

Making Everything Easier!™

Injection Moulding Part Design

FOR
DUMMIES®

Learn to:

- Design better parts by keeping the process in mind
- Incorporate functional features into your design
- Understand the language of injection moulding

Brought to you by

proto labs®

Thom Tremblay



proto labs®

Real Parts. Really Fast.™

Proto Labs is the world's fastest manufacturer of custom parts for prototyping and short-run production. We use proprietary computing technologies and automated manufacturing systems to ship parts in as little as one day after receipt of your 3D CAD model.

If you only need 1 to 10 parts, Proto Labs' CNC machining service can make your part in a range of plastic or metals. For 10 to 10,000 parts, Proto Labs' injection moulding service quickly provides parts in engineering grade resins.

The Proto Labs sales and customer service team aims to provide you with the information and support your project needs. We hope this book answers many of the questions you have about designing parts for injection moulding. Some information included is specific to the Proto Labs process and relates to designing so that Proto Labs can manufacture your parts quickly. If you want more tips, tricks, and in-depth information, check out the library on our website for design resources including whitepapers, videos, and articles.

www.protolabs.co.uk

Injection Moulding Part Design

FOR

DUMMIES®

PROTO LABS® SPECIAL EDITION

by Thom Tremblay



WILEY

John Wiley & Sons, Inc.

Injection Moulding Part Design For Dummies®, Proto Labs® Special Edition

Published by
John Wiley & Sons, Inc.
111 River Street
Hoboken, NJ 07030-5774

www.wiley.com

Copyright © 2012 by John Wiley & Sons, Inc., Hoboken, NJ

Published by John Wiley & Sons, Inc., Hoboken, NJ

No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, scanning or otherwise, except as permitted under Sections 107 or 108 of the 1976 United States Copyright Act, without the prior written permission of the Publisher. Requests to the Publisher for permission should be addressed to the Permissions Department, John Wiley & Sons, Inc., 111 River Street, Hoboken, NJ 07030, (201) 748-6011, fax (201) 748-6008, or online at <http://www.wiley.com/go/permissions>.

Trademarks: Wiley, the Wiley logo, For Dummies, the Dummies Man logo, A Reference for the Rest of Us!, The Dummies Way, Dummies.com, Making Everything Easier, and related trade dress are trademarks or registered trademarks of John Wiley & Sons, Inc. and/or its affiliates in the United States and other countries, and may not be used without written permission. Proto Labs is a registered trademark of Proto Labs, Inc., and may not be used without Proto Labs, Inc.'s permission. All other trademarks are the property of their respective owners. John Wiley & Sons, Inc., is not associated with any product or vendor mentioned in this book.

LIMIT OF LIABILITY/DISCLAIMER OF WARRANTY: THE PUBLISHER AND THE AUTHOR MAKE NO REPRESENTATIONS OR WARRANTIES WITH RESPECT TO THE ACCURACY OR COMPLETENESS OF THE CONTENTS OF THIS WORK AND SPECIFICALLY DISCLAIM ALL WARRANTIES, INCLUDING WITHOUT LIMITATION WARRANTIES OF FITNESS FOR A PARTICULAR PURPOSE. NO WARRANTY MAY BE CREATED OR EXTENDED BY SALES OR PROMOTIONAL MATERIALS. THE ADVICE AND STRATEGIES CONTAINED HEREIN MAY NOT BE SUITABLE FOR EVERY SITUATION. THIS WORK IS SOLD WITH THE UNDERSTANDING THAT THE PUBLISHER IS NOT ENGAGED IN RENDERING LEGAL, ACCOUNTING, OR OTHER PROFESSIONAL SERVICES. IF PROFESSIONAL ASSISTANCE IS REQUIRED, THE SERVICES OF A COMPETENT PROFESSIONAL PERSON SHOULD BE SOUGHT. NEITHER THE PUBLISHER NOR THE AUTHOR SHALL BE LIABLE FOR DAMAGES ARISING HEREFROM. THE FACT THAT AN ORGANIZATION OR WEBSITE IS REFERRED TO IN THIS WORK AS A CITATION AND/OR A POTENTIAL SOURCE OF FURTHER INFORMATION DOES NOT MEAN THAT THE AUTHOR OR THE PUBLISHER ENDORSES THE INFORMATION THE ORGANIZATION OR WEBSITE MAY PROVIDE OR RECOMMENDATIONS IT MAY MAKE. FURTHER, READERS SHOULD BE AWARE THAT INTERNET WEBSITES LISTED IN THIS WORK MAY HAVE CHANGED OR DISAPPEARED BETWEEN WHEN THIS WORK WAS WRITTEN AND WHEN IT IS READ.

For general information on our other products and services, please contact our Business Development Department in the U.S. at 317-572-3205. For details on how to create a custom *For Dummies* book for your business or organization, contact info@dummies.biz. For information about licensing the *For Dummies* brand for products or services, contact BrandedRights&Licenses@Wiley.com.

ISBN: 978-1-118-20401-6 (pbk); ISBN: 978-1-119-47557-6 (ebk)

Manufactured in the United States of America

10 9 8 7 6 5 4 3 2 1



WILEY

Table of Contents

Introduction1

About This Book	1
Foolish Assumptions	2
How This Book Is Organised	2
Chapter 1: Wrapping Your Head around Injection Moulded Parts.....	2
Chapter 2: Pinning Down Your Job Needs.....	2
Chapter 3: A Beginner's Guide to the Perfect Part	3
Chapter 4: Going Beyond a Straight-Pull Mould	3
Chapter 5: Adding the Finishing Touches	3
Chapter 6: Ten (Or So) Ways to Give Your Parts the Proto Labs Edge	3
Icons Used in This Book.....	4
Where to Go from Here	4

Chapter 1: Wrapping Your Head around Injection Moulded Parts5

Getting Grounded in Injection Moulding	6
Meeting the machine.....	6
Making the part	7
Keeping design in mind.....	7
Breaking Down the Part-Making Process	8
Starting the part with the mould	8
Talking straight about straight-pull moulds	8
Knowing the core from the cavity	9
Doing a runner	9
Getting out of the gates.....	12
Keeping your cool.....	14
Prepping Parts for Proto Labs.....	16
Quote system	16
Part size and configuration	16

Chapter 2: Pinning Down Your Job Needs17

Getting Materialistic	17
Matching resins to part functions	18
Reviewing resin properties.....	19
Considering blends.....	21

Pondering the Need for a Pretty Part	21
The A side: Form	22
The B side: Function	22
Facing the Details	23
Building great walls	23
Making smooth transitions	23
Matching thickness to material	24
Making an important draft pick	25
Draft-angle guidelines	26
Texture effects	27
Rounding the bends	27
Parting is such sweet sorrow	28
Making Parts at Warp Speed	29
Getting that sinking feeling	30
Filling the mould	31
Chapter 3: A Beginner's Guide to the Perfect Part 33	
Ordering the Ribs	33
Meeting the Boss	35
Feeling a Draft	36
Keeping the Fillets Consistent	37
Getting a Feel for Texture	37
Featuring Multiple Features	38
Making the Write Decision about Text	40
Chapter 4: Going Beyond a Straight-Pull Mould 43	
Solving Problems from the Side	43
Knowing when to use side-action cams	44
Seeing what sizes the cams can be	45
Shutting off the action	45
Adding Resistance with Bumpoffs	48
Making Mouldmaking Simpler with Inserts and Pins	49
Picking out inserts	49
Pinning down the core	49
Designing for High Productivity	51
Designing a mould for multiple parts	51
Accommodating side-actions in multi-cavity moulds	51
Mirror, Mirror: Making Mirrored Parts	52
Chapter 5: Adding the Finishing Touches 55	
Colouring Your Parts	55
Colouring from the outside: Painting	56
Colouring from the inside: Adding colourant	56
Designing a Big Finish	58



Chapter 6: Ten (Or So) Ways to Give Your Parts the Proto Labs Edge.....	59
Proto Labs Know-How.....	59
Access to Experts.....	60
Proprietary Technologies and Tools.....	60
Real Parts Made Rapidly	61
Mass-Production Readiness	61
Multiple Finishes	62
On Demand Manufacturing.....	62
Glossary	63

Publisher's Acknowledgments

We're proud of this book and of the people who worked on it. For details on how to create a custom *For Dummies* book for your business or organization, contact info@dummies.biz. For details on licensing the *For Dummies* brand for products or services, contact BrandedRights&Licenses@Wiley.com.

Some of the people who helped bring this book to market include the following:

Acquisitions, Editorial, and Vertical Websites

Senior Project Editor: Zoë Wykes

Development Editor: Kathy Simpson

Editorial Manager: Rev Mingle

Business Development Representative:
Kimberley Schumacker

Custom Publishing Project Specialist:
Michael Sullivan

Composition Services

Senior Project Coordinator: Kristie Rees

Layout and Graphics: Tim Detrick

Proofreaders: Lindsay Amones,
Melissa Cossell

Special Help: Grace Fairley,
Amanda Graham, Barbara Moore

Dummies Publishing, United Kingdom

Michelle Leete, Vice President, Consumer and Technology Publishing Director

David Palmer, Publisher

Publishing and Editorial for Technology Dummies

Richard Swadley, Vice President and Executive Group Publisher

Andy Cummings, Vice President and Publisher

Mary Bednarek, Executive Director, Acquisitions

Mary C. Corder, Editorial Director

Publishing and Editorial for Consumer Dummies

Kathleen Nebenhaus, Vice President and Executive Publisher

Composition Services

Debbie Stailey, Director of Composition Services

Business Development

Lisa Coleman, Director, New Market and Brand Development

Introduction



We live in the Age of Plastic. Consumer items, medical devices, and pretty much every other types of products you can think of contain plastics. Plastics form structural components or provide a personal touch to consumer goods, making products feel smooth and light in your hand.

That's all well and good. The problem is that it can take forever to get a plastic part made. First, you have to know how a plastic part is supposed to be designed; next, you have to find a company that can make the parts; then you have to wait until that company has time in its manufacturing schedule to start your job. After you've waited a few months, you may discover that you didn't know as much about designing plastic parts as you thought you did. Worse, sometimes the people who make your parts let you know what is wrong with your design *after* the mould has been cut. (If you're lucky, the people who make the part also cut the mould, so there's a small chance that fixing the mould will be a priority — but either way, you're likely to be paying a price for it.)

This scenario is a scary one, isn't it? It's about enough to make you want to find a woodcarver who can whittle your parts for you.

About This Book

Injection Moulding Part Design For Dummies, Proto Labs Special Edition, provides insight into the Proto Labs process that makes it possible for you to get plastic parts made, not in weeks but in as little as one day. In this book, I give you tips on how to design your parts to make them better and easier to manufacture. I also tell you about design options that allow even complex parts to be made quickly. Further, if all that material isn't enough, I provide an overview of some of the materials available and show how you can make your parts look like a million dollars without spending that much or waiting forever.

Foolish Assumptions

Engineering departments and even private individuals now have ready access to design tools that are light years ahead of what was available even a few years ago. Being able to develop a part in 3D on a computer can help you better visualise the part and evaluate what needs to be included in the design. This will help with making sure it can be manufactured and be functional in the real world. The challenge is having the background in plastic-parts design to take control of these design tools.

This book assumes that you have access to design tools like these, have an existing design that you want to produce, and don't have the capability to do the manufacturing yourself.

How This Book Is Organised

I've organised this book into six chapters, each written to stand on its own, so you don't necessarily have to read it from start to finish. Feel free to jump in wherever it makes the most sense for you to start, based on your experience.

Chapter 1: Wrapping Your Head around Injection Moulded Parts

In Chapter 1, I introduce some of the terminology and provide an overview of the process of making an injection moulded part.

Chapter 2: Pinning Down Your Job Needs

In Chapter 2, the focus is on the function of the part and how it affects the material, the finish, and even the design approach. The chapter also introduces more complex concepts such as material selection, draft angle, and adding fillets to your parts.

Chapter 3: A Beginner's Guide to the Perfect Part

Mould design considerations for creating injection moulded plastic parts are the main topics of Chapter 3. I cover the critical details of mouldmaking, such as ribs, walls, and bosses, and how to design features that affect the look of the part — like adding texture or text.

Chapter 4: Going Beyond a Straight-Pull Mould

If you think your part is too complicated to be made quickly, Chapter 4 is for you. This chapter covers ways to make advanced features in the injection moulding process and explains that when one technique doesn't work, there's another that might do the trick.

Chapter 5: Adding the Finishing Touches

Chapter 5 introduces the all-important subjects of colour and surface finish. Knowing what options you have is important so that your parts look as good as you imagined they would.

Chapter 6: Ten (Or So) Ways to Give Your Parts the Proto Labs Edge

This chapter discusses how expertise, experience, and technology work together to make it possible for you to get your parts made quickly and productively with Proto Labs injection moulding service.

Icons Used in This Book

Occasionally, you'll see special icons that call attention to important information. Don't bother double-clicking the page; just sit up and take note! Here's what you can expect.



The Remember icon points out important information to help you retain the details.



When you see the Tip icon, you can be sure that it offers information that will transform your perception of the universe — or that it just points out some information that could come in handy while you're designing parts.



Pay careful attention wherever you see the Warning icon because failing to heed this information could cost you time, money, and/or damaged parts.

Where to Go from Here

You can take a second look at the Table of Contents to see what subjects you're familiar with, and then start with a familiar topic. But because the Proto Labs approach to moulding plastic parts is unique, I suggest that you may well benefit by reading from the beginning.

What's the worst thing that could happen if you read straight through? Take a chance, and see whether you find the Proto Labs approach to moulding parts as interesting as I do.

Chapter 1

Wrapping Your Head around Injection Moulded Parts



In This Chapter

- ▶ Getting the fundamentals right
- ▶ Seeing how injection moulded parts are made
- ▶ Ready parts for injection moulding



No matter where you turn, you see things made out of plastic, and most of those things are composed of plastic parts. Clearly, people have been making plastic parts for a long time, gathering a lot of expertise along the way, and now manufacturers can make those parts quickly and well. If you're reading this book, you're probably thinking about getting some plastic parts made, and you want to know how you, too, can do so quickly and well.

