

HANDBOOK OF THERMOPLASTICS INJECTION MOULD DESIGN

P.S. Cracknell and R.W. Dyson



BLACKIE ACADEMIC & PROFESSIONAL

An Imprint of Chapman & Hall

Handbook of Thermoplastics Injection Mould Design

by

P.S. CRACKNELL and R.W. DYSON
London School of Polymer Technology
University of North London



BLACKIE ACADEMIC & PROFESSIONAL

An Imprint of Chapman & Hall

London · Glasgow · New York · Tokyo · Melbourne · Madras

Published by
Blackie Academic & Professional, an imprint of Chapman & Hall,
Wester Cleddens Road, Bishopbriggs, Glasgow G64 2NZ

Chapman & Hall, 2-6 Boundary Row, London SE1 8HN, UK

Blackie Academic & Professional, Wester Cleddens Road, Bishopbriggs,
Glasgow G64 2NZ, UK

Chapman & Hall Inc., 29 West 35th Street, New York NY10001, USA

Chapman & Hall Japan, Thomson Publishing Japan, Hirakawacho Nemoto
Building, 6F, 1-7-11 Hirakawa-cho, Chiyoda-ku, Tokyo 102, Japan

DA Book (Aust.) Pty Ltd., 648 Whitehorse Road, Mitcham 3132,
Victoria, Australia

Chapman & Hall India, R. Seshadri, 32 Second Main Road, CIT East,
Madras 600 035, India

First edition 1993

© Chapman & Hall, 1993

Typeset in 10/12 pt Times New Roman by Best-set Typesetter Ltd.,
Hong Kong

Printed in Great Britain by

St Edmundsbury Press Limited, Bury St Edmunds, Suffolk

ISBN 0 7514 0074 2

Apart from any fair dealing for the purposes of research or private study, or criticism or review, as permitted under the UK Copyright Designs and Patents Act, 1988, this publication may not be reproduced, stored, or transmitted, in any form or by any means, without the prior permission in writing of the publishers, or in the case of reprographic reproduction only in accordance with the terms of the licences issued by the Copyright Licensing Agency in the UK, or in accordance with the terms of licences issued by the appropriate Reproduction Rights Organization outside the UK. Enquiries concerning reproduction outside the terms stated here should be sent to the publishers at the Glasgow address printed on this page.

The publisher makes no representation, express or implied, with regard to the accuracy of the information contained in this book and cannot accept any legal responsibility or liability for any errors or omissions that may be made.

A catalogue record for this book is available from the British Library

Library of Congress Cataloging-in-Publication data available

**Handbook of
Thermoplastics Injection
Mould Design**

For Chris and Cathy

Preface

Injection moulding is one of the most important methods of manufacturing plastics products. Through the development of sophisticated micro-processor control systems, the modern injection moulding machine is capable of producing precision mouldings with close tolerances in large numbers and with excellent reproducibility. This capability, however, is often limited by the lack of a proper appreciation of mould design.

The mould, or tool as it is often called, is at the heart of the injection moulding process. Its basic function is to accept the plastic melt from the injection unit and cool it to the desired shape prior to ejection. It is not, however, simply a matter of the mould having an impression of the shape to be moulded. Many other factors have to be taken into account – for example, the ability to fill the mould impression properly and efficiently without inducing weaknesses in the moulding and the efficient cooling of the moulding in order to maximise production rates without diminishing the quality of the moulding. In addition, the type of mould, gate and runner system, and ejection system which will best meet the needs of a particular job specification have to be determined. In our experience lack of attention to such factors leads to the mould limiting the ability of the injection moulding machine and preventing the process as a whole from achieving its true potential.

Injection moulds should not be designed in isolation but should be part of a team effort involving the product designer, the mould designer, the mould maker and the injection moulder. Each partner must be aware of the others' part in the process as a whole. This handbook will, it is hoped, be of some use to each of these partners.

P.S.C.
R.W.D.

