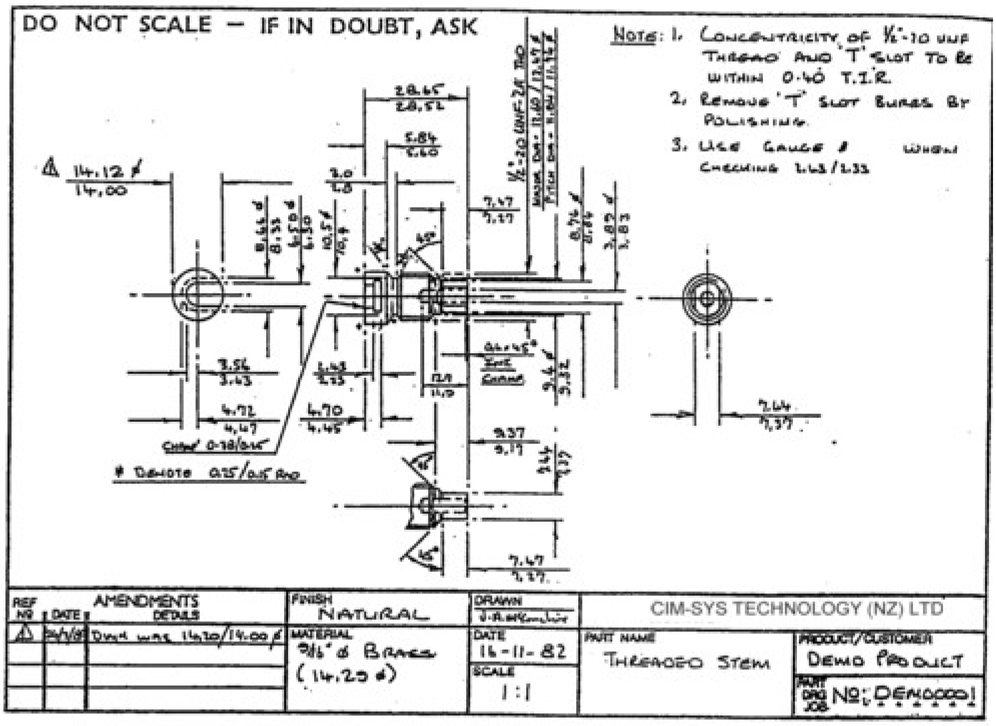
DESIGN FOR QUALITY

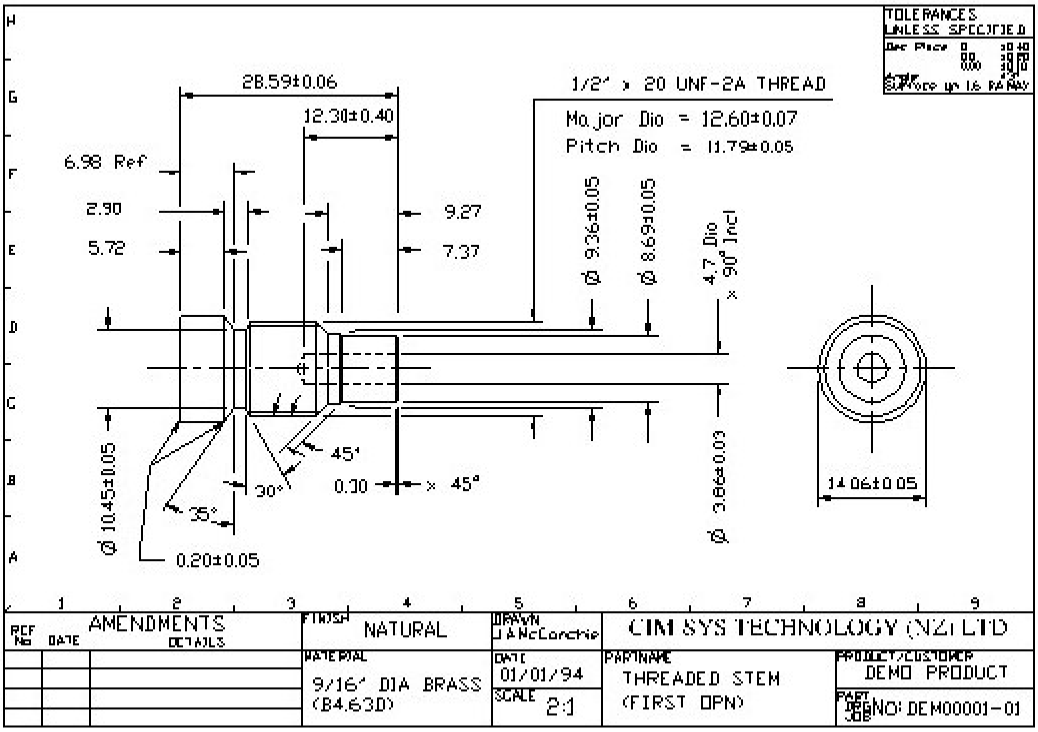
By Jim McConchie

To achieve a consistent quality product, it is essential to have a drawing which is easy to read and consistent in its dimensions, with the tolerances clearly stated, and