Most plastic parts are created in a process called *injection moulding*, which involves injecting molten plastic into an open space in a device called a *mould*. The name *injection moulding* isn't going to win any originality awards, but the process itself is creative because it helps people turn incredible ideas into real parts. Knowing how injection moulding works makes it easier to design parts for the process. This chapter gives you that understanding.

Getting Grounded in Injection Moulding

For all of the big-brained science involved in injection moulding, the process can be broken down into a few basic steps:

1. **Melt the plastic pellets.**
2. **Inject the melted plastic into the mould.**
3. **Let the moulded part cool.**
4. **Eject the finished part out of the mould.**

This section gives you a broad overview of these steps. For more detail, see “Breaking Down the Part-Making Process” later in this chapter — and the rest of this book.

Meeting the machine

Understanding how injection moulding works starts with understanding the equipment that makes the mould: the injection moulding machine (see Figure 1-1).

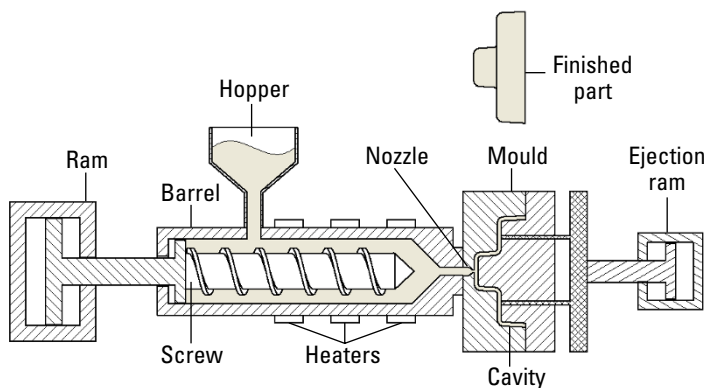


Figure 1-1: Schematic of a typical injection moulding machine.

Making the part

The injection moulding process starts with feeding pellets of plastic resin into the hopper of an injection moulding machine (see Figure 1-1), which melts those pellets through a combination of heat and pressure. The heat comes from electrical bands on the outside of the barrel. The pressure comes from a variable pitch screw in the barrel.

This screw drives the pellets from one end of the barrel toward the mould. The ram, similar to the hydraulic cylinders you might see on an earth-moving machine, builds the pressure needed to force the plastic into the mould. When the molten plastic resin is soft enough, the ram pushes the screw forward, driving the plastic through a small nozzle into a cavity in the mould, where the part takes shape.

After the mould is filled, it's left to cool.

Finally, when the plastic has cooled long enough for the part to harden, the mould is opened, and the part is ejected.

Keeping design in mind

Most of the work of designing an injection mould focuses on what happens between two points:

- ✓ When the ram moves forward
- ✓ When the mould is opened to produce the part

Most of this book focuses on how you can make sure that what comes out of the mould is what you need it to be. Keeping the mould in mind while you design a part helps you find ways to create high-quality parts cost-effectively.



If a part is designed in a way that allows it to be high quality in a prototype tool in low quantities, it's also going to be high quality when you want to produce a million of them. Also, understanding how a part will be used can save a dramatic amount of time by allowing you to look at various approaches.



Don't be too shocked if you're asked to change minor aspects of a part. When these requests come up, they come up for good reason, and the people who ask them don't take them lightly. Collaborating and working with experienced people ultimately gives you a better part in the long run.

Breaking Down the Part-Making Process

The previous section introduced the four basic steps of injection moulding, but there's much more to know. If you're interested in getting parts made, you need to understand all the details of the process because you need to pay extra attention to some fine points when you're developing parts to be moulded. This section gets you started on your detail work.

Starting the part with the mould

Because you can't have an injection-moulded plastic part without having an injection mould, it's important to start by thinking about how your mould will be made, how it will work, and how you can get the best possible part from it.

A mould has many components. At first glance, it looks like a bunch of metal plates sandwiched together — which, oddly enough, is pretty much what a mould is. The plates and pieces have different functions, however, and can require some really amazing skill to build.

Talking straight about straight-pull moulds

Although you will find some variations in injection moulds, the most common style is the *straight-pull mould*. This mould centres on a work piece made of at least two pieces of steel or aluminium, held together with mechanical clamps or hydraulic pressure while the plastic is injected and then pulled straight apart (surprise!) when the plastic has cooled enough to remove the part.



You can add pieces called *side-actions* (also called cams or slides) to the mould to create openings in the sides of parts or to create more complex features. Side-actions make it easier to create more complex parts while still using a basic mould.

Spinning the mould analogy

Here's another way to think about the halves of a mould: The halves are usually referred to as the A and B sides, much like the vinyl records of old. As with records, the A side is the

one that gets most of the attention, but the B side is probably more interesting. The two sides of the mould combine to enclose what is referred to as *the cavity*.

Knowing the core from the cavity

At the centre of the big mould assembly are the two halves that create the hollow where the melted plastic goes (see Figure 1-2).

These halves are the core and the cavity, and they work like this:

- ✓ **Core:** The *core* is usually the interior or noncosmetic side of the part. The core side also contains the ejection mechanism used to push the completed part out of the mould.
- ✓ **Cavity:** The *cavity* is the void inside the mould that the molten plastic fills. Plastic enters the mould from the cavity side and forms the final shape of the part. The cavity side usually forms the cosmetic side of the part.

To get the benefits of the Proto Labs process, all you need to do is focus on the design of the part and make sure that it follows the guidelines in this book.



Doing a runner

As “Making the part” explained earlier in this chapter, the molten plastic from the extruder is pushed into the mould, entering it through a series of channels called a *runner system* (see Figure 1-3). At different stages of the process, the parts of the channel have different names (sprue, runner, gate), but they're always part of the runner system.

Defining the runner system is one of the true arts of mould design. Being able to maintain a smooth flow of material while making sure the whole cavity fills — and fills correctly — is the sort of thing that makes rocket scientists nervous.

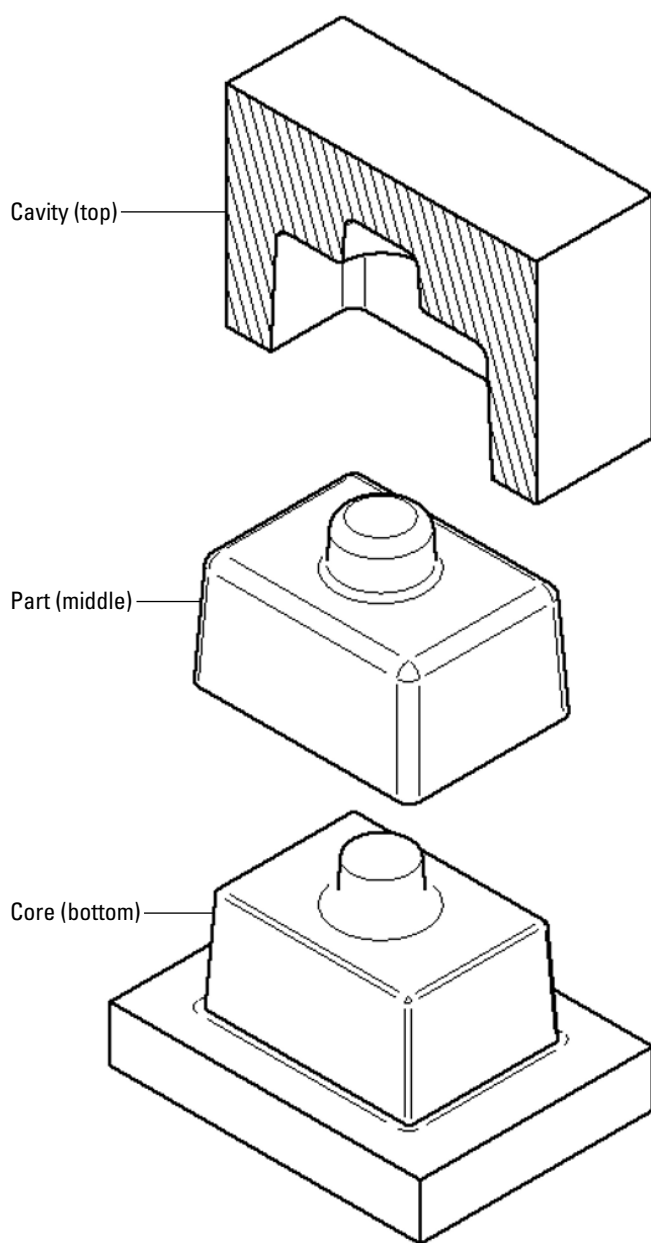


Figure 1-2: A part between a mould's core and cavity.

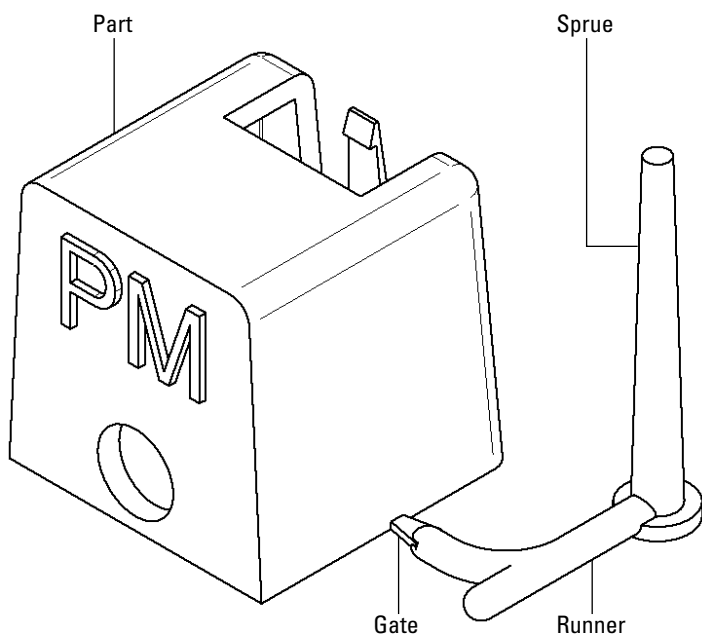


Figure 1-3: The runner system guides the melted plastic to the cavity that forms the part.

The runner system has to make sure that the mould can fill — but not too fast or too slow. It also has to make sure that the pressure doesn't get too high or too low. The runner system is usually defined when designing the mould.

If you've ever built a model car or an aeroplane, you've seen great examples of a runner system in the pieces to which the model parts are attached, as follows:

- ✓ **Sprue:** The *sprue* is the main channel through which the plastic enters the mould. It's typically larger than the other channels because the plastic for the entire part flows through it. In an aeroplane model kit, for example, the sprue is the thickest cylinder that rises above the rest of the plastic piece.
- ✓ **Runners:** The sprue connects to the *runners*, which connect all the parts and spread the plastic along the face where the halves of the mould meet. If you build a model

car from a kit, the runners are the sticks that run alongside all the parts and branch out from the sprue.

- ✓ **Gates:** The runners connect to the *gates*, which control the flow of the plastic into the cavity (I discuss this more in the next section). On your model car, the gate is where your part breaks off when you twist it.



You probably won't want people to see your finished part with the runner system attached. You can remove the part from the cool-looking runner system, and the runner can be appropriately admired or recycled by grinding it into pellets and running it back through the moulding machine to make the next part. It's like a polymer circle of life.

Getting out of the gates

After you create a runner system, the next elements of the mould that you need to work out are the *gates*, which are the connections where the runners meet the cavity. Gates come in a variety of shapes and can manoeuvre the plastic into the cavity in various ways.



The locations of gates are important because if you want a part to come out of the mould looking like you thought it would, you must make sure that the plastic flows to all parts of the cavity. If you put the gates in the wrong places, that flow won't happen.

Following are a few types of gates you may want to use:

- ✓ **Edge:** *Edge gates* port plastic into the cavity through the edge of the part (see Figure 1-4). Injecting the plastic through the edge leaves the runner and the part connected when the process is complete. It's easy to trim or break the part off the runner, but an edge gate leaves a small imperfection called a *vestige*.

If you need the edges of the part to remain clean, a couple of other gate types can keep your part looking cleaner.

- ✓ **Tunnel:** *Tunnel gates* inject the plastic into the cavity from a port that's cut into the core side of the mould and comes back up into a portion of the part (see Figure 1-5).
- ✓ **Post:** *Post gates* (see Figure 1-6) allow the plastic to be shot into the back of the mould via the paths of ejector pins (see the next section). The downside is that the ejector

will push the plastic out of the hole and leave it stuck to the part. If you need to, you can cut off the excess plastic, which is known as a *post* (hence the name post gate). So which came first, the post or the gate that formed it?

✓ **Hot tips:** *Hot tips* are gates that connect the sprue directly to the part. A tip is placed in the part's cavity and heated so that the part doesn't stick to it, but a dimple has to be added to the part to allow the plastic to flow out of it properly (see Figure 1-7). Moulds using hot tips waste almost no plastic.

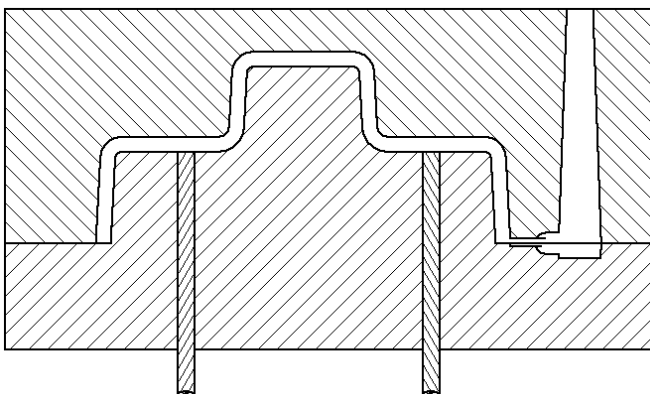


Figure 1-4: An edge gate.

