FORM TOOL DESIGN MANUAL

SOMMA TOOL COMPANY, INC.

109 Scott Road (At exit 25 off I-84) P.O. Box 2559
Waterbury, CT. 06725 USA
Phone: 203-753-2114 Fax: 203-756-5489

Internet - http://www.sommatool.com
E-mail - somma@sommatool.com

By: Herman R. Somma

Rev. 9/98

TECHNICAL MEMBER OF PRECISION MACHINED PRODUCTS ASSOCIATION
PMPA

SERVING THE SCREW MACHINE INDUSTRY SINCE 1939

FORM TOOL DESIGN MANUAL

—— CONTENTS ——

INTRODUCTION

--- CONTENTS ---

INTRODUCTION

“The heart of the screw machine”

A form tool is one of the simplest of all tools:

It is merely a piece of steel or carbide with a shape cut into it that is the mirror image of that which is to be produced on the part.

However, this simple form tool is a very important tool, the tool around which the screw machine is built. In actuality it is *“The heart of the screw machine”* since every job has some sort of a form in it. The work the form tool will perform is essential to determining how the job will be run, how the cams will be laid out, the time cycle and what the sequence of operations will be.

Because the form tool is so critical to the successful manufacture of the part, we believe it should be given the full attention it deserves. Any extra effort spent in designing or making the tool will be repaid many times over in improved quality of the parts, trouble free operation and longer tool life between grinds.

We trust that by answering the many questions that arise in the process of form tool engineering, this manual will enable you to design form tools that will achieve these results.

Note: Although this manual is primarily intended for the form tool designer, we believe it should also be of assistance to anyone involved in the engineering and/or operation of screw machines.

HOW TO USE THIS MANUAL

This manual is divided into 6 sections:

SECTION 1: (Pg. 7) This section addresses the problem of how to design the tool in those areas that do not control part dimensions.

There usually is no problem in determining the dimensions the part must be formed to, since they have to be specified on the part print. However, the part of the tool design that is for those portions of the tool that do not necessarily control the dimensions of the part, but that are critical to the function of the tool, are the areas which are the most troublesome to the tool designer.

To assist in designing these unspecified portions of the tool, Section 1 consists of a check list of most of the common situations encountered that must be determined before starting to design the tool.

SECTION 2: (Pg. 27) This section explains the mechanics of drawing the tool to make it wire EDM or toolmaker friendly. How to design the various types of tools (insert, circular, dovetail). How to calculate step corrections and how to compensate for the effect these corrections have on angles and radii.

SECTION 3: (Pg. 45) This section is loaded with helpful hints that will make the tool perform more efficiently or eliminate problems.

SECTION 4: (Pg. 57) This section is a compilation of many of the reference tables you will need to refer to in designing form tools — and all in one place.

SECTION 5: (Pg. 59) This section listing the most common of the various high speed steels, carbides and coatings available will assist you in deciding which type material the tool should be made from based on its application.

SECTION 6: (Pg. 62) This section explains the geometry of the cutting edge of the tool that makes it necessary to correct step differentials, as well as angles and radii, in order to produce the desired dimensions on the part. It is important to understand this tool geometry but is not essential to designing a tool.

These recommendations will assist you in being able to design a very functional tool that will produce a good part and be screw machine operator and tool maker friendly.

Good Luck!!

SECTION 1

CHECK LIST OF THINGS TO DETERMINE BEFORE STARTING TO DESIGN THE FORM TOOL

TYPE OF TOOL

SOMMA RESHARPENABLE INSERT FORM TOOLS/SHAVE TOOLS:

Specify thickness, width, length and face angles according to holder being used. Also, if used on a single spindle machine, whether forward or reverse (right or left hand) spindle rotation.

