Geometric Dimensioning & Tolerancing

ASME Y14.5M, 1994

REV. A

Table of Contents

1. General Rules

1.1. Rule 1 – Limits of Size

1.1.1. Individual Feature of Size

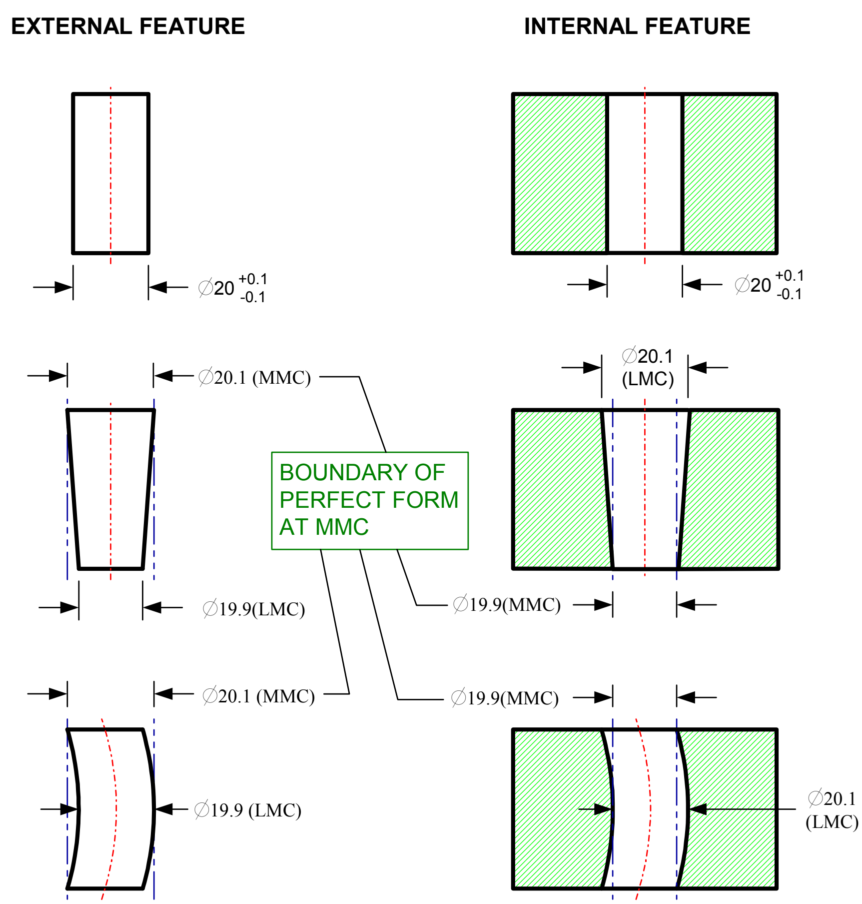
Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form and size are allowed.

1.1.2. Variations of Size

The actual size of an individual feature at any cross-section shall be within the specified tolerance of size.

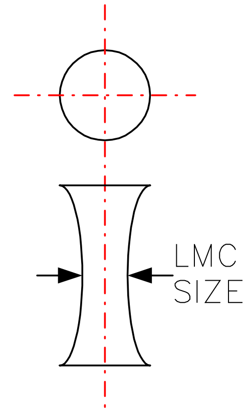
1.1.3. Variations of Form (Envelope Principle)

a) The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at Maximum Material Condition (MMC). This boundary is the true geometric form represented by the drawing. No variation in form is permitted if the feature is produced at its MMC limit of size.



b) Where the actual local size of a feature has departed from MMC toward Least Material Condition (LMC), a variation in form is allowed equal to the amount of such departure.

c) There is no requirement for a boundary of perfect form as LMC.



1.1.4. Relationship between Individual Features

The limits of size do not control the orientation or location relationship between individual features.

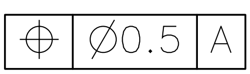
Features shown perpendicular, coaxial, or symmetrical to each other must be controlled for location or orientation to avoid incomplete drawing requirements.

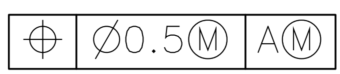
1.2. Rule 2 – Applicability of Feature Size

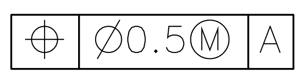
Applicability of material condition modifier (RFS, MMC, LMC) is limited to features subject to variations in size.

They may be datum features or other features whose axes or centre planes are controlled by geometric tolerances.

FOR ALL Applicable Geometric Tolerances: RFS applies will respect to the individual tolerance, datum reference, or both, where NO MODIFYING SYMBOL is specified.
. “ASME Y14.5-1994”





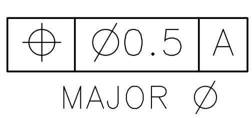


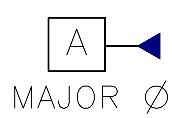
1.3. Rule 3

All other controls is implied Regardless of Feature Size (RFS).

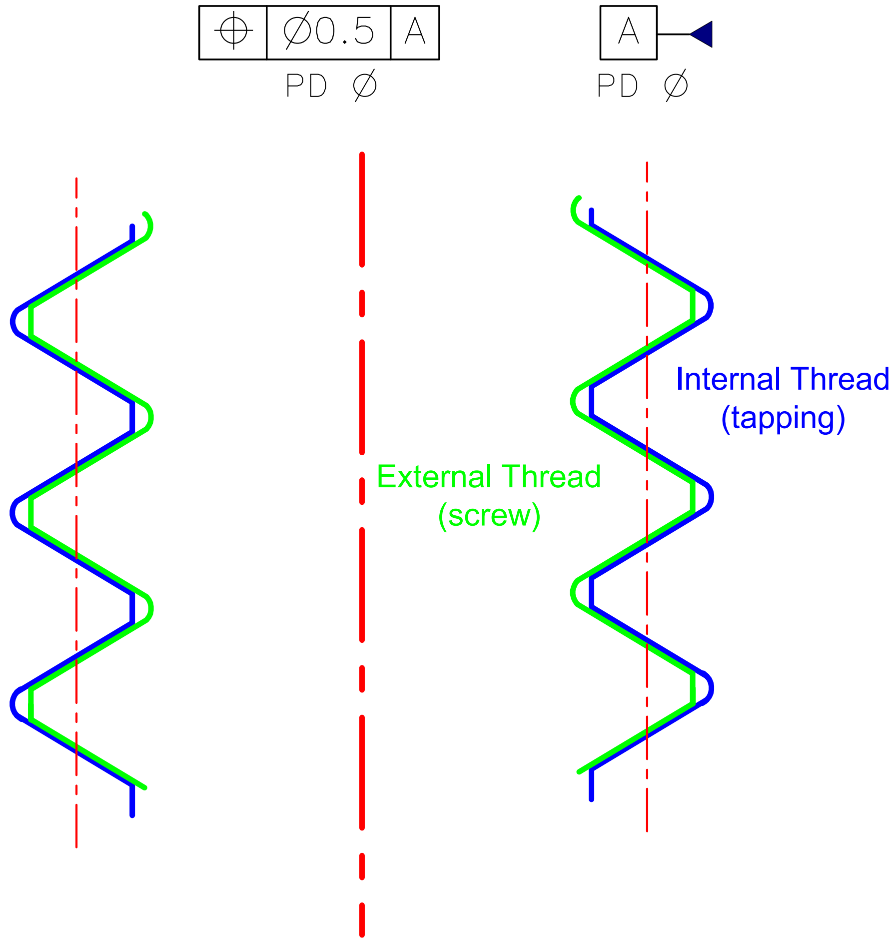
1.4. Pitch Rule

a) Each tolerance of orientation or position and datum reference specified for a screw thread applies to the axis of the thread derived from the pitch cylinder.





b) Each tolerance of orientation or position and datum reference specified for features other than screw threads, such as gears and splines, must designate the specific feature to which each applies.



1.5. Virtual Condition

A constant boundary generated by the collective effects of a size feature's specified MMC or LMC material condition and the geometric tolerance for that material condition.

The virtual condition of a feature is the extreme boundary of that feature which represents the 'worst case' for, typically, such concerns as a clearance of fit possibility relative to a mating part or situation.

PIN: VC = Size MMC + Tolerance
VC = Size LMC – Tolerance

HOLE: VC = Size MMC – Tolerance
VC = Size LMC + Tolerance

1.6. Exercise

1. A(n) ________________ is a numerical value expressed in appropriate units of measure, indicated on a drawing and in documents to define the size and/or geometric characteristics and/or locations of features of a part.

2. ________________ is a general term applied to a physical portion of a part.

3. Define Tolerance.

   ____
   ____
   ____
   ____

4. All Dimensions shall have a tolerance except for dimensions that are identified as:
   a) reference. b) maximum. c) minimum.
   d) stock sizes. e) all of the above.

5. What are the limit of the dimension 25±0.4? __________________

6. What is the tolerance of the dimension in question 5?____________

7. What is the nominal dimension of the dimension shown in question 5?
   ____

8. Give an example of an equal bilateral tolerance. ________________

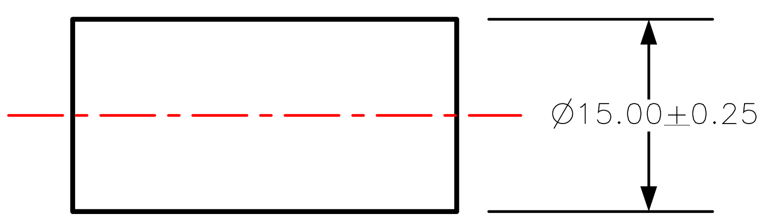
9. Give an example of an unequal bilateral tolerance. ______________

10. Give an example of a unilateral tolerance. ____________________

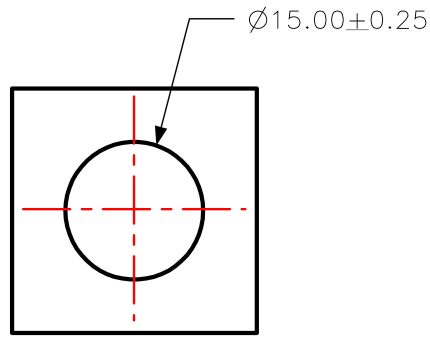
11. Define Maximum Material Condition (MMC).

    ____
    ____

12. What is the MMC of the feature shown below?



13. What is the MMC of the feature shown below? ________________



14. Define Least Material Condition (LMC).

____

____

____

15. What is the LMC of the feature shown in question 12? ____________

16. What is the LMC of the feature shown in question 13? ___________

17. List the three general groups related to the standard ANSI fits between mating parts.

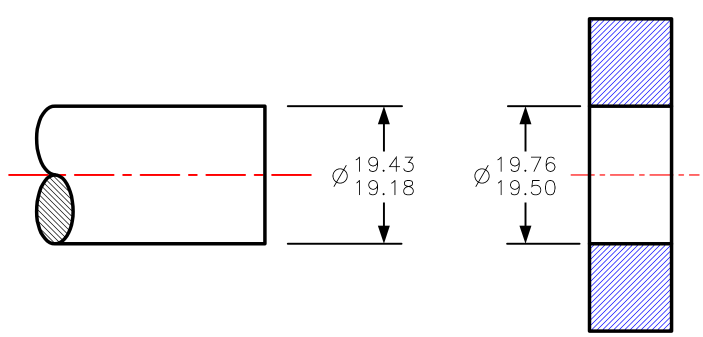
1) ____

2) ____

3) ____

18. Is the fit between the two parts shown below a clearance or a force fit?

____



2. Geometric Characteristics and Symbols

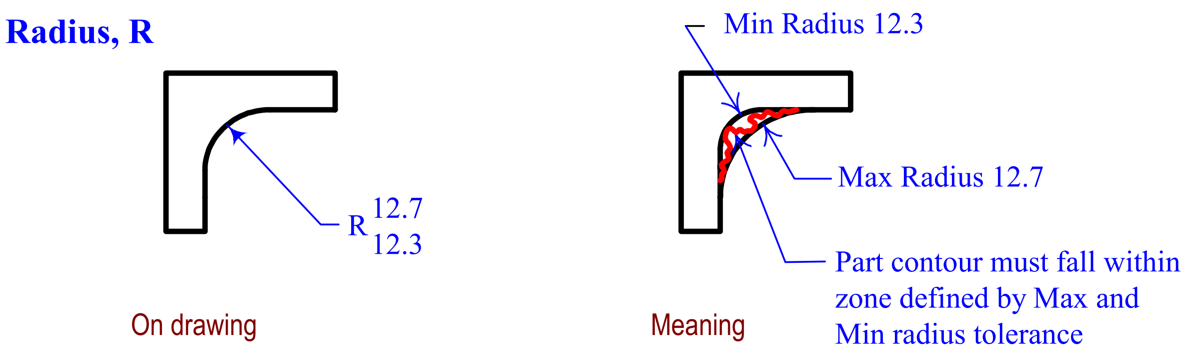
2.1. Symbol

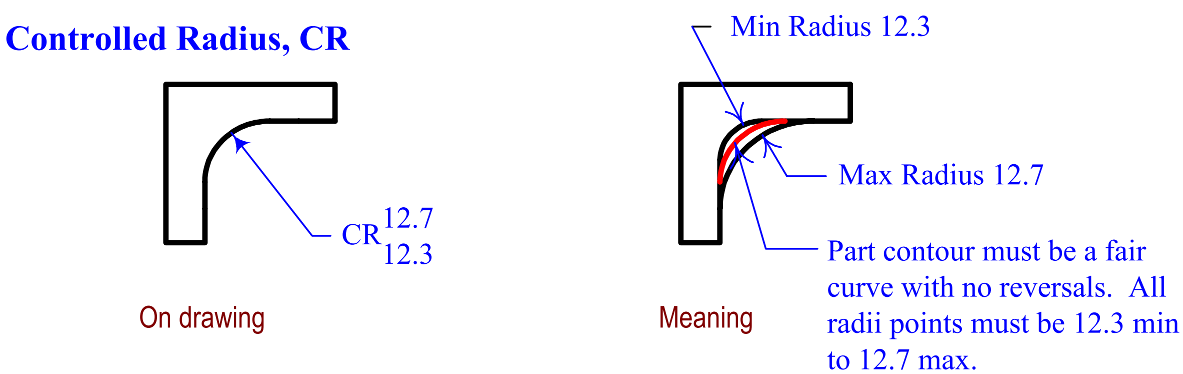
	Type of Tolerance	Characteristic	Symbol ASME Y14.5M-1994	Symbol ISO
For Individual Features	Form	Straightness	⏤	⏤
		Flatness	⏥	⏥
		Circularity	○	○
		Cylindricity	⌭	⌭
For Individual or Related Features	Profile	Profile of a Line	⌒	⌒
		Profile of a Surface	⌓	⌓
For Related Features	Orientation	Angularity	∠	∠
		Perpendicularity	⊥	⊥
		Parallelism	∥	∥
	Location	Position	⌖	⌖
		Concentricity	◎	◎
		Symmetry	⌯	⌯
	Runout	Circular Runout	↗	↗
		Total Runout	⌰	⌰

Symbol for:	ASME Y14.5M	ISO
At Maximum Material Condition	Ⓜ	Ⓜ
At Least Material Condition	Ⓛ	Ⓛ
Regardless of Feature Size	NONE	NONE
Projected Tolerance Zone	Ⓟ	Ⓟ
Diameter	∅	∅
Spherical Diameter	S∅	S∅
Square	□	□
Number of Places	X	X
Counterbore	⌴	⌴
Countersink	⌵	⌵
Depth	↧	↧
All Round	⌾	⌾
Between	↔	NONE
Arc Length	$\overset{\frown}{10}$	$\overset{\frown}{10}$
Radius	R	R
Spherical Radius	SR	SR
Controlled Radius	CR	NONE
Conical Taper	⌲	⌲
Slope	⌳	⌳
Tangent Plane	Ⓣ	Ⓣ
Free State	Ⓕ	Ⓕ
Statistical Tolerance	⟨ST⟩	NONE

Radius, Controlled Radius

There are two types of radii tolerance that can be applied, the radius and controlled radius. The radius (R) tolerance is for general applications. The controlled radius (CR) is used when it is necessary to place further restrictions on the shape of the radius, as in high stress applications.



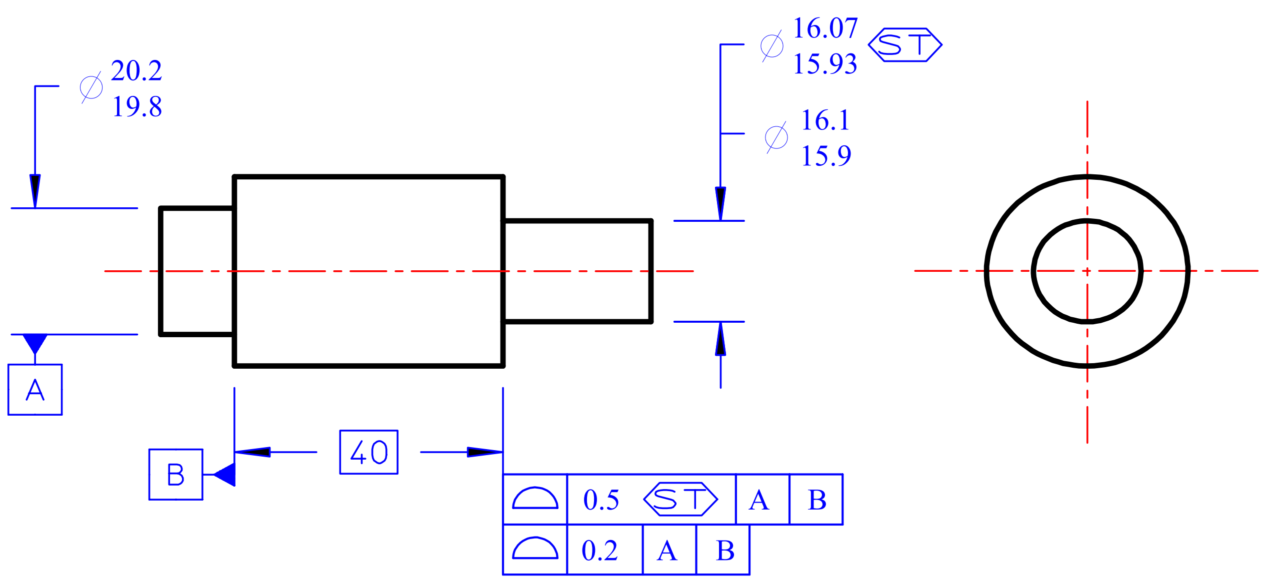


Statistical Tolerance

Often, tolerances are calculated on an arithmetic basis. Tolerances are assigned to individual features on a component by dividing the total assembly tolerance by the number of components and assigning a portion of this tolerance to each component. When tolerances are stacked up in this manner, the tolerance may become very restrictive or tight.

Statistical tolerancing is the assignment of tolerances to related components of an assembly on the basis of sound statistics. An example is, the assembly tolerance is equal to the square root of the sum of the squares of the individual tolerance.

Statistical Tolerance may be applied to features to increase tolerances and reduce manufacturing cost. To ensure compatibility, the larger tolerance identified by the statistical tolerance symbol may only be used where appropriate statistical process control will be used. A note such as the one shown below shall be placed on the drawing.



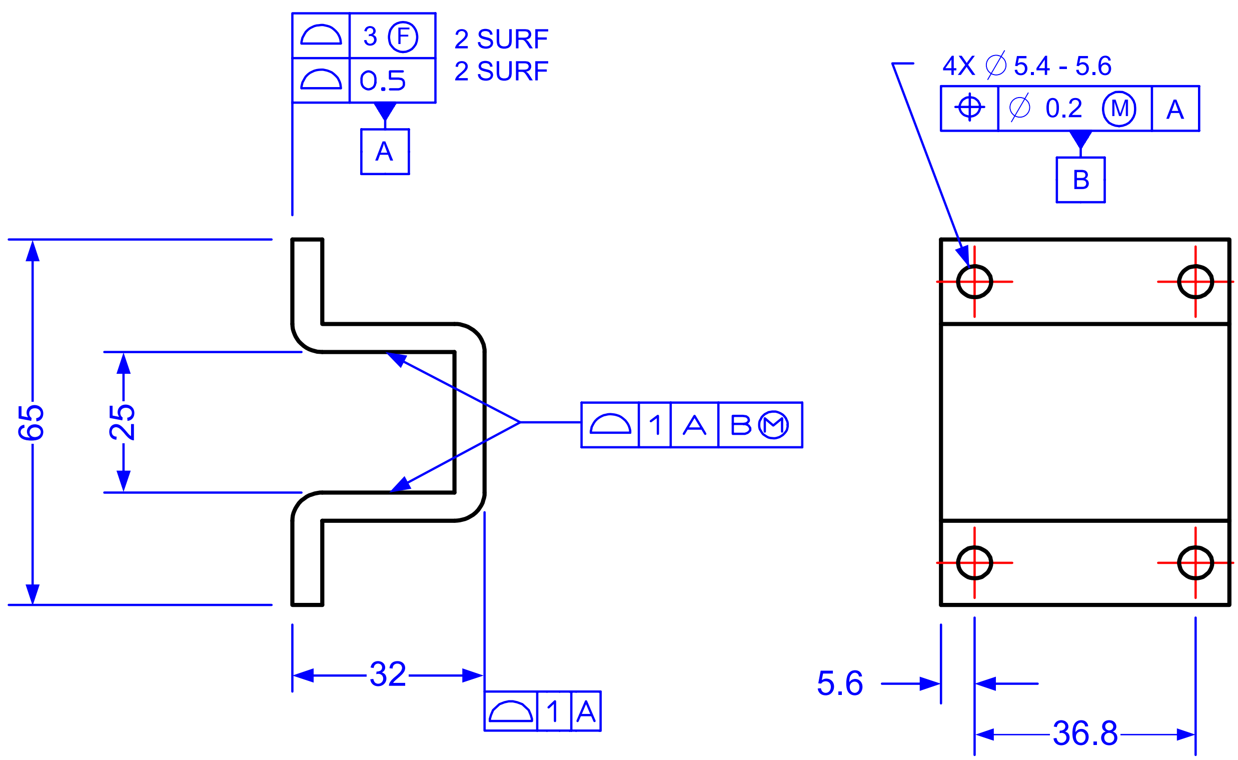
NOTE:
FEATURES INDENTIFIED AS STATISTICAL TOLERANCE ST SHALL BE PRODUCED WITH STATISTICAL PROCESS CONTROLS, OR TO THE MORE RESTRICTIVE ARITHMETIC LIMITS

Free State

Unless otherwise specified, all dimensioning and tolerancing applies in a free state condition with no restraint. Some parts, such as sheet metal, thin metal, plastics and rubber are non-rigid in nature. It may be necessary to specify design requirements on the part in a natural or free state as well as in a restrained condition. The restraint or force on the nonrighi9d parts is usually applied in such a manner to resemble or approximate the functional or mating requirements.

A note or specification on the drawing should explain how the part is restrained and the force required to facilitate the restraint. A sample note can be found on the drawing below.

The free state symbol means that dimensions and tolerances that have the free state symbol applied are checked in the free state and not in the restrained condition.



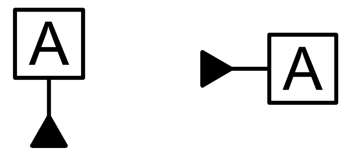
UNLESS OTHERWISE SPECIFIED, ALL UNTOLERANCED DIMENSIONS ARE BASIC. PART IS TO BE RESTRAINED ON DATUM A WITH 4 5M SCREWS

ASME Y14 Series	ISO Standards
Y14.2 – Lines & Lettering	3098
Y14.3 – Sections & Views	128
Y14.5 – Dimensioning & Tolerancing	129, 406, 1101, 1660, R1661, 2692, 5455, 5458, 5459, 7083, 8015, 10579; (also 14660-1 & 14660-2)
Y14.6 – Screw Thread Representation	6410-1, 6410-2, 6410-3
Y14.8 – Casting & Forgings	
Y14.36 – Surface Texture Symbols	1302

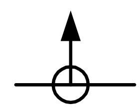
Basic Dimension
(theoretically exact dimension in ISO) $\boxed{65}$

Reference Dimension (auxiliary dimension in ISO) (68)

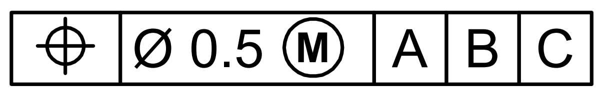
Datum Feature



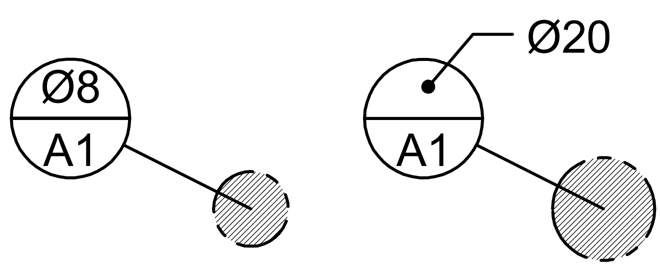
Dimension Origin



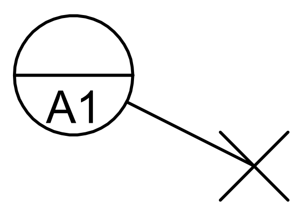
Feature Control Frame



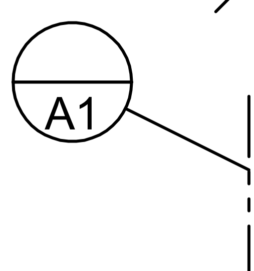
Datum Target Area



Datum Target Point



Datum Target Line



Geometric Charactieristic Overview

Datums	Type of Tolerance	Characteristic	Symbol	2D or 3D	Controls: Axis or Median Plane	Controls: Surface	Applicability of Feature Modifiers	Applicability of Datum Modifiers
Datums NOT allowed	Form	Straightness Line Element	⏤	2D		X	No	N/A
		Straightness Axis or Median Plane	⏤	3D	X		Yes	N/A
		Flatness	⏥	3D		X	No	N/A
		Circularity	○	2D		X	No	N/A
		Cylindricity	⌭	3D		X	No	N/A
Datums Required ①	Profile	Profile of a Line	⌒	2D		X	No	Yes if size features
		Profile of a Surface	⌓	3D		X	No	Yes if size features
	Location	Position	⌖	3D	X	③	Yes	Yes if size features
		Concentricity	◎	3D	②		No	No
		Symmetry	⌯	3D	②		No	No
Datums Requied	Orientation	Angularity	∠	3D ⑤	X	X	Yes if size features	Yes if size features
		Perpendicularity	⊥	3D ⑤	X	X	Yes if size features	Yes if size features
		Parallelism	∥	3D ⑤	X	X	Yes if size features	Yes if size features
	Runout ④	Circular Runout	↗	2D		X	No	No
		Total Runout	⌰	3D		X	No	No

① There are special case where position and profile may not require datums
② These characteristics control opposing median points
③ Can also control surface boundary
④ Can control form, orientation and location
⑤ These characteristics can be made 2D by writing "LINE ELEMENTS" under the feature control frame

2.2 Exercise

A dimensioning and tolerancing template is recommended for drawing proper symbols on this test and on future tests.

1. List the five basic types of geometric dimensioning and tolerancing symbols.
   a) ______________________________
   b) ______________________________
   c) ______________________________
   d) ______________________________
   e) ______________________________
2. Name the five types of geometric characteristic symbols.
   a) ______________________________
   b) ______________________________
   c) ______________________________
   d) ______________________________
   e) ______________________________

3. Name each of the following geometric characteristic symbols.

⏤	______________	◎	______________
⏥	______________	⌯	______________
○	______________	//	______________
⌭	______________	∠	______________
⌒	______________	⊥	______________
⌓	______________	↗	______________
⌖	______________	⌰	______________

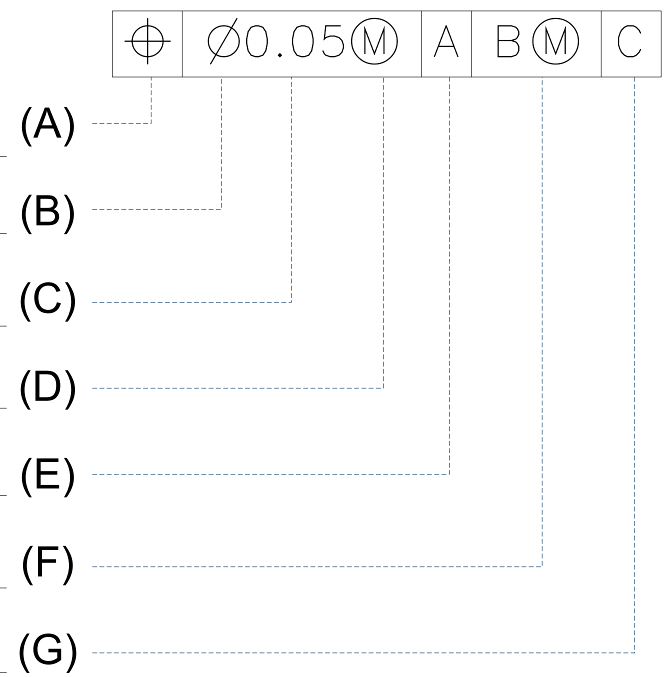
4. Any letter of the alphabet can be used to identify a datum except for ____, ____, or ____.

5. When may datum feature symbols be repeated on a drawing?

6. What information is placed in the lower half of the datum target symbol?

7. What information is placed in the top half of the datum target symbol?

8. Label the parts of the following feature control frame.



9. Completely define the term "basic dimension".

____

____

____

____

____

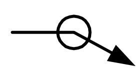
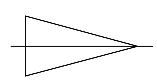
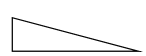
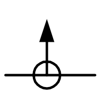
10. How are basic dimensions shown on a drawing?

____

____

____

11. Name the following symbols.

Ø	____________	◎	____________
R	____________	□	____________
SR	____________	(68)	____________
CR	____________	↧	____________
SØ	____________		____________
X	____________		____________
⌴	____________		____________
⌵	____________	⟨ST⟩	____________
	____________	$\boxed{65}$	____________

3. Datum

3.1. Datum Concepts

A datum is a theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location of geometric characteristics of features of a part are established. Datums are established by specified features or surfaces. Where orientation or position relationships are specified from a datum, the features involved are located with respect to this datum and not with respect to one another.

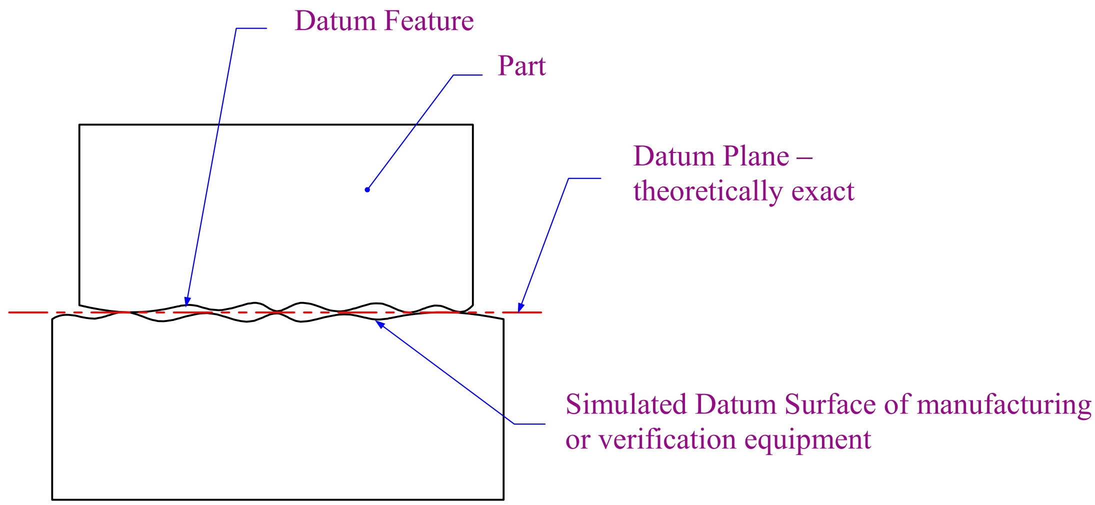
Every feature on a part can be considered a possible datum. That is, every feature shown on a drawing depicts a theoretically exact geometric shape as specified by the design requirements. However, a feature normally has no practical meaning as a datum unless it is actually used for some functional relationship between features. Thus a datum appearing on an engineering drawing can be considered to have a dual nature: it is (1) a “construction” datum, which is geometrically exact representation of any part feature, and (2) a “relationship” datum, which is any feature used as a basis for a functional relationship with other features on the part. Since the datum concept is used to establish relationships, the “relationship” datum is the only type used on engineering drawings.

By the above definition, a datum on an engineering drawing is always assumed to be “perfect”. However, since perfect parts cannot be produced, a datum on a physically produced part is assumed to exist in the contact of the actual feature surface with precise manufacturing or inspection equipment such as machine tables, surface plates, gage pins, etc. These are called datum simulators which create simulated datum planes, axes, etc., and, while not perfectly true, are usually of such high quality that they adequately simulate true references. This contact of the actual feature with precise equipment is also assumed to simulate functional contact with a mating part surface.

Datum feature: The actual surface of the part.

Simulated datum: The plane established by the inspection equipment such as a surface plate or inspection table.

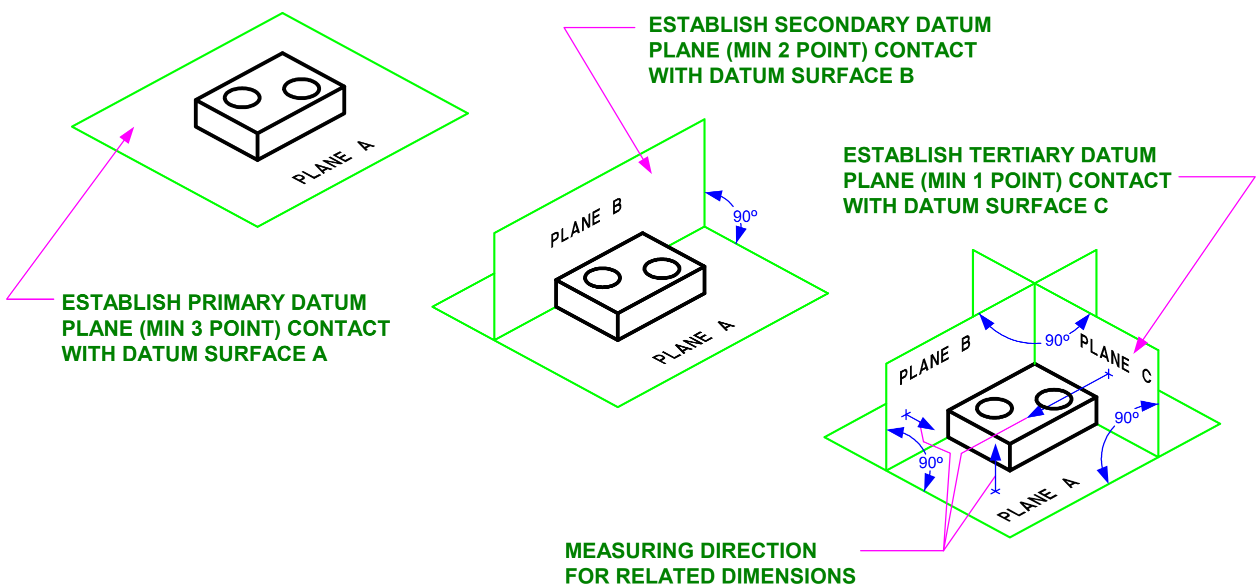
Datum plane: The theoretically exact plane established by the true geometric counterpart of the datum feature.

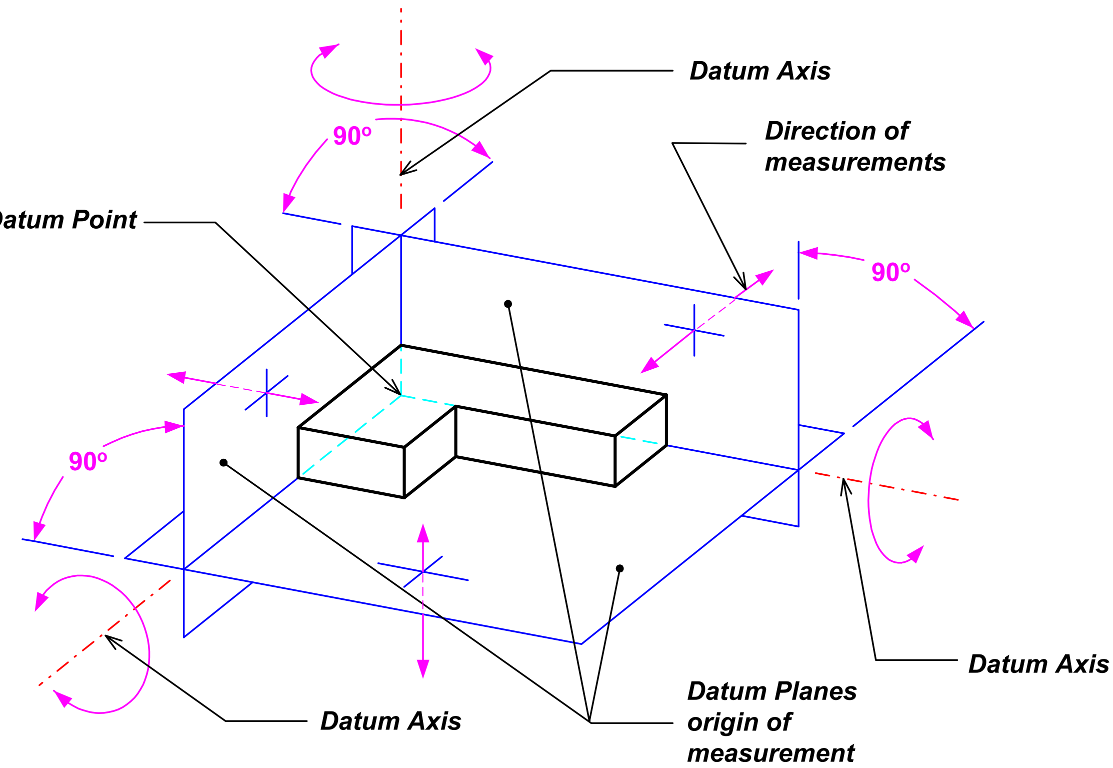


3.2. Establishing Datum Planes

Datum features are selected based on their importance to the design of the part.

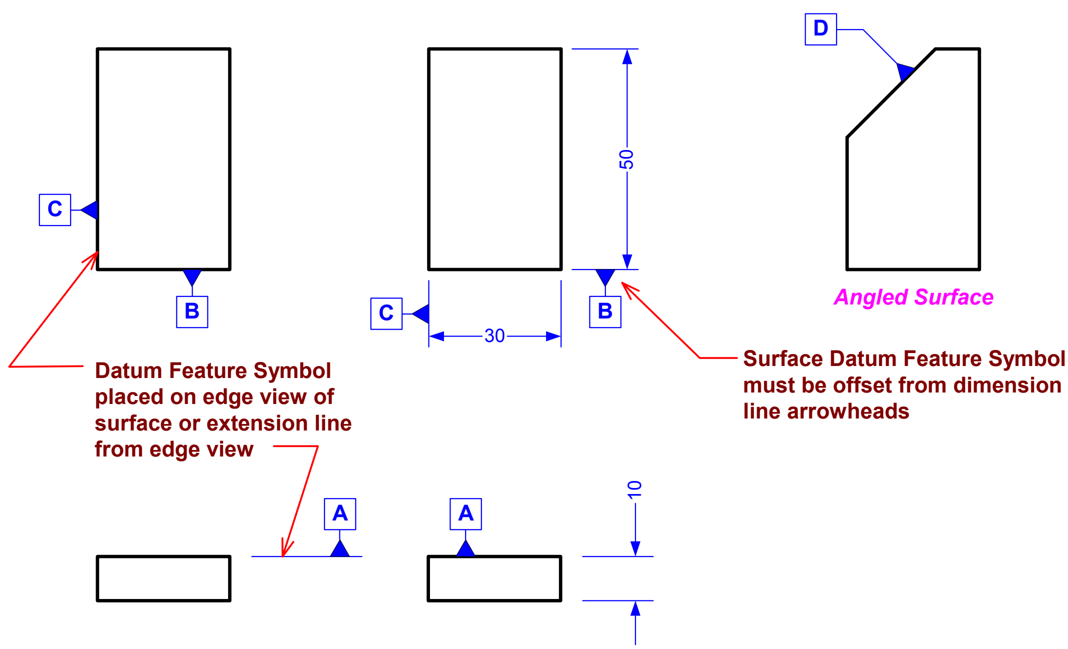
Generally three datum features are selected that are perpendicular to each other. These three datums are called the datum reference frame. The datums that make up the datum reference frame are referred to as the primary datum, secondary datum, and tertiary datum. As their names imply, the primary datum is the most important, followed by the other two in order of importance.