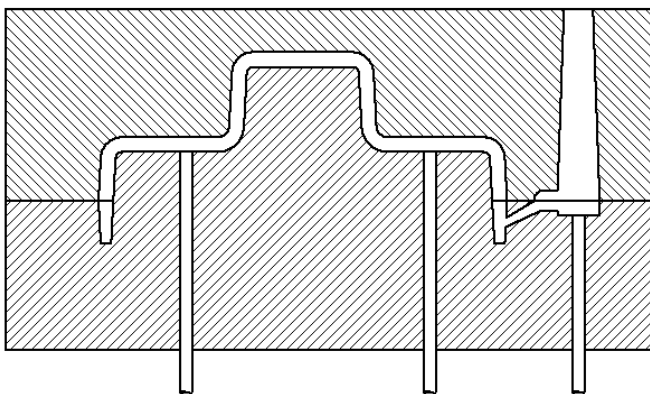


Figure 1-5: A tunnel gate is cut off when the part is ejected.

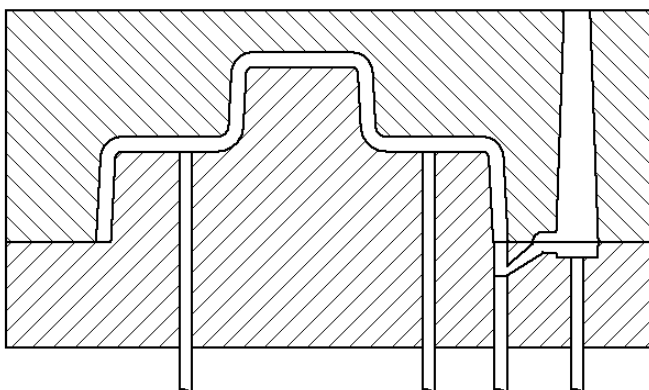


Figure 1-6: A post gate.

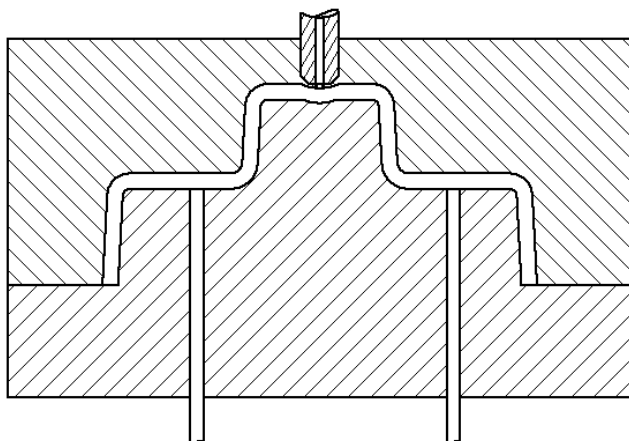


Figure 1-7: A hot tip gate.



A lot of thought is put into designing the cavity, the runner system, and the gate placement. Whether you need a short run of parts or a large-scale production, if a mould isn't designed to produce a quality part, it isn't useful.

Keeping your cool

Because the mould holds the plastic's heat a little too well, you need to cool the mould with water to help the part solidify

faster. Then, when the part has cooled and solidified, you can open the mould. You generally have to push the part out of the mould by using *ejector pins* (see Figure 1-8).



If you don't wait long enough before opening the mould, your part will be distorted — which is generally considered to be a bad thing.

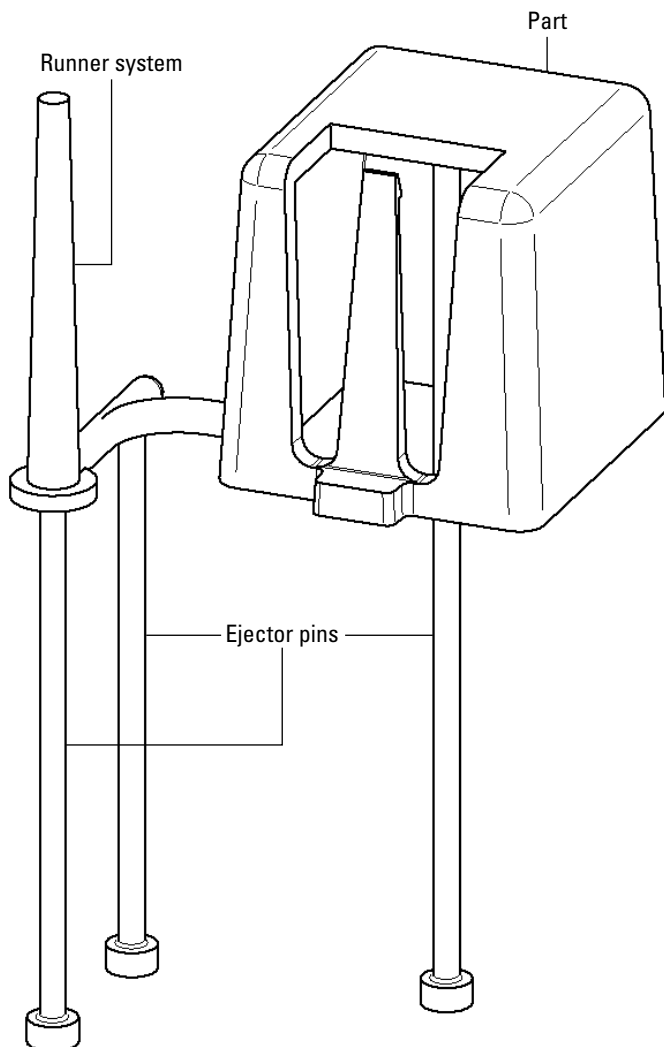


Figure 1-8: Use ejector pins to push the part out of the mould.

Prepping Parts for Proto Labs

To use the Proto Labs injection moulding process, you need to consider additional factors when you plan the part design. So what is this Proto Labs process? I'm glad you asked.

Quote system

The Proto Labs process starts with an automated quote system that lets you select the options you want and gives you pricing information along the way. Then Proto Labs uses your CAD model to build a mould from it quickly, which lets them mould your part in practically no time.

Part size and configuration

To get a plastic part created quickly, you need to limit a few things, such as the size of the mould and how much plastic you can put in it.

Accurately calculating the size of a part can be complicated. Proto Labs' online quoting system automatically determines the necessary size of the mould and whether it exceeds the maximum size that can be made using its process.



Now you know the basics of putting plastic into shape. Before you dig into the deep details of designing plastic parts, however, you need to know how to make a part simple and to the point so that it does its job correctly (see Chapter 2).

Minimising mass

In designing some parts, you may wish to minimise mass. Here are a few considerations:

- ✓ The thickness of the part's walls, the size and volume of internal features, or even large, solid portions
- of the part may cause the part to distort in undesired ways.
- ✓ Ribs or other reinforcing features may have to be added, removed, or relocated to make the part easier to mould.

Chapter 2

Pinning Down Your Job Needs

.....

In This Chapter

- ▶ Picking the perfect plastic
 - ▶ Making parts pretty (or not)
 - ▶ Sweating the small stuff
 - ▶ Fighting warp
-

What is your part going to do? Is it going to serve as the housing of a consumer product that people will hold for hours on end? Will it be a mount for a circuit board in that product that no one will see? Function affects form for any type of part, but with plastic parts in particular, it also affects finish — and even the design of the mould that shapes them.

This chapter helps you pin down just what you need your parts to do so that you know just how to design them.

Getting Materialistic

Before you can worry about the geometry of a plastic part, you have to know what type of plastic you want to use.

Many types of plastic materials are available, but in their raw form they're called *resins*. Resins can be combined with other resins or with filler materials to give plastics special properties. All these materials change after they've been injected into a mould. As the materials cool, the features of the part shrink and take on their physical (*mechanical*) properties. (For more on shrinkage, see "Making Parts at Warp Speed", later in this chapter.)

Matching resins to part functions

How do you determine what type of resin is best suited for a particular part? Consider what you want the part to do, as well as the following:

- ✓ **Mechanical properties of the part:** Mechanical properties include the strength of the part at normal and high temperatures and its impact resistance.

If your part needs minimal strength at room temperature and high impact resistance, for example, a huge number of resins in several categories can do the job.

- ✓ **Characteristics of resins:** Considering the characteristics of various resins, such as how resistant they are to deforming during cooling and how well they fill small features of a mould, also narrows the field.
- ✓ **Special considerations:** Your part may have additional specifications, such as a need for FDA or UL ratings, which can further constrain your choice of resin.
- ✓ **Cost:** If the resin that best meets your needs is too expensive or has other limiting factors, you may be able to

adjust the design so that the part can be made effectively from another resin.

- ✓ **Need for reinforcing fibres:** When additional strength is needed, fibres of different kinds can be added to the resin as it melts in the extruder (see Chapter 1). These fibres are injected into the mould along with the resin; they align with the flow of the material to reinforce the part, much as rebar reinforces concrete.



The addition of fibres creates more challenges in mould design, but it may allow you to use plastic instead of metal for some structural parts.

Every type of material has rules and guidelines that relate to how small a feature can be, in order for it to fill properly. Working with the experts at Proto Labs, you can use these guidelines to make it easier to prepare your parts for rapid production. If a critical feature is too small for a particular resin, they can provide you with the resources to help you choose a suitable resin.

This section gives you some guidelines on plastics to help you make the best choice for your product.

Reviewing resin properties

Table 2-1 lists some of the properties of common resin types, some of which you may already have heard of.

Table 2-1 Properties of Common Resins						
Resin Type	Strength	Impact Resistance	Dimensional Accuracy	Capability to Fill Small Features	Performance at High Mould Temperatures	Cost
Acetal	Medium	Medium	Fair	Fair	Fair	Medium
Acrylic	Medium	Low	Good	Fair	Good	Medium
Acrylonitrile butadiene styrene (ABS)	Low to medium	High	Good	Fair	Good	Low
High-density polyethylene (HDPE)	Low	High	Fair	Excellent	Good	Low
Polycarbonate (PC)	Medium	High	Good	Fair	Good	Medium to high
Polycarbonate/ABS alloy (PC/ABS)	Medium	High	Good to excellent	Fair	Good	Medium
Polypropylene (PP)	Low	High	Fair	Excellent	Good	Low
Polystyrene (PS)	Low to medium	Low	Good	Good	Good	Low

Considering blends

The material *is* your part, so its capability to perform to your needs isn't optional. If a standard material doesn't provide everything you need on its own, you may be able to find a blended resin that meets your needs.

As Table 2-1 shows, for example, when polycarbonate and ABS are combined, the new material is stronger and is capable of forming a part more accurately than ABS alone.

Pondering the Need for a Pretty Part

Next, consider how pretty your parts need to be. Parts that are going to be hidden from view don't have to please the human eye, and they can have more imperfections than you normally want visible (*cosmetic*) parts to have. Figure 2-1 shows the hidden B side of a plastic part; its features make the part work but don't necessarily make it pretty.

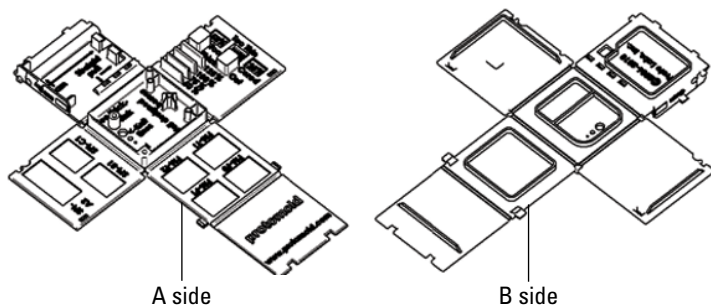


Figure 2-1: Precise features and fine edges need extra attention when you're creating a mould.



To make the A side of a cosmetic part look good, however, you have to put a lot of thought into the B side.

The A side: Form

Normally, the A side of a part is the cosmetic side and the B side is less cosmetic. This is easier to determine on parts with an “inside” and an “outside,” such as a case or enclosure. It’s not uncommon to find a part (like a lens) where both sides are cosmetic or a part with no apparent cosmetic sides (like a gear or structural piece).

Making parts with beautiful surfaces moves the science of mould design toward an art form. It takes years of experience accelerated by the latest technology to ensure that fine surfaces aren’t distorted by the injection moulding process. Guesswork isn’t an option, regardless of the lead time on a part.

Not all cosmetic parts are highly polished, of course. Some of the most elaborate plastic parts you’ll come across are parts you’re not supposed to see. A single plastic part inside electronic equipment or a power tool may replace several metal parts, and it may have complex features and mounting points to other parts that make a smooth finish impractical.

If you want your parts to have a textured finish, however, you introduce a whole different set of design considerations. You may have to change how the part is shaped to allow it to come out of the mould without spoiling the texture. I look more closely at this topic in Chapter 3.



Depending on the texture, you can even use the finish to conceal imperfections. Small flaws that would require a lot of changes to make a shiny part acceptable may not be noticeable when a texture is applied.

The B side: Function

As long as the positions and sizes of any structural elements are within specification, you can be flexible in creating moulds for hidden parts, putting gates and other features where they work best without worrying about how they look.

You also have more flexibility when you’re working with the experts at Proto Labs to make changes in a hidden part.

Changing the thickness of a wall or the location of a rib, for example, may make the part easier and less expensive to make without affecting its ability to perform its function.

Facing the Details

When you're looking at a well-designed injection-moulded plastic part, a few details may stand out:

- ✓ Most, if not all, of the main walls have consistent thickness.
- ✓ The faces of the part are angled around the main shape.
- ✓ Many of the part's corners are rounded.
- ✓ The part has a thin line around it.

These four details occur by design, not coincidence. Fortunately, you're not the first person who ever needed to have plastic parts made, so manufacturers know a lot about how these details need to be worked out. The following sections discuss these details one by one. Keep them in mind as you design your part.

Building great walls

Whenever possible, you should keep the thickness of a part's walls consistent — not just for simplicity but also for strength. Plastic shrinks as it cools, and it cools from the outside in. This can cause the walls to be pulled inward (called *sink*), or it can cause internal stresses or voids, or all of the above. If a mould has varying wall thicknesses, you have to work even harder on the design to keep the part looking like it's supposed to when it comes out of the mould.