Abbreviations

Elements

Al	aluminium
Be	beryllium
C	carbon
Cr	chromium
Co	cobalt
Cu	copper
Mn	manganese
Mo	molybdenum
Ni	nickel
Pb	lead
Si	silicon
S	sulphur
V	vanadium
W	tungsten

Polymers

ABS	acrylonitrile–butadiene–styrene
CA	cellulose acetate
FEP	fluorinated ethylene propylene
HIPS	rubber modified polystyrene (=TPS)
LCP	liquid crystal polymer
PA 6	nylon 6
PA 6,6	nylon 6,6
PA 11	nylon 11
PA 12	nylon 12
PAE	polyarylate
PBT	polybutylene terephthalate
PC	polycarbonate
PE-HD	high density polyethylene (HDPE)
PE-LD	low density polyethylene (LDPE)
PES	polyethersulphone
PMMA	polymethyl methacrylate
POM	polyacetal (acetal)

PP	polypropylene
PPO	polyphenylene oxide (modified)
PPS	polyphenylene sulphide
PS	polystyrene
PSU	polysulphone
PVC	polyvinylchloride
PVC-P	plasticised PVC
PVC-U	unplasticised PVC (=UPVC)
SAN	styreneacrylonitrile
TPS	rubber modified polystyrene (=HIPS)

Miscellaneous

AISI	American Iron and Steel Institute
BS	British Standard
DIA	diameter
DIN	German standards
DRG	drawing
ISI	Swedish Standardisering Kommission
QA	quality assurance

Contents

1	Product and mould	1
1.1	Introduction	1
1.2	Shrinkage	1
1.2.1	Mould shrinkage	1
1.2.2	Sinking	3
1.2.3	Internal stress and warping	4
1.3	Wall thickness	5
1.4	General features	7
1.4.1	Taper and draft angle	7
1.4.2	Corners	8
1.4.3	Ribs	8
1.4.4	Bosses	8
1.4.5	Weld lines	9
1.5	Undercuts	9
1.6	Surface finish	10
2	Thermal considerations	12
2.1	Introduction	12
2.2	Heat considerations	12
2.3	Specific enthalpy curves	14
2.4	Cooling	15
2.5	Cooling rates	16
2.6	Mould cooling	19
3	Fluid flow	21
3.1	Introduction	21
3.2	Cooling systems	21
3.3	Sample calculations	21
3.4	Viscosity	22
3.5	Limitations of Poiseuille's equation	24
3.5.1	Turbulent flow	24
3.5.2	Channel cross-section	25
3.6	Polymer melts	27
3.7	Examples of the use of flow equations and data	28
3.8	Channel cross-section	30
3.9	Filling the mould impression – other factors	31
4	The injection moulding machine and its influence on mould design	34
4.1	Introduction	34
4.2	Machine function	34
4.2.1	The machine base unit	34
4.2.2	The injection unit	35
4.2.3	The clamp unit	37
4.3	Machine types and configurations	39
4.4	Machine specification	41

5	Understanding moulds	42
5.1	Introduction	42
5.2	Tooling terminology	42
5.2.1	The mould impression	42
5.2.2	The mould core	42
5.2.3	The mould cavity	43
5.2.4	The split line	43
5.2.5	Venting	45
5.2.6	Ejection	46
5.2.7	Back plate	47
5.2.8	Sprue bush	47
5.2.9	Register ring	48
5.2.10	Tool location	48
5.3	Mould types	48
5.4	Choosing the correct mould	49
5.5	Mould specification	51
6	The two-plate mould	52
6.1	Introduction	52
6.2	Tool construction	52
7	Runner and gate design	56
7.1	Introduction	56
7.2	Freeze flow characteristics	56
7.3	Runner configurations	57
7.4	Gate positioning and design	58
7.5	Gate types	59
7.6	Edge gate	59
7.7	Fan gate	61
7.8	Diaphragm gate	61
7.9	Ring gate	62
7.10	Spoke gate	63
7.11	Tunnel or submarine gate	63
7.12	Pin point gate	65
7.13	Tab gate	65
7.14	Flash or film gate	66
8	Mould cooling	67
8.1	Introduction	67
8.2	Cooling requirement	67
8.3	Mould cooling time	69
8.3.1	Estimation of cooling time	70
8.4	Cooling media (or coolants)	71
8.5	Conductive thermal properties of mould construction materials	71
8.6	Cooling – design options	72
8.6.1	Location of cooling channels	72
8.6.2	Flat plane cooling	73
8.6.3	Spiral cooling	73
8.6.4	Finger or bubbler cooling	75
8.6.5	Baffle cooling	75
9	Ejection	77
9.1	Introduction	77
9.2	Choice of ejection method	77

