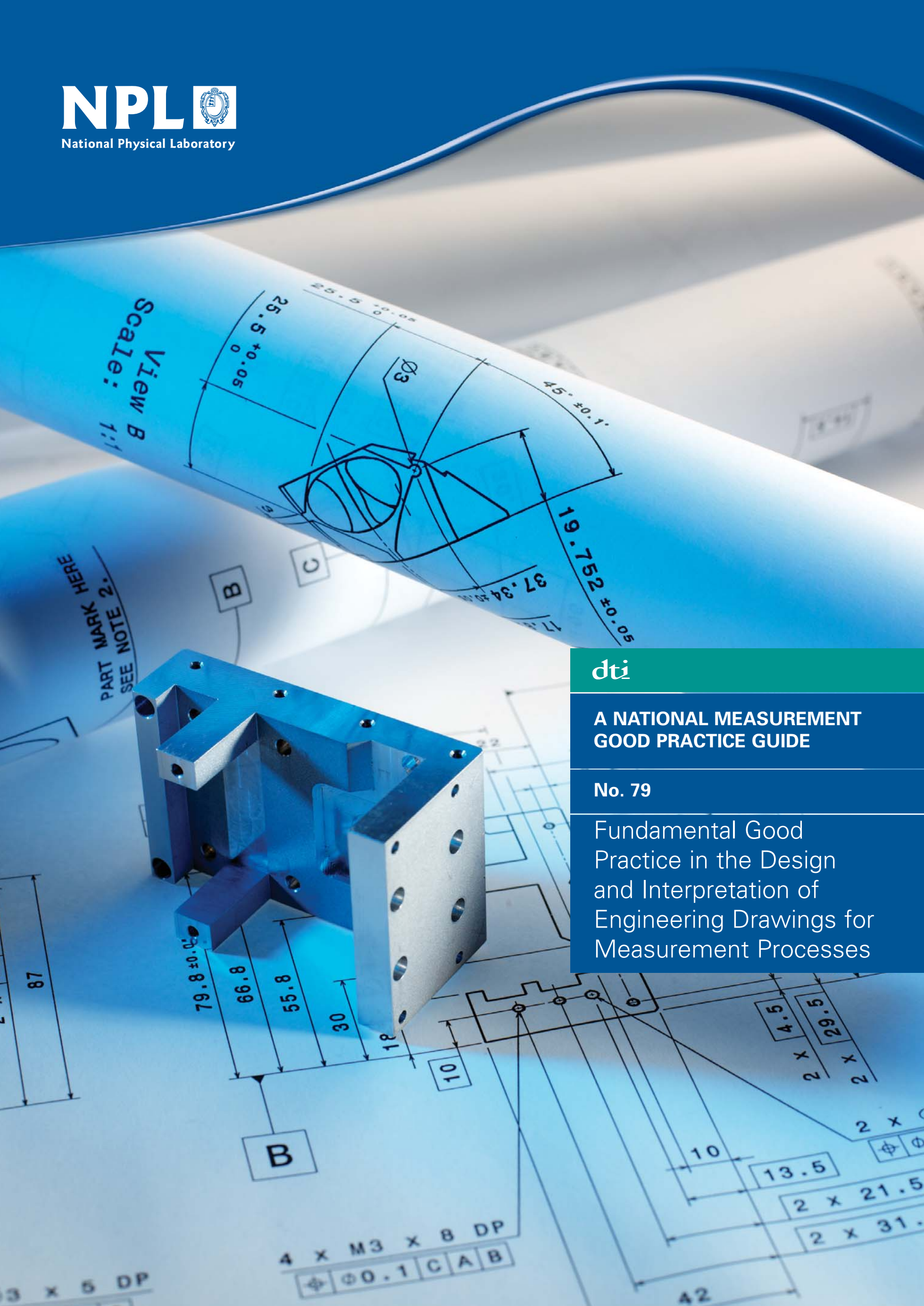




dti

A NATIONAL MEASUREMENT GOOD PRACTICE GUIDE

No. 79

Fundamental Good
Practice in the Design
and Interpretation of
Engineering Drawings for
Measurement Processes



The DTI drives our ambition of 'prosperity for all' by working to create the best environment for business success in the UK. We help people and companies become more productive by promoting enterprise, innovation and creativity.

We champion UK business at home and abroad. We invest heavily in world-class science and technology. We protect the rights of working people and consumers. And we stand up for fair and open markets in the UK, Europe and the world.

This Guide was developed by the National Physical Laboratory on behalf of the NMS.

Measurement Good Practice Guide No. 79

Fundamental Good Practice in the Design and Interpretation of Engineering Drawings for Measurement Processes

David Flack
Engineering Measurement Team
Engineering and Process Control Division

Keith Bevan
Bevan Training and Assessment Services Limited

ABSTRACT

This good practice guide is written for engineers, designers and metrology technicians who wish to understand the basics of the interpretation of engineering drawings in relation to the measurement process. After reading this guide designers should have a better understanding of the measurement process and metrology technicians should be in a better position to interpret the aims of the designer.

© Crown Copyright 2005
Reproduced with the permission of the Controller of HMSO
and Queen's Printer for Scotland

July 2005

ISSN 1368-6550

National Physical Laboratory
Hampton Road, Teddington, Middlesex, TW11 0LW

Acknowledgements

This document has been produced for the Department of Trade and Industry's National Measurement System Policy Unit under contract number GBBK/C/08/17. Thanks are due to Hexagon Metrology, Tesa Technology and Renishaw for providing some of the images and to Dr Richard Leach (NPL) and Prof. Paul Scott (Taylor Hobson) for suggesting improvements to this guide.

Contents

Introduction.....	11
What this guide is about, and what it isn't?.....	12
Holistic approach to production	12
An introduction to Geometrical Product Specification	12
Interpreting a drawing in preparation for manufacture	16
Interpreting a drawing in preparation for measurement	17
The standard reference temperature	18
Design.....	21
The designer's role – an introduction to the modern design process utilising CAD, FEA and mathematical modelling.....	22
Design interpretation to make manufacturing and measurement easier.....	23
So why do dimensions require tolerances?	26
Measurement considerations when designing components.....	26
Can I design the component to make measurement easier?	26
Design changes to aid holding the component.....	26
Design changes to aid access to features.....	27
Design changes to allow repositioning to be used	27
Manufacture	29
An introduction to the manufacturing of components.....	30
Overview of basic manufacturing and machining processes	31
Stock removal techniques.....	31
Volume production processes	35
Monitoring the process	35
Trend monitoring and statistical control.....	36
Process control and measurement feedback.....	37
Drawings	39
A simple overview of what information is being conveyed, for example, size, position, shape and surface texture.....	40
Introduction	40
Co-ordinate systems	40
General tolerances	41
Geometric tolerancing.....	41
Geometric references (datum features)	43
Feature control frames.....	45

Geometric characteristics	46
Form tolerances	47
Tolerances of location	50
Maximum material condition	51
Tolerances of orientation.....	55
Tolerances of run-out	57
Surface texture.....	58
Establishing a co-ordinate system (datum) for a component	59
Can the datum be easily measured?.....	63
The virtual datum	63
Why is a partial arc a bad datum?	64
Measuring partial arcs on CMMs – the problems	64
Design consideration – partial arcs	65
Does the tolerance really need to be that tight?.....	66
Least squares or minimum zone	66
Inspection.....	69
Ensuring that the components meet the requirements.....	70
Sampling versus 100% inspection.....	70
Choosing the appropriate measurement tool	70
Is it possible to measure the dimensions specified on the drawing to ascertain if the tolerance has been met?	71
The importance of traceability	71
Uncertainty of measurement	73
Determining conformance with a specification - ISO 14253 decision rules....	73
Summary of ISO 14253.....	77
Published Standards	79
List of published standards.....	80
Glossary of terms	81
Glossary of terms.....	82
Appendices.....	85
Appendix A Links to other useful sources of information	86
A.1 National and International Organisations.....	86
A.1.1 National Physical Laboratory	86
A.1.2 National Institute of Science and Technology (NIST).....	86
A.1.3 EUROMET.....	86
A.1.4 Institute for Geometrical Product Specification.....	87
A.2 Networks	87

A.2.1	Dimensional Metrology Awareness Club (DMAC).....	87
A.2.2	Software Support for Metrology Programme (SSfM).....	87
A.3	National and International Standards	87
A.3.1	British Standards Institution (BSI)	87
A.3.2	International Organisation for Standardization (ISO)	88
A.4	Traceability.....	88
A.5	National Measurement Partnership (NMP).....	89
A.6	Training courses	89
A.6.1	E-training.....	90
A.7	Further reading	92
Appendix B	Further reading	93
B.1	Maximum material condition – an example.....	93
B.2	Statistical Process Control.....	94
B.2.1	Mean	94
B.2.2	Range	94
B.2.3	Standard deviation	94
B.2.4	Normal bell shaped curve	94
B.2.5	Histogram and control chart	95
B.2.6	Capability.....	96
B.2.7	Six Sigma.....	97

List of Figures

Figure 1	A typical manufactured component (Hexagon Metrology).....	13
Figure 2	The General concept of GPS tolerancing	14
Figure 3	The GPS tolerancing model.....	15
Figure 4	A set of gauge blocks (Hexagon Metrology)	18
Figure 5	A co-ordinate measuring machine.....	18
Figure 6	Design on a CAD system (Hexagon Metrology)	22
Figure 7	A typical drawing	23
Figure 8	A micrometer (Tesa Technology)	24
Figure 9	Callipers (Tesa Technology)	24
Figure 10	Bore gauges (Tesa Technology).....	24
Figure 11	A dial indicator (Tesa Technology).....	24
Figure 12	A roundness-measuring instrument.....	25
Figure 13	A mounted sphere.....	27
Figure 14	Attaching spheres allows repositioning to be used.....	27
Figure 15	An item too big for the CMM being measured by repositioning (the white spheres are used for repositioning).....	27
Figure 16	On-machine gauging (Renishaw)	31
Figure 17	On-machine gauging (Renishaw)	31
Figure 18	Gauging instrumentation (Tesa Technology).....	31

Figure 19 A typical lathe.....	32
Figure 20 A typical milling machine	33
Figure 21 A typical wire cutting machine.....	34
Figure 22 Wire cutting in action	34
Figure 23 Inspecting a batch (Hexagon Metrology)	36
Figure 24 Basic control chart layout.....	37
Figure 25 Cartesian co-ordinate system.....	40
Figure 26 Classification of geometrical tolerances.....	42
Figure 27 Overview of geometrical tolerances	43
Figure 28 Datum triangles and datum letters	44
Figure 29 Datum axis.....	44
Figure 30 Datum surface or feature extension of surface	45
Figure 31 Datum target frames	45
Figure 32 A feature control (tolerance) frame and the information it can contain.....	46
Figure 33 A simple feature control frame.....	46
Figure 34 Circularity (roundness) symbol	47
Figure 35 Circularity (roundness) definition	47
Figure 36 Straightness symbol.....	47
Figure 37 Straightness definition	48
Figure 38 Flatness symbol	48
Figure 39 Flatness definition	48
Figure 40 Cylindricity symbol	49
Figure 41 Cylindricity definition	49
Figure 42 Profile symbols.....	49
Figure 43 Profile tolerance zones bilateral and uni-lateral	49
Figure 44 Line profile tolerance	50
Figure 45 Position tolerances.....	50
Figure 46 Cylindrical tolerance zone	51
Figure 47 The MMC symbol placement.....	51
Figure 48 The concept of maximum material condition.....	52
Figure 49 MMC example frames	53
Figure 50 Concentricity/co-axiality symbol	53
Figure 51 Concentricity definition.....	53
Figure 52 Symmetry feature control frame.....	54
Figure 53 Symmetry of a slot.....	54
Figure 54 Parallelism symbol	55
Figure 55 Example of a parallelism tolerance	55
Figure 56 Parallelism with cylindrical tolerance zone.....	55
Figure 57 Perpendicularity symbol.....	56
Figure 58 Example of a perpendicularity tolerance	56
Figure 59 Example of a perpendicularity tolerance.....	56
Figure 60 Perpendicularity with a cylindrical tolerance zone	57

Figure 61 Angularity symbol	57
Figure 62 Angularity definition	57
Figure 63 Run-out symbol	58
Figure 64 Run-out can be in the circular or axial direction	58
Figure 65 Total run-out	58
Figure 66 Surface texture symbol	58
Figure 67 Measuring surface texture	59
Figure 68 A surface texture measuring instrument	59
Figure 69 Aligning a component - exaggerated (Hexagon Metrology)	59
Figure 70 Defining the axis of rotation (Hexagon Metrology)	61
Figure 71 The affect of mis-alignment (Hexagon Metrology)	61
Figure 72 Creating a line to rotate about the z-axis (Hexagon Metrology)	62
Figure 73 Setting the origin (Hexagon Metrology)	62
Figure 74 A 3-2-1 alignment	63
Figure 75 A partial arc	64
Figure 76 Ten uniformly spaced measurements on a partial arc, showing the (nominal) circle of which it is part	65
Figure 77 A partial arc with a form tolerance	66
Figure 78 Traceability chain for gauge blocks	72
Figure 79 The two results and their uncertainties	74
Figure 80 Uncertainty of measurement: the uncertainty range reduces the conformance and non-conformance zones (Copyright BSI – extract from BS EN ISO 14253- 1:1999)	75
Figure 81 Conformance or non-conformance	76
Figure 82 E-training via the NPL website	90
Figure 83 The introductory screen to the analogue-probing module	90
Figure 84 The drawing requirements and actual measured values obtained	93
Figure 85 The calculations of the hole position both before and after the application of the MMC principle	93
Figure 86 Normal or bell shaped curve	95
Figure 87 A histogram	96
Figure 88 A control chart	96

Fundamental good practice in the interpretation of engineering drawings

Preface

The authors hope that after reading this Good Practice Guide you will be able to make better measurements of the size or shape of an object. The content is written at a simpler technical level than many of the standard textbooks on “Engineering Design” so that a wider audience can understand it. We are not trying to replace a whole raft of good textbooks, operator’s manuals, specifications and standards, rather present an overview of good practice and techniques.

“Metrology is not just a process of measurement that is applied to an end product. It should also be one of the considerations taken into account at the design stage. According to the Geometrical Product Specification (GPS) model, tolerancing and uncertainty issues should be taken into account during all stages of design, manufacture and testing. The most compelling reason is that it is often considerably more expensive to re-engineer a product at a later stage when it is found that it is difficult to measure, compared to designing at the start with the needs of metrology in mind.” Dr Richard Leach 2003

GOOD MEASUREMENT PRACTICE

There are six guiding principles to good measurement practice that have been defined by NPL. They are:

The Right Measurements: *Measurements should only be made to satisfy agreed and well-specified requirements.*

The Right Tools: *Measurements should be made using equipment and methods that have been demonstrated to be fit for purpose.*

The Right People: *Measurement staff should be competent, properly qualified and well informed.*

Regular Review: *There should be both internal and independent assessment of the technical performance of all measurement facilities and procedures.*

Demonstratable Consistency: *Measurements made in one location should be consistent with those made elsewhere.*

The Right Procedures: *Well-defined procedures consistent with national or international standards should be in place for all measurements.*

Introduction

1

IN THIS CHAPTER

- What this guide is about, and what it isn't.
- Holistic approach to production.
- An introduction to Geometrical Product Specification.
- Chains of GPS standards.
- Interpreting a drawing in preparation for measurement, the standard reference temperature.

This measurement good practice guide will provide an overview of the three main stages of production – design, manufacture and inspection – with a view to providing a holistic approach to the communications required to achieve optimum production within a modern manufacturing concern.

What this guide is about, and what it isn't?

It is intended that this guide should give enough information so that the metrologist can interpret the designer's specification and to give the designer some background to modern measurement methods. This good practice guide is not intended to be an authoritative guide on producing engineering drawings.

Holistic approach to production

An holistic approach to the production process can only be achieved if everyone in the process communicates adequately. The primary means of communication is the engineering drawing and it is important that all key players in the process are involved right from the start of the design.

“Metrology is not just a process of measurement that is applied to an end product. It should also be one of the considerations taken into account at the design stage. According to the Geometrical Product Specification (GPS) model, tolerancing and uncertainty issues should be taken into account during all stages of design, manufacture and testing. The most compelling reason is that it is often considerably more expensive to re-engineer a product at a later stage when it is found that it is difficult to measure, compared to designing at the start with the needs of metrology in mind.” Dr Richard Leach, 2003.

One significant problem for many people working in modern manufacturing is that the whole process has become so complicated that very few people understand it all – the trend is for people to become specialised.

A product must meet design requirements, product specifications and standards. It must then be manufactured by the most environmentally friendly and economical methods. Quality must be built in at every stage from design through to assembly, instead of only testing upon completion. To be competitive, production methods must be flexible so that responses can be made to market demand, product types, quantity and rate of production requirements. New developments in materials, production methods and computer aided manufacture must be evaluated and implemented as deemed appropriate. A manufacturing organisation is a large and complex system that requires feedback from all levels to ensure optimum use of all its materials, machines, energy, capital, labour and technology resources.

An introduction to Geometrical Product Specification

Before going too deeply into the subject of engineering drawings you need to know a little bit about Geometrical Product Specification or GPS. The idea of GPS is to give assurance in obtaining the following essential properties of a product:

- ❑ functionality,
- ❑ safety,
- ❑ dependability,
- ❑ interchangeability.

GPS is implemented through a series of standards. GPS standards have been divided into four groups:

- ❑ Fundamental GPS standards (fundamental rules for dimensioning and tolerancing).
- ❑ Global GPS standards (for example, ISO 1 on the standard reference temperature).
- ❑ General GPS standards (for example, the chain of standards listed on page 16).
- ❑ Complementary GPS standards (for example, technical rules for drawing indications).

This section will give a brief introduction to GPS. A more comprehensive description can be found in *Geometric Product Specifications – course for Technical Universities* (see Appendix A.7 for full details).

The primary means of communication within a production environment is an engineering drawing that provides a clear and unambiguous definition of the part geometry and specifies the limits of imperfection of that geometry. These limits are referred to as the tolerance.



Figure 1 A typical manufactured component (Hexagon Metrology)

At the first design stages of a component, the designer imagines the product to be an ideal, perfectly manufactured object. All component parts are assumed to be of perfect form and size. Manufacturing processes cause component parts to vary in many different ways. For example, there can be variations in dimension, form and surface texture. These variations can have a great effect on the functionality of the component. It is, therefore, critical that the definitions of geometry are standardised and understood, so that the variation that is inherent in manufacturing processes can be taken into account to minimise waste products. To be able

to understand the geometrical variations within component parts a set of requirements have been produced. These requirements are the part of GPS, covering sizes and dimensions, geometrical tolerances and geometrical properties of surfaces (see Figure 2).

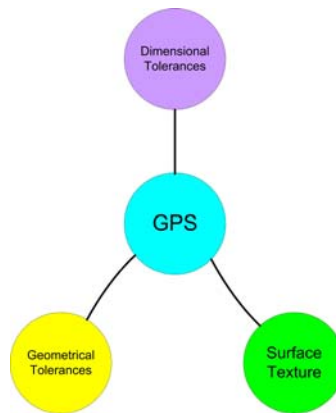


Figure 2 The General concept of GPS tolerancing

The fundamental effects on the fit, function, safety and quality of component parts can be addressed by implementing the concepts of GPS.

GPS is an internationally accepted concept covering all of the different requirements indicated on a technical drawing relating to the geometry of industrial workpieces (for example, size, distance, radius, angle, form, orientation, location, run-out, surface roughness, surface waviness, surface defects, edges, *etc.*) and all related verification principles, measuring instruments and their calibration. Expressed more simply this entails specifying the requirements for the micro and macro geometry of a product (workpiece) with associated requirements for verification and calibration of related measuring instruments.

As explained in the introduction of ISO 14660-1:1999:

Geometrical features exist in three “worlds”:

- ❑ *the world of specification, where several representations of the future workpiece are imagined by the designer;*
- ❑ *the world of the workpiece, the physical world;*
- ❑ *the world of inspection, where a representation of a given workpiece is used through sampling of the workpiece by measuring instruments.*

It is important to understand the relationship between these three worlds. ISO 14660 defines standardized terminology for geometrical features in each world as well as standardized terminology for communicating the relationship between each world.

The Duality Principle is also important to mention here. It is defined in ISO/CD 14659 as

A specification is defined independently of any measurement procedure or – equipment. The measurement and equipment is fully controlled by the specification (specification operator – verification operator).

All the rules are defined for specifications only – and metrology shall apply to the rules – deviations/differences will be part of the uncertainty of measurement.

NOTE

During 1996 the Technical Committee ISO/TC 213 *Dimensional and Geometrical Product Specifications and Verification* was established. This technical committee was established to prepare the GPS documentation for the International Organization for Standardization (see A.3.2). It so happens that there is also another committee in Europe known as CEN/TC 290 that works in parallel, so to keep continuity between the organisations an agreement was made between the two technical committees to prepare identical documents on GPS. Most countries adopt these international standards on GPS as national standards.

It is important that all persons involved in design, manufacturing and metrology must have knowledge of the requirements of GPS. It is critical that the communication between the relevant engineering departments involved in GPS is clear, precise, accurate, and that each department understands the product design requirements.

The GPS standards contain fundamental technical rules that define technical drawings. The fundamental elements included within the GPS models are shown in Figure 3.