which is drawn in a common projection ie. third angle projection. If first angle is used this must be clearly stated.

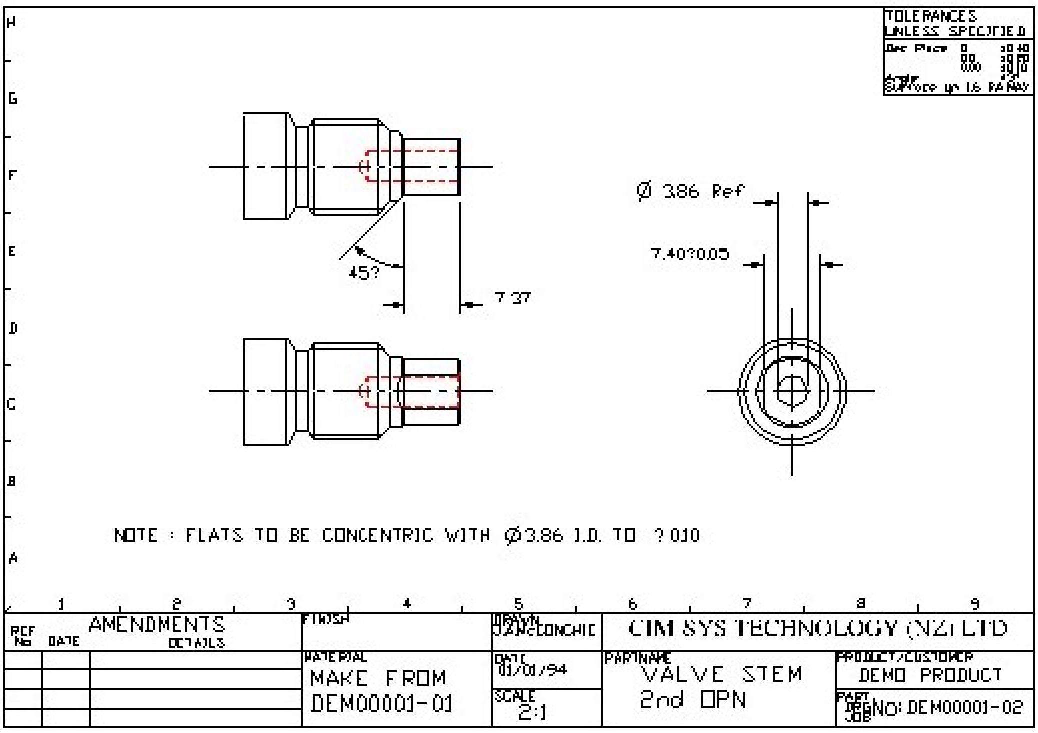
If the product is made in numerous operations, break the drawings down to match the operations, to minimise confusion.