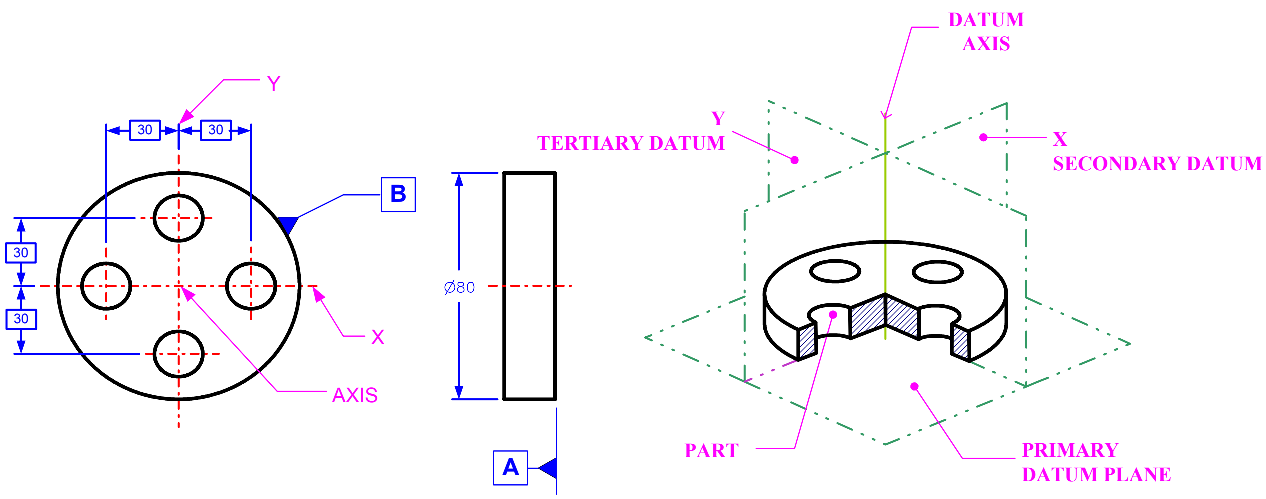
3.3. Datum Identification

When a surface is used to establish a datum plane on a part, the datum feature symbol is placed on the edge view of the surface or on an extension line in the view where the surface appears as a line. A leader line may also be used to connect the datum feature symbol to the view in some applications.

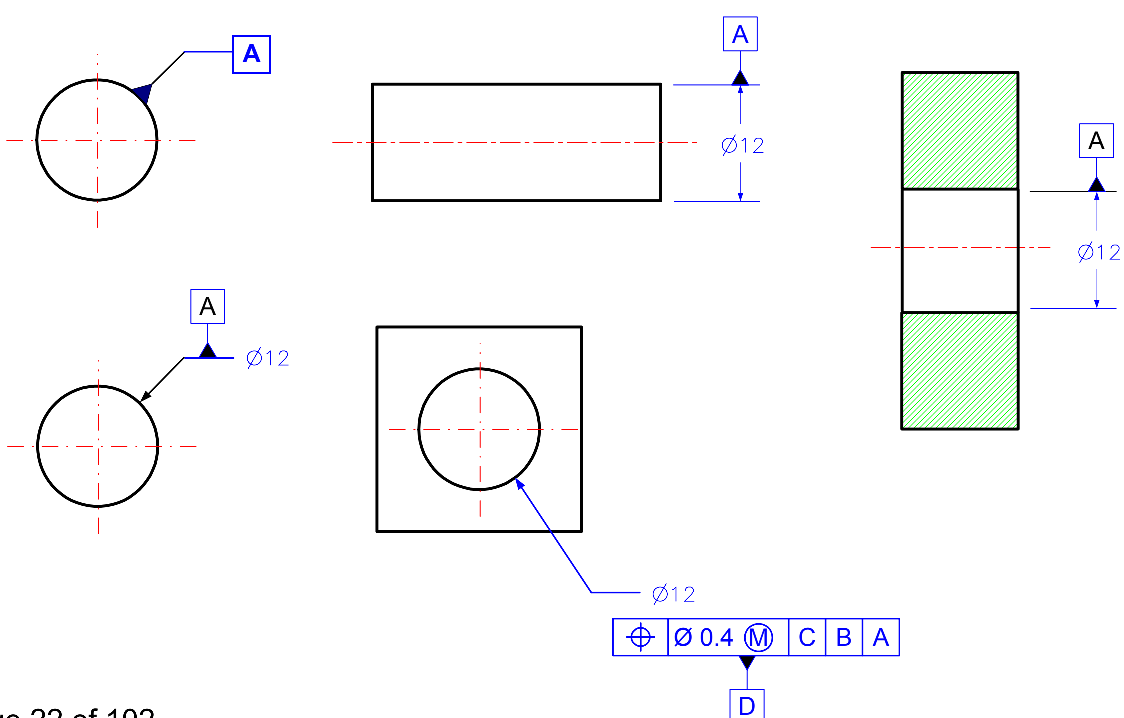


3.4. Datum Axis

A cylindrical object may be a datum feature. When the cylindrical datum feature is used, the centre axis is known as the datum axis. There are two theoretical planes intersecting at 90º. These planes are represented by the centrelines of the drawing. Where these planes intersect is referred to as the datum axis. The datum axis is the origin for related dimensions, while the X and Y planes indicate the direction of measurement. A datum plane is added to the end of the object to establish the datum frame.

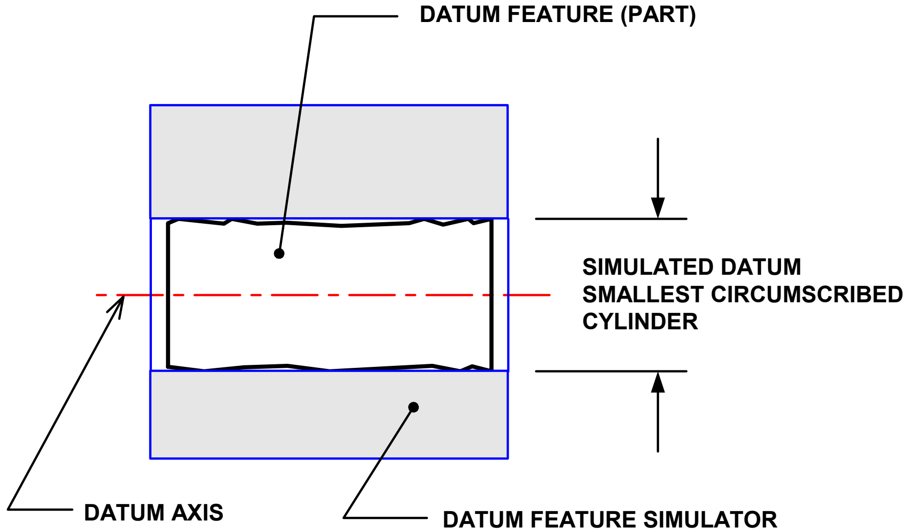


Placement of the Datum Feature Symbol for a Datum Axis

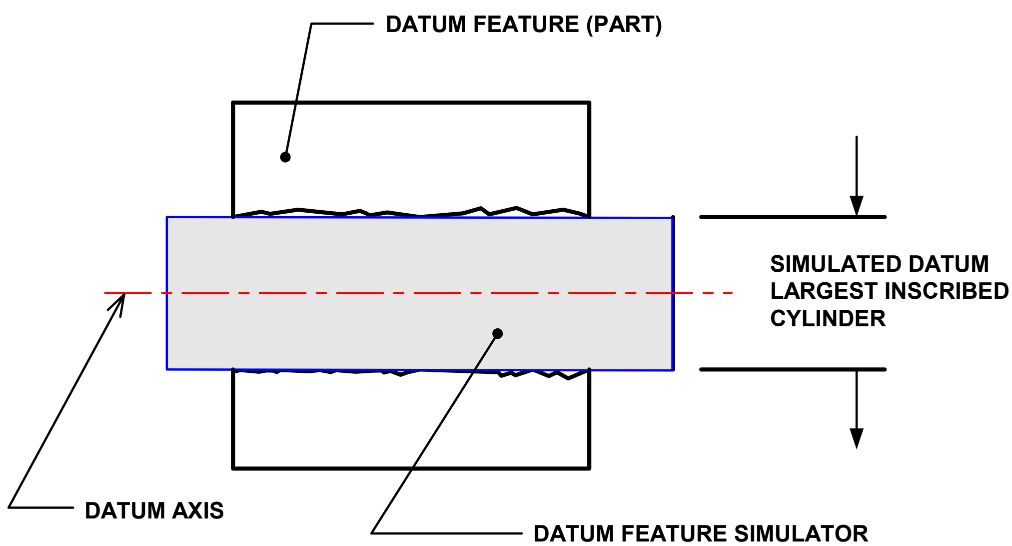


Simulated datum axis

The simulated datum axis is the axis of a perfect cylindrical inspection device that contacts the datum feature surface. For an external datum feature, the inspection device is the smallest (MMC) circumscribed cylinder. The inspection device for an internal datum feature is the largest (MMC) inscribed cylinder.



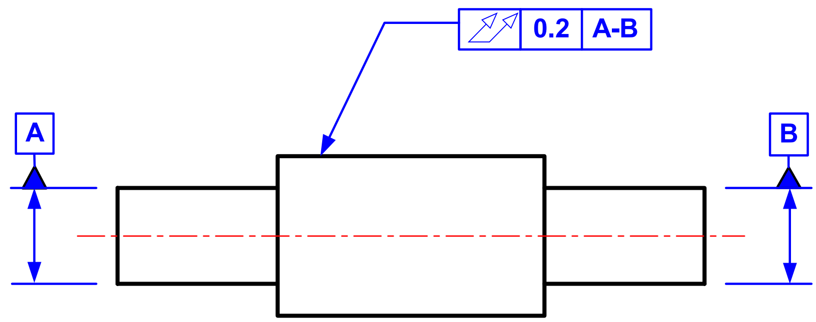
Simulated datum axis for an external datum feature



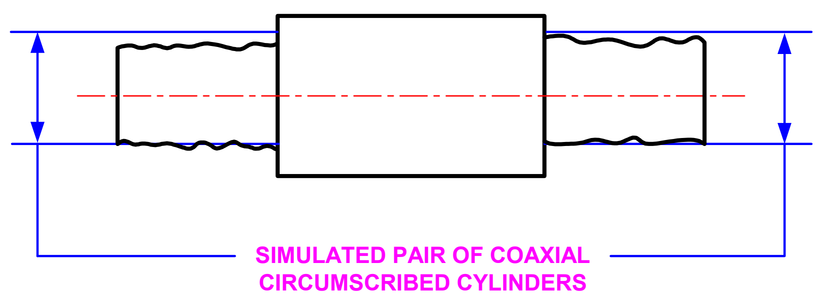
Simulated datum axis for an internal datum feature

3.5. Coaxial Datum Features

Coaxial means two or more cylindrical shapes that share a common axis. Coaxial datum features exist when a single datum axis is established by two datum features that are coaxial. When more than one datum feature is used to establish a single datum, the datum reference letters are separated by a dash and placed in one compartment of the feature control frame. These datum reference letters are of equal importance and may be placed in any order.



THE DRAWING



THE MEANING

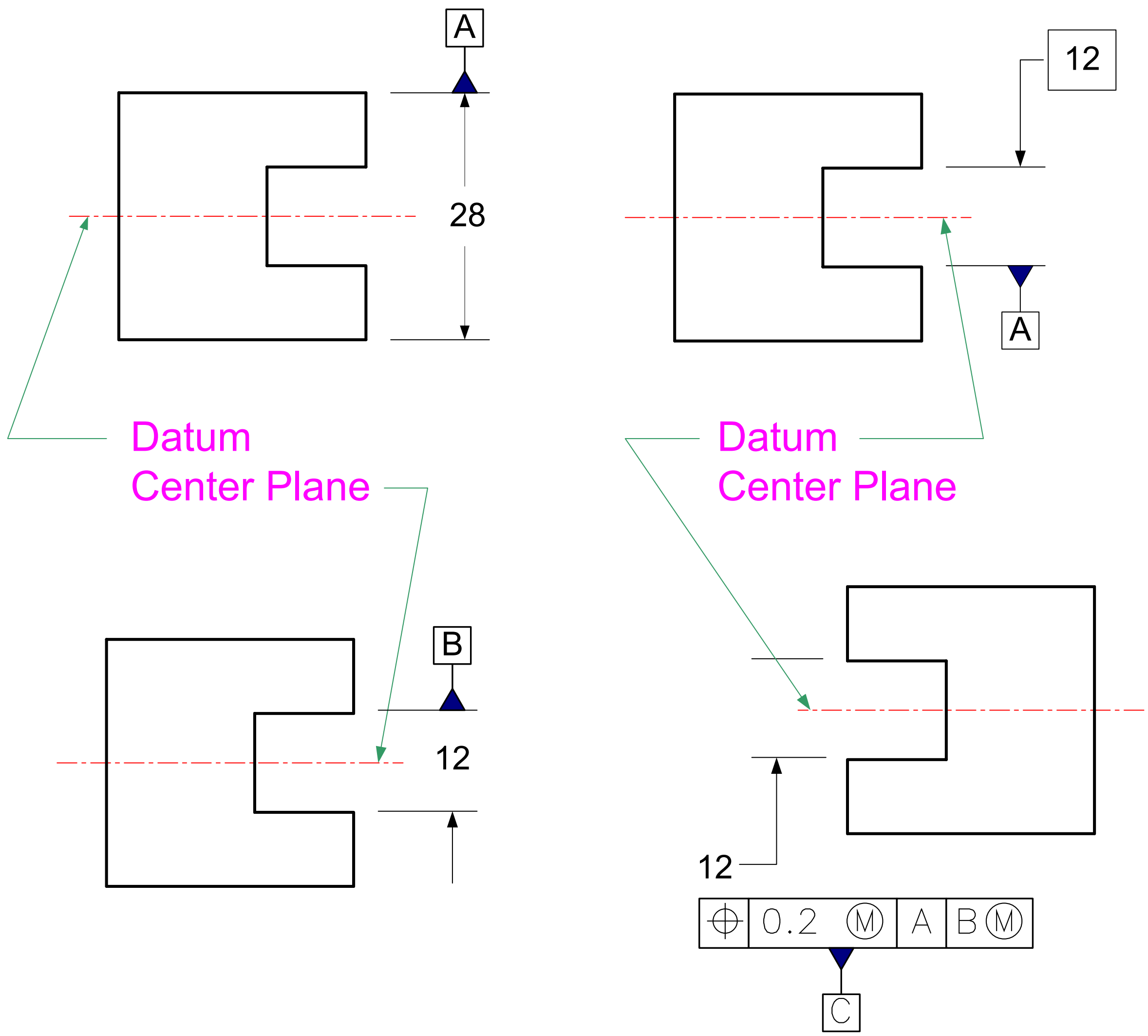
3.6. Datum Axis of Screw Threads, Gears, and Splines

When a screw thread is used as a datum axis, the datum axis is established from the pitch cylinder unless otherwise specified. If another feature of the screw thread is desired, then note “MAJOR DIA” or “MINOR DIA” is placed next to the datum feature symbol.

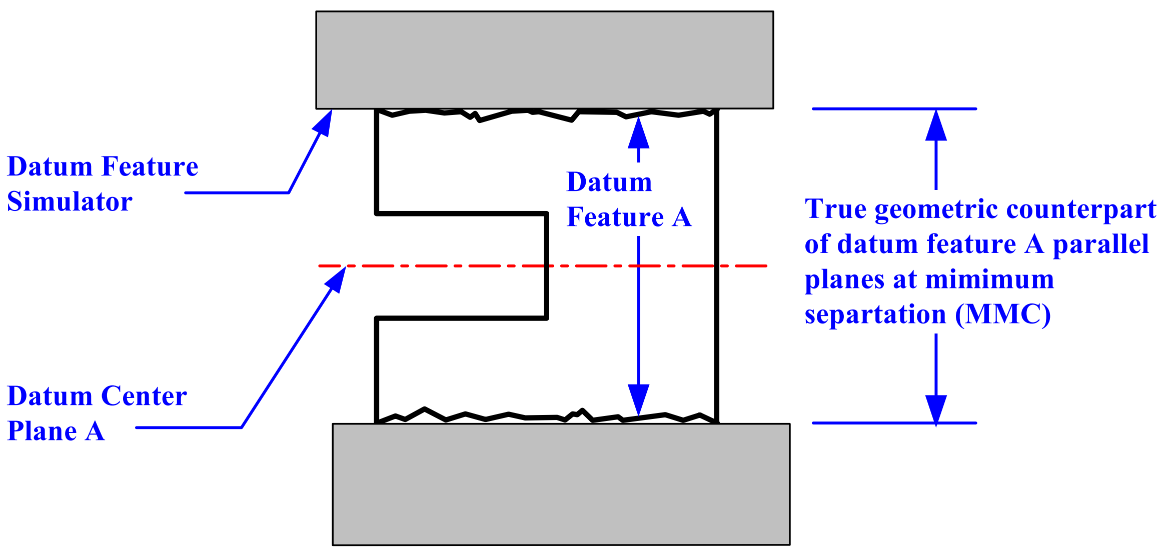
A specific feature such as the major diameter should be identified when a gear or spline is used as a datum axis. When this is done, the note “MAJOR DIA”, “MINOR DIA”, or “PITCH DIA” is placed next to the datum feature symbol as appropriate. The use of a screw thread, gear, or spline should be avoided for use as a datum axis unless necessary.

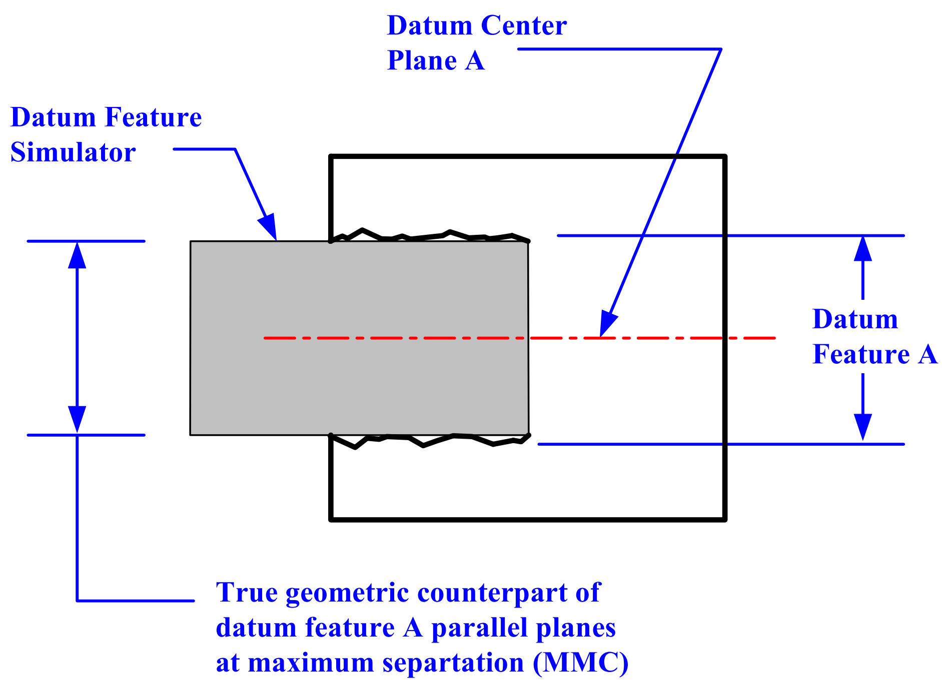
3.7. Datum Center Plane

Elements on a rectangular shaped symmetrical part or feature may be located and dimensioned in relationship to a datum centre plane. The representation and related meaning of datum center plane symbols are as shown in the following.



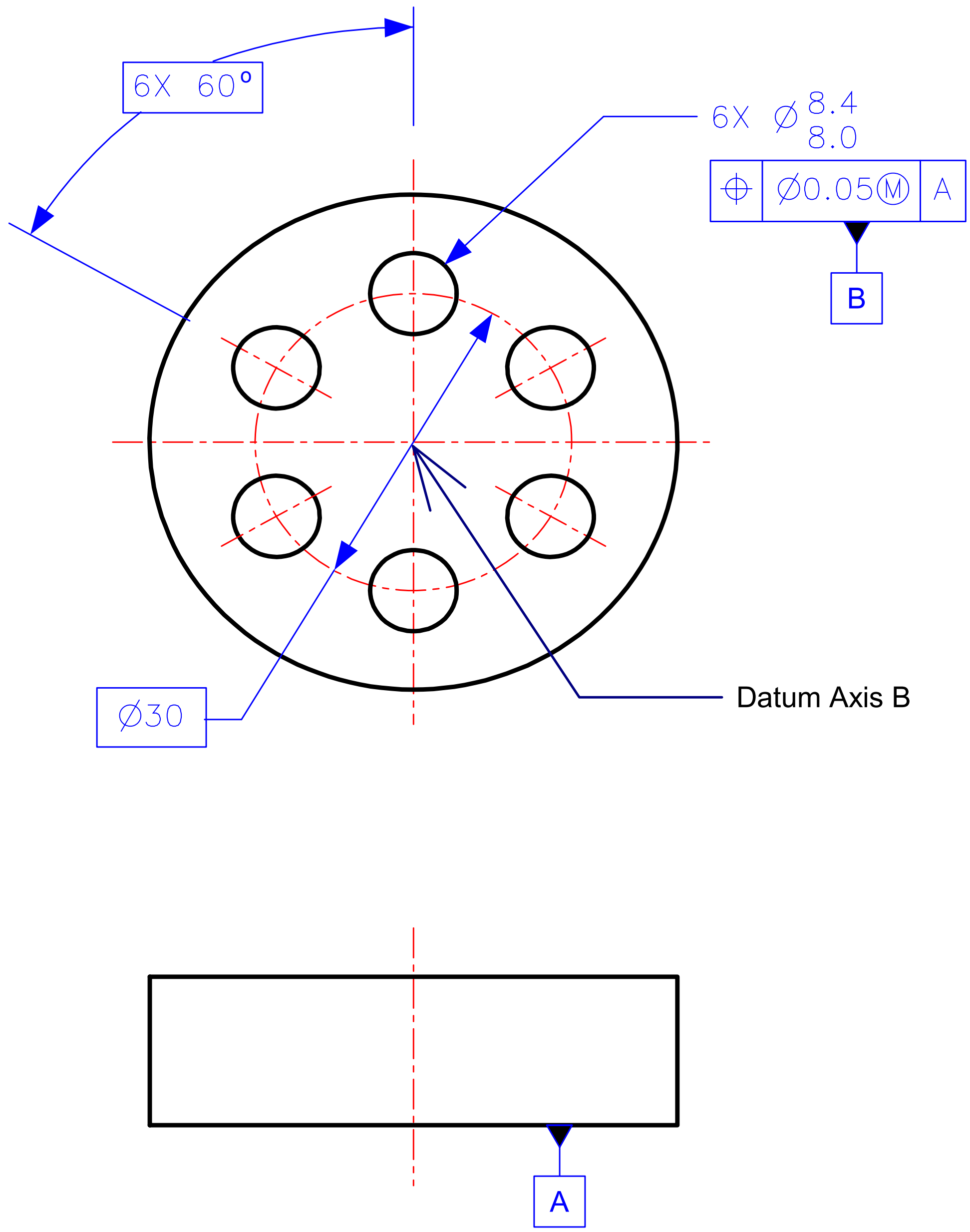
The simulated datum centre plane is the centre plane of a perfect rectangular inspection device that contacts the datum feature surface. For an external datum feature the datum centre plane is established by two parallel planes at minimum (MMC) separation. For an internal datum feature, the datum centre plane is established by two parallel planes at maximum (MMC) separation.





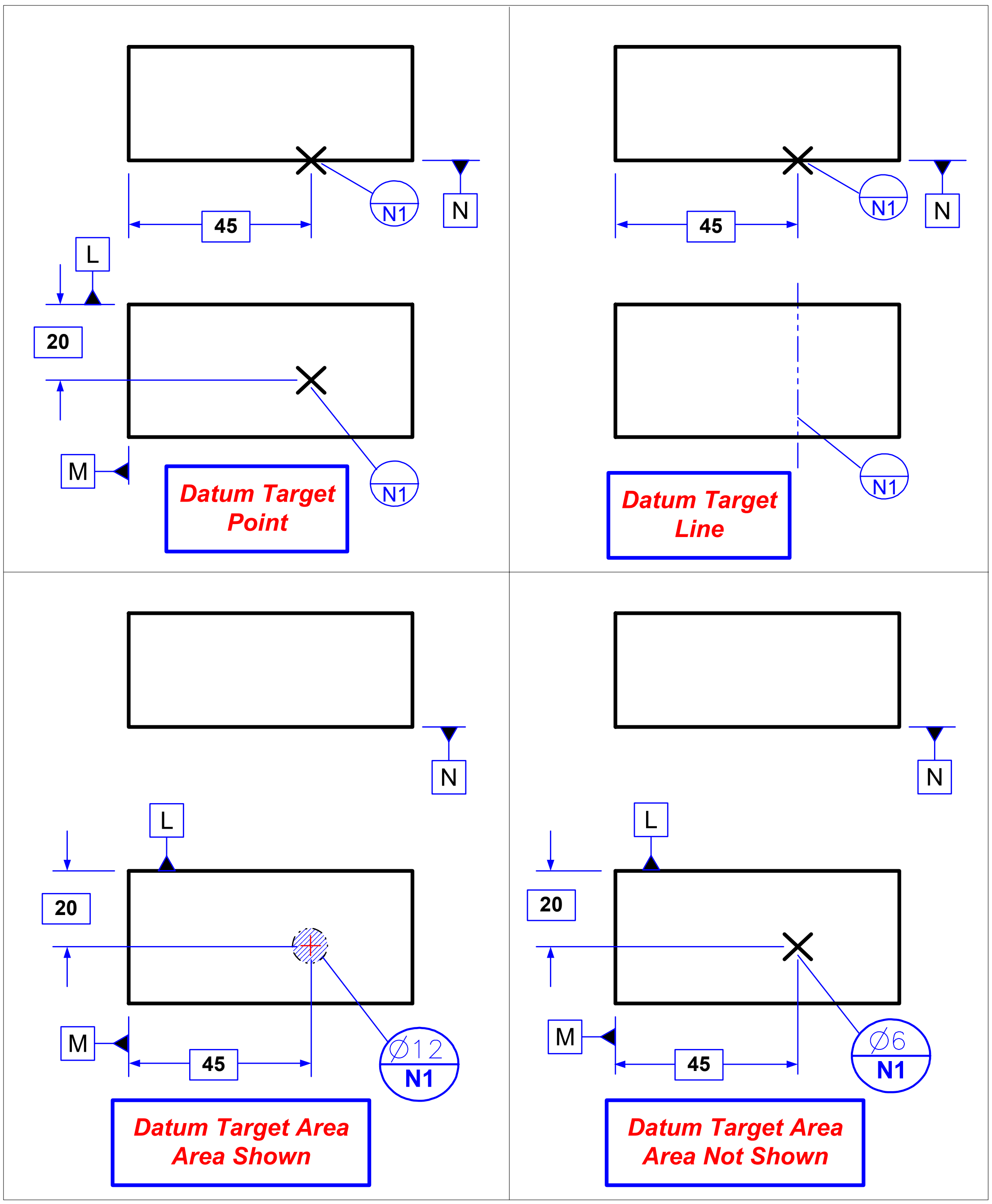
3.8. Pattern of Holes as a Datum

The center of a pattern of features, such as the holes in the part may be specified as the datum axis when the datum feature symbol is placed under, and attached to, the middle of the feature control frame. In this application, the datum axis is the center of the holes as a group.

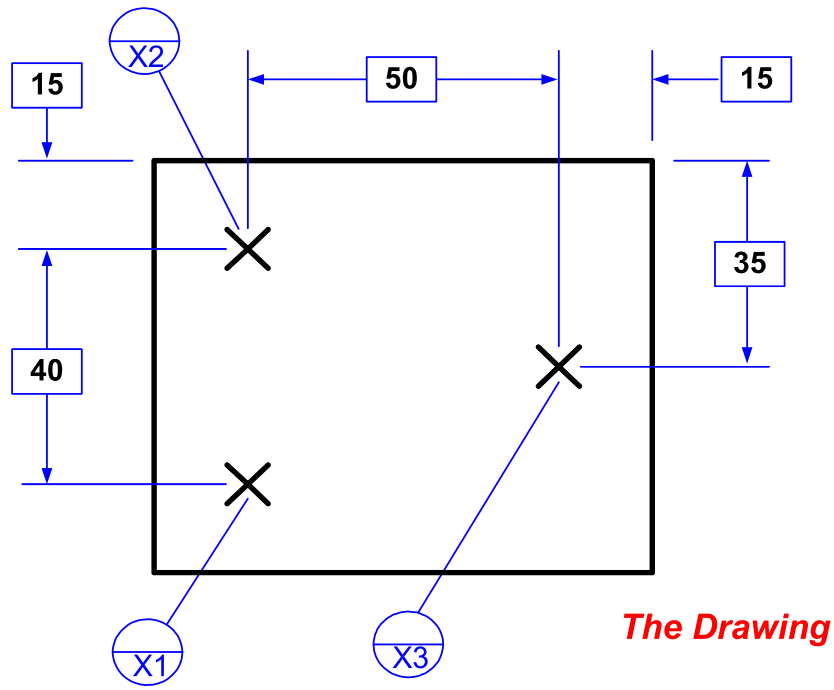


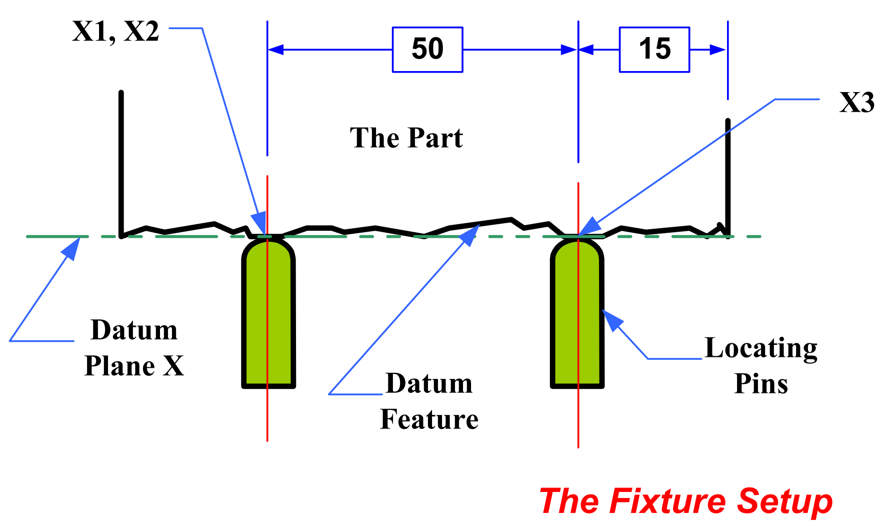
3.9. Datum Targets

In many situations it is not possible to establish an entire surface, or entire surfaces, as datums. When this happens, then datum targets may be used to establish datum planes. This procedure is especially useful on parts with surface or contour irregularities, such as some sheet metal, sand cast, or forged parts that are subject to bowing or warpage. This method can also be applied to weldments where heat may cause warpage. Datum targets are designated points, lines, or surface areas that are used to establish the datum reference frame.

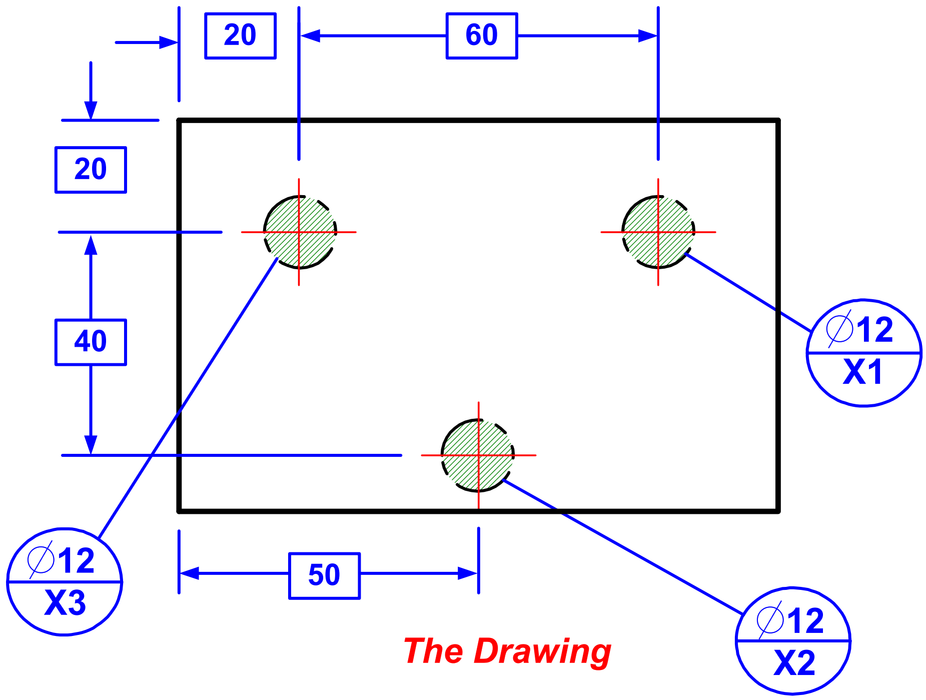


When **datum target points** are used on a drawing to identify a datum plane, the datum plane is established by locating pins at the datum tangent points. The locating pins are rounded or pointed standard tooling hardware.

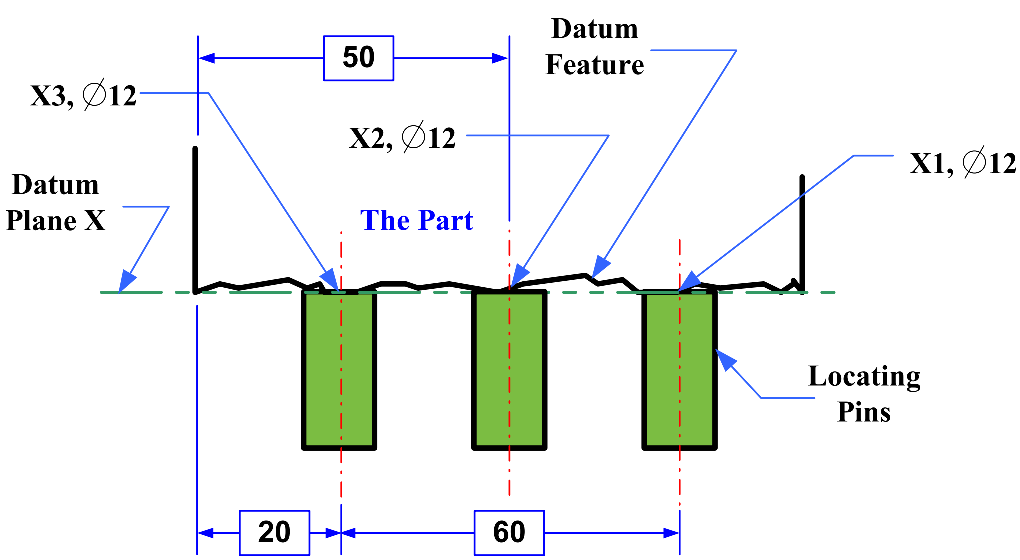




Areas of contact may also be used to establish datums. The shape of the **datum target area** is outlined by phantom lines with section lines through the area. Circular areas are dimensioned with basic or tolerance dimensions to located the center. The diameter of the target area is provided in the upper half of the datum target symbol or with a leader and dot pointing to the upper half. The locating pins for target areas are flat end tooling pins with the pin diameter equal to the specified size of the target area.

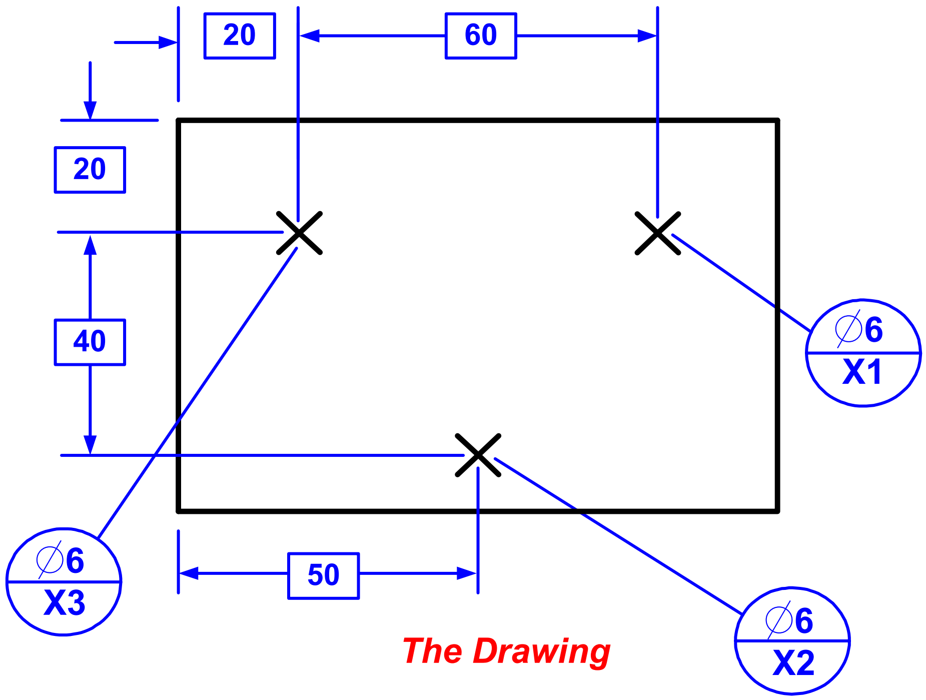


The Drawing

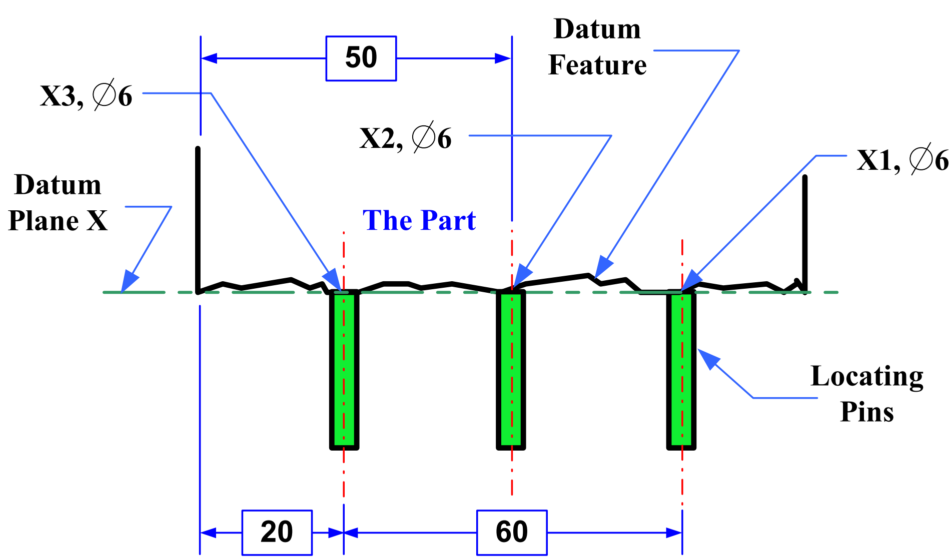


The Fixture Setup

When the area is too small to accurately or clearly display on a drawing, a datum target point is used at the center location. The top half of the datum target symbol identifies the diameter of the target area.

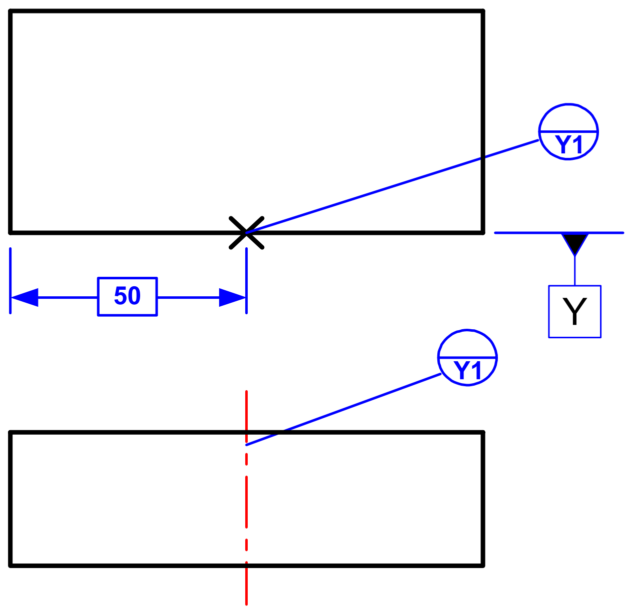


The Drawing

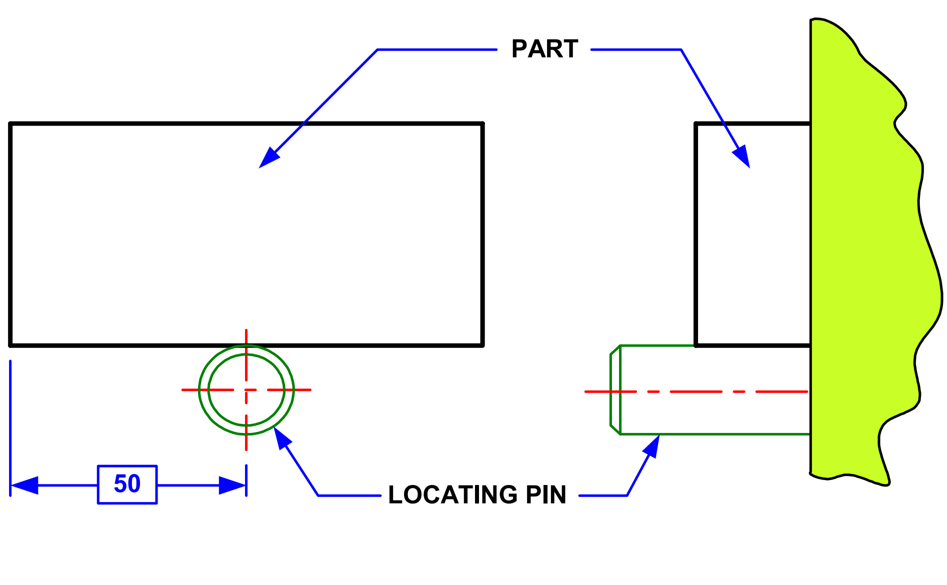


The Fixture Setup

A **datum target line** is indicated by the target point symbol “X” on the edge view of the surface and by a phantom line on the surface view. If the locating pins are cylindrical, then the datum target line is along the tangency where the pins meet the part. The pins may also be knife-edged. A surface is often placed at 90º to the pin to create the datum reference frame.



