UNIVERSITY OF JOHANNESBURG

FACULTY OF ART, DESIGN AND ARCHITECTURE
DEPARTMENT OF INDUSTRIAL DESIGN

ENGINEERING DRAWING MANUAL
SUPPLEMENT TO ENGINEERING MEDIA 2
SUBJECT CODE: KDM21–1

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This manual supplements and interprets the SABS code of practice for engineering drawing 0111 part 1, but does not replace it.

Introduction

Engineering drawing is a visual language that enables designers of all disciplines to specify the exact shape, size, materials and finishes a product should conform to. Like any language, it only works because the people using it have a common understanding of the rules and the vocabulary of the language. The rules of engineering drawing are based on a well-reasoned system of representing shapes and their dimensions as fully as possible with the greatest degree of accuracy, the smallest possibility of error, and using the least amount of information.

Assessment criteria for engineering drawing

Your drawings will be assessed against the three primary criteria listed here. If you keep these criteria in mind when you create engineering drawings, you will be in a good position to evaluate the correctness of your drawings and to make corrections as you draw. This is the best way to learn, and to earn good grades.

Thoroughness: All the information in your drawings should give a complete picture of the size, shape, complexity and manufacturing requirements of the parts drawn. Thoroughness is a matter of providing only the information that is required. Too much information is as bad as too little information. When just the right amount of information is present, then the drawing is complete.

To be thorough you will need to edit and review your engineering drawings a number of times, weed out the incorrect information and add in missing information.

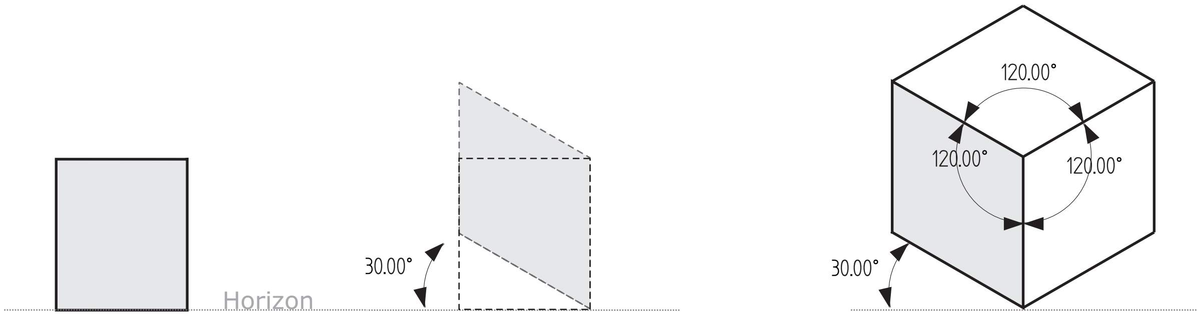
Clarity: Clarity is a matter of including all relevant information using the fewest views and the fewest dimensions with the least amount of clutter. Any information that is confusing, ambiguous, or redundant (a repeat of information given elsewhere) must be eliminated.

Difficulty factor: Your drawings must address the level of difficulty that suits the learning requirements of the project. Drawings that avoid addressing the problems set out in the brief cannot be assessed against the learning objectives, and will be penalised even if they are in other respects ‘correct’.

1 Axonometric and isometric projection

There are two methods of graphic (two-dimensional) projection used in engineering drawing, **axonometric** and **orthographic**.

The two methods are related. An orthographic projection is a flat view of an object seen perpendicular to that view. Figure 1 shows how the front view (orthographic projection) of a cube is skewed so that it can form part of a three-dimensional view of the cube



1. Orthographic front view of a cube

2. Front view is skewed to form a parallelogram

3. The skewed orthographic front view becomes the front of the axonometric (isometric) cube. The skewed top and right side views are joined to the front view to form an isometric cube.

Figure 1: orthographic to axonometric projection

1.1 Axonometric projection

Axonometric projection uses parallel lines to represent three-dimensional objects. This is different to perspective which uses lines that meet at a vanishing point. In axonometric projection the y axis is normally vertical and the x and z axes are at an angle to the horizontal. The three visible faces of an object are aligned along their respective x, y, and z axes. The most notable feature of such projection is that because parallel lines remain parallel to one another, rectangular shapes are represented using parallelograms. **Isometric projection** is a particular format of axonometric projection in which the X and Z axes are always at 30°to the horizontal. The word *isometric* comes from the Greek for *equal measure*, so the three axes are equally divided into 120° segments.

An axonometric projection is not the same as a perspective view, since no foreshortening takes place, the vertical lines remain vertical and the horizontals are always represented at an acute (less than 90°) angle to the horizon. Some of the three-dimensional shapes in this manual are depicted using isometric projection, and some use axonometric projections that are not isometric. For your own engineering drawings you may use any axonometric projection, including isometric projection, to help explain your orthographic views, but you may not use a perspective view. This is to ensure a measure of uniformity in your drawings. Exaggerated perspective can lead to misleading representations of three-dimensional shapes.

Axonometric/isometric projections are useful for explaining the three dimensional shape being depicted in orthographic views, and for exploded views, but they are not very suitable for dimensioning. An axonometric projection can sometimes be used for dimensioning very basic shapes, but it is nevertheless a cumbersome method because although **true sizes** of some of the geometry can be dimensioned, **true shapes** cannot. For instance, in the above depiction of a cube, the height of each side could be accurately dimensioned, but since each side is represented by a diamond neither the width nor the squareness of the sides can be depicted accurately. It is especially cumbersome to use axonometric projections to describe lines which are not either straight or perpendicular to each other, such as circles, arcs, and diagonal lines.

1.2 Orthographic projection

1.2.1 Motion parallax

Our eyes allow us to see the world in three dimensions, or 'in perspective'. This kind of seeing is called binocular vision. Without it we would have no sense of depth. A drawback of binocular vision is that we are unable to observe two different points of an object at the same time. We cannot truly see flat shapes. One point on a shape will always appear closer than the other, depending on where we are relative to the two points.

This phenomenon is called motion parallax. You might know it as the parallax error. You will have seen how this works when you mark off a measurement using a ruler. You need to move your head directly above each measurement you wish to mark off to avoid making inaccurate measurements or marks. By moving your head directly above the point on the ruler, you are placing your eyes perpendicular (at a 90˚ angle) to that point. Orthographic projection is just a way of putting an observer at 90˚, or perpendicular, to prescribed views of an object.

An orthographic view works by artificially negating motion parallax. The view of the object that the observer (you or I) sees is 'flattened' so that the observer is always perpendicular to all points being observed. It is now possible to see and measure both the 'true' shape and the 'true' dimensions of the shape.

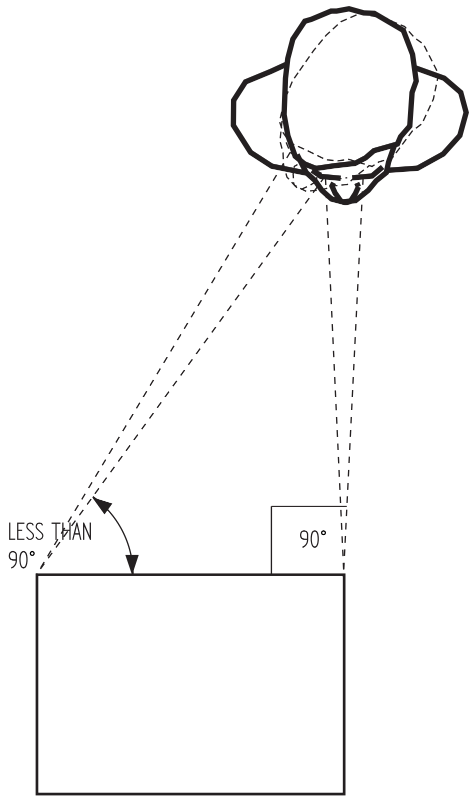


Figure 2. Motion parallax: the human inability to observe two points on an object at the same time. The dotted lines show that a person standing still would need to turn his head to see the furthest visible point of this block. This point is further away than the point which is he is perpendicular to.

1.2.2 Cartesian co-ordinates and Cartesian planes

Orthographic projection is based on Cartesian co-ordinates. Cartesian co-ordinates and Cartesian planes are named after the french philosopher and mathematician René Descartes.

A Cartesian plane is simply a flat surface that is partitioned into a grid of horizontal and vertical lines. The lines are numbered. The lines at zero are called the axes. The vertical axis is the y axis (also called the ordinate) and the horizontal axis is the x axis (also called the abscissa). Lines to the right of the y axis and those above the x axis are numbered positively, and lines to the left of the y-axis and those below the x-axis are numbered negatively.

A line originating from the x axis is called an x co-ordinate, and one from the y axis a y co-ordinate. A point on a Cartesian plane must be described using two intersecting lines, one from either axis. Their intersection is called a set of co-ordinates. Any geometric shape can be described mathematically, using numbers (dimensions), by mapping the co-ordinates of the significant points of that shape. It is very important to note, as can be seen in figure 3, that the positions are mapped by always starting at the zero point, or origin, rather than adding one position onto the end of another. This origin is called a datum in engineering drawing. Datums are indispensable for accurate dimensioning because, if used correctly, they limit the number and severity of errors that can be made. This important idea is the basis for all dimensioning rules used in engineering drawing.

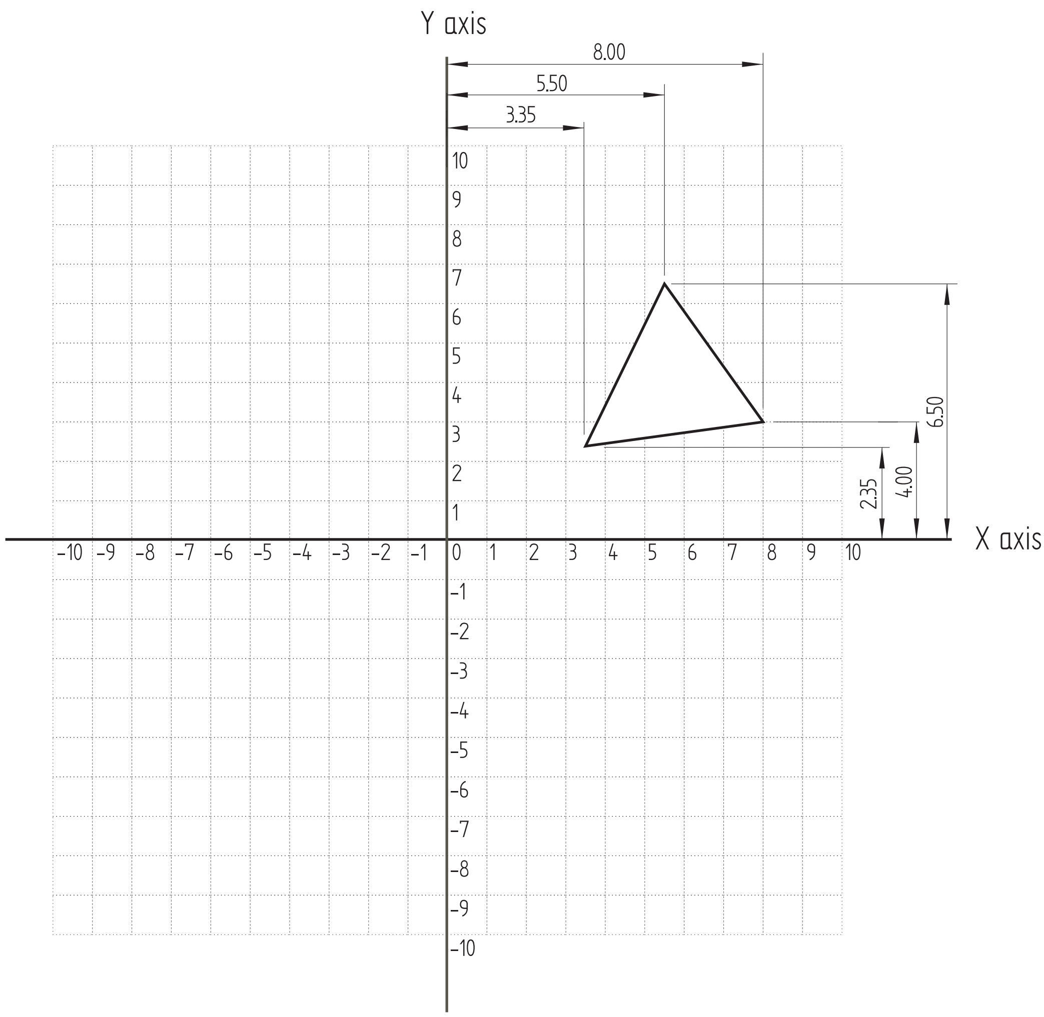
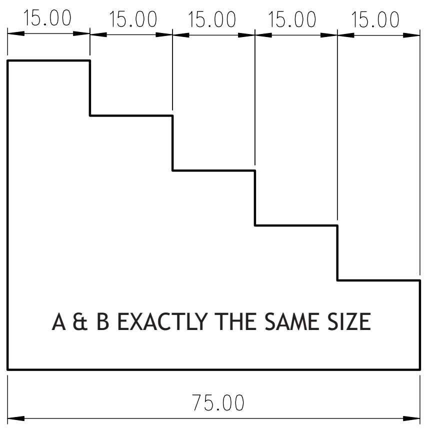
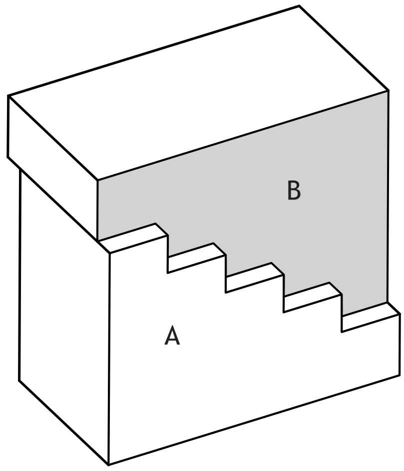


