate without support pillars will allow 14 in^2 of projected cavity area without flashing (top illustration). The use of one row of support pillars (middle illustration) will quadruple the projected cavity area to 56 in^2 . Two rows of support pillars (bottom illustration) will provide nine times the projected area or 126 in^2 .

10.21 Tolerances for Thermoplastic Molded Parts

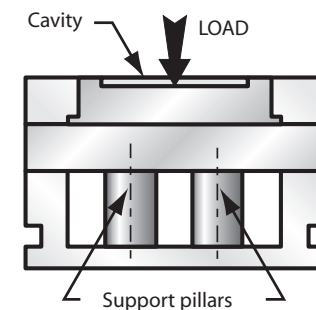
Does my company need and can it afford the tolerances I have specified? A realistic view of the purchase cost of tolerances often helps avoid high manufacturing costs without affecting the performance of the thermoplastic injection molded products.

It may be unreasonable economically to specify fine production tolerances on a molded product when it is designed to operate within a wide range of environmental conditions. Dimensional changes of the product caused by temperature variations alone can be three to four times as great as the specified tolerances. Specifying fine production tolerances does not make too much difference in many applications such as bearings and gears. Fine tolerances with thermoplastics are not as vital as with metals because of the resiliency of thermoplastic materials.

The following are some general product design suggestions for the selection of production tolerances:



One row of support pillars increases the capacity to support projected cavity area four times

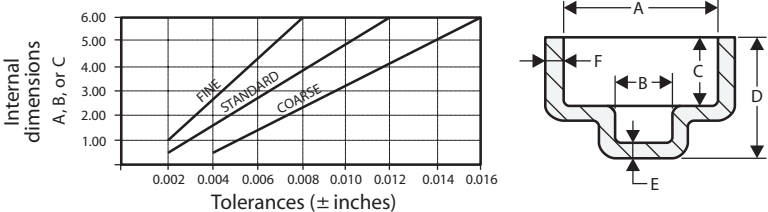


Two rows of support pillars increase the capacity to support projected cavity area nine times

Figure 10-157 Increase in load capacity using mold support pillars (Courtesy: DME)

- The product design for a thermoplastic molded part should indicate the conditions under which the dimensions shown must be held.
- In a product design drawing, the overall tolerances for a thermoplastic molded part should be shown in in/in, not in fixed values. A title block should read, "All decimal dimensions $\pm 0.00X$ in/in ($\pm 0.00X$ mm/mm), unless otherwise specified", not $\pm 0.00X$ in ($\pm 0.00X$ mm).
- Only fine tolerances should be specified in the design drawing.
- When compromises in tolerances could be acceptable from a performance standpoint, the tolerances in question should be discussed with the molding process engineer.

Figure 10-158 shows the tolerances developed by the plastics molding industry for a thermoplastic injection molded part. The purpose of these specifications is to assist the product designer in obtaining a quick and preliminary analysis for the different molding tolerances found in thermoplastic injection molded parts.

ENGLISH UNITS Suggested tolerances (inches)					
	For dimensions over 6.00 inch add (±) 0.001 (Fine), 0.002 (Standard) or 0.003 (Coarse) inch per added inch			Tolerances ± (inches)	
	External height D	Single Cavity D = lower than 1.00	Fine	Standard	Coarse
		Multiple Cavity D = lower than 1.00	0.002	0.004	0.006
		D greater than 1.00 add per inch	0.003	0.005	0.007
	Sidewall thickness F = lower than 0.30	For C = lower than 1.00	0.002	0.003	0.004
		C greater than 1.00 add per inch	0.001	0.002	0.003
	Bottom wall thickness E	E = lower than 0.10	0.002	0.004	0.006
		E = 0.10 to 0.20	0.004	0.005	0.007
		E = 0.20 to 0.30	0.006	0.007	0.008

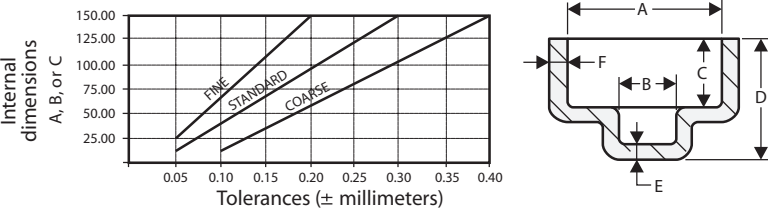
METRIC UNITS Suggested tolerances (millimeters)					
	For dimensions over 150.00 mm add (±) 0.02 (Fine), 0.05 (Standard) or 0.08 (Coarse) mm per added mm			Tolerances ± (millimeters)	
	External height D	Single Cavity D = lower than 25.00	Fine	Standard	Coarse
		Multiple Cavity D = lower than 25.00	0.05	0.10	0.15
		D greater than 25.00 add per mm	0.08	0.13	0.18
	Sidewall thickness F = lower than 7.50	For C = lower than 25.00	0.002	0.003	0.004
		C greater than 25.00 add per mm	0.05	0.08	0.10
	Bottom wall thickness E	E = lower than 2.50	0.001	0.002	0.003
		E = 2.50 to 5.00	0.05	0.10	0.15
		E = 5.00 to 7.50	0.10	0.13	0.18
		0.15	0.18	0.20	