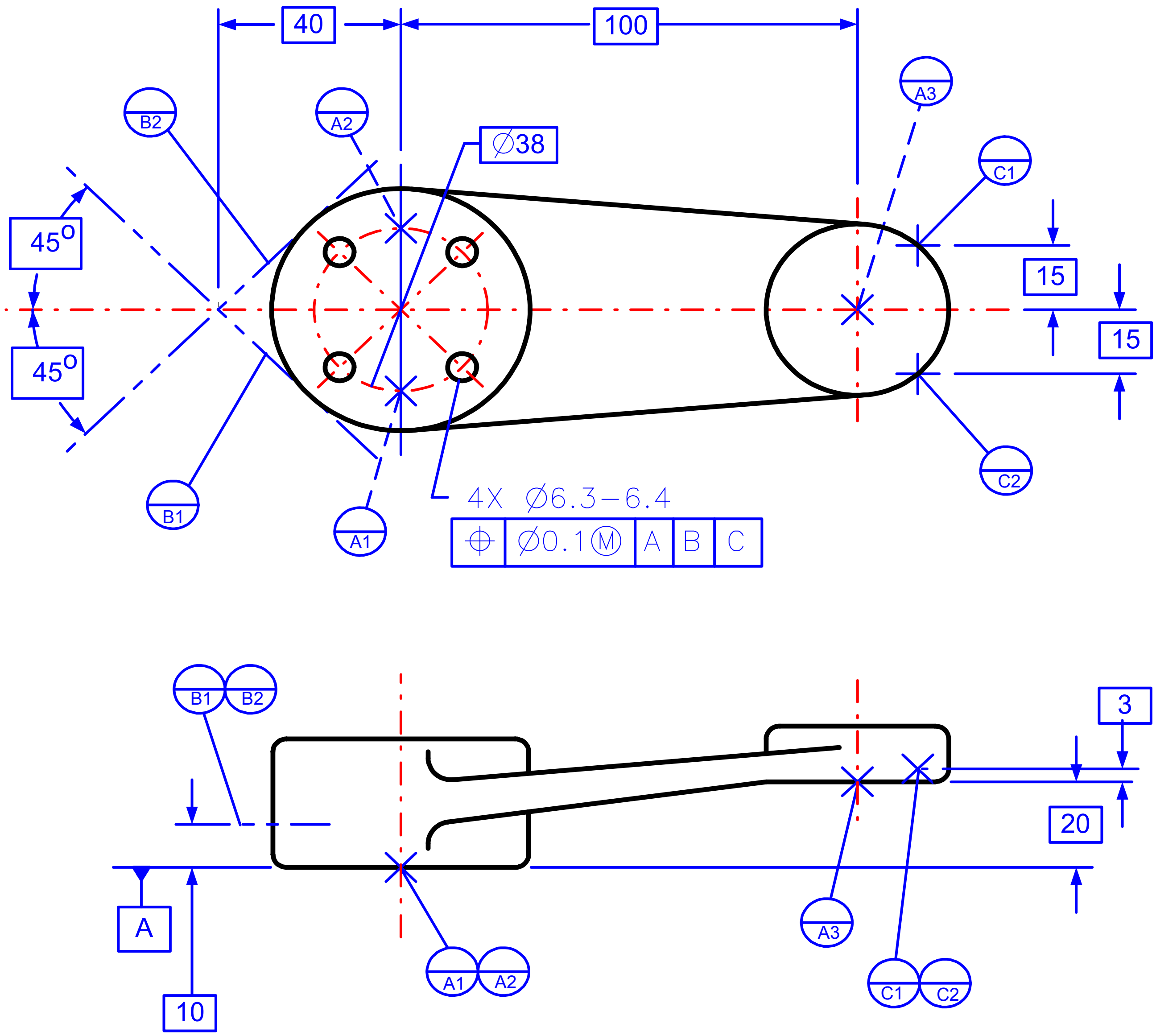
The Drawing



The Fixture Setup

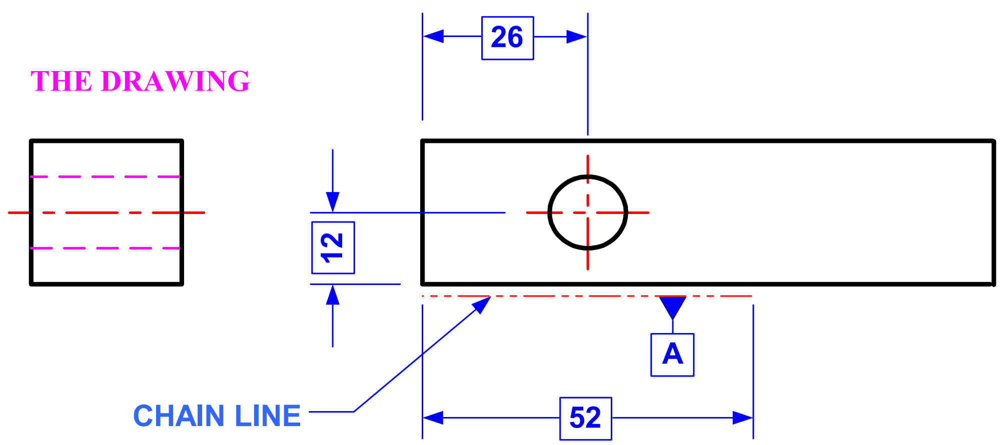
Example 1

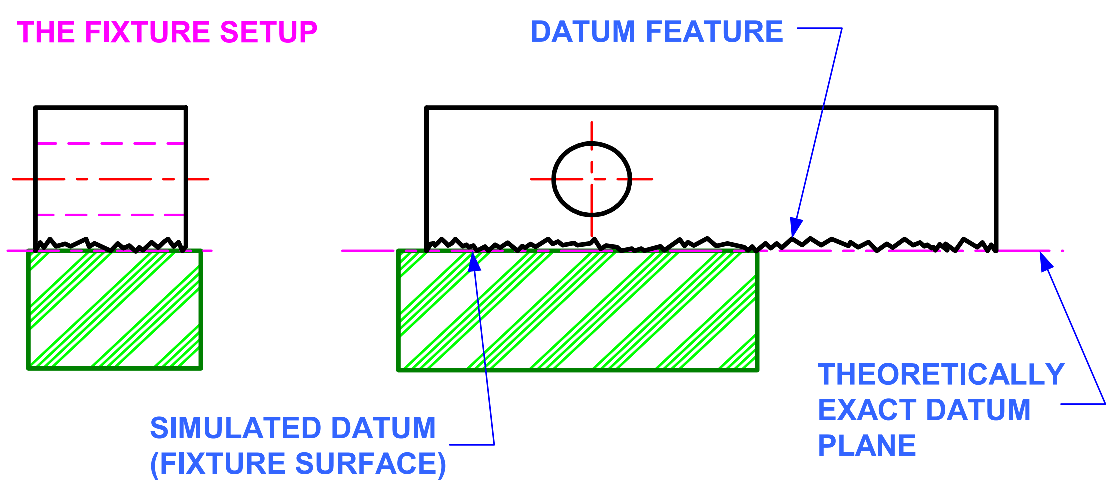
From ASME Y14.5M-1994, p78



3.10. Partial Datum Surface

A portion of a surface may be used as a datum. For example, this may be done when a part has a hole or group of holes at one end where it may not be necessary to establish the entire surface as a datum to effectively locate the features. This may be accomplished on a drawing using a chain line dimensioned with basic dimensions to show the location and extent of the partial datum surface. The datum feature symbol is attached to the chain line. The datum plane is then established at the location of the chain line.





3.11. Exercise

1. List the 3 primary items that are considered Datum features on an object or part.

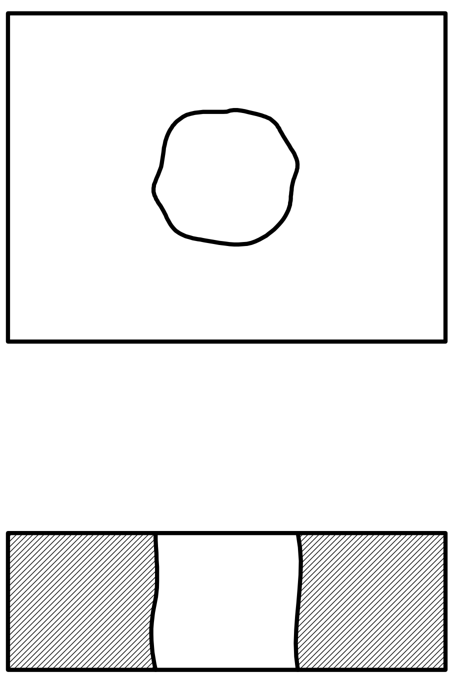
   ____

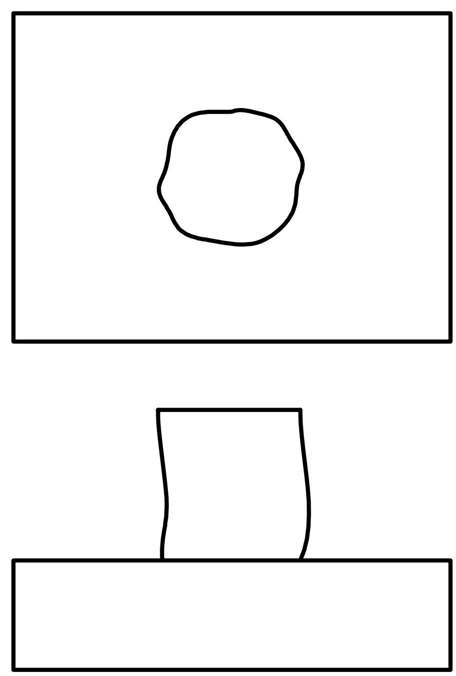
2. Draw the symbol that is known as the Datum Reference Symbol.

3. The primary datum requires a minimum of _________ points.
   The secondary datum requires a minimum of _________ points.
   The tertiary datum requires a minimum of _________ points.

4. Below are examples of a hole (Figure 1) and a pin (Figure 2) that will be identified as datum features. Sketch on the figure and explain how the datum axis for each would be determined.

5.





On the following, Figure 3, identify the: datum feature, part, simulated datum, and the datum plane.

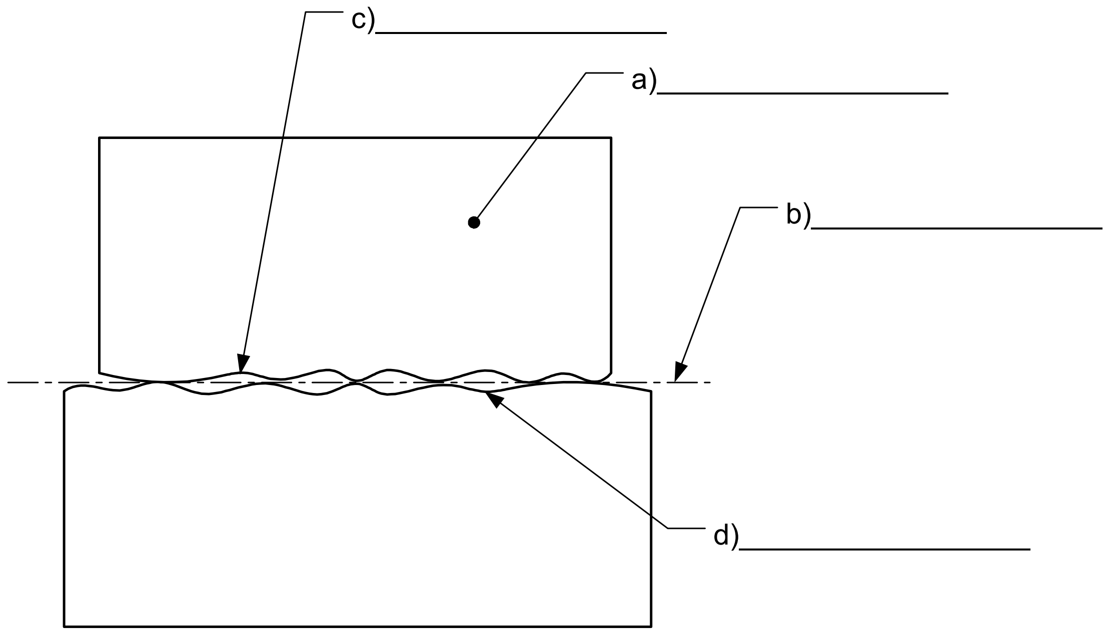


Figure 3

On the following exercises, using the drawing provided on next page (Figure 4),

6. Specify the left hand edge as Datum A.

7. Specify the ∅12 hole as Datum D.

8. Specify the right hand edge as Datum G.

9. On the bottom surface, specify a partial Datum K over a distance of 40 from the right edge of the part.

10. Specify the right hand edge of the 13 slot as Datum M.

11. Specify the 13 slot as Datum P.

12. Specify the two ∅6 holes as Datum S.

13. Datum features may be either features of size or features without size. On the drawing, identify features of size by placing a 'Z' next to them, and identify the features without size by placing an 'x' next to them.

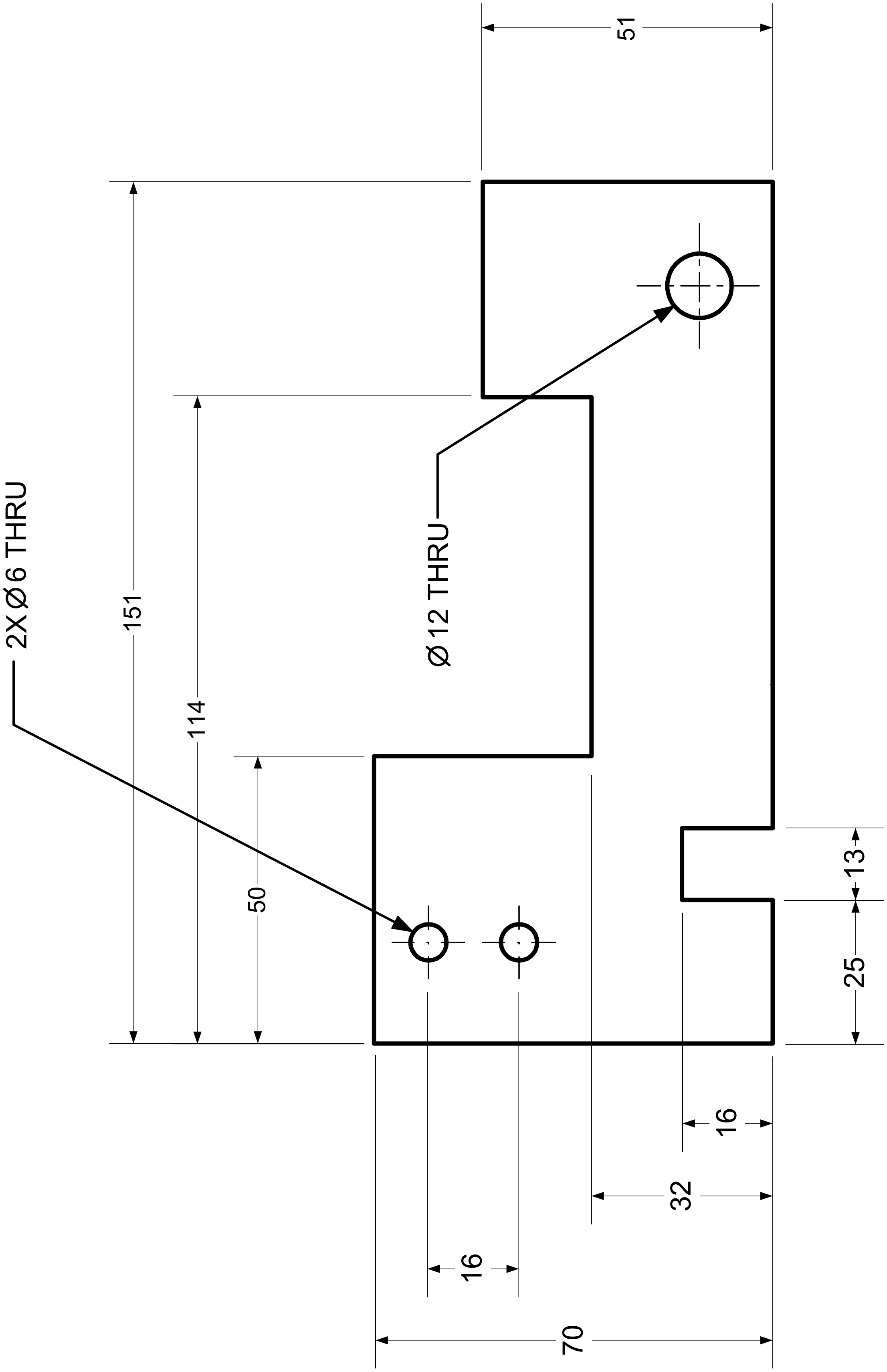
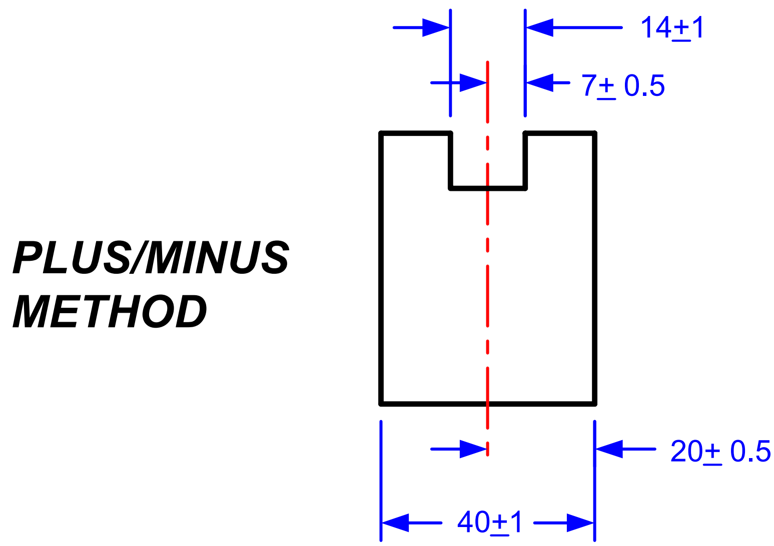
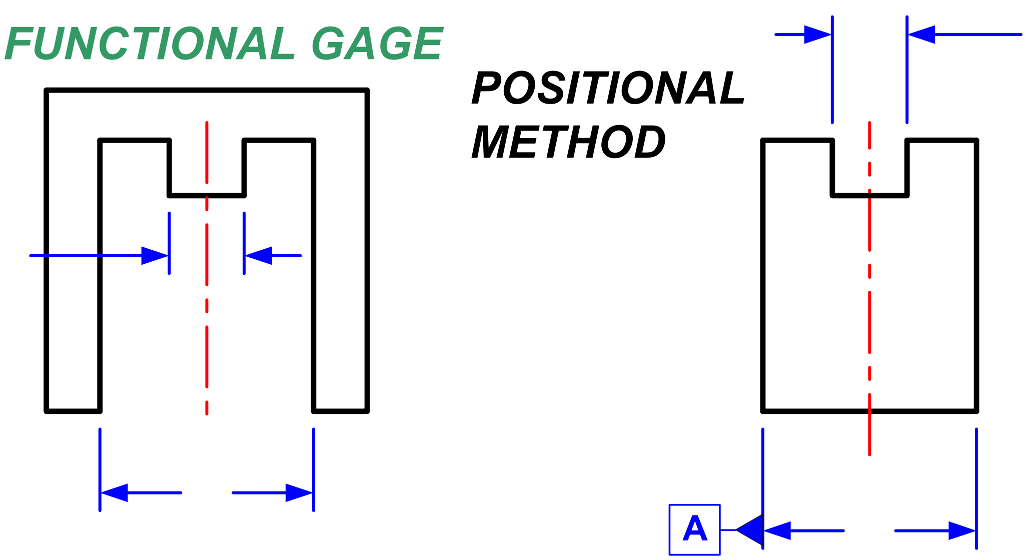


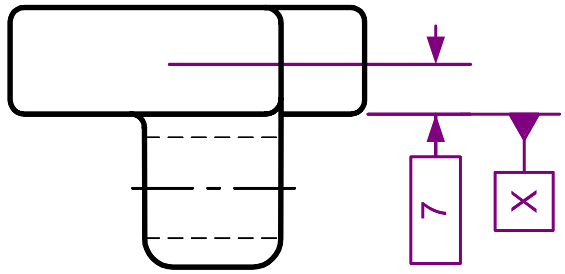
Figure 4

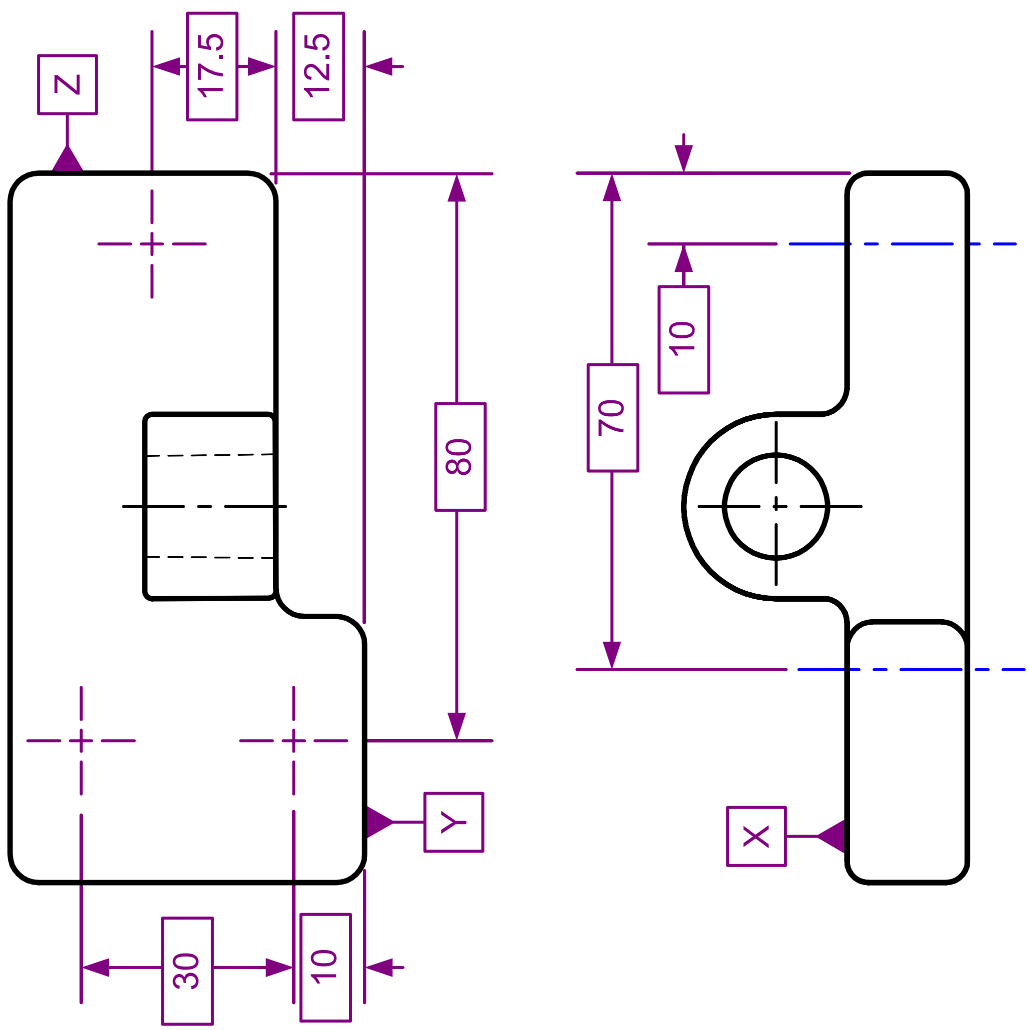
14. What is the relationship between the center plane of the slot and the center plane of the part? What is the total location tolerance that the center plane of the slot vary from the center plane of the part? Is design intent clear?





The picture below represents a cast part. It was determined that the part should have datum targets specified to standardise the initial machining set-up. On the drawing next page, sketch the datum targets in proper format as you would expect to see them on an engineering drawing. Surface X should have three ∅10 target pads, Surface Y should have two targets lines of contact and Surface Z should have one point of contact. Arrange these targets on the indicated surfaces to your preference. Show all basic dimensions and just estimate the distances.


7
X

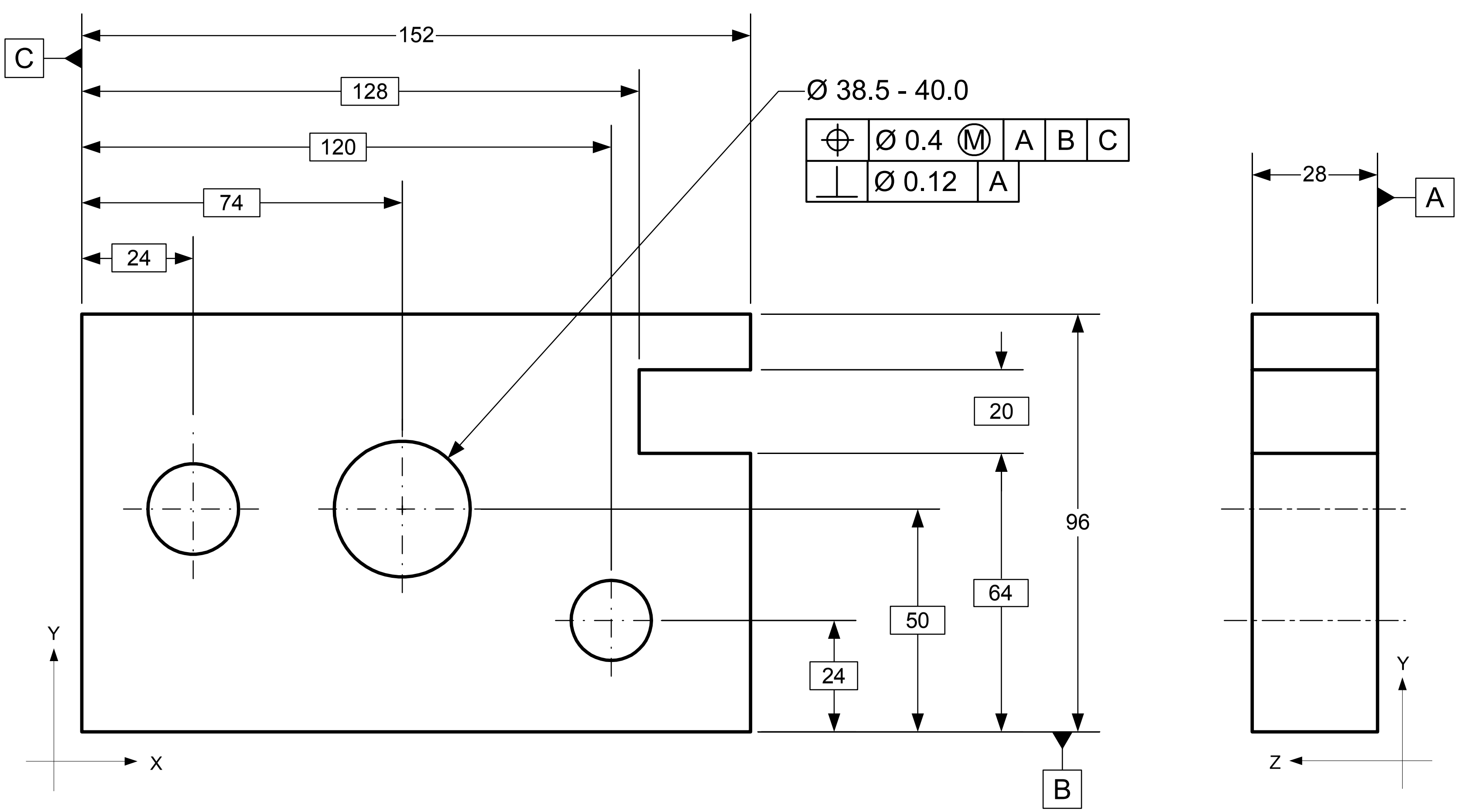

Z
17.5
12.5
80
70
10
Y
X
30
10

Understanding Datum Reference Frame Application (DRF)

Example 1

Degrees of Freedom Matrix

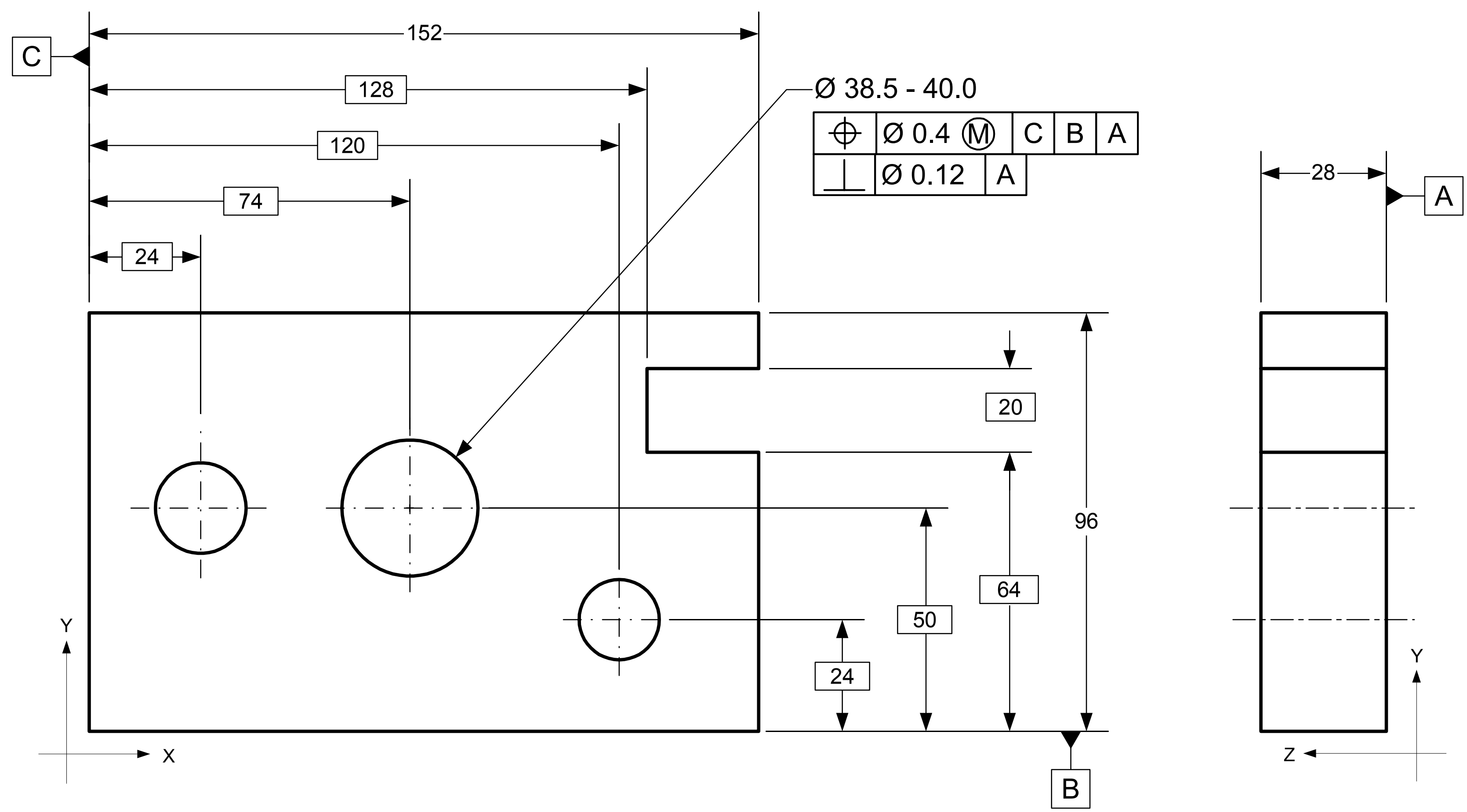
Datum Features	Rx	Ry	Rz	Tx	Ty	Tz



Understanding Datum Reference Frame Application (DRF)
Example 2

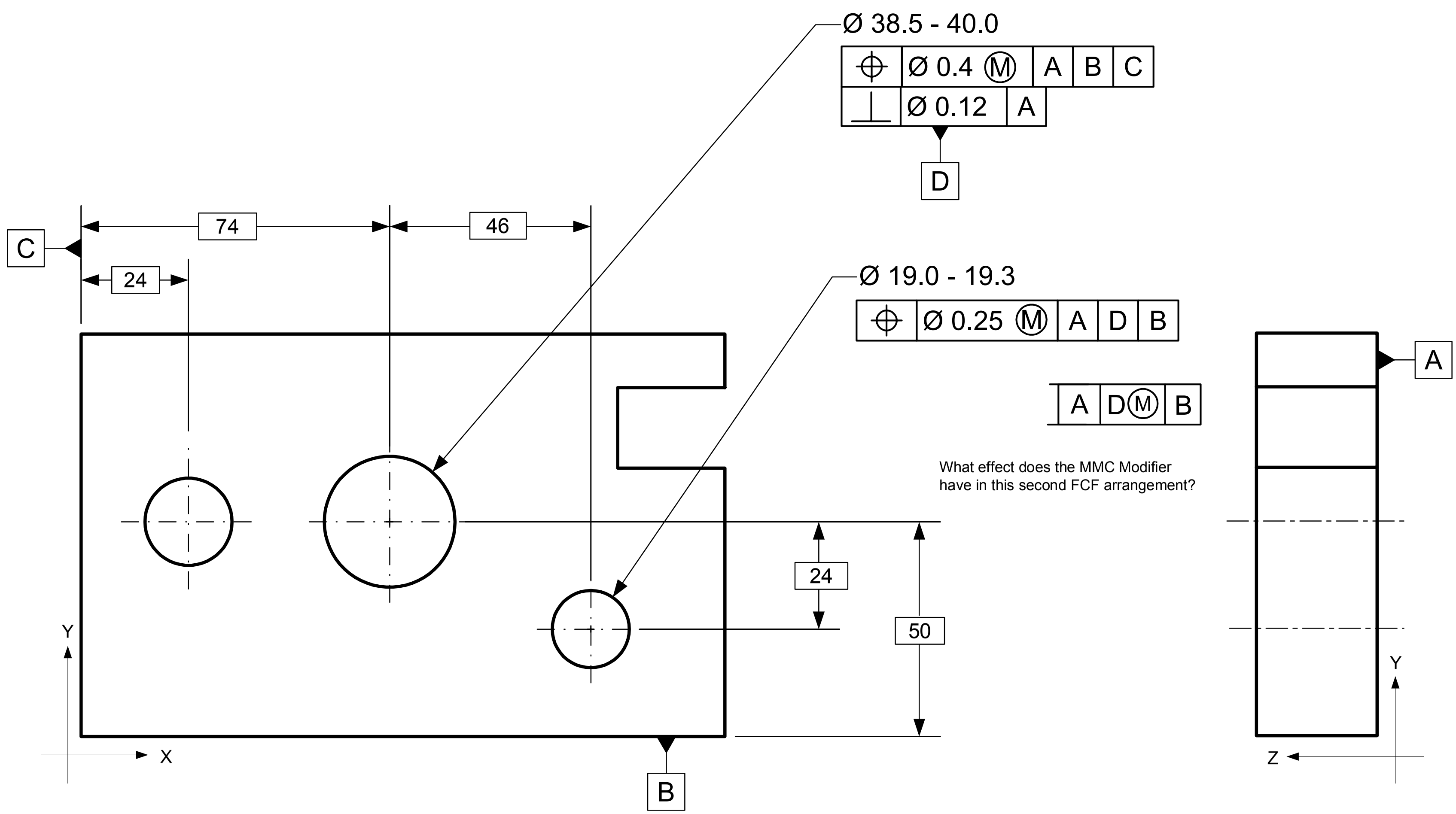
Degrees of Freedom Matrix

Datum Features	Rx	Ry	Rz	Tx	Ty	Tz



Understanding Datum Reference Frame Application (DRF)
Example 3

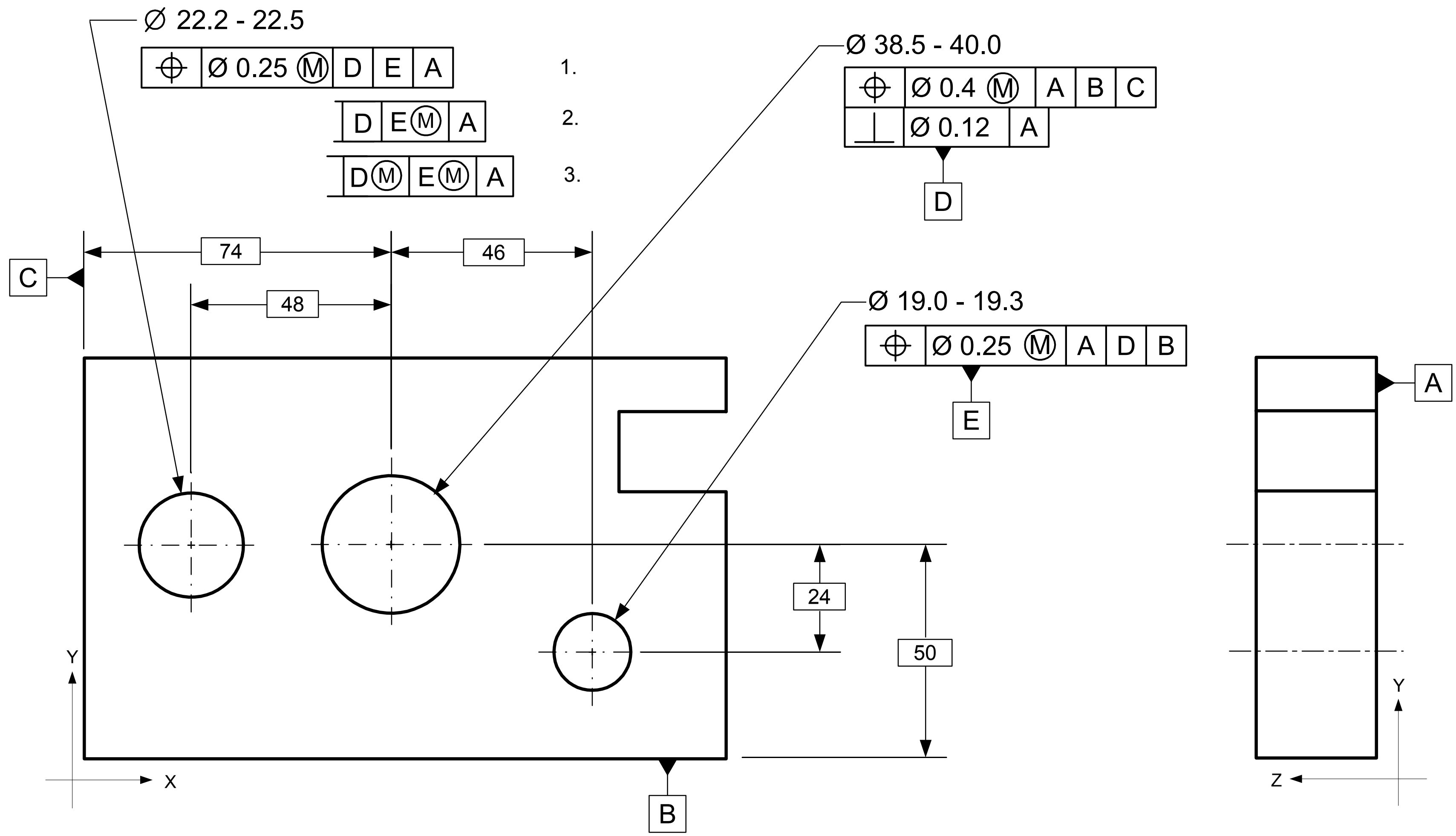
Datum Features	Rx	Ry	Rz	Tx	Ty	Tz



Understanding Datum Reference Frame Application (DRF)
Example 4

Datum Features

	Rx	Ry	Rz	Tx	Ty	Tz

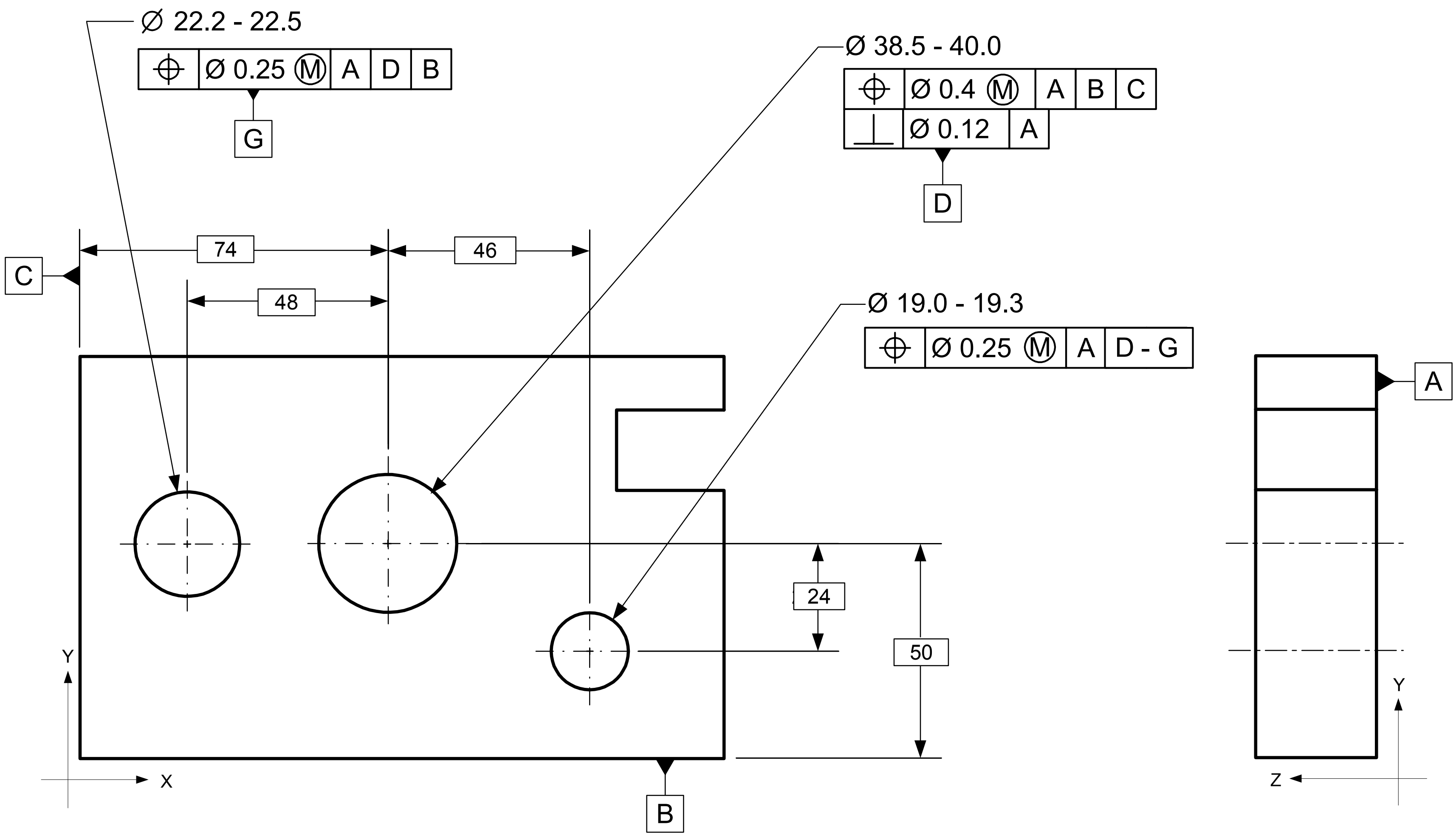


Understanding Datum Reference Frame Application (DRF)