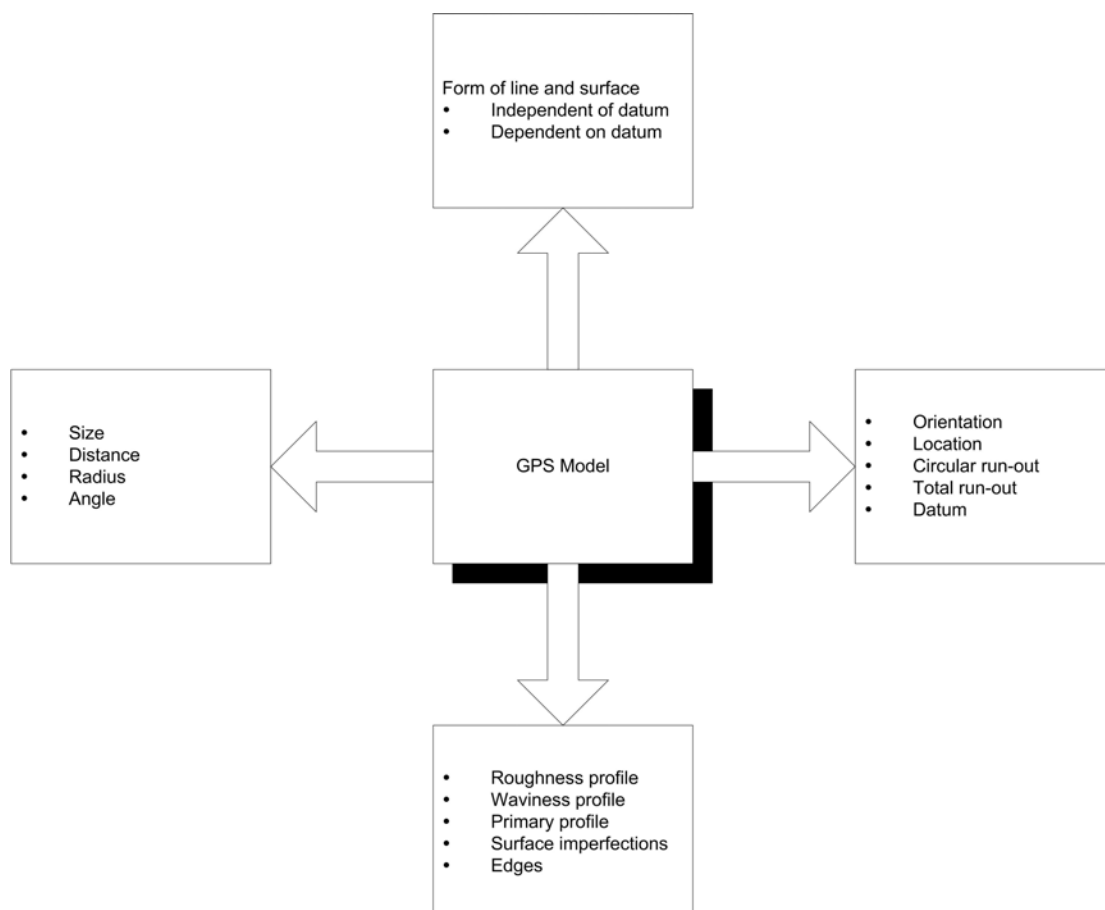


Figure 3 The GPS tolerancing model

The GPS standard technical rules are organised into six chain links for any given characteristic and are defined as listed below:

1. The rules, symbols and how to understand the specifications of product documentation.
2. Theoretical definitions of tolerances and their numerical values.
3. Geometry of a non-ideal, real workpiece defined in relation to tolerance symbols on the drawing.
4. The conformance, non-conformance of real workpiece deviations to specification taking into account measurement uncertainty.
5. General approach to measurement equipment types and requirements.
6. Calibration standards, procedures and requirements of the measuring equipment used and their link to national and international standards.

All of the above links relate to the design, manufacturing and measurement process.

More recently the GPS master plan has been modified to take into account the Duality Principal and now includes seven links:

Column 1: Product documentation indication – codification

Column 2: Definition of tolerances - theoretical definition and values

Column 3: Definitions of characteristics of extracted features - specification operators

Column 4: Comparison

Column 5: Measured value of characteristics of extracted features - verification operator

Column 6: Metrological characteristics of measurement equipment

Column 7: Calibration and verification of metrological characteristics of measurement equipment

Specification			Comparison	Verification		
1	2	3	4	5	6	7

Necessary for unambiguous contract	Necessary for verification only
------------------------------------	---------------------------------

Go to the Institute for Geometrical Product Specification website www.ifgps.com and click on resources for more information about GPS.

Interpreting a drawing in preparation for manufacture

It is not usually the prerogative of the designer to decide the details of the machining of a component, although it is often possible to foretell the sequence of some of the manufacturing processes involved. From knowing the manufacturing sequence the designer can identify the manufacturing datum face(s)¹, and from this, the required machining dimensions. The datum faces will be those faces used to hold the component during manufacture.

¹ Datum – a theoretically exact feature to which other toleranced features are related.

It is common practice in various industries to produce stage drawings. The datum features that are used to produce components stage by stage may not be the same as the finished drawing datum features. For example, a hole may have been produced in the component to allow for the product to line up on a fixture to produce other features, for example, turbine blades slots. Once the slots have then been produced, the hole could be in a feature that is then removed before the component is fully completed.

The planning of the manufacturing process based on the stage or final drawings is vital to the success of producing quality component parts. Many areas have to be considered before the manufacturing process begins. Consideration of the following is important:

- What is the required method of manufacture?
- Availability of resources, such as machines, tooling, personnel, equipment.
- How do we hold the component?
- Is fixturing required?
- Do we use in-line measurement?
- Gauging or measuring instruments?
- What are the best instruments to use?
- Have we considered the influences of measurement uncertainty?
- Will training be required?

Once these questions have been answered the production process can begin. It is now important to think about how to monitor the process. We will know our current capability, but can we control our processes consistently? It may be that as part of the manufacturing process we will have to use statistical techniques to assist us.

Interpreting a drawing in preparation for measurement

The importance of interpreting the design requirements cannot be stressed too highly during preparation for measurement. Identifying the geometric characteristics of the component and the datum features that make up the co-ordinate system is critical to a successful measurement strategy. When making measurements you should make use of datum features identified in drawings, technical documents or computer aided design (CAD)² models that relate directly to the component.

Datum features on a drawing are normally an important characteristic – a locating or positioning feature. A datum feature could be a face (a surface), a centre line (an axis), or a series of characteristics that collectively make up a datum system. The datum system may be easy to set up when using conventional measuring equipment, such as a surface table in conjunction with angle plates, dial gauges, height gauges and gauge blocks (Figure 4).

Alternatively, the use of CAD data may be a requirement of the measurement process and, therefore, computerised measuring equipment such as the co-ordinate measuring machine (CMM), may need to be used (Figure 5).

² Sometimes CAD is referred to as computer aided drafting, CADD referring to computer aided drafting and design. CAD and CADD are essentially the same.



Figure 4 A set of gauge blocks (Hexagon Metrology)



Figure 5 A co-ordinate measuring machine

When setting up a datum for measurement, as will become apparent later, it is preferable to choose as datum features the surfaces that were used in the manufacturing process to hold the component. This choice relates the inspection results directly to the manufacturing process.

The features of any component can be defined in two ways, relative to a datum position or positions (absolute), or relative to one another (incremental). The co-ordinate system should be clearly defined whether on a physical drawing or CAD model.

From the drawing the measurement strategy must be determined for the geometric characteristics and the co-ordinate system. Account must be taken of environmental considerations such as temperature effects, equipment required and the associated uncertainties in relation to the stated specifications. These points will be covered in more detail throughout this guide.

The standard reference temperature

The most important GPS standard is ISO 1:2002 *Geometrical Product Specifications (GPS) – Standard reference temperature for geometrical product specification and verification*. This

specification defines the standard reference temperature for all dimensional measurement and all other GPS standards refer to it. GPS standard ISO 1:2002 states:

The standard temperature for geometrical product specification and verification is fixed at 20 °C.

What does this mean for the designer, the manufacturer and the dimensional metrologist?

For the designer it means that all dimensions and tolerances on the drawing apply to a component that is at a temperature of 20 °C. If the component normally operates at a higher temperature the designer will need to correct the desired sizes at this higher temperature to 20 °C for the drawing. During manufacture the component must be measured close to 20 °C, how close to 20 °C depends on the materials and the tolerances involved.

Final inspection will either be made:

- in a temperature controlled room at 20 °C;
- by comparison with known artefacts of similar material at a temperature close to 20 °C;
- by a measuring machine that measures the component temperature and makes appropriate corrections or
- by the operator making the measurement at some other temperature and making manual corrections.

NOTE



Temperature

How much the part material changes size for a given temperature change is known as the coefficient of linear thermal expansion. For a typical material such as steel this is expressed as $11.6 \times 10^{-6} \text{ }^{\circ}\text{C}^{-1}$. To correct a length to 20 °C use the following equation:

$$L_{20} = L_T + (20 - T) \times \alpha \times L_T$$

where L is length, T is the temperature at which the length was measured and α is the expansion coefficient.

For example, a steel bar that is measured as 300.015 mm at a temperature of 23.4 °C has a length of

$$300.015 + (20 - 23.4) \times 11.6 \times 10^{-6} \times 300.015 = 300.003 \text{ mm}.$$

Design

2

IN THIS CHAPTER

- The designers role - an introduction to the modern design process utilising CAD, FEA and mathematical modelling.
- Manufacturing considerations when designing a component.
- Can I design the component to make manufacture easier?
- Measurement considerations when designing a component.
- Can I design the component to make measurement easier?
- Design changes to aid holding the component.
- Design changes to aid access to features.
- Design changes to allow repositioning to be used.

The designer's role – an introduction to the modern design process utilising CAD, FEA and mathematical modelling

The design process begins with a list of functional requirements and goes on to the technical specification. At this point the designer is beginning to create the solution to the customer's problem and can imagine the component that he wants to have made.

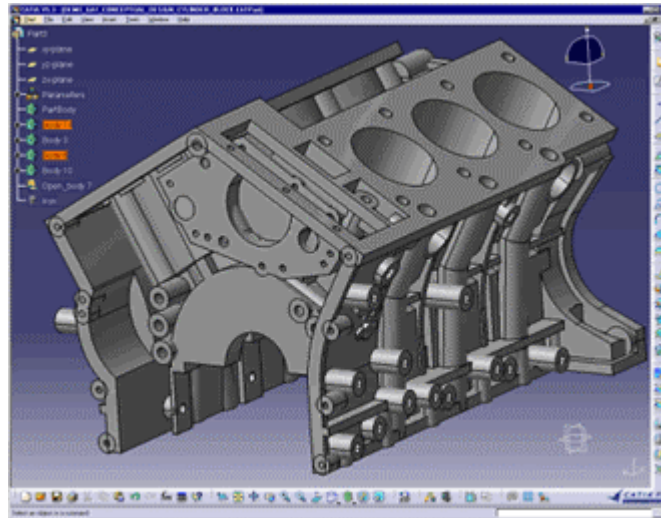


Figure 6 Design on a CAD system (Hexagon Metrology)

The design can be done by conventional methods such as the production of a technical drawing or by use of computer aided design (CAD). CAD is an electronic tool that enables the designer to make quick and accurate drawings with the use of a computer. The CAD system allows the designer to construct a three dimensional representation of the component that can be viewed from many different angles to check the functionality and appearance (as shown in Figure 6).

Advanced CAD systems will check interference between components, model the motion of moving parts and calculate stress concentrations and resonant frequencies of components and assemblies.

In order to have the component made the designer must construct a co-ordinate system to allow dimensioning of the component and to provide the detailed information to the manufacturing engineer.

A co-ordinate system is usually a conventional Cartesian³ co-ordinate system with three orthogonal axes – conventionally denoted x , y and z (see Chapter 4). In some situations, a cylindrical co-ordinate system (with a radial distance, an angle and a height) would make sense and in other rare situations a spherical co-ordinate system (with a radial distance, and two angles) could be used.

³ Named after René Descartes

The modern designer utilises CAD, finite element analysis (FEA)⁴ and mathematical modelling to fulfil the design requirements and product specifications. Consideration of many different aspects should be addressed for the benefit of all involved in the process such as:

- Manufacturing considerations when designing a component.
 - Can I design the component to make manufacture easier?
- Measurement considerations when designing a component.
 - Can I design the component to make measurement easier?
 - Design changes to aid holding the component.
 - Design changes to aid access to features.

These points will now be covered in more detail.

Design interpretation to make manufacturing and measurement easier

Inadequate drawings lead to delays in production, rework on the assembly floor and reliance on a subcontractor who has the know-how to deliver what is required.

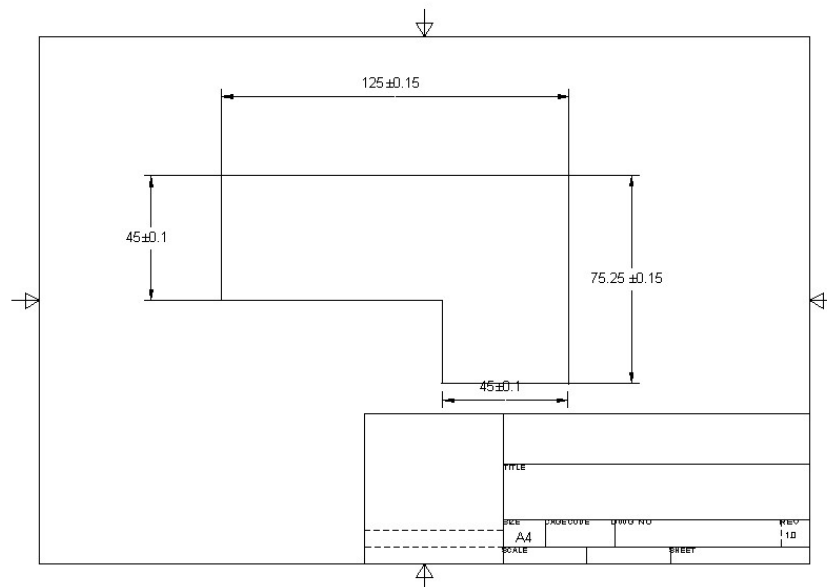


Figure 7 A typical drawing

There are international and national standards regarding the conventions and symbols to be used on engineering drawings and it is not the intention of this best practice guide to duplicate them here – rather we shall provide an overview of what is important to communicate, how to communicate it, and how to translate that knowledge into sensible actions for the shop floor. Figure 7 shows part of a typical drawing.

⁴ FEA, or finite element analysis, is a technique for predicting how structures and materials respond to environmental factors such as forces, heat and vibration.

The basic problem with interpreting engineering drawings is that not all the people involved in the design, manufacture and inspection of a component have read the 1700+ pages of the various drawing standards such as BS 8888, the various ISO standards, ASME Y14.5, to name but a few. As a result, few people really fully understand what has been written on a drawing or what should be inferred from it – they certainly could not agree on everything if they were sitting around a table together discussing a drawing.

The purpose of an engineering drawing is to show the requirements of the design function, with clear and relevant information so that the product can be manufactured and inspected to those requirements. The methods used in the design process should be clear and concise so as not to cause ambiguity in the interpretation of the design and this should in turn allow everyone involved in the process to interpret the design requirements.

Traditionally measurement of a component meant using techniques such as height gauges, micrometers, gauge blocks and other reference artefacts (as shown in Figure 8 to Figure 11).



Figure 8 A micrometer (Tesa Technology)



Figure 9 Callipers (Tesa Technology)



Figure 10 Bore gauges (Tesa Technology)



Figure 11 A dial indicator (Tesa Technology)

Today the technique that has almost totally replaced these traditional techniques is the use of a co-ordinate measuring machine (Figure 5). Other techniques that the modern metrologist has in his armoury include devices such as roundness-measuring instruments (Figure 12).

Conventional metrology generally involves sampling a few points or estimating the change in reading of a dial indicator as it was moved over the surface. Modern instruments allow almost complete coverage of the surface and, therefore, allow for easier determination of form errors and more accurate determination of for instance mean diameter.



Figure 12 A roundness-measuring instrument

When it comes to measurement there are many issues that can cause problems, for example, the measurement of a partial arc. The problem is in this case that the drawing specification requires the measurement of the diameter of a circle but only part of the circle can be accessed (a partial arc) – the rest of the circle is ‘virtual’. Partial arcs are particularly sensitive to small variations in the shape of the arc and the apparent radius and position of the centre will change radically. We will discuss these issues in more depth in Chapter 5.

The dimensions and tolerances stated on the drawing can sometimes cause confusion. If a technical drawing or CAD model stated that a diameter of 20.0 mm was required to be produced, what methods and equipment would be used to manufacture and measure the part? The choice will come down to many factors, but consideration of the number of decimal places specified against the tolerance band plays a major part in your decision rules.

For example, what measuring instruments might you use to measure the following diameters:

1. Diameter 20.0 ± 0.2 mm,
2. Diameter 20.00 ± 0.02 mm?

Diameter 1 could be measured using a calliper and diameter 2 a micrometer. This highlights the importance of the influence the tolerance has on your measurement strategy.

NOTE



The application of $\pm$ limit specifications usually causes large specification uncertainty. It must be emphasised that any $\pm$ limit specification, generally speaking, can be completely substituted by geometrical dimensioning and tolerancing as per ISO 1101 and its related standards. Consequently, it is recommended that the use of $\pm$ limit specifications be restricted to features of size only.

So why do dimensions require tolerances?

The design drawings or CAD models of component parts identify the ideal dimensions required within given specifications. It is known that manufacturing processes inherently produce variation, whether caused by the machine tool, tooling, materials, operator, poor gauging, lack of maintenance, measuring equipment or lack of training. When measurements are made during the manufacturing process it is important to minimise this variation. Variation can be minimised by having a good understanding of the requirements of the design drawing, including tolerance types. Emphasis must be put on measuring techniques and equipment that are fit for purpose to verify these dimensions, an understanding of any associated measurement uncertainties is imperative.

Measurement considerations when designing components

When designing a component the designer must constantly ask the question ‘Can this item be measured?’

Tolerances may have been specified for which the technology does not exist to prove conformance/non-conformance. If the technology does exist your company may not have access to it or the measurements could be extremely expensive and may not have been budgeted for.

Has enough room been allowed for to allow access to the features that need measuring? Are the tolerances reasonable? Are datum features large enough? Have tolerances for partial features (for example, partial arcs) been specified in the most appropriate way?

Early in the design process the designer should contact the laboratory performing the measurements, particularly if it is outside of their own organisation. Small changes at the design stage could prevent a costly quote for the measurement later.

Can I design the component to make measurement easier?

The answer to this question is inevitably yes. A number of design features may be added that do not affect the function of the component that will make life easier for the person performing the measurements. The designer should speak to the person performing the measurements prior to finalising the drawing. Small changes made prior to finalising the drawing may save lots of time later. An example of this would be tightening the surface texture tolerance, which could reduce the variability in the measurement process.

Design changes to aid holding the component

A problem often encountered with small components is how to hold them for measurement without distorting them or blocking access to features. Sometimes the simple addition of some holes, a flange or a spigot at the design stage can make measurement so much easier. The designer may even consider designing a fixture especially for the purpose of measurement.

Design changes to aid access to features

Design changes to allow access for measurement devices is something a designer should always consider. This may be the simple addition of a small viewing hole or the removal of non-functional material for instance widening a slot to allow access for micrometer anvils.

Design changes to allow repositioning to be used

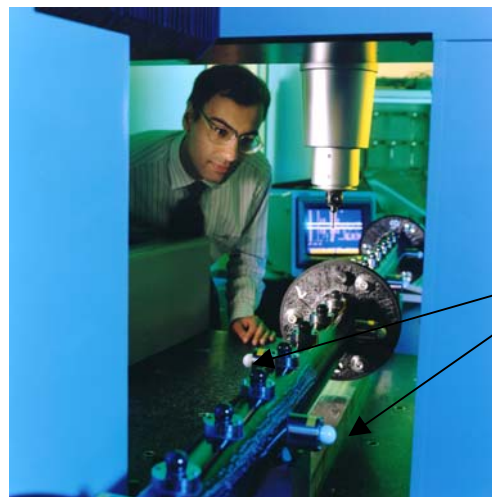
A powerful technique called repositioning allows components that would be otherwise too big for a particular CMM to be measured in two sections. By designing tapped holes into the component where reference spheres (Figure 13 and Figure 14) can be attached, measurements on the two sections can be made relative to one co-ordinate system. Figure 15 shows an item being measured that is too big for this particular CMM. The operator is measuring the separation of the black spheres. By attaching spheres to the side of the component several measurements can be stitched together in to a common frame of reference.



Figure 13 A mounted sphere



Figure 14 Attaching spheres allows repositioning to be used.



Repositioning
spheres (white)

Figure 15 An item too big for the CMM being measured by repositioning (the white spheres are used for repositioning)

For more information on repositioning see Cox M G, *et al* 1997 *Measurement of artefacts using repositioning methods* NPL Report CLM2.

Manufacture

3

IN THIS CHAPTER

- The manufacturing engineer's role – an introduction to the manufacturing of components.
- Basic machining processes.
- Volume production processes - die casting.
- Measurement during manufacture, manual and in-line.
- Trend monitoring and statistical control.
- Measurement feedback in the process.

An introduction to the manufacturing of components

Once a component has been designed it can now be manufactured. When dealing with the manufacture of components there are many things that need to be considered. Not only has the method of manufacture to be considered but measurement considerations and process control must be accounted for. This chapter aims to give an overview for both the designer and the metrologist of the different machining processes. A list of some of the machining processes that you may come across is given in Table 1.

Table 1 Common manufacturing processes

Metal casting
Forming and shaping processes • rolling, • forging, • extrusion and drawing, • sheet metal forming, • sintering, • rapid prototyping.
Material removal processes • turning, • drilling, • milling, • grinding.
Joining processes • welding, • brazing, • bonding, • mechanical fastening.