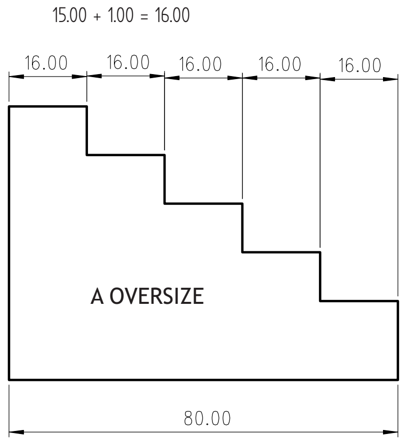
Figure 3. **A Cartesian plane**. Position, position, position. The co-ordinates (positions) of the vertices (meeting points) of a shape describe the dimensions of the shape, and at the same time these co-ordinates show the position of the shape relative to a datum (origin, or zero point).

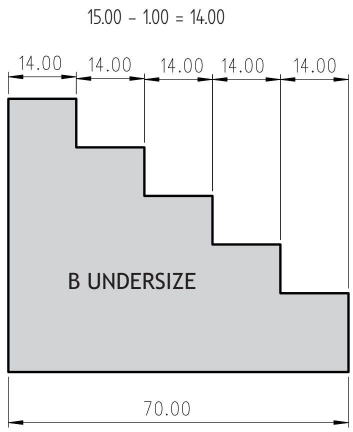
Ideally, two parts made to the same dimensions should fit together perfectly..

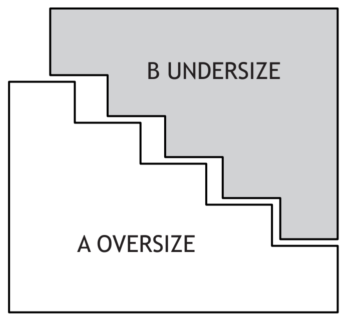


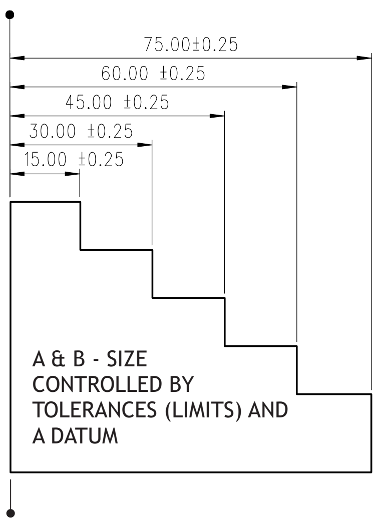


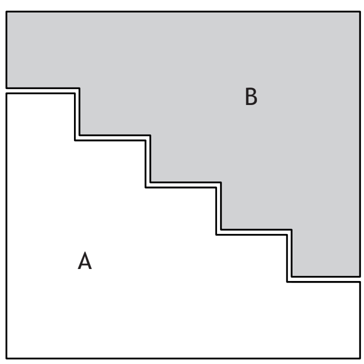
In reality slight errors in manufacturing mean the two parts will be slightly different. In the illustration shown here, an allowance is made for a 1.00 mm error for each dimension. As you can see, if, for part A, each dimension grew by 1.00 mm, and for part B each dimension shrunk by 1.00 mm, the parts would be grossly mismatched.











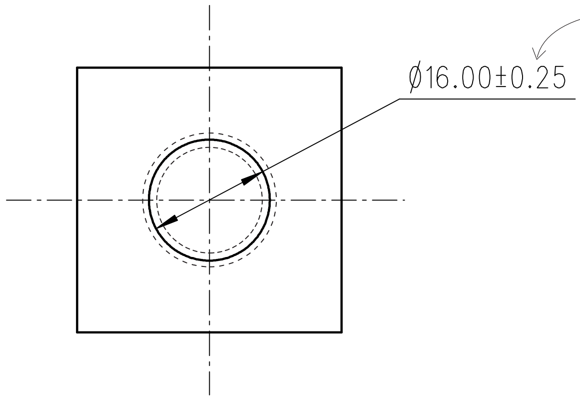
The accumulation of errors seen in the above example can be drastically reduced by starting all dimensions from the same point, rather than starting each new dimension from where the old one ends. This 'same point' is called a datum. The 'allowance for error' is called a tolerance. A tolerance 'limits' the amount of error allowed.

In the example here part A is oversize to the maximum and part B undersize to the maximum. They fit together neatly with a very small gap between mating faces. This gap is also called a tolerance.

Almost all modern manufacturing uses these principles for making the products that you will be designing.

Tolerances can be one of two kinds, *size* tolerances (often referred to as *limits of size*), and *fit* tolerances.

Size tolerances specify how much larger or smaller a dimension can be. Size tolerances apply to both position and size dimensions.



This size tolerance specifies that the hole may be 0.25 mm smaller in diameter, or 0.25 mm larger in diameter. It can also be read as:

Ø16.25 (= upper limit of size)
O15.75 (= lower limit of size)

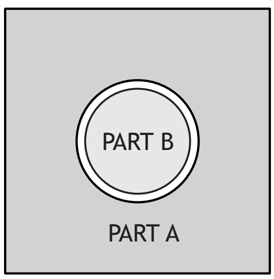
Fit tolerances specify how close the fit between mating parts can be. There are two types:

1. Clearance fit

   If the gap is positive, that is the smaller part fits into the larger part with some play between them, then it is called a clearance fit.

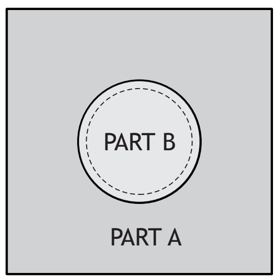
2. Interference fit
   If the gap is negative, the 'smaller' part will need to be forced into the 'larger' one and the two parts will not be able to move relative to one another without further force being applied. This is called an interference fit

CLEARANCE FIT



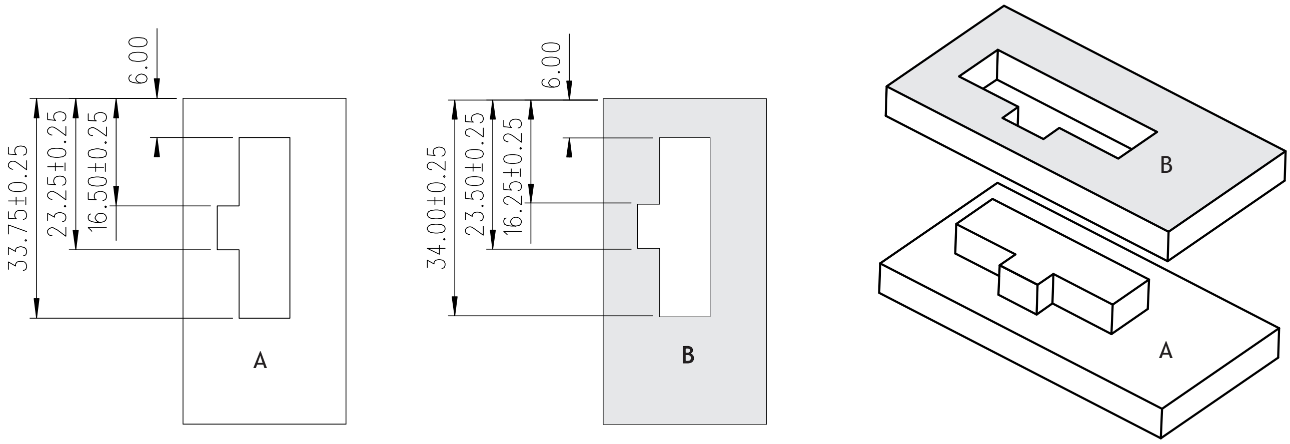
Part B is smaller than part A. The gap between the parts is the clearance.

INTERFERENCE FIT

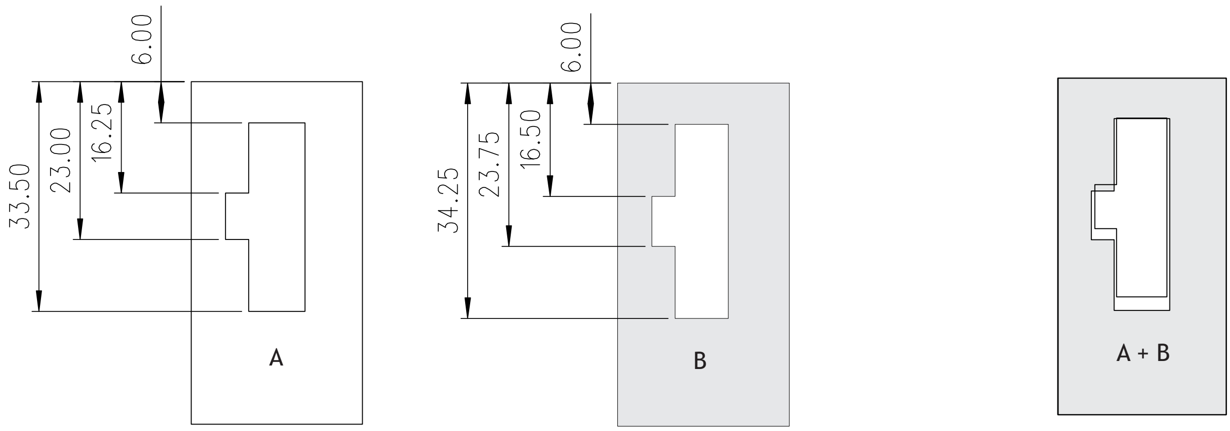


Part B is larger than part A. There is no gap between the parts. Instead, they interfere with one another and so have an interference fit.

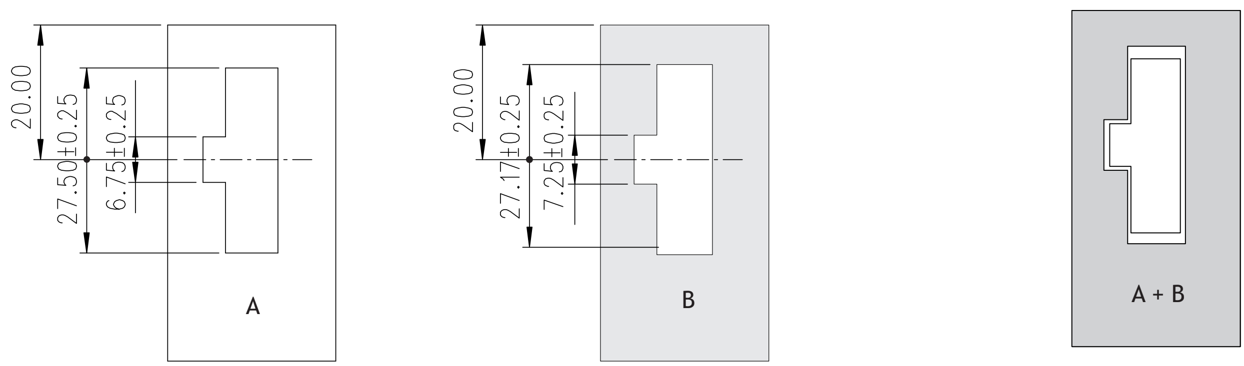
SECONDARY DATUMS



Parts A and B must fit together with some clearance between them. For the sake of illustration, assume the first dimension to be fixed at 6.00 mm, and the clearance to be a nominal 0.25 mm. The allowable tolerance for all the other dimensions shown is ± 0.25 mm.