9.3	Ejection methods	79
9.3.1	Pins and blades	79
9.3.2	Ejector sleeves	80
9.3.3	Valve headed ejectors	80
9.3.4	Stripper ring and plate ejection	83
9.3.5	Air ejection	83
9.4	Estimation of ejection force	84
10	The three-plate mould	86
10.1	Introduction	86
10.2	Why choose a three-plate mould?	86
10.3	Tool construction	87
11	Runnerless moulds	90
11.1	Introduction	90
11.2	Advantages of the hot-runner mould	90
11.3	Hot-runner systems	91
11.4	The externally heated hot manifold mould	91
11.5	The internally heated manifold mould	93
11.6	The insulated hot-runner mould	95
12	Undercut moulds	97
12.1	Introduction	97
12.2	Core pulling	97
12.3	Core pulling actuation methods	97
12.3.1	Cam pin actuation	97
12.3.2	Lost action cam pins	100
12.3.3	Action wedges	101
12.3.4	Hydraulic core pulling	102
12.3.5	Pneumatic core pulling	104
12.3.6	Electro-mechanical core pulling	105
13	Standard mould parts	107
13.1	Introduction	107
13.2	Why use standard mould parts?	107
13.3	Standard parts and assemblies	108
13.4	Standard elements	109
13.5	Accessory components	110
13.5.1	Accessory cooling components	110
13.5.2	Accessory ejection components	112
13.5.3	Mould feeding systems	112
13.5.4	Core pulling parts and accessories	113
14	Prototype moulds	114
14.1	Introduction	114
14.2	The case for a prototype mould	114
14.2.1	Moulding aspects of the prototype component	114
14.2.2	Mould design and constructional aspects highlighted by prototype moulding	115
14.3	Prototype mould tool materials and construction	117
14.4	Prototype tool construction	118
14.4.1	Inserted bolster prototyping	118
14.4.2	Modular prototype mould tool construction	120

15	Mould tool materials	121
15.1	Introduction	121
15.2	Mould construction material requirements	121
15.3	Mould construction materials	122
15.3.1	Ferrous mould construction materials	122
15.3.2	Alloy steels	122
15.3.3	General purpose mould steels	123
15.4	Non-ferrous mould construction materials	125
	Index	129

1 Product and mould

1.1 Introduction

The mould impression is the part of the mould that accepts molten plastic from the injection unit either directly via a sprue or via a sprue and runner. It is in the impression that the molten plastic cools and assumes the desired shape. In designing an impression to make a particular product, a number of factors need to be considered. It is not simply a matter of making the impression conform precisely to the product design. Some factors, for example, are: the impression must be capable of being filled as easily as possible; the moulding must be as free of stress as possible; the plastic shrinks on cooling and the moulding must be removable from the open mould. While the mould designer may be able to allow for these factors in designing a mould impression to make a product, the designer must be capable of appreciating the limiting factors and, where necessary, must be able to advise the product designer of difficulties and suggest remedies. The following sections outline the salient factors which need to be considered.

1.2 Shrinkage

1.2.1 *Mould shrinkage*

All materials shrink on cooling due to thermal contraction. The shrinkage of the plastic on cooling from the melt temperature to the mould temperature is known, somewhat perversely, as the mould shrinkage. The principal reason for mould shrinkage is thermal contraction which is measured by the thermal expansion coefficient of the plastic. The expansion coefficients of plastics materials are high compared with metals (Table 1.1). Typically, a 100°C rise/drop in temperature will produce an increase/decrease of between 0.001 and 0.02 mm/mm depending on the material. Although this is small, it should not be ignored. Additionally, crystallisable thermoplastics shrink on crystallising, the amount of additional shrinkage depending upon the amount of crystallinity developed which in some polymers is very much dependent on the rate of cooling. For example, PET hardly crystallises at all when cooled rapidly unless it is seeded but slow cooling can produce up to about 50%