Following are dimensions of standard inserts:

Insert Type	Width	Master Length	Thickness Resharpenable	Thickness Throwaway	Front Clearance	Top Rake	Form Depth Maximum
Form	.3745	.630	5/16	1/8	12 Deg	5 Deg	9/32
Form	.7495	.630	5/16	1/8	12 Deg	5 Deg	9/32
Form	1.1245	.630	5/16	1/8	12 Deg	5 Deg	9/32
Form	1.2495	.956	3/8	1/8	12 Deg	5 Deg	7/16
Form	1.7495	.956	3/8	1/8	12 Deg	5 Deg	7/16
Shave	1.1245	.875	3/8	1/8	0 Deg	2.5 Deg	7/16
Shave	1.7495	1.125	3/8	1/8	0 Deg	0 Deg	9/16
Cutoff	.130	1.250	1/2	—	12 Deg	5 Deg	7/16

CIRCULAR FORM/SHAVE TOOLS:

Specify major diameter, offset of cutting edge from center line of tool, pinholes, keyways, side teeth or other requirements according to machine or toolholder being used on, also top rake, if any. Also whether for forward or reverse (right or left hand) rotation.

Following are blank specification for various screw machines.

Machine	Diameter	Center Hole	Offset	Min. Width	Pinholes No.	Pinholes Dia.	Pinholes Depth	Pinholes Circle
B&S No. 00	1.750	3/8-16 Thd	.125	1/4	8	5/32	5/16	1-1/16
No. 0	2.250	1/2-13 Thd	.156	5/16	6	3/16	5/16	1-3/8
No. 2	3.000	5/8-11 Thd	.250	3/8	6	3/16	3/8	1-1/2
Dav. Form	2 .000	9/16 Ream	.125	1/4	6	9/64	thru	1-1/4
Dav. Shave	1.301	7/16-20 L.H.	.062	1/4	--	--	--	--

See machine manufacturers specifications or Somma catalog for blanks for National Acme, New Britain, Cleveland and #4 and #6 B&S machines.

DOVETAIL FORM/SHAVE TOOLS:

Specify dovetail width to theoretical sharp corners, depth of dovetail, inner corner radii and outer corner flats as required for the dovetail holder it is being used on. Also other special requirements, such as location and size of tapped hole for jack screw or keyways if called for. Specify overall height, width, length, front clearance angle, top rake angle, if any, offset of form of tool towards spindle. Also if used on a single spindle machine, whether for forward or reverse (right or left hand) spindle rotation.

See Somma catalog for specifications of available standard blanks made to machine manufacturers specifications.

RIGHT HAND OR LEFT HAND SPINDLE ROTATION

RIGHT HAND ROTATION (ALSO CALLED FORWARD OR COUNTERCLOCKWISE):

Multi-spindle machines including Davenports always rotate in a forward direction. There fore, all form tools, shave tools or other cutting tools must be right hand cut.

Because these machines only run right hand, tapping operations can only be performed by rotating the tapping head in the same direction as the spindle at a slower speed for tapping in and then at a higher speed than the spindle (same direction) for backing out the tap.

LEFT HAND ROTATION (ALSO CALLED REVERSE, BACKWARD OR CLOCKWISE):

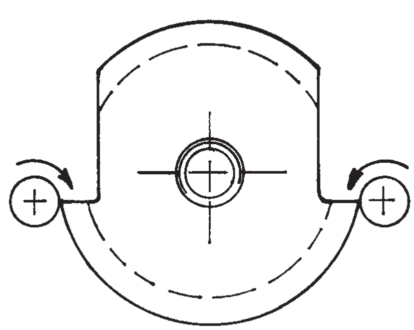
Single spindle machines (Brown & Sharpe Type) run forward in most cases. However, when tapping, or threading externally with a solid die that does not open, the forming must be done with the spindle running in reverse, therefore, the form tools must be left hand cut.

The reason the machine runs in reverse when forming is that there is only a choice of two speeds (RPM) in any cycle. Therefore, since the threading "in" has to be right hand and at the slower speed, the threading "out" has to be left hand and at the higher speed. Therefore, all other operations including forming, drilling, etc. must be performed in the left hand mode because they require the higher spindle speed.

