



BS 5100 : 1974

UDC 621.753.1

BEST COPY AVAILABLE

**Guide to the
Principles of
geometrical tolerancing**

This British Standard, having been approved by the Mechanical Engineering Industry Standards Committee, was published under the authority of the Executive Board on 8 May, 1974.

© British Standards Institution, 1974
ISBN: 0 580 08045 5

Copyright

Users of British Standards are reminded that copyright subsists in all BSI publications. No part of this publication may be reproduced in any form without the prior permission in writing of BSI. This does not preclude the free use, in the course of implementing the standard, of necessary details such as symbols and size, type or grade designations. Enquiries should be addressed to the BSI Secretariat.

Contract requirements

Attention is drawn to the fact that this British Standard does not purport to include all the necessary provisions of a contract.

Revision of British Standards

British Standards are revised, when necessary, by the issue either of amendment slips or of revised editions. It is important that users of British Standards should ascertain that they are in possession of the latest amendments or editions.

The following BSI references relate to the work on this standard: Committee reference MEE/10/1.

Draft for approval 73/38036

8702 3 1 5K IWD



BS 5100 : May 1974

UDC 621.753.1

Guide to the
Principles of
geometrical tolerancing
(Conforming to BS 308 : Part 3 : 1972)

Amendments issued since publication

Amd. No.	Date of issue	Text affected

British Standards Institution
2 Park Street · London W1A 2BS
Telephone 01-629 9000
Telex 266933

BS 5100 : 1974

British Standard Guide to the Principles of geometrical tolerancing

Contents

	Page
Foreword	3
1. Tolerance symbols	4
2. Tolerance frame	6
3. The feature controlled	7
4. Datum	9
5. Datum features	9
6. Dimensions which define true positions – datum dimensions	13
7. Tolerances of straightness	14
8. Tolerances of flatness	15
9. Tolerances of roundness	16
10. Tolerances of cylindricity	17
11. Profile tolerance of a line	18
12. Profile tolerance of a surface	19
13. Tolerances of parallelism	20
14. Tolerances of squareness	21
15. Tolerances of angularity	22
16. Tolerances of position	23
17. Tolerances of concentricity	24
18. Tolerances of symmetry	25
19. Tolerances of run-out	26
20. Maximum material condition 	28

BS 5100 : 1974**Foreword**

Geometrical tolerancing was included in the 1953 edition of BS 308 'Engineering drawing practice'; the method of application to drawings involved marking the dimensions defining true position TP, and using prescribed plain language abbreviations POSN TOL etc., to label the tolerances. Following increasing world use of the system, the International Organization for Standardization (ISO) agreed on a symbol notation and this was given for information in an appendix to BS 308:1964. This notation has now been introduced as the standard convention in BS 308:Part 3:1972 and it is to this standard that readers are referred for more information if required. Meanwhile this booklet is intended to serve as an introduction to the subject for those who are going to be craftsmen, draughtsmen and engineers or for those who already are, but who are more familiar with traditional practices in engineering drawing than with geometrical tolerance techniques.

BS 5100 : 1974

1. Tolerance symbols*

	Type of tolerance	Characteristic to be tolerated	Symbol	Ref. in this book	Ref. in BS 308: Part 3
For single features	Form	Straightness	—	7	3.1.1
		Flatness		8	3.1.2
		Roundness		9	3.1.3
		Cylindricity		10	3.1.4
		Profile of a line		11	3.1.5
		Profile of a surface		12	3.1.6
For related features	Attitude	Parallelism		13	3.2.1
		Squareness		14	3.2.2
		Angularity		15	3.2.3