Figure 10-158 Tolerances for thermoplastic injection molded parts

10.21.1 Factors Affecting Dimensional Control Tolerances

Thermoplastic injection molded product final tolerances are more complicated to predict or calculate. Tolerance is a result of product design geometries, product size, wall thickness, type of thermoplastic, resin molding characteristics, linear coefficient of thermal expansion, mold/post molding shrinkage of resin, amount of fiber glass or mineral reinforcements (orientation), material handling, molding machine, auxiliary equipment, mold design, molding conditions, and end use environment conditions.

The following variables are responsible for dimensional changes in thermoplastic injection molded product and the inability to hold dimensions:

Thermoplastic resin

- Thermoplastic melt viscosity variations require process temperatures adjustments. This can occur by deliberate or accidental mixing of lots of virgin material having different viscosity properties.
- Virgin material moisture content variations affect the drying process conditions and cause changes in melt viscosity.
- Types and levels of reinforcements, melt flow rates, impact resistant additives, lubricants, and colorants may not be uniformly dispersed.
- Variations in size and geometry of the virgin resin pellets.
- Different amounts of reground material (sprues, runners, defective parts) of different sizes (longs and fines) with different moisture content that are poorly blended with the virgin resin.

Material handling

- Drying process affects the melt flow rate and molding parameters.
- Condensation of virgin resin (cold warehouse, warm molding area).
- Resin contaminated with fines and longs or other materials.
- Molded parts after ejection are subjected to different cooling methods: ejected parts fall in a box, parts placed facing up or down on the work table while others are immediately packed, etc.
- Transporting the molded parts through hot, dry, wet or cold locations.
- Fans or open windows causing drafts in the work area.
- Considerable atmospheric changes in humidity and temperature.

Injection molding machine

- Erratic performance of the electrical and hydraulic systems.
- Nozzle temperature problems; cold nozzle causes short shots; hot nozzle burns the melt changing its viscosity
- Worn clamp platen bearings (mold moves causing concentricity and vertical dimension variations).
- Worn plastifying screw, check valve, or barrel causing melt variations and back flow leakage (short shots and poor density of molded parts).
- Shot-to-shot variations in the quantity of material fed to the cylinder.

Mold

- Gates too small and/or too long (premature melt freeze-off caused by a pulsing of the melt flow rate within the cavities).
- Runner system too long or too complex (excessive injection pressure drop causing melt flow pulsations during filling the cavities).
- Poor or improper mold venting.
- Deflections of the core insert, cavity insert sidewall, or support plate.
- Poor temperature control of the cores and cavities surfaces.
- Worn mold components such as leader pins, ejection system, wedge blocks, binding, etc.

Auxiliary Equipment

- Mold temperature fluctuations of the mold thermolator, or the size of the thermolator too small.
- Mold temperature control equipment can not maintain a uniform temperature of the cavities at required levels.
- Defective dryers. Defective desiccant beds position control, desiccant pellets do not absorb moisture (too old and/or contaminated), defective thermocouples, shut off valves, filters or blowers. Dryer temperature too low for drying the resin or variations in drying temperature and time exist. Processing dry air return too high for the efficiency of the desiccant material. Drying equipment needs an after-cooler on the processing air return to improve the drying efficiency of the desiccant beds, allowing the dryer to operate at higher drying temperatures (250 to 350 °F).
- Inadequate chipper design or poor maintenance. The same chipper is used to grind different materials (contamination). Reground material of different sizes (longs and fines).
- No vibrating screen equipment to separate the reground material in large sizes and fines.
- Nonuniform metering, blending, and reground material ratio used with the virgin resin.

Mold Operating Conditions

- Molding cycles not uniform (change of shifts, delays between cycles).
- Deliberate change in any controllable molding variable (melt/mold temperatures, screw back pressure, screw return velocity, injection pressure, packing pressure, screw forward time, injection speed, etc.).
- Molding cycle too short, resulting in some unmelted material that could block the gates or enter inside the cavities (low back pressure, low compression screw, low barrel temperature profile).
- Injection molding filling rate too slow.
- Mold surface cavity temperature variations.

10.22 General Specifications for Mold Construction for Thermoplastic Injection Molding Resins

Special consideration must be given to mold construction, mold quality, and mold performance. The mold vendor is responsible for calculating mold shrinkage, pressures, and mold stresses for each mold. All the mold components have to be sized and constructed accordingly. The mold vendor is responsible for any corrections that are required if the mold does not function properly.