Making smooth transitions

If a face needs to have different thicknesses, or if the thickness of adjacent faces must be different (see Figure 2-2), you need to make sure that the transitions are smooth to limit the stresses on the walls and minimise the differences in shrinkage as the material cools. Sharp corners also cause stress risers in the part, becoming a potential failure point.

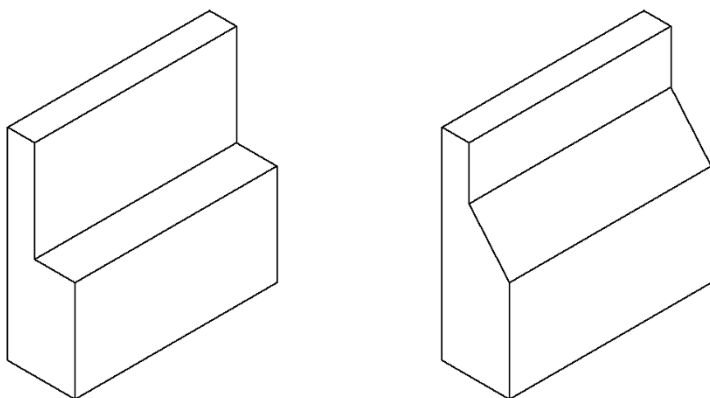


Figure 2-2: Changing a face thickness should be done with a transition.

Matching thickness to material

Another consideration is the material you'll be using because the ability to make very thick walls or other features without warping is a characteristic that differs from resin to resin. Table 2-2 shows some examples of recommended wall thickness for various types of resins.

Table 2-2 Recommended Resin by Wall Thickness

<i>Wall Thickness (mm)</i>	<i>Resin Type</i>
0.6–3.8	Acrylic (PMMA)
0.6–3.8	Polypropylene (PP)
0.8–3.0	Acetal (POM)
0.9–3.8	Polystyrene (PS)
1.0–3.1	Polycarbonate (PC)
1.1–3.6	Acrylonitrile butadiene styrene (ABS)



TIP

As you can see, the resins listed in Table 2-2 have a lot of overlap. If you can incorporate these common wall thicknesses into your design, you keep your resin options open wider. This approach may seem to be a bit “chicken or egg,” but when you give it a little time to sink in, it makes the whole process easier.



It's better to err on the side of making features thin because if an initial test moulding shows that a feature is too thin, it's easier to machine the mould to make that feature thicker than it would be to make a thick feature thinner, which could require recutting the entire mould.

Making an important draft pick

In a straight-pull mould (see Chapter 1), the side walls of the part are more or less parallel to the mould's *direction of pull* — the direction in which the part will be pulled out of the mould. Notice that I say *more or less*, not *exactly*. To ensure that the part comes out of the mould smoothly, you need to make sure the walls *aren't* exactly parallel: They need to slant as they're cut into the mould so that the part will come out of the mould easily (see Figure 2-3).

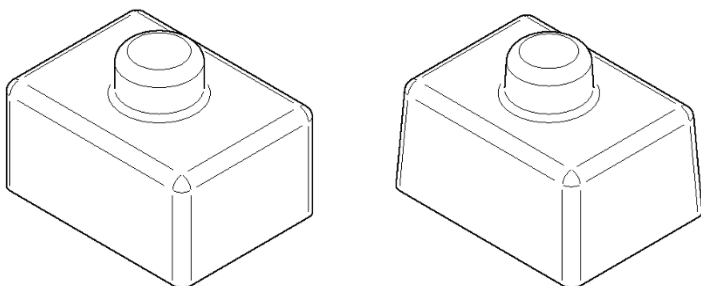


Figure 2-3: Parallel walls (left) create friction in the direction of pull; tapered walls (right) reduce it.

If you don't include enough wall angle — a feature called *draft* — you can get cosmetic flaws in the part, called *drag marks*, that are the result of the part sticking to the mould as the part is ejected. See Figure 2-4.

The key to using the draft concept properly is knowing what angle the draft needs to be. In keeping with the long-standing tradition of creative naming in injection moulding, this angle is called *draft angle*, and it's another characteristic that you need to think about when you design a part.

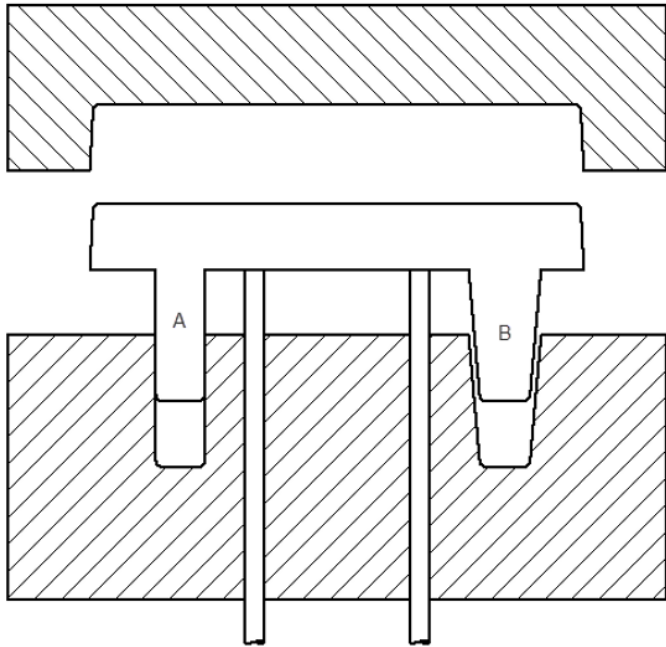


Figure 2-4: The undrafted rib (A) scrapes all the way out, while the drafted rib (B) pops out unblemished.

Draft-angle guidelines

People who've been making moulds for a long time have worked out some really good guidelines for draft angle, and those guidelines are pretty straightforward — at least, as long as no surface textures are involved (see “Getting Materialistic” earlier in this chapter). For plastic without a surface texture, you can base draft angles on the height of the feature. A wall that's 50mm high can have a 2-degree draft angle; one that's 75mm high can have a 3-degree draft angle, and so on.

This angle is applied on each face, so you also need to keep track of that wall-thickness thing (see “Building great walls” earlier in this chapter). It's not uncommon for a part designer to start with a good wall thickness but not consider the effect of draft, so the end of the wall or rib comes out too thin for the material or the base of the wall or rib comes out too thick.

Texture effects

If your part has textured faces, however, the draft-angle guidelines change. You need to provide enough of an angle so that when a textured face shrinks away from the wall, it shrinks far enough away to keep the texture from getting hung up in the tiny pockets on the mould surface that created it. The rougher the texture, the greater the draft angle needs to be to prevent drag marks.

Proto Labs' quoting system analyses the draft angles of your part and shows you areas that aren't drafted enough to use texture. A face using a light bead blast texture may need only 3 degrees of draft, whereas a medium bead blast texture may need 5 degrees of draft.

Rounding the bends

Rounded corners (see Figure 2-5) allow the material to flow more efficiently during injection and also reduce the stresses on the material during cooling, which helps reduce the part's tendency to bow, warp, or develop fragile corners.

The key to reducing these problems is maintaining thickness through the corner. Making the radius of *interior fillets* (the insides of corners) on the part at least half the thickness of the material is helpful. More is better, but it's important to keep the exterior radius in ratios so that thickness remains consistent. Keep the following tips in mind:

- ✓ The interior radius should be at least half of the wall thickness. The exterior radius should be the inside radius plus the wall thickness.
- ✓ Using a larger-exterior radius without adjusting the interior radius will thin the wall around the corner.
- ✓ Using a smaller outside radius will create a thickening around the corner that can also obstruct the flow of resin in the mould.
- ✓ It's also important to include a radius on edges of interior features to aid the flow of the plastic and to improve the strength of the part. This also helps to reduce wear in the mould itself.

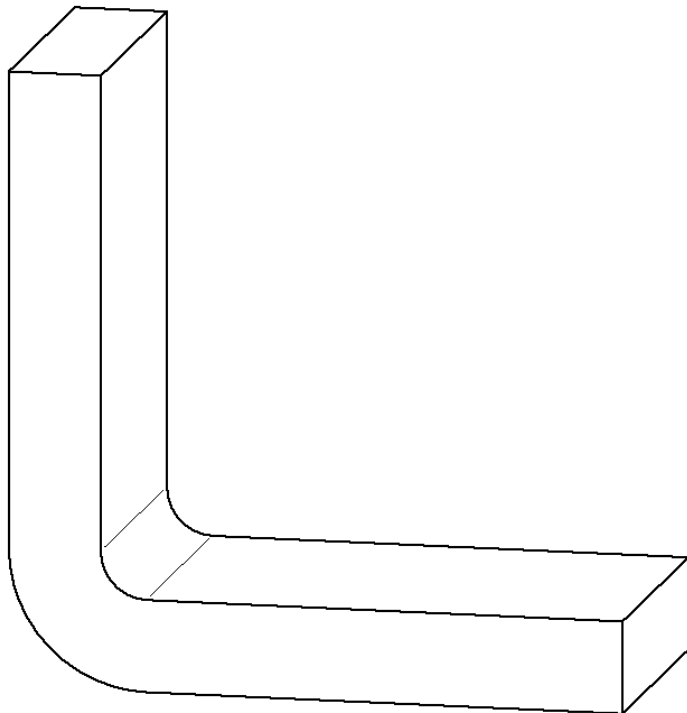


Figure 2-5: Round corners reduce stresses inside the mould.



Not every corner of a part is visible from the outside. The B side of the part may have a lot of edges that need to be rounded.

Parting is such sweet sorrow

The two halves of the mould meet at what is called the *parting line*. Depending on how the mould designer created the A and B sides, there may be a thin blemish around the outside of your part (see Figure 2-6). If you need to have a sharp edge on a part, it's best to try to have that edge make up part of the parting line because it reduces or eliminates the blemish.

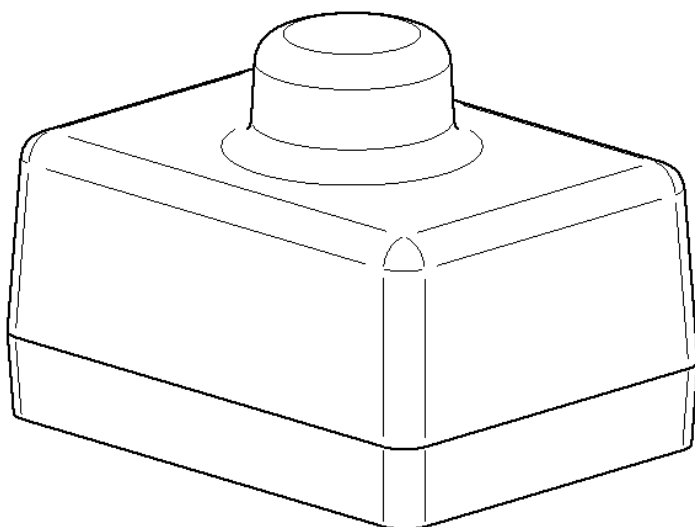


Figure 2-6: A parting line may appear as a fine edge on a part.

The faces where the parts of the mould meet don't need to have a radius. If you need to apply a radius to the end of a wall that ends where the mould separates, you should set the parting line back from that actual face and place it where the radius you need comes tangent to the next face. Moving the parting line eliminates what would otherwise be a razor-sharp edge, which will wear quickly. This is important even for a mould that won't be used to make thousands of parts because things that are good for the mould are good for the part.

Making Parts at Warp Speed

A common problem with an injection-moulded plastic part is warp. *Warp* occurs when the features of a part bend, or the whole part bends, as the material cools (see Figure 2-7). This distortion can happen if portions of the part have inconsistent material thickness, or sharp corners, or thick or thin areas, and so on.

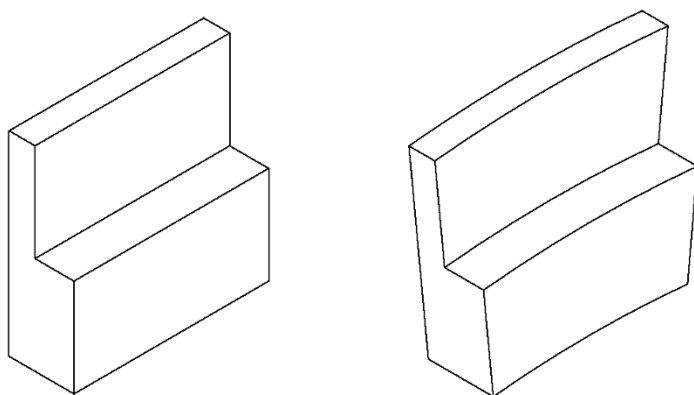


Figure 2-7: A normal part (left) can get warped (right) during cooling.

You can sometimes reduce part warping by changing the locations and shapes of gates or by enhancing the mould's cooling capacity.

Getting that sinking feeling

Sink (see Figure 2-8) is a type of warp that happens in the middle of a face when the material is too thick. The hot material fills the cavity completely, but when it cools, it contracts toward the centre of the feature's volume, causing the surface to sink. (I just don't know where they came up with that name.)

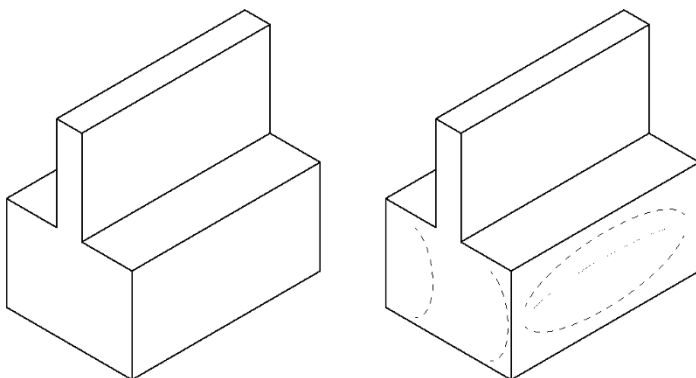


Figure 2-8: Sinking causes hot material in a mould to . . . well, sink.

Filling the mould

If your part has many holes and other features, the resin may have to flow around those features in several directions. When these separate flows meet, they may not bond completely, so you may see deformations or discolourations. These imperfections are called *knit lines*, *weld lines*, or (when plastics come from more than one gate) *meld lines*. For some types of internal parts, you may not need to worry about these lines.