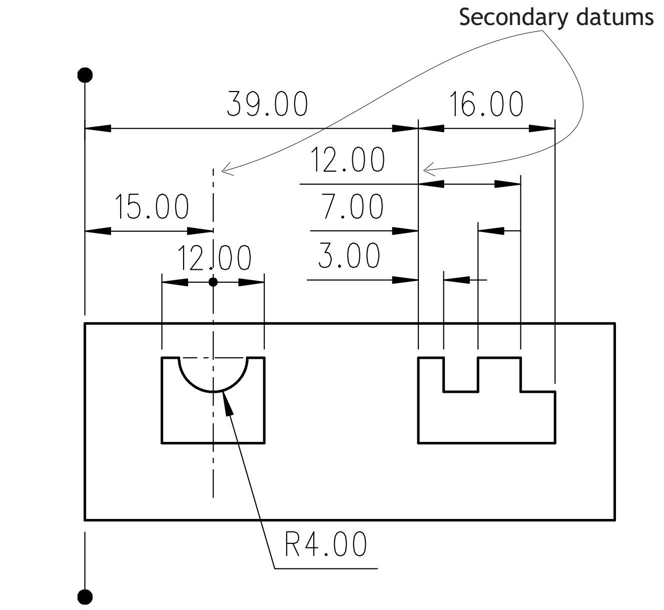


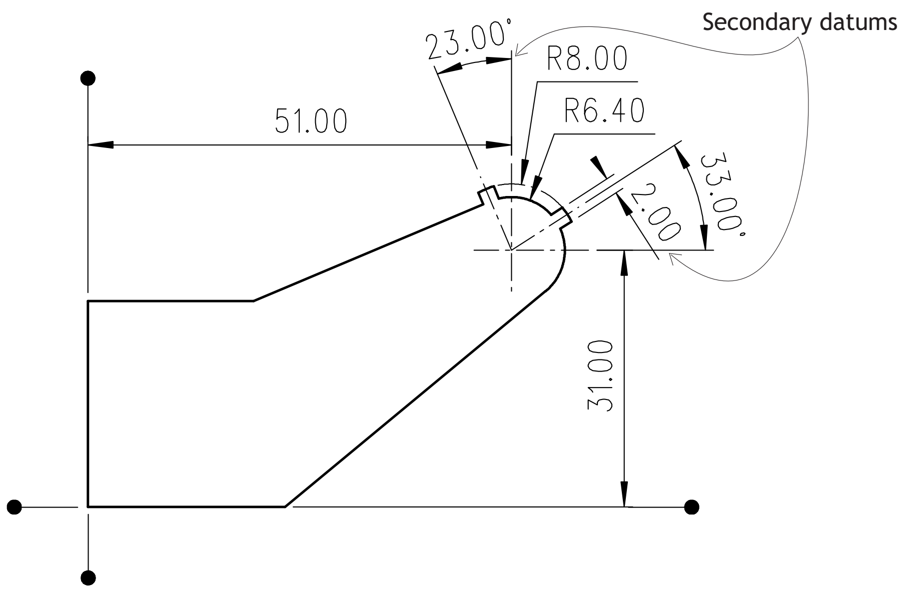
If the dimensions on part A shrink by 0.25 mm, and those on part B grow by 0.25 mm, then the parts will not fit together as they should. This is in spite of following the rule that dimensions should start from a common datum.



However, if on both parts, a secondary datum is established on the centre line that divides the keyed rectangle, then it becomes possible to control and reduce the amount of potential mismatch. Of course in reality this secondary datum will also need to have a tolerance attached to it. Here it is left out for the sake of illustration.

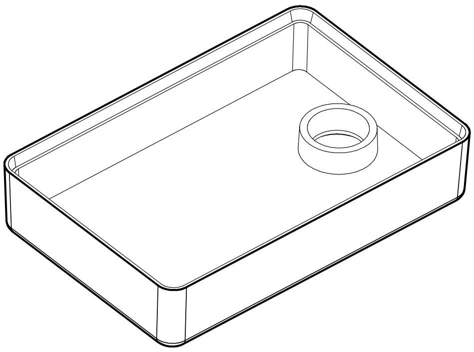
A second advantage of this way of dimensioning is that it simplifies the dimensions and makes them easier to read. Further, because there are fewer dimensions the tolerance on the dimension describing the secondary datum can be a lot smaller, so eliminating any mismatch of positions when assembled.


Secondary datums
39.00
16.00
12.00
15.00
7.00
12.00
3.00
R4.00


Secondary datums
23.00°
R8.00
51.00
R6.40
33.00°
2.00
31.00

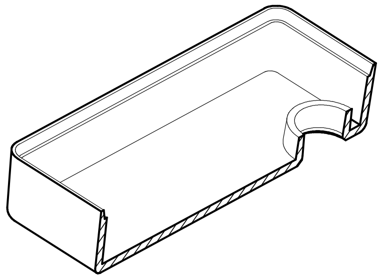
A

Here is an isometric view of an injection moulded part.



B

Here, the part is sectioned in half so you can get an idea of what the shapes of the die set will look like if they were also sectioned.

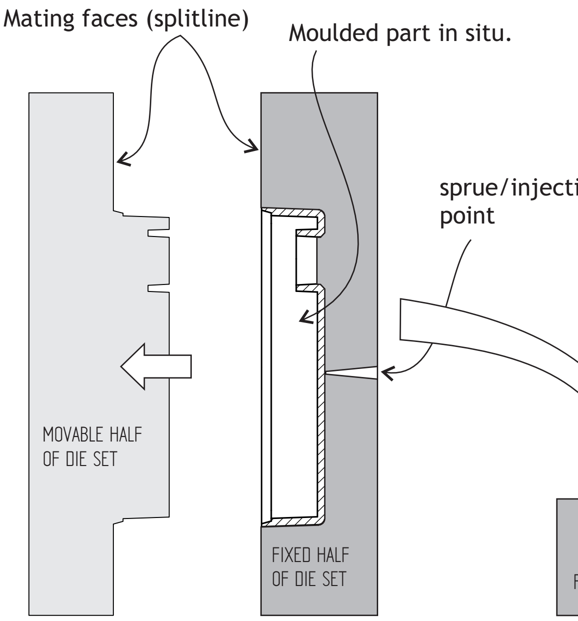


C

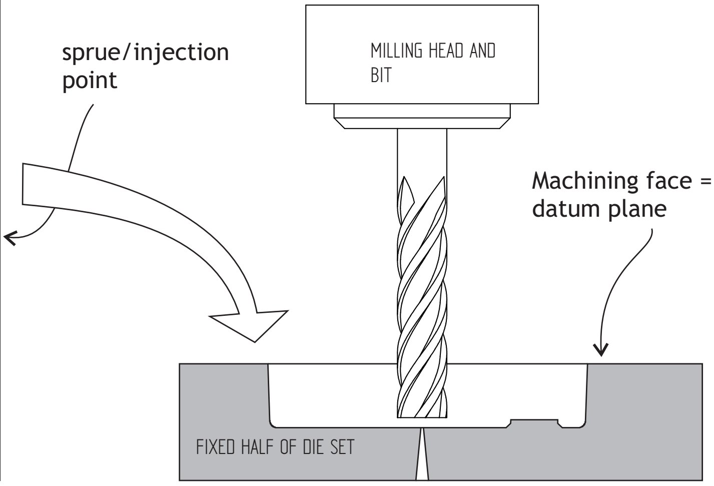
The die set would be orientated vertically in the injection moulding machine.......

D

.....but when each part of the set is being cut, it must be orientated horizontally. So, it is the machining face that must be chosen as the datum plane in your drawing. This is because the machining face is the only place a machinist can start cutting from. It is also the face of the die set that mates with the corresponding face on the other half of the die set. This face will be seen as the split line on the final moulded product.



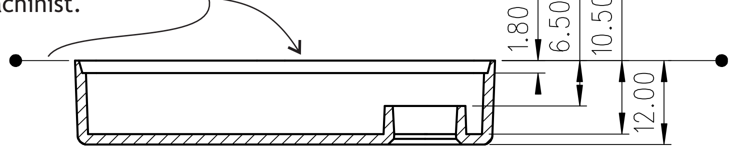
SECTION VIEW OF DIE SET IN OPEN POSITION



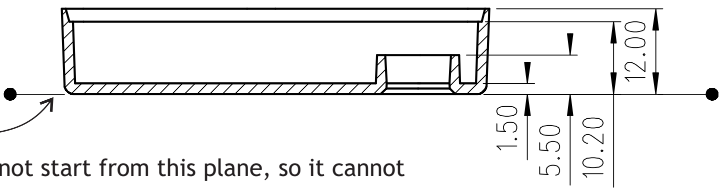
E

Orthographic view and datum arranged horizontally to make dimensions easiest to read for the machinist.

CORRECT



INCORRECT



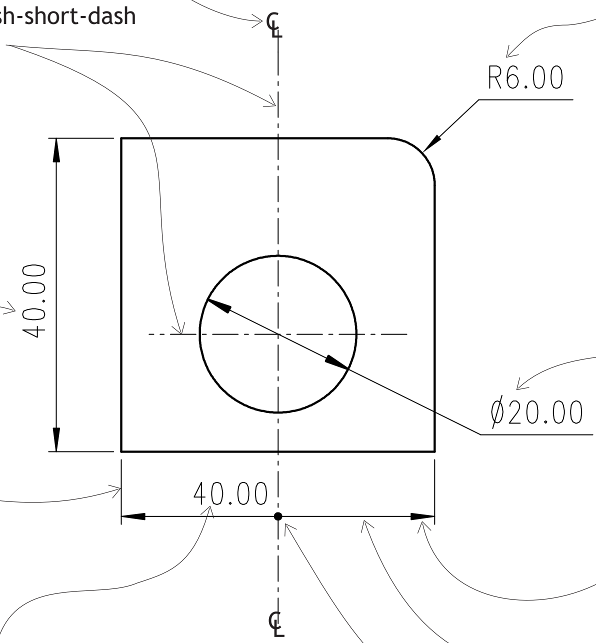
A machinist cannot start from this plane, so it cannot be a datum.

THE PARTS OF DIMENSIONS

The ℄ sign marks an axis of symmetry. An axis of symmetry is a centre line indicating that the outline of a view is essentially symmetrical. It simplifies dimensioning and makes it more precise.

A radial dimension (for arcs/ incomplete circles) is always preceded by the R sign and is always placed horizontally. Its dimension line/leader line should be a diagonal.

Centre line - a thin long-dash-short-dash line that must run across the shape that it divides.



A diametric dimension (for full circles) is always preceded by the Ø sign, and is always placed horizontally. Its dimension line/leader line should be a diagonal.

Vertical dimensions must be readable from the right

Extension line

Arrows should be filled (black), 3.00 mm high and 1.00 mm wide

Dimension line

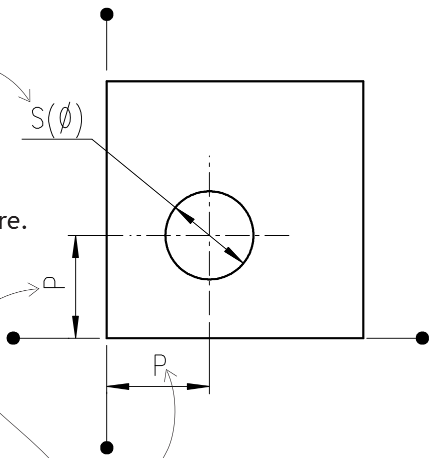
Dimension value. Always place the numbers 'above' and parallel with the dimension line. Always place two decimals (numbers) behind the point, and a zero before the point if the value is less than 1.