Example 5

Degrees of Freedom Matrix

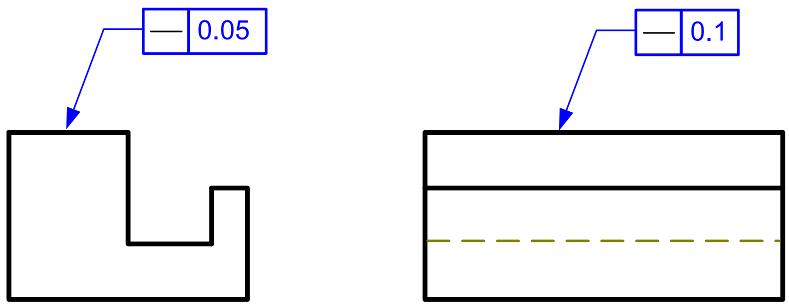
Features						



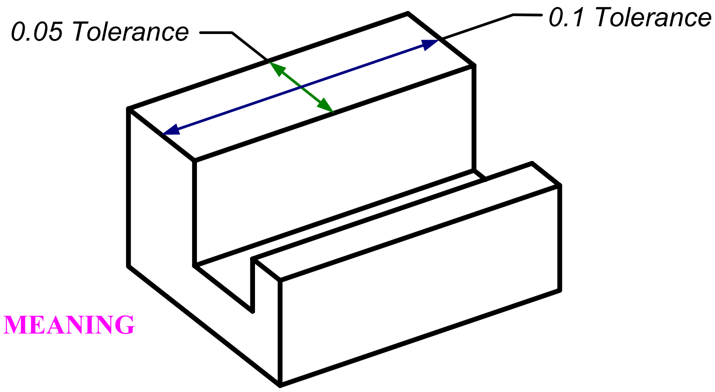
4. Form Tolerance

4.1. Straightness

Line Element – Plane Surface



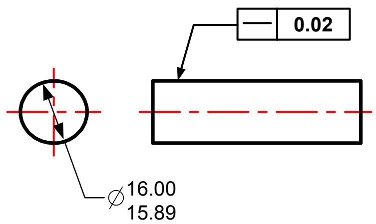
ON THE DRAWING



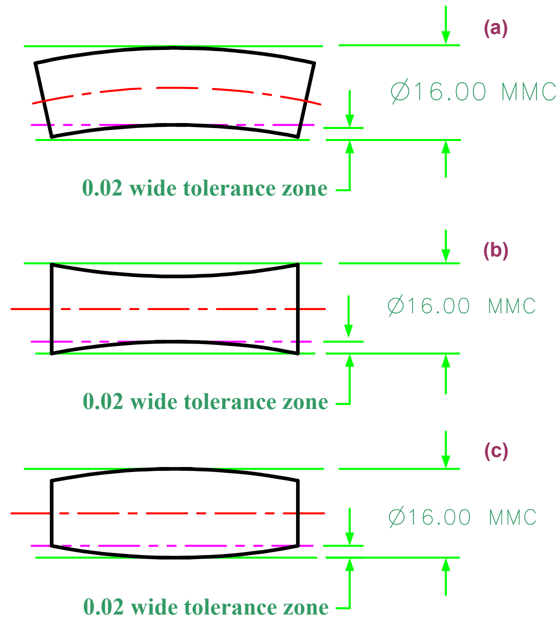
MEANING

Each longitudinal element of the surface must lie between two parallel lines 0.05 apart in the left view and 0.1 in the right view of the drawing.

Line Element – Cylinder



ON THE DRAWING

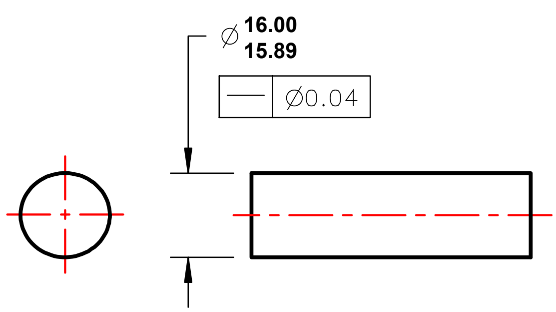


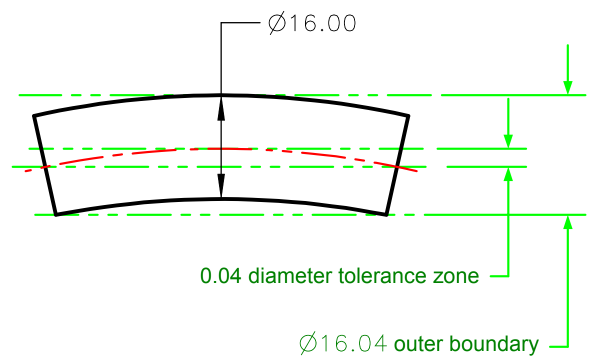
MEANING

Each longitudinal element of the surface must lie between two parallel lines 0.02 apart where the two lines and the nominal axis of the part share a specified limits of size and the boundary of perfect form at MMC 16.00

Note: Waisting (b) or barreling (c) of the surface, though within the straightness tolerance, must not exceed the limits of size of the feature

Axis at Regardless of Feature Size (RFS)



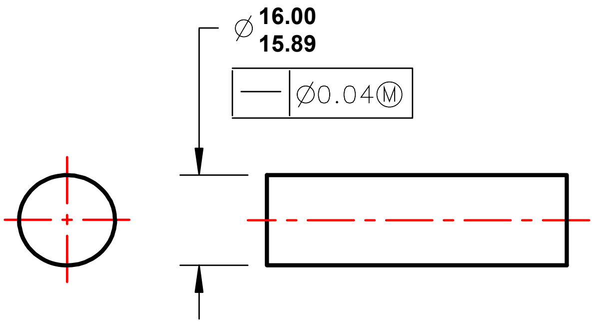


ON THE DRAWING

MEANING

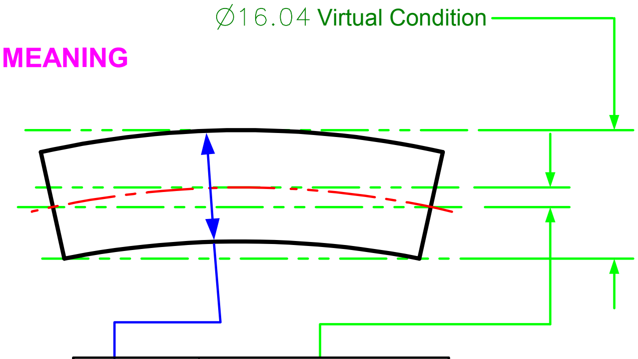
The derived median line of the feature's actual local size must lie within a cylindrical tolerance zone of 0.04 diameter, regardless of the feature size. Each circular element of the surface must be within the specified limits of size.

Axis at Maximum Material Condition (MMC)



ON THE DRAWING

MEANING

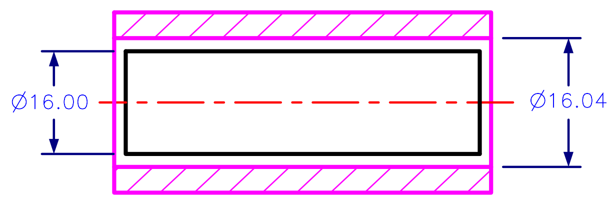


Feature Size	Diameter tolerance zone allowed
16.00	0.04
15.99	0.05
15.98	0.06
↓	↓
15.90	0.14
15.89	0.15

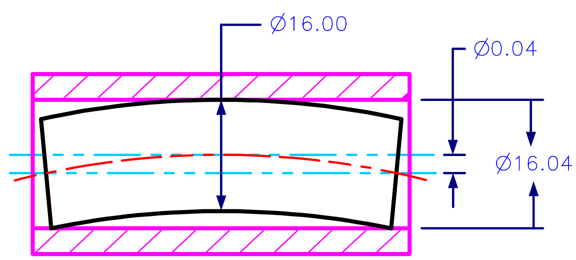
The derived median line of the feature's actual local size must lie within a cylindrical tolerance zone of 0.04 diameter at MMC. As each actual local size departs from MMC, an increase in the local diameter of the tolerance cylinder is allowed which is equal to the amount of such departure. Each circular element of the surface must be within the specified limits of size.

Acceptance Boundary

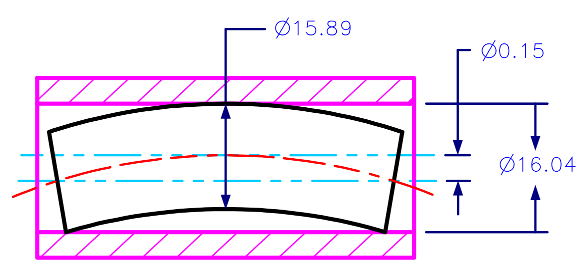
- The maximum diameter of the pin with perfect form is shown in a gage with a 16.04 diameter hole.



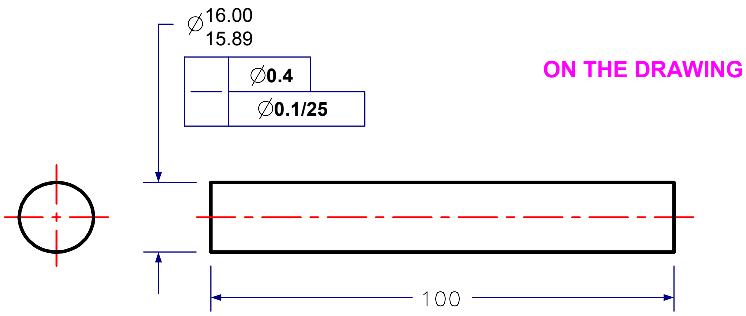
- With the pin at maximum diameter 16.00, the gage will accept the pin with up to 0.04 variation in straightness.

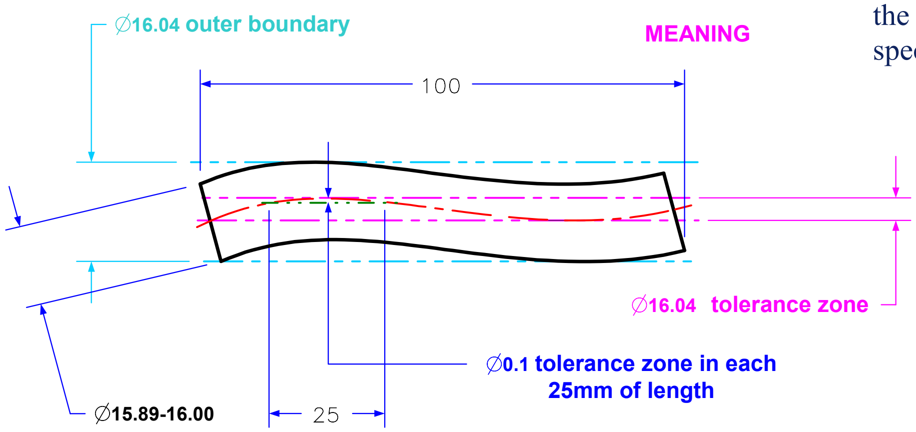


- With the pin at minimum diameter 15.89, the gage will accept the pin with up to 0.15 variation in straightness.



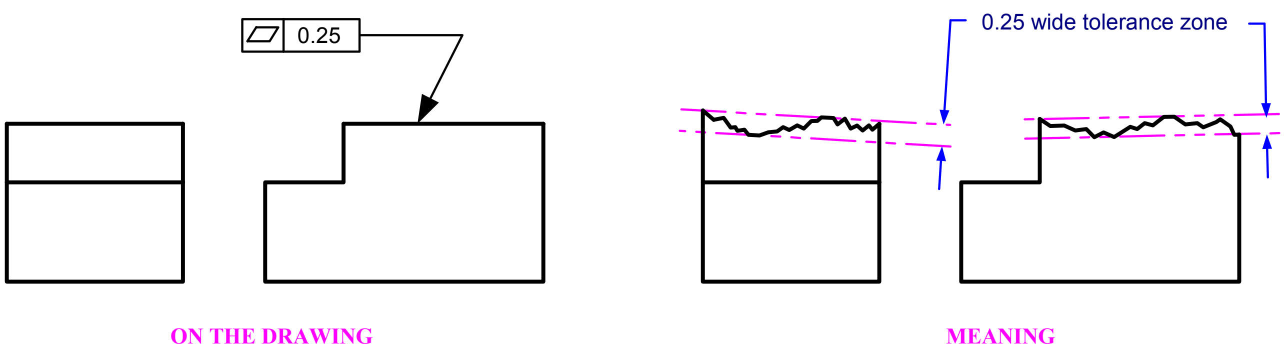
Per Unit Length





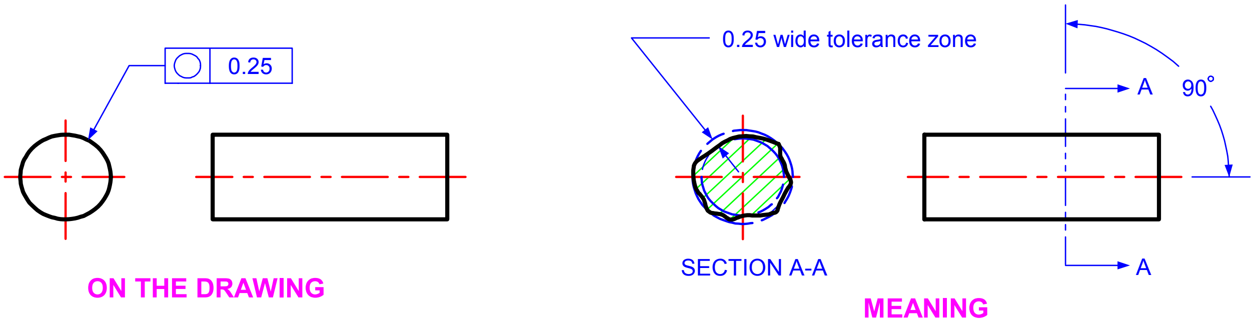
The derived median line of the feature's actual local size must lie within a cylindrical tolerance zone of ∅0.4 for the total 100mm of length and within a 0.1 cylindrical tolerance zone for any 25mm length, regardless of feature size. Each circular element of the surface must be within the specified limits of size.

4.2. Flatness

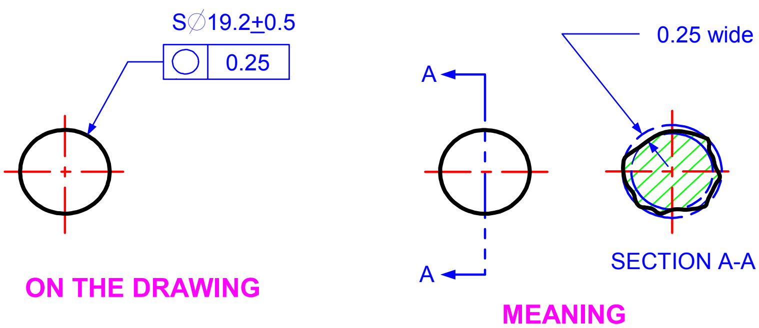


The surface must lie between two parallel planes 0.25 apart. The surface must be within the specified limits of size.

4.3. Circularity (Roundness)

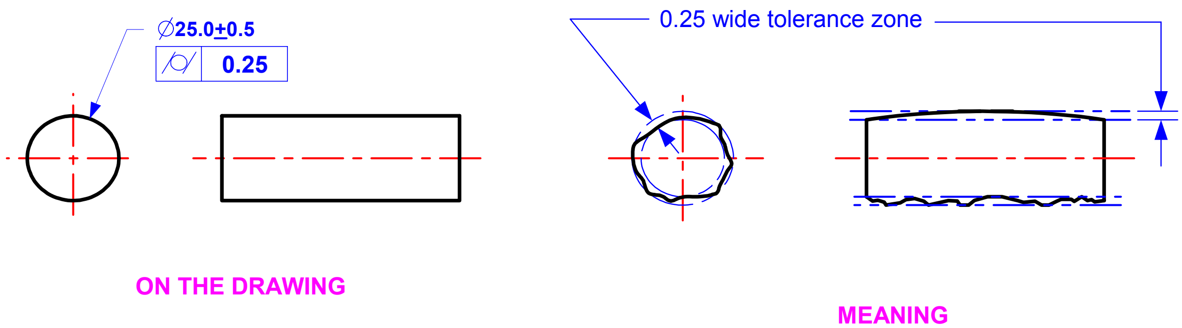


Each circular element of the surface in a plane perpendicular to an axis must lie between two concentric circles, one having a radius 0.25 larger than the other. Each circular element of the surface must be within the specified limits of size.



Each circular element of the surface in a plane passing through a common center must lie between two concentric circles, one having a radius 0.25 larger than the other. Each circular element of the surface must be within the specified limits of size.

4.4. Cylindricity



The cylindrical surface must lie between two concentric cylinders, one having a radius 0.25 larger than the other. The surface must be within the specified limits of size.

4.5. Exercise

1. On Figure 1(a), indicate control of element straightness by use of Rule #1 so that maximum possible error is no more than mm if the feature maximum size is ∅16mm.

2. On Figure 1(b), indicate an element straightness maximum of 0.012mm.

3. What is the circularity (roundness) of this pin? ______________________________

4. On Figure 1(c), indicate that axis straightness may violate Rule #1 and allow a total bend of up to 0.4mm.

5. On Figure 1(d), assume that the pin will assemble with the hole shown in 1 (e).

   The condition of ________________ is often desired. Indicate this with a straightness tolerance of 0.4mm.

6. What is the cylindricity of this pin? _____________________

7. What is the Virtual Condition of the pin for the requirement of question 5? _____

8. On Figure 2, indicate on the bottom surface a control that requires all elements and points relative to each other be within a tolerance zone that is two planes

   which are 0.05mm apart. This control would be called ______________________

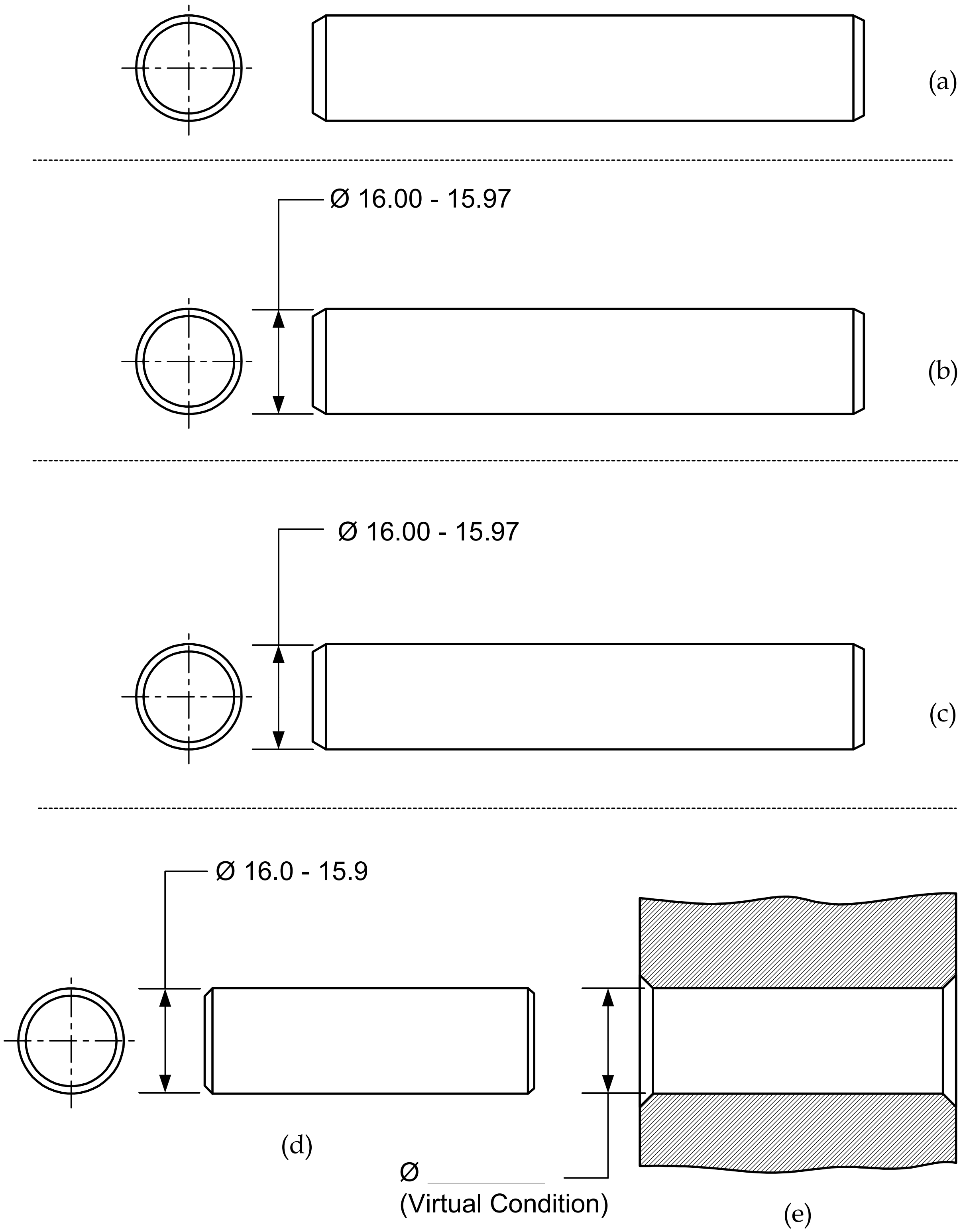


Figure 1

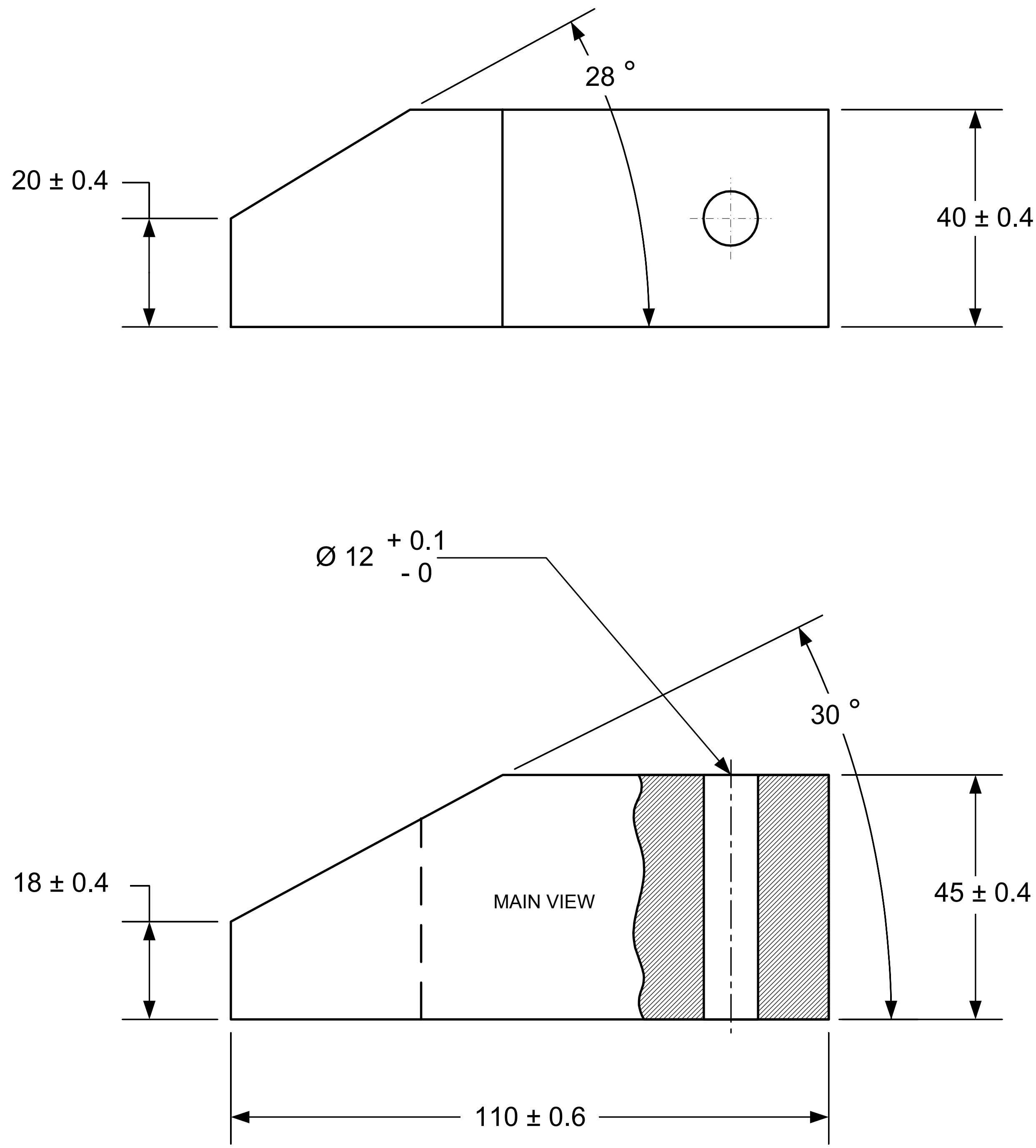
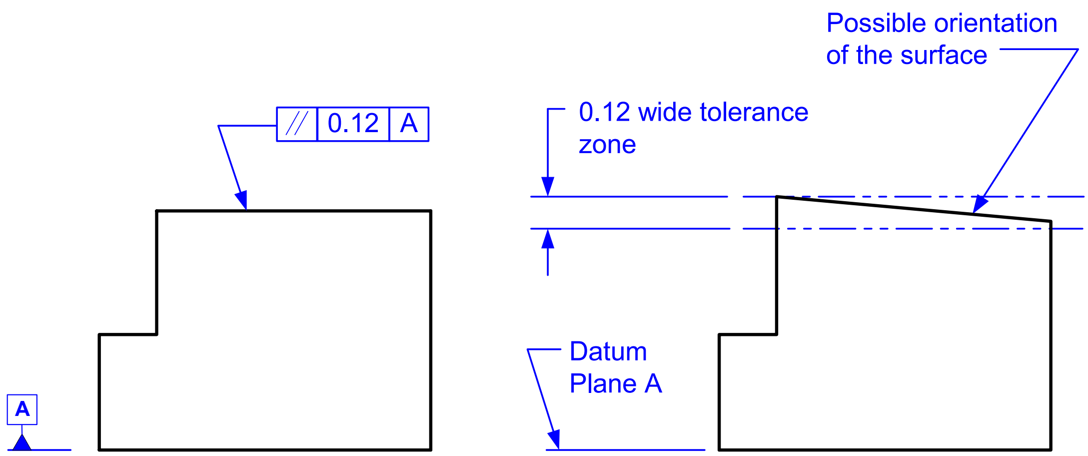


Figure 2

5. Orientation Tolerance

5.1. Parallelism

Surface Plane

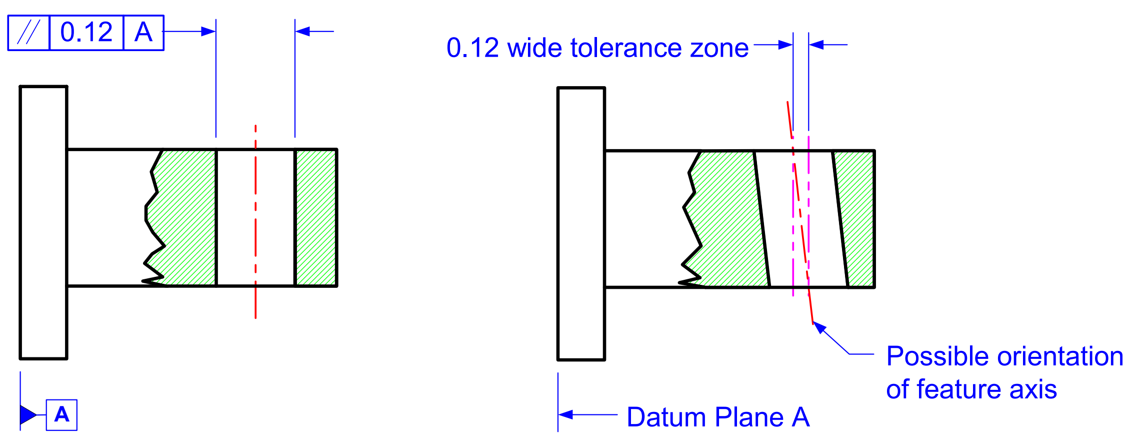


ON THE DRAWING MEANING

The surface must lie between two parallel planes 0.12 apart which are parallel to datum plane A. The surface must be within the specified limits of size.

Axis related to a Surface Plane

ON THE DRAWING MEANING

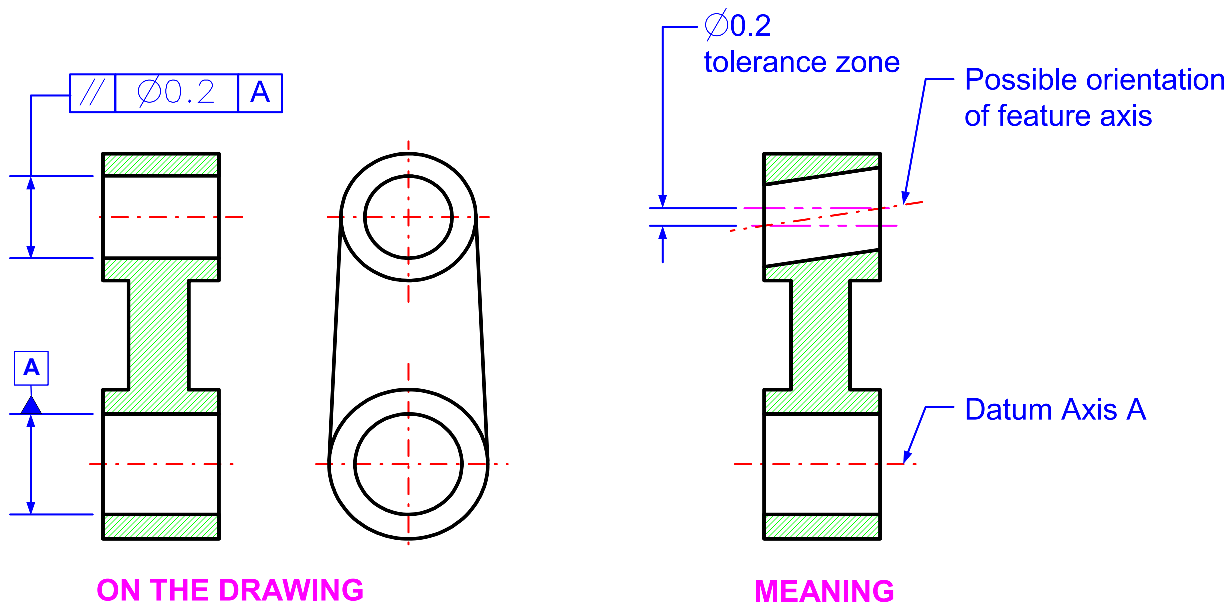


Regardless of feature size, the feature axis must lie between two parallel planes 0.12 apart which are parallel to datum plane A. The feature axis must be within the specified tolerance of location.

What would be the result if a diameter symbol was added to the callout?

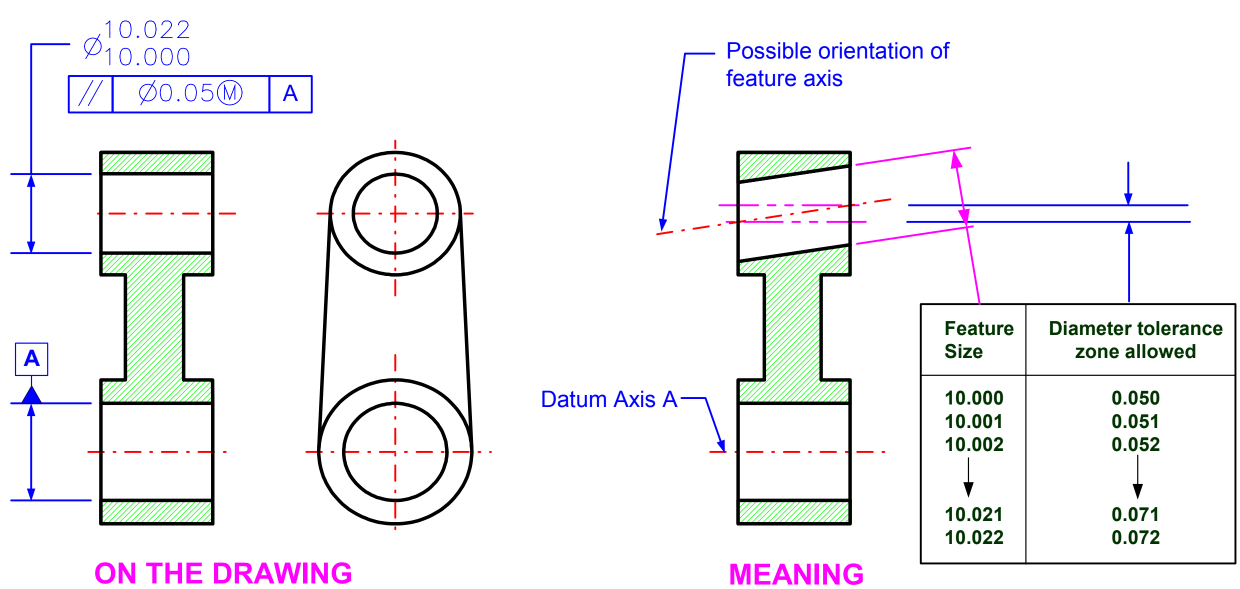
//	Ø 0.12	A

Axis related to another Axis at Regardless of Feature Size (RFS)



Regardless of feature size, the feature axis must lie within a 0.2 diameter cylindrical zone parallel to datum axis A. The feature axis must be within the specified tolerance of location.

Axis related to another Axis at Maximum Material Condition (MMC)

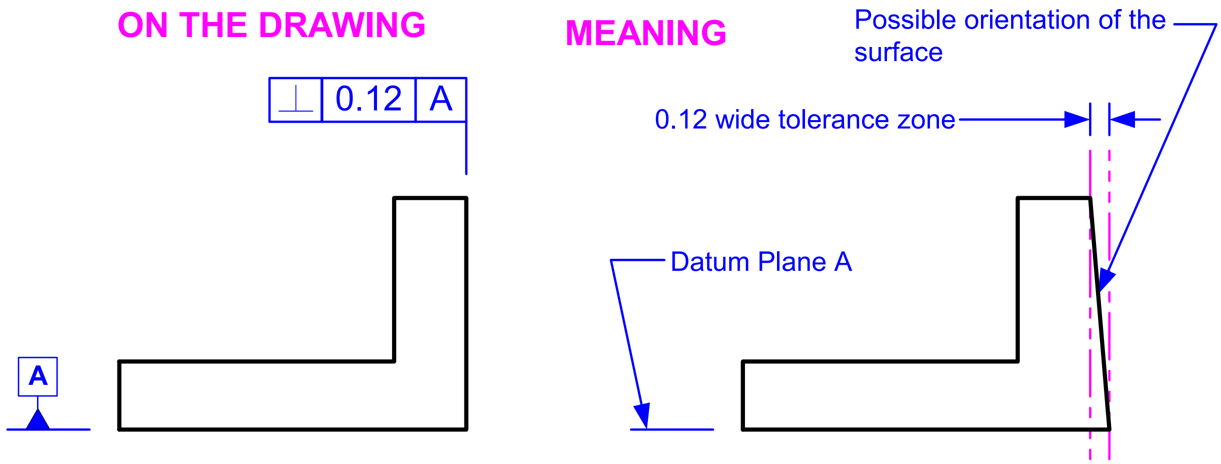


Feature Size	Diameter tolerance zone allowed
10.000	0.050
10.001	0.051
10.002	0.052
↓	↓
10.021	0.071
10.022	0.072

Where the feature is at maximum material condition (10.000), the maximum parallelism tolerance is n0.050. Where the feature departs from its MMC size, an increase in the parallelism tolerance is allowed which is equal to the amount of such departure. The feature axis must be within the specified tolerance of location.