10.22.1 Mold Design Requirements

Metric and English units are used for dimensioning the thermoplastic injection molded product designs and details.

All components used in the mold must be to the American National Standards in inch for molds that will be operating in the USA manufacturing facilities. For molds that will be operating in other countries, metric standards (millimeter) are required (metric to English conversions are not acceptable).

Unacceptable Examples:

- ⇒ 10 mm ejector pin converted to 0.3937 in diameter.
- ⇒ 50 mm dimension converted to 1.9685 in.

Unless otherwise specified, allowable tolerances on all the cavities, cores, slides and the related components are:

- ⇒ Two (2) place decimal dimensions = ± 0.01 in.
- ⇒ Three (3) place decimal dimensions = ± 0.001 in.
- ⇒ Four (4) place decimal dimensions = ± 0.0005 in.

All cavity and core pockets must be machined in and must be square and perpendicular within ± 0.001 in.

All cavity and core blocks must be square and perpendicular within ± 0.001 in. The fitting must be done by surface grinding and must fit correctly in the cavity or core pockets. Hand grinding the cavities or core pockets or the cavities and core blocks is prohibited.

10.22.2 Mold Drawing Standards

All drawings can be on roll size “Velum” (maximum of 52.00 in long) or on a computer file. The company border and title block drawing format is used.

The first sheet of each set of drawings is a “Table of Contents”, listing what is drawn on each sheet in the set of drawings.

There must be at least one plan view for the ejector half of the cavity and one plan view for the fixed half of the cavity. One view showing the fixed half plan and half ejector plans are not acceptable.

Label plan views with “Top” to show which side is the top of the mold when set in the injection molding machine.

A minimum of two section views must be shown.

Assembly drawings will be ballooned, showing frame and cavity detail numbers. If a detail of the mold is drawn, the balloon will have a horizontal dividing line with the detail number above the sheet number, on which the detail is drawn below.

On each sheet, below the title block, the detail numbers and description must appear in the margin.

All components that are not standard should be detailed. Any standard component that requires modification must also be detailed (e.g., ejector pins, core pins).

The shut off height of mold must be at least 1.00 in larger than the minimum shut off height of the molding machine and at least 1.00 in shorter than the maximum daylight of the machine.

For a family type mold, where parts are very similar, and a master cavity and frame are used to produce different parts, the company tool engineer can advise on what type of molds are required. The vendor will supply detail cavity changeover instruction sheets.

Outlines of the molding machine tie bars need to be shown in phantom lines in the plan views of both the fixed and moving halves of the mold.

All water cooling baffles and diverter plugs must be detailed.

All water cooling baffles and diverter plugs must be detailed.

All runner systems (sprue, sprue puller, runner with cold slug pockets and ejectors, and gates) must be detailed.

All venting systems must be detailed.

A full set of approval drawings will be required for all assembly sheets; design detail drawings for some hot runner mold systems will be required at the completion date. The tool engineer will date and initial a full set of these drawings at this time.

Documentation for Mold Modifications

The tool drawing change letter has no relationship to the product change letter, only the change notice number does.

The changed area of the drawing is identified with the change letter in a triangle.

There may be changes not related to an engineering change, such as for mold improvements. These are recorded in the same manner.

The company change notice block is required on every sheet, placed in the upper right hand corner of the sheet. Provide a minimum of eight lines of entry (tool engineer to provide format.)

Mold Shrinkage

Mold shrinkage values for each specific thermoplastic resin are the responsibility of the tool vendor to provide! The tool engineer or the resin supplier does not make shrinkage recommendations to the mold builder.

Center-to-center distances between cored holes, slots, bosses, etc. are to be held to the part print mean dimensions plus the applied mold shrinkage rate for a particular dimension of the thermoplastic resin.

Cored holes and openings in the part should be dimensioned on the tool drawing to the mean dimension, plus 50% of product print tolerances above mean, plus mold shrinkage rates. This will allow for metal removal on mold insert, if necessary.

Product design having outside fit conditions will be dimensioned to mean dimensions, minus 50% of product print tolerance below mean, plus the thermoplastic resin mold shrinkage factor.

Molded Product and Mold Cavity Identification

Each cavity must be identified as required by the engineering product design print (tool engineer to advise).

The company trademark is required on molded products and positioned as shown on the product design drawing.

Cavity numbering sequence must be on the assembly drawing.

In cases when there is more than one mold of the same tool number, the following mold and cavity identification will be required:

- ⇒ Mold # 1, Tool Number – “A” cavity 1 and 2.
- ⇒ Mold # 2, Tool Number – “B” cavity 3 and 4.
- ⇒ Mold # 3, Tool Number – “C” cavity 5 and 6.
- ⇒ If cavity numbers go up to nine, underline cavities 6 and 9.