However, if the thread is left hand the opposite is true, meaning that the threading operation would be left hand, slow speed and all other operations including forming would be right hand.

Also, if using an opening die head there is no reason to reverse the spindle to unthread the die and forming is done in the high speed, right hand mode.

Note: This does not affect circular tools since they can be sharpened for right or left hand cutting. See Figure #1. However, carbide tipped circular tools, dovetail tools, or insert type tools must be designed accordingly.



L.H. **Figure #1** R.H.

FINISHING TOOL OR ROUGHING TOOL

FINISHING TOOL:

Finishing tools are designed to produce the part exactly to the dimensions as specified.

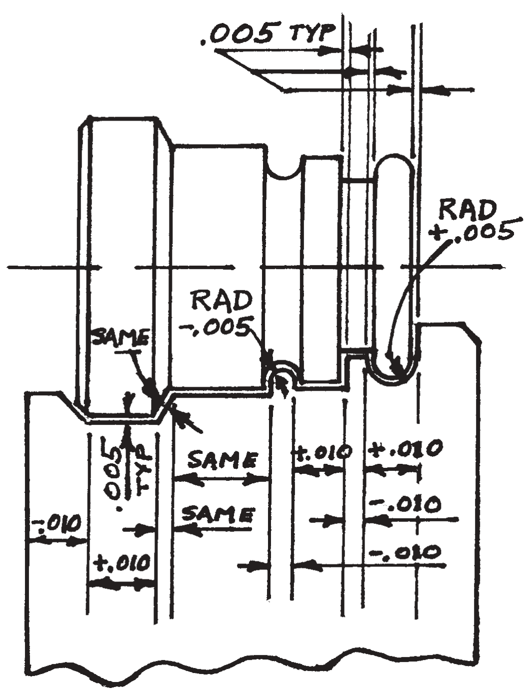
ROUGHING TOOLS:

Roughing tools are used when the final dimensions are produced by a shave tool (also called "size tool") or when the stock removal is divided between two or more tools.

ROUGHING FOR SHAVING TOOLS:

When the form tool prepares the part for a shave tool, the form tool has to be designed so as to leave all dimensions oversize. The standard practice is for the rough form tool to make all diameters .010" oversize and to leave .005" stock to be removed on all side vertical surfaces, angles and radii. See Figure #2.

Note: There are times when the shave tool finishes only one critical diameter of the part. In this case, the form tool leaves .010" on the critical diameter but finishes all the other dimensions to size.



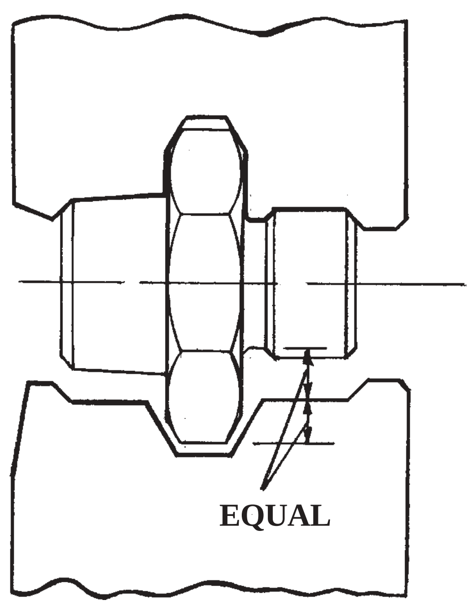
ROUGHING TOOL
FOR SHAVING

Figure #2

ROUGHING FOR EQUALIZATION OF STOCK REMOVAL:

Since all operations are performed simultaneously in the different spindle positions on multiple spindle machines, the time to make one part is equal to the time it takes to perform the longest operation. For example, if .500” has to be removed from the diameter of a part, instead of taking the .500” off with one tool in one position, the cut is divided between two tools in different positions with the first tool taking .250”, and the second tool taking off the balance of .250”. This means that the time needed to remove the material will be cut in half, with a corresponding reduction in the time it takes to make the part. Also, since the tools are only doing half the work, they give better finishes and sizes and last longer between sharpenings. The first tool should follow the general outline of the part but need not be as detailed as the second tool, thereby reducing its cost. See Figure #3.