If your part has dead-end areas in the mould, such as deep ribs and bosses, air may become trapped. As the plastic compresses the air, the temperature rises, which means that the feature may not fill, or the increase in the air's temperature may cause a burn mark on the feature, or both. To prevent this, vents are often added to dead ends in moulds.

Also, in “Making smooth transitions,” earlier in this chapter, I point out the importance of keeping transitions between thicknesses smooth. If the mould isn't designed properly, variations in thickness can prevent plastic from filling part of the mould and can leave open portions in the part (see Figure 2-9).

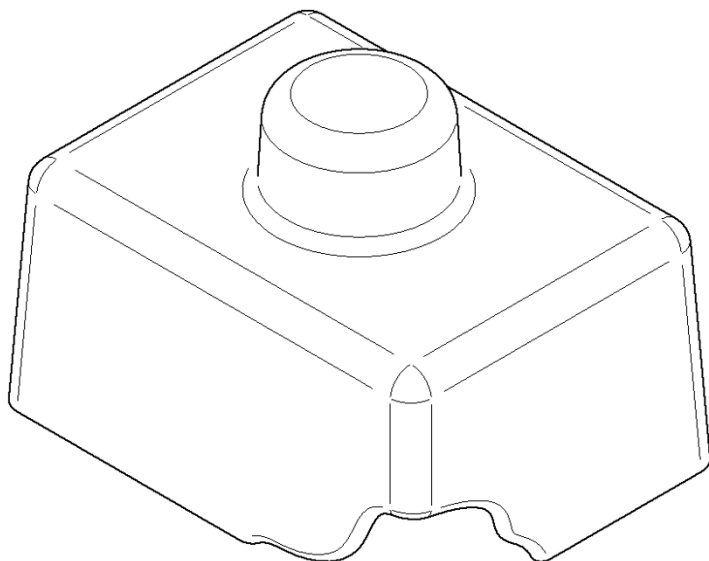


Figure 2-9: If the plastic isn't in the mould, it isn't in the finished part.

What could possibly go wrong?

Everything on a plastic part deforms. The mould itself is built larger than the part needs to be because the part will shrink into the proper shape as the material cools. If a mould is made correctly, the completed part will be so close to the designed part that any difference will be hard to detect.

If the work is rushed, however, and the mould isn't designed to help the

part, the final part may not be right. This is why most people who produce plastic parts don't attempt to make parts rapidly. They haven't developed processes to build consistent outcomes into the initial design. If the design is done correctly, however, plastic parts can be made quickly and accurately.

Chapter 3

A Beginner's Guide to the Perfect Part

In This Chapter

- ▶ Studying the anatomy of ribs, walls, and bosses
 - ▶ Keeping your fillets consistent
 - ▶ Feeling out texture
 - ▶ Choosing a fine font for text
-

The previous chapters talk about the process by which plastic parts are made, as well as some big problems involved in part-making and a few approaches that make the process easier. This chapter looks at the details of preparing parts for rapid moulding, focuses on the features of plastic parts, and suggests design approaches that can make manufacturing those parts easier.

Ordering the Ribs

Quick — find something plastic, and open it. Almost every plastic part you can find has at least a few ribs in it. *Ribs* are the internal walls of a part. They have many functions and play many roles. They support the outer surface and often create bridges between features. Ribs usually have a function after the part is out of the mould, and they also help to make the moulding process easier by adding channels for the plastic to flow through.

Just like walls, ribs have some quality-control guidelines:

- ✓ **Thickness:** If ribs are too thick, they cause a sink in the surface opposite the rib when they cool. (For details on sink, see Chapter 2.)

To help prevent this problem, a rib thickness of 40–60 percent of the adjacent wall's thickness (see Figure 3-1) is recommended so that the rib cools first. If you think your ribs should be thicker than the recommended size but you're not sure, it's better to try the thinner feature in case it has to be thickened. To make a rib thicker, you just cut a bigger groove in the mould. It's hard to add material to the mould to make the rib thinner.



Rib thickness isn't a constant, because you may need to include draft too, so consider the thickness of the base of the rib for your measurement. (For more about draft, see the section "Feeling a Draft," later on in this chapter.)

- ✓ **Draft angle:** While you're already having fun with your calculator, you also need to make sure that the draft angle (see Chapter 2) doesn't cause the rib to taper to a sharp edge. Not only would the rib be difficult to machine into the mould, but you might get a nasty cut as well. That wouldn't be fun at all.

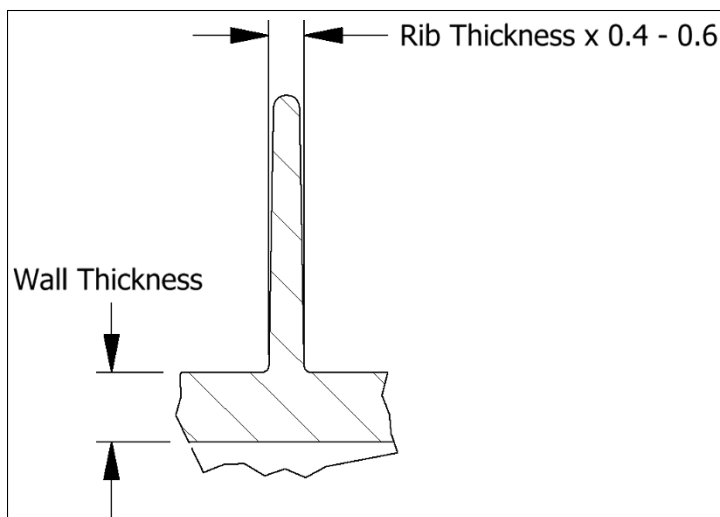


Figure 3-1: A rib should be 40–60 percent as thick as the adjacent wall.

Meeting the Boss

After the ribs, the most common internal features of plastic parts are probably the *bosses*. A boss is a raised portion of the part. It rises above the surrounding geometry and can be on either the A side or the B side depending on its role. Bosses perform several functions, including supporting other parts and creating a place for screws to connect to or pass through.

A really good way to think about a boss is to think of it as a rib that goes in a circle (see Figure 3-2). In the figure, you see how you must consider draft when establishing the thickness of the feature.

As with ribs (see “Ordering the Ribs” earlier in this chapter), you want to make sure that the boss doesn’t have too much wall thickness. If you’re concerned that a boss won’t be strong enough, you can add ribs to support it. With a hollow boss, you should make sure that the opening in the middle goes all the way down to the connecting face (see Figure 3-2) so that the part doesn’t have an unplanned thick spot.

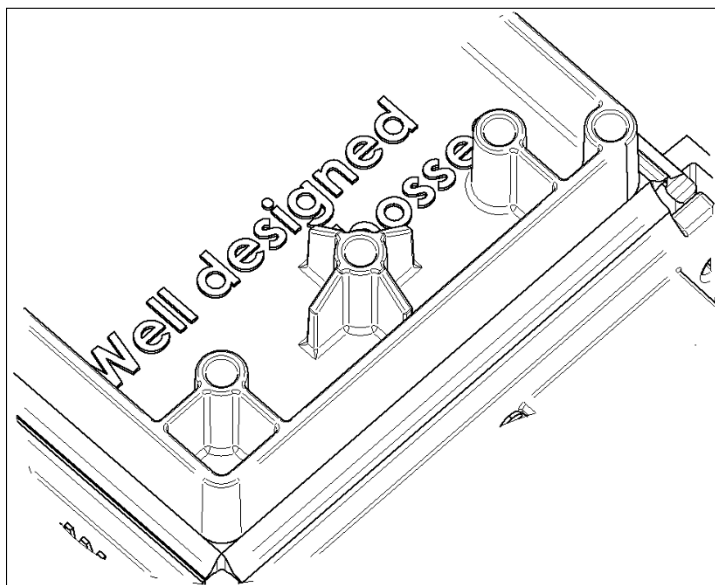


Figure 3-2: A collection of well-designed bosses.

As with all features, excessive material in bosses leads to sink marks when the part cools. (Check out the earlier section “Ordering the Ribs” for guidelines about rib thickness.) A common source of sinking is when a partially hollow boss has too much material at the bottom. The bottom of the boss should follow the thickness of the adjacent face if it’s not going all the way through, as shown in Figure 3-3.

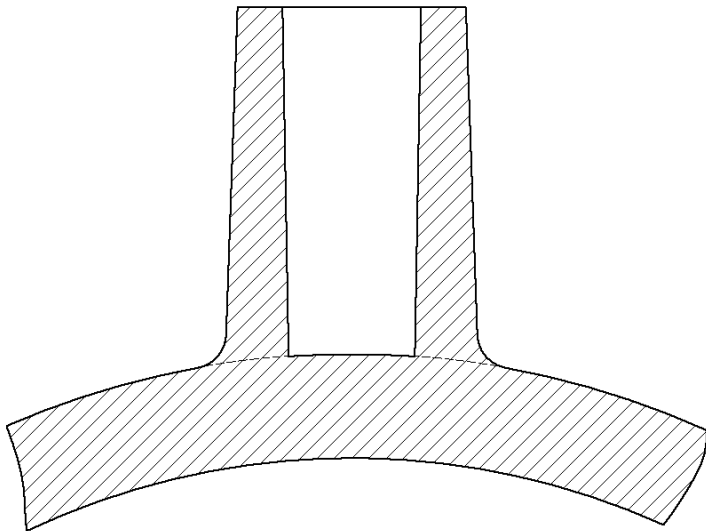


Figure 3-3: Allow the bottom of a hollow boss to match the natural floor.

Feeling a Draft

If ribs, walls, or the part as a whole have to be cut deeply into the mould, you may have to change the draft angle as the cut gets deeper. In Chapter 2, I talk about a 50mm rib needing a 2-degree draft and a 75mm rib needing a 3-degree draft. This can be difficult to machine into the mould because small diameter endmills break when they are used to mill deep ribs. Adding draft allows a larger endmill to be used, which will be less likely to break.

Possible solutions include stepping the feature so that the cutter can get into the mould deeply enough to give the edge of the feature the required width. Sometimes, the parting line

itself (see Chapter 2) can be shifted to reduce the cut depth, as shown in Figure 3-4. This method can also keep features from getting too thin or thick by eliminating the entire draft from being in one direction.

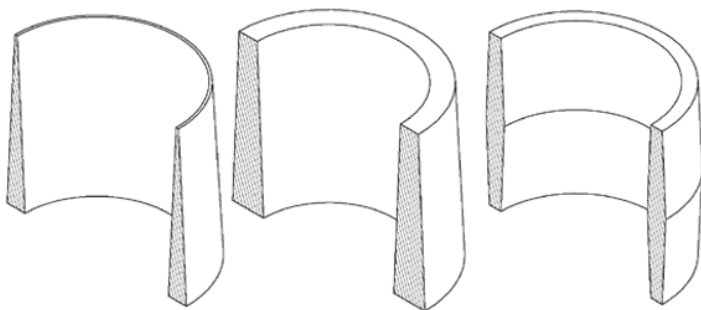


Figure 3-4: Wall height may need to be divided to reduce the effect of draft.



In general, using 3 degrees or more of draft will make things much easier all around and eliminate additional work to accommodate deeper cuts.

Keeping the Fillets Consistent

The fillets on a part are important for controlling material flow — and for one other important task. To keep costs down and to speed the process of creating your plastic parts, it's very helpful to minimise the radii used on the parts' fillets. Minimising the fillet may be in conflict with other plastic design best practices and/or mechanical engineering principles.



Consider focusing on an interior radius of one half of the standard wall thickness and an exterior radius of the thickness $\times 1.5$. (If you've read Chapter 2, it's like déjà vu. It's still a good tip worth repeating though.)

Getting a Feel for Texture

Choosing a texture for your part really depends on the application. A coarse texture can help conceal imperfection, and it can also make the part pleasant to touch and easy to keep

looking clean. At the opposite end of the scale is the high-grade polish that's used for so many of the plastic parts of consumer electronics.

Even though you're looking to get a part made quickly, you can still include a texture or polished finish to gain a good understanding of what your part will look like when it is in high-volume production.

Also, you can use more than one texture on a part. On the A side, for example, you may want a medium texture to provide a softer look, but you might also need to apply a nameplate in a recess. Because the nameplate will adhere much better to a smooth surface than to a textured one, you can leave the texture off the recess area for functional reasons. Or you may want to vary the texture for aesthetic reasons.

As for the B side, you may not even need to think about a texture, depending on the part and how pretty you need it to be (see Chapter 2).



Keep in mind that when using texture, design elements like draft angle may have to be noticeably different. As mentioned in Chapter 2, draft angle is related to height, so a 25mm-tall rib needs only 1 degree of draft when not textured and would need 5 degrees of draft with a coarse texture. These changes also apply to bosses and other tall, thin features that have to be drafted. Drafting is done to prevent drag marks on the texture when the part is pulled out of the mould.

Proto Labs offers several standard textures that give you a variety of choices but still permit parts to be produced quickly. These textures range from as milled machining marks to a mirror finish.

Featuring Multiple Features

At the risk of repeating myself, parts need to have consistent wall thickness. But how do you achieve consistent thickness when you need a big feature on the top of your part or need to create something like a roller (see Figure 3-5)? Simple: Just build your part from a series of walls (see Figure 3-6) rather than make it one big solid feature.

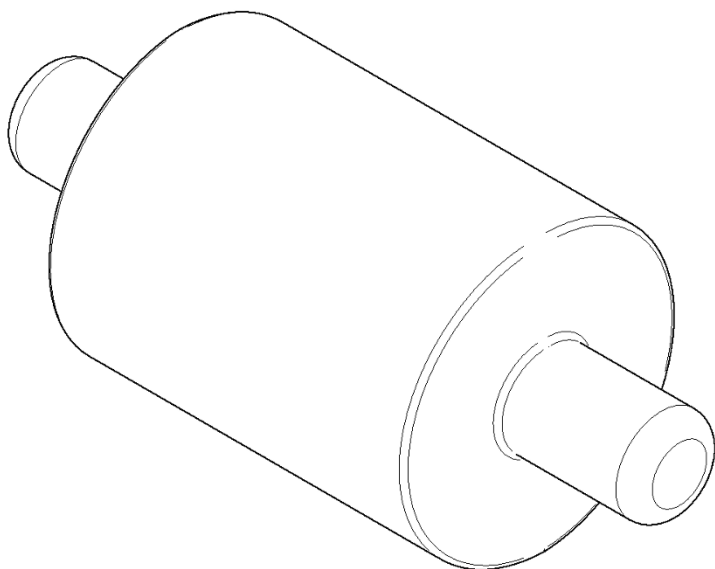


Figure 3-5: To make a spacer . . .