10.22.3 Required Types of Tool Steels for Mold Construction

Types of Tool Steels

Mold base custom designs, use P-20/P-21 prehardened 34–36 Rc.

For all special structural components, use AISI 4140 prehardened 28–32 Rc (extension nozzle bushing, clamp plates).

After the tool vendor has shown that the mold has been debugged and the pre-production results have been reviewed and approved by the tool engineer, the mold components can be plated.

Electroless nickel plate all mold bases, clamping plates, locating rings, and ejector plates, at the completion of the first molding trials, after making the necessary adjustments and corrections to the mold. Preharden and polish the cavities/cores/slides.

DME, HASCO, NATIONAL mold components are to be used (e.g., core pins, return pins, extension nozzles, etc.).

Types of tool steels, hardness and their applications are shown in Table 10-2 and 10-12 (the usage of other types of tool steels requires the approval of tool engineer):

The tool engineer reserves the right to ask for formal certification of any type of steel and hardness used in the mold by the mold vendor.

Nitrating is required for all the cavities, cores, inserts, slides, and slide carriers. Typical case depths between 0.015 in and 0.025 in are recommended. Special cases must be reviewed with the tool engineer before nitrating. Before nitrating, all standing steel, all shut-off, or any special areas of the cavity or core must be digitized. The related components must be measured and documented on the

Table 10-12 Types of Tool Steels for Molds

Material	Hardness	Application
H-13 / S7 / A2	Rc 48–52	Cavities/cores inserts
P-20 / P-21	Rc 28–36	Mold base
Nitrated H-13	Rc 65–74	Ejector/leader pins
D-2 / CPM 10V	Rc 60–62	Gate inserts
A2 / A6 / P6 / O2	Rc 48–52	Slides
Ampco 940	Rc 28–30	Inserts/cores in poor cooling areas
A2 / A6 / P2	Rc 56–58	Wear plates
SAE 1020	Rc 30–35	Ejector plate

tool drawings. (**Note:** Break all unnecessary sharp edges, 0.030 in $\times$ 45°, on any nitrate components to reduce chipping!).

Welding

No welding will be done without the prior approval from the tool engineer.

Before any welding can be approved, the vendor must provide the proper procedures in writing to the tool engineer. Those procedures are to be in line with the type of steel to be welded and the details of the items, such as:

- Surface preparation and cleaning
- Preheat treatment or annealing
- Preheating
- Post-weld surface treatment
- Post-weld heat treatment
- Post-weld cleaning
- Filler material

Before welding, the vendor must submit an updated copy of the “Request to Weld” document to the tool engineer.

Welding will not be allowed, except in those cases where the tool engineer has signed approval on the document before the welding operation. A copy of the signed document will be given to the tool engineer immediately following each approval or denial of the “Request to Weld” documents.

The original of all “Request to Weld” documents used during the complete mold building, tryout, and “MQ1” will be supplied to the tool engineer with the tool drawings. Each original sheet is to be signed and dated by the vendor. This will be the vendor’s statement in writing regarding the weld history of the mold.

Any welds done in the mold outside this specification and/or not in concert with its requirements will result in replacement of the welded items at the vendor’s expense including any fitting, tryout, debug, or re-qualification costs incurred.

Finishing

Cavities, inserts, and cores must be finished as stated on the specification sheet. Surfaces must be free of nicks, scratches, undercuts, and all final polishing is to be done in the direction of the draw.

Mechanically remove all EDM finished areas and thermally stress-relieve per material suppliers recommendations.

Meta-Lax is not an acceptable method for stress relieving EDM residual internal stresses.

10.22.4 Mold Construction Requirements

Locating Ring

Locating rings need to be the nominal size required by the type of injection molding machines to be used. Locating rings are to be made of SAE 6521 or 6524 steels, hardened to Rc 28-32, and documented on the Q.C. Plate.

Locating rings are not required for vertical clamp and horizontal injection type molds. Tool Engineer will provide molding machine information.

Sprue Bushing

The nozzle radius for sprue bushings should be 0.75 in or 0.50 in spherical (highly polished), made of SAE 6145 steel hardened Rc 43-45, ground and polished, unless otherwise specified by the tool engineer.

The sprue length must be kept to a minimum (extension nozzle type of design is recommended). Circumstances requiring a longer sprue length must be made known to the tool engineer. All sprue bushings for horizontal clamp and horizontal injection type of molds, in which radial orientation is critical, must be keyed to keep sprue bushing from rotating. The sprue bushing small inside diameter should be between 0.031 in and 0.062 in larger than the front inside diameter (orifice) of the injection molding machine nozzle.