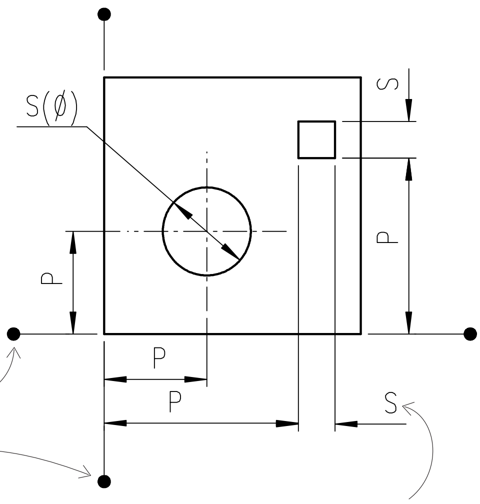
Centre dot - use when either a centre line or an axis of symmetry divides a dimension into two equal parts.

SIZE AND POSITION DIMENSIONS

Any feature or shape requires at least three dimensions: two ***position*** dimensions (an X and a Y dimension, otherwise known as a set of co-ordinates) and a ***size*** dimension. The position dimensions must always start from datums.

A size dimension shows how large the feature is. Circles and arcs must be marked by symbols that show what their shapes are.





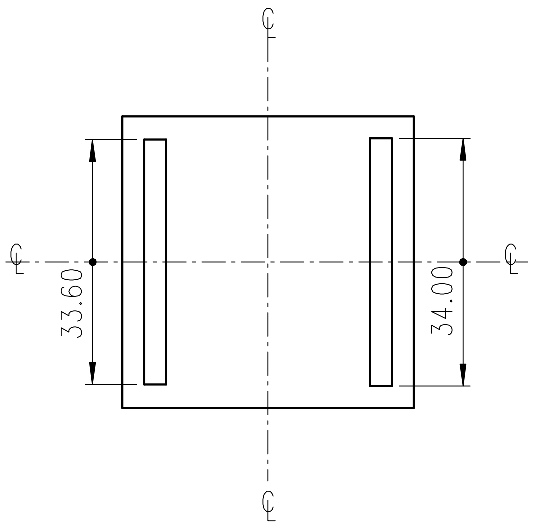
Position dimensions show *where* to place the feature, as measured from a datum.

Datum planes must be marked by a dot of Ø1.50 mm on the end of an extension line, placed outside all dimensions.

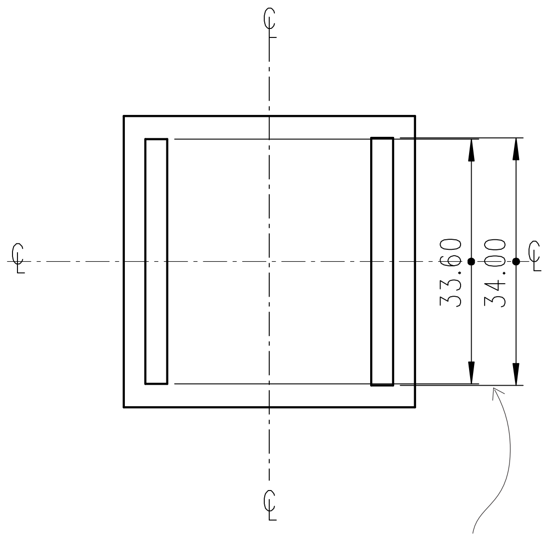
Size dimensions that are not arcs or circles are not usually marked as to their shape.

RULE 1. Dimensions must take the shortest route and cross as few lines as possible.

CORRECT



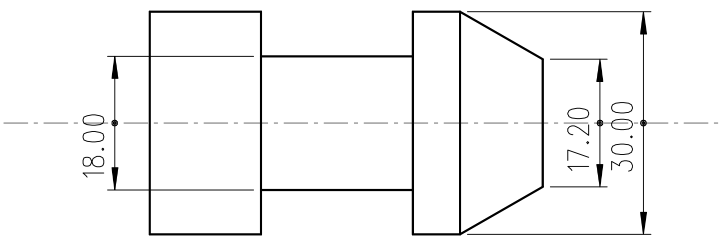
INCORRECT



This dimension line creates confusion by crossing more lines than it needs to, and by being too close to other lines. Dimensions become cluttered and lack clarity.

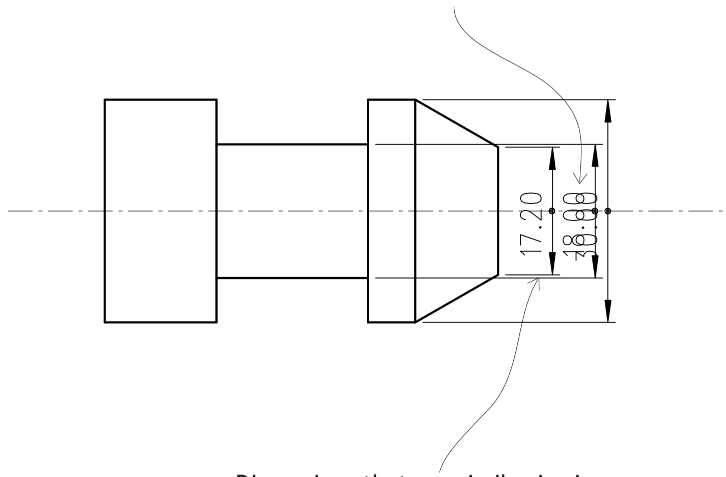
RULE 2. Dimensions should not overlap.
Dimensions of similar size should not be placed near one another.

CORRECT



INCORRECT

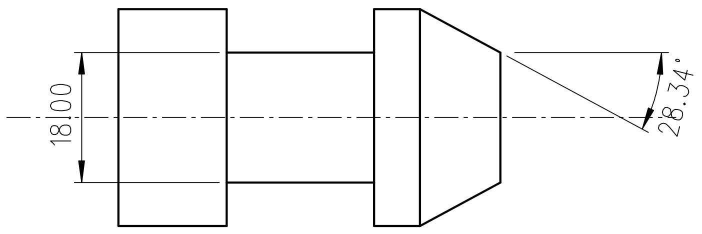
Dimensions that overlap are confusing or unreadable.



Dimensions that are similar in size are difficult, and often impossible, to trace.

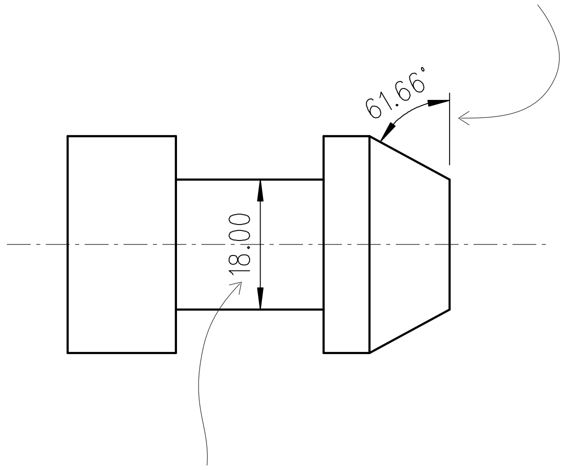
RULE 3. Dimensions should use extension lines wherever possible.

CORRECT



INCORRECT

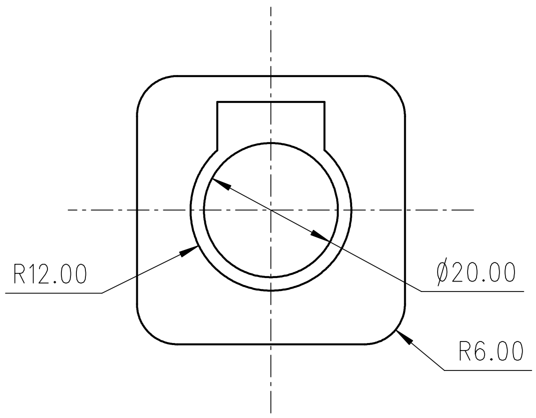
Angled dimensions should have two extension lines



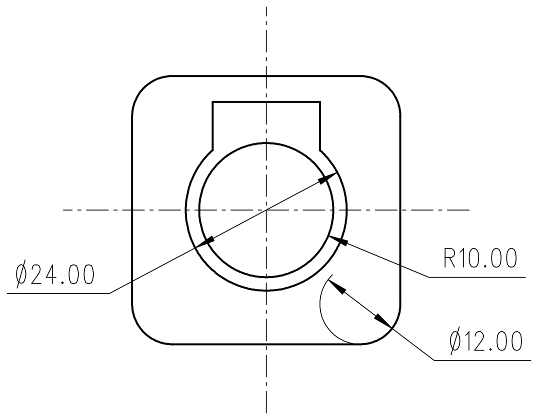
A dimension should not be inside a shape unless it is unavoidable

RULE 4. Diametric and radial dimensions are not interchangeable

CORRECT



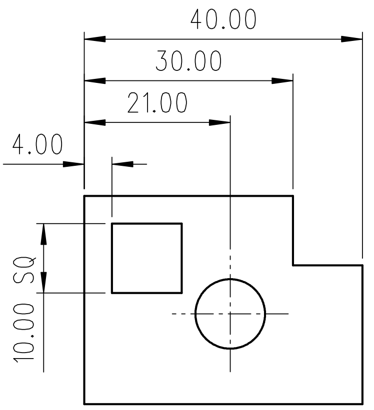
INCORRECT



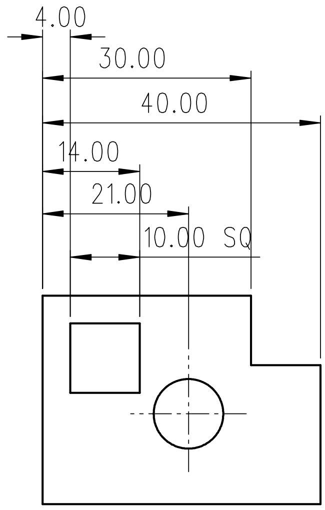
These dimensions give false information about the geometry of the features.

RULE 5. Dimensions should be arranged from smallest (closest to the outline) to largest (furthest from the outline).
Dimensions should be arranged so that they are as clear as possible.

CORRECT

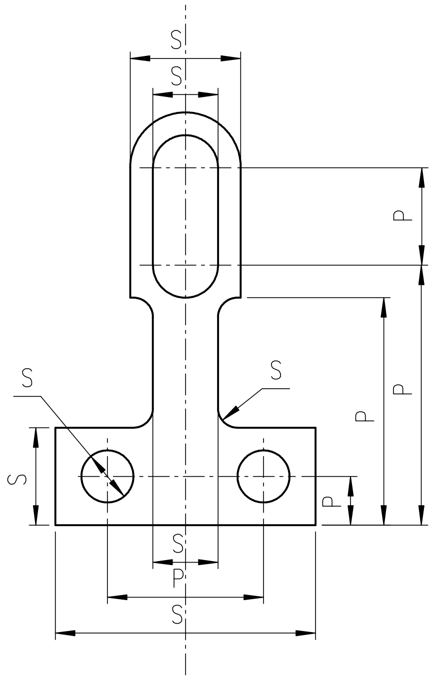


INCORRECT

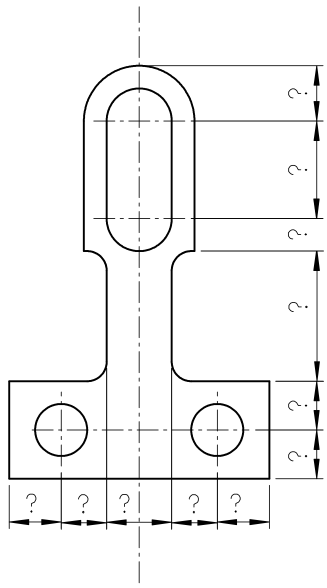


RULE 6. Separate the size dimensions from the position dimensions as far as possible.