Roughing tools can also be used on single spindle machines if the slide positions are available. In this case, they should be designed following the same rules as for roughing for shave tools except it is a good idea to leave diameters .020/.030” oversize and .010/.020” on the side surfaces. Roughing will improve production because the tools will last longer between sharpenings, and maintain better size control and finish.



PARTIAL FORMING
ON MULTIPLE
SPINDLE MACHINES

Figure #3

PORTION OF PART TO BE FORMED

SINGLE OR MULTIPLE TOOLS:

It is most desirable to form the complete part with one form tool whenever possible. This simplifies set up and locks all the length and diameter dimensions into a single tool. However, if the length of the part in relation to its smallest diameter is such that the part would deflect or break off in forming, the form should be distributed between two or more form tools using the following as a guide.

DIAMETER TO LENGTH OF CUT RATIO:

Length of form being cut outboard of the smallest diameter should be no longer than three times the smallest solid diameter. Although the forming operation is usually performed before the drilling operation, if the smallest diameter has a hole through it prior to forming, the length of form should be no more than three times the total of the wall thickness (total wall thickness = smallest diameter minus diameter of hole through it). See Figure #4.

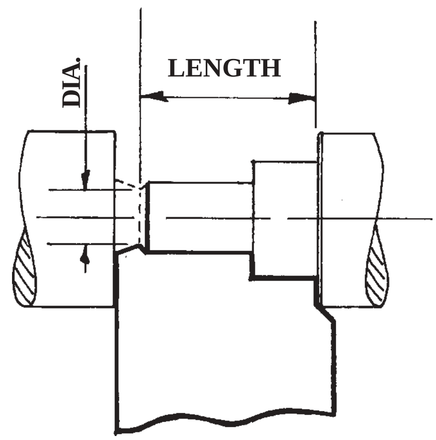


Figure #4

Ratio can be increased to 4 to 8 times smallest diameter if part is supported from the turret while being formed; examples of supports: rod in hole, drilling or reaming while forming, round support with hole bored to diameter of front end of part, etc.

Note: These ratios are a guide and may vary due to feed rate, type of material being cut, tool geometry or machine condition. Also at times the division of the form may have to violate these rules in order to have tool straddle a critical "king" length to maintain tolerances.

COMBINATION CUTOFF AND FORM TOOL:

Unlike multiple spindle machines that can have four or more forming tool slides, single spindle machines (B&S type) have only a front and a rear cross slide that can be used for form tools, plus one or two vertical slides that are normally used for cutoff blades or other light duty operations. Therefore, on single spindle machines, it is very often advantageous to incorporate a cutoff blade into the form tool, thereby freeing up one of the other slides for some other operation. This can also simplify the setup and running of the job by eliminating the need for a separate cutoff tool. This type tool can cutoff the part and also form the front end of the next part for any length within the diameter to length ratio as explained previously.

CHAMFERING END OF PART BEING CUTOFF:

As the cutoff blade approaches center the diameter still holding the part becomes progressively smaller and weaker. If the cutoff tool attempts to chamfer the part being cutoff, the added cutting pressure will tend to break part off sooner and inconsistently. Therefore, this practice is not recommended. See Figure #5.

Forming on outboard side of the cutoff blade is however very common on Davenport machines. This is because these machines are equipped with a pickoff collet that closes down on the part and rotates it at the same speed as the main spindle. Since the part being cutoff is held securely and is rotating at the same speed as the part still in the spindle, it can be readily formed and in fact, eliminates the need for an angle on the cutoff blade. See Figure #6.