Table 1.1 Dimensional stability data

Material	Thermal expansion (mm/mm K ⁻¹ × 10 ⁶)	Mould shrinkage (%)	Water absorp. (%)
ABS (rigid)	80	0.3–0.8	0.3
Acetal	80	2.0–3.5	0.2
Cellulose acetate	100	0.3–0.7	2–6
Fluorinated ethylene propylene	90	3.0–6.0	0
Nylon 6,6	120	1.5–2.0	1.5
Nylon 6	100	1.0–1.5	1.6
Nylon 11	150	1.2	0.4
Nylon 12	104	1.0	0.3
Polybutyleneterephthalate	90	1.5–2.0	0.2
Polycarbonate	70	0.6–0.8	0.16
Polyethylene (LD)	170	2.0–3.5	0.02
Polyethylene (HD)	120	2.0–3.5	0.01
Polymethylmethacrylate	85	0.1–0.8	0.35
Polypropylene	110	1.5–2.5	0.01
Polyphenylene oxide (modified)	55	0.5–0.7	0.1
Polystyrene (GP)	70	0.2–0.6	0.2
Polystyrene (rubber modified)	120	0.2–0.8	0.2
Polyethersulphone	55	0.6–0.8	0.15
Polyvinyl chloride (rigid)	55	0.1–0.5	0.05
Styrene acrylonitrile	70	0.2–0.5	0.3
Steel	11–13		

The above are typical values for unfilled grades

crystallinity. The rate of cooling therefore determines the total amount of shrinkage as well as the properties of the product.

The use of fillers (mineral powders, glass fibres, etc.) can reduce the amount of shrinkage on moulding because they have much lower thermal expansion coefficients. However, processability may be adversely affected as well as dimensional stability.

It is common practice to quote a figure for *mould shrinkage* either in mm/mm or as a percentage for plastics materials (Table 1.1). Such figures should be regarded as indicative. The precise shrinkage observed will depend on *temperature drop, rate of cooling, shaping pressures* and *anisotropy due to orientation*.

Anisotropy arises primarily from molecular orientation produced during flow (chapter 3). The consequence is that shrinkage is greater in the flow (orientation) direction than in the cross-flow (transverse) direction. The difference in shrinkage depends on the material and the production methods. The inclusion of fibres also produces anisotropy. Since mineral fibres (glass and carbon) shrink less than plastics, this tends to negate the differential shrinkage of the plastic and at fibre loadings of above about 20%, it is common to find that the differential is reversed, i.e. shrinkage is greater in the transverse direction.

In dimensioning the mould, the mould dimensions should be slightly oversized compared with the product dimensions. The variability of shrinkage means that product tolerances should be as generous as other requirements permit otherwise tight control of the moulding process is required. Since amorphous materials shrink less than semi-crystalline materials, these materials are preferred where close product tolerances are necessary.

Other shrinkage factors which may need to be considered in dimensioning the mould are briefly listed below.

- (a) The mould may not be at room temperature. Allowance should be made for the cooling of the moulding to room temperature. For amorphous plastics, this is simply a matter of thermal contraction and this may be estimated from the thermal expansion coefficient. Semi-crystalline materials may contract more due to further crystallisation on cooling.
- (b) Even when cooled in the mould to room temperatures, semi-crystalline materials may shrink over a period of time after ejection as a result of further crystallisation. This is known as post-mould shrinkage which, though usually small (less than 0.01%), may have to be taken into account for high precision products. One remedy is to ensure that crystallisation is completed during moulding by, for example, increasing the cooling time.
- (c) If the product is dimensioned for service at elevated temperatures, this may need to be taken into account when deciding upon mould dimensions.

1.2.2 *Sinking*

The moulding of thermoplastics in thick sections presents other shrinkage problems, especially where mould shrinkage is high (e.g. highly crystalline materials). In a thick section of an injection moulded product, for example, the outside layers in contact with the cold mould cool rapidly. The inside layers remain hot for longer because polymers are poor thermal conductors. As the centre layers cool, shrinkage occurs (often to a greater extent than the quickly cooled outer layers) and in doing so pulls the outer layers away from the mould wall causing sinking at the surface (Figure 1.1a).