When dealing with all manufacturing processes the following needs to be considered:

- Identify the requirements of the production process, whether low volume production, such as one-off component parts used in motor sport, medium volume production such as twenty parts in a batch for the motor industry or high volume production such as thousands of fasteners for the aircraft industry.
- Are measurements taken during the manufacturing process, such as the use of basic hand tools (for example, callipers, micrometers, plug gauges, *etc.*), or is in-line gauging used? In-line gauging can be measurements using hard gauges designed to verify specific dimensions or CMMs linked to a cell where the workpiece may be loaded by conveyor or robot. Figure 16 and Figure 17 show examples of on-machine gauging. Figure 18 shows equipment that might be used in hard gauging.



Figure 16 On-machine gauging (Renishaw)



Figure 17 On-machine gauging (Renishaw)



Figure 18 Gauging instrumentation (Tesa Technology)

Overview of basic manufacturing and machining processes

Stock removal techniques

There are numerous methods for removing material from stock in order to produce a component. The most common processes, along with a brief description, are listed below.

The lathe

A lathe is a machine tool that rotates the workpiece material so that when abrasive or cutting tools are applied to the workpiece, it can be shaped to produce an object that has rotational symmetry about the axis of rotation. Examples of objects that can be produced on a lathe include, table legs, crankshafts or camshafts. Figure 19 shows a typical lathe in use.

The lathe holds the stock material in a chuck and rotates the stock relative to a stationary tool. Translating the saddle and cross-slide that support the tool post in an axial and/or radial direction generates an interaction between the stock material and the tool that removes material as swarf. The lathe is generally suited to making cylindrical components. While the component can be parted off and reversed in the chuck to machine the other end, it is unwise to assign tight tolerances to the concentricity of holes bored from each end. Similarly the thickness of a flange may vary if it is machined in two set-ups.



Figure 19 A typical lathe

Three-axis milling machine

The three-axis milling machine utilises a rotating tool or cutter to remove material from the component. Various configurations of three-axis milling machine exist, but typically the spindle that carries the cutter is moved to provide the vertical axis, while the component is attached to a moving bed that provides the two horizontal axes of motion in mutually orthogonal directions. The component can be held in a vice, a chuck or by clamping it to the bed of the machine. The milling machine is suited to making a wide range of components but has limited access to the component at any given time during the process, although this can be improved by using a rotary table or inclinable bed to support the component. Figure 20 shows a typical three-axis milling machine.



Figure 20 A typical milling machine

Five-axis computer numerical control machine tool

In the case of a five-axis milling machine two orthogonal rotary axes are added to a three-axis milling machine and under computer numerical control these machines are capable of cutting complex three-dimensional shapes such as centrifugal compressor disks or propellers. Five-axis milling machines are capable of machining five faces of a component in one set-up. The component may be held in a vice, a chuck or clamped to a faceplate, or the face of the rotary table depending on the configuration of the particular machine tool.

Wire cutting

Wire cutting utilises an electro-chemical process to allow a thin conducting wire to cut its way through a conductive material while the component is mounted on a two-axis stage. The process is ideal for flat two-dimensional components with intricate outlines. Figure 21 shows a typical wire cutting machine. In Figure 22 you can see the cutting wire. There is a requirement for a pilot hole to feed the wire through at the start. More sophisticated versions of the machine offer a limited two-axis tilt facility in the head that allows the machining of tapers and cones. The process is capable of high accuracy machining and of producing thin sections with little danger of part failure under machining loads and deformations.



Figure 21 A typical wire cutting machine

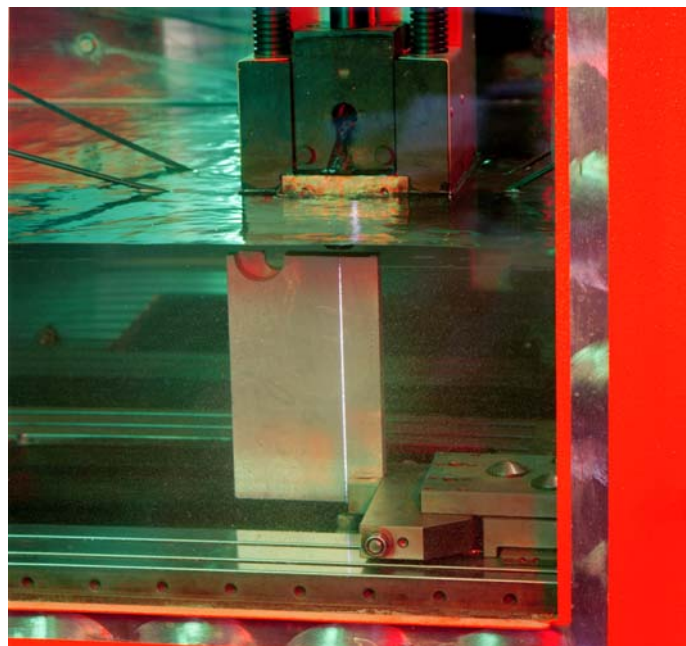


Figure 22 Wire cutting in action

Electro discharge machining (EDM)

EDM employs a similar principle to wire cutting, but uses a graphite or copper electrode instead of a wire. The electrode or die is slowly lowered into the material and excess material is electro-chemically eroded. It is usual to use a roughing die followed by a finishing die as the die itself is consumed – all be it at a slower rate than the component. This technique is very useful for manufacturing moulds and tooling for injecting moulding systems. For example, look at the detail in an Airfix™ plastic model kit.

Volume production processes

Casting

Die-casting techniques are suitable for medium to large production runs and allow the manufacture of components at near finished size, which minimises the machining required. What the designer can do to simplify manufacture is to incorporate clamping and fixturing features that may be temporary or permanent parts of the component. The designer can also incorporate features to facilitate the inspection process, for example, three small flats which can be probed to define a datum plane – if they are not functional features then they can be fully floated in a three dimensional best fitting routine during the inspection process (for example, Smartfit^{TM5}). Such software allows the inspector to determine if the required part can be machined out of the casting.

Monitoring the process

When dealing with any process it is important to link the information obtained from the process to continuous improvement. This can be done in many ways, and by many quality tools and techniques. These different tools and techniques can be utilised with many different types of measuring equipment ranging from basic hand tools, CMMs or purpose built gauges. In all cases it is important to understand what is really being monitored with these tools and techniques, *i.e.*, process variation.

This monitoring process is illustrated by considering the case of inspecting a batch of 500 components by picking 50 components from a box of 500 and measuring them. The statistics you derive from those measurements – such as the average size and the standard deviation of the sample will allow you to accept or reject the batch depending on the sampling plan that you have selected and the particular criteria used. However, sampling in this way will not have told you a great deal about the manufacturing process. If we now change the inspection process to one where we measure every tenth component off the production line, not only can we determine the average size and standard deviation, we can also determine trends. This trend monitoring is a powerful tool in the control of the process and allows the operator to make timely adjustments to the machine so that tolerances are not exceeded.

⁵ Trademark of Kotem Technologies Inc Hwww.kotem.comH



Figure 23 Inspecting a batch (Hexagon Metrology)

Trend monitoring and statistical control

The way that the parameters of a component vary will influence many decisions during the production process. As we are all aware variation exists in all walks of life. Variation is all around us ranging from the difference in the height of people, to the different types, colours and shapes of motor vehicles. So when dealing with variation in manufacturing it is important to monitor the way in which each individual products and processes vary, for example:

Why do dimensions vary on components?

What are the causes of variation within the methods of manufacture?

The use of statistical techniques to monitor the variation can help considerably to reduce the variation within the process and link the design, manufacture and measurement aspects of the process.

These monitoring processes use terminology such as mean, range, standard deviation, histogram, control chart, capability and Six Sigma (see Appendix B.2). All of these terms are different descriptions of tools and techniques used to monitor the process.

Statistical Process Control (SPC) is the term used to cover the application of statistics to the control of industrial processes. In its simplest form this may involve measuring the size of every tenth item off the production line and measuring and recording the dimension on a graph that has the upper and lower tolerances marked on it. By taking note of the trends displayed on the graph (Figure 24) it is then possible to predict when the process is going to produce components with dimensions that exceed the permitted tolerances, and take corrective actions such as adjusting the tool setting.

Process control and measurement feedback

When measurements are taken at either the manufacturing stage or the inspection stage trend monitoring and SPC can take place. Measuring equipment, such as micrometers, callipers, in-line gauging or even a CMM, can be connected to computers that can be networked together. The option is then available to use software that can monitor the measurement results and statistical data obtained. The quality tools and techniques described in Appendix B.2 can be used to link the data back to the process. The use of both resultant values and calculations, combined with graphs, are very important to create an environment of prevention.

The data obtained from the measuring equipment and SPC can be fed back into the production process to allow for the relevant adjustments to be made to the machines, thus giving the opportunity to implement a prevention strategy.

Figure 24 shows a basic SPC control chart that plots the process variation with time. The basic aim of SPC is to minimise variation.

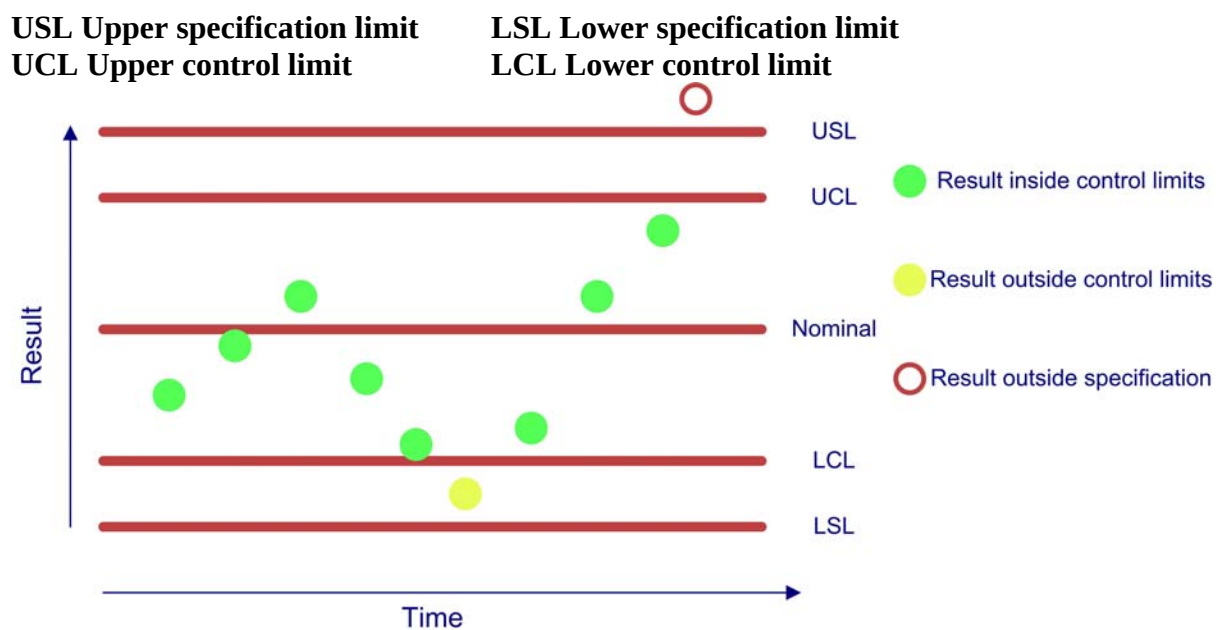


Figure 24 Basic control chart layout

Drawings

4

IN THIS CHAPTER

- A simple overview of what information is being conveyed – size, shape, surface finish, *etc.*
- Co-ordinate systems – Cartesian, cylindrical, spherical, local versus global.
- Dimensioning, establishing the datum, can the datum chosen be easily measured, virtual datums.
- Why is a partial arc a bad datum? Size of the datum feature.
- Does the tolerance really need to be that tight?
- Least squares or minimum zone?

The engineering drawing is the most common method for communicating the designer's thoughts to the engineer and the metrologist. This chapter aims to give a brief introduction to the drawing language.

A simple overview of what information is being conveyed, for example, size, position, shape and surface texture

Introduction

Once manufactured a real component can vary in many ways from the specified design. The size can vary in many ways, for example, bore diameters, overall length, the shape can vary, for example, cylinders may be barrel shaped; the surface texture may not be perfectly smooth. Form deviation arises from generally repeatable aspects of the manufacturing-machine performance such as machine tool elasticity and workpiece fixturing. These variations are acceptable as long as they fall within the specified limits. The purpose of the drawing is to convey how much variation is allowable.

Co-ordinate systems

Firstly we will briefly introduce the co-ordinate systems you may come across on a drawing. These can be Cartesian, cylindrical or spherical and each can be global or local.

Co-ordinate systems – Cartesian, cylindrical, spherical

A co-ordinate system is most commonly a conventional Cartesian system with three orthogonal axes (*i.e.* at mutual right angles to each other) conventionally labelled x , y and z (as shown in Figure 25).

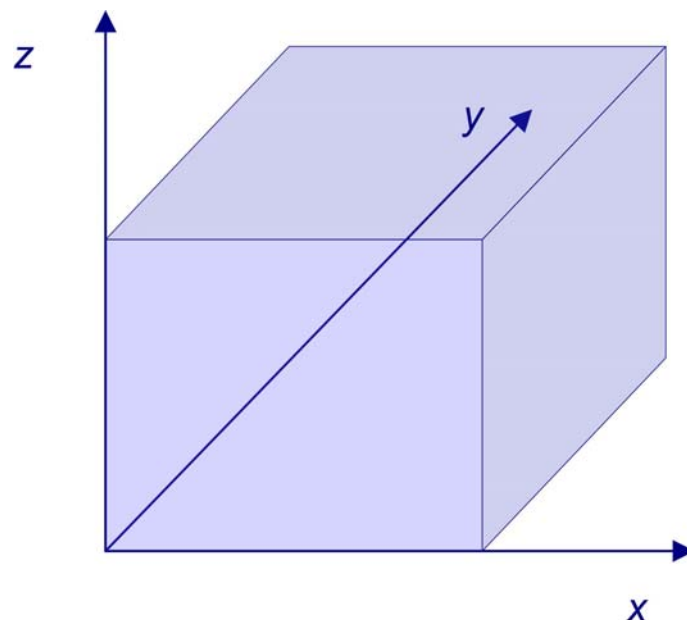


Figure 25 Cartesian co-ordinate system

In some situations, a cylindrical co-ordinate system (with a radial distance, an angle and a height) would make sense and in other rare situations a spherical co-ordinate system (with a radial distance and two angles) could be used.

Cartesian co-ordinates, x , y , and z , can be expressed in spherical co-ordinates. The azimuth and elevation are angular displacements measured from the positive x -axis, and the x - y plane, respectively; and R is the distance from the origin to a point.

Global versus local

Co-ordinate systems can be either global or local. A *global* co-ordinate system can be thought of as an absolute reference frame.

In many cases, it may be necessary to establish your own co-ordinate system, whose origin is offset from the global origin, or whose orientation differs from that of the predefined global systems. Such user defined co-ordinate systems are known as *local* co-ordinate systems.

Both these co-ordinate systems can be Cartesian, cylindrical or spherical.

As an example, if a drawing describes an aircraft, the global co-ordinate system may be based on say the aircraft nose cone. However, individual components may have local co-ordinate systems.

General tolerances

Tolerance in engineering is an allowance made for imperfections in a manufactured object. A specification might call for a cylinder with a nominal diameter of 100 mm, but will also state a tolerance such as ± 0.1 mm. This means that any diameter with a value in the range 99.9 mm to 100.1 mm is acceptable. It would not be reasonable to specify a diameter with a value of exactly 100 mm, because such a cylinder cannot be made.

Most critical features on a drawing will be individually toleranced. However, other features will not have either an adjacent tolerance or a geometric tolerance indicated, these are normally expressed as a general tolerance. The value of this general tolerance is normally indicated on a specific area of the drawing. For example, it could be stated as ± 0.250 mm unless otherwise stated for a linear dimension or $\pm 1^\circ$ for angular dimensions. To be specific about the functional dimensions, the use of geometric tolerances becomes the next step.

Geometric tolerancing

The purpose of geometric tolerancing is to describe the geometry of products and their relationship between various functional parts or assemblies. Geometric tolerancing is used in conjunction with conventional drawing practices.

A universal language of symbols exists for geometrical tolerancing, much like the international system of road signs that advise drivers how to navigate the roads. Geometrical tolerancing symbols allow a design engineer to precisely and logically describe part features in a way they can be accurately manufactured and inspected.

The greatest impact on improvements to a process through the use of geometric tolerancing can be shown in the quality, cost and delivery of your product. Developments within the CAD and CMM worlds, in standards, quality tools and techniques have put the onus on GPS to provide a more generic link in the design, quality and manufacturing processes.

The use of geometric tolerancing on CAD designs and engineering drawings can bring benefits in many ways, for example:

- Datums and datum reference systems are used to define the design requirements in relation to the component dimensions and their subsequent mating parts.
- Universally accepted symbols and terms, to reduce confusion.
- The dimensions and related tolerances are based on their functionality.
- Dimensional tolerance methods that decrease tolerance 'stack up'.
- Provides information that can be used to control tooling and assembly interfaces.

When utilising geometrical tolerancing and dealing with the component's real geometrical surface you can categorise the deviations from the nominal shape, orientation and location into either a *single* or *related to a datum* feature geometric tolerance type (Figure 26). The single classification relates to form tolerances that are not normally related to a datum, for example, straightness. It is possible to look at the profile tolerance types as form tolerances as they do not always relate to a datum.

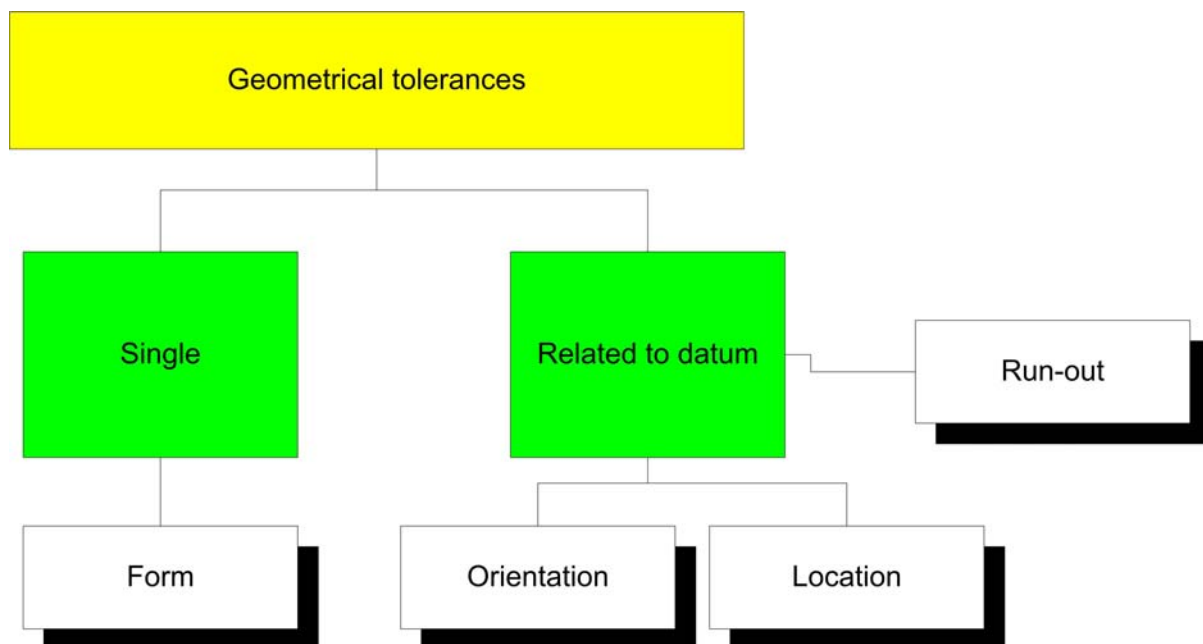


Figure 26 Classification of geometrical tolerances

The following section gives an overview of the fundamental requirements of geometrical tolerancing. Figure 27 gives an overview in graphic form.