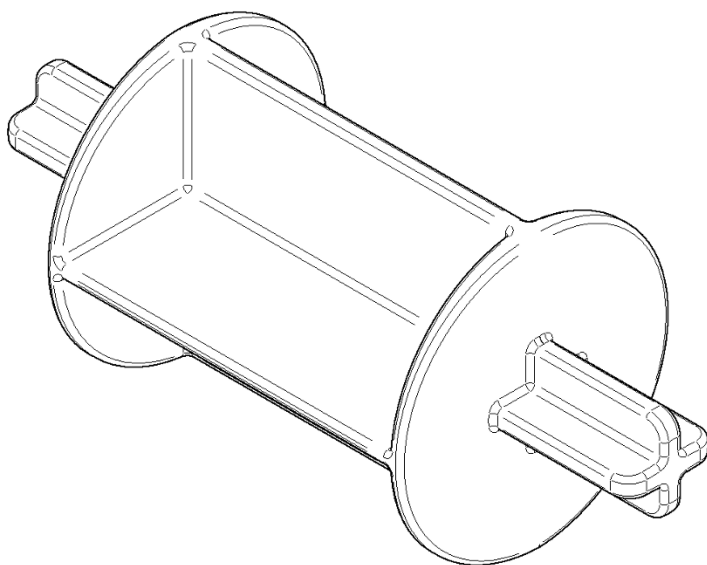


Figure 3-6: . . . make a series of walls.

By eliminating volume that could distort the part as it cools (see Chapter 2), you can do a better job of keeping the critical faces of the final part in place. Because this isn't a normal train of thought, going through the process of designing parts for injection moulding will help you to recognise opportunities to approach the design of your part a little differently. This will make it easier to design your other parts with "mouldability" in mind.

This process is different from machining material away from a solid piece of metal to create the final part. In injection moulding, the part is created by flowing plastic material into a mould, so rather than have machinists drill a hole in a block, you form a core that constructs the hole by flowing the material around it.

Making the Write Decision about Text

Putting text on a part usually isn't a casual decision. Text often provides directions, safety warnings, or some sort of logo or branding. All these things are important to get right. But was your company logo created to allow its inverse to be machined easily into a mould? I'm guessing not.

If you need to put text on a part, here are a few tips to keep in mind:

- ✓ **Text size:** You need to make sure that the text is large enough for the resin that you choose to fill that part of the mould. You also need to make sure that the text is large enough for the cutting tool used to make your mould to move within (see Figure 3-7). For raised text on your part, the smallest mill size used in the injection moulding process is 0.4mm.
- ✓ **Legibility:** To make text simple to machine and read, I suggest using a bold sans serif font, such as Arial or Century Gothic (see Figure 3-8). If your design system controls the size of fonts using points as a unit, use 20 points or more. This will make the text a little large for small parts, but it's necessary to keep mould-cutting times short. Otherwise, you should measure the smallest feature that needs to be cut for your text and make sure

it is wider than 0.4mm for the cutting tool — and be sure to leave room for the tool to move inside the cut. Simply having the groove the same width of the tool doesn't allow the tool to work properly.

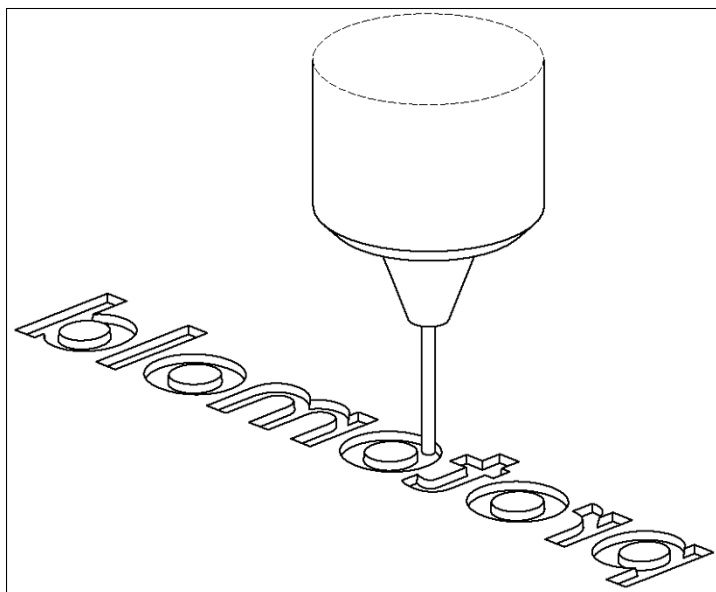


Figure 3-7: Machining text into a mould.

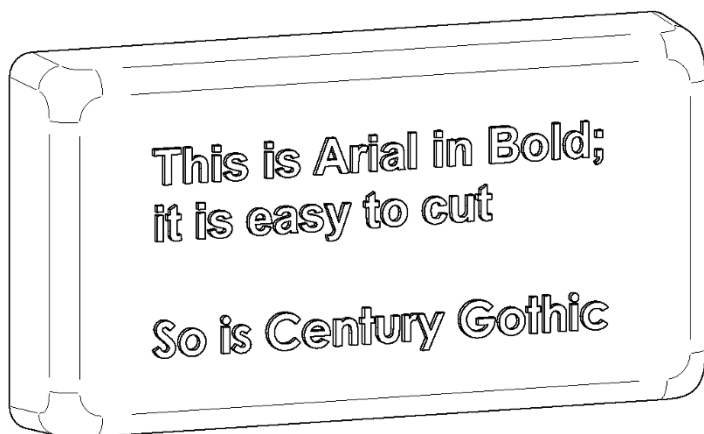


Figure 3-8: Examples of good fonts to use for text.



Using capital letters can help, too. Dotting a small *i* is tricky.

- ✓ **Draft:** Make sure to include draft on the side of the text. A draft angle of 2–5 degrees is recommended.
- ✓ **Height:** The text shouldn't rise too far above the surrounding surface. Try to keep text height to 0.4mm or less. Otherwise, you start to create cuts that are quite deep for a very small mill.



Declining an engraving invitation

Proto Labs recommends that you don't use engraved text on a part. To add engraving, machinists have to cut the material of the part down to create the raised letters on the mould. This process is very time consuming (and, therefore, expensive).

Also, it's extremely difficult to polish the mould near and between the letters, so the surrounding surface of the part may be inconsistent. Even a textured surface, which can hide some imperfections, may not look as good as you want it to.

Chapter 4

Going Beyond a Straight-Pull Mould

In This Chapter

- ▶ Making your parts move with side-action cams
- ▶ Working with bumpoffs
- ▶ Simplifying mouldmaking with inserts

The simple action of the straight-pull mould works well for many parts, but if you need something a bit more complex, there are ways to accomplish your goal. A couple of techniques make it possible to create more-complex parts and still make them quickly.

Proto Labs has also figured out ways to make specialised features in the mould that are too difficult to create in a straight-pull mould. Various types of insert add flexibility to moulds and keep costs down at the same time.

This chapter covers a few ways to get more complexity than you can with a simple straight-pull mould.

Solving Problems from the Side

To create these tricky features, you can build moving elements called *side-actions* into the mould (see Figure 4-1). In the mould, a cam mechanism moves a slide that moves part of the mould into position as the mould closes and then moves the slide back away from the part as the mould opens again. Because of the mechanism that moves it, you will sometimes hear the hardware of a side-action referred to as

a *cam* or *side-action cam*. Other options, such as shutoffs (see the upcoming section “Shutting off the action”), close off the cavity on the sides of a part but are open down to the parting line. This makes it possible for a feature on the part to overhang and be built using the standard straight-pull mould.

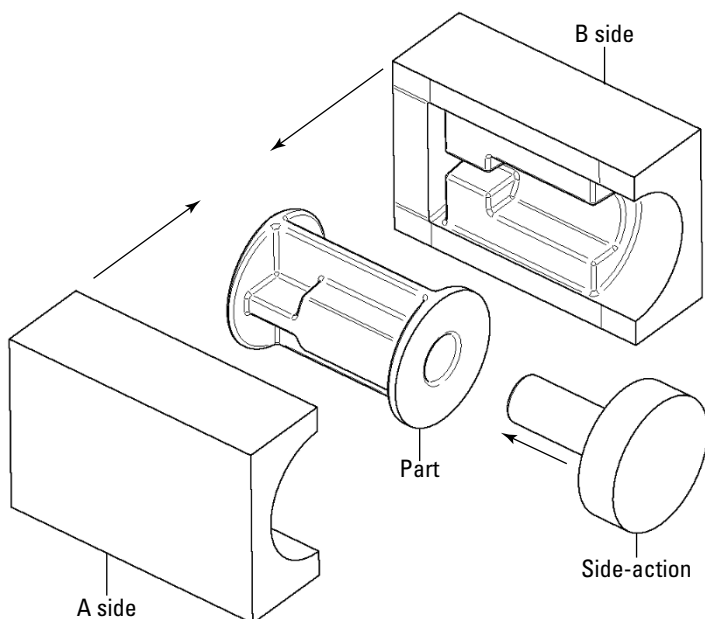


Figure 4-1: A side-action allows you to make features that can't be made otherwise.

Knowing when to use side-action cams

How do you know whether side-action cams are a good option for you? Here are a few guidelines:

- ✓ There needs to be room to pull the mould section clear of the part, including space for the cam mechanism. This usually means the undercut feature must be on the outside of the part.

- ✓ The cam's movement has to be perpendicular to the opening direction of the mould. Creating a side-action that would work at any angle to the part is too complex for a quick-turnaround part.

The feature that the side-action is making does not have to sit on the parting line, but the cam needs to come from the parting line up to the feature. This will create a witness line on your part.

- ✓ The faces that make up the interior of your part's feature need to be drafted in the direction that the side-action moves so the side-action can withdraw without distorting the part.

Seeing what sizes the cams can be

You can use multiple side-actions in a mould, but the slide and the clearance for it take up volume, which leaves less room for your part. This limitation makes sense because you've got limited space to begin with.

Shutting off the action

A *shutoff* uses the normal open and close action of the mould and a portion of the A side to make contact with the B side of the part to form a contact area. This can create an open area in the wall of the part rather than using a side-action to block part of the cavity. The wall that contains the opening can still have a draft applied to it.

A shutoff is also different from other mould geometry like ribs and bosses because it allows the designer to take a different approach to designing the part for moulding.

One of my favourite examples is a *clip*. Normally, you think of a clip as a post sitting on a nice solid face, as shown in Figure 4-2. Creating the locking portion of the clip, however, may require an elaborate side-action, which may be impossible to make because it is unreachable or obstructed by other features.

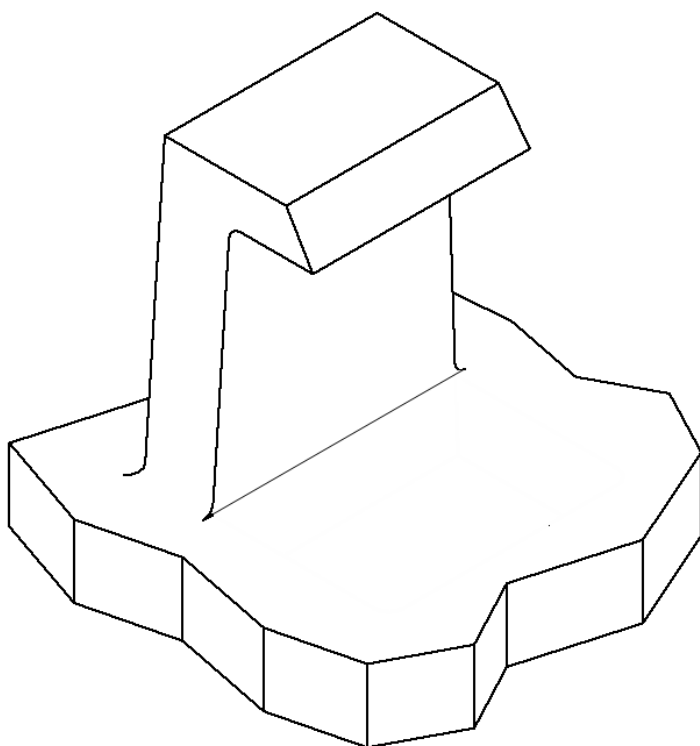


Figure 4-2: As designed, this clip looks really nice, but it's hard to make.

More ways to use side-actions

You can also use side-actions to create a face on a part that has no draft. Because you can include more than one side-action cam, you can have more than one face that's perpendicular to the mould opening. You can even have a texture or text on that face, and because that part of the mould moves out of the way

before the part is ejected, you don't have to worry about drag marks.

I like one other novel use for a side-action: using it to create a hollow interior for a long, narrow part rather than making a deep cut into the mould face. See Figure 4-1 for an example.

Instead, you can use a shutoff to create a hole in the finished part by filling the space below the edge of the clip, as shown in Figure 4-3. This clip will be easier to make.