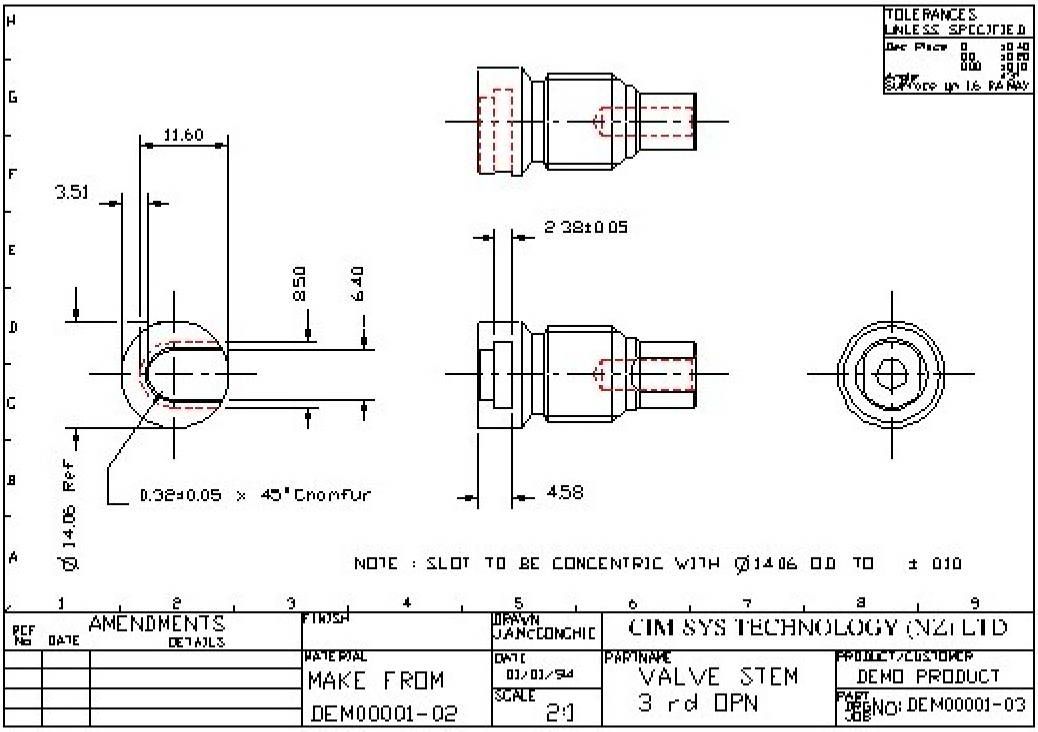
Original drawing with all dimensions.



Revised drawing for operation one



Revised drawing for operation two



Revised drawing for operation three

Now the operator does not have to determine which dimensions are relevant to the operation from all the other dimensions.

All dimensions should be to the target or mid tolerance and only bilaterally tolerances used.

Unilateral tolerances were used when two parts fitted together and the shaft component was dimensioned to show its biggest allowable size and conversely the hole was toleranced to show its smallest allowable size so that the required clearance/interference was always obvious.

When the machine was set to produce the product, the setter would realise the importance of this and set the shaft towards bottom limit and the hole towards top limit to allow for tool wear.

Unfortunately, because of natural variations within the process as now evident since practising Statistical Process Control (SPC) half of the products would have been out of specification until the tool wore down a bit. Hence our decision to use only bilateral tolerances and stating that the target / mid tolerance is what we set to and provided the process is capable, all the products produced will be to specification.

See Figure 4.1 for examples of tolerances.