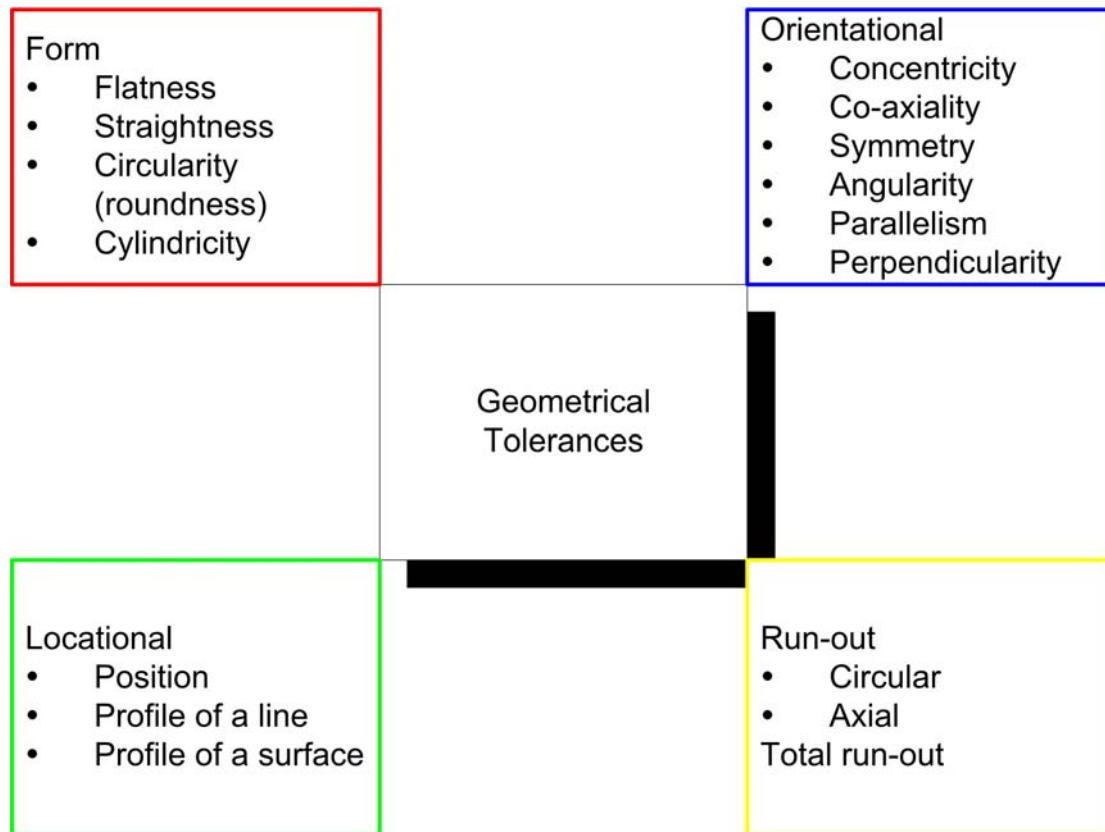


Figure 27 Overview of geometrical tolerances

The utilisation of geometrical tolerancing on drawings is by:

- Geometric references (datum features).
- Feature control frames.
- Geometric characteristics (symbols).
- Tolerance shapes.
- Tolerance zones (values).

Each of these points in the list above will now be explained in more detail.

Geometric references (datum features)

The datum features can be identified from the component drawing or CAD model. The datum features are normally expressed on the technical drawing in a filled or an open triangle. The identification contained within the box is normally shown as a capital letter as shown in Figure 28. The letter will occur in the feature control frame (Figure 33) of tolerances related to this datum. The feature control frame should identify all the datum features required. When measurement takes place it is important to be able to create a co-ordinate system related to the component. The terminology used to identify the datum features in geometrical tolerancing is:

1. *The primary datum* - this is defined as a feature or features used for the levelling of the component normally on a surface or an axis.

2. *The secondary datum* - this is defined as a feature or features used for the rotation of the component part relative to the primary datum.
3. *The tertiary datum* - this is defined as a feature or features used to complete the co-ordinate system in relation to the primary and secondary datums.

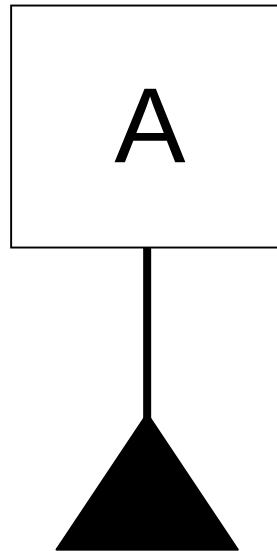


Figure 28 Datum triangles and datum letters

The datums will be positioned on the technical drawing differently depending on the specific requirements and functionality of the feature or features. It is important to identify these requirements so as not to make fundamental errors when manufacturing or measuring the component. For example, care must be taken as to whether an axis, surface/feature extension, or target datum is the required reference for the co-ordinate system. In Figure 29 the datum is the axis of the component while in Figure 30 the datum is a feature extension.

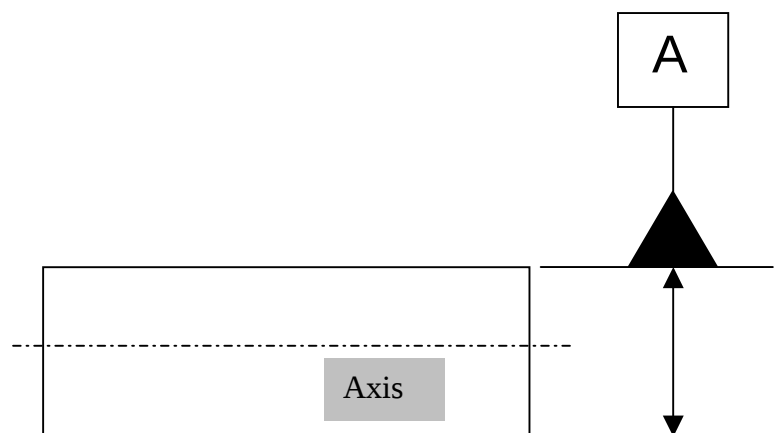


Figure 29 Datum axis

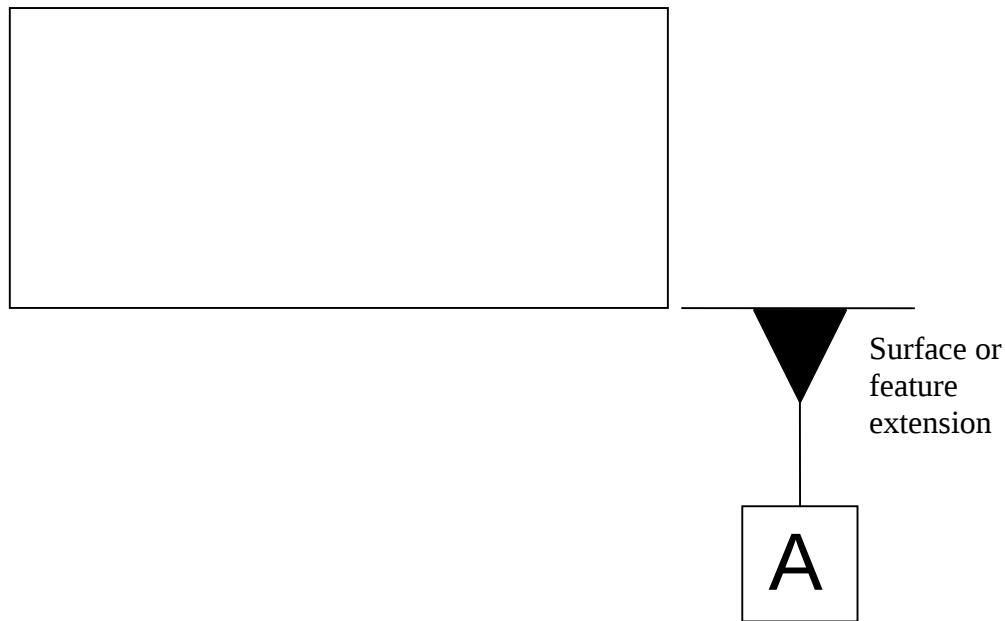


Figure 30 Datum surface or feature extension of surface

Where a component is rough machined, datum target points are used to establish a datum. Datum target frames are used to define the points, lines, or areas where the manufacturing locations or measurement points should be defined from, to create the co-ordinate system. Figure 31 shows an example of a datum target frame. This example shows the location zone for manufacture or measurement to be anywhere within the circular area of a diameter of 8 mm with a theoretical centre being at 15 mm by 15 mm from the corner. A1 is the reference name (datum feature and datum target number). The target is an area so it is hatched.

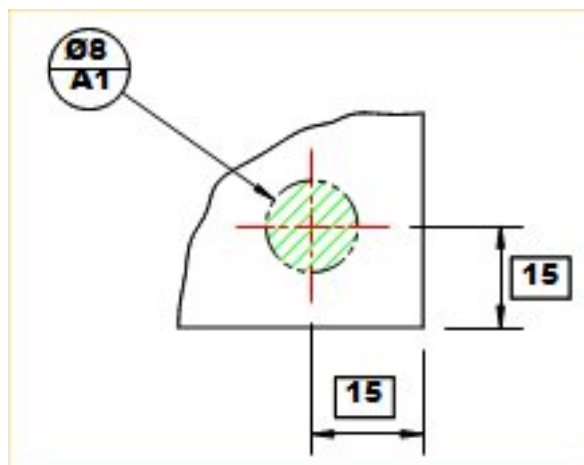


Figure 31 Datum target frames

Feature control frames

The feature control frame (or tolerance frame) is a rectangular frame divided in to a series of compartments that contain various pieces of information regarding the technical requirements

of the dimensions to be manufactured and measured (Figure 33). The feature control frame contains the tolerated characteristic symbol, shape of the tolerance zone, limits of production variability and datum references where applicable (Figure 32).

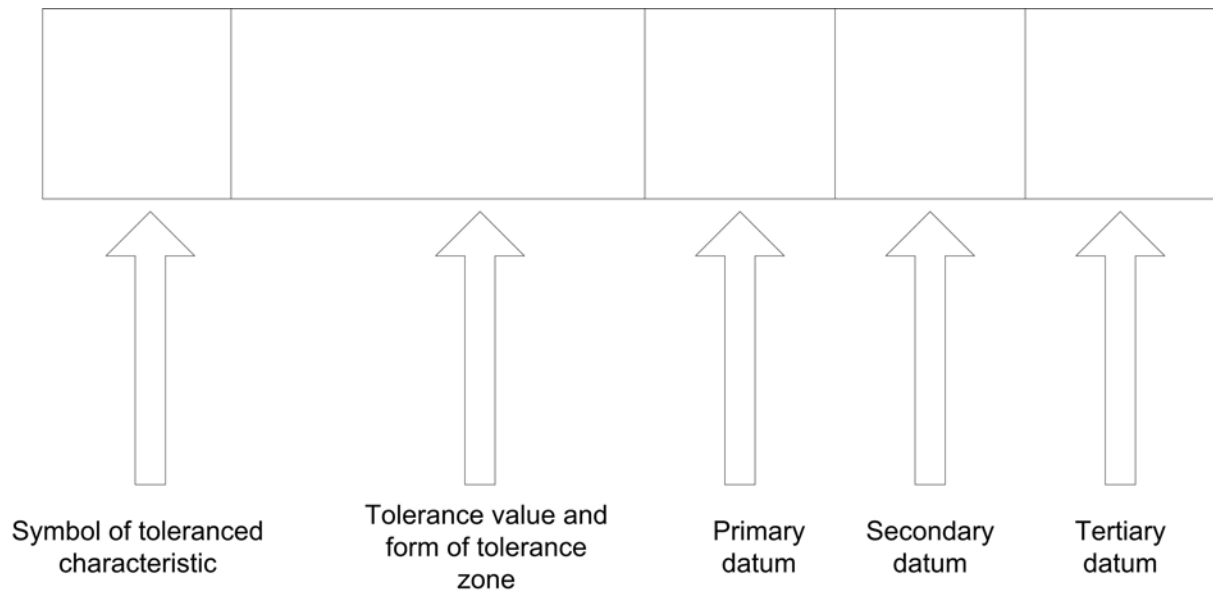


Figure 32 A feature control (tolerance) frame and the information it can contain

The number of compartments in the feature control frame can vary. This is dependent on the characteristic type used, whether single or related and what the functional requirements are. For example, for a single tolerance type the feature control frame can be as shown in Figure 33.



Figure 33 A simple feature control frame

Geometric characteristics

The first compartment of the feature control frame contains the symbol of the tolerated characteristic. The geometric characteristics are represented by a series of symbols that can be used to describe the design requirements of the feature or features to be manufactured or measured. We will now go through a number of these in turn and give an indication of how they could be measured.

NOTE



CMM measurement of geometry

When using a CMM to measure geometric form, the density of points used should reflect the typical form errors expected from the machining process.

Form tolerances

Circularity (roundness)

Figure 34 shows the symbol used to indicate roundness. Two concentric circles bound the tolerance zone specified (Figure 35). In Figure 34 the profile must lie between two concentric circles 0.038 mm apart. During inspection measurements could be obtained by the use of a roundness-measuring machine (Figure 12) or by the use of a CMM. A roundness-measuring machine will in general rotate the component while a transducer in contact with the surface records the deviations of the surface from a perfect circle. A CMM will contact at many points around the circumference of the surface and then fit a circle to the data to allow roundness to be assessed. Both these measurement techniques make use of filters and it is important that all in the production process should agree the choice of filter. The density of points used should reflect the typical form errors expected from the machining process.

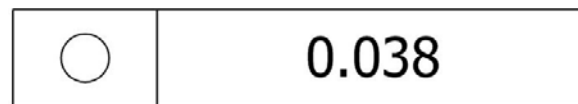


Figure 34 Circularity (roundness) symbol

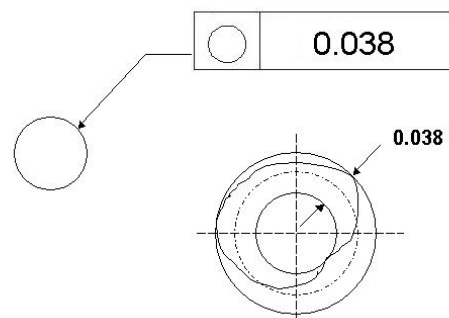


Figure 35 Circularity (roundness) definition

Straightness

The symbol for straightness is shown in Figure 36. A straightness tolerance specifies a tolerance zone bounded by two parallel lines (Figure 37). During inspection measurements could be obtained by the use of a dial indicator in conjunction with other basic measuring tools. Alternatively a line could be probed on the surface using a CMM.

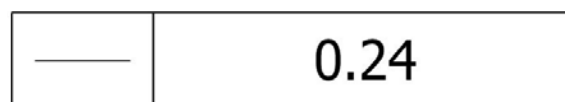


Figure 36 Straightness symbol

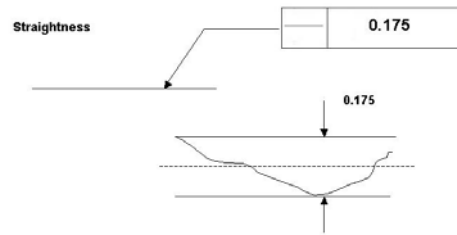


Figure 37 Straightness definition

Flatness

The symbol for flatness is shown in Figure 38. A flatness tolerance specifies a tolerance zone bounded by two parallel planes (Figure 39). During inspection measurement results could be obtained by the use of flatness measuring equipment such as optical flats, precision levels or a dial indicator in conjunction with other basic measuring tools. Alternatively the surface could be probed with a pattern of points using a CMM.

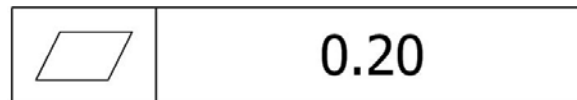


Figure 38 Flatness symbol

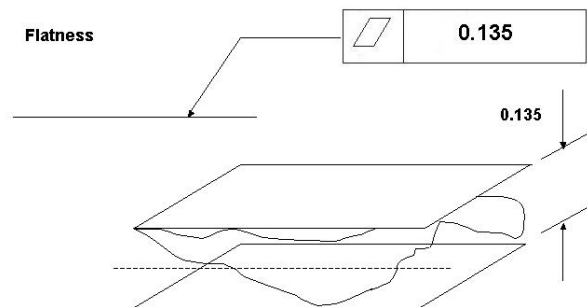


Figure 39 Flatness definition

Cylindricity

The symbol for cylindricity is shown in Figure 40. A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie (Figure 41). During inspection measurements could be obtained by the use of a roundness-measuring machine. Alternatively the surface could be probed with a pattern of points using a CMM.



Figure 40 Cylindricity symbol

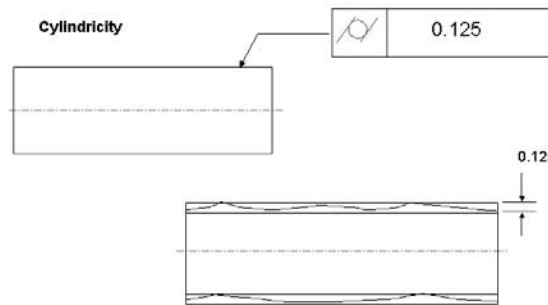
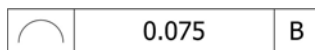


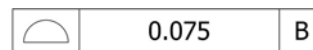
Figure 41 Cylindricity definition

Profile

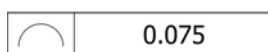
The profile tolerance is a method of controlling irregular surfaces, lines, arcs or normal planes (Figure 42). Profiles can be applied to individual line elements or the entire surface of a part. The profile tolerance specifies a uniform boundary along the true profile within which the elements of the surface must lie. Measurement results could be obtained by the use of a dial indicator and master artefact in conjunction with other basic measuring tools, a profile projector, contour-measuring instrument, various non-contact measuring devices or by the use of a CMM.



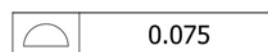
Profile of a line - related tolerance



Profile of a surface - related tolerance



Profile of a line - single tolerance



Profile of a surface - single tolerance

Figure 42 Profile symbols

The profile tolerance can be bi-lateral or unilateral depending on the position of the leader lines in relation to the nominal and the tolerance zone (see Figure 43).

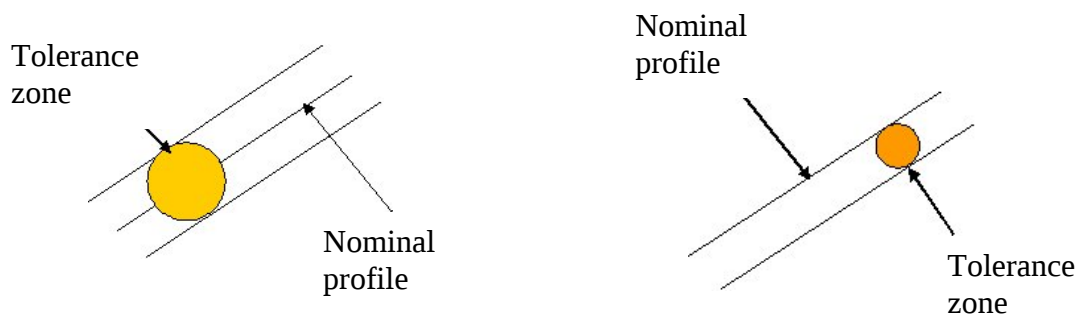


Figure 43 Profile tolerance zones bilateral and uni-lateral

The left hand image in Figure 43 shows how the tolerance zone for a line profile is always defined between two lines which envelope a set of imaginary circles of diameter, t , whose centres lie on the nominal profile.

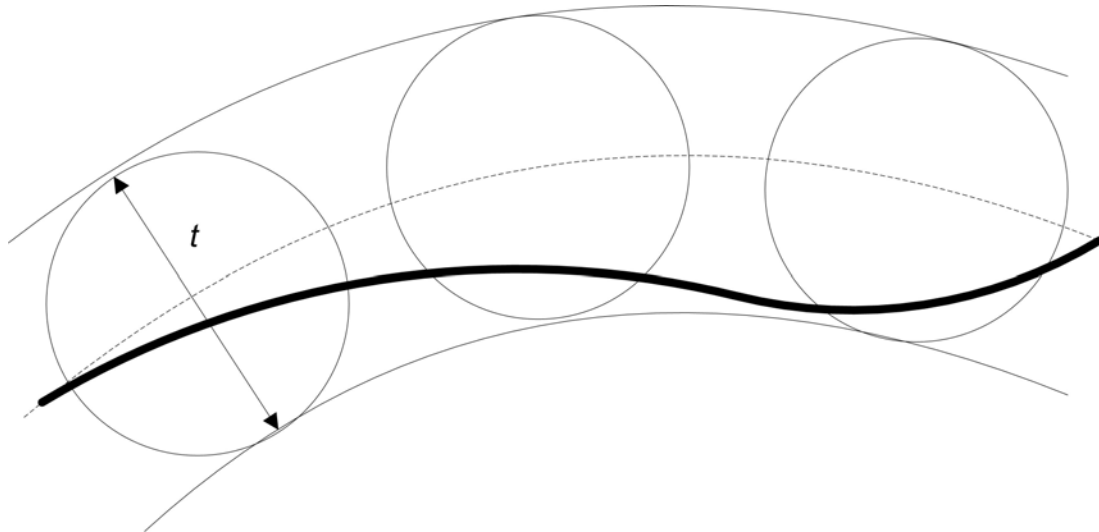


Figure 44 Line profile tolerance

In Figure 44 the heavy line (real profile) lies between circles of the prescribed tolerance placed on the geometrically exact form (dashed line).

Tolerances of location

Tolerances of location indicate the permissible variation in the location of a feature in relation to another feature or datum (location, symmetry and concentricity). Basic dimensions establish the true position from datum features and between interrelated features. Figure 45 shows a drawing of a component with a hole dimensioned from datum B and C.

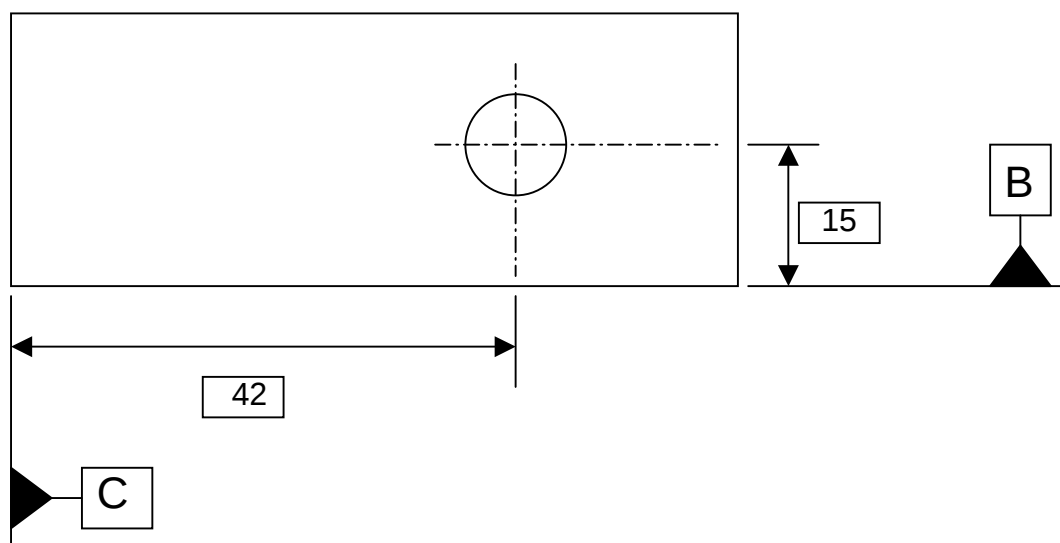


Figure 45 Position tolerances

A positional tolerance is the total permissible variation in location of a feature about its theoretically exact location with respect to the datum. For cylindrical features, such as holes and outside diameters, the positional tolerance is generally the diameter of the tolerance zone in which the axis of the feature must lie.