CORRECT

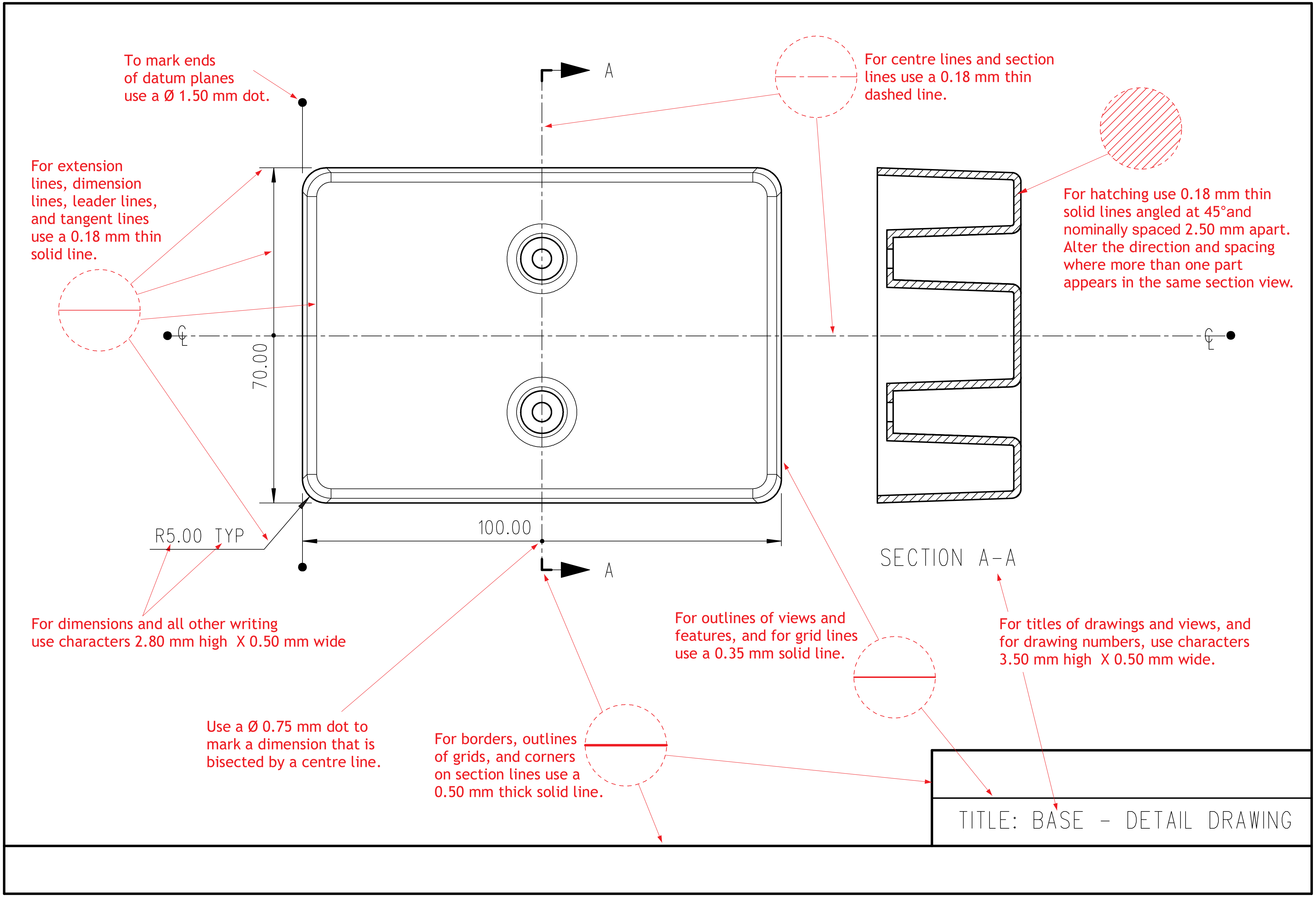


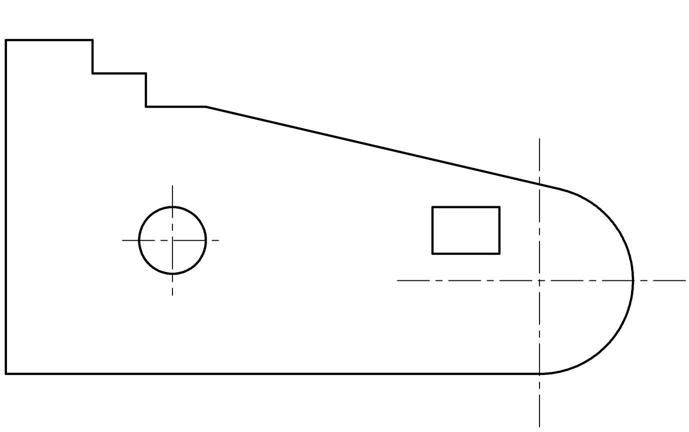
INCORRECT



(after S.A.B.S. 0111-1:42)

STYLE GUIDE

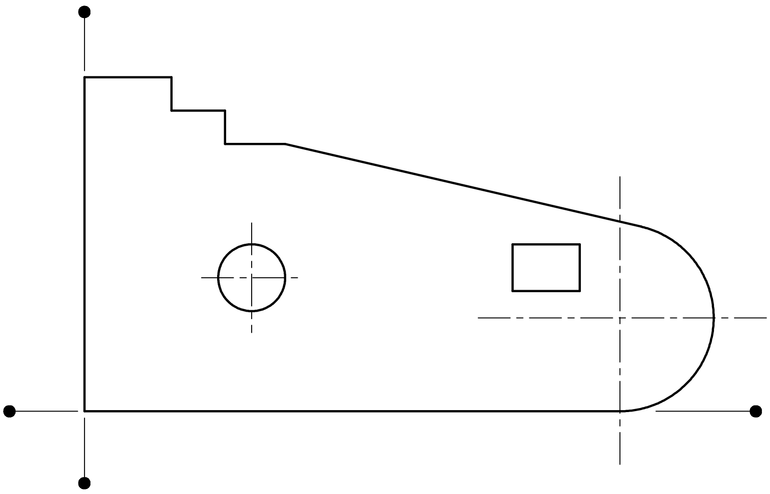

To mark ends of datum planes use a Ø 1.50 mm dot.
For centre lines and section lines use a 0.18 mm thin dashed line.
For extension lines, dimension lines, leader lines, and tangent lines use a 0.18 mm thin solid line.
For hatching use 0.18 mm thin solid lines angled at 45°and nominally spaced 2.50 mm apart. Alter the direction and spacing where more than one part appears in the same section view.
A
A
70.00
100.00
R5.00 TYP
SECTION A–A
For dimensions and all other writing use characters 2.80 mm high X 0.50 mm wide
For outlines of views and features, and for grid lines use a 0.35 mm solid line.
For titles of drawings and views, and for drawing numbers, use characters 3.50 mm high X 0.50 mm wide.
Use a Ø 0.75 mm dot to mark a dimension that is bisected by a centre line.
For borders, outlines of grids, and corners on section lines use a 0.50 mm thick solid line.
TITLE: BASE – DETAIL DRAWING



STEP 1

Place centre lines.

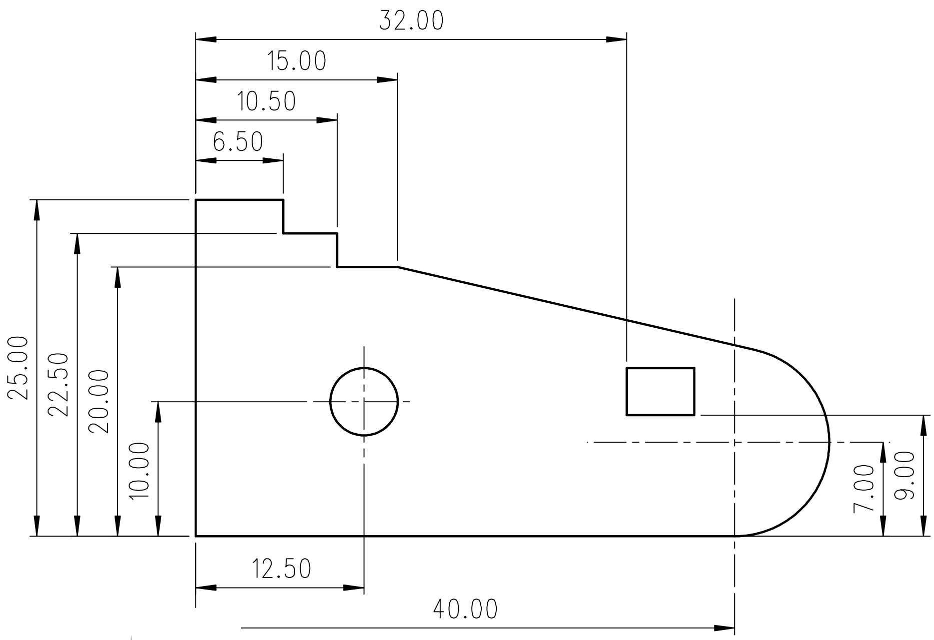
Note - centre lines must always cross over the outlines of the features they bisect.



STEP 2

Identify datums.

Note - at this point don't draw datums, just identify the correct ones to start dimensioning from



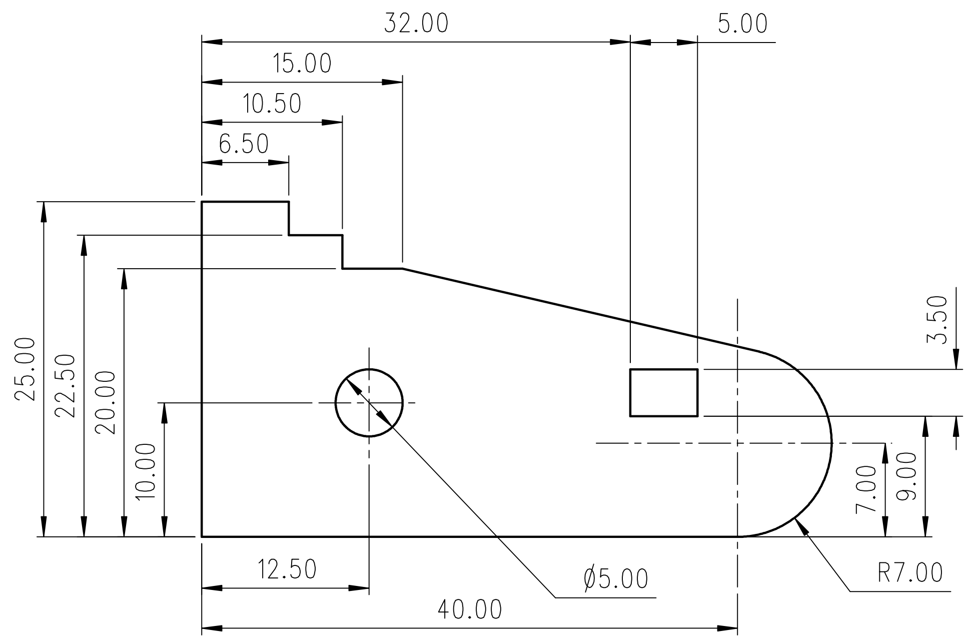
STEP 3

Place position dimensions

Note - each position dimension must start from a datum.

Place dimensions from smallest to largest

Place dimensions so that the extension lines are as short as possible, and so that they cross as few lines as possible.



STEP 4

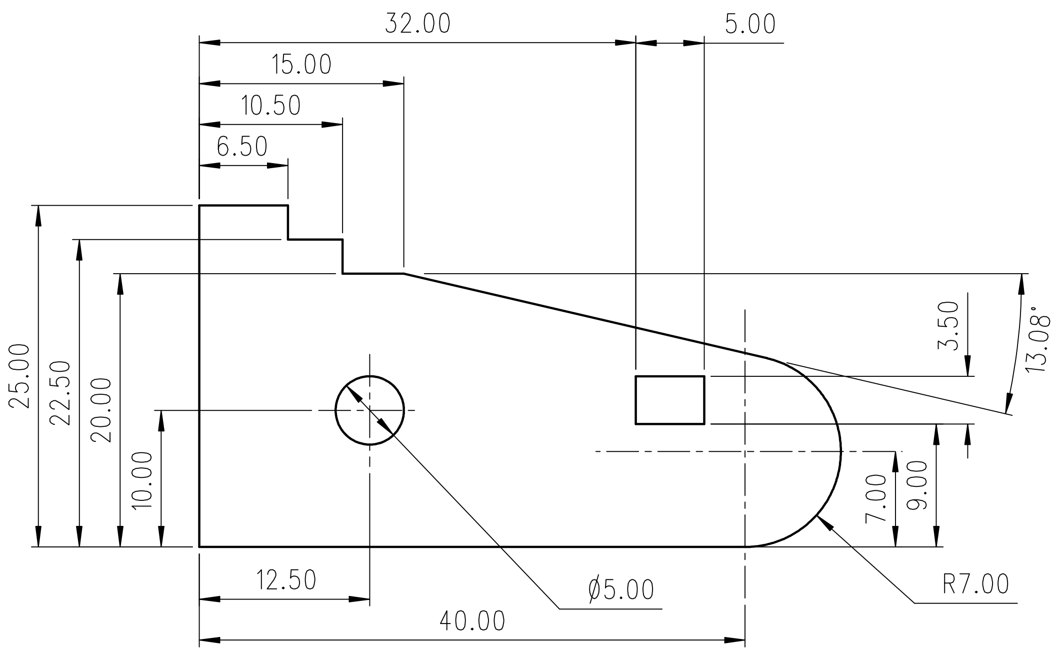
Place size dimensions.