In extreme cases (e.g. polypropylene) cavitation can occur as well as, or instead of, sinking. The inside of the cooling section shrinks away from itself producing voids or cavities which may result in essentially hollow centred sections which severely weakens the product (Figure 1.1b). Sinking is reduced by

- reducing part thickness;

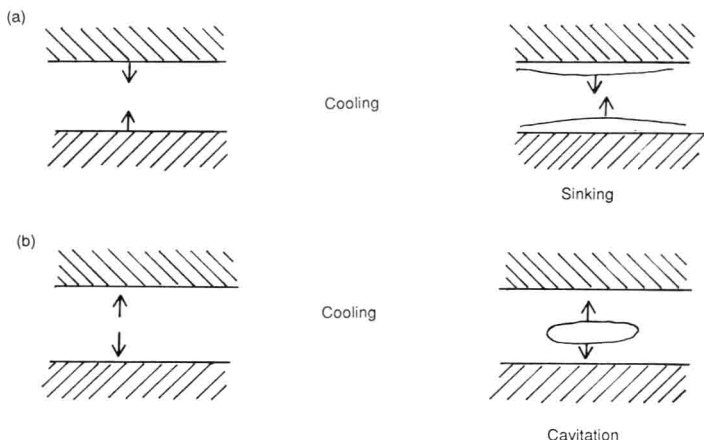


Figure 1.1 Showing forces producing sinking at the moulding surface and cavitation inside the moulding when thick sections are cooled from the melt temperature.

- incorporating fillers;
- maintaining an internal pressure during cooling.

The most convenient way of producing an internal pressure is by the incorporation of a gas in the melt. This can be done by dispersing a blowing agent (e.g. an azo-carbamide) into the polymer which decomposes during processing to provide the gas (nitrogen). The product is a slightly cellular moulding. An alternative is the gas (nitrogen) injection technique (e.g. Cinpress) which forms a tunnel of gas through the centre of thick sections. An advantage of the latter is that surface quality is not impaired through the escape of gas through the moulding surface as happens with blowing agents.

1.2.3 *Internal stress and warping*

When thick sections are moulded, rapid changes in temperature can cause thermally induced stresses due to differential expansion. If surface layers cool faster than the interior through poor conduction, the contraction of the surface will be greater than the interior thereby setting up stresses which can lead to warping and even failure in service at less than predicted stress levels. Thermal stresses induced during manufacture can be reduced by annealing during moulding or after moulding.

Internal stresses can also arise from flow induced anisotropy. Anisotropy arises from two principal sources: molecular orientation and the alignment of directional fillers such as fibres. Molecular orientation results from melt flow where the polymer chains are forced to change from their

random coil state (ideally) to an elongated coil. The degree of elongation depends primarily upon the nature of the polymer and the shear rate (stress) experienced in flow. Shear rate depends on channel dimensions and increases as the channel cross-section decreases. Typical shear rates in practice are $1000\text{--}5000\text{ s}^{-1}$ in injection moulding. The degree of coil distortion can be quite marked. On emerging from the channel (e.g. gate), the elongated coil will attempt to revert to the relaxed random coil state and if it can do so, the product will be isotropic. In practice, this ability to revert is hindered by

- loss of mobility in the polymer due to cooling;
- continued flow into the mould cavity.

The result is frozen molecular orientation in the general direction of flow. The degree of orientation is low compared with that which is deliberately induced in fibre and film production, but it can nevertheless be significant.

Frozen orientation produces stress which weakens the product and causes failure at lower applied stress levels since cracks can propagate more easily in the flow direction. However, stiffness is increased in the flow direction. Frozen orientation can also lead to dimensional instability. The application of heat induces molecular relaxation which produces warping or even gross distortion.

Molecular anisotropy can be minimised by using generous flow channels, low shear rates and slow cooling. Thin sections should be avoided. Fibres (usually glass) of length $0.3\text{--}0.5\text{ mm}$ incorporated into thermoplastics as reinforcements increase the anisotropic effects described above because the fibres tend to orientate in the same flow direction as the polymer chains. Mineral powders of aspect ratio greater than unity (e.g. talc) also contribute to anisotropy but less so than fibres.

1.3 Wall thickness

Component wall thickness requires several competing factors to be taken into consideration. The mechanical requirements of a section may dictate a certain wall thickness. However, thick sections should be avoided because of the requirement to remove heat efficiently (the cooling part of the cycle is the time that limits the production rate), the need to avoid sinking, cavitation and warping. Thin sections should be avoided because of the need for the melt to flow easily to fill the cavity. Economic factors often dictate the final wall thickness selected (amount of material used, etc.). If a wall is too thin for the strength properties required, the wall can be reinforced with ribs (see below). Flow characteristics vary from one plastics material to another and Table 1.2 gives typical values for wall thicknesses.