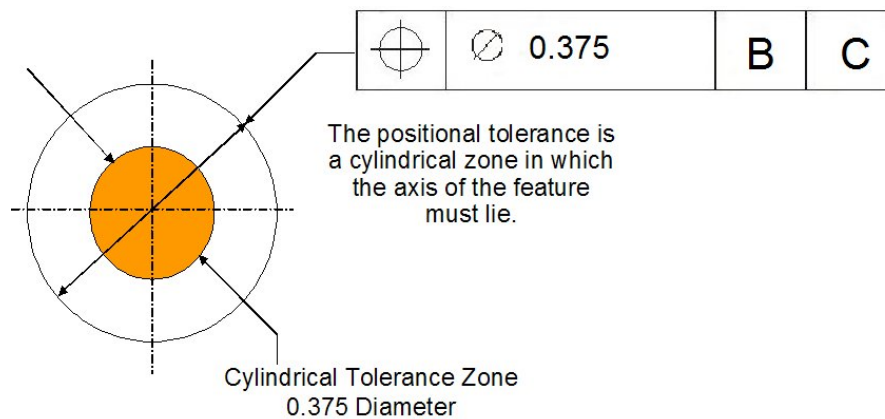


Figure 46 Cylindrical tolerance zone

In Figure 46 the tolerance zone is defined by a cylinder of diameter 0.375 mm. The axis of the cylindrical tolerance zone is set relative to datum B and datum C.

Maximum material condition

The maximum material condition (MMC) principle describes the condition where a geometric characteristic has the most amount of material. It is relevant when the assembly of two components is important. If the MMC principle is applied to an internal feature then this is defined as the smallest allowable size, if applied to an external feature then this is defined as the largest allowable size.

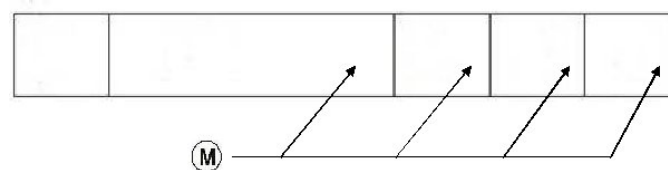


Figure 47 The MMC symbol placement

The MMC symbol can be positioned within the tolerance frame as shown in Figure 47 either next to the required tolerance or any applied datum symbol.

MMC is a principle used when the assembly of component parts is important and provides the combination of the smallest hole and the largest pin to fit together as an assembly. After this principle has been established it is known as the virtual condition – it defines the worst condition for the hole and pin to assemble.

In Figure 48 the top image shows the minimum metal condition, *i.e.*, the male feature at its smallest size together with the female feature at its maximum size. The bottom image shows the male feature at its largest size together with the female feature at its minimum size.

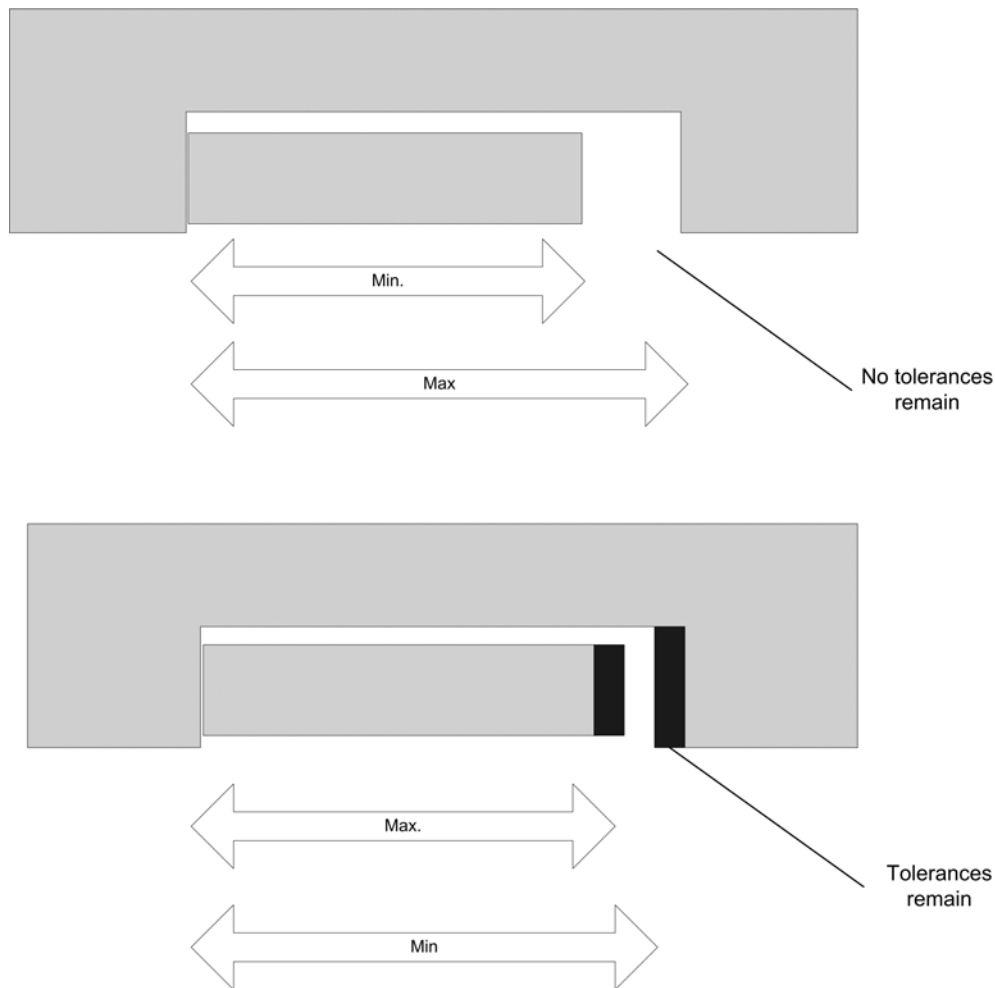


Figure 48 The concept of maximum material condition

When the two features are at their virtual condition they would just fit together and any deviation from this could result in both features going together more easily. It is important to remember that this principle is dependent on the features remaining within the tolerance of size.

The virtual condition is generated by combining the MMC condition and the geometric tolerance. If applied to an internal feature the virtual condition would be the MMC condition value minus the geometric tolerance. Alternatively if applied to an external feature the virtual condition would be the sum of the MMC condition value and the geometric tolerance. See *BS ISO 2692:1988 Technical drawings. Geometrical tolerancing. Maximum material principle* for more information. Figure 49 shows some example frames that use the MMC symbol.

	Maximum material condition applied to the measured feature
	Maximum material condition applied to the datum feature
	Maximum material condition applied to both the measured feature and the datum feature

Figure 49 MMC example frames

Concentricity/co-axiality

The symbol for concentricity and co-axiality is shown in Figure 50. The concentricity/co-axiality tolerance zone is related to either the centre point of two features (concentricity) or the centre axes of two features (co-axiality) such as circles or cylinders (Figure 51). During inspection measurement results could be obtained by the use of a roundness-measuring machine or by the use of the CMM. For concentricity two circles would be measured. For co-axiality one cylinder surface is measured in relation to another cylindrical surface's axis.



Figure 50 Concentricity/co-axiality symbol

In the example feature control frame (Figure 50) the actual axis of the tested feature shall be within a cylindrical zone of diameter 0.3 mm, the axis of which is datum feature B.

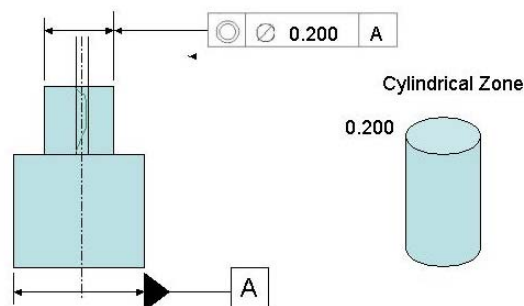


Figure 51 Concentricity definition

Symmetry

The symbol for symmetry is shown in Figure 52. For features that are not of cylindrical form, such as slots, the positional tolerance is the total width of the tolerance zone in which the centre plane of the feature must lie. During inspection measurement results could be obtained by the use of a dial indicator in conjunction with other basic measuring tools or by the use of a CMM.

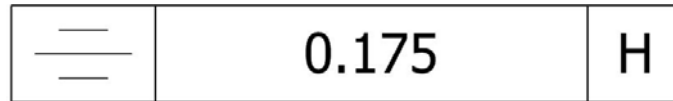


Figure 52 Symmetry feature control frame

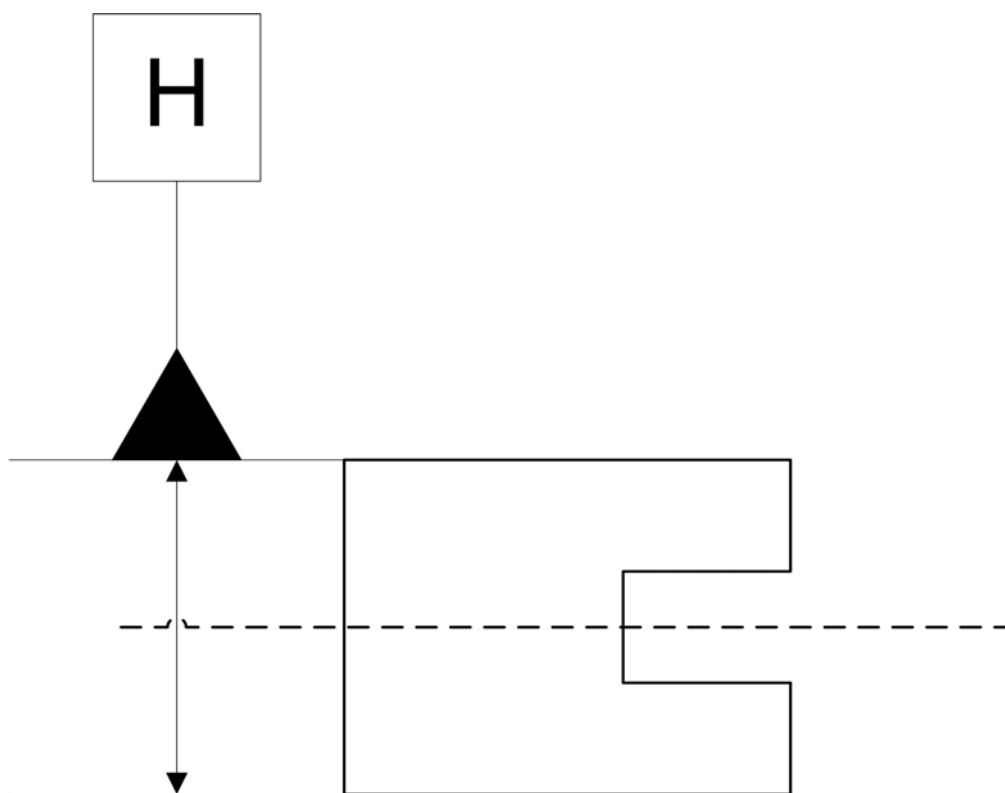


Figure 53 Symmetry of a slot

Applying the symmetry control frame shown in Figure 52 to the slot in Figure 53, the tolerance zone is limited by two parallel planes a distance 0.175 mm apart symmetrically disposed about the median plane (dotted line). When measured the derived median surface of the slot shall be contained within two parallel planes 0.175 mm apart, which are symmetrically disposed about the datum median plane, H.

Tolerances of orientation

Parallelism

The symbol for parallelism is shown in Figure 54. The parallelism tolerance zone can be defined by using two planes or two lines parallel to a datum plane or axis (Figure 55). During inspection measurement results could be obtained by the use of a dial indicator in conjunction with other basic measuring tools or by the use of a CMM. Remember that the parallelism of surface A to datum B is not the same as the parallelism of surface B to datum A.

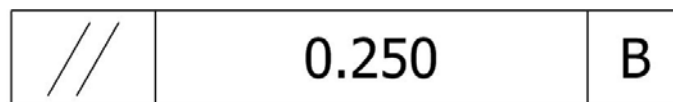


Figure 54 Parallelism symbol

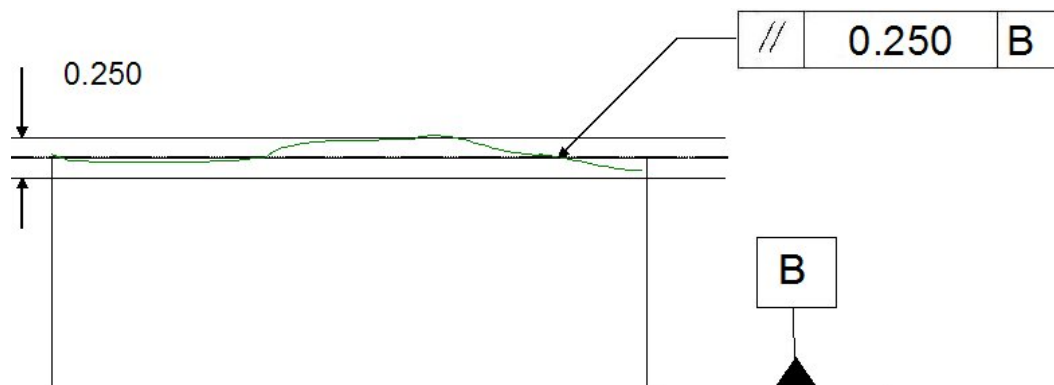


Figure 55 Example of a parallelism tolerance

Note that the tolerance zone can be specified as cylindrical when applied to an axis (Figure 56).

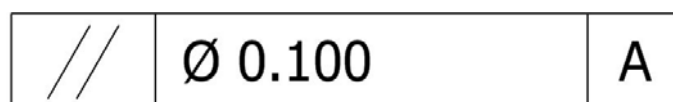


Figure 56 Parallelism with cylindrical tolerance zone

Perpendicularity

The symbol for perpendicularity is shown in Figure 57. The perpendicularity tolerance zone can be defined by using two planes or two lines perpendicular to a datum plane or axis (as shown in Figure 58 and Figure 59). During inspection measurement results could be obtained by the use of a dial indicator in conjunction with other basic measuring tools, a roundness-

measuring instrument or by the use of a CMM. Remember the perpendicularity of surface A to datum B is not the same as the perpendicularity of surface B to datum A.

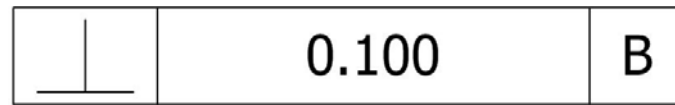


Figure 57 Perpendicularity symbol

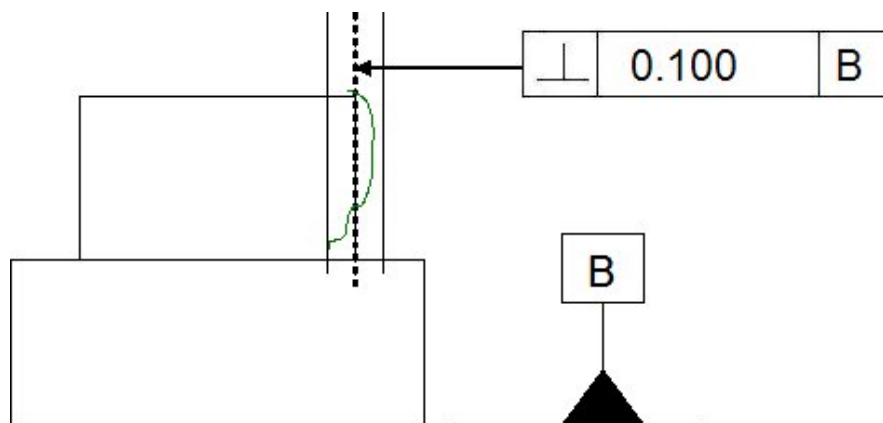


Figure 58 Example of a perpendicularity tolerance

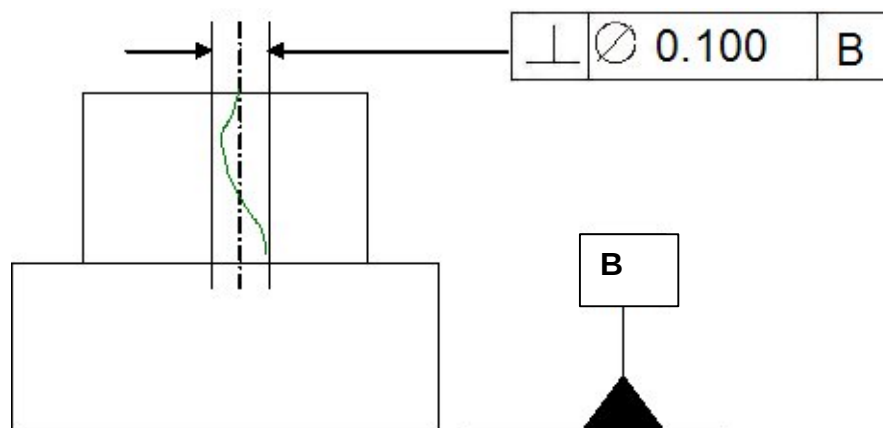


Figure 59 Example of a perpendicularity tolerance

Note that the tolerance zone can be specified as cylindrical when applied to an axis (Figure 60).

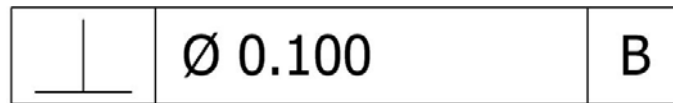


Figure 60 Perpendicularity with a cylindrical tolerance zone

Angularity

Figure 61 shows the symbol for angularity. Angularity is the condition of a surface or axis at a specified angle (other than 90°) from a datum plane or axis (datum B in this case). Two parallel planes at the specified basic angle relative to a datum plane or axis define the tolerance zone (Figure 62). During inspection measurement results could be obtained by the use of a dial indicator and sine bar in conjunction with other basic measuring tools or by the use of a CMM.

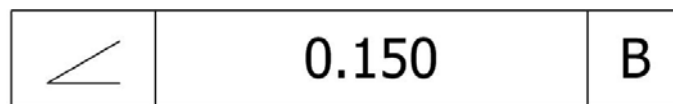


Figure 61 Angularity symbol

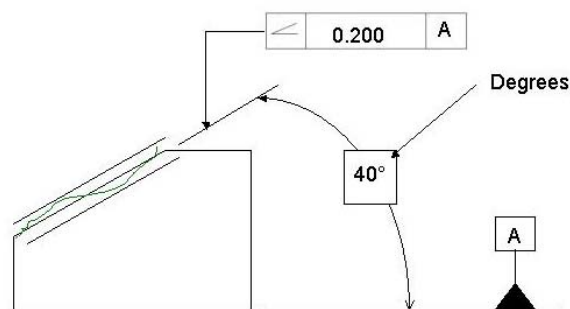


Figure 62 Angularity definition

Tolerances of run-out

Run-out

The total run-out symbol is shown in Figure 63. The run-out tolerance provides control of the circular characteristics of a surface. This type of tolerance is applied independently at any circular measuring position on both the diameter or surface areas, as the part is rotated through 360°. During inspection, measurement results could be obtained by the use of a dial indicator and vee-block in conjunction with other basic measuring tools, a roundness-measuring machine or by the use of a CMM. As shown in Figure 64, run-out can be either circular or axial.

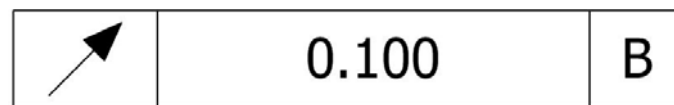


Figure 63 Run-out symbol

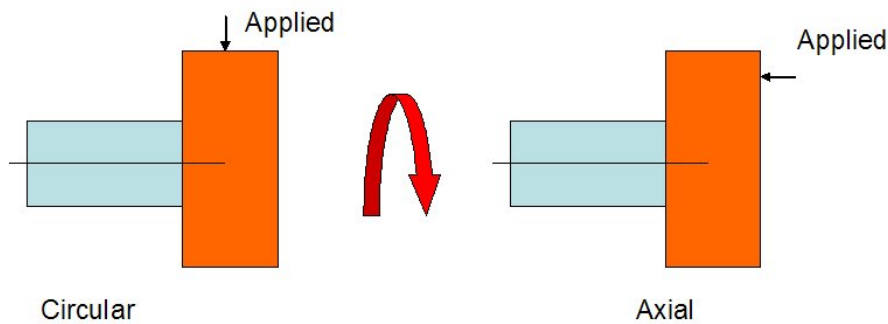


Figure 64 Run-out can be in the circular or axial direction

Total run-out

The total run-out symbol is shown in Figure 65.