Note - make sure position dimensions are in place before placing size dimensions.

A size dimension must be associated with its position dimensions.

Make sure extension lines cross over as few other lines as possible.

Radial dimensions must be placed diagonally and their arrows must not be near the meeting point of two lines

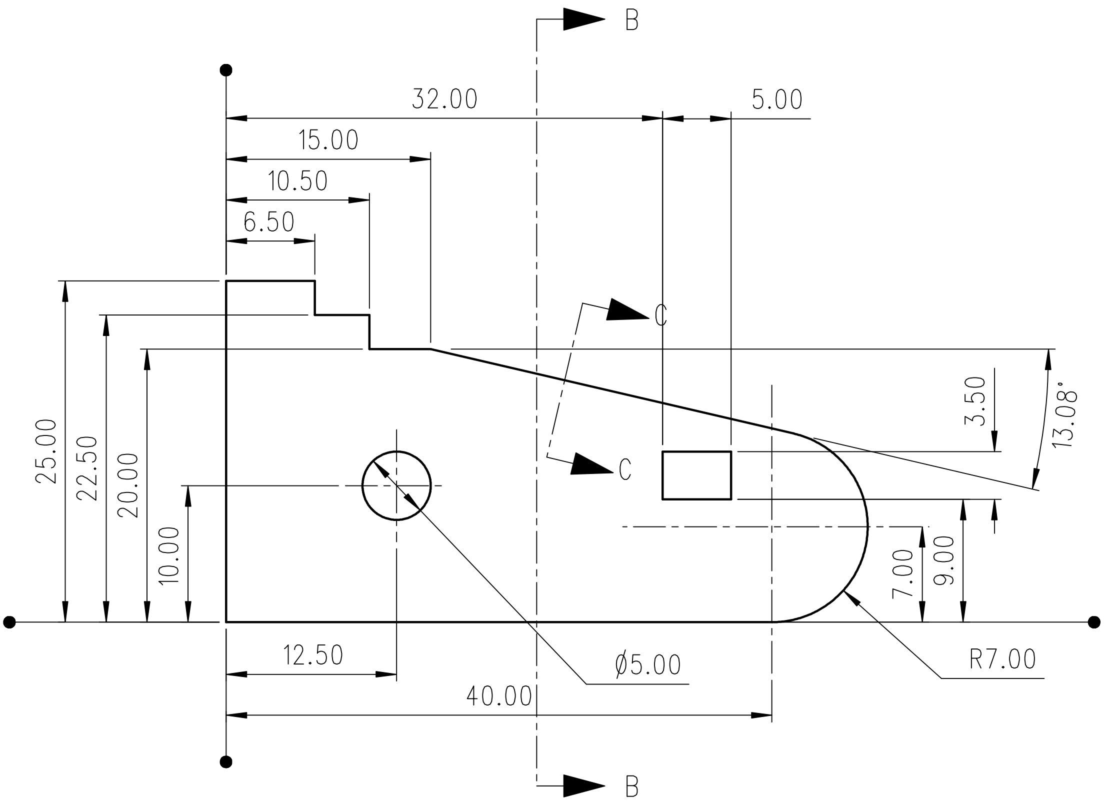


STEP 5

Place angle dimensions. An angle dimension is also a size dimension

Note - an angle dimension must start from a horizontal or a vertical plane.

It is usually best to place angle dimensions outside other dimensions. Although the angled extension line might cross over many lines, this will not affect clarity too much, and this way is usually clearer than squeezing the dimension inside other dimensions.



STEP 6

Mark the ends of datums with a Ø1.50 mm dot at the end of a short extension line.

STEP 7

Place section lines

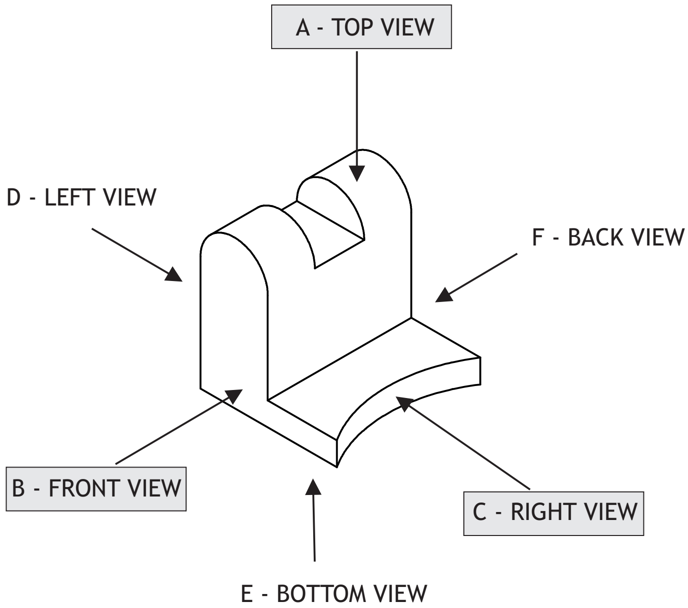
Note - section lines for full sections should be placed outside of all other dimensions.

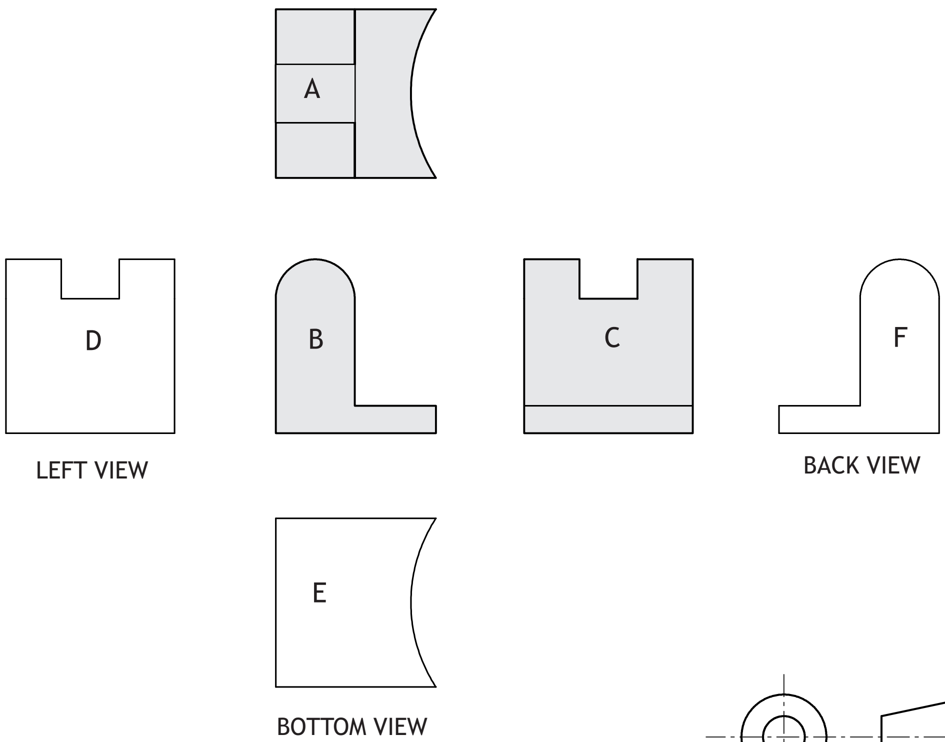
Section lines for partial sections should have their ends placed close to the features they cut.

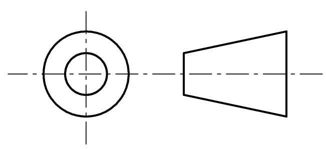
Section lines must be perpendicular to the features they cut, otherwise you will not get a section view with true shapes or sizes.

You will be using the third angle method of projection for this subject. This system consists of three primary views and three secondary views. The primary views are A, B, and C. The secondary views are D, E, and F.

For most drawings you create, you will need at least two of the primary views. Primary views are never labelled, whereas secondary views usually are labelled. This is because the third angle symbol that appears in the borders shows that the primary views will be arranged according to this system.





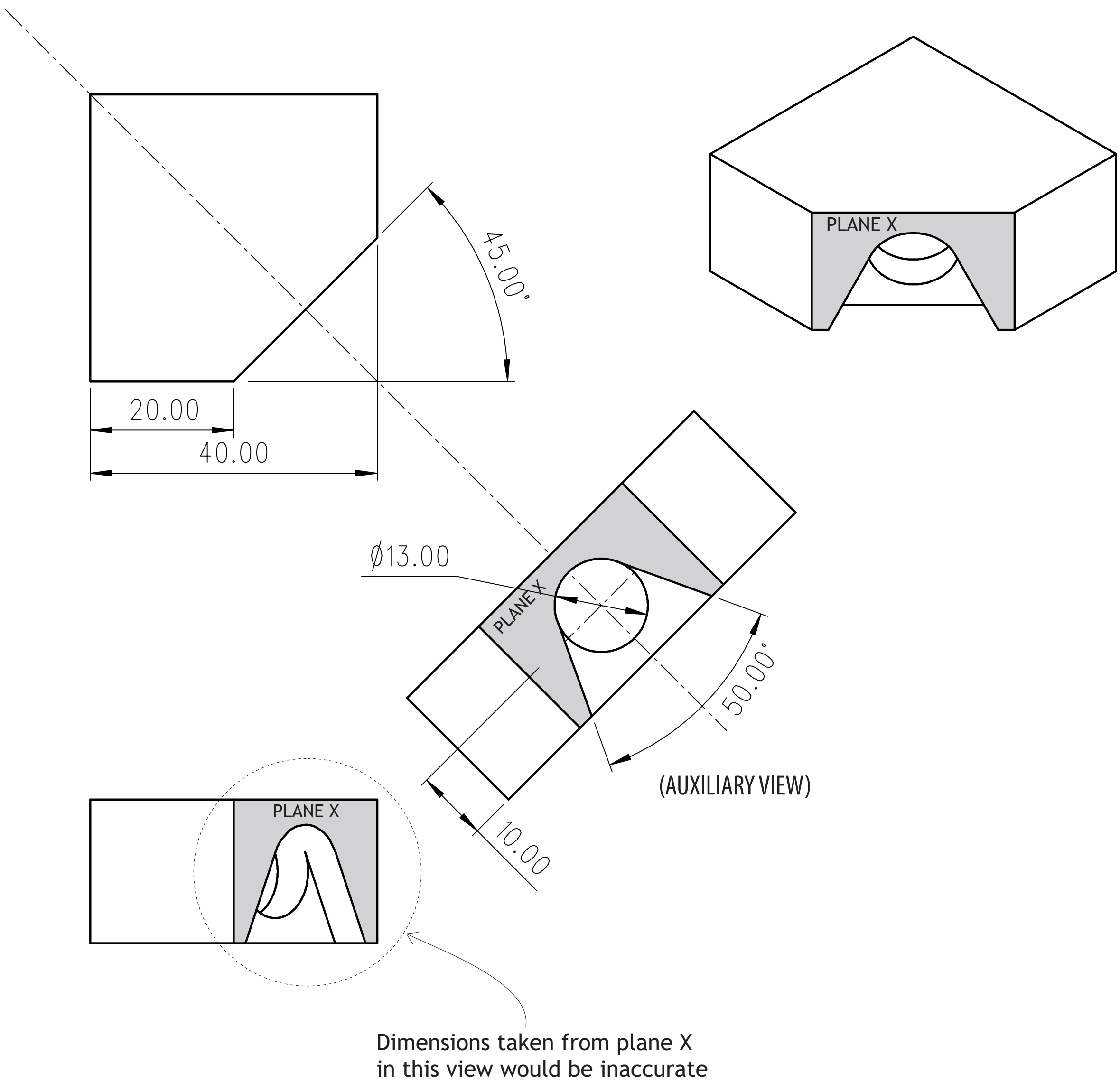


(3RD ANGLE SYMBOL)

The only dimensions that are ‘true’, meaning ‘accurate’, are those measured from true views.