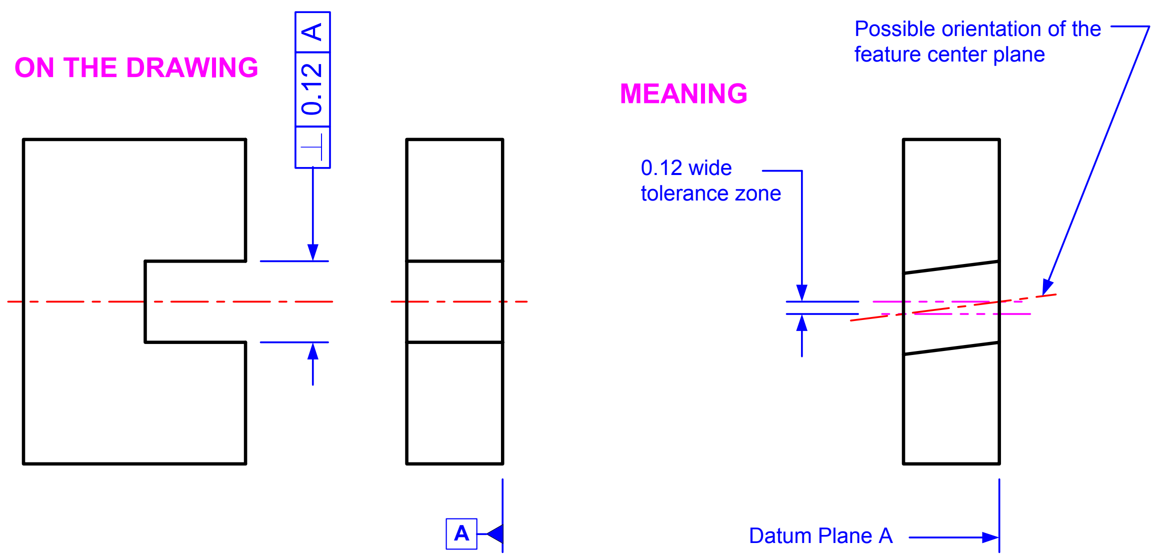
5.2. Perpendicularity

Surface Plane



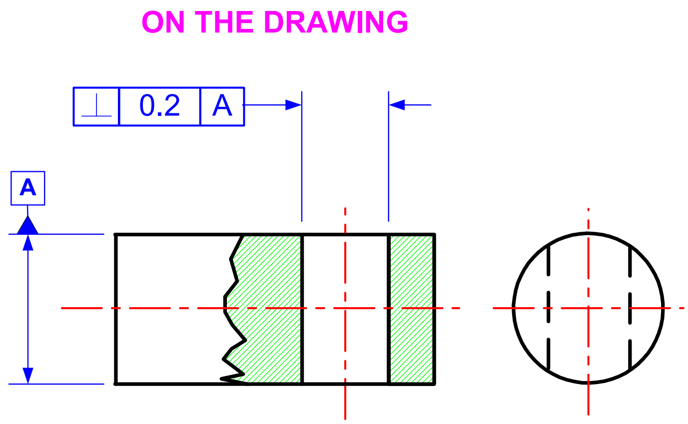
The surface must lie between two parallel planes 0.12 apart which are perpendicular to datum plane A. The surface must be within the specified limits of size.

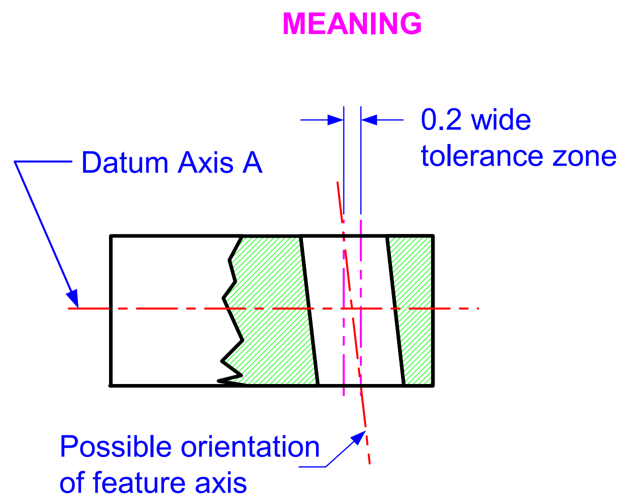
Center Plane



Regardless of feature size, the feature center plane must lie between two parallel planes 0.12 apart which are perpendicular to datum plane A. The feature center plane must be within the specified tolerance of location.

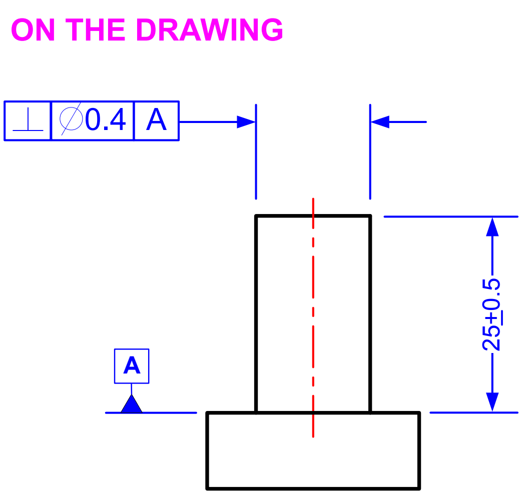
Axis to Axis

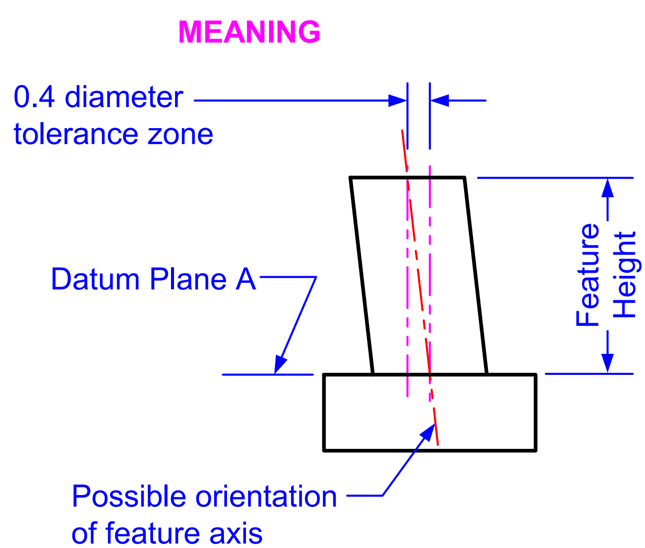




Regardless of feature size, the feature axis must lie between two parallel planes 0.2 apart which are perpendicular to datum axis A. The feature axis must be within the specified tolerance of location.
Note: This applies only to the view on which it is specified.

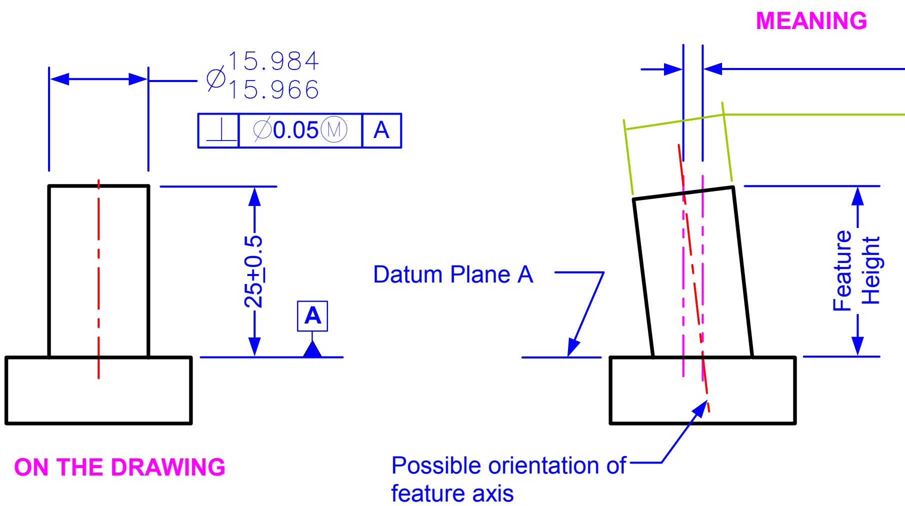
Axis to Plane (RFS)





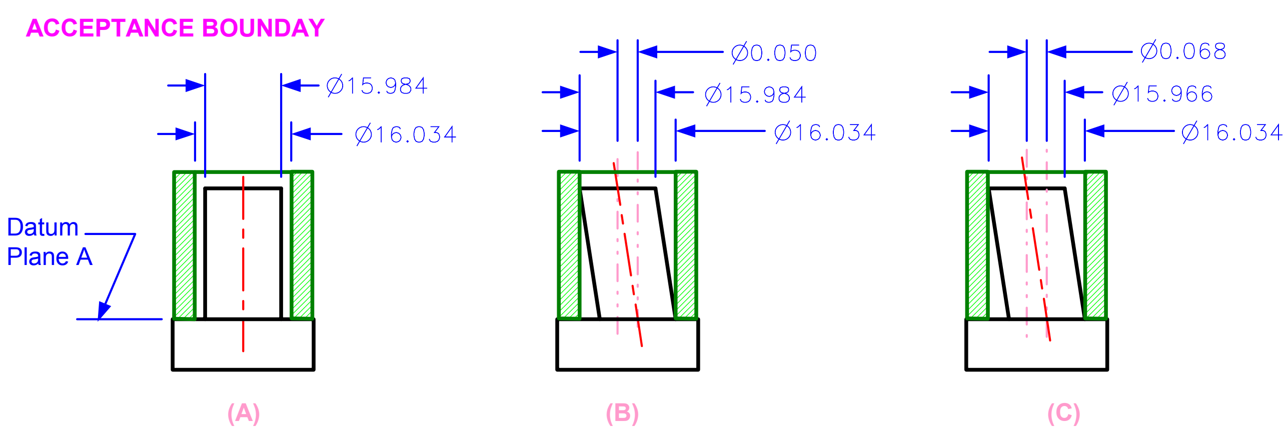
Regardless of feature size, the feature axis must lie within a cylindrical zone 0.4 diameter which is perpendicular to and projects from datum plane A for the feature height. The feature axis must be within the specified tolerance of location.

Axis to Plane (MMC)



Feature Size	Diameter tolerance zone allowed
15.984	0.050
15.983	0.051
15.982	0.052
↓	↓
15.976	0.067
15.966	0.068

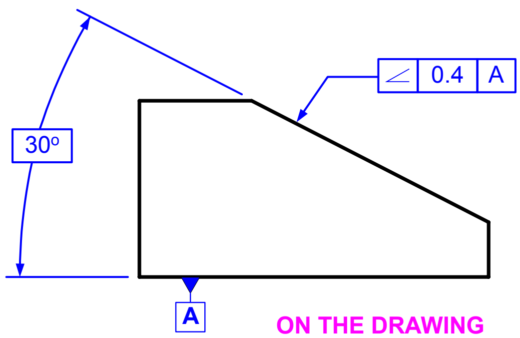
Where the feature is at maximum material condition (15.984), the maximum perpendicularity tolerance is n0.050. Where the feature departs from its MMC size, an increase in the perpendicularity tolerance is allowed which is equal to the amount of such departure. The feature axis must be within the specified tolerance of location.

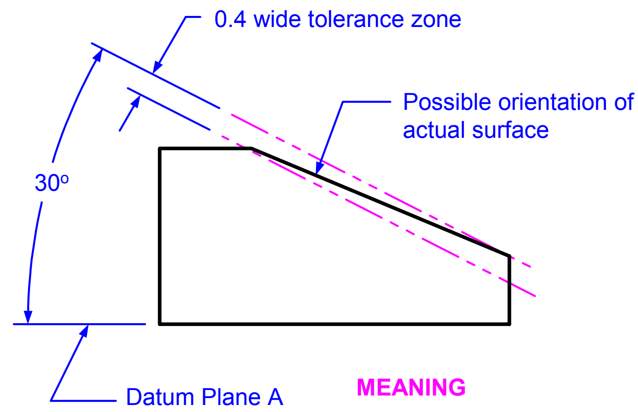


(A)The maximum diameter pin with perfect orientation is shown in a gage with a 16.034 diameter hole.
(B)With the pin at maximum diameter (15.984), the gage will accept the part with up to 0.05 variation in perpendicularity.
(C)The pin is at minimum diameter (15.966), and the variation in perpendicularity may increase to 0.068 and the part will be acceptable.

5.3. Angularity

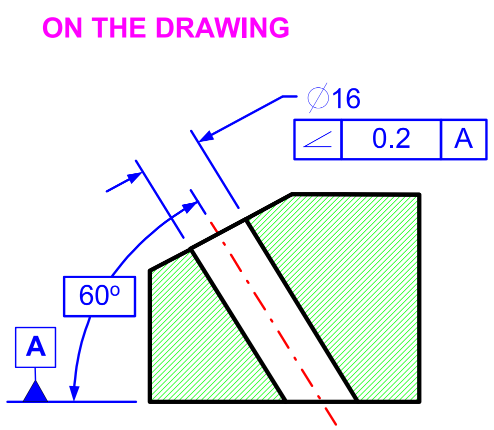
Surface Plane

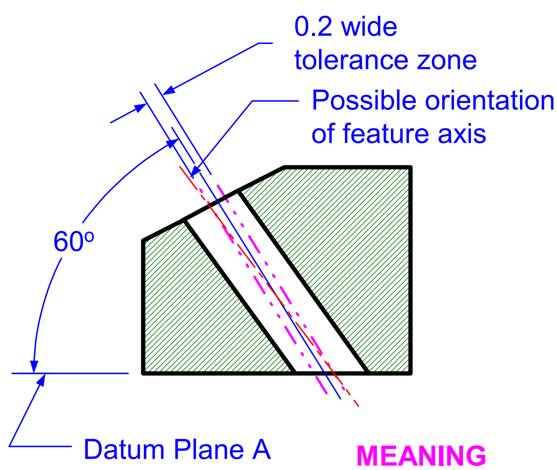




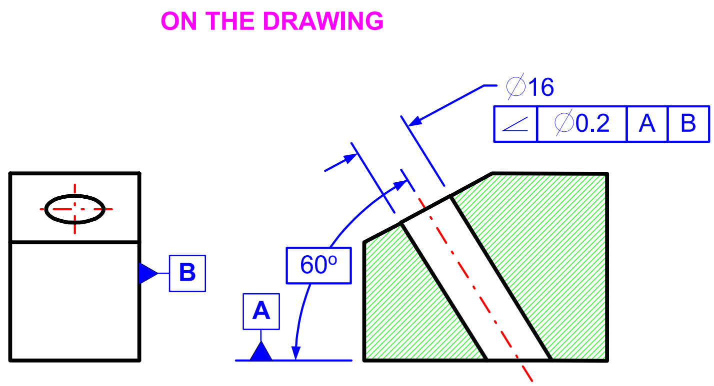
The surface must lie between two parallel planes 0.4 apart which are inclined at 30º to datum plane A. The surface must be within the specified limits of size.

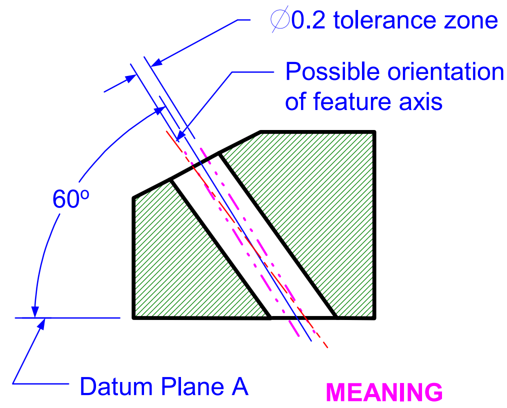
Axis to Surface Plane





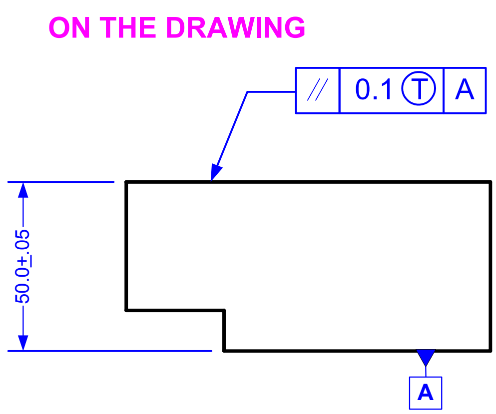
Regardless of feature size, the feature axis must lie between two parallel planes 0.2 apart which are inclined 60º to datum plane A. The feature axis must be within the specified tolerance of location.

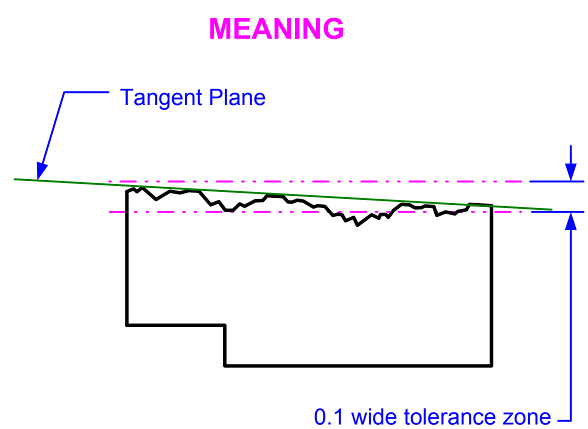




Regardless of feature size, the feature axis must lie within a 0.2 diameter cylindrical zone inclined 60° to datum plane A. The feature axis must be within the specified tolerance of location.

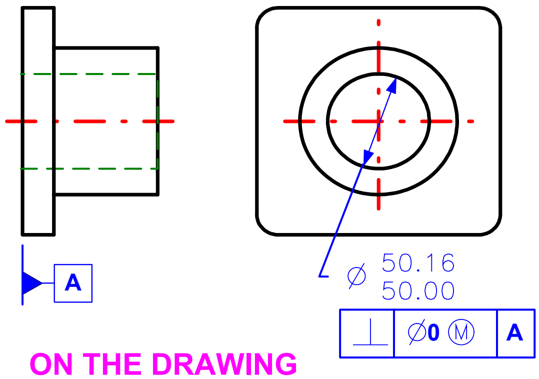
5.4. Use of Tangent Plane Symbol



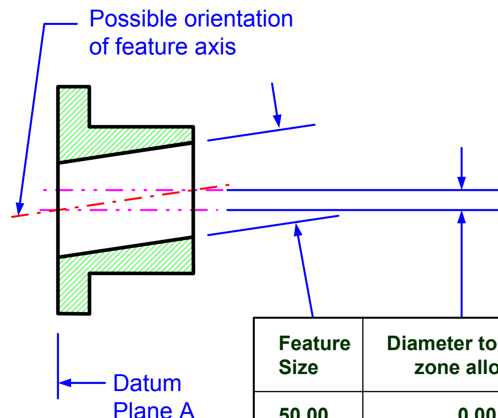


A plane contacting the high points of the surface shall lie within two parallel planes 0.1 apart. The surface must be within the specified limits of size.

5.5. Use of Zero Tolerance at MMC



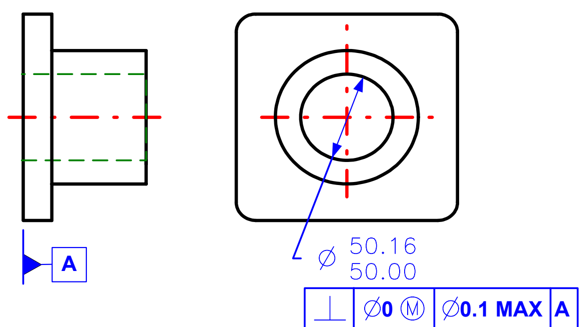
ON THE DRAWING



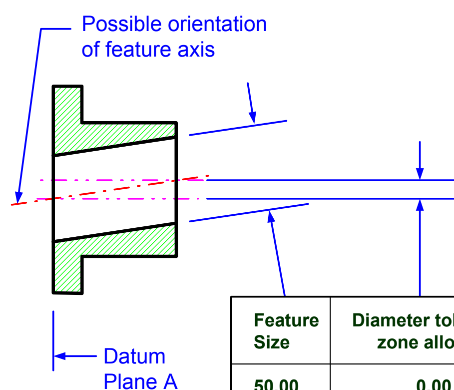
MEANING

Feature Size	Diameter tolerance zone allowed
50.00	0.00
50.01	0.01
50.02	0.02
↓	↓
50.15	0.15
50.16	0.16

Where the feature is at maximum material condition (50.00), its axis must be perpendicular to datum plane A. Where the feature departs from its MMC size, a perpendicularity tolerance is allowed which is equal to the amount of such departure. The feature axis must be within the specified tolerance of location.



ON THE DRAWING



MEANING

Feature Size	Diameter tolerance zone allowed
50.00	0.00
50.01	0.01
50.02	0.02
↓	↓
50.10	0.10
↓	↓
50.16	0.10

Where the feature is at maximum material condition (50.00), its axis must be perpendicular to datum plane A. Where the feature departs from its MMC size, a perpendicularity tolerance is allowed which is equal to the amount of such departure, up to the 0.1 maximum. The feature axis must be within the specified tolerance of location.

5.6. Exercise

1. On Figure 2, indicate that the right vertical surface in the main view is to be square to the lower surface within 0.08mm.

2. Show below (sketch) how the tolerance zone is established for the requirement of question 1.

3. Working on Figure 2, indicate that the right vertical surface is to be square with the front surface within 0.08mm.

4. Assume that in Figure 2, the ∅12mm hole has been located with position dimensions and tolerance. Add an orientation tolerance to control the relationship of the hole to the bottom surface within ∅0.08mm total.

5. Sketch how the tolerance zone is established for the requirement of question 4.

6. What is the total permissible perpendicularity if the hole size is produced at ∅12mm? ____________. If the hole is produced at ∅12.1mm? _____________

7. Suppose the perpendicularity of the produced feature was allowed to increase as the size of the feature increased, how would that be indicated?

8. What THEN is the total permissible perpendicularity if the hole size is produced at ∅12mm? __________. If the hole is produced at ∅12.05mm? ____________. If the hole is produced at ∅12.1mm? _____________.

9. On Figure 2, indicate requirements to control the angles within a total tolerance of 0.1mm.

10. Sketch how the tolerance zone is established for the 30° angle.

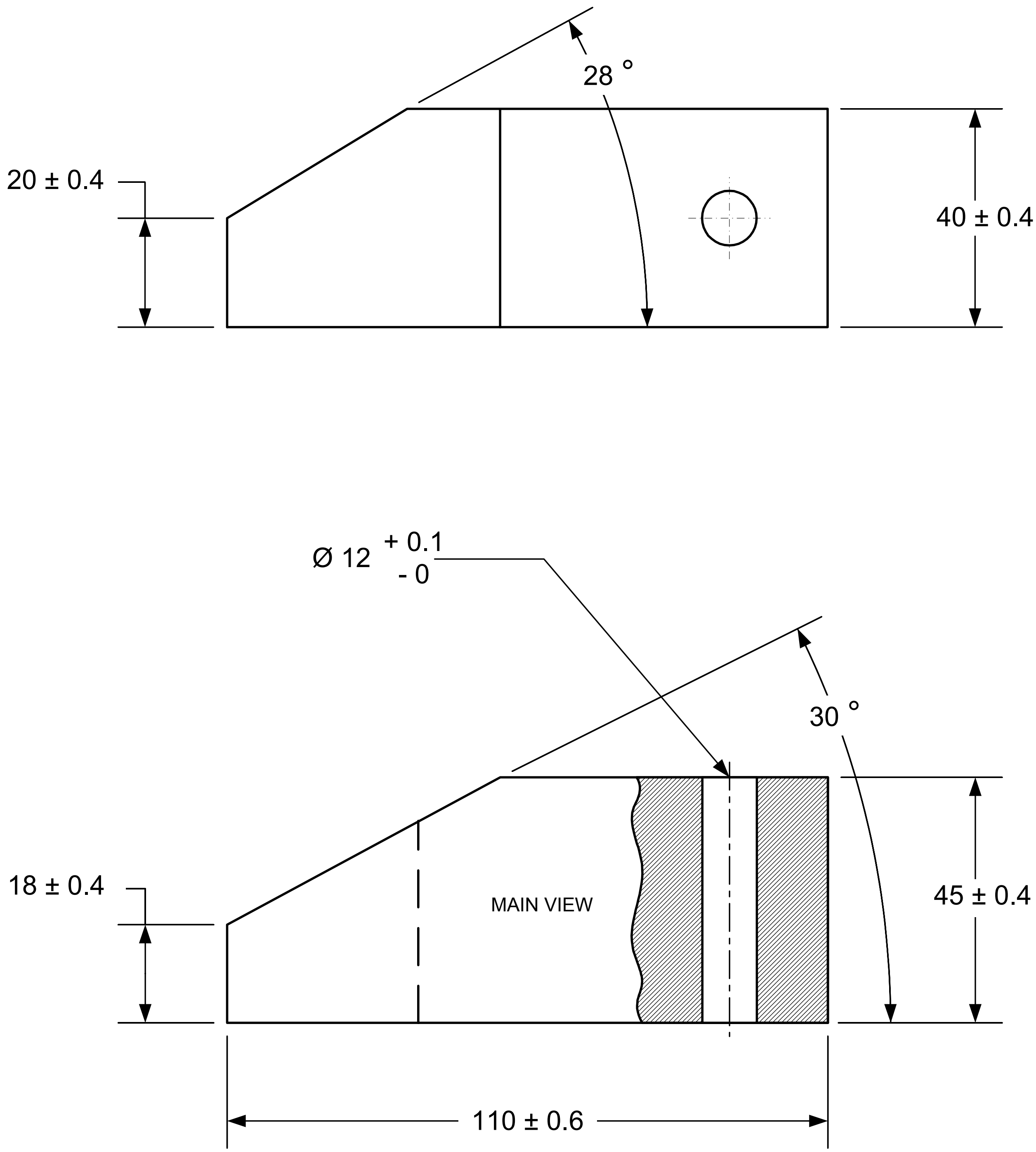
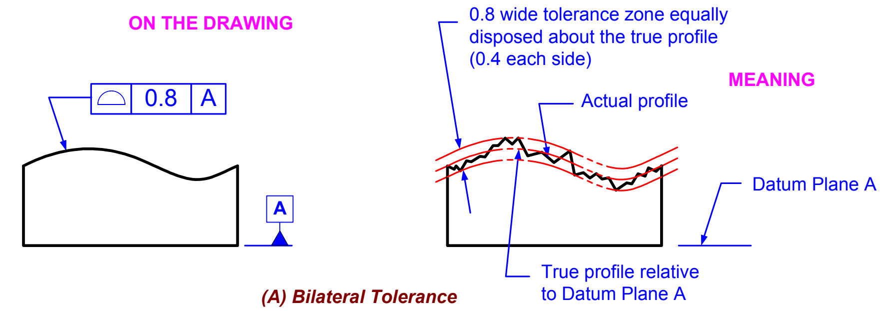


Figure 2

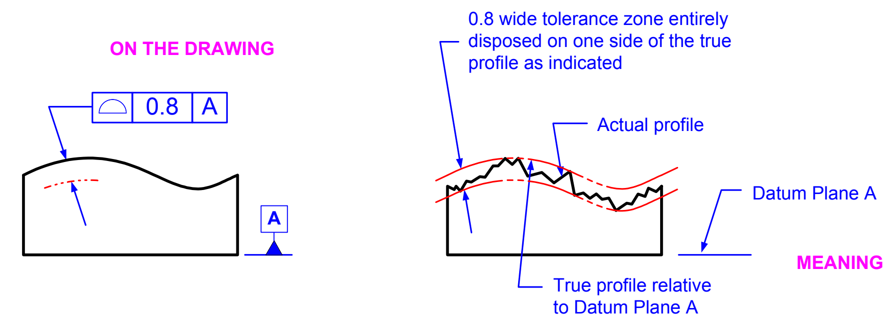
6. Profile Tolerance

6.1. Bilateral & Unilateral

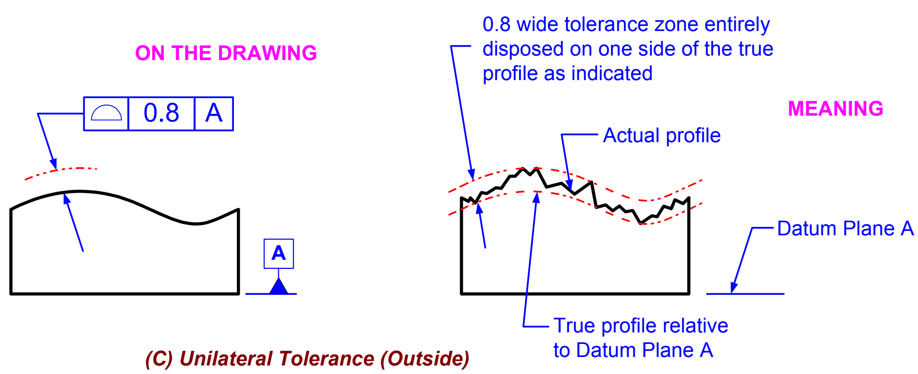
Profile of a Line ⌒ Profile of a Surface ⌓



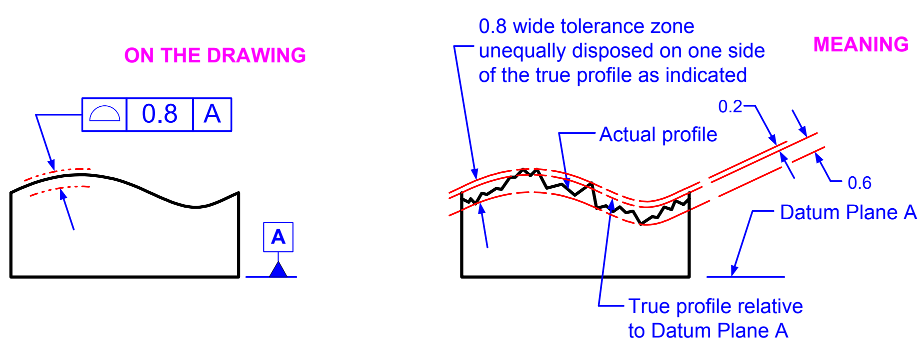
(A) Bilateral Tolerance



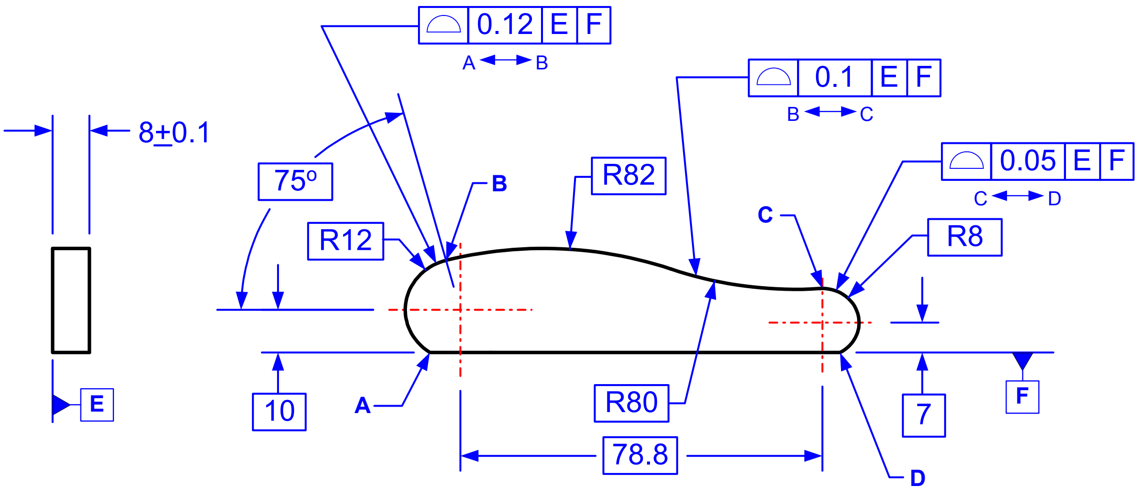
(B) Unilateral Tolerance (Inside)



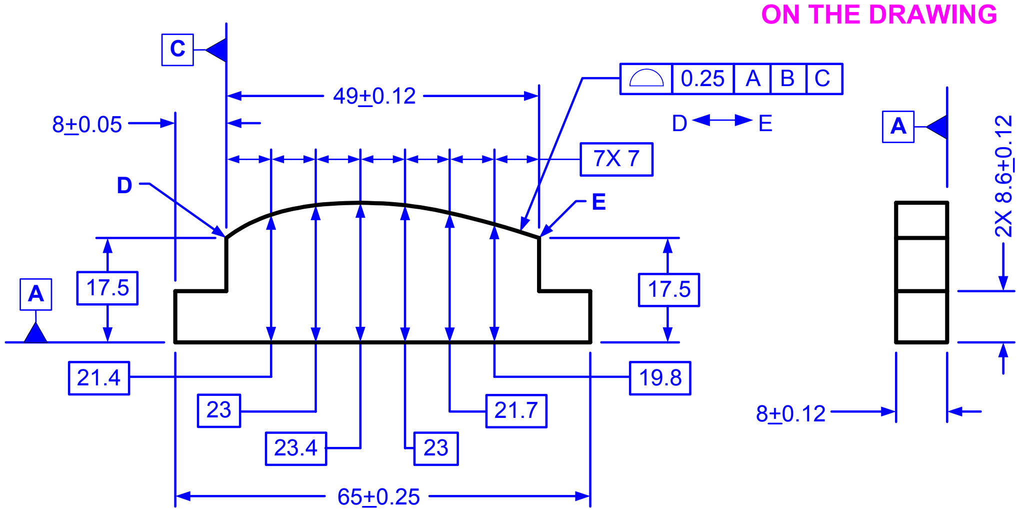
(C) Unilateral Tolerance (Outside)

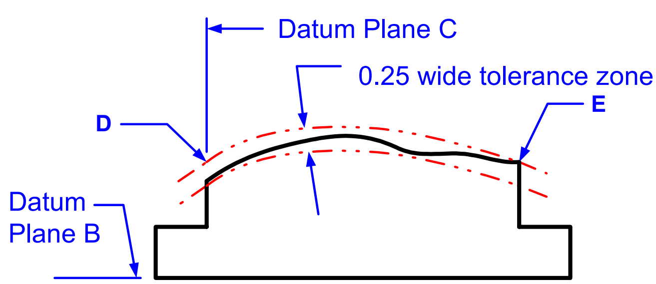


(D) Bilateral Tolerance unequal distribution

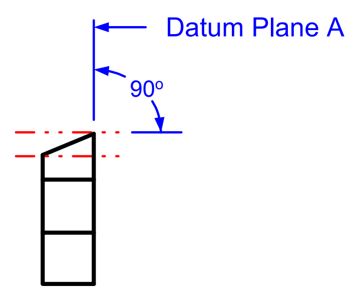


ON THE DRAWING

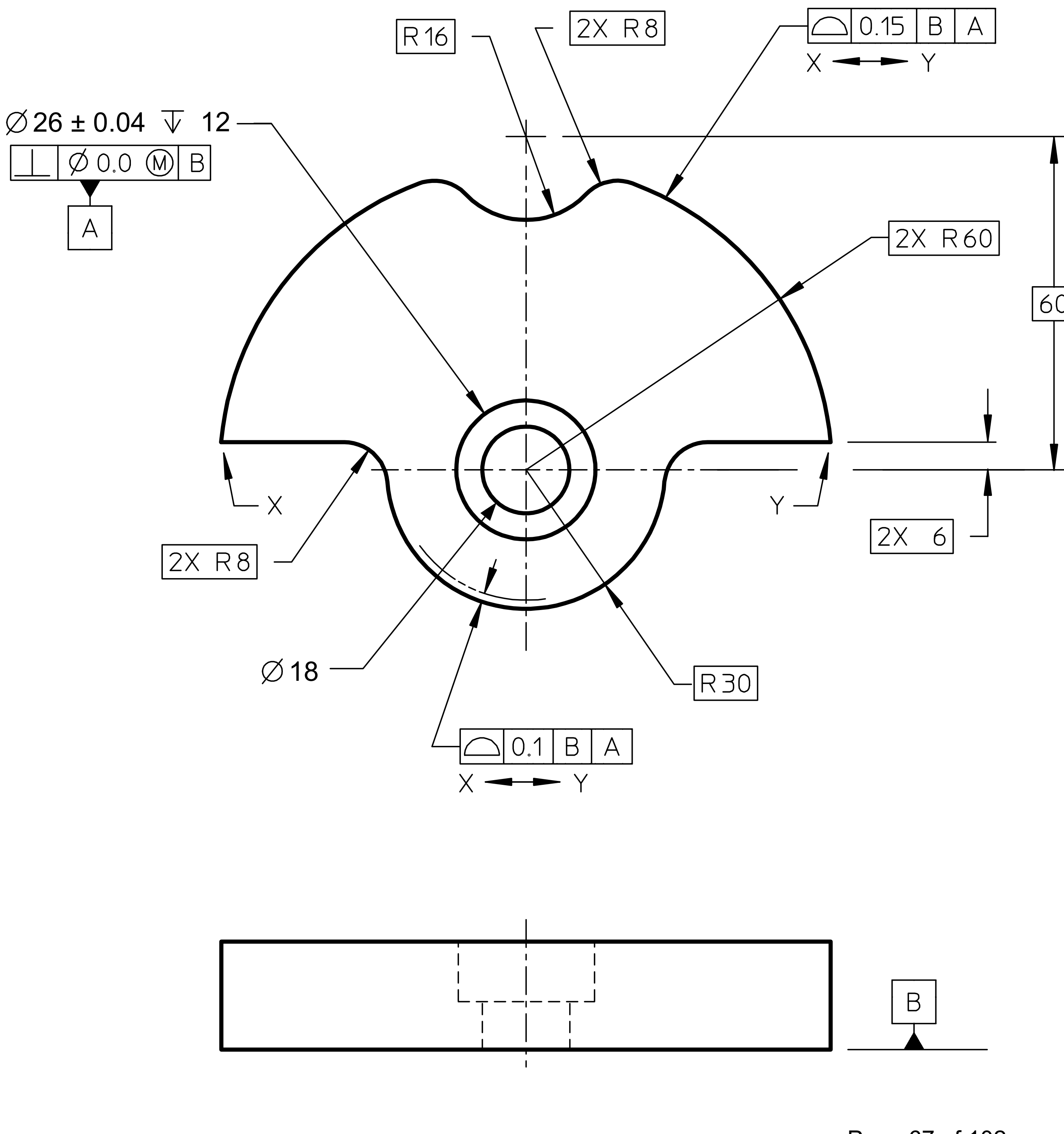


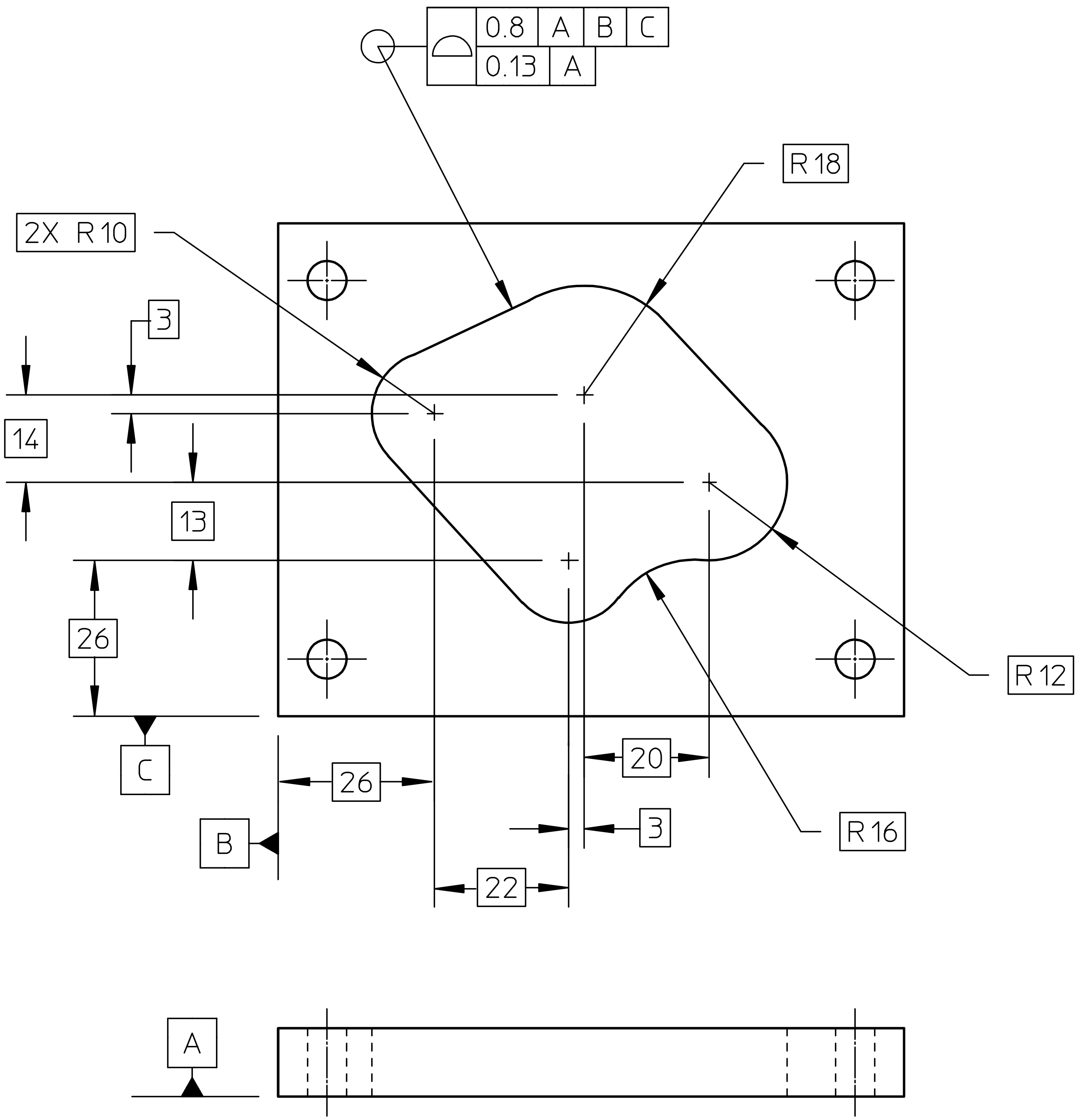


MEANING

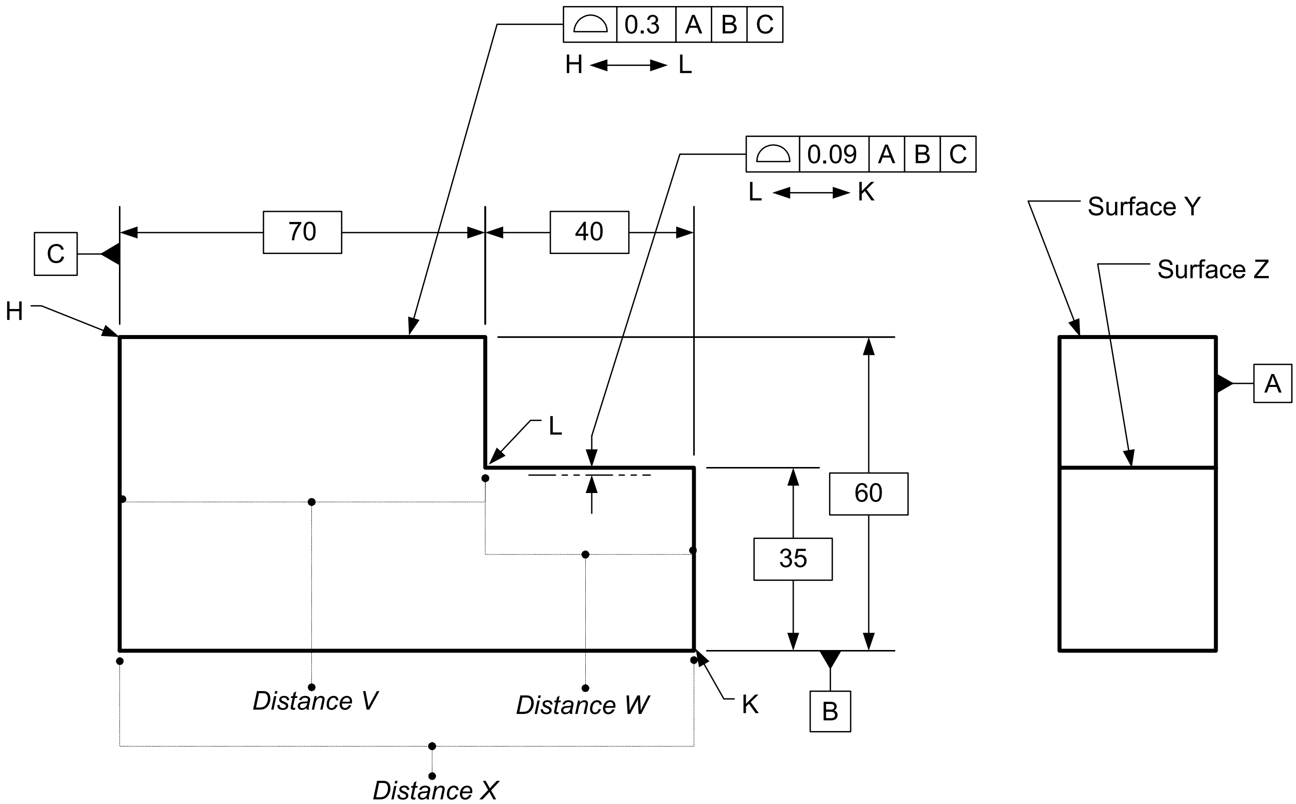


The surface between points D and E must lie between two profile boundaries 0.25 apart, perpendicular to datum plane A, equally disposed about the true profile and positioned with respect to datum planes B and C.