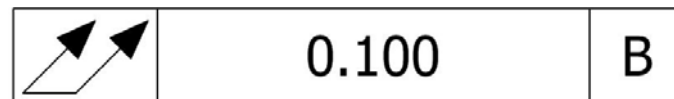


Figure 65 Total run-out symbol

For run-out different sections of the tolerated surface are independent. This is not the case for total run-out where the zone positions for different sections are strictly related, *i.e.*, all have the same zero point. Total run-out is applied in a similar way to run-out, it becomes composite by controlling more than one geometry. As an example, it can combine circularity, cylindricity, and co-axiality if applied to the outside diameter in Figure 64.

Surface texture

It is important to be aware that, not only can all the tolerances discussed so far effect the functionality of the component part, but that the surface texture of each geometric characteristic can cause dramatic consequences to the overall performance of a component part and its relevant assemblies.

Is the component part too smooth or too rough to function, do we know the parameters required to achieve this performance? There are many different types of parameters and specifications related to surface texture (for example Ra , Wa , Rq , Rsk) and to discuss them is beyond the scope of this guide. The general symbol used to identify surface texture is shown in Figure 66.

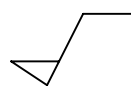


Figure 66 Surface texture symbol

In order to ensure that the surface texture requirement is expressed without ambiguity further information is added to the graphical symbol. It is important to have a good knowledge of the surface texture parameters and their requirements. Identification on the drawing of the evaluation parameters is critical. It will be almost certain that to measure surface texture some sort of surface texture machine will be used, where the measurement is taken using a stylus as shown in Figure 67 and Figure 68. Alternatively an optical surface texture measuring instrument could be used.



Figure 67 Measuring surface texture



Figure 68 A surface texture measuring instrument

For further information of the measurement of surface texture, parameter definitions, filters and uncertainties see Leach R K 2001 *The measurement of surface texture using stylus instruments* Measurement good practice guide No. 37 (NPL).

Establishing a co-ordinate system (datum) for a component

Earlier in this chapter the drawing datum and local and global co-ordinate systems were discussed. This short section will explain how the datum is used when measuring a component.

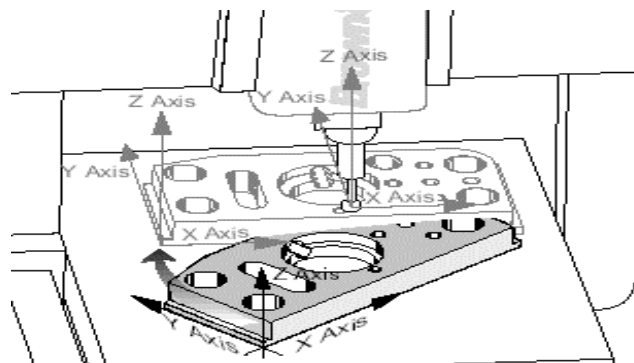


Figure 69 Aligning a component - exaggerated (Hexagon Metrology)

NOTE

In mechanical engineering, degrees of freedom (DOF) describe flexibility of motion. A component that has complete freedom of motion has six degrees of freedom.

Three degrees of freedom are translation - the ability to move in each of three dimensions (conventionally x , y and z).

The other three degrees are rotation, around the three perpendicular axes (pitch, roll and yaw).

It is important to ensure that the personnel carrying out manufacture and measurement accurately define the co-ordinate system used. The software used to assist in creating a co-ordinate system can vary between different types of machine tools and different types of co-ordinate measuring machines, but the fundamental methods for setting up the different types of co-ordinate systems do not change.

Why is alignment important?

If you were asked to measure the length of a wall with a tape measure, you would, without thinking align the tape roughly parallel with the floor and measure from one end of the wall to the other. You would not dream of holding the tape in the top corner and measuring to the opposite bottom corner. Without realising it, you had performed a simple alignment by measuring parallel to the floor.

A good co-ordinate system is as critical in accurate measurement of your parts as an accurate machine and a qualified probing system. The understanding of a co-ordinate system is critical to successful measurement. Just as a part must be properly set up in a tool room inspection area before it can be checked out, it must also be correctly aligned by the CMM operator.

What is probe qualification?

The probing system on the CMM will need to be qualified using a reference sphere or other known artefact since all probings must be corrected by the stylus tip diameter. Qualification involves determining the diameters of the stylus tips and the distances between them. This effectively creates an imaginary probing sphere of zero dimension that can access any point. Note that the effective diameter of the stylus sphere is always smaller than its actual size as it takes into account bending of the shank.

There are three steps to an alignment and it is important that the steps are carried out in the correct order:

1. Level to the part
2. Rotate to the axis
3. Set the origins

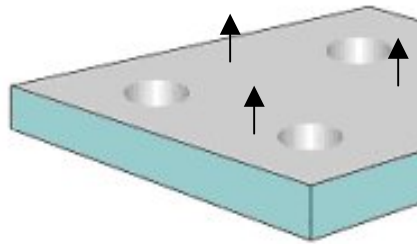


Figure 70 Defining the axis of rotation (Hexagon Metrology)

The first step in any alignment is to level to the part. This step is intended to ensure that the co-ordinate system for the part is perpendicular to the part and not to the axes of the machine (as shown in Figure 69).

For most parts levelling to the part involves contacting the top surface at three or more points (Figure 70) and fitting a surface. This surface is used to define an axis normal to the surface which is mathematically aligned with the z-axis. This axis is then used as a rotation axis for further alignment.

It is wrong to assume that the part is sitting flat or perpendicular to the vertical axis of the machine, there could be dirt under the part or the face that the part is sitting on is not parallel to the datum face.

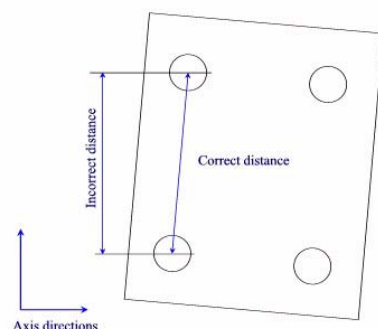


Figure 71 The affect of mis-alignment (Hexagon Metrology)

Now that the part is levelled, the next step of the alignment can be performed, which is rotate to the axis (Figure 71). It may be necessary to physically align the part to an axis of the machine, or alternatively if software is being used the software will mathematically adjust the

part rather than doing it physically. Once again the drawing will normally identify the alignment features.

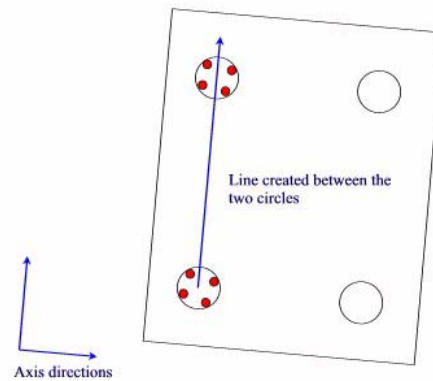


Figure 72 Creating a line to rotate about the z-axis (Hexagon Metrology)

In this example (Figure 72) two of the holes were measured and a line created through their centres. In the alignment software, the line is then rotated so that there is no longer an offset between the holes. This has the effect of rotating the CMM axes (not physically) so that the direction is “relative” to our part alignment.

Setting the origins

The origin is once again detailed on the drawing, it is a position or feature which is at $x = 0$, $y = 0$ and $z = 0$ (the bore in Figure 73). Sometimes it is necessary to use offsets from known features, but alignment software takes care of this.

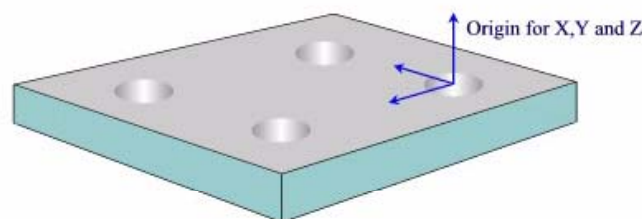


Figure 73 Setting the origin (Hexagon Metrology)

The result is a final co-ordinate system that is aligned to the part and has the origin in the correct position. This type of alignment is often referred to as a 3-2-1 alignment (Figure 74) and it is generally applicable to parts that are cuboid in nature. Aligning a cylindrical shape requires the axis to be defined and then the origin.

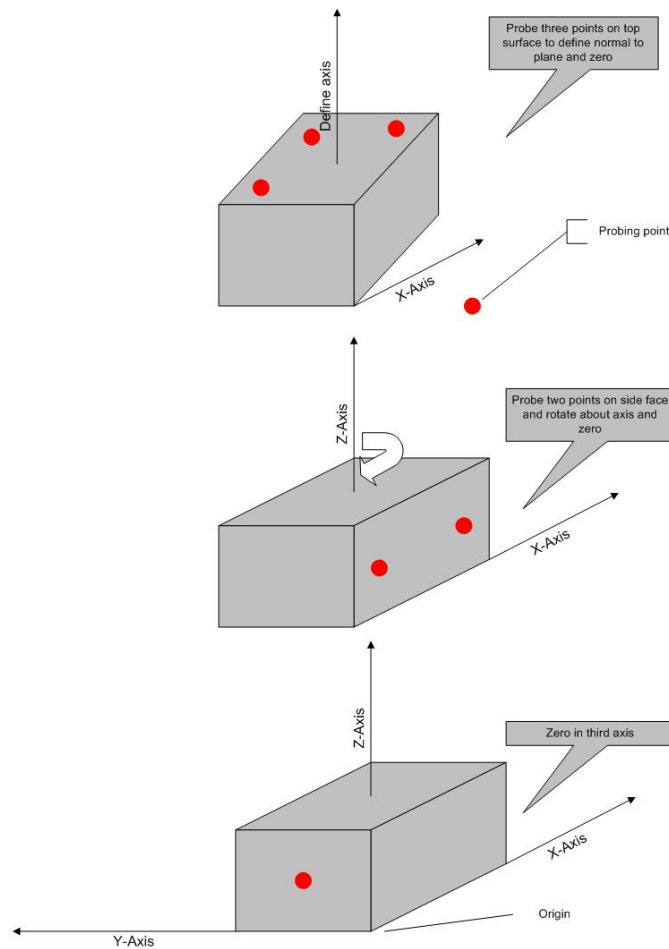


Figure 74 A 3-2-1 alignment

Can the datum be easily measured?

When designing a component the designer should make sure that the datum surfaces can be easily measured or fixtured. Datum surfaces should be substantial in size compared with the feature being measured. If datum features are small CMM measurement errors can play a larger part than if a larger datum was chosen. The important ratio is that between the size of the datum feature and the longest distance being measured.

As an example of the above, if the length of a 5 mm diameter, 300 mm long bar is being measured it would be inappropriate to use the 5 mm face as a datum feature for alignment. It would be better to use the axis of the bar. Small measurement errors have less effect over 300 mm than they would over 5 mm.

The virtual datum

When measuring components you may be faced with engineering drawings where the datum cannot be measured. That is the datum is not actually on the component. For instance a component may have an internal radius cut from one corner. The centre of this circle may lie outside of the boundaries of the component. This will make accurate measurement difficult and designers should avoid such situations if possible.

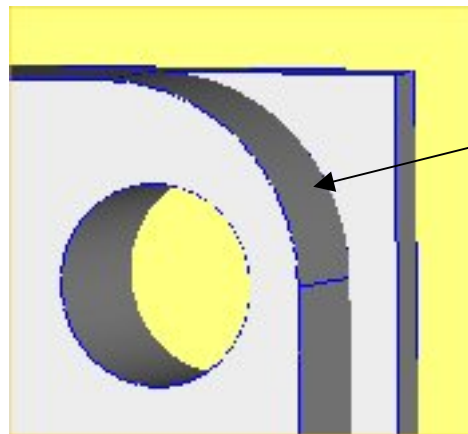
Why is a partial arc a bad datum?

In general a partial feature is one that constitutes a fraction of a complete feature. Either an actual feature might be of this form, or, because of access difficulties, it might be possible to measure only part of a complete feature. Examples of partial arcs are an arc of a circle, a patch of a sphere and a frustum⁶ of a cone. For instance, a partial (circular) arc might represent a corner radius (Figure 75). Such features can be more difficult to measure than full features. Because of the incomplete nature of the surface, errors can occur when trying to predict the centre and radius of the best-fit circle from the co-ordinate data from a CMM. Such a feature is, therefore, a bad choice for a datum.

Try this for yourself



The problems with partial arcs can be demonstrated by measuring a circular item, of say 25 mm radius, by contacting the surface at 20 points around the circumference. Note the centre co-ordinates and radius. Now make a measurement, but this time contact at 20 points in a 45° sector of the ring. Depending on the form deviations in the surface the results could be quite different.



Partial arc

Figure 75 A partial arc

Measuring partial arcs on CMMs – the problems

If a partial feature is to be measured relative to some other datum then it is often better to fit a circle equal to the specified radius and then look at the deviations of form from this circle. Consider a partial (circular) arc, measured at ten points uniformly spaced over part of a nominally circular feature, as shown in Figure 76.

⁶ A frustum of a cone is a solid figure formed from a cone by removing a slice at the top parallel to the base.

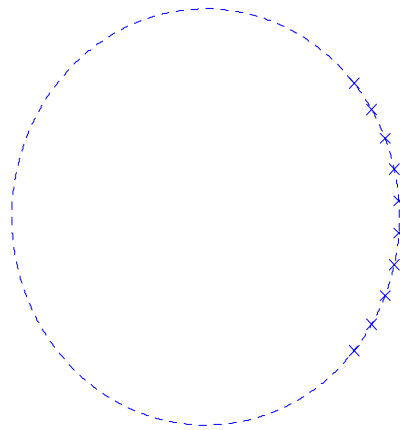


Figure 76 Ten uniformly spaced measurements on a partial arc, showing the (nominal) circle of which it is part

Suppose a least-squares circle is fitted to these points to obtain the radius and centre co-ordinates of the circle of which the arc is part. These circle parameters will have uncertainties associated with them as a consequence of the CMM measurement uncertainties. These uncertainties can be considerably greater when determined from such partial arc data as opposed to the use of measurements giving sensible coverage of a complete circle.

A partial arc will subtend a certain angle at the centre of the circle. Suppose the length of an arc is halved, thus halving the subtended angle. Ten uniformly spaced measurements are taken as before, but over this shorter arc. The resulting uncertainty of the computed radius is increased by a factor of approximately four, with a comparable statement concerning the uncertainty of the centre co-ordinates. This result applies for circular arcs that subtend any angle up to approximately 80° .

The significance of this result can be seen by applying it to an arc subtending, say, 80° , and then one subtending, say, 5° . The uncertainty in the radius determined for the latter case is greater than that for the former by a factor of over 250.

Design consideration – partial arcs

In such extreme circumstances of a 5° arc, it must be questioned whether the above circle parameters (circle centre co-ordinates and radius) are appropriate for an actual measurement task. To determine the radius and centre of a partial arc to within a small uncertainty might require an accuracy of co-ordinate measurement that is not readily available. A design specification in terms of such quantities can be regarded as unreasonable. A specification that required the form deviation (departure from circularity) to meet a certain tolerance would be much more reasonable (Figure 77). Determining whether the measurements of the arc indicate acceptance in this sense is simpler and forms a better approach. The arc would certainly not be a good choice as datum.

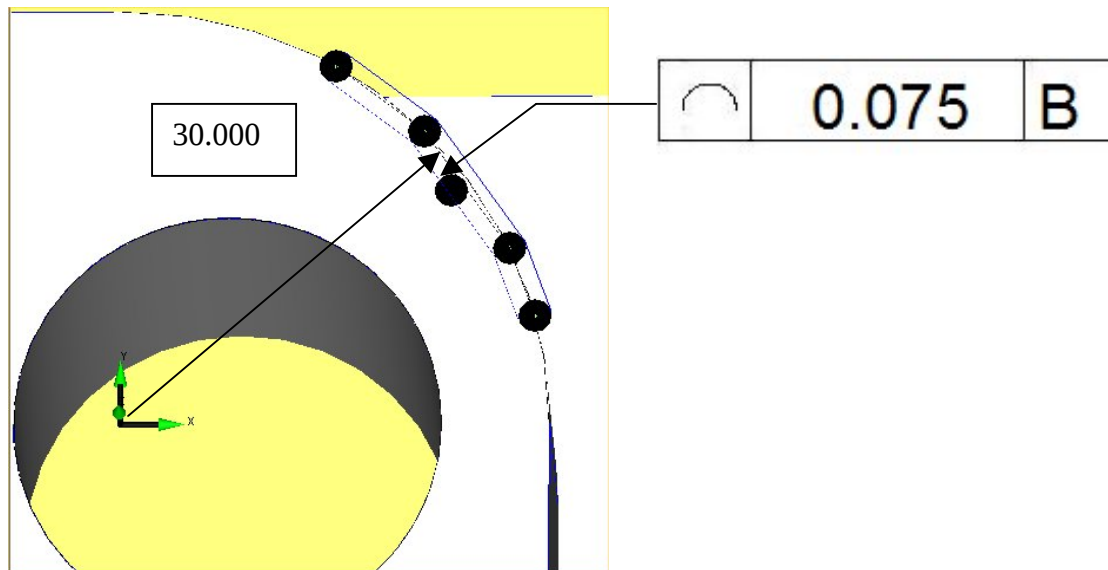


Figure 77 A partial arc with a form tolerance

Does the tolerance really need to be that tight?

When putting a dimension on a drawing the designer should think of the manufacturing and measurement implications. For instance if a diameter is toleranced at ± 0.001 mm this would mean that, as a rule-of-thumb, measurements need to be made with a measurement uncertainty at least ten times better, *i.e.*, with a measurement uncertainty of 0.000 1 mm. An uncertainty this low is only available from National Measurement Institutes such as NPL. To achieve this uncertainty will need specification of good surface texture and form (cylindricity in this case). The question the designer should ask is ‘Does the tolerance need to be this tight for the part to function correctly?’

Least squares or minimum zone

When evaluating form on CMMs the least-squares method has traditionally been used. This method was used historically as the least-squares method resulted in shorter computation times than minimum zone computations.

Minimum zone?



The minimum zone is defined for a circle as positioned to just enclose the measured profile such that their radial departure is a minimum.

According to ISO 1101:2004 the reference feature should only be determined by the minimum zone method. If a reference feature is determined by another method (for example, least squares) it can be assumed that the peak to valley is larger than the minimum zone requirement. Least squares should only be used where minimum zone is not available (for example, on older CMMs) or when the method has been agreed in advance. If no evaluation rule is indicated for form testing then the minimum zone method should be used especially in arbitration cases.

Inspection

5

IN THIS CHAPTER

- Ensuring that the components meet the requirements.
- Sampling versus 100% inspection.
- Choosing the appropriate measurement tool.
- Is it possible to measure the dimensions specified on the drawing to ascertain if the tolerance has been met?
- Determining conformance with a specification, UKAS rules, ISO 14253 decision rules.

The reason for inspection is to check that the component that has been manufactured meets the specification detailed on the drawing. This chapter will highlight some of the considerations that need to be taken into account during the inspection process.

Ensuring that the components meet the requirements

Inspection is important to prove that the components meet the design requirements. There are many things to consider including the type of measuring tool to be used, its uncertainty, its availability and capability to mention but a few.

Sampling versus 100% inspection

Before an inspection strategy is designed the decision needs to be made about whether 100% inspection or sampling⁷ should be used. Obviously for some industries 100% inspection is impracticable because those industries are producing items far quicker than they can measure them. It is an impossible task to measure everything and it is not that critical to measure all parts because the process has been proven to be stable using the quality tools and techniques previously mentioned. Alternatively, in some industrial applications it is critical to measure every dimension on every part due to the fact that the component may be safety critical, for example, aircraft components, medical component, *etc.* So the argument for sampling is dependent on the application of the component and their relevant safety requirements.

Choosing the appropriate measurement tool

When the measurement process takes place the choice of measurement tool for that application is important, sometimes a choice is available, sometimes not. The choice is dependent on the factors required from the design drawing or CAD model. Measurement uncertainties have to be considered. If we consider, BS EN ISO14253-1 *Geometrical Product Specifications – Inspection by measurement of workpieces and measuring equipment* we can look at the decision rules for proving conformance or non-conformance with specifications. From these rules we can choose the measurement equipment based on the uncertainty of measurement we need to achieve to prove conformance or non-conformance. In addition, speed of measurement may be an issue. It is also important not to tie up expensive and accurate equipment on making measurements that could be made easier and to the required accuracy on cheaper and less accurate equipment. For example, the use of a micrometer may be preferable to using a CMM if the tolerance is only 0.3 mm.

⁷ Sampling is that part of statistical practice concerned with the measurement of individual components so as to yield some knowledge about the rest of the population.

Is it possible to measure the dimensions specified on the drawing to ascertain if the tolerance has been met?

When measuring a component it is important that the appropriate equipment is available to ascertain conformance. As stated previously the starting point is ISO 14253-1 as will be discussed later.

It is the inspector's job to check that the designer has not specified tolerances for which the technology does not exist to prove conformance/non-conformance. Early in the design stage the measurement process for each toleranced feature should have been agreed. New measuring equipment should have been purchased and evaluated as necessary. Alternatively the possibility of sub-contract measurement should have been investigated. If the technology does not exist within your company to perform the measurements and you have to sub-contract the cost of measurement should be budgeted for early on. The need to sub-contract probably suggests the measurement is difficult and, therefore, may be costly.

If ascertaining compliance using the ISO 14253 rules is going to be difficult given the equipment at your disposal it may be worth checking with the designer that the tolerance needs to be that tight for the part to function correctly. Are the tolerances reasonable?