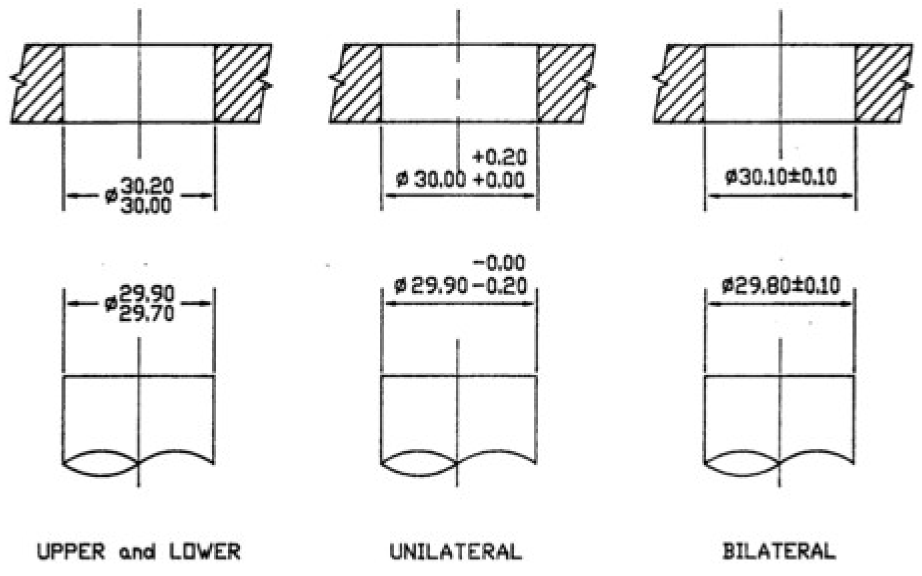


Figure 4.1 - TOLERANCING

Keep notes and comments to a minimum and dimension the drawing with this very important question in mind

HOW WOULD I MEASURE IT ?

It is common practice to dimension holes to their centre line.

But in practice, how do you measure this with standard measuring instruments ?. It would require the operator to perform some maths which could lead to errors. With modern digital measuring equipment and the appropriate software it is possible to measure the hole, then measure from the edge of the hole to the datum point and the software can calculate the actual distance and show what the actual centre distance is. Where this equipment is not available and when the tolerance allows, dimension the production drawing from the edge of the hole so that the operator has a direct measurement.