Vented Cold Runner System

The cross section area of the runners must be fully round (modified trapezoidal type or other geometries must be approved by the tool engineer). Layout and size of runners are to be calculated for a specific mold layout and type of thermoplastic resin by joint effort between the tool vendor, tool engineer, and resin supplier's technical representative. All cold runner layouts on the molds require a balanced melt channel distribution, unless otherwise approved by the tool engineer. A step down can be made in runner size diameters to obtain an efficient pressure drop of the runner, and an ideal cavity pressure required by the thermoplastic melt to mold the products. All cold slug pockets required for the runner system should be vented at the sprue puller, at the ends of the main and the secondary runners. All runners should have vented ejector pins; they must be smooth, free of nicks or imperfections and polished in the direction of flow. Runners along mating surfaces of two inserts are not acceptable. In such cases, a runner plate must be provided.

Use of Mold Flow and Mold Shrinkage analysis software programs is highly recommended and encouraged whenever possible.

Gates and Mold Venting Systems

The types of gates and mold venting systems are determined by the type of mold ejection system, the part geometry, resin type, and dimensional control required by the application.

A separate set of mold venting system detail drawings are required, showing the vent channels, geometries and dimensions for venting the cavity/core inserts, all runner's cold slug pockets, sprue puller, and ejector pins. The venting layout should show venting channel intersections and end at the edge of mold or venting connected to side of mold, via a clear port designed for this function.

"Ejector pin venting", "sintered vents", "logic seal venting system" or "vacuum venting system" are venting systems developed by several companies. These venting systems are commercially available and should be considered for mold design applications. All are highly recommended for deep/blind and thin (less than 0.020 in) walled products, difficult to fill applications to improve weld line strength, reduce flashing problems, reduce injection pressure, and to improve surface finishing.

Vented Sprue Puller

Reversed taper or "Z" type sprue pullers with ring groove vent channels are recommended for horizontal clamp and horizontal injection, or for vertical clamp and vertical injection type molds. Sprue pullers are not required for vertical clamp and horizontal injection type molds. The sprue puller is replaced by a vertical ejector pin system to free the runner and the cavities from the lower cavity plate.

Layout of Mold Cavity and Core Inserts

A symmetrical mold cavity layout in the middle of the mold will provide uniform distribution and balanced clamp force to counteract the polymer injection pressures that may cause a mold cavity shift. Allow space between cavity inserts for runners, waterlines, return pins, ejection system, support pillars, etc. for mold construction strength.

Each cavity insert should be mounted in a single opening "picture frame" of the cavity plates; two or more cavity inserts cannot be mounted in a single large opening, separated by spacers. The cavity plate sidewalls will have excessive large deflections causing serious molding problems. Lift holes are required for each cavity insert half. Holes will be sized correctly for the weight to be lifted. Each cavity and core insert requires two precision construction holes, holding the size and location to within 0.001 in. Hole sizes are determined by the size of the cavity insert blocks. Only standard sized holes are acceptable. The cavity inserts should be mounted at least 0.003 to 0.005 in above the mold base cavity plates.

The cavity insert sidewalls should have a maximum deflection of 0.001 in, when the cavity is subjected to 10,000 psi injection pressure.

The cavity plate compounded sidewalls should have a maximum deflection of 0.0001 in, when the cavity is subjected to 20,000 psi injection pressure to avoid deflections at the parting line causing flashing and ejection problems.

Cavity Insert Surface Contact Area (Shut Off)

Cavity inserts contact surface areas minus the runner and venting areas should require less than 20% of the yield strength of the steel used for the cavity inserts. These values will provide the maximum clamping force or size of the injection molding machine for the mold that can be used. Picture frame type inserts are recommended for the construction of the mold. If the cavity inserts are offset, support pillars must be added to the mold base frame to balance the loading on the machine tie bars.

Straight or severe angle vertical cavity insert contact areas are not acceptable. The tool engineer is to be notified of any deviations from the horizontal cavity insert contact area or face to face shut off.

Draft Angles

Usually, draft allowance should be per product print specifications. The responsible tool engineer must always confirm the need for draft, or how much draft to be used in each specific part design. In cases where a draft angle has not been specified, or the required draft angle may be the cause of problems during the molding process, the tool engineer must be consulted. This may be the case, for example, if the specified draft angle interferes with the geometry of the part, the draft angle is too small to eject a textured wall, the core draft angle is too small (the molded part cannot be removed from the core, or the part inside diameter sticks to the core, producing blisters, rough surface and poor dimensions).

Chamfer Edges

Chamfers along parting surface break lines should be 0.125 in $\times$ 45°. Provide pry bar slots at main parting line of the mold.

Specifications for Pipe Tapped Holes and Pipe Plugs

All external and internal pipe threads are to be NPT type, also known as “National Pipe Tapered” or “American National Standard Taper Pipe Thread”. NPT threads have superior strength and sealing abilities. These attributes are due to the interference that occurs at all engaged roots and crests of the threads. This interference or sealing action will prevent spiral leakage and allow for pressure tight joints without totally relying on the use of sealing compounds. NPT style threads are tapered 0.75 in per foot and do require greater care and accuracy. When designing and manufacturing the mold it is necessary for external and internal threads to have full thread height for length of engagement. After the threads are assembled, the top of the NPT dry seal type pressure plug must be flush with or below the top of the threaded hole.