For a view to be true it must be one that is perpendicular to a plane.

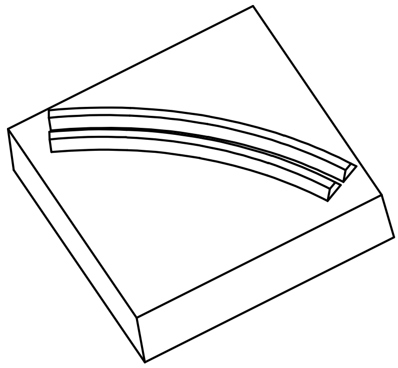
The plane X and the planes parallel to it, in the drawing below, are only true in the auxiliary view. An auxiliary view is any view that requires a viewpoint that is not one of the six standard views. It is always a view that is perpendicular to a plane, and this usually means it is orientated at a diagonal.



SECTIONING - PERPENDICULARITY

Section planes must always be placed so that they are perpendicular to the features they bisect. A section view is a form of auxiliary view. It must follow the same rules about showing true views and true dimensions.

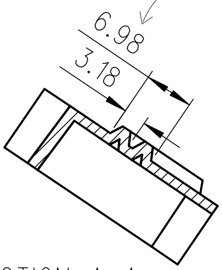
The slab below has two concentric semi-circular rails. The only way to provide accurate dimensions for the profiles for these rails is to section them. The correct and incorrect methods are shown at bottom.



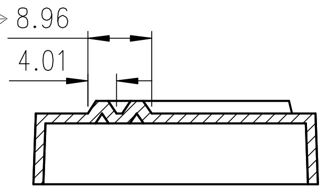
CORRECT

Note the difference in values

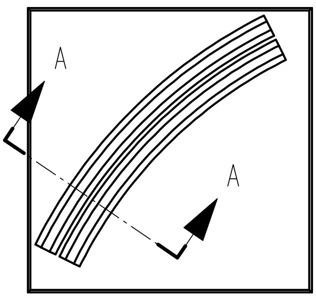
INCORRECT



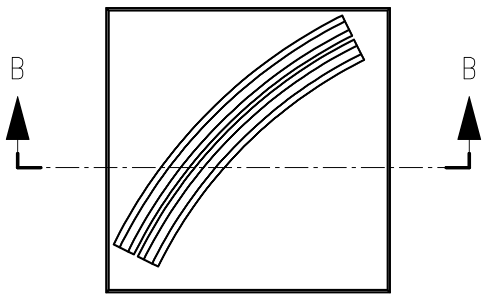
SECTION A-A



SECTION B-B



Section plane A-A is perpendicular to the rails.

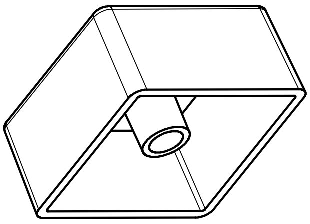


Section plane B-B is perpendicular to the slab, but not to the rails. The section view is not a true view of the rails, so both the dimensions and the profiles are incorrect.

SECTIONING - SELECTING VIEWS TO SECTION FROM

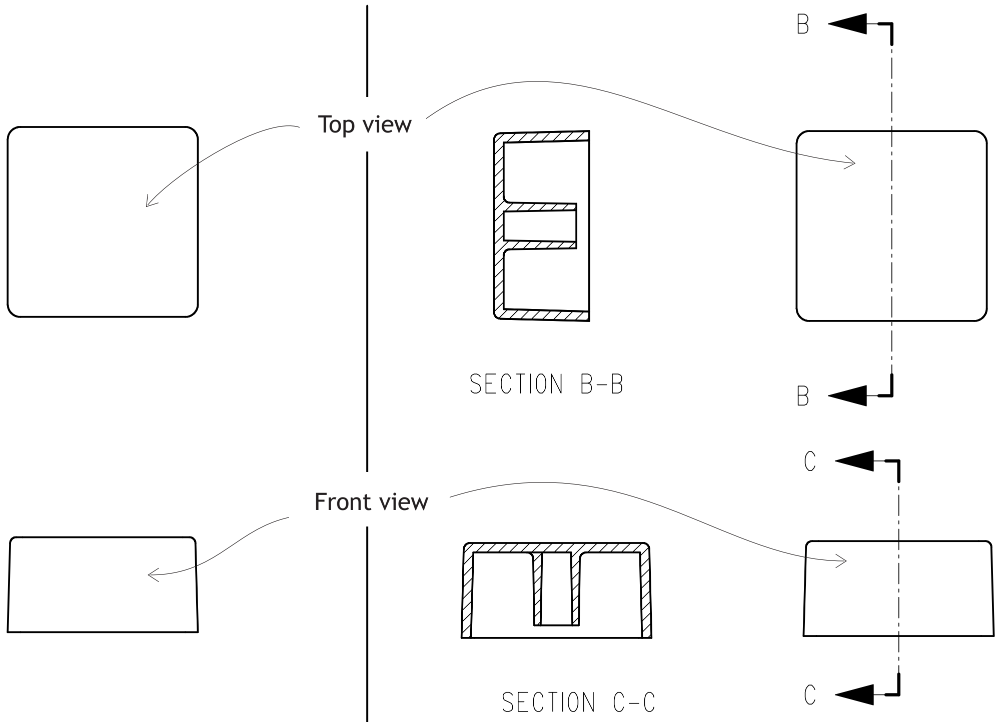
A section plane must always be placed in a view where the features that are being sectioned can be seen.

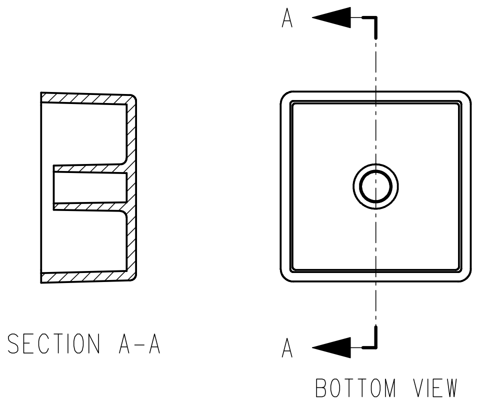
The isometric view here shows the undersides of a moulded part. The correct and incorrect of methods for placing section planes of this part are shown at the bottom.



CORRECT

INCORRECT

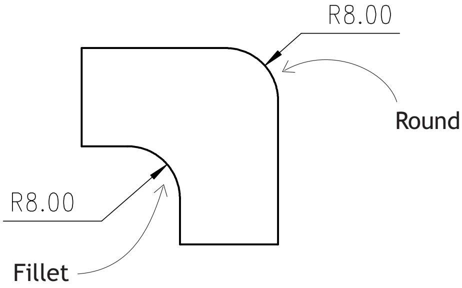




A bottom view here is essential for showing important details.

Sectioning from either the top or front view is incorrect because the geometry of the sectioned features is hidden in these views.

CORRECT

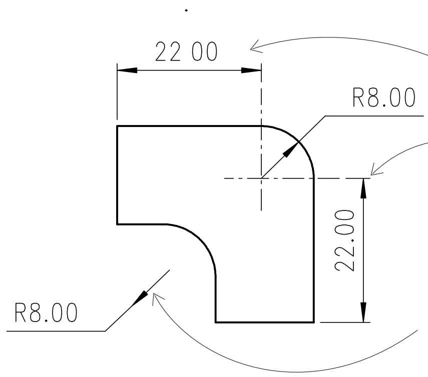


Fillets and rounds are arcs that:
a. connect two lines that are not parallel to one another
b. meet both lines at a tangent.

Never give positions or centre lines for fillets or for rounds, because their tangency is already a guarantee of their positions.

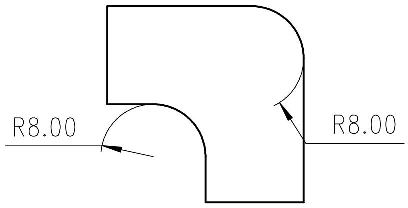
Providing position dimensions for fillets or rounds is incorrect because:
a. they are redundant (they repeat information that is already given)
b. they clutter the drawing.

INCORRECT

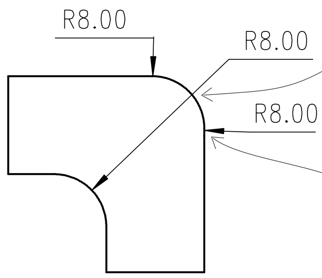


Position is redundant.

Position is implied(redundant), but also unclear, so is potentially confusing.



Cluttered



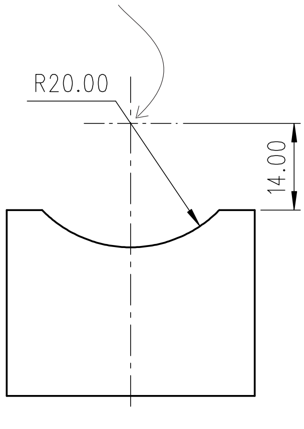
Dimension crosses lines unnecessarily.

Dimension points to the junction of two lines, therefore not clear.

DIMENSIONING CENTRE-LINE DEPENDENT ARCS

CORRECT

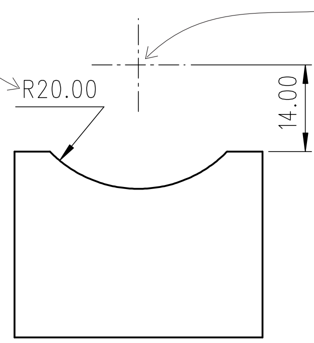
An arc dimension must pass through the centre lines that position it.



INCORRECT

This arc dimension is not connected to any centre lines, so it is a floating (position less) dimension.

These centre lines do not bisect any feature so they are meaningless.



PLACING DIMENSIONS IN CORRECT VIEWS

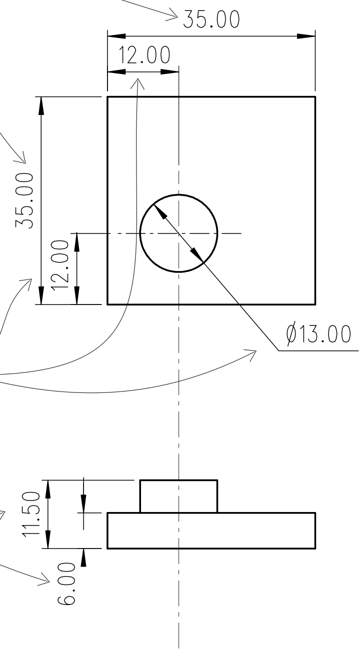
Dimensions should always be placed in the view in which both the *size* and the *shape* of the features are most obvious.

CORRECT

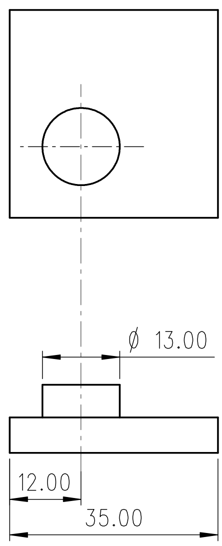
These two dimensions describe the geometry and size of the square.

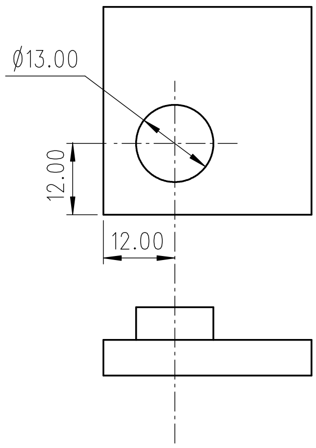
These three dimensions describe the position, geometry, and size of the circle relative to the square.

These two dimensions describe the heights of the features.