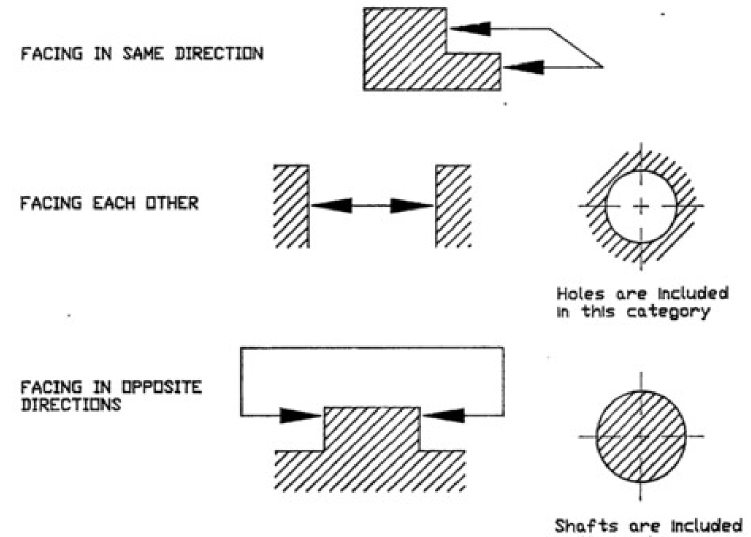


Figure 4.2 - PLANES of MEASUREMENT

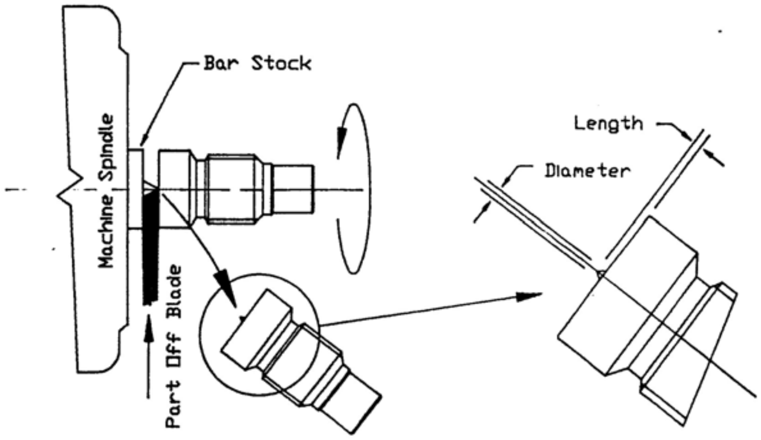
The possible planes of measurement using standard verniers and micrometers.

A high percentage of products are produced on automatic or CNC lathes from bar stock.
As the product is parted off (i.e. last operation is to cut it off the bar) a pip is left behind in the product as it falls away. If this part off pip is likely to cause problems then a tolerance on how big the pip is allowed to be should be added to the drawing. Various materials cause varying sized pips.

		High Speed Steel		Carbide	
Material	Size	Length	Diameter	Length	Diameter
Brass	Small	0.00	0.00	N/A	N/A
	Large	0.20	0.30	N/A	N/A
Leaded	Small	0,20	0.30	0.30	0.40
Steel	Large	0.30	0.50	0.40	0.60
Free Cutting	Small,	0.30	0.40	0.40	0.60
Steel	Large	0.40	0.60	0.50	0.80
Stainless	ALL	3.00	0.50	3.00	0.80
Steel	Remnants of swarfe remain attached to pip.				

As the cutting edge of the part off tool is required to be very sharp, it will not last very long. As the tool wears the pip will increase in size.

This pip can be removed by back drilling/milling on the Auto/CNC Lathe or by drilling, milling or linishing as a second operation. These additional operations add to the cost.



As it is impossible to make perfectly identical products because of variations during the process, caused by changes in temperature, wear and material variation, the designer must be fully aware of the various processes and their limitations. It is essential that the product design is such that it can be made using standard processes with normally achievable tolerances. All too often drawings are sent out for production with unnecessarily tight tolerances.

When tolerances each dimension the following questions need to be answered

(a) Does it fit inside something ?

(b) Does something fit inside it ?

(c) Does it act as a bearing surface ?

(d) Does the appearance matter ?

For (a) and (b) the type of fit required ie. clearance, transition or interference dictates the size of the allowable tolerance. But be realistic, is the tolerance achievable!! If there is no contact with any other part a larger tolerance can be applied.

For (c) and (d) - The surface finish is important for bearing life, the finish required dictates the process and affects the cost. Modern CNC machine tools with carbide/ceramic tooling can achieve very good surface finishes similar to grinding but at a reduced cost. If the appearance is important then a surface finish should be specified, but again, be realistic.

The Tolerance dictates which type of measuring instrument must be used to measure the product during production. As a guide the instrument must be at least ten times more capable than the tolerance to be measured. For example, to measure a Tolerance of +/- 0.01 mm, the measuring instrument must have an accuracy/resolution of +/- 0.001 mm to ensure that the results are a true representation of the product and allow for small variations in the environment, operator feel and instrument repeatability. It is extremely important that the operator can trust the results and make the correct adjustments to the process.