As was discussed earlier, the measurement of partial features on a drawing can be problematical. If partial features are wrongly specified on the drawing it is worth discussing with the designer if the features can be specified in a more appropriate way.

Datum features are critical to all measurements and you should make sure that any datum features on the drawing are of the appropriate size. As stated earlier small measurement errors have larger effect when measuring a datum that is small in size compared to the feature being evaluated.

Finally, sometimes measurement and inspection is made far more complex than it need be due to problems of access to features. Ideally such problems can be anticipated and solved at design time.

The importance of traceability

Before you start any inspection you will have to make sure that all your equipment is traceable.

PD 6461-1:1995 defines traceability as follows:

property of the results of a measurement whereby it can be related to stated references, usually national or international standards, through an unbroken chain of comparisons all having stated uncertainties

The important word here is *uncertainty*. As we will soon see, you cannot ascertain conformance unless you know the uncertainty of measurement.

Traceability is usually assured by having your equipment calibrated at a UKAS accredited laboratory, at a national laboratory or in-house against traceable standards. Traceability is important to ensure that components made in different organisations assemble correctly.

A typical traceability chain may take the following form:

- ❑ Micrometer checked in-house against gauge blocks calibrated at a UKAS accredited laboratory.
- ❑ UKAS laboratory calibrates the customer's gauge blocks by comparison against a set calibrated at NPL by interferometry.
- ❑ The gauge block laboratory at NPL has the lasers used in interferometry, thermometers, pressure measuring system, *etc.*, calibrated against the appropriate primary standard.

Figure 78 shows the traceability chain for gauge blocks in diagrammatic form.

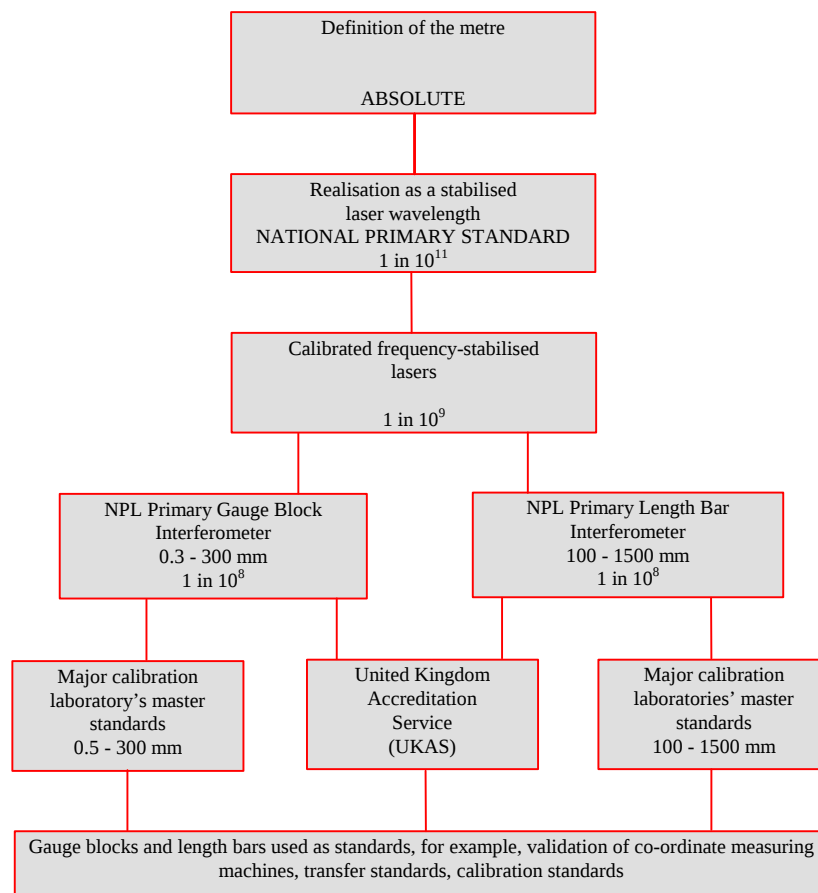


Figure 78 Traceability chain for gauge blocks

At the head of the traceability chain for length measurements is the definition of the metre (1983):

The metre is the length of the path travelled by light in vacuum during a time interval of 1/299 792 458 of a second.

For more information on traceability see Appendix A.4.

Uncertainty of measurement

Uncertainty of measurement is covered in Bell, S A *A beginner's guide to uncertainty in measurement* Measurement Good Practice Guide No. 11 (Issue 2), NPL, March 2001. If the reader is unfamiliar with measurement uncertainty it is advised they read this guide before reading the next section.

The definition of uncertainty is:

parameter, associated with the result of a measurement, that characterizes the dispersion of the values that could reasonably be attributed to the measurand

(PD 6461-4:2004 General metrology. Practical guide to measurement uncertainty)

Determining conformance with a specification - ISO 14253 decision rules

When making a measurement you may think that it is a simple matter of the result falling within the tolerance band to prove conformance. This is not the case as the following example shows.

The designer has specified that a hole should be $50 \text{ mm} \pm 0.005 \text{ mm}$ (top and bottom lines in Figure 79). The first operator measures the size with a traceable micrometer as 50.004 mm and states that the hole conforms to the drawing. However, the foreman, looking at this result examines the uncertainty of the micrometer. The measurement uncertainty of the micrometer is 0.003 mm and applying this uncertainty he realises that the actual size could lie between 50.001 mm and 50.007 mm . He gets the hole remeasured on a bore comparator that has a 0.001 mm uncertainty. The measurement comes out at 50.006 mm and conformance is not proven. As a general rule the measurement uncertainty of the equipment should be no greater than ten percent of the tolerance band.

Note that in this case both measurement results agree to within their uncertainties. For measurement 1, however, the measured value is less than the uncertainty away from the upper specification limit (USL) and no real information has been obtained about whether the true value is inside or outside the specification limits.

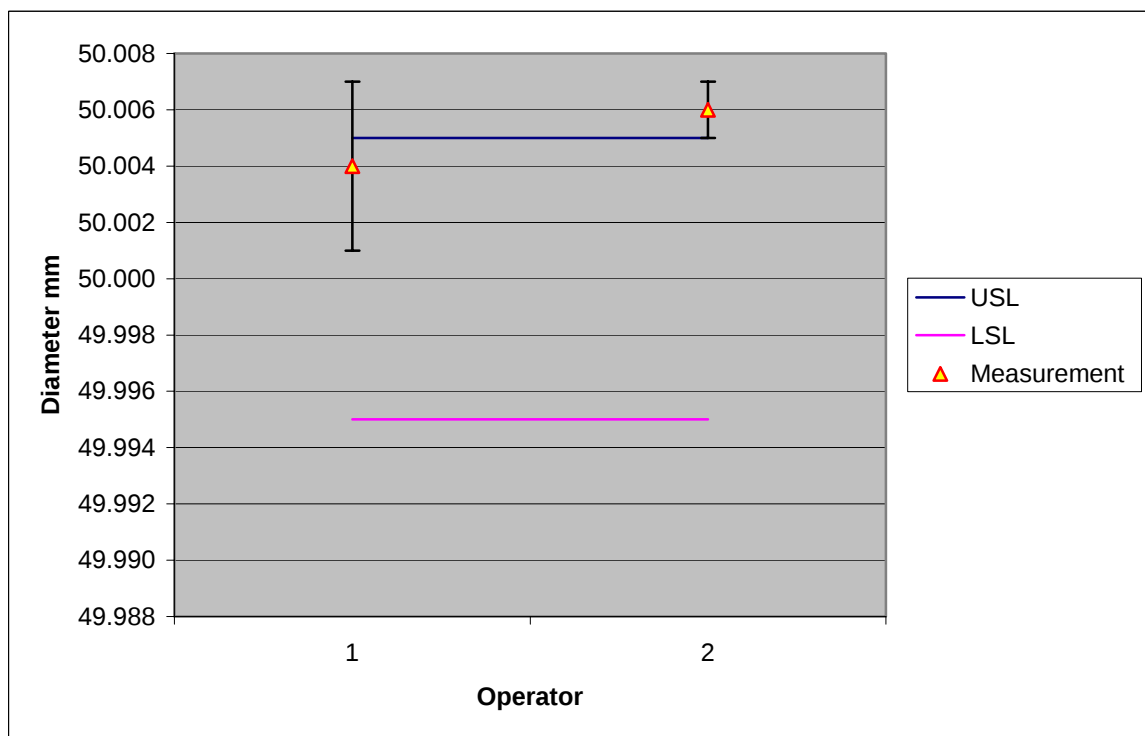


Figure 79 The two results and their uncertainties

ISO 14253 recommends that the following rules be applied for the most important specifications controlling the function of the workpiece or the measuring equipment.

At the design stage the terms “in specification” and “out of specification” refer to the areas separated by the upper and lower tolerance (double sided) or either LSL or USL for a one sided specification (see Figure 80 areas 1 and 2, line C).

When dealing with the manufacturing or measurement stages of the process the LSL and USL are added to by the measurement uncertainty. The conformance or non-conformance ranges are reduced due by the uncertainty (see Figure 80, line D).

These rules are to be applied when no other rules are in existence between supplier and customer. ISO 14253 allows for other rules to be agreed between customer and supplier. These rules must be fully documented.

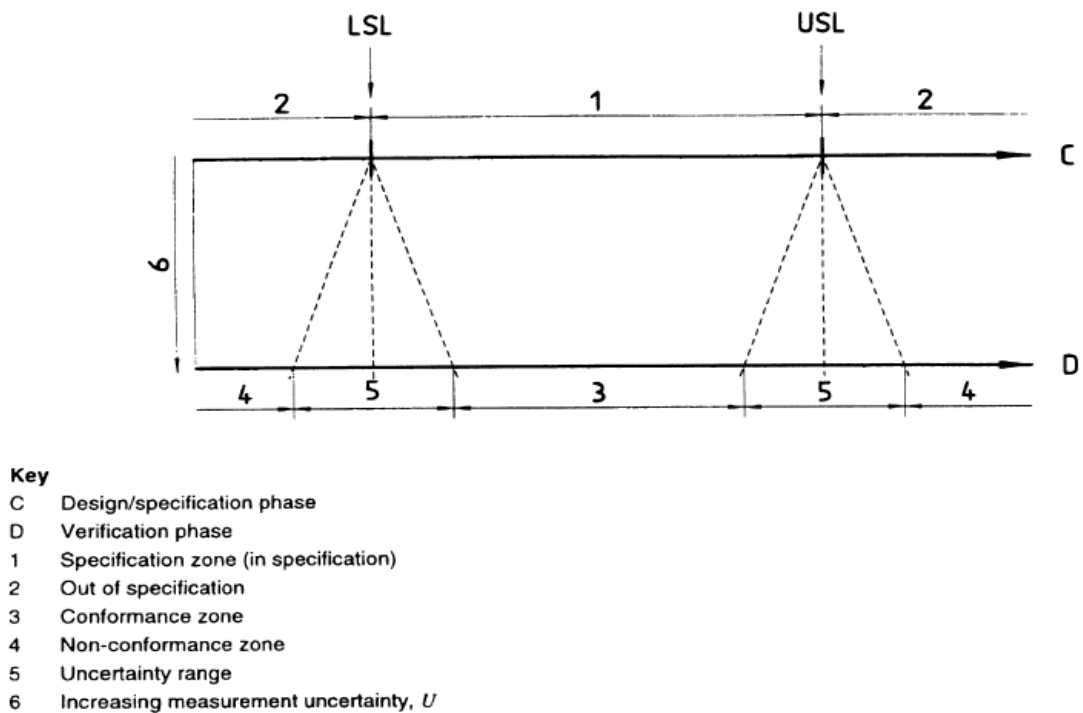


Figure 80 Uncertainty of measurement: the uncertainty range reduces the conformance and non-conformance zones (Copyright BSI – extract from BS EN ISO 14253-1:1999)

Conformance with a specification is proved when the result of measurement, complete statement,⁸ falls within the tolerance zone or within the maximum permissible error of the specification for measuring equipment (for example, the maximum permissible error of a CMM). Conformance is also proven when the measurement result falls within the tolerance zone reduced on either side by the expanded uncertainty. The conformance zone is linked to the LSL, USL and actual expanded uncertainty.

Non-conformance with a specification is proved when the result of measurement, complete statement, falls outside the tolerance zone or outside the maximum permissible error of the specification for measuring equipment. Non-conformance is also proven when the measurement result is outside the tolerance zone increased on either side by the expanded uncertainty. The non-conformance zone (4 in Figure 80) is linked to the USL, LSL and expanded uncertainty.

Neither conformance nor non-conformance with a specification can be proven when the result of measurement, complete statement, includes one of the specification limits (for example, measurement 1 in Figure 79 and Figure 81).

It is important that the principle behind these rules is applied to a supplier/customer relationship where the uncertainty of measurement always counts against the party

⁸ A complete statement of the result of a measurement includes information about the uncertainty of measurement.

who is providing the proof of conformance or non-conformance, *i.e.* the party making the measurement. That is to say the supplier will reduce the tolerance by their measurement uncertainty to prove conformance. The customer will increase the tolerance by their measurement uncertainty to prove non-conformance.

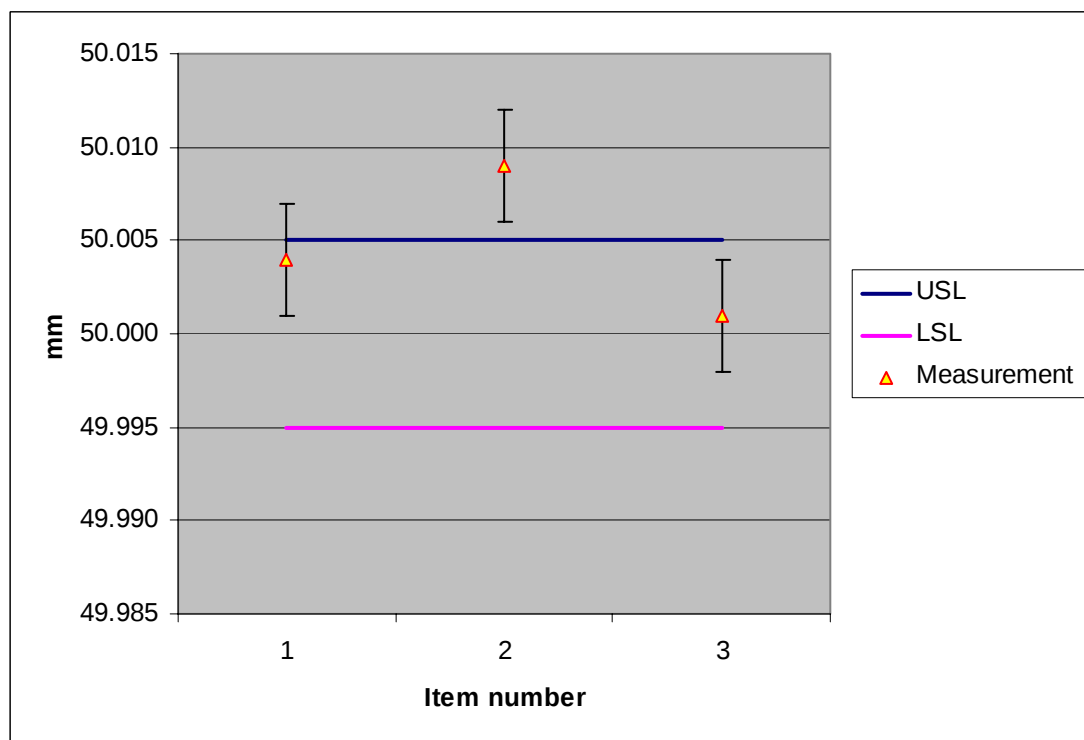


Figure 81 Conformance or non-conformance

Referring to Figure 81, three items have been measured. The purple line shows the LSL, the blue line the USL.

Measurement of item 1 - neither conformance nor non-conformance with a specification can be proven

Measurement of item 2 – non-conformance is proven

Measurement of item 3 – conformance is proven

In the case of item 1 the result of measurement, complete statement straddles the USL and neither conformance nor non-conformance with a specification can be proven. In the case of item 2 the result of measurement, complete statement is above the USL and so non-conformance is proven. In the case of item 3 the result of measurement, complete statement is above the LSL and below the USL and so conformance is proven.

Summary of ISO 14253

ISO 14253 can be summed up in the following statements:

The supplier shall prove conformance in accordance with clause 5.2⁹ of BS EN ISO 14253 using their estimated uncertainty of measurement.

The customer shall prove non-conformance in accordance with clause 5.3¹⁰ of BS EN ISO 14253 using their estimated uncertainty of measurement.

When evaluating the measurement result the uncertainty is always at the disadvantage of the party with onus on proof.

In the past the measurement uncertainty has been ignored when ascertaining conformance with a specification as long as the uncertainty was 1/10th of the specification width. This procedure is no longer acceptable.

⁹ Rule for proving conformance with specification

¹⁰ Rule for proving non-conformance with specifications

Published Standards

6

IN THIS CHAPTER

- A list of published standards.

List of published standards

The following is a list of published standards relevant to this guide. In the UK standards are available from British Standards Institution (BSI).

ISO 1:2002 Geometrical Product Specifications (GPS) -- Standard reference temperature for geometrical product specification and verification

ISO 2692:1988 Technical drawings -- Geometrical tolerancing -- Maximum material principle

ISO 406:1987 Technical drawings -- Tolerancing of linear and angular dimensions

ISO 1101:2004 Geometrical Product Specifications (GPS) -- Geometrical tolerancing -- Tolerances of form, orientation, location and run-out

ISO 1302:2002 Geometrical Product Specifications (GPS) -- Indication of surface texture in technical product documentation

ISO 14253-1:1998 Geometrical Product Specifications (GPS) -- Inspection by measurement of workpieces and measuring equipment -- Part 1: Decision rules for proving conformance or non-conformance with specifications

ISO/TS 14253-2:1999 Geometrical Product Specifications (GPS) -- Inspection by measurement of workpieces and measuring equipment -- Part 2: Guide to the estimation of uncertainty in GPS measurement, in calibration of measuring equipment and in product verification

ISO/TS 14253-3:2002 Geometrical Product Specifications (GPS) -- Inspection by measurement of workpieces and measuring equipment -- Part 3: Guidelines for achieving agreements on measurement uncertainty statements

BS EN ISO 14660-1:2000 Geometrical Product Specifications (GPS). Geometrical features. General terms and definitions

BS EN ISO 14660-2:2000 Geometrical Product Specifications (GPS). Geometrical features. Extracted median line of a cylinder and a cone, extracted median surface, local size of an extracted feature

Glossary of terms

7

IN THIS CHAPTER

- A glossary of terms.

Glossary of terms

CAD	Computer aided design is a computer based method used to produce design drawings and specifications of components and assemblies.
Cartesian co-ordinate system	Co-ordinate system with three orthogonal axes – normally defined as x , y and z at mutual right angles to each other.
CMM	A measuring system with the means to move a probing system and capability to determine spatial co-ordinates on a workpiece surface. (ISO 10360)
Co-ordinate system	A co-ordinate system is used to define where the product dimensions originate from. This provides the information for the manufacturing and measurement engineers to be able to set up the component part as designed.
Cylindrical co-ordinate system	A co-ordinate system defined with a radial distance, an angle and a height.
Datum	A theoretically exact geometric reference (such as an axis, a plane, a straight line, <i>etc.</i>), to which toleranced features are related. Datums may be based on one or more datum features of a part. (ISO 5459)
Datum features	A real feature of a part (such as an edge, a surface, or a hole <i>etc.</i>), which is used to establish the location of a datum. (ISO 5459)
Engineering drawing	An engineering drawing shows the requirements of the design function, with clear and relevant information so that the product can be manufactured and inspected to those requirements.

Geometric characteristics	... a series of symbols that can be used to describe the design requirements of the feature or features to be manufactured or measured.
GPS	Geometrical Product Specification covers requirements on sizes, geometrical tolerances and geometrical properties of surfaces.
LSL	Lower specification limit defines the smallest allowable value for the dimension to be within specification.
MMC	Maximum material condition is a principle used in conjunction with the assembly of different component parts that provides the combination of the smallest hole and the largest pin to fit together as an assembly.
Non-contact	Probing system which needs no material contact with a surface being measured in order to function (ISO 10360).
Result of measurement	...value attributed to a measurand, obtained by measurement.
Size	A number expressing, in a particular unit, the numerical value of a linear dimension.
Spherical co-ordinate system	Spherical co-ordinate system defined with a radial distance, and two angles.
Thermal expansion	How much the material changes length for a given temperature change is known as the coefficient of linear thermal expansion.
Tolerance	... difference between the upper and lower tolerance limits.

Traceability	... property of the results of a measurement whereby it can be related to stated references, usually national or international standards, through an unbroken chain of comparisons all having stated uncertainties.
USL	...upper specification limit defines the largest allowable value for the dimension to be within specification.
Uncertainty	Measurement uncertainty defines a quantifiable variation from the true value within a confidence level.
Uncertainty of measurement	...parameter, associated with the result of a measurement that characterizes the dispersion of the values that could reasonably be attributed to the measurand. (ISO 14253-1)
Variation	... phenomenon that the value of a characteristic is not constant within one individual feature or within a set of workpieces. (ISO 17450-1)
Value	...magnitude of a particular quantity generally expressed as a unit of measurement multiplied by a number.
Virtual condition	... limiting boundary of perfect form permitted by the drawing data for the feature; the condition is generated by the collective effect of the maximum material size and the geometric tolerances. (ISO 2692)
Virtual size	... dimension defining the virtual condition of a feature.

Appendices

8

IN THIS CHAPTER

- Appendix A Links to other useful sources of information.
- Appendix B Further examples.

