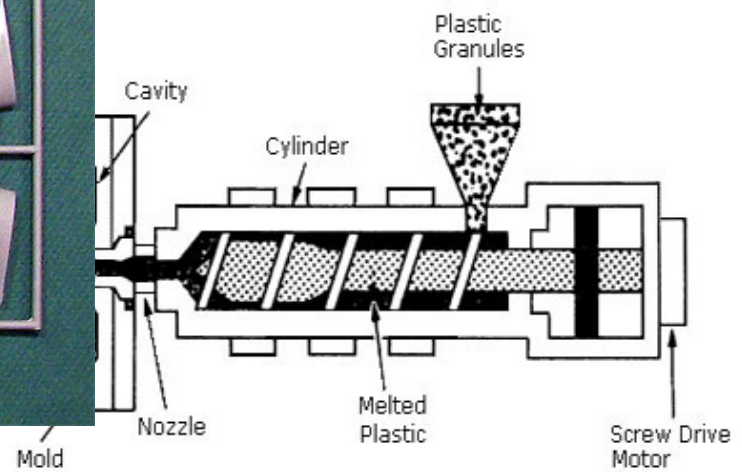
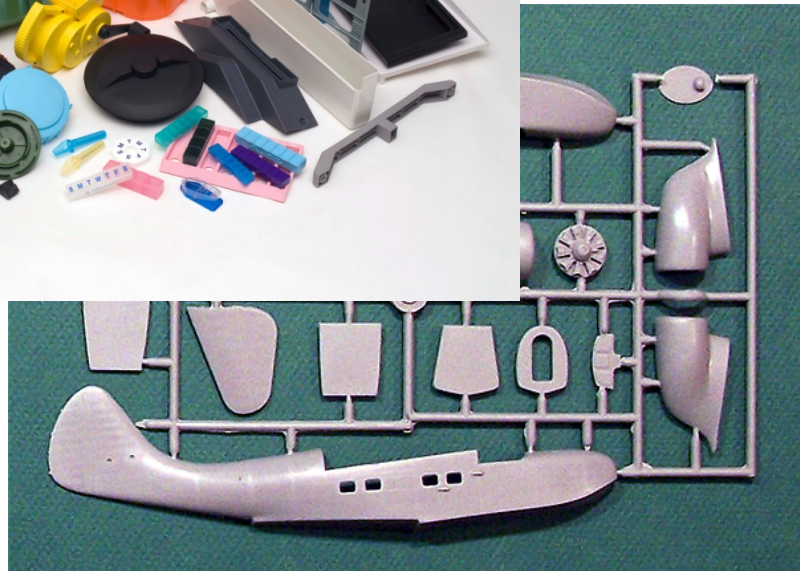


Fundamentals of Design for Plastic Injection Molding

Kelly Bramble



**Fundamentals
of
Design for Plastic Injection Molding**

Copyright, Engineers Edge, LLC
www.engineersedge.com

All rights reserved. No part of this training program may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, recording, or any information storage and retrieval system, without permission in writing from the publisher. While every precaution has been taken with respect to the accuracy of the technical subject matter, Engineers Edge and the author assume no liability with respect to the use and application of the technical information contained herein.

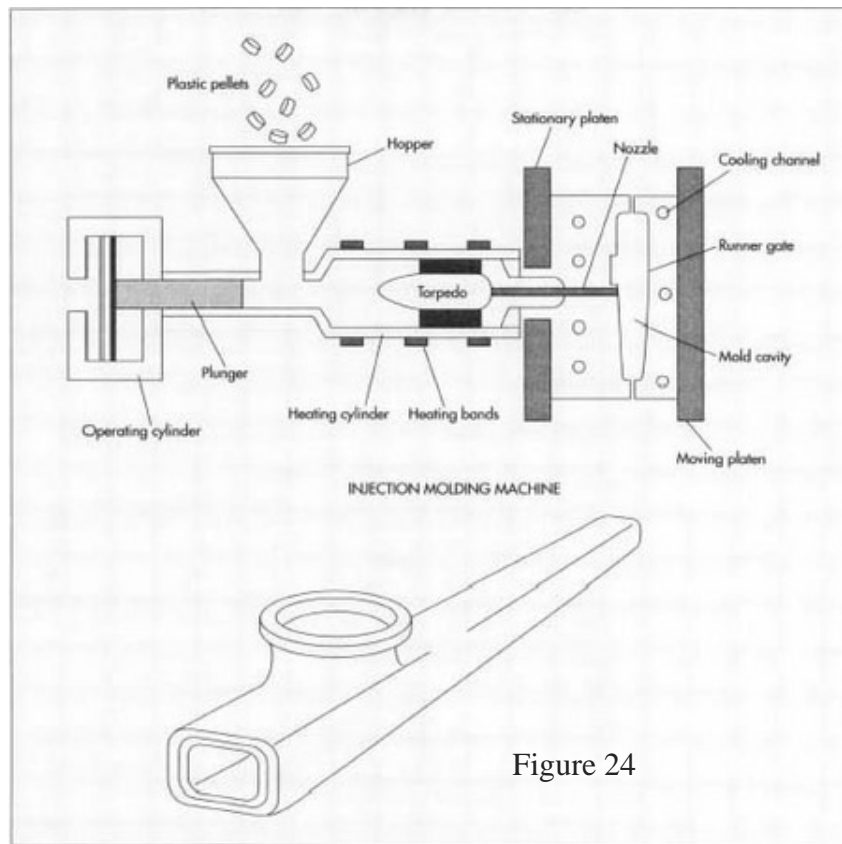
Sold and Distributed by:

Engineers Edge, www.engineersedge.com,
PDH Storm, www.pdhstorm.com

Revision A

Plastic Injection Molding Manufacturing Process

Injection molding involves taking plastic in the form of pellets or granules and heating this material until a liquid is obtained. Then the plastic liquid is forced into a split-die chamber/mold where it is allowed to "cool" into the desired shape. The mold is then opened and the part is ejected, at which time the cycle is repeated. Though temperature varies with different plastics and molding conditions, a typical melting temperature for the plastic is approximately 180°C (350°F). The mold tool is typically made from steel and often includes water cooling features. The injection molding process takes about 50 seconds per part, where most of this time is required for the part to cool sufficiently before being ejected from the molding tool. The illustration below depicts the injection molding process and tooling.



Definitions:

Parting Line: Location at where the perspective injection molding tool halves meet. Usually, a line or "Flash" is visible on the molded part at the parting line location.

Thermosets - are plastics that undergo chemical change during processing to become permanently insoluble and infusible. Thermoset material will undergo or has undergone a chemical reaction by the action of heat, catalysts, ultra-violet light, etc., leading to a relatively infusible state. Typical of the plastics in the thermosetting family are the amines (melamine and urea), most polyesters, alkyds, epoxies, and phenolics.

Thermoplastics - are resins that repeatedly soften when heated and harden when cooled. This characteristic facilitates the forming of the material into desired shape when the thermoplastic is placed within a die or fixture of the desired final geometry.

Plastic Injection Molding Design Guidelines

To insure the quality of the final product, it is necessary to start out with quality components. Injection molded parts can be molded to a high quality standard by focusing on these areas of plastic technology:

1) Correct Part Design, 2) Accurate Selection of Material, 3) Proper Plastic Processing.

Only by drawing on expertise from these three areas of plastic technology can a product designer create quality molded parts that maximize performance and are cost effective.

If there was only one rule for the injection molding process it would have to be "maintain uniform wall thickness". Here are some examples of problems associated with part designs that feature non-uniform wall thicknesses:

Sink marks due to uneven shrinkage

Sink marks result from a wall yielding to the still shrinking interior mass.



Figure 24.2

Stress due to uneven shrinkage

A part with non-uniform wall thickness will cool unevenly, resulting in high molded-in stress.



Figure 24.4

Voids due to uneven shrinkage

The already cooled section will not yield to the shrinking action of the cooling interior mass causing voids in the thick portion of the part.



Figure 24.6

Warpage due to uneven shrinkage

Concentrated stress at the junction of high & low shrinkage areas may cause a part to warp.



Figure 24.8

Plastic parts are all from the mold. Sir good definition for removed from the of 1/2 degrees is normal for plastic

To view the remainder of the course material and complete the quiz to get PDH credit and certificate, you must purchase the course.

*First, close this window and click **“Create an Account”** or **“Login”** located on the right side of the webpage then select the link at the bottom of the webpage:*

“Send payment via Paypal or Credit Card”

Many times the to cure this problem means of increasing sink marks or s

Rib thickness should be recommended.

allow part ejection or removal (slightly) to mold tooling cores. A the molded plastic part to be from the mold. A draft angle degrees per side are considered



design. One of the easiest ways to add ribs is a local way and economical way of adding ribs without causing

al wall thickness is

4.12