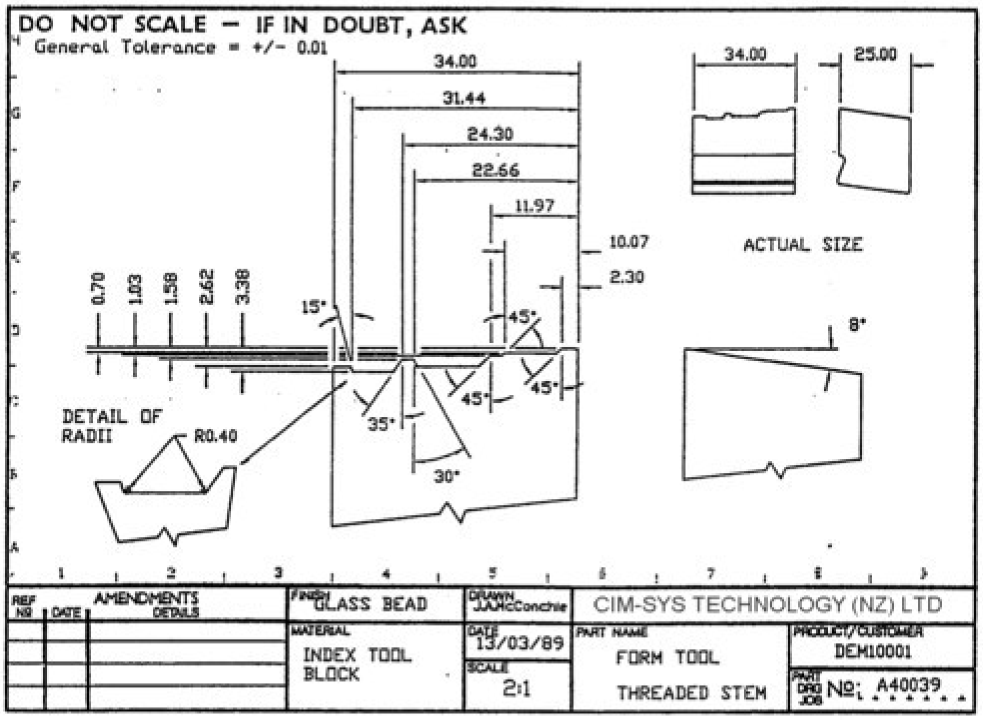
The shape of the product also governs how it is to be measured. A simple product such as a bush with parallel sides and square across the ends is easy and quick to measure, but a complex shape with lots of curves and angles is extremely difficult. If this complex shape is critical to its operation , such as the curve on a turbine blade, then expensive three dimensional equipment is required.

The cheaper alternative is the Optical Profile Projector, but for real time measurement these are time consuming unles a CNC controlled or Digital Vison unit is used. The product may need to be sectioned to enable internal dimensions to be checked, which is counter productive.

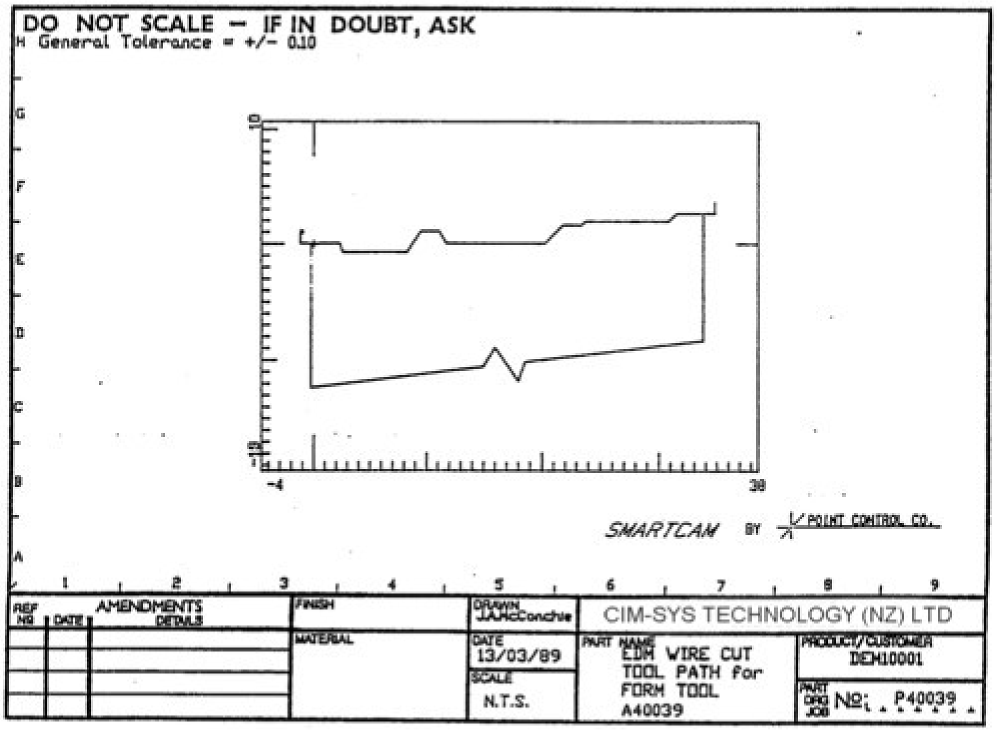
To minimise the number of characteristics that need to be checked, form tools for external shapes and stepped drills or trepanning tools should be used.