If a design or manufacturing compromise needs to be incorporated into the tool and there is not enough room for full length engagement of a standard NPT dryseal type pressure plug, a flush type NPT pressure plug may be substituted. The “Dryseal pressure tight flush” type plug has a taper of 0.875 in per foot and provides a high pressure seal through the differences in taper and is designed to remain flush with the top of the 0.75 in taper NPT threaded hole. All holes to be threaded NPT style should be countersunk, drilled, and taper reamed. Incomplete or torn threads in an NPT threaded hole must not be accepted.

Mold Temperature Control

Cooling channels must be drilled from one side of the mold to the other. The plugs need to be easy to remove and the channels easily accessible for maintenance and cleaning. The mold cooling layout proposal must be approved by the tool engineer. The use of a mold cool temperature control analysis software program is recommended whenever applicable.

All waterline openings other than outlets or inlets will be closed with brass pressure plugs, using Teflon® tape as a thread sealant. Plugs must be below the surface to eliminate any interference. Modifying the plugs is not acceptable.

A separate drawing is required showing only the mold cooling systems in the cavity plates, cavity and core inserts.

All the internal diverter passages must be plugged by slip fitting 416 stainless steel or brass plugs, braised to a 0.125 in diameter, 416 stainless steel, or brass rods, that will hold the plugs in place. The rods will have a brazed threaded attachment at the pipe tap end of the hole. To ease plug removal, use “Neverseize” or an equivalent thread compound on plugs at assembly. Waterline connections need to be on the side of the mold and are to be tapped with a standard NPT thread to connect with the mold temperature control hoses’ male plug connectors. Special waterline fittings of correct sizes should be used on the mold cooling channels. These fittings do not restrict the coolant flow rates. The mold builder is responsible for providing and installing all waterline connectors and pipes. These connectors should be compatible with those available at the mold tryout. Use only brass pipes and fittings. (Common types of steel pipes and pipe fittings of any type are not acceptable).

Waterline entrances and exits are to be stamped on the mold base with suffixes indicating “IN” and “OUT”. Mold drawings must also show the layout of cooling channels and same designations for “In”s and “Out”s. Permanently identify hose fittings on both ends.

Use Parker Hannifin Teflon®/stainless steel braided hose # 919 with manufactured end fittings also in stainless steel or equivalent. The size of the hoses will be determined by the mold cooling analysis and the turbulence flow required by the application.

Leader Pins and Bushings

Leader pin diameters should be sized based on the leader pin length and mold weight. All leader pins must be placed in the side of the mold that will provide the least restriction to part removal. Leader pins must engage bushings by at least one pin diameter before cam pins engage and/or telescoping mold sections engage, or any other mold sections engage.

Provide slug clearance behind leader pin bushings to clear out possible slugs from mold.

One leader pin is to be offset to ensure proper assembly of the mold halves.

Core Pins

- Mounting:* Core pin bodies must be held to the shortest practical length, and held in place by the head.

Where radial orientation is critical, a positive lock at the head of the pin must be provided. Locking by means of pinning into a recess is not acceptable.

Core pins must be mounted to provide access for replacement without major mold disassembly. If this is not practical, the problem must be made known to the tool engineer.

Core pins mounted in assembly with sleeve ejectors need to be retained in the ejector bottom plate, backed up by a plate held by two screws. Do not use set screws to back up core pins. The bottom plate must be doweled to the ejector retainer block to establish and maintain core alignment. All core pin holes must be jig ground. The core pin outside diameter will be located both on the bottom of the core pin and next to the counterbore for the core pin head. Use letters for identifying core pins. Lettering of core pins must also be shown on the mold drawings.
- Material:* Use DME “CX” core pins (Rc. 50-55) and nitrated moving core pins.

Vented Ejector Pins

Location and size of vented ejector pins (ring groove vent channels) will be approved by the tool engineer. Contoured ejector pins must be approved by tool engineer. Contoured pins will be positively locked radially. Locking by means of pinning is not acceptable.

All ejector pin holes must be jig ground in the cavity block.

An ejector pin outside diameter will be located both on the bottom of the ejector pin and beside the counter bore for the ejector pin head. Use numbers for identification and include on the mold drawings.

Use "THX" Thru Hard DME ejector pins (Rc 65-74).

Ejector Blades

Ejector blades are used only in areas specified by the tool engineer.

Ejector Sleeves

Use DME or equivalent ejector sleeves.