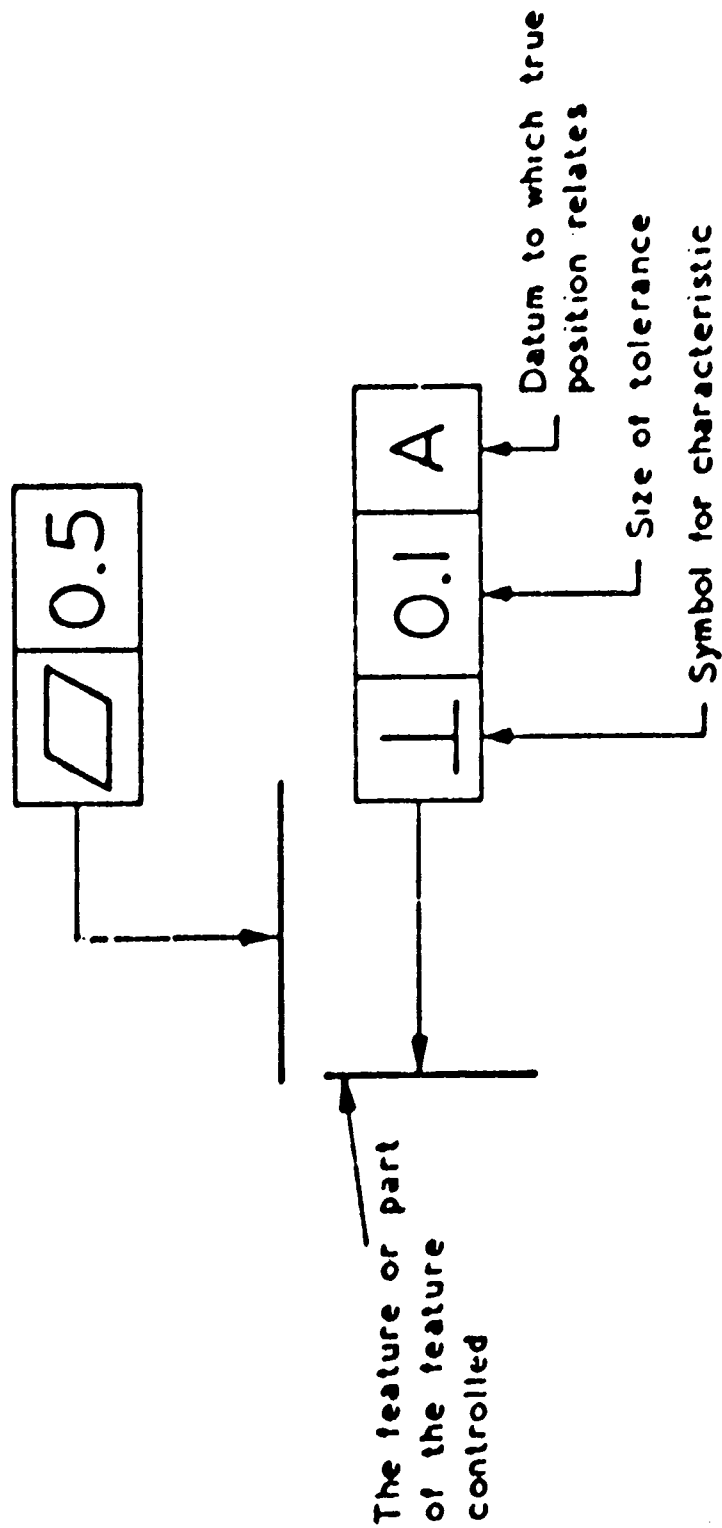
BS 5100:1974

For related features	Location	Position		16	3.3.1
		Concentricity		17	3.3.2
		Symmetry		18	3.3.3
	Composite	Run-out		19	4
Maximum material condition					
				20	2.6
Boxed dimension (dimension which defines true position)					
				6	2.7.1
Indication of datum feature (to be used only as shown in Section 5)					
				5	2.4.1
Indication of circular or cylindrical tolerance zone					
				See 14	2.2.3

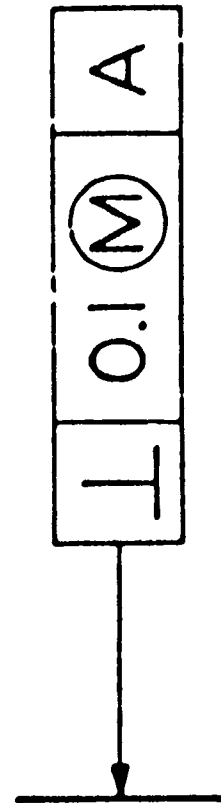
*See Clause 2 of BS 308 : Part 3

BS 5100 : 1974

2. Tolerance frame*



All tolerances apply regardless of feature size unless modified by $\textcircled{M}$ and this would be indicated in such a way as:

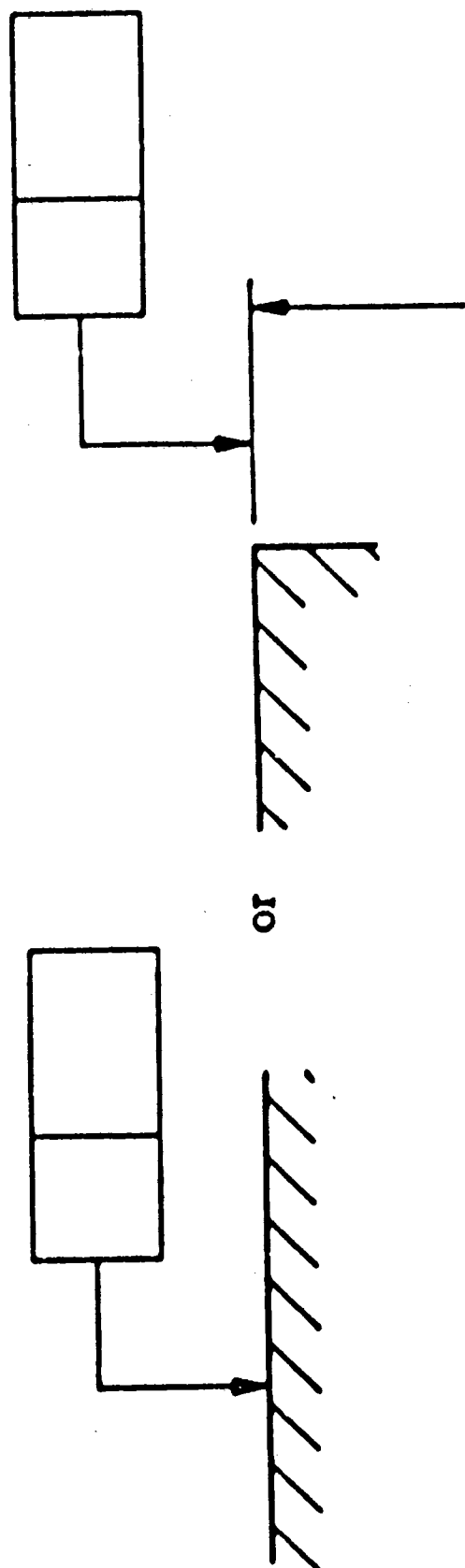


*See 2.2 of BS 308 : Part 3

BS 5100 : 1974

3. The feature controlled*

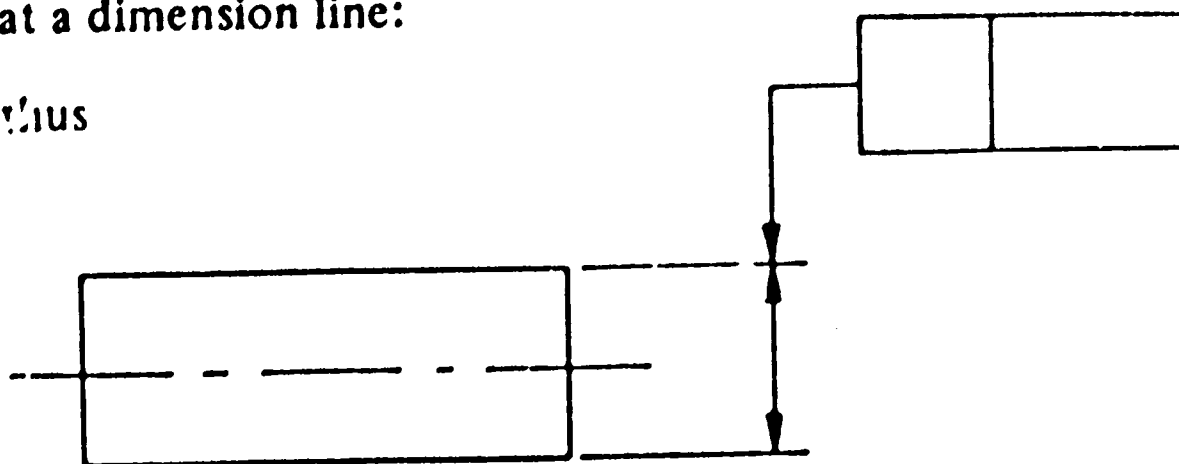
The feature controlled by tolerance is indicated at the surface or at an extension line but *not* at a dimension line, thus:



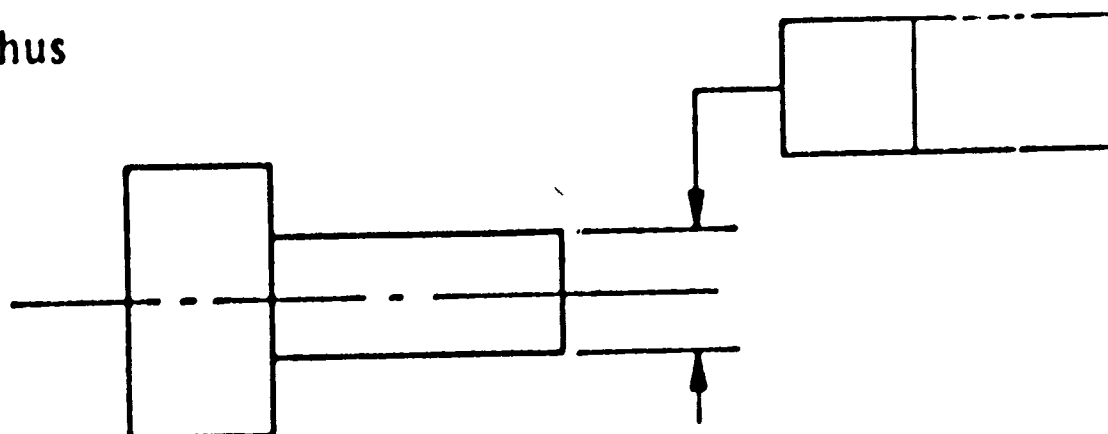
BS 5100 : 1974

If the feature is an axis or median plane it is indicated at a dimension line:

thus

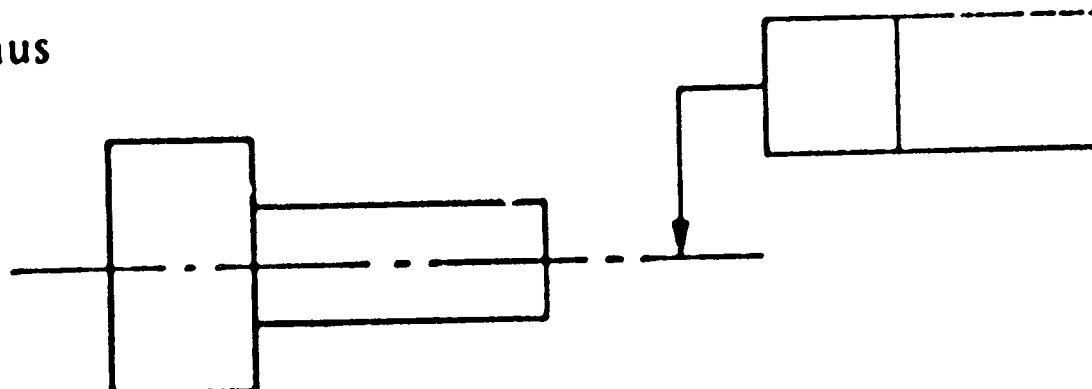


or thus



in this case the axis controlled is the axis of the smaller diameter.

or thus



in this case it is the common axis which is controlled.