Appendix A Links to other useful sources of information

A.1 National and International Organisations

A.1.1 National Physical Laboratory

The National Physical Laboratory (NPL) is a world-leading centre in the development and application of highly accurate measurement techniques. As the UK's national standards laboratory, NPL underpins the National Measurement System (NMS), ensuring consistency and traceability of measurements throughout the UK. NPL offers a unique range of measurement services, contract research, consultancy and training services. Other areas of expertise include the design and characterisation of engineering materials, and mathematical software, especially its application to measurement and instrumentation.

For more information on the wide range of metrology services, facilities and research work carried out at NPL either visit the NPL web site at www.npl.co.uk or contact the NPL Helpline at

Tel: 020 8943 6880, Fax: 020 8943 6458, E-mail: enquiry@npl.co.uk

A.1.2 National Institute of Science and Technology (NIST)

NIST is the equivalent of NPL in the United States of America. The NIST web site at www.nist.gov often contains documents relevant to this guide in pdf format.

A.1.3 EUROMET

EUROMET is a cooperative organisation between the national metrology institutes in the EU including the European Commission, EFTA and EU Accession States. The participating metrology institutes collaborate in EUROMET, with the objective of promoting the co-ordination of metrological activities and services with the purpose of achieving higher efficiency.

The main aims of EUROMET are:

- to encourage cooperation in the development of national standards and measuring methods;
- to optimise the use of resources and services and
- to improve measurement facilities and make them accessible to all members.

For more information visit the EUROMET web site at: www.euromet.ch

A.1.4 Institute for Geometrical Product Specification

More information about GPS can be found at the Institute for Geometrical Product Specification website www.ifgps.com. Click on resources for more information on GPS.

A.2 Networks

A.2.1 Dimensional Metrology Awareness Club (DMAC)

DMAC is an industry-focused club. The club is managed by NPL on behalf of the DTI and has the following objectives:

- to provide a focused platform for interaction across all the technical areas covered by the Engineering Measurement Programme;
- to enable members to learn about the latest developments in their own as well as related areas of dimensional metrology;
- to provide a mechanism to encourage and facilitate interaction and exchange of ideas with other member organisations and
- to give members the opportunity to provide input to, and influence the development of future NMS programmes.

For further information visit the DMAC web site at: www.npl.co.uk/npl/clubs/dmac/

A.2.2 Software Support for Metrology Programme (SSfM)

SSfM is a metrology-focused programme comprising projects designed to meet the requirements of NMS programmes. The overall aim of the SSfM Programme is to tackle a wide range of generic issues, some of which are problems in metrology that require the application of established software engineering practices, whilst others require advances in mathematics, software engineering or theoretical physics. The programme, thus, includes work in metrology, mathematics, software and theoretical physics, with strong links between the various disciplines.

The SSfM Club is aimed at users and suppliers of metrology software, giving them a say in the direction of the Programme. It is the focal point for the distribution of general information arising from the Programme.

Further details can be found at: www.npl.co.uk/ssfm/

A.3 National and International Standards

A.3.1 British Standards Institution (BSI)

BSI started in 1901 as a committee of engineers determined to standardise the number and type of steel sections in order to make British manufacturers more efficient and competitive. The BSI Group is now the oldest and arguably the most prestigious national standards body in the world and is among the world's leading commodity and product testing organisations.

A.3.2 International Organisation for Standardization (ISO)

The International Organization for Standardization (ISO) is a worldwide federation of national standards bodies from some 140 countries.

The mission of ISO is to promote the development of standardisation and related activities in the world with a view to facilitating the international exchange of goods and services, and to developing cooperation in the spheres of intellectual, scientific, technological and economic activity.

ISO's work results in international agreements that are published as International Standards.

Further information on ISO can be found at: www.iso.ch

A.4 Traceability

Traceability in measurement is the concept of establishing a valid calibration of a measuring instrument or measurement standard, by a step-by-step comparison with better standards up to an accepted or specified standard. In general, the concept of traceability implies eventual reference to an appropriate national or international standard.

The National Physical Laboratory is the United Kingdom's national standards laboratory. It operates at the heart of the National Measurement System (NMS) which is the infrastructure designed to ensure accuracy and consistency in every physical measurement made in the UK. Chains of traceability link UK companies' measurements directly to national standards held at NPL.

For the majority of industrial applications, companies can establish a link to national measurement standards through the calibration and testing services offered by United Kingdom Accreditation Service (UKAS) accredited laboratories, which are in turn traceable to NPL. However, for challenging or novel measurements to the highest standards of accuracy, which are not catered for by UKAS-accredited laboratories, NPL can often provide a traceable measurement solution directly to industry.

UKAS is the sole national body recognised by government for the accreditation of testing and calibration laboratories, certification and inspection bodies. A not-for-profit company, limited by guarantee, UKAS operates under a Memorandum of Understanding with the government through the Department of Trade and Industry.

UKAS accreditation demonstrates the integrity and competence of organisations providing calibration, testing, inspection and certification services.

Further information on UKAS can be found at: www.ukas.com.

A.5 National Measurement Partnership (NMP)

The National Measurement Partnership programme is an initiative of the Department of Trade and Industry with the aim of developing the organisational infrastructure of the UK's National Measurement System.

The objectives of this programme are to:

- increase the uptake of accredited calibration in the UK;
- increase the qualified skill base in measurement and
- increase user access to measurement expertise.

To achieve these objectives, the programme is:

- establishing a network of accredited UKAS Calibration Laboratories;
- expanding the Competing Precisely initiative to provide increased measurement advice;
- establishing a National Measurement Helpline;
- managing the roll-out of measurement training in the form of a National Vocational Qualification ;
- establishing a National Measurement Forum to steer and support the programme.

The National Measurement Partnership has produced a number of products to help promote good measurement practice, including case histories, good practice guides and self-assessment checklists.

For further details see www.nmpuk.co.uk.

A.6 Training courses

Training courses are available on some of the material detailed in this guide. Training is available from Keith Bevan, one of the authors of this guide.

Keith runs Bevan Training and Assessment Services (BTAS) and has 25 years of experience in industry and training. BTAS provides industry-focused training and assessment services predominantly in the Engineering Production and Technical Support areas of National Vocational Qualifications (NVQ). The methods of delivery include:

- On-line and on site training courses available in metrology
- GPS training

For further details see www.bevantraining.co.uk.

A.6.1 E-training

For details of NPL's e-learning materials see www.npl.co.uk/e_training/ (Figure 82). The training focuses on the area of coordinate measuring machine (CMM) probing and measurement strategies. Particular attention is being paid to ensuring that the material developed and the training and assessment methodologies used are suitable to contribute to the underpinning knowledge for a Technical Services NVQ. Figure 83 shows an example screen from the e-training material.

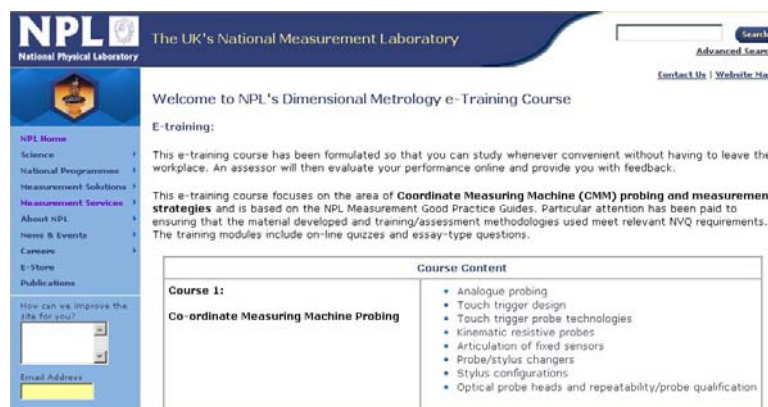


Figure 82 E-training via the NPL website

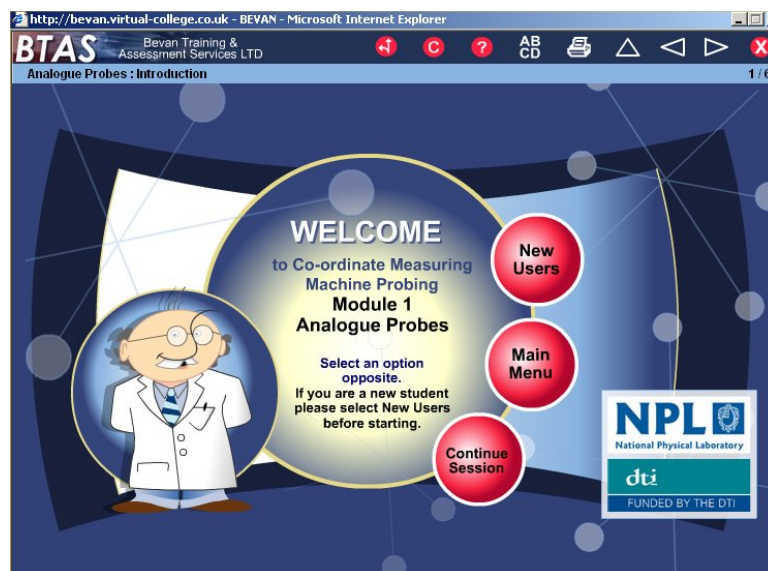


Figure 83 The introductory screen to the analogue-probing module

A.7 Further reading

Busch T, Harlow R, Thompson R L 1998 *Fundamentals of Dimensional Metrology, Third Edition*, (New York: Delmar Publishers)

Humieny Z (editor) *et al.* 2001 *Geometrical Product Specifications – Course for Technical Universities*, (Warsaw: Warsaw University of Technology Printing House)

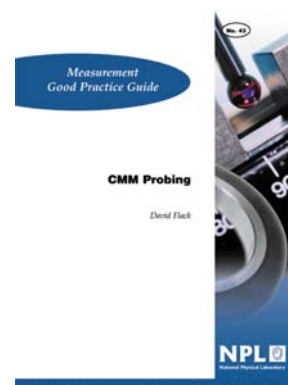
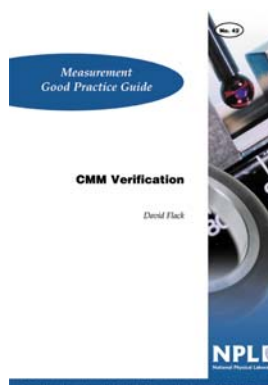
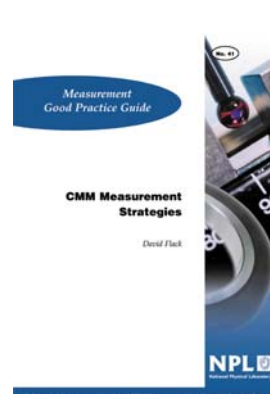
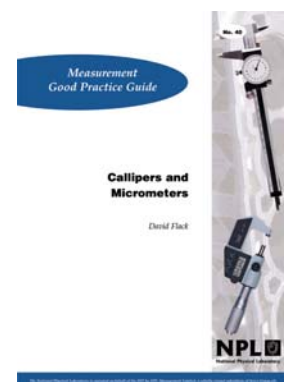
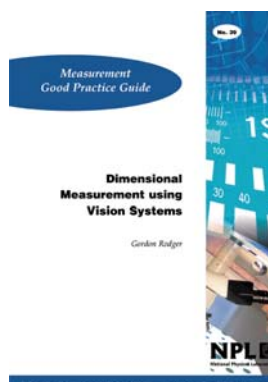
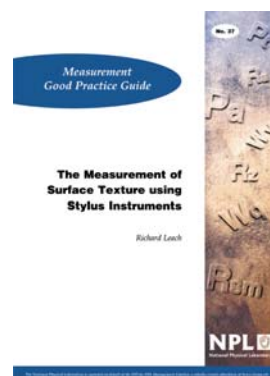
Green P 2005 *The Geometrical Tolerancing Desk Reference: Creating and Interpreting ISO Standard Technical Drawings*, (Newnes)

Leach R K 2001 *The measurement of surface texture using stylus instruments* Measurement good practice guide No. 37 (NPL)

Bell S A 2001 *A beginner's guide to uncertainty in measurement* Measurement good practice guide No. 11 Issue 2 (NPL)

PD 6461-4:2004 General metrology. Practical guide to measurement uncertainty

PD 6461-3:1995 Guide to the expression of uncertainty in measurement (GUM)



Appendix B Further reading

B.1 Maximum material condition – an example

An example of how the principle of maximum material condition can be of benefit is shown in Figure 84 and Figure 85. These diagrams show how the tolerance zone is increased and the hole position found acceptable to the required design function as shown in.

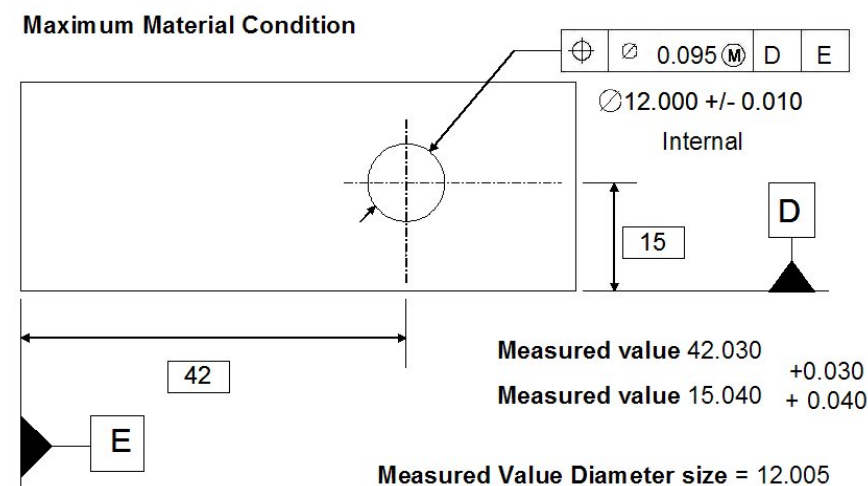


Figure 84 The drawing requirements and actual measured values obtained

Geometric Dimensioning & Tolerancing

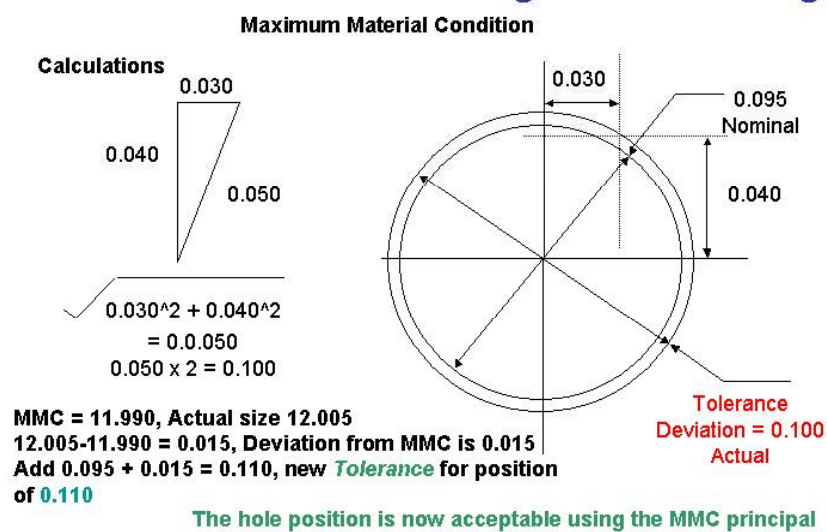


Figure 85 The calculations of the hole position both before and after the application of the MMC principle

B.2 Statistical Process Control

In chapter 3 a brief mention was made of some SPC terms. This section aims to explain in a little more detail what those terms mean.

B.2.1 Mean

The mean is a mathematical average of the readings taken. The mean is used in calculations required for further evaluation of capability of component dimensions. It is expressed mathematically as follows:

$$\bar{x} = \frac{\sum_{i=1}^n x_i}{n} = \frac{x_1 + x_2 + x_3 + x_4 + \dots + x_n}{n}$$

That is to say it is the sum of all the values divided by the number of values. Note the useful sigma notation.

B.2.2 Range

Difference between the smallest and largest values obtained during measurement

B.2.3 Standard deviation

Mathematical calculation of the variation obtained from the results. As the variation increases so does the standard deviation. It is effectively the root mean square deviation of the results. It is a useful way of characterising the reliability of the results. It is often referred to as sigma or σ .

B.2.4 Normal bell shaped curve

There are various ways to specify a random variable. The most visual is the probability density function. The probability density function represents how likely each value of the random variable is.

The normal distribution, also called the Gaussian distribution, is an extremely important probability distribution. The distributions have the same general form, differing in their location and scale parameters; the mean (or average) and standard deviation (variability), respectively. A plot of the distribution is often called the bell curve because the graph of its probability density resembles a bell (Figure 86).

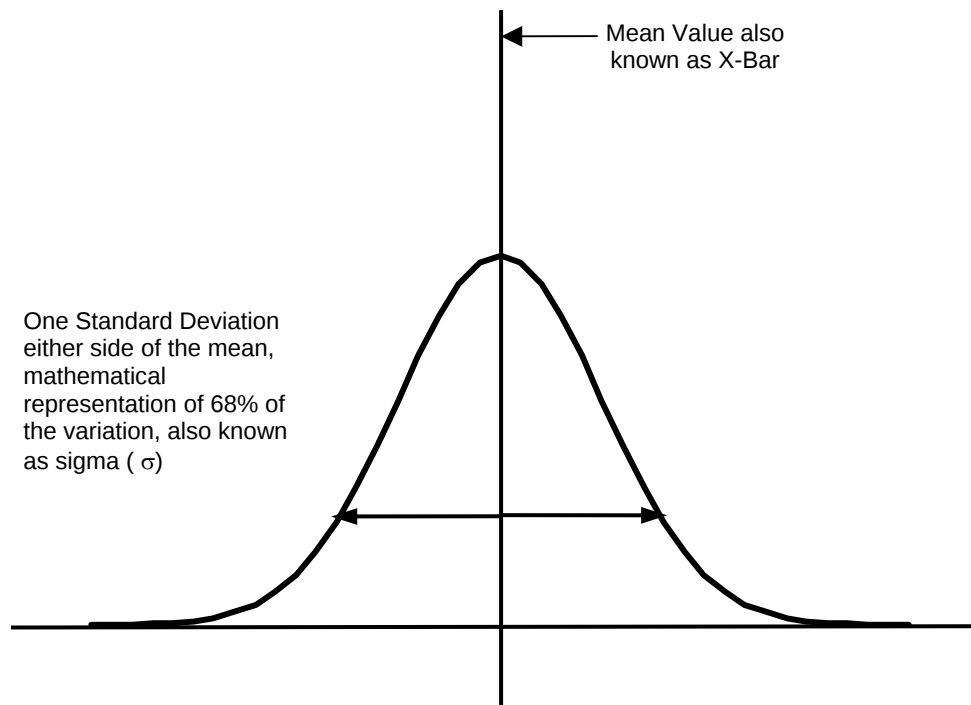


Figure 86 Normal or bell shaped curve

Some notable qualities of the normal distribution are:

The density function is symmetric about its mean value.

68.27% of the area under the curve is within one standard deviation of the mean.

95.45% of the area is within two standard deviations.

99.73% of the area is within three standard deviations.

B.2.5 Histogram and control chart

Histograms and control charts are two different types of graphs used to present the results. The histogram develops a similar shape to the curve shown above (Figure 86) in standard deviation, but is designated into classes (Figure 87), it can link to the tolerance and, therefore, capability. The control chart (Figure 88) is used to control the process once capability has been proven. Control limits are defined to allow for reaction to problems within the process before the dimensions are out of specification. The values from the process are plotted over a time period.

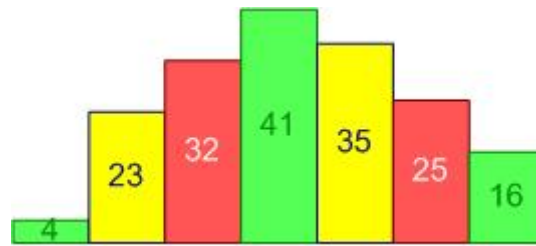


Figure 87 A histogram



Figure 88 A control chart

B.2.6 Capability

One method of expressing capability is a calculation based on the variation, and the tolerance (known as CP) and a calculation based on the variation, tolerance and location (known as CPK). These parameters used to assess the capability of a process

Cp = Process Capability. A simple and straightforward indicator of process capability.
 Cpk = Process Capability Index. Adjustment of Cp for the effect of non-centered distribution.

Cpk is an index which measures how close a process is running to its specification limits, relative to the natural variability of the process. The larger the index, the less likely it is that any item will be outside the specification.

B.2.7 Six Sigma

Statistical techniques are used in the Six Sigma¹¹ approach to process improvement. The classical view of quality was that 99.73% parts should be acceptable (three sigma), but the six-sigma view of quality is 99.99966% parts should be acceptable (six sigma). The relationship between the process capability and the part per million defects is shown in Table 2. The methodology of Six Sigma is to provide a mechanism to improve customer satisfaction by providing almost perfect products and services.

Table 2 Process capability

Process Capability (sigma)	Defects per million opportunities (ppm)
2	308,507
3	66,807
4	6,210
5	233
6	3.4

The Six Sigma concept looks at improvements that can be made by reducing variation.

Six Sigma is split into two main parts:

1. Design for six sigma.
2. Six Sigma (process).

The use of 'Design for six Sigma' is to ensure that new products fit into six sigma processes. Where as 'Six Sigma process' looks at the variation in the current processes and helps you identify, from the data how improvements can be made by reducing the variation in those processes

Both these techniques should run parallel, not only in-house but also through your supply chain. A step-by-step approach to produce successful reductions from the data gathered is important.

¹¹ Six Sigma is a trade mark of Motorola