Return Pins

Standard, hardened and nitrided, DME or equivalent return pins should be used. The minimum diameter of pins should be 0.625 in for molds designed for machines smaller than 550 t, 1.00 in diameter for larger molds; unless otherwise approved by the tool engineer. A minimum of four (4) pins per mold is required.

Return pin contact surfaces will be flat and perpendicular to mold travel, 0.062 in $\times$ 45° chamfer is required at the end of the pins. The maximum allowable span from pin to pin is 18.00 in.

Ejector Guide Posts and Bushings

All ejector plates are guided by a minimum of four hardened guide posts and bronze plated steel bushings. Guide posts are to be mounted in the cavity holder block and bushings in the ejector plates.

Ejector Plate Return

Ejector plates are to be spring returned. Accelerated or positive return mechanisms will be used in addition to the return spring when needed to prevent possible mold damage. Use a "Square D" Limit Switch (Class 9007 Type B 62G) mounted to sense that the ejector plate has returned.

Support Pillars

Support pillars should be placed as close as possible to the points of maximum loads, under the cavities, under the projected surface of runner and sprue systems, and under all slides. Support pillars are to be fastened by 3/8"-16 socket head screws to the clamp plate. Preload, if necessary, will be determined for each mold. Support pillars should be of adequate size and number to reduce movement of the mold during all phases of the molding cycle.

Stop Buttons

Standard "Stop" buttons should be used.

Mold General Assembly

All loose components that are to be assembled with the mold must be screwed, doweled, or keyed non-symmetrically for error proof assembly.

Wherever possible, cavity details are to be made removable from the mold without taking the mold out of the machine. Cavity and core inserts must extend from the parting line into the cavity pockets.

Angled Pin Slides

Mechanically actuated slides, properly heeled, are preferred to hydraulically actuated slides and should be used where possible. All sliding surfaces have a hardened steel-to-bronze or hard-on-hard surface contact and have provisions for lubrication. Heel surfaces will have hardened steel wear plates acting against bearing surfaces.

Slides opening to the outside of the mold must have positive stop blocks to prevent accidental drop out from the mold. A positive system to keep the slides in the proper orientation when the mold is open must also be incorporated. DME or HASCO standard slide retainers or custom built spring plunger locks are to be used (tool engineer to advise). All sliding mold components must have a provision for lubrication.

Use an angled lead-in on cam pins instead of a full radius.

Hoist Rings and Hoist Ring Holes

Safety is the number one priority when selecting and installing hoist rings onto the molds. Select hoist rings rated for a minimum of one and a half times the total weight of the mold. Hoist rings must be permanently mounted to the mold and should have means of securing them on the mold during the normal molding operation. The mold must balance within 5° in any direction when suspended on the hoist ring. If more than one hoist ring is installed on the mold, each hoist ring must be rated above the total weight of the mold. If a spacer block is required to elevate the hoist ring, the socket head cap screws securing the spacer block to the mold must have threaded engagement a minimum of twice the diameter and be sized so that each bolt can handle twice the total weight of the mold. Spacer block mounting bolts must be torqued to the recommended torque loads. Use DME hoist rings for molds up to 10,000 pounds and Danly hoist rings for molds between 10,000 and 20,000 pounds. Stamp in the location of the main lift ring ("Main lift ring location"), also stamp in the correct lift ring installation torque requirement ("Torque XXX foot-pounds"). Stamp tool number on the lift ring spacer block, if the mold is so equipped. Each mold will be provided with a minimum of two lift holes on each face of the cover and ejector half of the mold; i.e., top, operator side, bottom, and side opposite the operator.

Table 10-13 Recommended Lift Holes and Hoist Ring Tapped Hole Sizes

Thread size	4-ft capacity
3/4"-10 UNC	500.00 to 2,000 lb
1"-8 UNC	2,000 to 6,000 lb
1-1/4"-7 UNC	6,000 to 12,000 lb
1-1/2"-6 UNC	12,000 to 18,000 lb

Four (4) lift holes must be provided in each of the Q.C. clamp plates. Each lift hole must be rated for the total weight of the mold. Any modifications to the lift rings or the lift rings spacer block mounting bolts are strictly prohibited!

Slides and sections that weigh more than 50.00 pounds must have lifter holes (drilled and tapped) for removal and installation purposes. Number and location are dependent upon specific application.

Each mold half must be individually balanced for set-up purposes. In instances where putting lift holes on the centerline of gravity is physically impossible, three or more lift holes may be required for adequate balancing.

Mold Halves Interlocks

To ensure proper mold halves alignment, always use DME or HASCO interlocks. A minimum of four interlocks will be required per mold. Interlocks are to be positioned to ensure alignment of the mold halves, top to bottom and left to right. The tool engineer has to agree with the vendor, as to type, size, and location of the mold interlocks to be used.