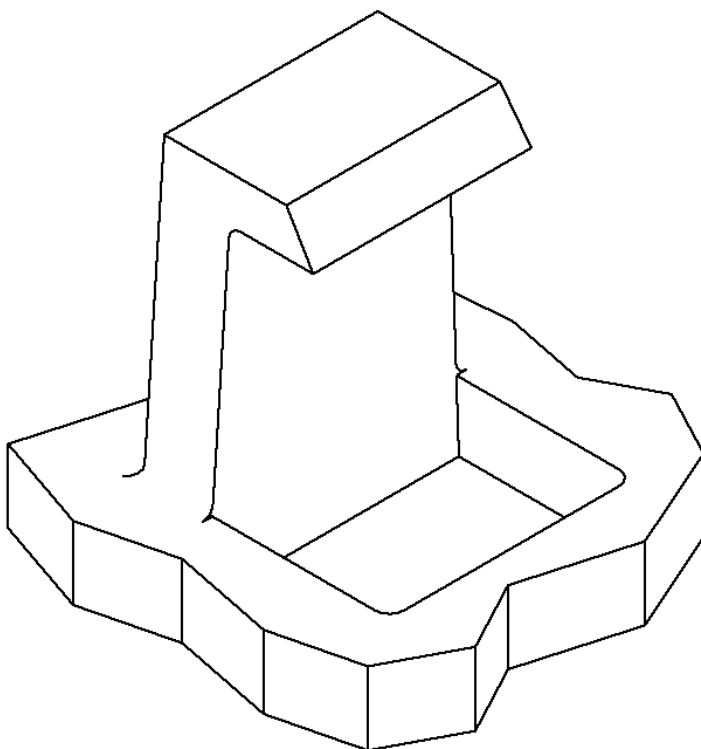


Figure 4-3: Creating a hole at the base of the clip makes it possible to use a sliding shutoff.



The portion of the mould that makes the underside of the clip is on the B side of the mould and must fit tightly against the A side of the mould to create the miniature cavity creating the clip. You must have a minimum 3-degree draft on the faces that meet when the mould is closed.



If you want to know for certain whether the geometry on the side of your part can be made with a side-action or a shutoff, when you upload your part into the Proto Labs system, you will be advised of this as part of your auto quote.

Adding Resistance with Bumpoffs

Side-action cams and shutoffs are primarily for making external side geometry. Sometimes neither will do the job. There are other tricks to use when you need to solve more difficult design challenges. There are simply cases where keeping a clearance for the mould to open smoothly will not allow the part to be functional. To create a lip on the inside of a part, for example, you must put a recess in the mould to create a feature that the adjacent piece of the part won't clear without resistance. This type of feature can sometimes be made in a rapid injection moulding tool with something called a *bumpoff*. No, this isn't a reference to organised crime. Just think of a bumpoff as being a feature that makes the part more difficult to eject.

As you see in Figure 4-4, the recess that makes a bumpoff is just machined in because a side-action can't be added to the inside.

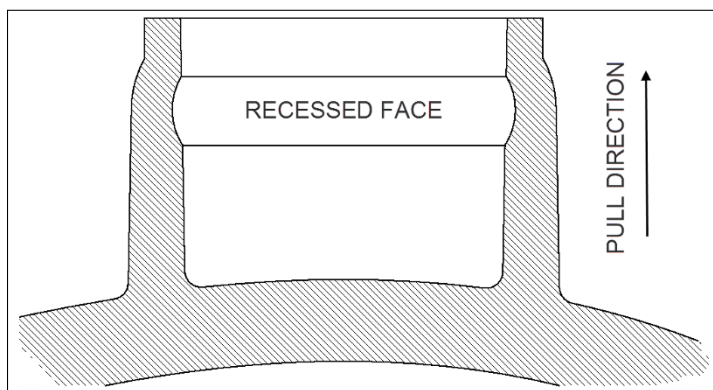


Figure 4-4: Bumpoffs are built from recesses in a mould.

As the part cools in the mould, the material shrinks onto the recessed face, resulting in resistance to ejection. For that reason, you can make the recess only so deep before damaging the surface of the part during ejection. It's best to work with the experts at Proto Labs when you design bumpoffs into your part, to gauge what sizes will work with the material you are going to use.

Making Mouldmaking Simpler with Inserts and Pins

That's enough talk about the fancy stuff for a while. The bottom line is this: The easier it is to make the mould for your part, the faster and less expensive it will be to make that part. Proto Labs has developed some techniques that save machining time and improve quality. These techniques involve adding temporary parts to the mould, as you'll see in this section.

Picking out inserts

Another way to build part features that can't be made in the motion of the mould is to use special mould components called *inserts* or *pickouts*. These pickouts are placed in the mould before injection and come out of the mould with the part; then the pickouts are removed from the plastic part manually and put back in the mould. Will the creativity in naming mould components never cease?

Pickouts don't have to align to the mould in any particular fashion, so just imagine the possibilities. Features that are normal to a drafted face or placed on curved faces at an angle to the parting line can't be made with side-actions or shutoffs. Figure 4-5 shows a great example of a boss that really needs a pickout to work properly. The only real limitation that pickouts have is that they need to be able to remain in place while the mould fills, so they have to be mounted securely, and they have to be able to be manually removed from each part.

Pinning down the core

Tall, thin features can be risky things to machine into a mould. Small holes going through or into a part require this type of geometry. These types of feature are costly to machine, and the risk of bending or breaking them off is high. For that reason, the machinist has to slow his or her work dramatically to make them.

That's where core pins can come in. Core pins can be machined outside the mould (which makes it easier to maintain quality) and then inserted into the mould (see Figure 4-6).

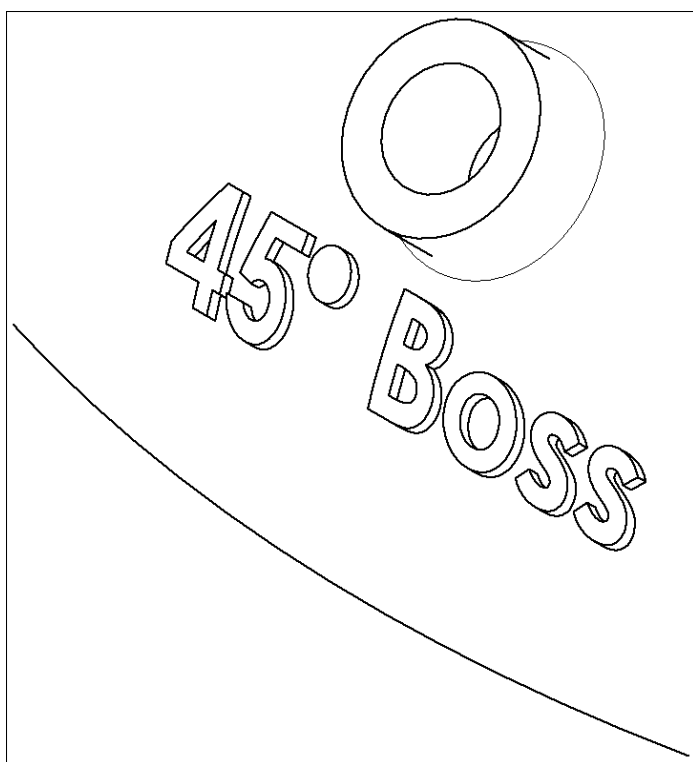


Figure 4-5: A pickout works perfectly to create this angled boss on a curved face.

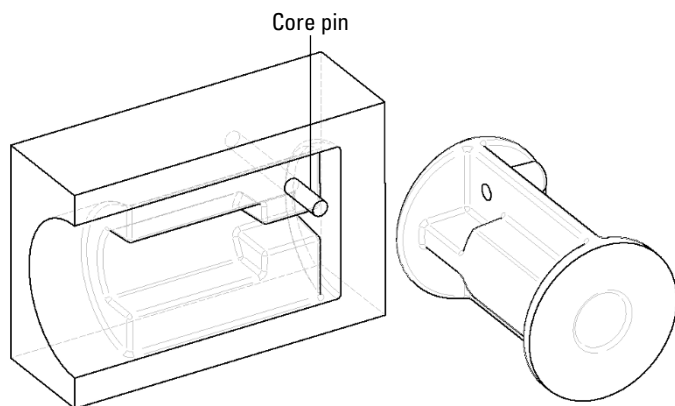


Figure 4-6: Long, thin cores can be added to the mould after the cavity sides are machined.



If you need to have tight control of a feature that's set deeply into the mould, core pins can be a great way to maintain a steeper draft angle or other type of control.

Designing for High Productivity

Just because a mould can accommodate a limited volume of material, it doesn't mean that you can get only one part per production cycle.

If you need to make a large number of small parts, for example, consider making your mould to accommodate several parts per cycle. The cost of machining this type of mould may be higher than the norm, but getting more parts faster may offset the price difference.

Designing a mould for multiple parts

Designing the mould's core and cavity for multiple parts, as shown in Figure 4-7, is a relatively simple process. You have to make some additional calculations to ensure consistent quality from cavity to cavity, but all in all, this technique is a great way to get more for your money.

Accommodating side-actions in multi-cavity moulds

Adding extra cuts in the mould to make multiple parts can make side-actions difficult to place. Depending on the size and range of motion of the side-action, it's still possible, but sometimes it just cannot work. In such a case, you can still create special features in the mould by using pickouts and core pins in the same fashion for several cavities as you can for one.

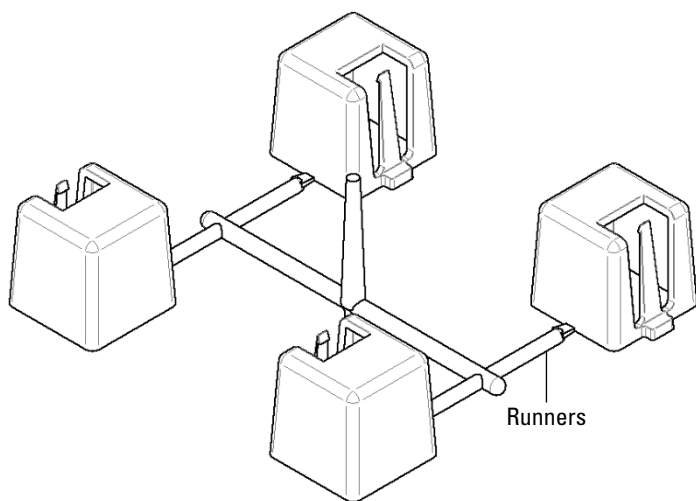


Figure 4-7: Creating multiple parts at the same time requires more runners and ejectors.

Mirror, Mirror: Making Mirrored Parts

In cases where the main body of a couple of parts is the same, you should explore ways of using one part to make two mating parts. If designed properly, the parts can be connected using features that will allow you to hinge or interlock them together.

By using a little creativity, you may be able to design the halves of a hinged container as two iterations of the same part (see Figure 4-8). Creating this one type of part will save you from having to create a second mould.

For example, you might create a hinge with a pin toward one end of the part and a hinge with a matching hole near the other. When two copies of this part are positioned properly, the hook on one hinge mates with the shaft on the corresponding hinge.

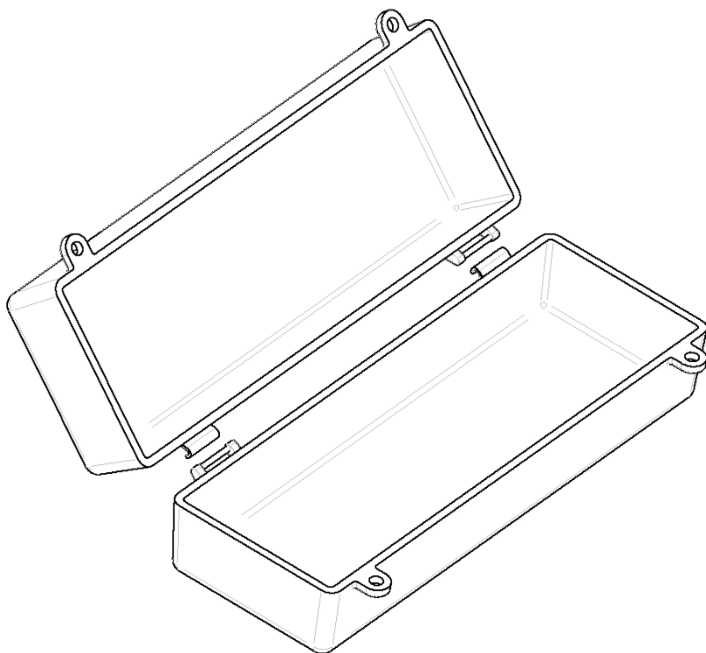


Figure 4-8: Two identical parts can replace two separate parts.

Chapter 5

Adding the Finishing Touches

In This Chapter

- ▶ Adding colour
 - ▶ Finishing the job
-

With your design nearly complete, you're probably really excited about getting the manufactured part in your hands (and quickly, too). But because how a part looks can be as important as how well it's built, you may be wondering what colour your part will be or what options you have for colouring it.

You've come to the right chapter. Here, I discuss adding colour and texture to give your parts the best possible finish.

Colouring Your Parts

You're probably so accustomed to seeing brightly coloured plastic parts that you may be surprised to find out that most plastics don't have natural colour. With all the options you have for manufacturing your part, I don't think you'll be surprised to find that you also have several options for getting the colour you want.

There are two basic ways to colour a part:

- ✓ Paint the part after it's been removed from the mould.
- ✓ Add colourant to the resin before it's injected into the mould.

Which method is better? The answer depends on what you want your part to do, how quickly you need it, and what you plan to do with it. The following sections discuss both methods.

Colouring from the outside: Painting

If you need to match other components' colours precisely, or if you want to use a colour that's specific to your company or customer, painting may be the best option.

Painting is fairly straightforward, allowing you to match any existing painted parts. The only extra step involves preparing the surface for painting.

If some surfaces need to mate very closely with other surfaces, they can be masked before painting. Masking may be necessary because paint adds a small amount of thickness to the part, which could change the join. This adds a step to the production of the part, but the overall process is still likely to be faster than trying to develop the correct colour in the resin.

Painting also allows you to add a finer finish to the part without having to create a highly polished finish in the mould cavity.

Colouring from the inside: Adding colourant

The other way to add colour to plastic parts is to add colourant to the resin. To make the process quick and efficient, Proto Labs offers a selection of standard colourants. Some of my favourite colourants are Banana, Cherry Red, Coconut, Fresh Green, Honey Beige, Mushroom, and Nectarine. (Do those names make you hungry too?) Proto Labs also offers a wide selection of reds, greens, and blues, as well as several transparent colours for use with clear resins. When colouring resins, it works best to start with a resin available in a natural or clear colour.