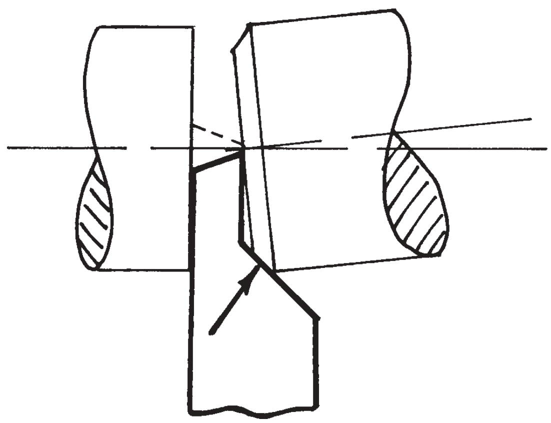


Figure #5

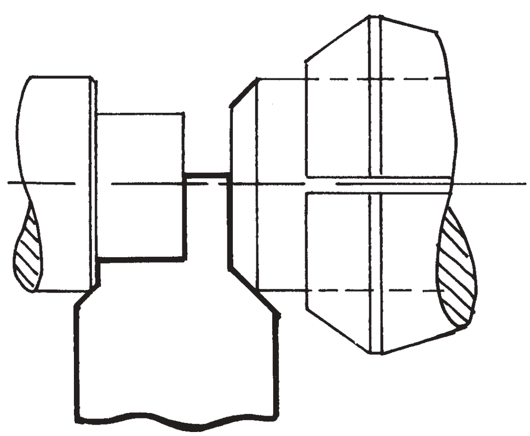


Figure #6

DETERMINING CORRECT CUTOFF BLADE WIDTH

IMPORTANCE OF CUTOFF BLADE WIDTH:

For efficient job performance and to save material the cutoff blade width should be the narrowest blade that will cutoff the part without breaking off, deflecting or burning out.

RECOMMENDED CUTOFF THICKNESS:

The following chart shows the blade widths we have found to work out successfully in most cases.

Stock Diameter	Blade Thickness
1/8	.030
3/16	.040
1/4	.050
5/16	.060
3/8	.070
1/2	.080

Stock Diameter	Blade Thickness
5/8	.090
3/4	.100
1"	.120
1-1/4	.140
1-1/2	.160

Note: These widths are only a guide and may have to be increased or at times can be decreased depending on the part configuration. See Section 3 - "REDUCING CUTOFF BLADE THICKNESS".

PURPOSE OF ANGLES ON CUTOFF BLADES:

An angle is put on the cutoff blade with the high point on the side of the blade that is against the back of the part being cutoff. This angle forms a cone on the front end of the part with the smallest end of this cone at the back end of the part falling off. Since the weakest point is where the part will separate from the bar, this means that the breakoff diameter will be very small and will leave a minimum burr. After the completed part is separated from the bar, the tool continues to advance until the lower end of the angle passes center of the front of the next piece still in the machine, producing a clean flat face. See Figure #7.

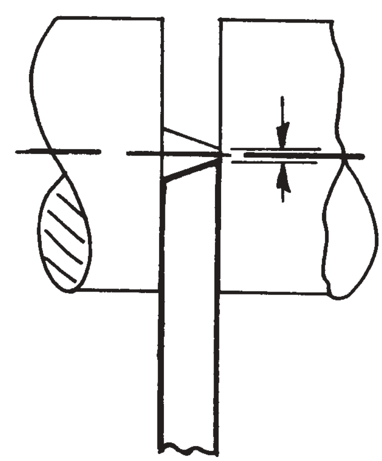


Figure #7

Obviously the steeper the cutoff angle, the smaller the break off point will be. This is very desirable except, that depending on how difficult to machine the material is, if the angle is too steep the heat generated in cutting will tend to burn out or wear out the point prematurely because there is less body to carry away the heat generated.

The standard practice for cutoff angles is as follows:

15 Degrees for steel, bronze, nickel and other difficult to machine materials.
23 Degrees for brass, aluminum, copper, zinc and other easy to machine materials.