R16
2X R8
0.15 B A
X Y
∅26 ± 0.04 12
∅0.0 Ⓜ B
A
2X R60
60
X
Y
2X R8
2X 6
∅18
R30
0.1 B A
X Y
B


0.8 A B C
0.13 A
R 18
2X R 10
3
14
13
26
C
26
B
22
20
3
R 12
R 16
A

6.2. Exercise



1. On the part shown above, what is the minimum and maximum of the following distances in relation to the datum reference frame, as allowed by the profile callout?

   (a) Distance V: Minimum ________________ Maximum ________________

   (b) Distance W: Minimum ________________ Maximum ________________

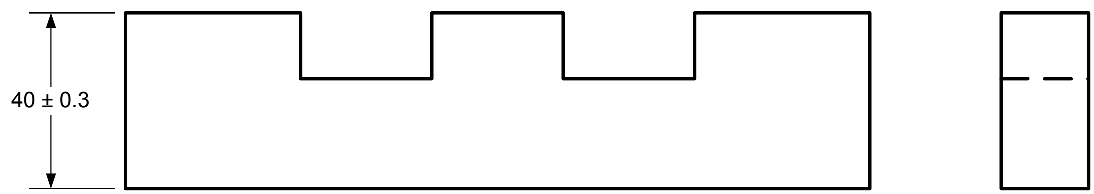
   (c) Distance X: Minimum ________________ Maximum ________________

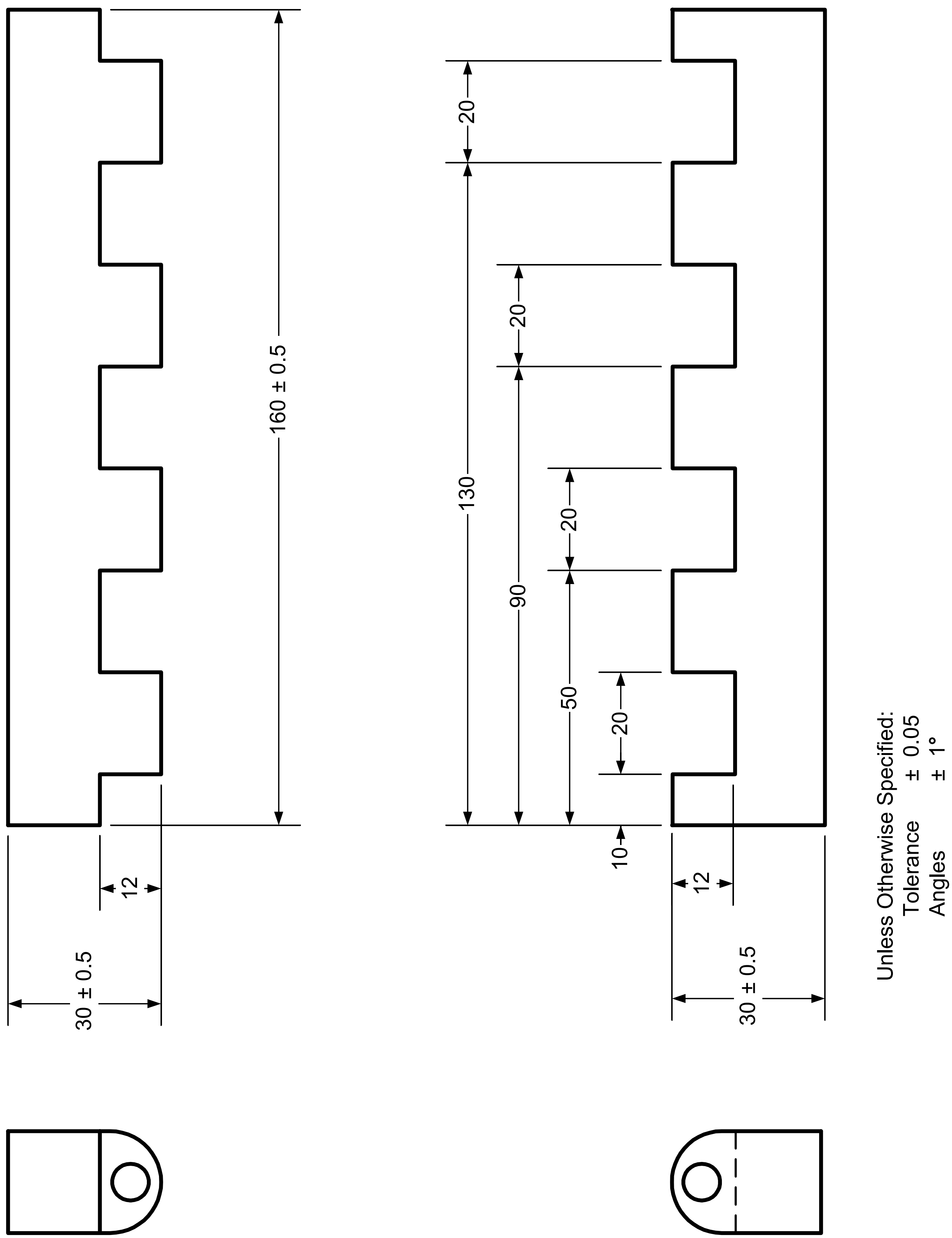
2. On the same part, considering the applicable profile callouts, what is the maximum perpendicularity of the following surfaces in relation to Datum A?

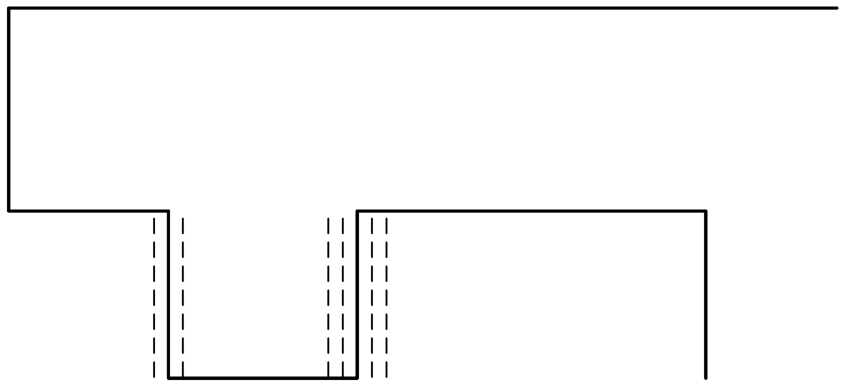
   (a) Surface Y: ________________

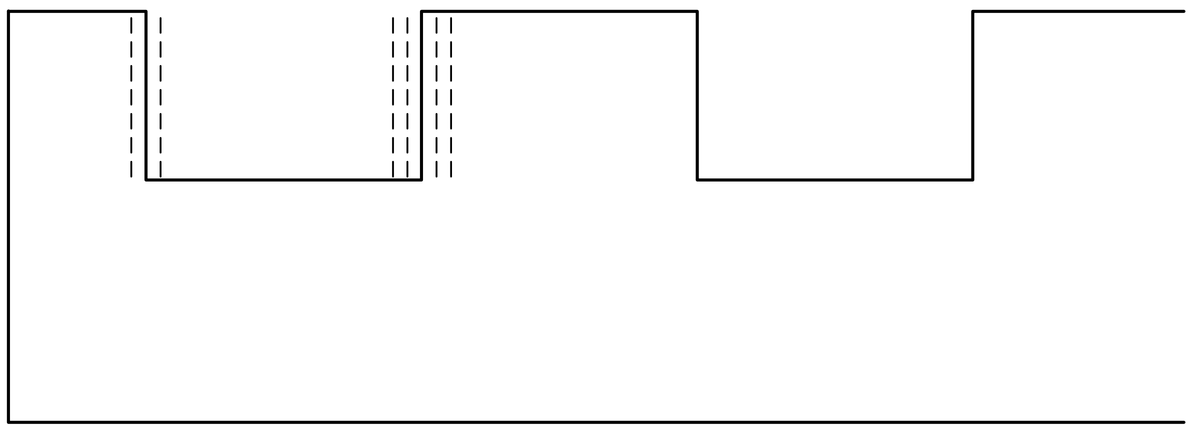
   (b) Surface Z W: ________________

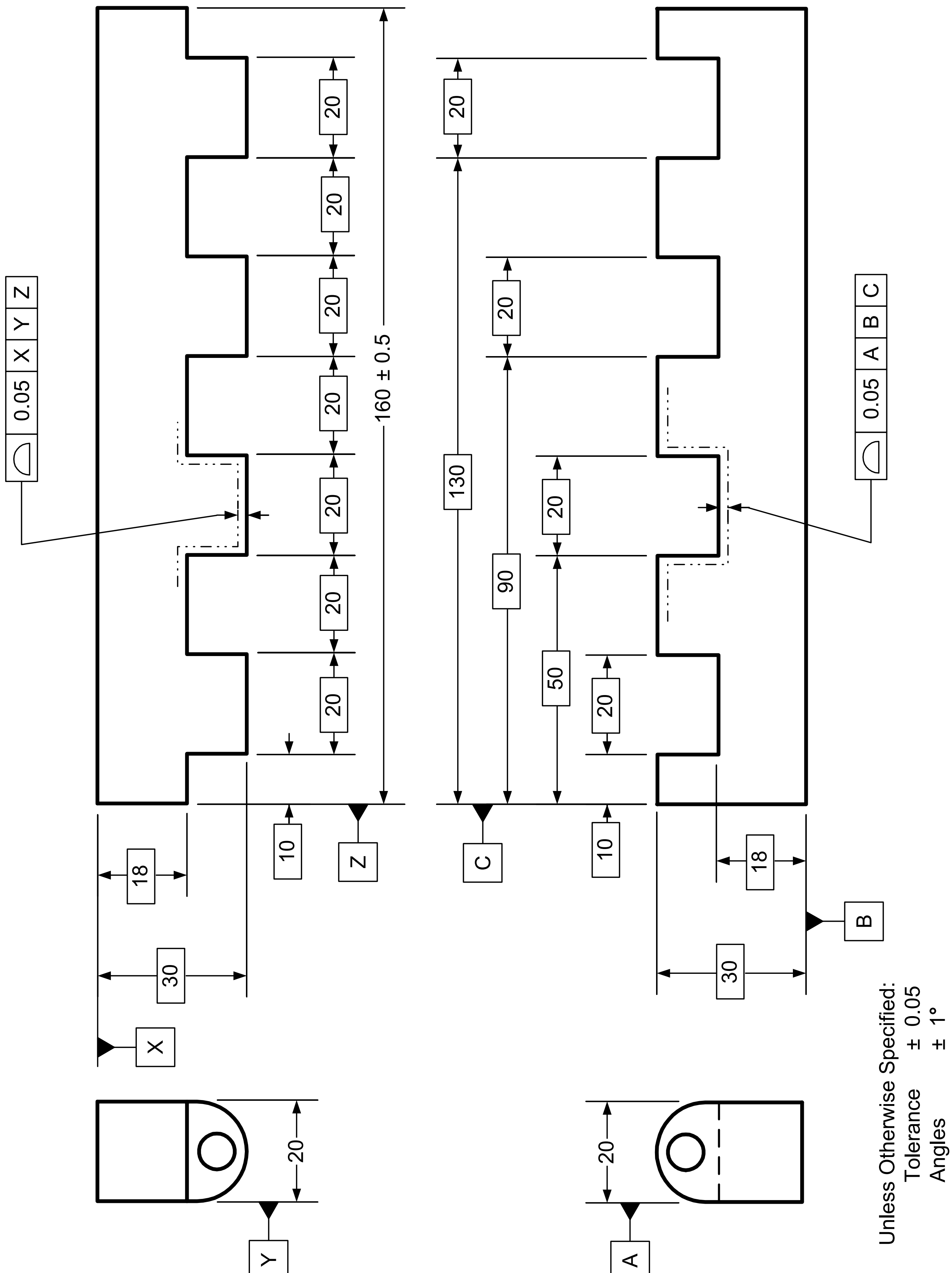
3. Profile of a Surface can also be used with a size tolerance to refine the size or shape. Following is an example where the three top surfaces are to be coplanar (in-line) within 0.3mm and in relation to the bottom surface of the part. Each surface is to flat within 0.1mm. Define these requirements on the drawing.

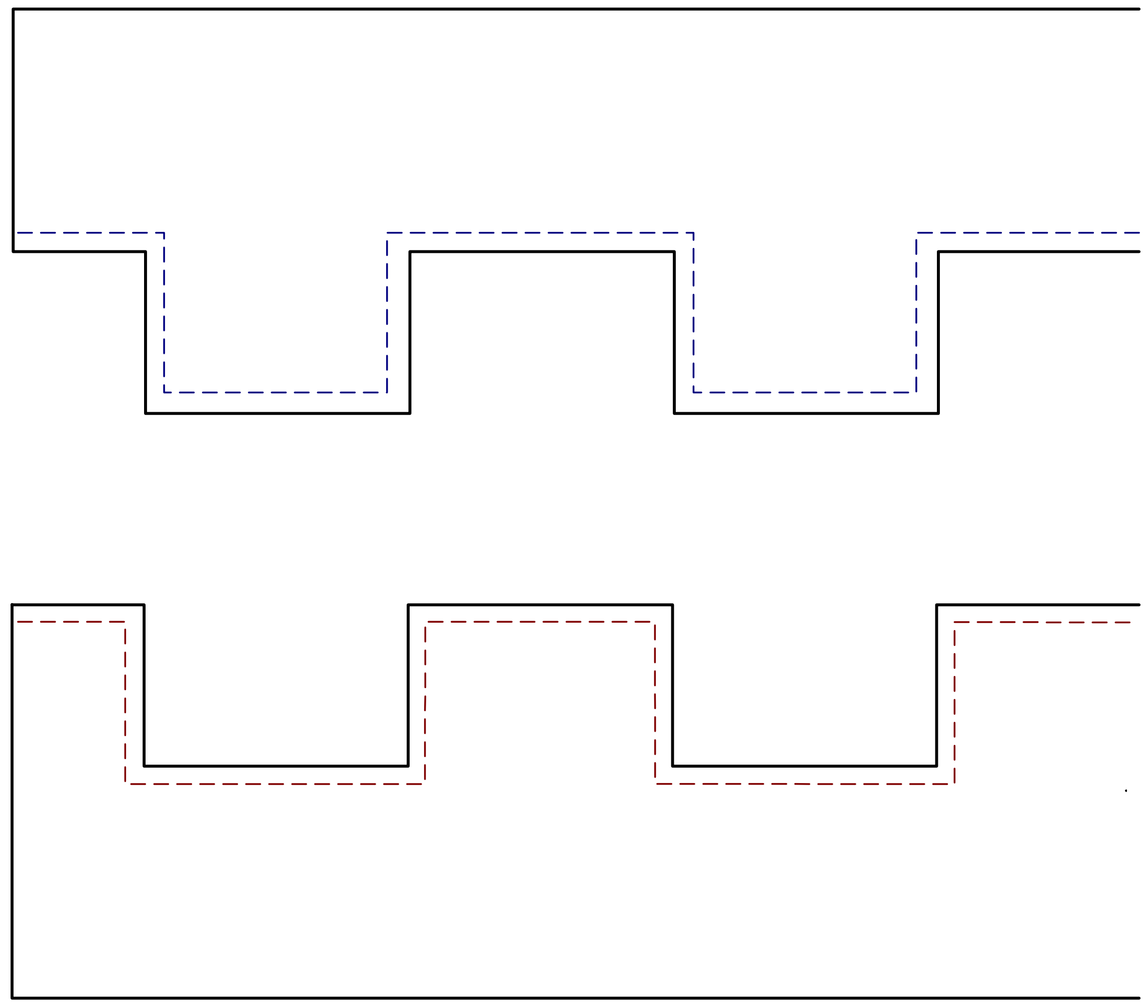



160 ± 0.5
12
30 ± 0.5
20
130
20
90
20
50
20
10
12
30 ± 0.5
Unless Otherwise Specified:
Tolerance ± 0.05
Angles ± 1°






0.05 X Y Z
0.05 A B C
160 ± 0.5
130
90
50
20
18
30
10
X
Y
Z
A
B
C
Unless Otherwise Specified:
Tolerance ± 0.05
Angles ± 1°



7. Runout Tolerance

7.1. Coaxial Features

There are three types of coaxial feature control. Proper selection is based upon which of the below controls best suits the functional design requirement.

Runout - Use where part feature surfaces in a rotational consideration must relate to a datum axis. Runout is applicable only on an RFS basis.

Runout

↗	0.5	A

Total Runout

⌰	0.5	A

(Surface to axis control, RFS)

Position - Use where part feature surfaces relate to a datum axis on a functional or inter-changeability basis; typically mating parts are involved. Position is normally applied only on an MMC basis (occasionally an RFS datum is used).

⌖	⌀0.5Ⓜ	AⓂ

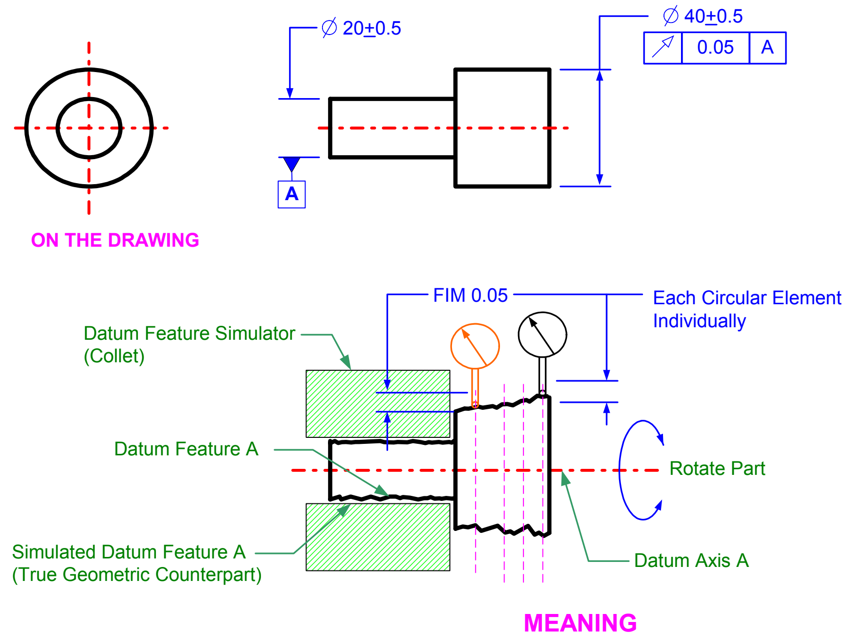
(Axis to axis control, MMC)

Concentricity - Use where part feature axis / axes in a rotational consideration must relate to a datum axis. Concentricity is applicable only on an RFS basis.

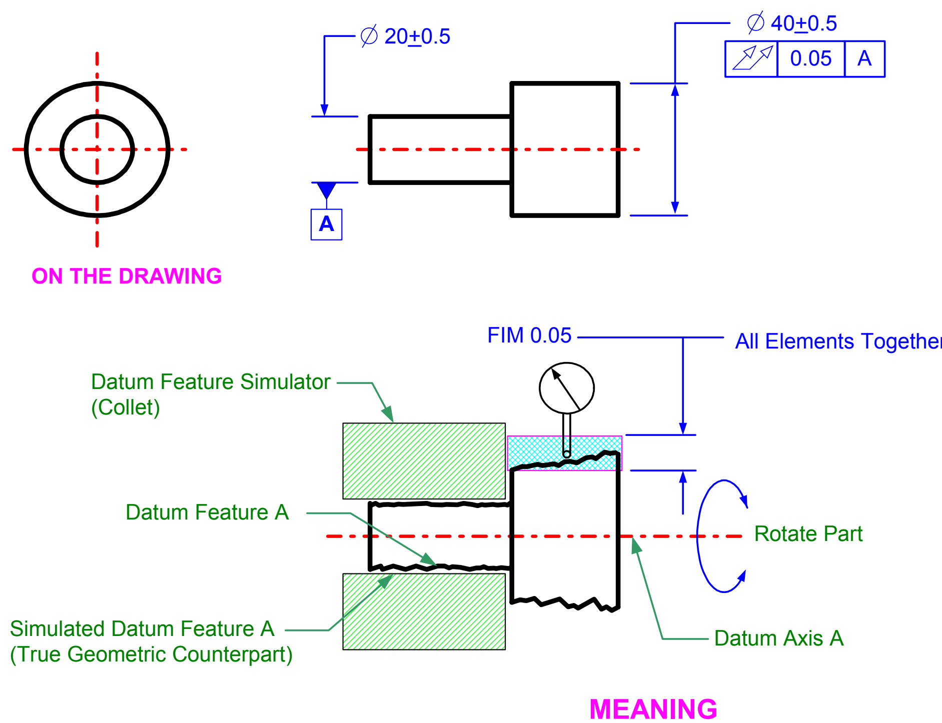
◎	⌀0.5	A

(Axis to axis control, RFS)

7.2. Runout



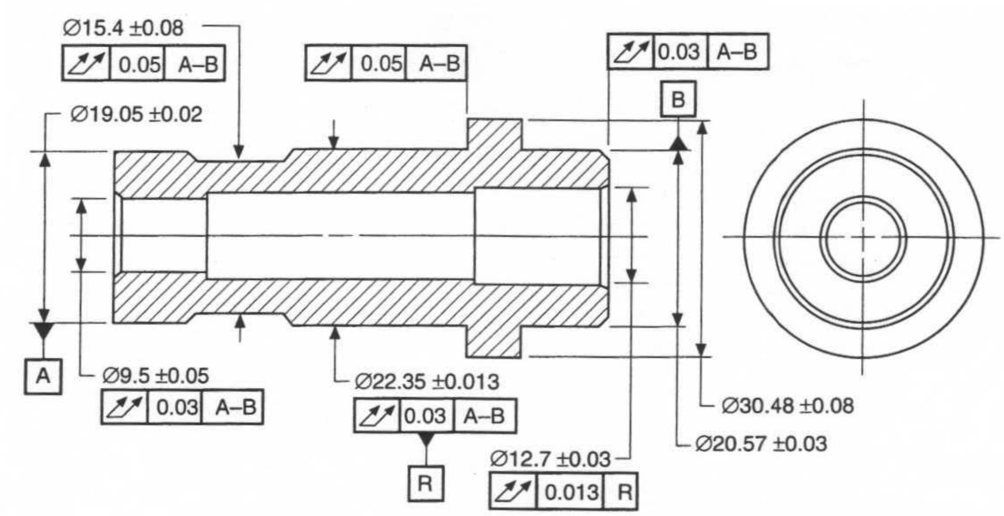
Runout: Each circular element of the feature must be within the runout tolerance and within 0.05 wide tolerance zone (FIM) in relation to datum axis A

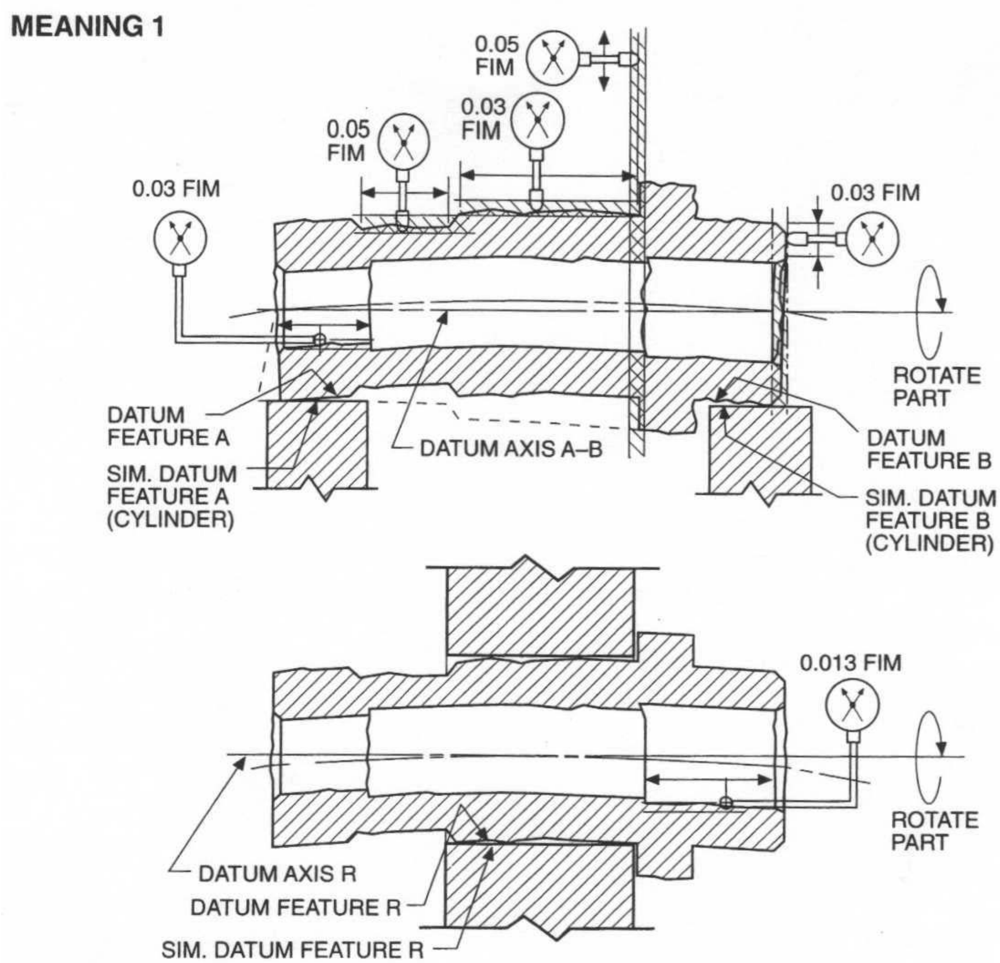


Total Runout: All surface elements, total, across entire surface must be within the runout tolerance and within 0.05 wide tolerance zone (FIM) in relation to datum axis A.

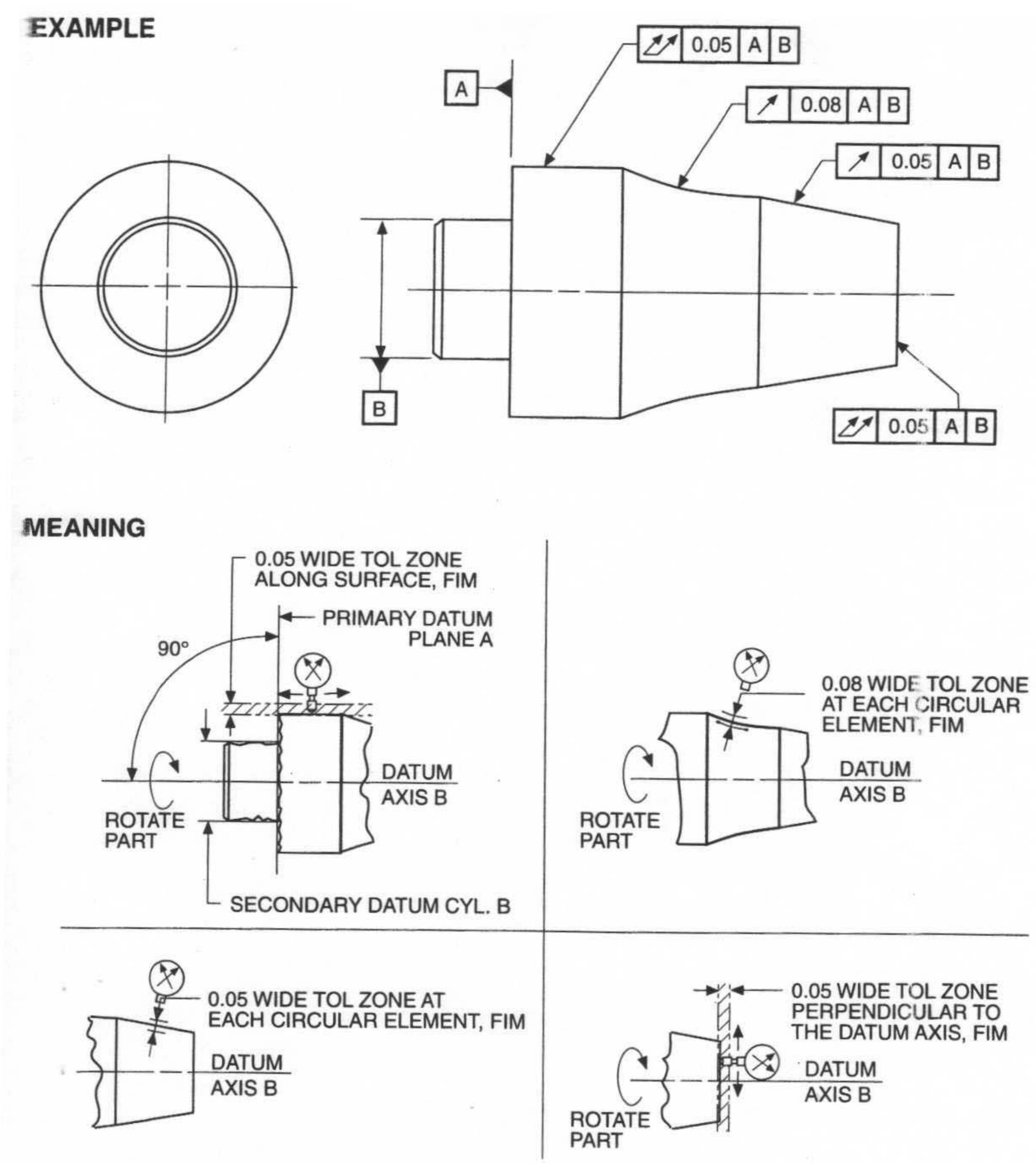
7.3. Examples

Part Mounted on Two Functional Diameters

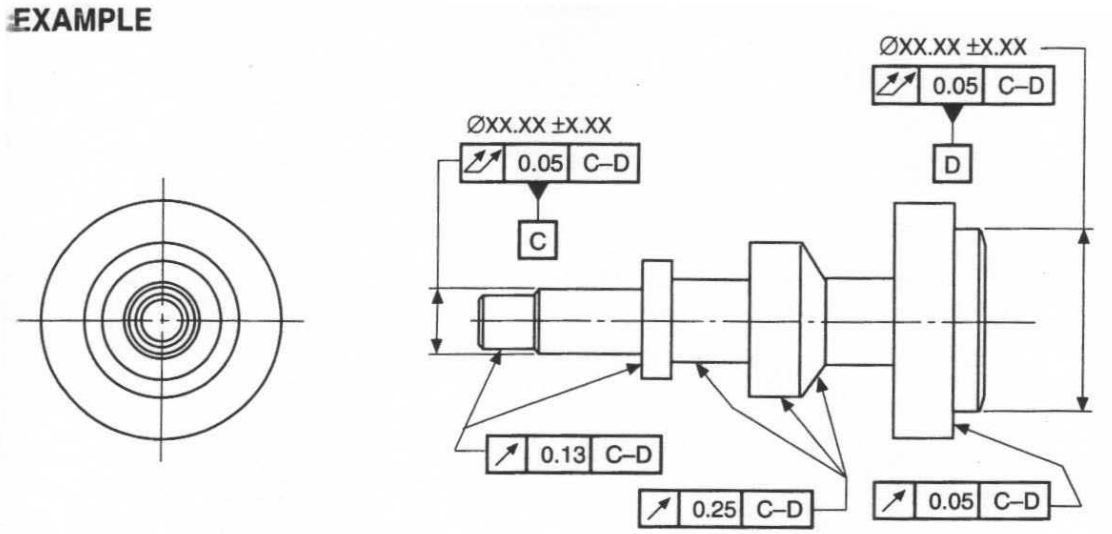


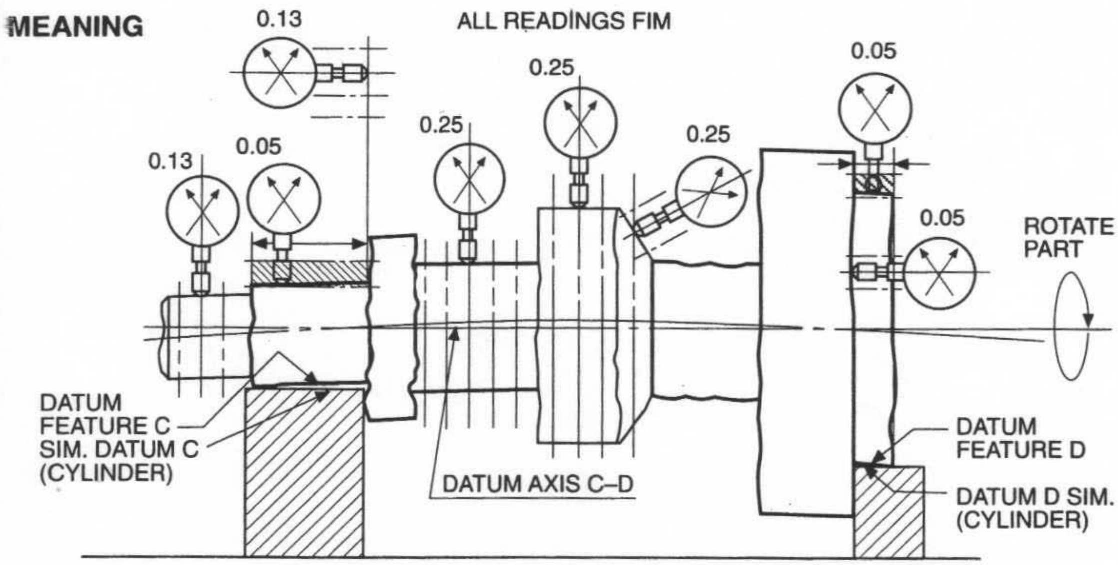


Part Mounted on Functional Face Surface (Datum) and Diameter (Datum)



Part Mounted on Two Functional Diameters

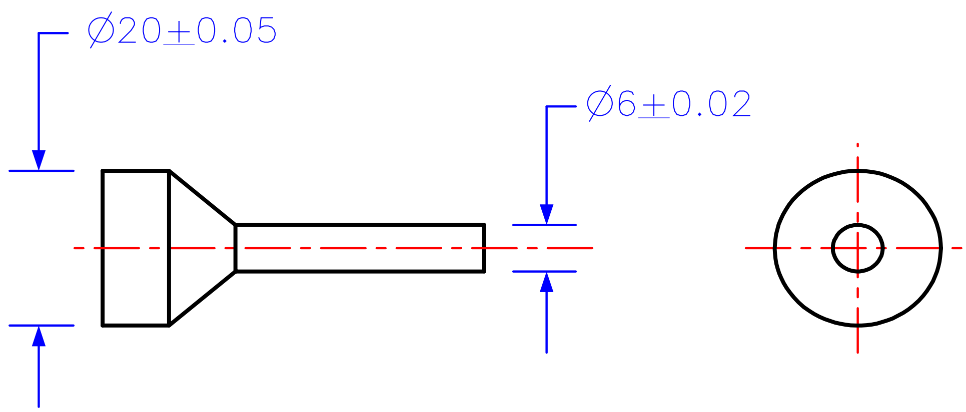




—EACH CIRCULAR ELEMENT INDIVIDUALLY—
—DATUM DIAMETERS C AND D ALL ELEMENTS TOGETHER (TOTAL)—

7.4. Exercise

1. What is the coaxiality requirement between the two diameters as expressed on the following drawing sketch? ______________________________



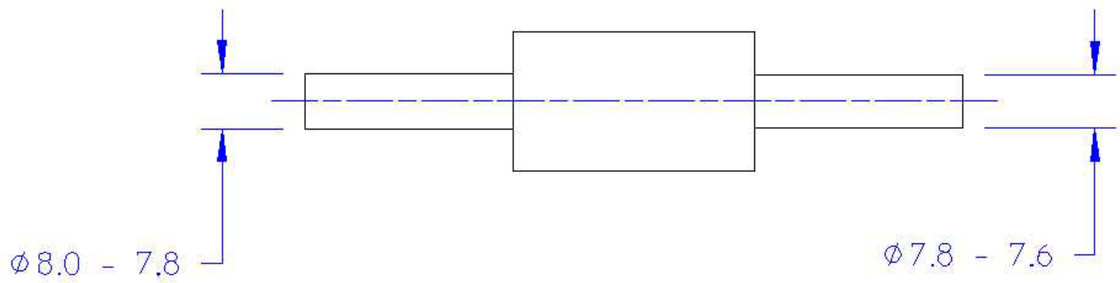
2. On the sketch, specify a 0.12mm circular runout requirement on the large
3. After defining the runout requirement, what is the maximum circularity (roundness) for the larger diameter? ____________________________________

4. With the specified runout requirement, what is the maximum position of the large diameter in relation to the small diameter? _____________________________

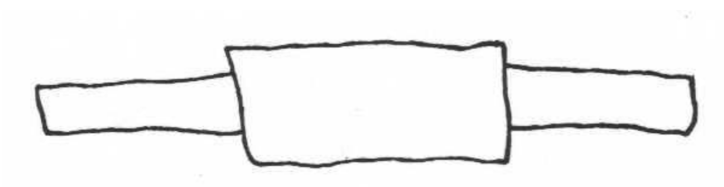
5. On the sketch, specify a runout requirement to make the left face perpendicular to the Datum axis within 0.1mm total.

6. On the figure below, sketch how the two runout requirements would be verified.

7. On the part drawing below, specify a 0.12mm total runout relating the large diameter to both of the two small diameters together.