Mold Cavity Pressure Monitoring

Button type pressure transducers are to be used. The tool engineer will select what type and how many pressure transducers to be used in the mold, if required. The connectors for the pressure transducers will be part of the system. Each connector will be identified by a number showing the transducer's location in the mold. A bolt on the protective plate is required over the connectors. The controller will be mounted on the operator's side of the mold. A metal plaque will be attached to the operator's side of the mold showing the location of each pressure transducer.

Mold Identification Tag

A metal mold identification tag is to be completed and mounted on the operator side of the mold. All metal plaques and identification tags are to be mounted on the mold using 1/4"-20 UNC screws.

Cavity or Core Inserts Sidewall Thickness

Wall thickness must be considered with extreme care to keep the cavity or core insert from deflecting, cracking, or breaking. The cavity insert wall thickness should be calculated based on the maximum length and depth of the cavity and the type of tool steel. The maximum sidewall deflection of 0.001 in is recommended for a cavity injection pressure of 10,000 psi. This cavity insert contact area should have a width large enough to hold the dimension requirements for the mold runner, gate, and venting system selected for this application.

Water baffle holes or any internal channels must have a full radius.

Any cavity insert thin walls or a weak structure area deemed to be a potential problem must be reported to the tool engineer.

Standard Mold Tool Steels

All standard tool steel and hardness areas must be reviewed with the tool engineer. A decision must be made if the standard tool steels are acceptable or different types of materials (stainless steel, high thermal conductivity copper alloys, wear resistant, etc.) are required for the mold base, cavity/core inserts, or other components of the mold.

10.23 Mold Tryout – Debug – Approvals – “MQ1” Requirements

10.23.1 Mold Tryout or Evaluation

The mold vendor delivers parts from the first molding run for evaluation. To identify the severity of injection molding problems, ten samples are required of the molded component in its print specified material and ten samples in its natural color version. A complete layout inspection of those parts molded in print specified material will accompany the samples.

The mold vendor will identify any known defects and show whether they will be addressed by mold modification or requests for dispensation. Be advised that samples with missing or undersized fillet radii are unacceptable. Dispensation for noncritical features and modifications to mating parts will be considered. The tool engineer will also consider vendor recommendations to improve the molded part's functionality or molding performance of the parts.

10.23.2 Mold Debug Procedures

The mold vendor is required to eliminate all dimensional, mechanical, electrical, hydraulic, and pneumatic errors, plus all water leaks and malfunctions in the tool. The mold vendor will develop the operating parameters with the tool engineer and modify the tooling as required to provide approved parts. Please note that some part drawings specify a moisture content and thermal conditioning before inspection.

The mold prints must be revised to show the final details of any modification made to produce molded components. At the conclusion of this phase, all molds will operate entirely as specified and designed and make molded components to the specified dimensions. The mold vendor is responsible for running as many tests as are necessary to approve the mold.

10.23.3 Approval of Molded Parts and Pre-Production Molding Process

Molded components supplied by the mold vendor upon completion of the debug phase are to be made in the mold under conditions closely approximating the intended molding production conditions for that mold.

These molded components should not be supplied to the tool engineer until the mold vendor feels that they conform to the part drawing(s) or computer files. Support layout documentation must accompany these molded parts. The mold must be running at the specified cycle time and all molding parameters must be documented at this time. A 100% part layout of each cavity is required when the mold is running to the correct molding conditions. Inspection and documentation of all molded component print dimensions are required.

10.23.4 Mold Cavity and Core Surface Temperatures

The mold vendor is required to provide a mold surface temperature map for both the cavity and the core for each half of the mold. Readings need to be taken when the mold is running in a molding production mode.

10.23.5 “MQ1” Requirements

The “MQ1” production run is conducted after the mold vendor has verified that the mold has been properly debugged and the pre-production results have been reviewed and approved. The “MQ1” run is a continuous run for a six hour period, starting from the time when the mold and molding machine are production ready (wide adjustment range of the molding parameters).

Ten subgroups of ten consecutive samples are to be pulled and labeled from each cavity at 30 minute intervals (the total number of samples from each cavity is 100 pieces). Material certification, including moisture analysis, is required for “MQ1” before run.

Measure all “KPC” dimensions with the same equipment being used at the company’s production facility. Provide 100% layout confirming the molded components to print from two molded components, for each cavity. The “KPC” dimensions must have the mean centered with a “PPK” greater than 1.67 from all the above samples and be statistically shown.

Any failures in the “MQ1” production run, or the sample parts dimensional correctness, will require a new “MQ1” production run. The tool engineer must be present for all mold tryouts, debug and the “MQ1” molding production run. The “MQ1” needs to be run on equipment that accurately reflects the intended production equipment. Parameters are those intended to be used in production. Any deviation will require written approval by the tool engineer.

The mold supplier is responsible for conducting as many mold trials as necessary to bring the molded components up to an “MQ1” level. One additional mold tryout will be required after the steel hardening process of the mold is completed and the mold is reassembled for the final time.