FACING OFF PART

INTO HOLE:

Pass hole or countersink diameter .010 on the side (.020" on diameter)— if hole is over 1" diameter increase to .015" on side (.030" on diameter). See Figure #8.

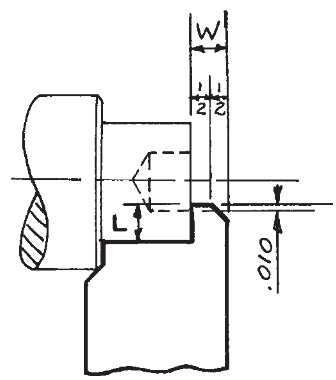


Figure #8

PAST CENTER:

Pass center according to stock size as follows: See Figure #9.

Diameter	Pass Center
Up to 1/8"	.005"
to 3/8"	.008"
to 3/4"	.010"
to 1"	.012"
Over 1"	.015"

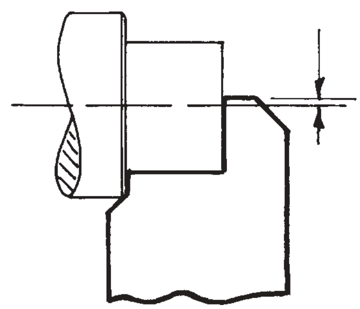


Figure #9

These past center distances also apply for combination cutoff and form tools. See Figure #10.

Note: When facing off the front end of a part with a carbide tool, especially when cutting a difficult to machine material, tool life and part finish can be improved by adding a .010"/.015" x 45 degree chamfer on the leading edge of the face-off portion of the tool. Of course the base diameter has to be reduced accordingly.

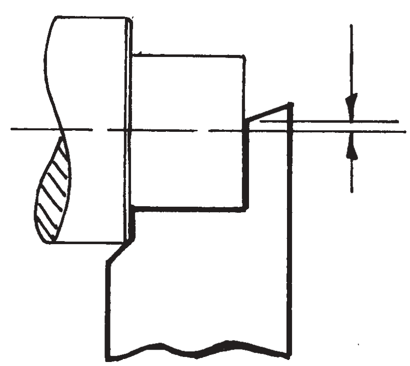


Figure #10

WIDTH OF FACEOFF:

Total width should be no less than the length of the face off portion of tool. This width should be approximately half flat and the balance should run off at 45 degrees. This is so that face off shoulder will not interfere with other tools in the setup. Ex: The flute of the drill when drilling at the same time as forming. See Figure #8.

EXTENDING PAST OUTBOARD END OF PART

CHAMFER CORNER:

If tool extends past the outboard end of the part, it will most likely roll a burr on the front surface. To remove this burr, a step should be added to the tool that will chamfer the end corner with either an angle or a radius.

If no chamfer is specified, corner should be chamfered at .005” x 45 degrees. To accomplish this a .015” x 45 degree projection is added to the tool starting .005” short of the length of the shoulder being chamfered and continue with a flat surface past the end of the 45 degree shoulder for a distance of approximately the same width of the cutoff tool being used. See Figure #11.

For parts 1” diameter and over, the chamfer can be increased to .010” and the projection to .025” x 45 degrees.

If a radius chamfer is preferred, a projection .025” high should be added to the tool. This projection should have a .010” radius in the corner ending at the end of the length of the shoulder to be chamfered. Also, an angle of 10 degrees to the face of the part and tangent to the .010” radius should be specified. This angle will allow for and blend with any variations in the length of the part. A flat surface should extend past this radius projection for a distance approximately the same width as the cutoff tool being used. See Figure #12.

For parts 1” diameter and over, the radius can be increased to .015/.020” and the projection to .050/.060”.