8. On the figure below, sketch how the runout requirement would be verified.



9. Can runout be used without a datum feature reference?____________________

10. Can the Ⓜ or Ⓛ modifiers be used with runout?

____

11. What is the main difference between circular and total runout?

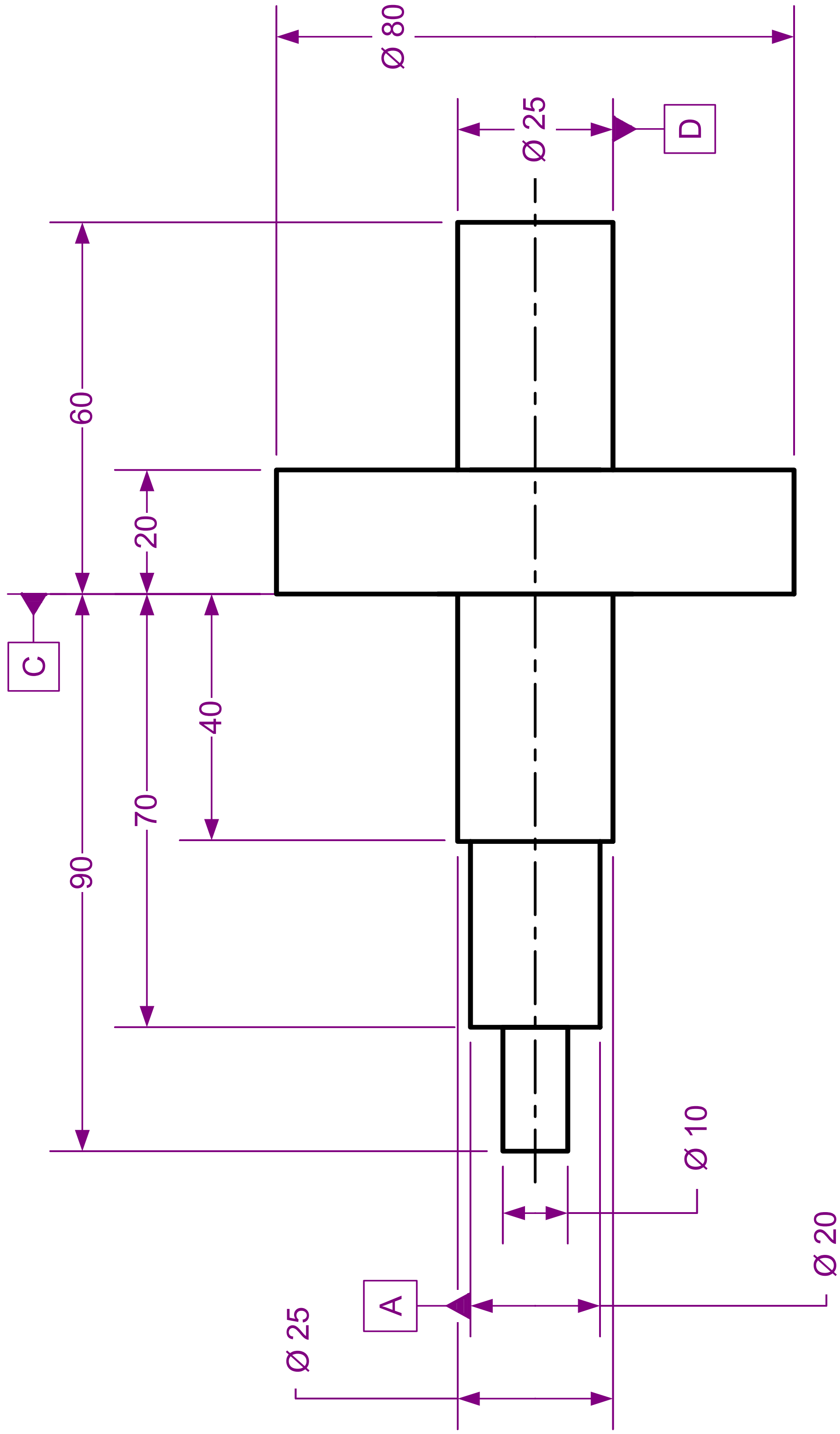
____

12. What is the main difference between runout and concentricity?

____

13. What is the main difference between concentricity and position?

____



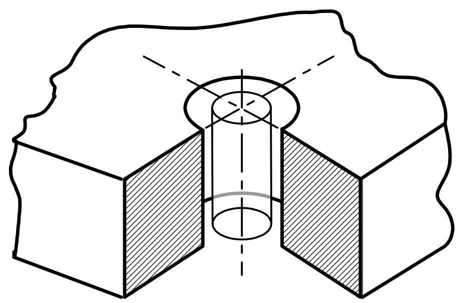
Ø 80
Ø 25
D
60
20
C
40
70
90
Ø 10
Ø 20
A
Ø 25

8. Location Tolerance

8.1. Position

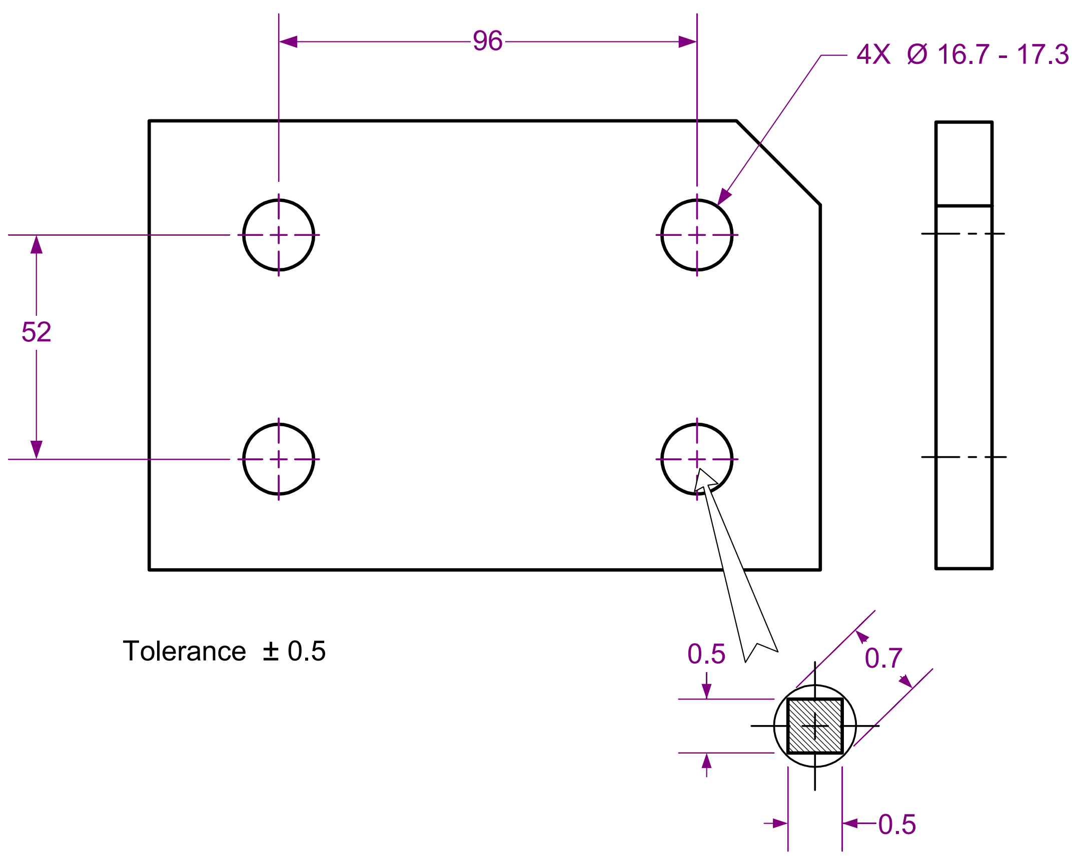
Hole Verification

Remember that all features have depth. Therefore, when doing design or making measurements, the tolerance zone must be considered from one end of the zone to the other.

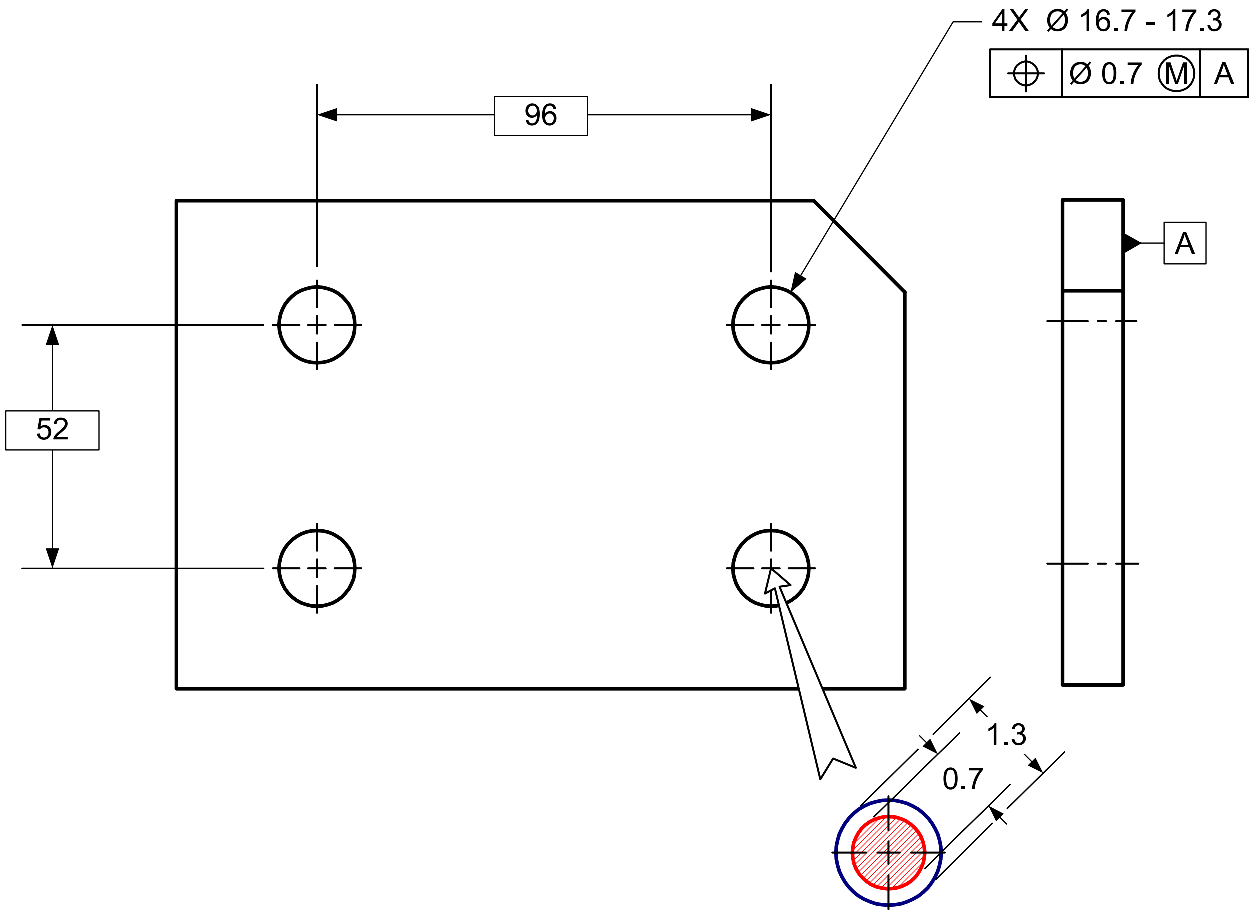


Position vs. Plus/Minus

Coordinate Tolerancing



Position Tolerancing
(Round zone)



- Each hole has its own positional tolerance zone. The zone size is dependent on the size of the produced hole.
- When the hole is produced at its MMC size, the positional tolerance zone is the tolerance stated in the FCF.
- If the hole is produced at something larger than the MMC size, the positional tolerance zone is stated FCF tolerance PLUS the amount that the hole is larger than MMC.

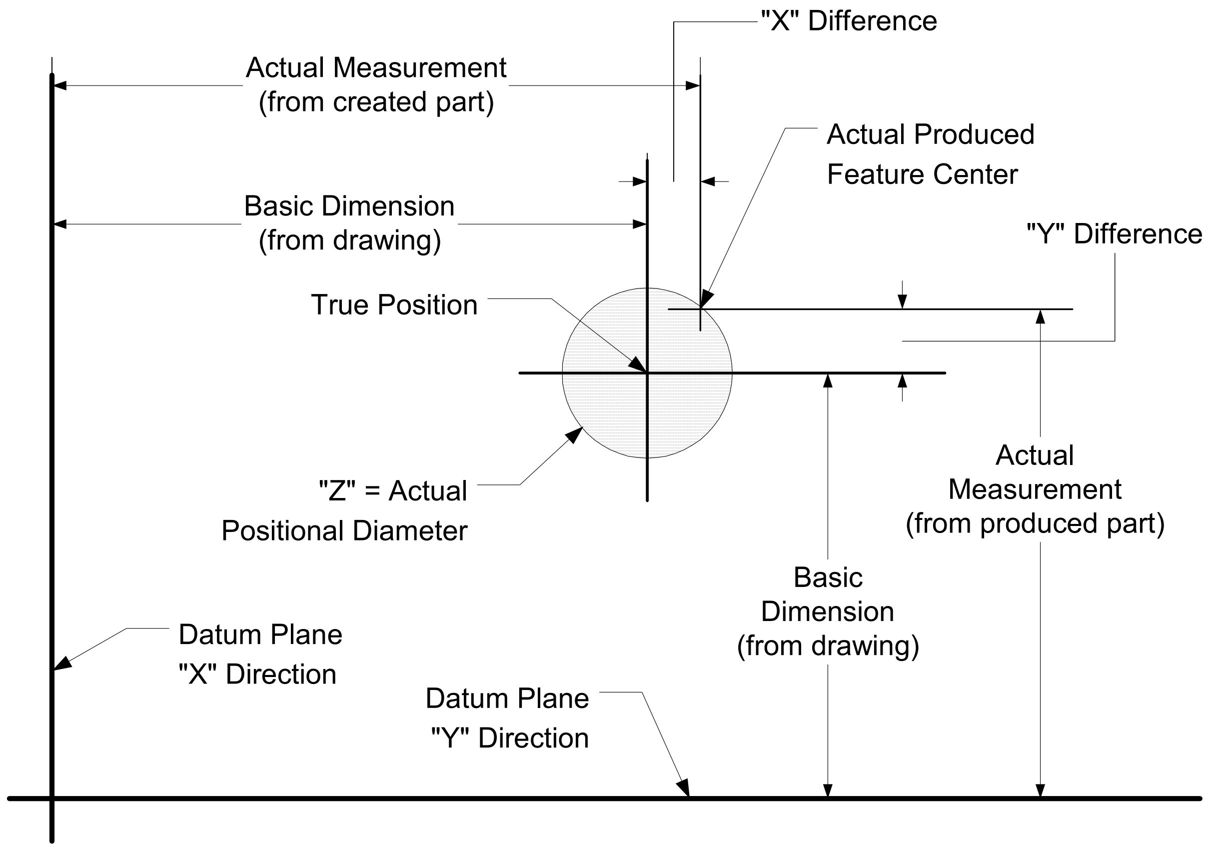
Example Features:

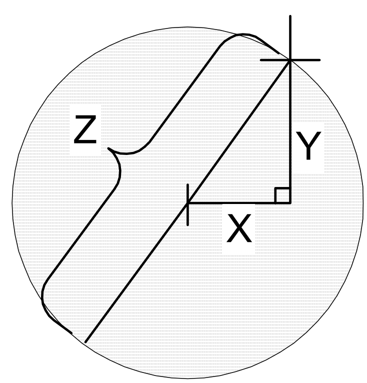
If Hole #1 = ∅ 16.7	Then tolerance zone	=
#2 = ∅ 16.9	tolerance zone	=
#3 = ∅ 17.2	tolerance zone	=
#4 = ∅ 17.4	tolerance zone	=

Position Zone

CONVERSION OF COORDINATE MEASUREMENTS TO POSITION LOCATION

0.20	0.400	0.402	0.404	0.408	0.412	0.418	0.424	0.431	0.439	0.447	0.457	0.466	0.477	0.488	0.500	0.512	0.525	0.538	0.552	0.566
0.19	0.381	0.382	0.385	0.388	0.393	0.398	0.405	0.412	0.420	0.429	0.439	0.449	0.460	0.472	0.484	0.497	0.510	0.523	0.537	0.552
0.18	0.361	0.362	0.365	0.369	0.374	0.379	0.386	0.394	0.402	0.412	0.422	0.433	0.444	0.456	0.469	0.482	0.495	0.509	0.523	0.538
0.17	0.341	0.342	0.345	0.349	0.354	0.361	0.368	0.376	0.385	0.394	0.405	0.416	0.428	0.440	0.453	0.467	0.481	0.495	0.510	0.525
0.16	0.321	0.322	0.326	0.330	0.335	0.342	0.349	0.358	0.367	0.377	0.388	0.400	0.412	0.425	0.439	0.453	0.467	0.482	0.497	0.512
0.15	0.301	0.303	0.306	0.310	0.316	0.323	0.331	0.340	0.350	0.361	0.372	0.384	0.397	0.410	0.424	0.439	0.453	0.469	0.484	0.500
0.14	0.281	0.283	0.286	0.291	0.297	0.305	0.313	0.322	0.333	0.344	0.356	0.369	0.382	0.396	0.410	0.425	0.440	0.456	0.472	0.488
0.13	0.261	0.263	0.267	0.272	0.279	0.286	0.295	0.305	0.316	0.328	0.341	0.354	0.368	0.382	0.397	0.412	0.428	0.444	0.460	0.477
0.12	0.241	0.243	0.247	0.253	0.260	0.268	0.278	0.288	0.300	0.312	0.326	0.339	0.354	0.369	0.384	0.400	0.416	0.433	0.449	0.466
0.11	0.221	0.224	0.228	0.234	0.242	0.251	0.261	0.272	0.284	0.297	0.311	0.326	0.341	0.356	0.372	0.388	0.405	0.422	0.439	0.457
0.10	0.201	0.204	0.209	0.215	0.224	0.233	0.244	0.256	0.269	0.283	0.297	0.312	0.328	0.344	0.361	0.377	0.394	0.412	0.429	0.447
0.09	0.181	0.184	0.190	0.197	0.206	0.216	0.228	0.241	0.255	0.269	0.284	0.300	0.316	0.333	0.350	0.367	0.385	0.402	0.420	0.439
0.08	0.161	0.165	0.171	0.179	0.189	0.200	0.213	0.226	0.241	0.256	0.272	0.288	0.305	0.322	0.340	0.358	0.376	0.394	0.412	0.431
0.07	0.141	0.146	0.152	0.161	0.172	0.184	0.198	0.213	0.228	0.244	0.261	0.278	0.295	0.313	0.331	0.349	0.368	0.386	0.405	0.424
0.06	0.122	0.126	0.134	0.144	0.156	0.170	0.184	0.200	0.216	0.233	0.251	0.268	0.286	0.305	0.323	0.342	0.361	0.379	0.398	0.418
0.05	0.102	0.108	0.117	0.128	0.141	0.156	0.172	0.189	0.206	0.224	0.242	0.260	0.279	0.297	0.316	0.335	0.354	0.374	0.393	0.412
0.04	0.082	0.089	0.100	0.113	0.128	0.144	0.161	0.179	0.197	0.215	0.234	0.253	0.272	0.291	0.310	0.330	0.349	0.369	0.388	0.408
0.03	0.063	0.072	0.085	0.100	0.117	0.134	0.152	0.171	0.190	0.209	0.228	0.247	0.267	0.286	0.306	0.326	0.345	0.365	0.385	0.404
0.02	0.045	0.057	0.072	0.089	0.108	0.126	0.146	0.165	0.184	0.204	0.224	0.243	0.263	0.283	0.303	0.322	0.342	0.362	0.382	0.402
0.01	0.028	0.045	0.063	0.082	0.102	0.122	0.141	0.161	0.181	0.201	0.221	0.241	0.261	0.281	0.301	0.321	0.341	0.361	0.381	0.400
X / Y	0.01	0.02	0.03	0.04	0.05	0.06	0.07	0.08	0.09	0.10	0.11	0.12	0.13	0.14	0.15	0.16	0.17	0.18	0.19	0.20

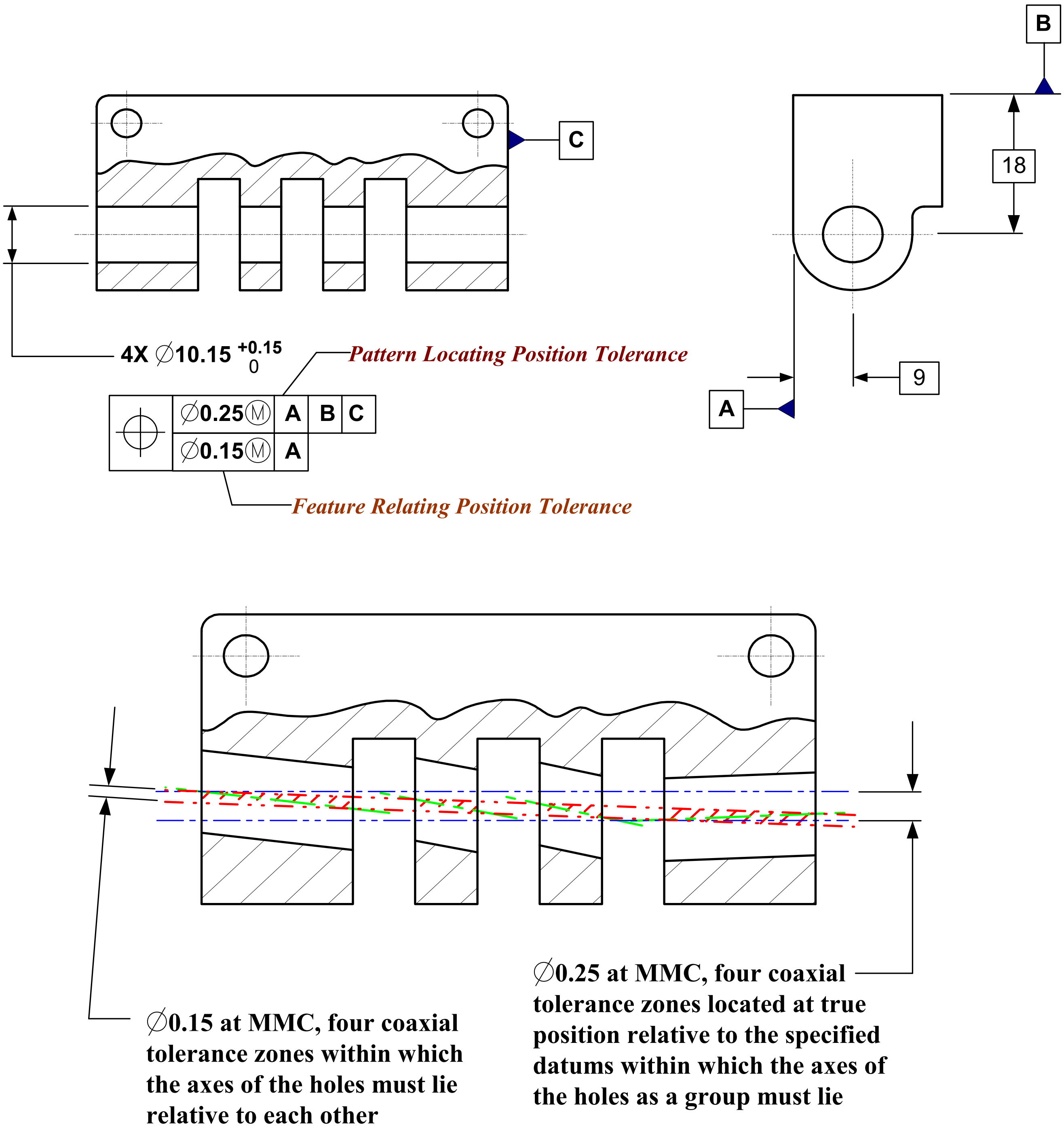




$$\text{Formula}: Z = 2\sqrt{X^2 + Y^2}$$

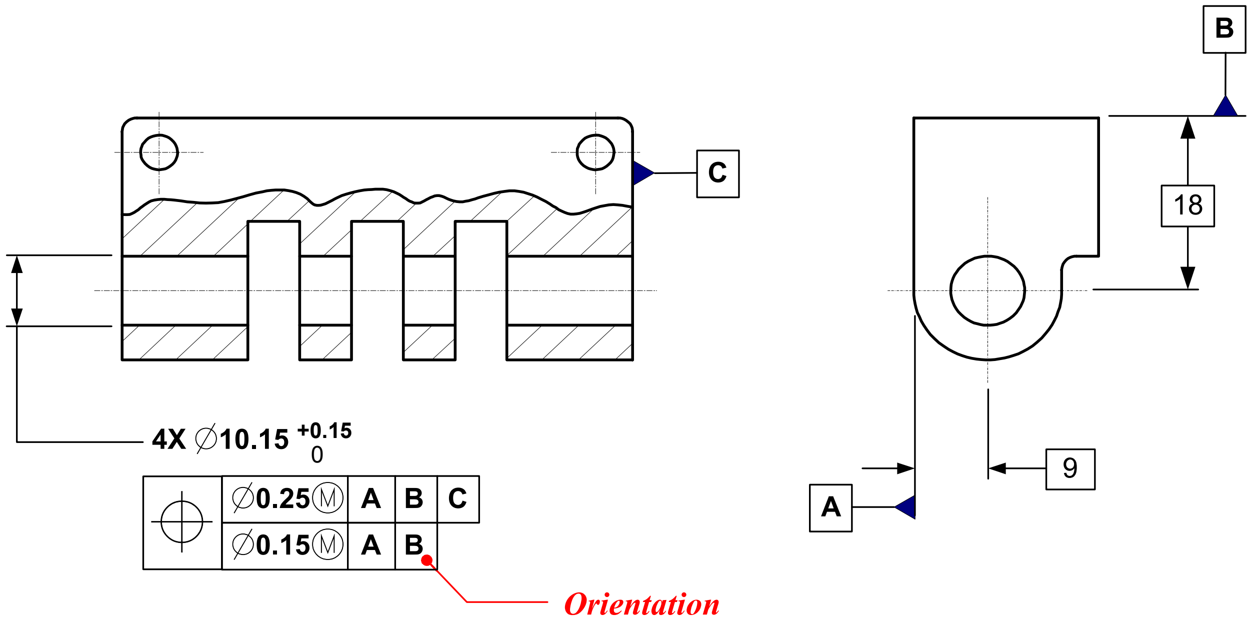
8.2. Composite Positional Tolerance

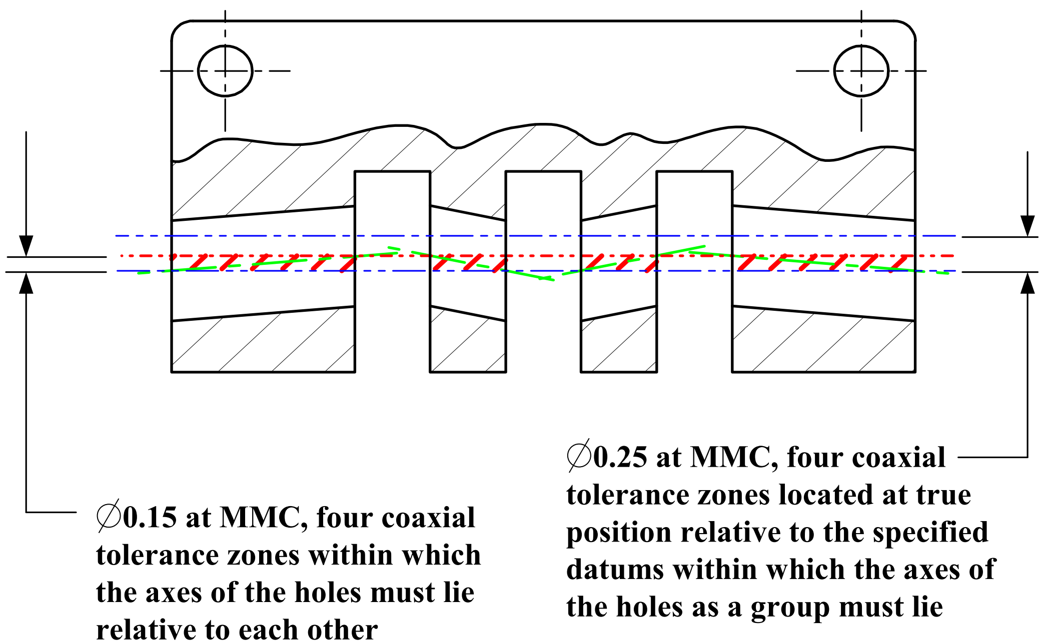
Position, Axis to Surface, Coaxial
Axis to Axis, Coaxial



Positional Tolerancing for Coaxial Holes of Same Size

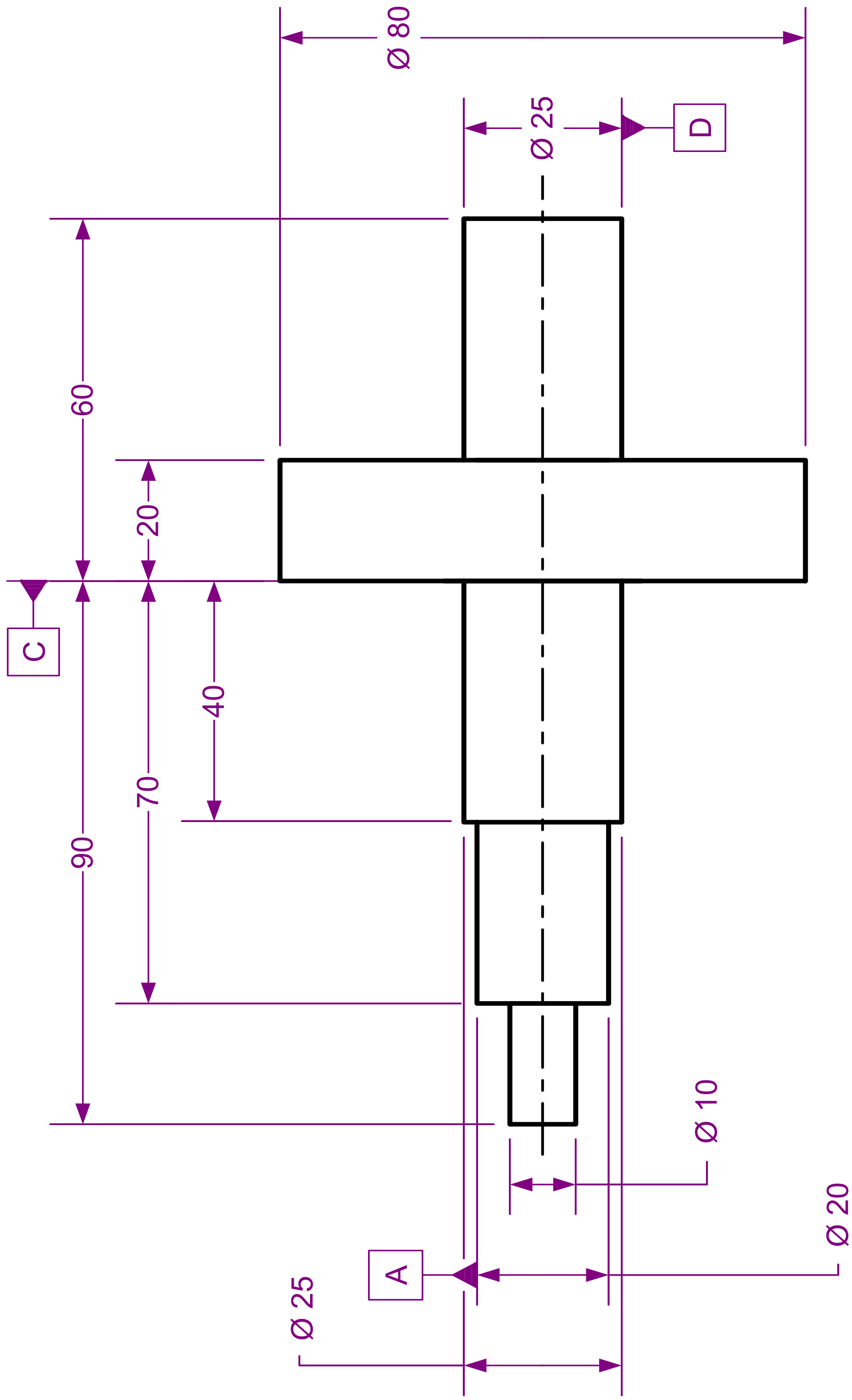
Position, Axis to Surface, Coaxial
Axis to Axis, Coaxial

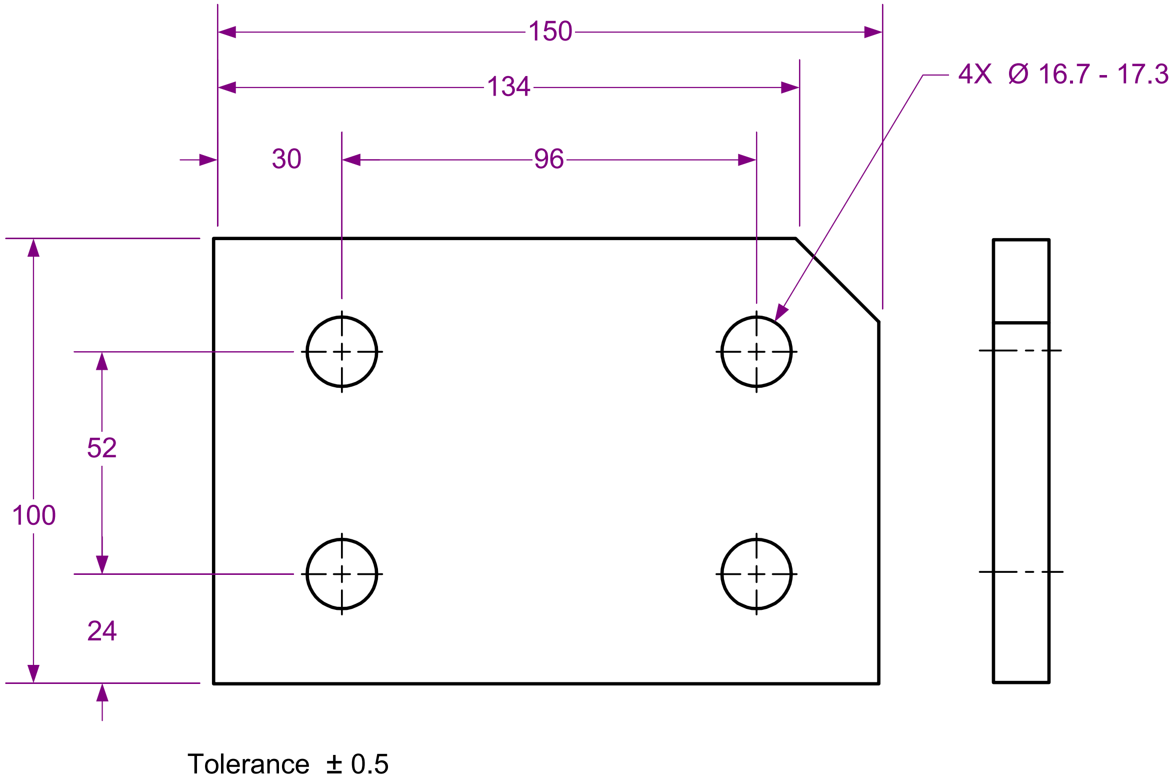


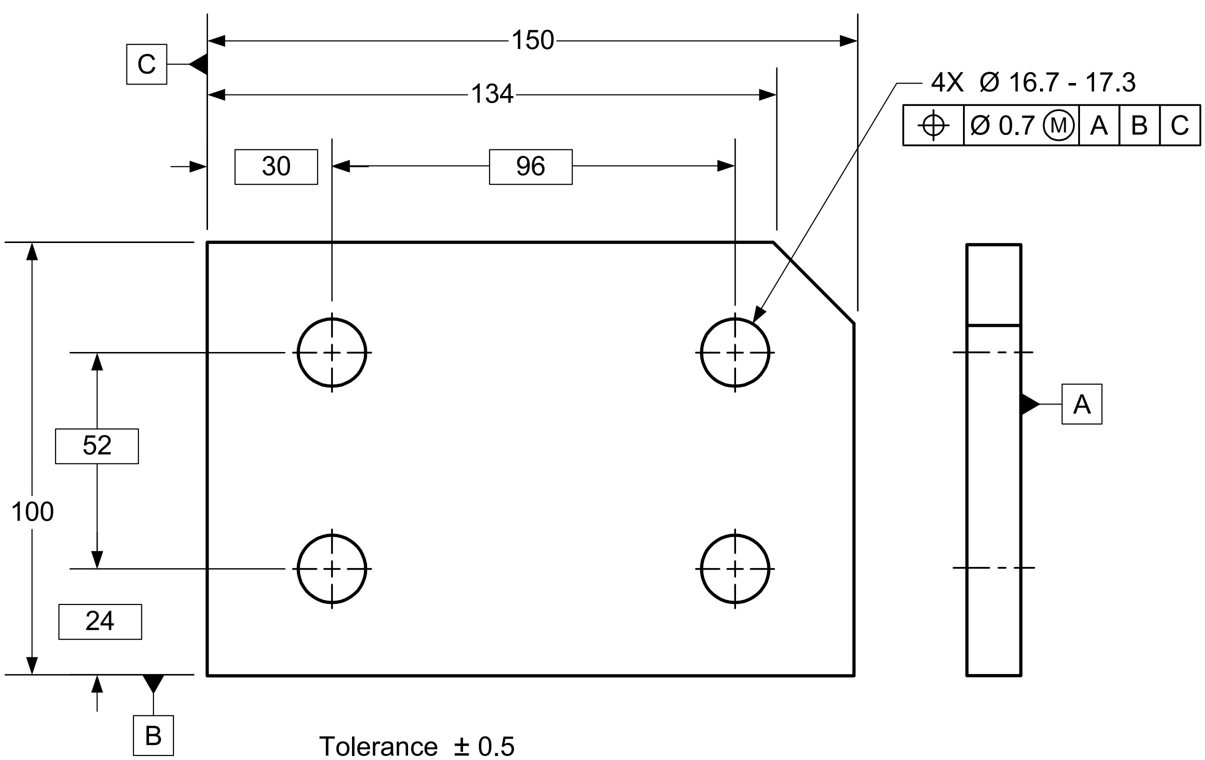


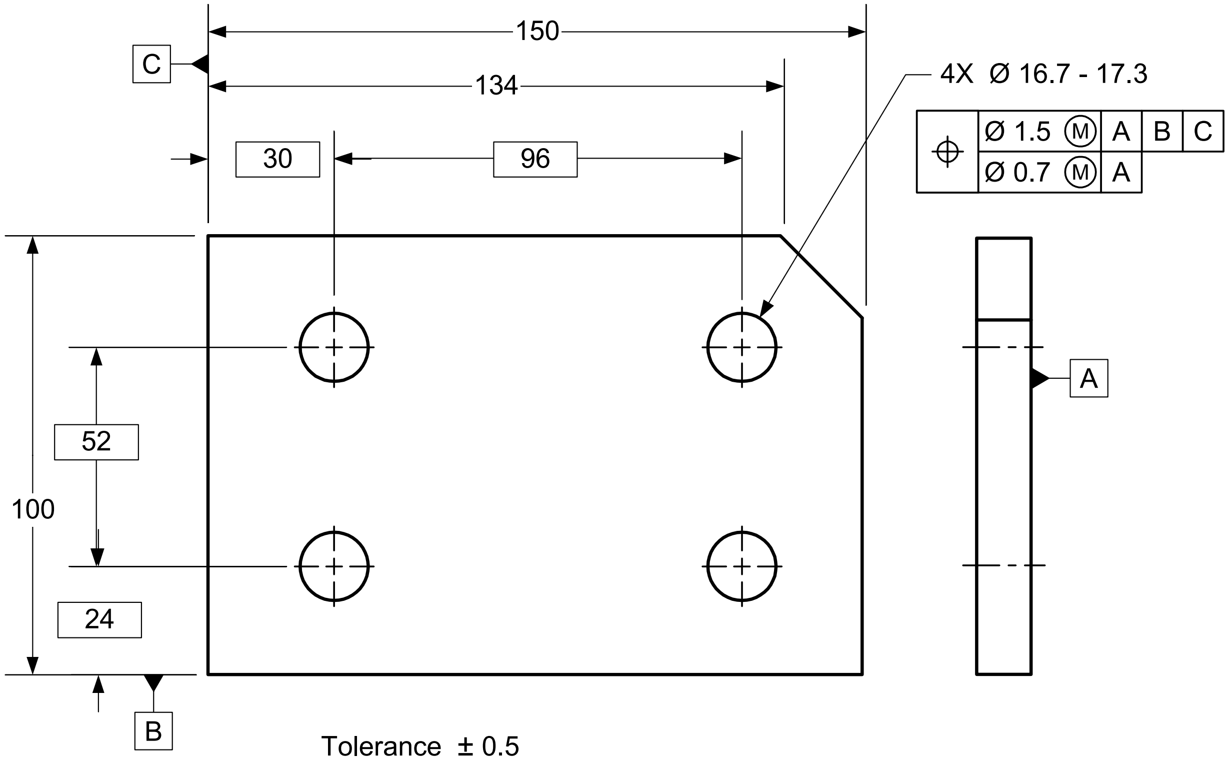
Positional Tolerancing for Coaxial Holes of Same Size, Partial (Parallelism) Refinement of Feature-Relating Axis