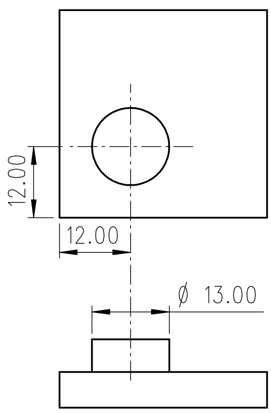


INCORRECT





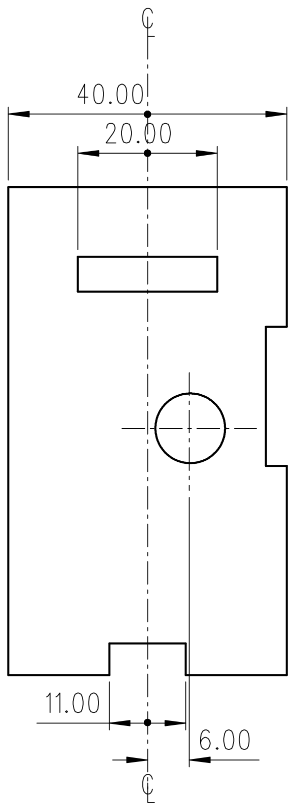
Size dimensions should always be clustered with their position dimensions. The two types of dimensions must not be separated and must not appear in two different views.



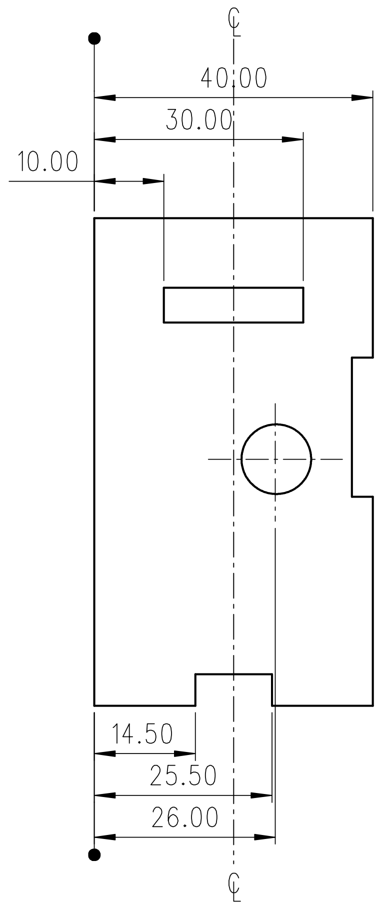
DIMENSIONING SYMMETRICAL SHAPES

The shape below is essentially symmetrical in its vertical axis, even though there are two features that are not symmetrically placed. The axis of symmetry is the correct datum, and the dimensions that are divided equally by this axis are marked as such by small dots placed at the intersection of the dimension line and the axis.

CORRECT

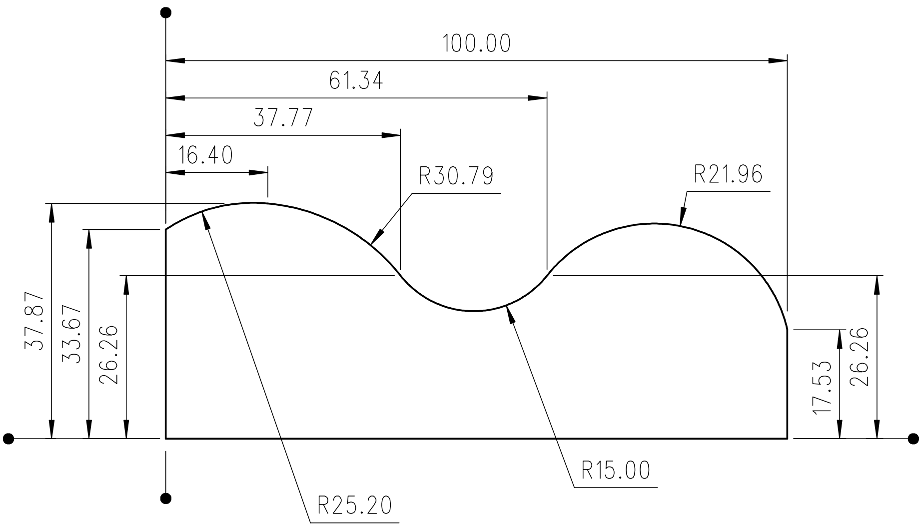


INCORRECT



DIMENSIONING MULTIPLE TANGENTIAL ARCS

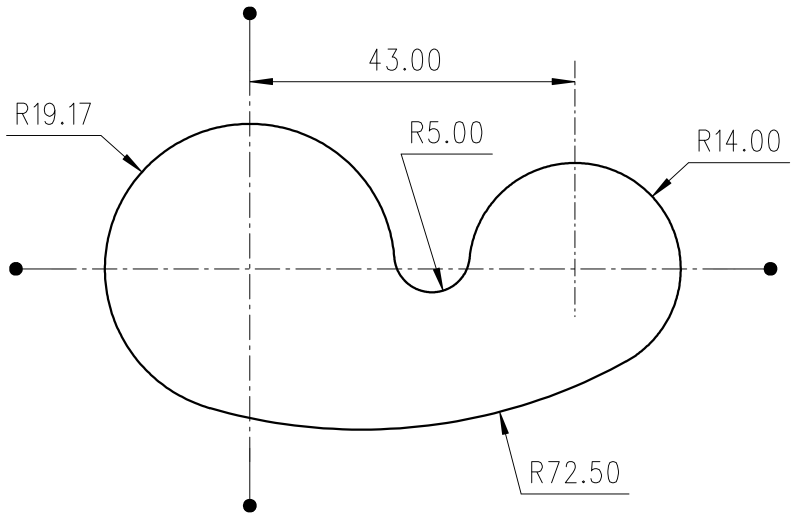
Smooth curves that undulate, or that resemble ellipses or parabolas, should be made into multiple tangential arcs in your CAD drawing to be suitable for manufacturing. Describing the start and end positions of these arcs is a simple matter of plotting their co-ordinates, as in this example.



For shapes with no straight edge or axis of symmetry available as a datum, use the centre of the most prominent arc as a datum.

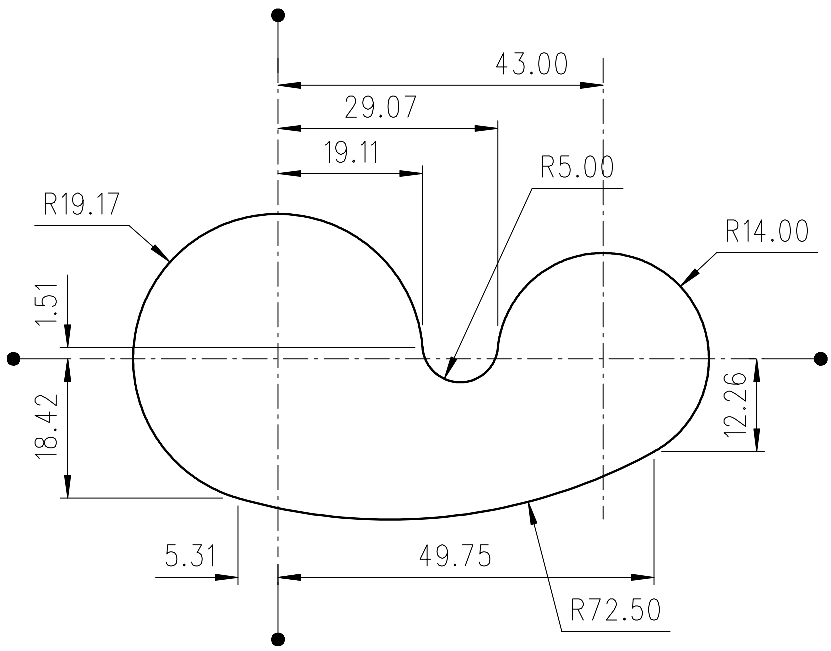
Believe it or not, the shape below is fully dimensioned. It only requires the description of the positions of the two fuller arcs relative to one another, and the radial sizes of all of the arcs, to be complete.

CORRECT



By contrast, the drawing below is over-dimensioned, therefore not as clear as it could be. The position dimensions (other than the distance between centres) are all redundant because the shallow arcs are, in fact, a fillet (R5.00) and a round (R72.50) respectively.

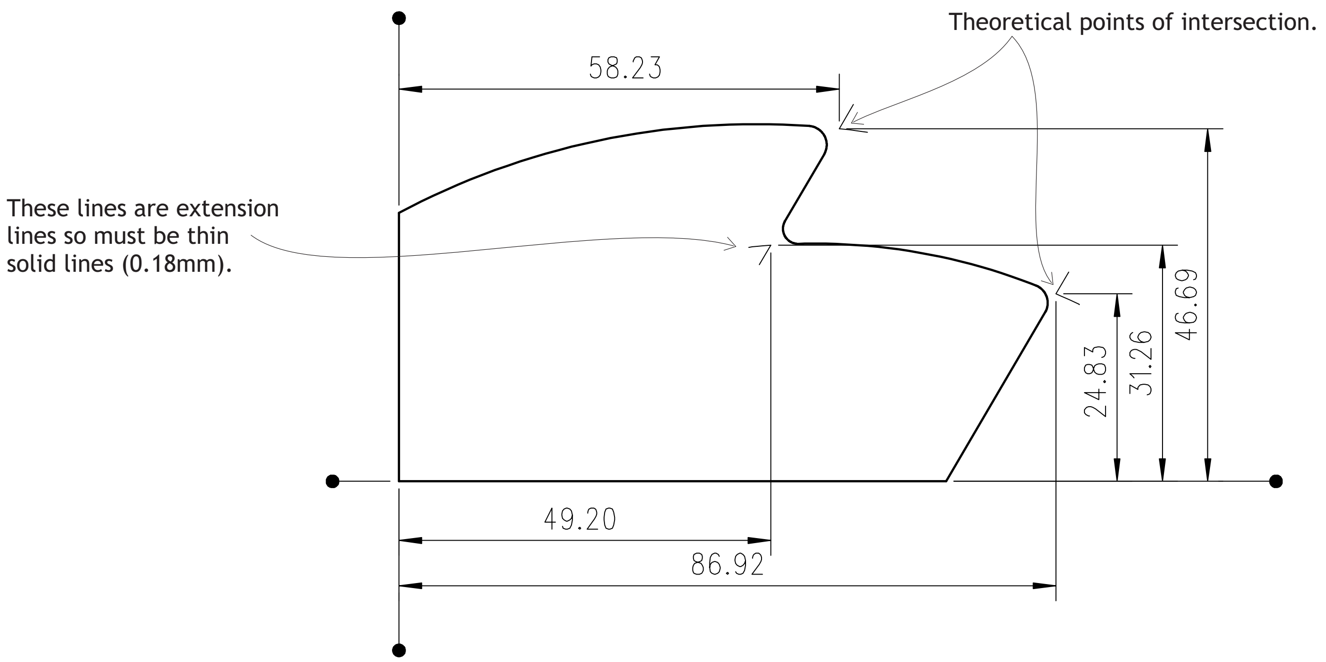
POOR PRACTICE



DIMENSIONING POSITIONS OF THEORETICAL INTERSECTIONS

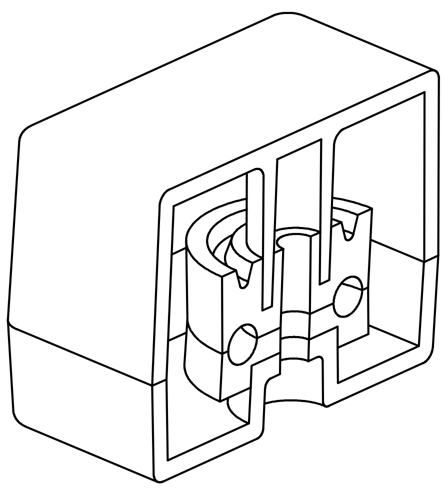
The following only applies for fillets and rounds other than those on right angle corners.

Where two lines (vertical, horizontal, diagonal, or radial) are truncated by a fillet or a round, the theoretical point of intersection must be used for dimensioning the position. This point needs to be drawn in manually (CAD programmes don't do it automatically) by extending both lines a short distance beyond the point of intersection.

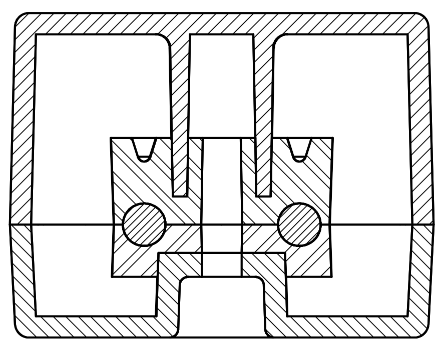


SECTIONING ASSEMBLIES

The parts in a sectioned assembly must be differentiated from one another by alternating the direction of the hatching (by 90°), and by altering the spacing of the hatching.



ISOMETRIC SECTION A-A



SECTION A-A