The colours stocked by Proto Labs may not provide an absolute match for the colour you want, but they can give a very close approximation of what your part will look like. Using the stocked colour allows you to save money and still get your part quickly.

To make plastics in one of these colours, Proto Labs technicians mix a small amount of dye pellets with the resin. This technique is referred to as a *salt-and-pepper mix* because of the way the raw resin and colourants look in the hopper (see Figure 5-1 for a black-and-white depiction). It doesn't take a lot of colourant to do the job; typically, about 3 percent of the total volume of resin will work. This mixture blends while the plastic melts in the barrel of the extruder before injection.



Figure 5-1: A little colourant goes a long way.



Resins that have special properties (such as flame resistance) or that are used for special applications (such as medical or food-storage use) may lose some of those features when colourants are added, and Proto Labs doesn't stock specialized colourants that don't interfere with them. Make sure that colourants won't affect the function of your finished part. See the nearby sidebar "Adding colour with precompounded resins."



If you need to match a specific colour precisely, it's best to go to a custom colour compounder and get precoloured resin. Be sure to factor in enough time for this process!

Adding colour with precompounded resins

Different types of resin respond differently to being coloured. Resins that take a high temperature to melt, for example, are harder to mix with colourant consistently. Also, depending on the colour you select, there may be a risk of swirling or inconsistency in the colour of the finished part. To solve these potential problems, you can choose a precompounded resin.

Precompounded resins are made by mixing colourant with resin, melting and extruding the mixture, grinding up the resulting plastic, and repeating the process until the colour is consistent and exactly what you want.

Precompounded resins can also maintain the special properties that Proto Labs can't guarantee when it mixes resin with stock colourants.

Designing a Big Finish

Plastic parts can come in a variety of finishes. Whether you need a part to look polished or have a softer look, you can define the specific finish — for the entire part or for portions of it. You can even mix finishes in different areas.

Proto Labs offers various finishes, as shown in Table 5-1.

Table 5-1 Proto Labs Surface Finishes and Textures

<i>Finish Callout</i>	<i>Description</i>
PM-F0	Noncosmetic; finish to Proto Labs discretion
PM-F1	Low cosmetic; most tool marks are removed
SPI-C1	600 grit stone, 10-12 Ra
SPI-B1	600 grit paper, 2-3 Ra
SPI-A2	Grade #2 Diamond Buff, 1-2 Ra
PM-T1	Proto Labs texture, SPI-C1 followed by light bead blast
PM-T2	Proto Labs texture, SPI-C1 followed by medium bead blast

Chapter 6

Ten (Or So) Ways to Give Your Parts the Proto Labs Edge

.....

In This Chapter

- ▶ Drawing on professional expertise
 - ▶ Getting real parts, not prototypes
 - ▶ Adapting for different-length runs
-

This chapter delves into the advantages of working with professionals when you need to have prototype parts created and the advantages of using the Proto Labs solution.

Proto Labs Know-How

This book isn't about how to get a part made quickly or even how to get a good part made quickly; it's about getting a high-quality plastic part made, period.

The suggestions and options presented here are best practices for rapid injection moulding. They're based on Proto Labs' years of experience in helping people get their finished ideas in their hands faster than they believed possible.

These methods can help anyone who wants to know how to make injection-moulded plastic parts. But best practices are just part of the equation. To get parts made, you also need people.

Access to Experts

No book or “how to” document can compare with the experienced eye of an expert or craftsman. It’s all well and good to follow guidelines, but if you’re not already experienced in injection moulding, you could be applying the wrong guidelines to your design.



You need to work with people who can help you review your design and your needs. Experts can share information that may give you a different perspective on what rules apply to your specific situation. If you’re even thinking about having a plastic part made, it’s important to touch base with these experts to get their input.

Even if you have experience in injection moulding, Proto Labs’ professionals can bring a fresh viewpoint to your situation and help you to see things in a new way. Injection moulding design processes and availability of materials are changing all the time, and people who think plastics day and night are great people to have on your side.

Proprietary Technologies and Tools

Having best practices to follow and experienced people to assist you isn’t enough. What’s missing? Ah, yes: technology.

Manufacturing technologies are improving by the minute, and when they’re used to improve the performance of already knowledgeable professionals in the field, it’s a powerful result. Today’s specialised software can analyse your part to see whether it can be manufactured as is. If it can be, the next step is to let you know, quickly, what it will cost.

The analysis is even more effective when it includes engineering data on a select set of materials that will be processed in a known, consistent way. Having the more specific engineering data provided by Proto Labs allows better quality prediction and a better understanding of cost.

Real Parts Made Rapidly

Rapid prototyping is typically done by building facsimiles of parts using liquids solidified by laser — powders and plastics that are melted into a stacked string are both common. These simulated parts have their place and are good at allowing someone who isn't involved with the part design to have an idea of the shape and size that the part will be. However, if you need a prototype that can be tested for form, fit, and function, these rapid prototypes (RPs) cannot convey the way the actual material feels, represent correct colour, or create a consistent finish. RPs are also unable to show more than one finish on the same part.

Some RP systems can use the same material that you'd choose for injection moulding, but the material doesn't give the parts the same physical characteristics as an injection moulded part. If you need to evaluate the mechanical performance of a part, be sure to keep this in mind.

The process of the plastic flowing into the mould has an effect on its strength and performance. The material used in a part that's made outside of a mould doesn't experience the same physical transformation as injection-moulded plastic. It, too, may be affected by cooling but in different ways. You can't get the same performance from a rapid prototype part that you get from a rapidly injection-moulded part made by Proto Labs.

Mass-Production Readiness

This point has been made before but it's worth repeating: A part that can be injection moulded by using rapid turnaround production methods will be a high-quality part that can be produced by using mass production tooling as well.



Injection moulding is injection moulding. The only real differences are the cost of the mould, more flexibility with mould size, and the ability to use more advanced tooling like side-actions in a mould that takes weeks or months to build. Historically, people have waited weeks or even months to find out that their part needed substantial changes and the mould

had to be scrapped. With Proto Labs, you can quickly and precisely prove out your design and, if needed, get low to medium on demand parts, which could satisfy your production quantities or enable you to grow towards serial production.

Multiple Finishes

Proto Labs can make a quick turnaround injection-moulded part with multiple finishes. It can help you test side-actions or other tooling options and see what will work for the long haul. The amount the plastic will shrink and how the final size of the part changes can be measured, and any sink or imperfections that appear, can be corrected before the mould is cut. In other words, you'll know for certain that your design looks as good as you expect it to.

Depending on the role your design plays, the fit and finish of the part is an increasingly important consideration as more and more people are accustomed to the “feel” of high-quality parts and are developing a more discriminating eye for fit and finish.

On Demand Manufacturing

If a company can potentially create your part in a day, it stands to reason that the same company could make a lot of parts over an extended period. All it takes is willingness and the equipment to do so.

If you don't need a huge number of parts made, ask the people at Proto Labs, about on demand manufacturing and how many parts they could make over a longer period of time. They already know that you know your design, and they've become familiar with what you want your part to do.

Glossary



A side: Sometimes called the “cavity”, this is the half of the mould that usually creates the exterior of a cosmetic part. The A side usually does not have moving parts built into it.

B side: Sometimes called the “core”, this is the half of the mould where ejectors, side-action cams, and other complex components are located. On a cosmetic part, the B side usually creates the inside of the part.

bloom: A cosmetic imperfection that is created where the resin is injected into the part, usually visible as a blotchy discoloration on the finished part at the site of the gate.

cam: See *side-action*.

cavity: The void between the A side and the B side that is filled to create the injection-moulded part. The A side of the mould is also sometimes called the cavity.

core: A portion of the mould that goes inside a cavity to form the interior of a hollow part. Cores are normally found on the B side of a mould, and so the B side is sometimes called the core.

core-cavity: A term used to describe a mould created by mating A side and B side mould halves.

core pin: A fixed element in the mould that creates a void in the part. It is often easier to machine a core pin as a separate element and add it to the A side or B side as needed. Steel core pins are sometimes used in aluminium moulds to create tall, thin cores that might be too fragile if machined out of the bulk aluminium of the mould.

draft: A taper applied to the faces of the part that prevent them from being parallel to the motion of the mould opening. This keeps the part from being damaged due to the scraping as the part is ejected out of the mould.

edge gate: An opening aligned with the parting line of the mould where resin flows into the cavity. Edge gates are typically placed on an outside edge of the part.

ejection: The final stage of the injection moulding process where the completed part is pushed from the mould using pins or other mechanisms.

ejector pins: Pins installed in the B side of the mould that push the part out of the mould when the part has cooled sufficiently.

family mould: A mould where more than one cavity is cut into the mould to allow for multiple parts made of the same material to be formed in one cycle. Typically, each cavity forms a different part number. See also *multi-cavity mould*.

finish: A specific type of surface treatment applied to some or all faces of the part. This treatment can range from a smooth, polished finish to a highly contoured pattern that can obscure surface imperfections or create a better-feeling part.

flash: Resin that leaks into a fine gap in the parting lines of the mould to create an undesired thin layer of plastic.

gate: The generic term for the portion of the mould where resin enters the mould cavity.

hot tip gate: A specialised gate that injects the resin into a face on the A side of the mould. This type of gate doesn't require a runner or sprue.

insert: A portion of the mould that is installed permanently after machining the mould base, or temporarily between mould cycles.

knit lines: See *weld lines*.

meld lines: See *weld lines*.

multi-cavity mould: A mould where more than one cavity is cut into the mould to allow for multiple parts to be formed in one cycle. Typically, if a mould is called "multi-cavity," the cavities are all the same part number. See also *family mould*.

On-Demand Manufacturing: Low volume production of end use parts, where production may continue for several years.

parting line: The edge of a part where the mould separates.

pickouts: A mould insert that remains stuck to the ejected part and has to be pulled out of the part and placed back into the mould before the next cycle.

post gate: A specialised gate that uses a hole that an ejector pin passes through to inject resin into the mould cavity. This leaves a post vestige that usually needs to be trimmed.

press: An injection moulding machine.

ram: A hydraulic mechanism that pushes the screw forward in the barrel and forces resin into the mould.

resin: A generic name for chemical compounds that, when injected, form a plastic part. Sometimes just called “plastic.”

rib: A thin, wall-like feature parallel to the mould opening direction, common on plastic parts and used to add support to walls or bosses.

runner: A channel that resin passes through from the sprue to the gate(s). Typically, runners are parallel to, and contained within, the parting surfaces of the mould.

shrink: As it cools, a plastic part will change size. Knowing the amount a resin will shrink allows you to build a cavity larger than the final part will be, so that the part will shrink to the final desired dimensions.

shutoff: A feature that forms an internal through-hole in a part by bringing the A side and B side in contact, preventing the flow of resin into the through-hole.

side-action: A portion of the mould that is pushed into place as the mould closes, using a cam-actuated slide. Typically, side-actions are used to resolve an undercut, or sometimes to allow an undrafted outside wall. As the mould opens, the side-action pulls away from the part, allowing the part to be ejected.

sink: Dimples or other distortions in the surface of the part as different areas of the part cool at different rates. These are most commonly caused by excessive material thickness.

sprue: The first stage in the resin distribution system, where the resin enters the mould. The sprue is perpendicular to the parting faces of the mould and brings resin to the runners, which are typically in the parting surfaces of the mould.

stitch lines: See *weld lines*.

straight-pull mould: A mould that uses only two halves to form a cavity that resin is injected into. Generally, this term refers to moulds with no side actions or other special features used to resolve undercuts.

tab gate: See *edge gate*.

texture: See *finish*.

tunnel gate: A gate that is cut through the body of one side of the mould to create a gate that does not leave a mark on the exterior face of the part.

vestige: After moulding, the plastic runner system (or in the case of a hot tip gate, a small dimple of plastic) will remain connected to the part at the location of the gate(s). After the runner is trimmed off (or the hot tip dimple is trimmed), a small imperfection called a “vestige” remains on the part.

undercut: A portion of the part that shadows another portion of the part, creating an interlock between the part and one or both of the mould halves. An example is a hole perpendicular to the mould opening direction bored into the side of a part. An undercut prevents the part from being ejected, or the mould from opening, or both.

warp: The curving or bending of a part as it cools that results from stresses as different portions of the part cool and shrink at different rates. Parts made using filled resins may also warp due to the way the fillers align during resin flow. Fillers often shrink at different rates than the matrix resin, and aligned fibres can introduce anisotropic stresses.

weld lines: Also known as “stitch lines” or “knit lines” and when multiple gates are present, “meld lines”. These are imperfections in the part where separated flows of cooling material meet and rejoin, often resulting in incomplete bonds and/or a visible line.

You can get *real* parts, really fast!

Have you ever been asked to develop a new product or prototype and given a ridiculously tight timeline? Are you struggling with the long wait associated with injection moulding, when you really need to get to market first? Do you wish you had some guidance to help you better understand the process of making plastic parts so that you could save time, money, and frustration? This book gives you the tools you need to identify design considerations that can drastically improve the quality of your part and the speed at which it can be manufactured.

- *Getting better parts faster* — by understanding the basic principles of the injection moulding process
- *Designing for mouldability* — and knowing the easy-to-follow rules that can help you avoid costly mistakes
- *Addressing your options* — and getting the look and feel you want
- *Working with the experts* — and learning the language necessary to communicate effectively about your project

Thom Tremblay is a design software specialist with a leading software producer. He has authored several books on 3D CAD. He has also worked with hundreds of companies to help them understand how changes in design and engineering software affect their processes and bottom line. In his spare time, he rides motorcycles long distances and likes to pretend he's a rock star.



Open the book and find:

- How injection moulding works
- How to make a better prototype
- How to have your parts look the way you expect
- How to get it all in less time than you ever thought possible

Go to Dummies.com

for videos, step-by-step examples,
how-to articles, or to shop!

For Dummies®
A Branded Imprint of



ISBN: 978-1-118-20401-6
Book not for resale

WILEY END USER LICENSE AGREEMENT

Go to www.wiley.com/go/eula to access Wiley's ebook EULA.