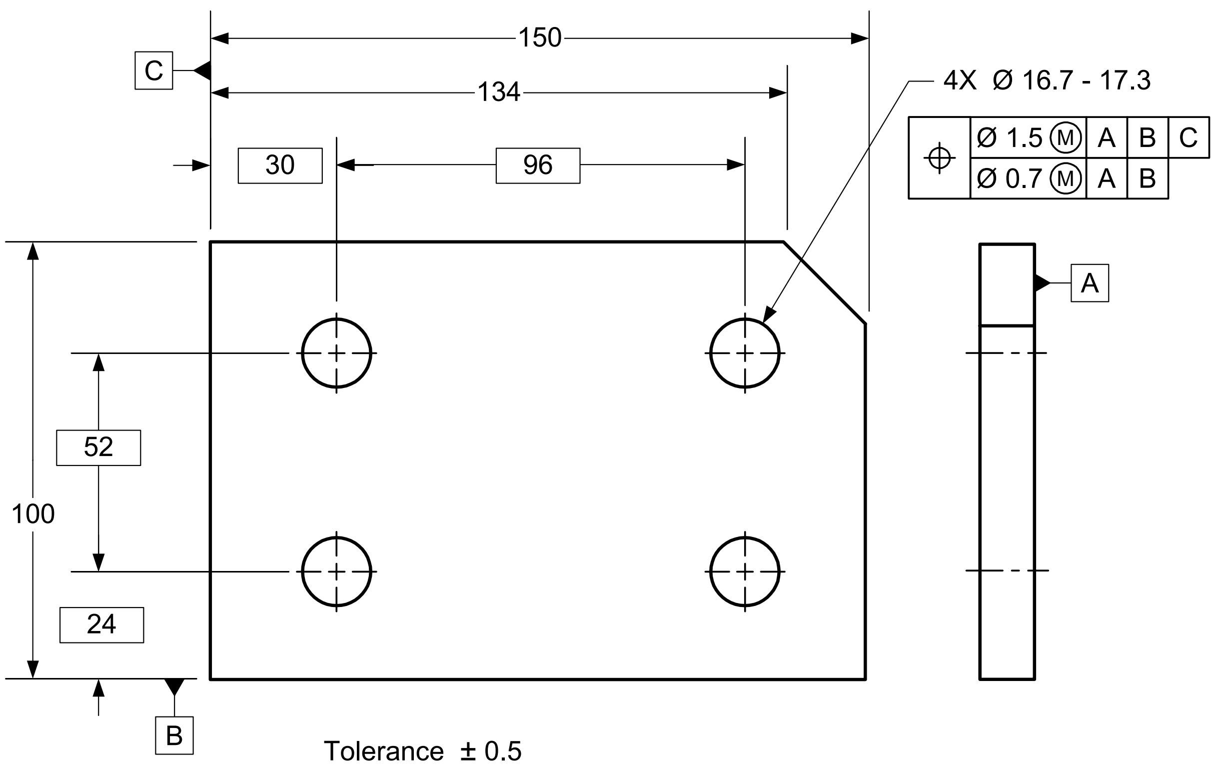
Position, Axis to Axis, Coaxial

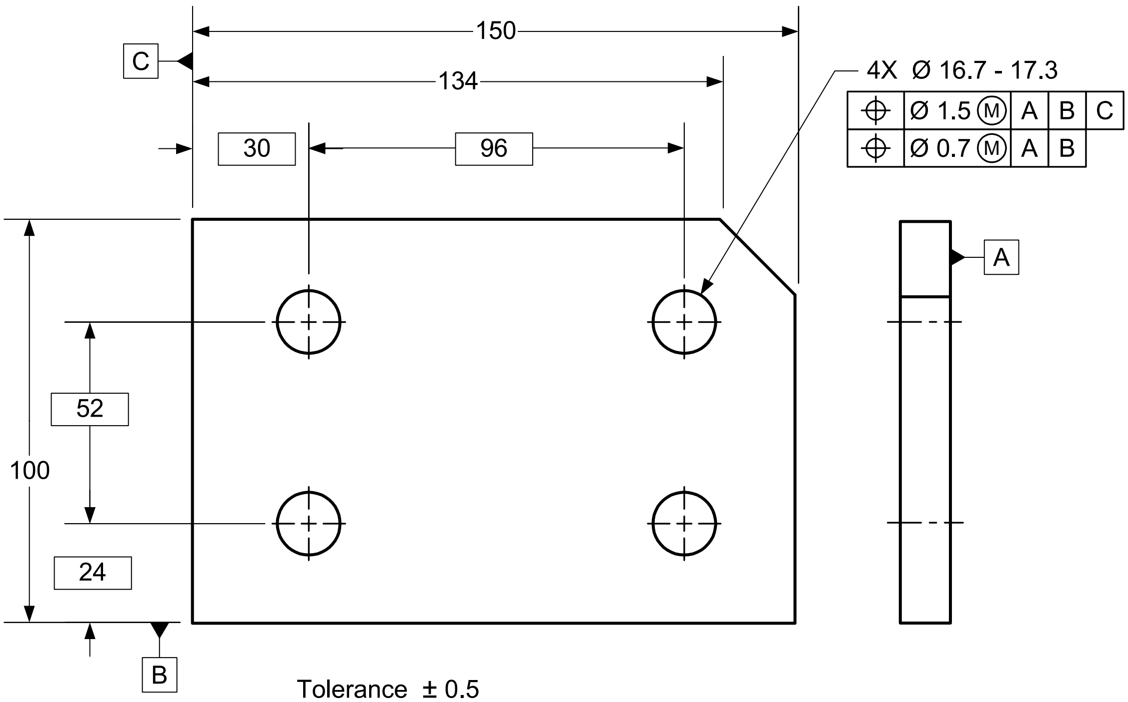



150
134
30
96
4X Ø 16.7 - 17.3
100
52
24
Tolerance ± 0.5


C
150
134
30
96
4X Ø 16.7 - 17.3
Ø 0.7 Ⓜ A B C
A
100
52
24
B
Tolerance ± 0.5

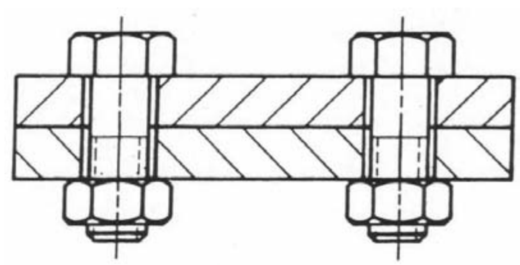

150
134
C
30
96
4X Ø 16.7 - 17.3
Ø 1.5 Ⓜ A B C
Ø 0.7 Ⓜ A
A
52
100
24
B
Tolerance ± 0.5


150
134
C
30
96
4X Ø 16.7 - 17.3
Ø 1.5 Ⓜ A B C
Ø 0.7 Ⓜ A B
A
52
100
24
B
Tolerance ± 0.5



8.3. Position Tolerance Calculation

Floating Fastener

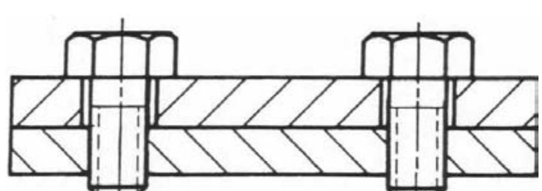


To calculate position tolerance with fastener and hole size known:

$$\mathbf{T} = \mathbf{H} - \mathbf{F}$$

Where T = tolerance, H = MMC hole, and F = MMC fastener

Fixed Fastener



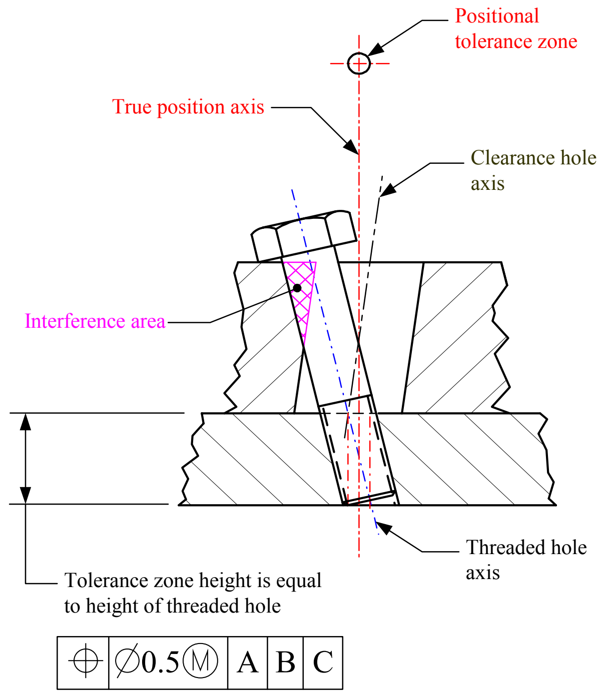
To calculate position tolerance with fastener and hole size known:

$$\mathbf{T} = \frac{\mathbf{H} - \mathbf{F}}{\mathbf{2}}$$

Where T = tolerance, H = MMC hole, and F = MMC fastener

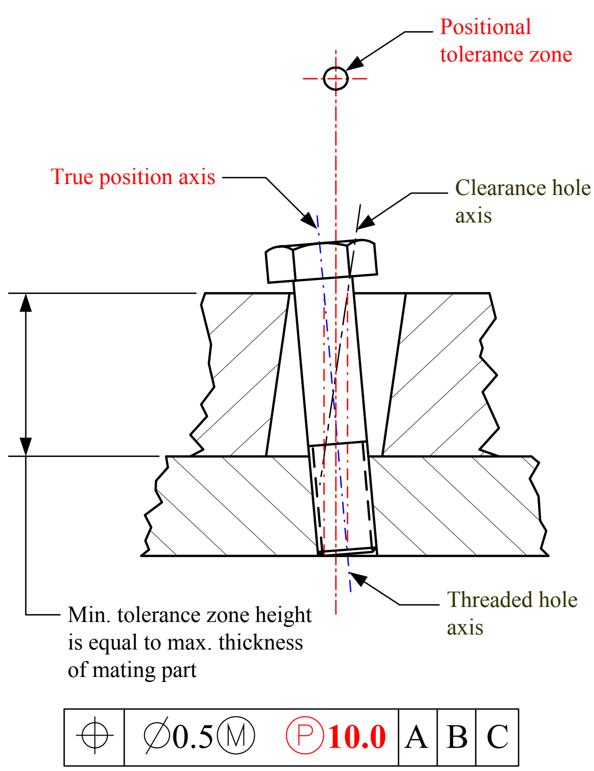
8.4. Projected Tolerance Zone

(1)



Interference diagram, fastener and hole

(2)

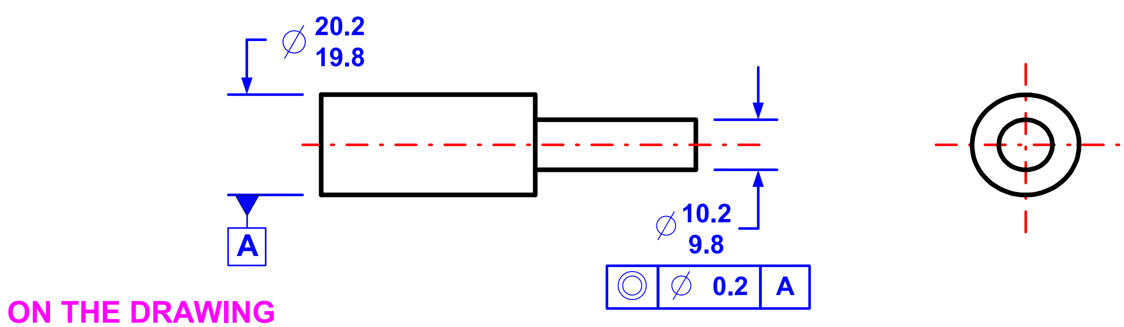


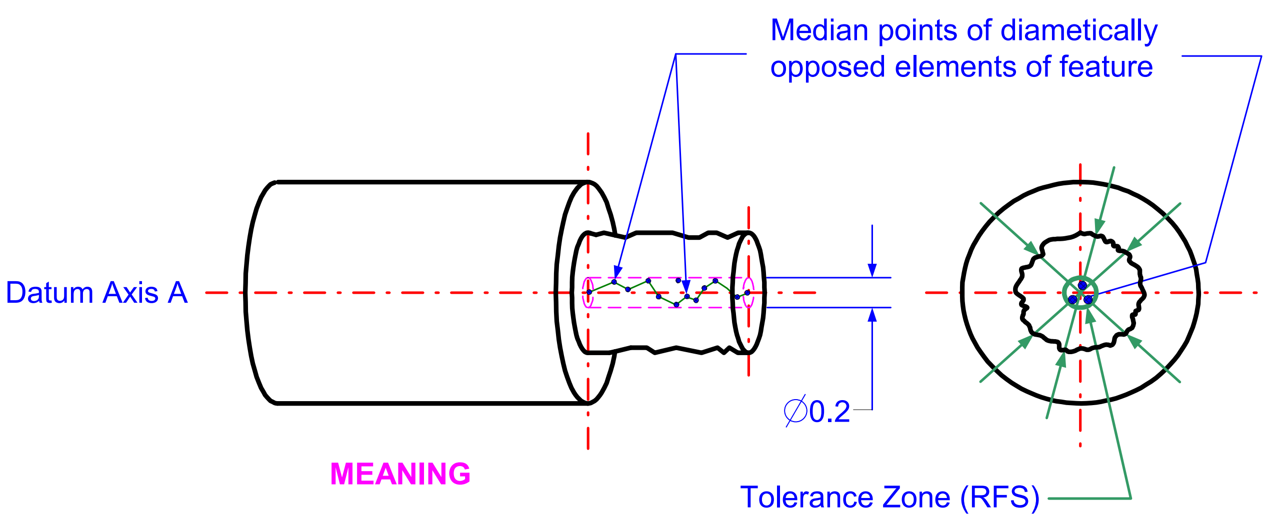
Basis for projected tolerance zone

8.5. Concentricity

Concentricity is that condition where the median points of all diametrically opposed elements of a figure of revolution (or corresponding-located elements of two or more radially disposed features) are congruent with the axis (or center point) of a datum.

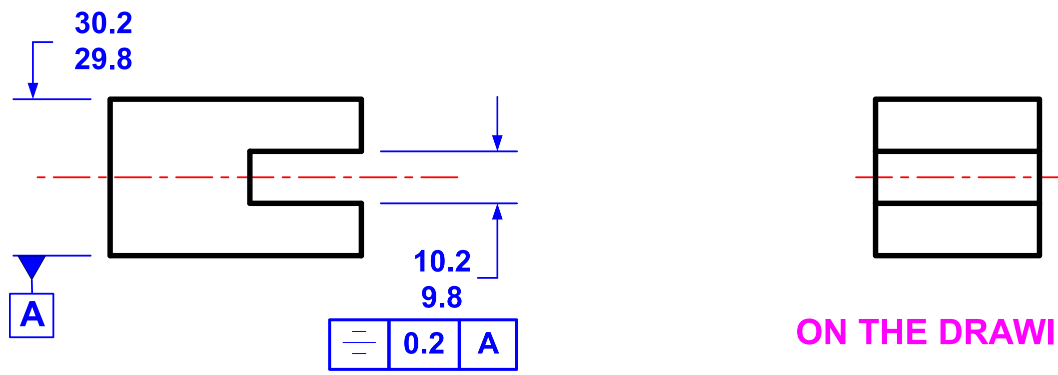
A concentricity tolerance is a cylindrical (or spherical) tolerance zone whose axis (or center point) coincides with the axis (or center point) of the datum feature. The median points of all correspondingly-located elements of the feature being controlled, regardless of feature size, must be within the cylindrical tolerance zone. The specified tolerance and the datum reference can only apply on an RFS basis.





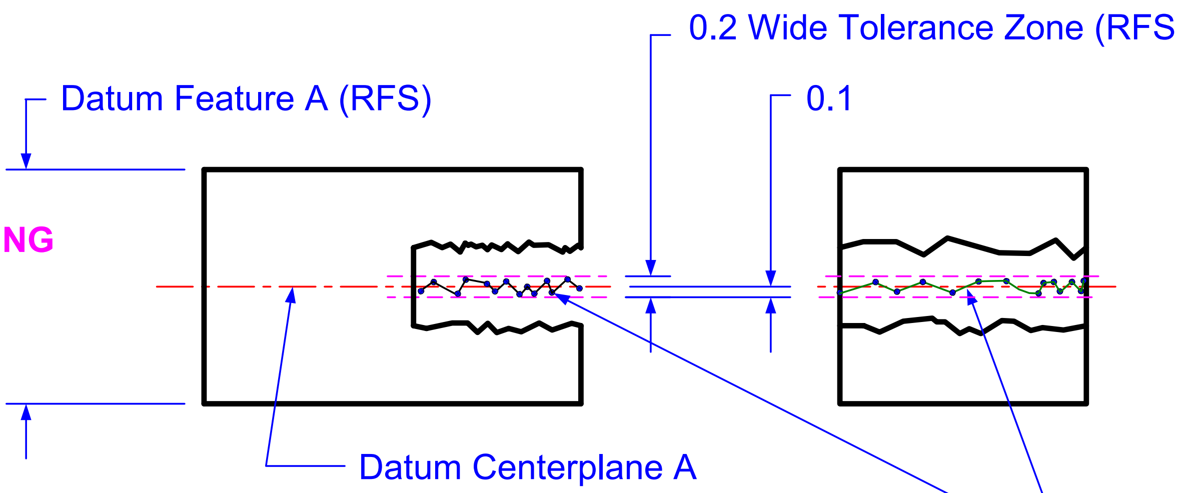
8.6. Symmetry

Symmetry is that condition where the median points of all opposed or correspondingly located elements of two or more feature surfaces are congruent with the axis or center plane of a datum feature. The material condition RFS only is to apply.



ON THE DRAWING

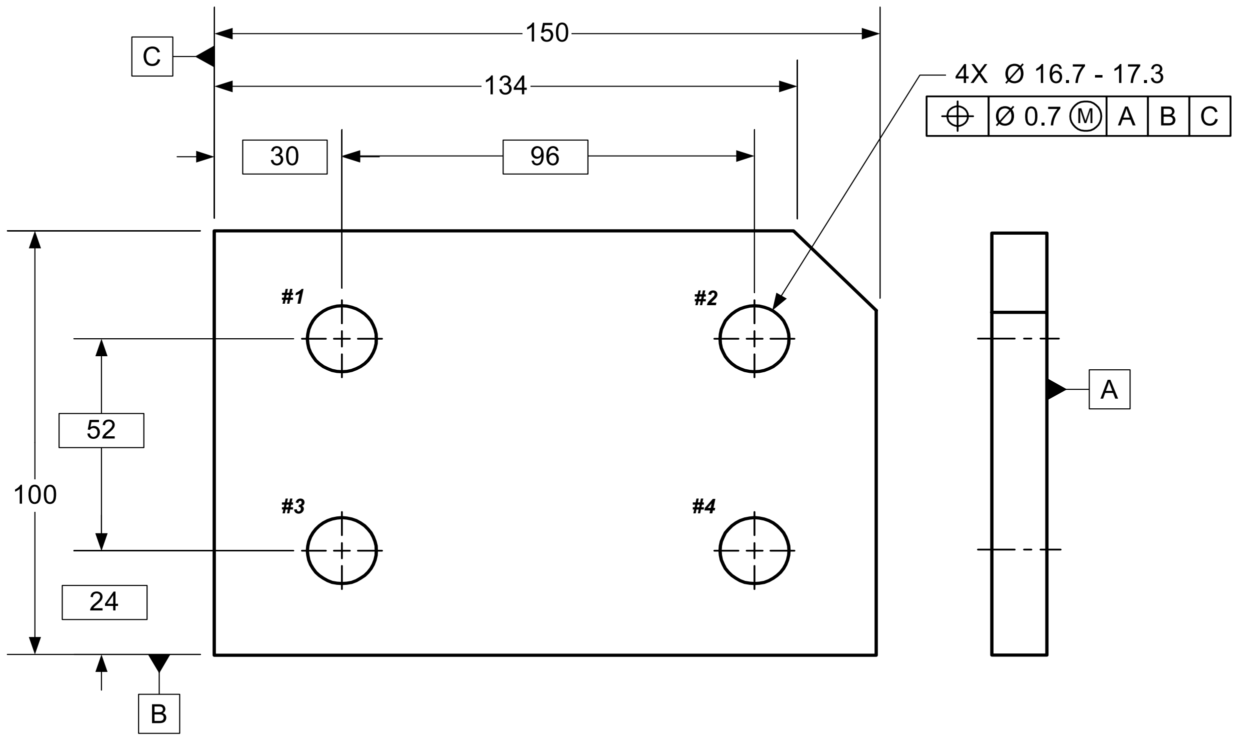
MEANING



All median points of opposed elements of the slot must lie within the 0.2 wide tolerance zone, RFS. The tolerance zone being established by two paralle planes equally disposed about datum centerplane A, RFS.

8.7. Exercise

1.

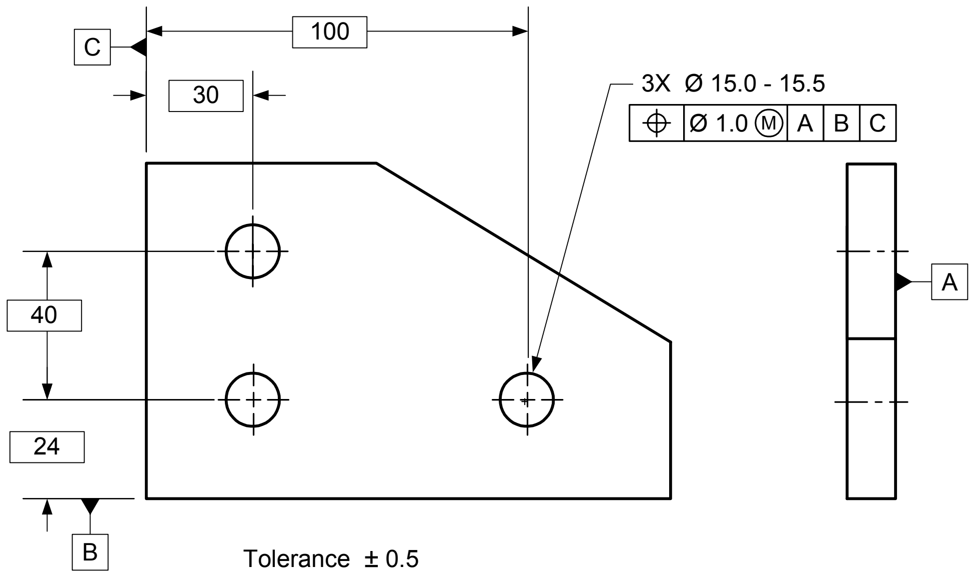


Inspection Report			
Hole #	Actual "X" Location	Actual "Y" Location	Hole Size
1	29.75	76.5	17.2
2	126.3	75.85	17
3	30.4	24.43	16.9
4	125.91	23.48	17.1

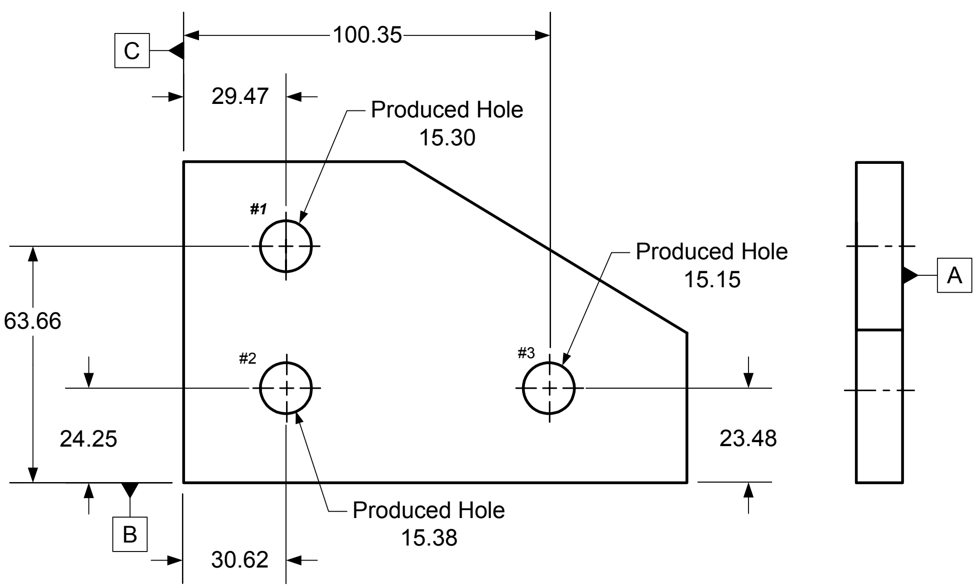
Hole #	Actual Location		Hole Size	Position Tol.	Accept	Reject
1	X =	Y =				
2	X =	Y =				
3	X =	Y =				
4	X =	Y =				

2.

Drawing Requirements

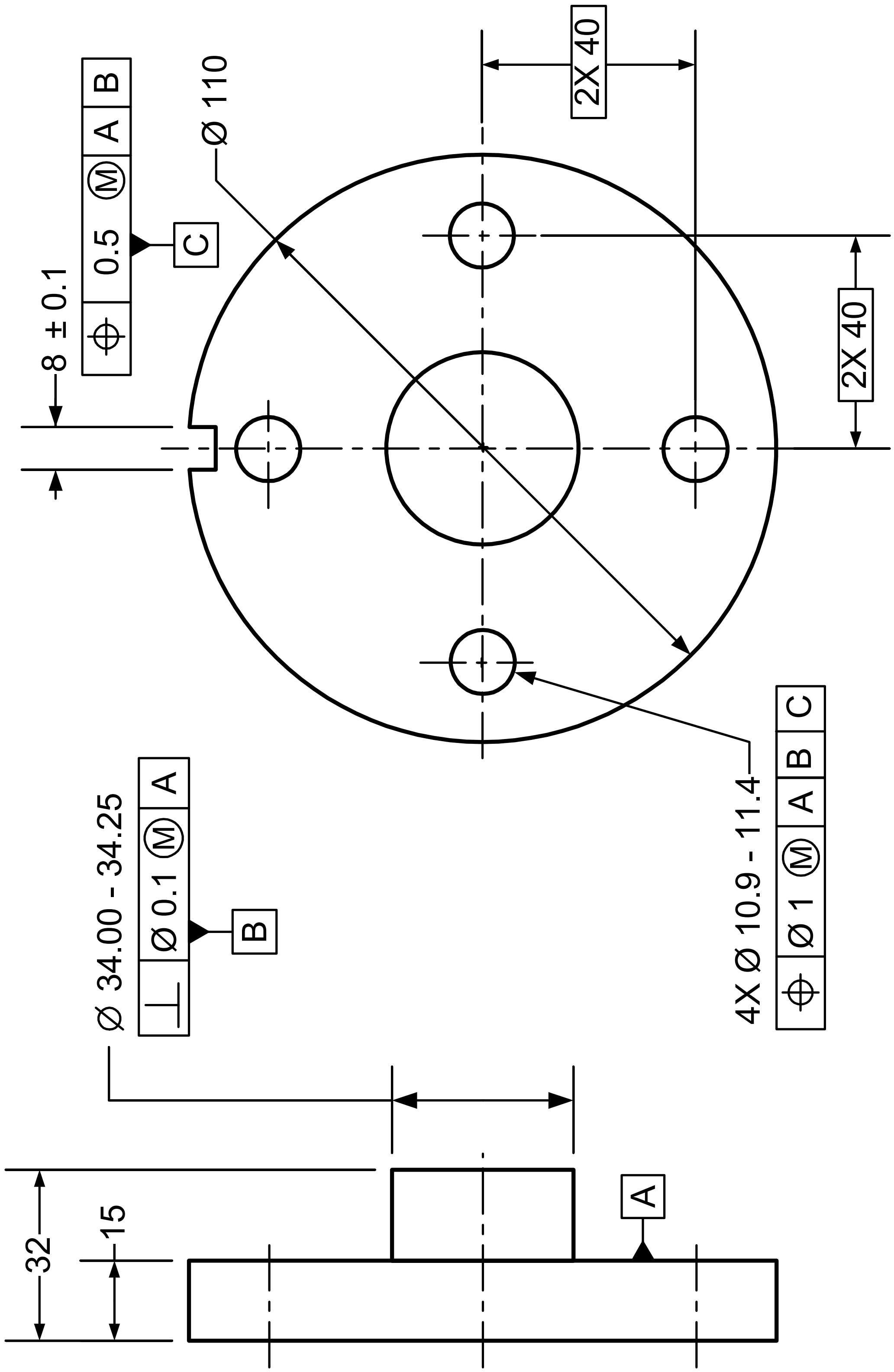


Produced Part



Hole #	Hole MMC	Hole Actual Size	Position Tolerance Allowed	"X" Distance	"Y" Distance	Position Location	Accept	Reject
1								
2								
3								

3.


2X 40
Ø 110
2X 40
8 ± 0.1
0.5 Ⓜ A B
C
Ø 34.00 - 34.25
Ø 0.1 Ⓜ A
B
4X Ø 10.9 - 11.4
Ø 1 Ⓜ A B C
32
15
A

Inspection Report for Hub Cover				
Item #	Actual Size	X Dimension	Y Dimension	Comments
B	34.09	+0.1	+0.1	
C	8.05	0.59	---	
1	11.25	+0.3	39.72	
2	11.3	40.25	+0.55	
3	11.04	-0.21	40.54	
4	11.28	40.48	+0.26	

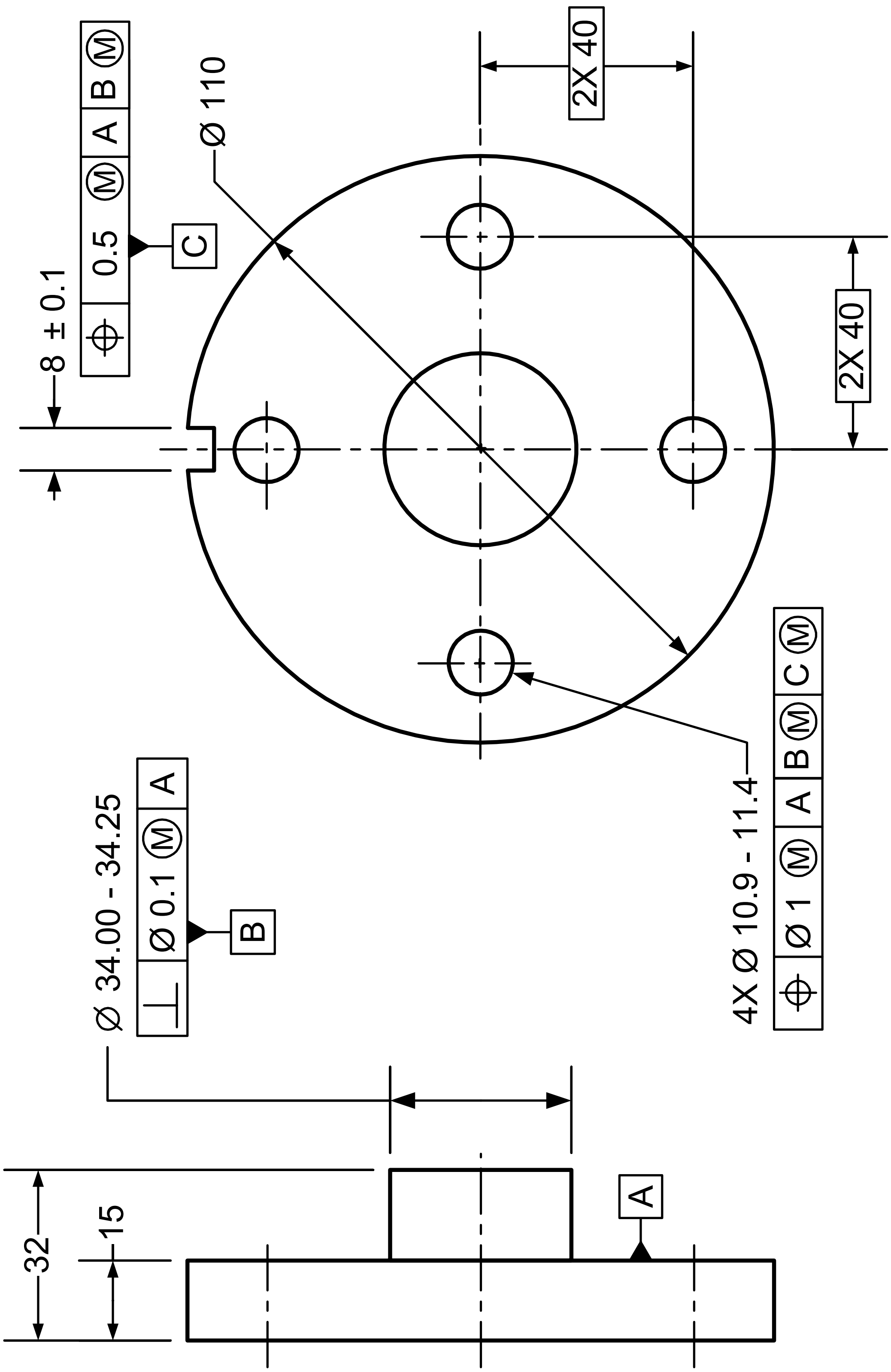
Check Sheet
Layout Inspection

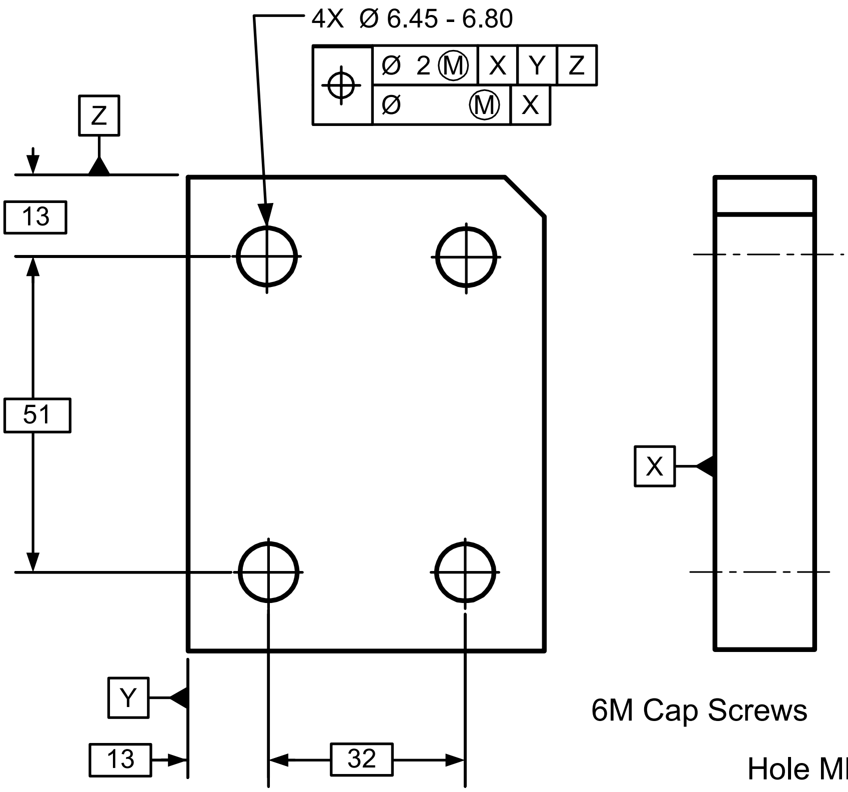
Part No. ______________ Part Name ______________ Operation No. ________ Rev. _____

Requested by ______________ Lot No. ______________ Serial No. ____________

Geometric Verification Check

#	Feature	Feature Size			Allowable	Actual Location			Accept	Reject	
		MMC Size	Actual Size	Size Δ	Zone	± X Δ	± Y Δ	Zone			

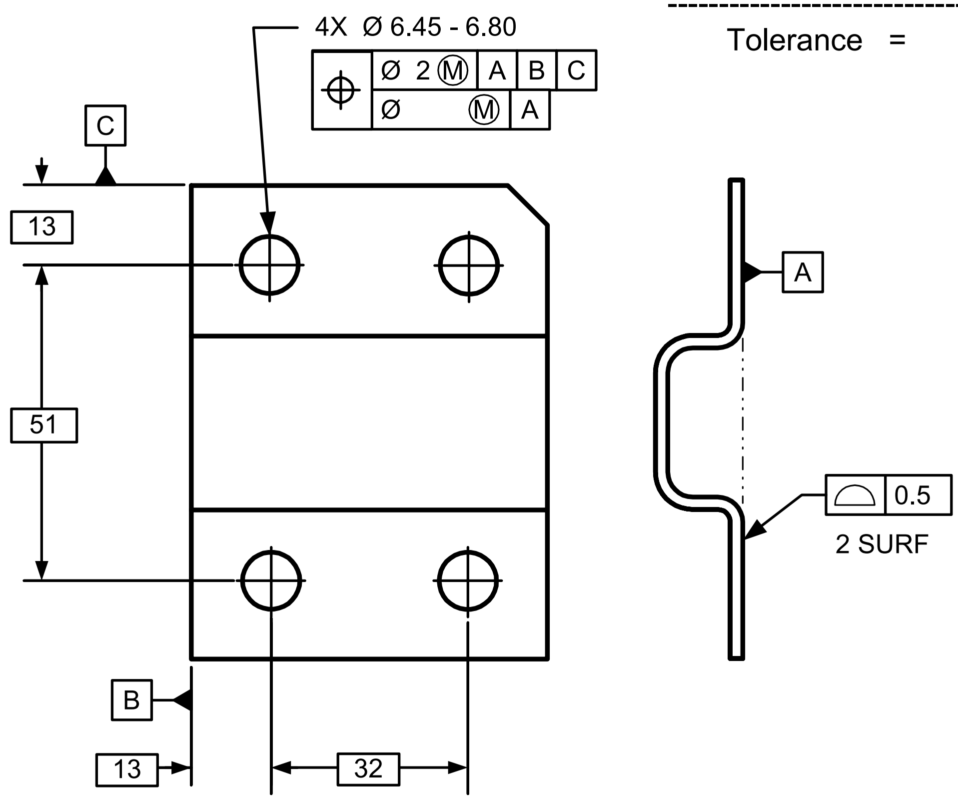

Ø 110
2X 40
2X 40
8 ± 0.1
⌖ 0.5 Ⓜ A B Ⓜ
C
4X Ø 10.9 - 11.4
⌖ Ø 1 Ⓜ A B Ⓜ C Ⓜ
Ø 34.00 - 34.25
⊥ Ø 0.1 Ⓜ A
B
A
32
15

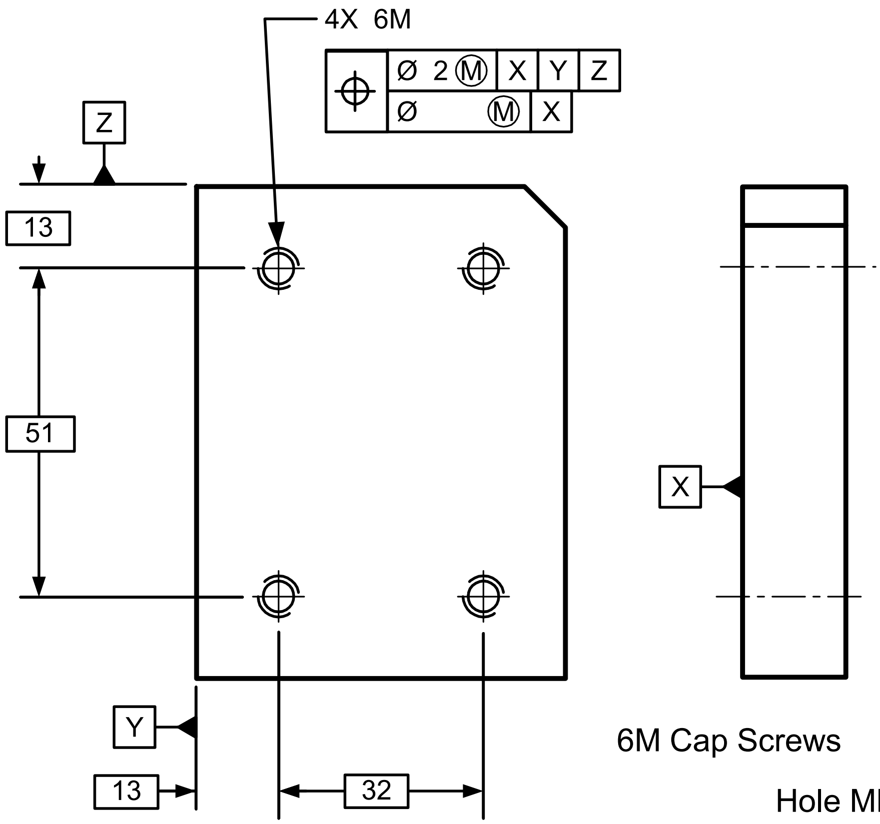


6M Cap Screws

Hole MMC =
- Screw MMC =

Tolerance =





6M Cap Screws

Hole MMC =
- Screw MMC =

Tolerance =