With modern drafting tools like CAD (Computer Aided Draughting and CAM (Computer Aided Manufacturing) it is possible to design the product on CAD, drawn to mid tolerance, and then design the tooling at the same time. The digital data is then transferred to a CAM package and the machine code to run a CNC EDM Wire Cut machine or CNC Grinder is produced to make the tool within a tolerance of ten times the product tolerance. Alternatively large scale optical templates can be plotted to enable the toolmaker to grind the tooling on conventional tool and cutter grinders with optical comparators. Scale templates for checking the product on a profile projector can also be produced to facilitate Quality checks. By accurately producing the tooling only two dimensions per tool need be checked once the process is in control. Provided the tool is undamaged and tool wear is monitored the products will meet their specification.

If the shape is complex but not critical it could be possible to either cast or forge the product and only check the overall dimensions. The greater tolerances for these processes must be allowed for and jigging or chucking carefully designed to allow for the variations.



Form Tool Drawing derived from CAD drawing DEM00001-01



Tool Path for Wire EDM Cutting of Tool from High Speed steel or Carbide.

GENERAL NOTES ON DIMENSIONING

CHAMFERS :

Internal chamfers should be dimensioned across their diameter with the included angle stated. External chamfers can be dimensioned as above or on their length with the angle to the axis stated or dimensioned.

Chamfers for external threads should be dimensioned as a diameter by the included angle as there is no intersecting point left once the thread has been cut.

RADII :

Internal radii, where not critical, should state a maximum radius allowable. If critical, the actual radii with tolerance should be stated.

If carbide tooling is used, the most common Radii is 0.4mm with alternatives being : 0.2 (restricted to a small range of tips 0.8 and 1.2 For High Speed Steel the radii is ground to suit, 0.1mm radii being a practical minimum to allow for tool edge breakdown between tool re-sharpening

Undercutting of corners can be achieved by the use of carbide grooving tools with a range of widths of 1mm to 5mm and corner radii of 0.15mm to 0.30mm. Stress raising caused by undercutting should be considered during the design phase.

If burrs are acceptable, a size should be specified i.e. 0.1mm max. or a chamfer specified to ensure that all edges are free of burrs.

Note: Chamfers are preferred for external corners and Radii for internal corners.

Carbide has a very high resistance to wear as compared to High Speed Steel, hence it is preferable to utilise carbide wherever possible. This ensures that the components are dimensionally stable for a longer period and if indexible inserts are used the down time is reduced when the tooling needs to be replaced.

THREADS

The specification for threads should include the size, type and class of thread. i.e. M10 x 1.5 - 6g for a 10 mm dia by 1.5 pitch external ISO Coarse Thread to class 6g.

i.e. 1/2" x 20 UNF - 2B for a 1/2" dia by 20 TPI internal Unified thread to class 2B.

The drawing should include the major and effective diameters for external threads and the minor diameter for internal threads as these dimensions can be readily measured.

MATERIAL STOCK SIZE :

If the material stock size to the manufacturers specifications is acceptable then the word STOCK should be shown beneath the dimension. When the outside dimension of the component is critical and the required tolerance is tighter than the manufacturers specification or the dimension is smaller than the stock material, a tolerance must be applied to that dimension.