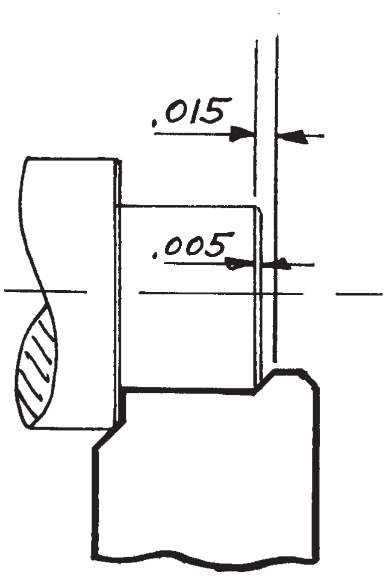


Figure #11

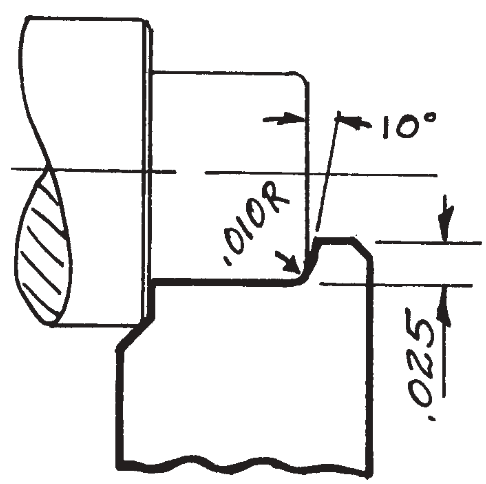


Figure #12

EXTENDING PAST BACK END OF PART

CHAMFER CORNER:

If tool extends past the cutoff end of the part it will most likely throw a burr on the back corner when it is cutoff. To remove this burr, a projection that will produce either a 45 degree chamfer or a radius as described for the outboard end should be added to the cutoff end of the tool. See Figure #13.

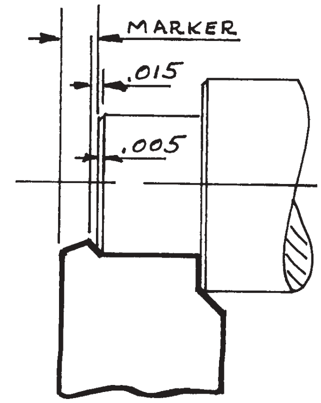


Figure #13

DEPTH OF PROJECTION:

The depth of this projection into the part may have to be minimized if it affects the "diameter to length ratio" as previously explained. This projection is included as part of the total "marker" width. See Figure #13.

CUTOFF MARKER:

A shoulder known as a "marker" should be added to the tool past this projection to ease the work of the cutoff tool. The distance this shoulder extends past the end of the part can be either greater or less than the width of the cutoff tool depending on how the job is laid out.

WIDER THAN CUTOFF:

For multiple spindle machines, the total width of the marker (portion of projection that extends beyond chamfer on part plus added shoulder) should be .005/.010" wider than the cutoff blade thickness. This is because all tools advance the same distance at the cutting feed rate, therefore, the cutoff blade dives in at a high feed rate and only reduces to the slow feed rate while cutting through the remaining wall thickness. This reduces cycle time and can also save material by allowing the use of a narrower cutoff blade. See Figure #14.

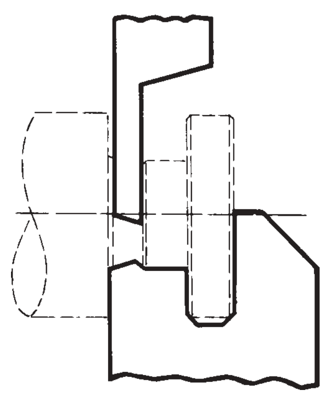


Figure #14

Usually the job layout provides for a way to remove the step that this will leave on the face of the next part. However, if for some reason the job layout does not provide a way to remove this step, the marker then must be narrower than the cutoff blade.

Since single spindle machines do not have as many slides as multiple spindle machines, the marker can only be wider than the cutoff width when the diameter it turns to is larger than the front end diameter of the next part or the step is faced off or the drilled away in a subsequent operation. See Figure #15.

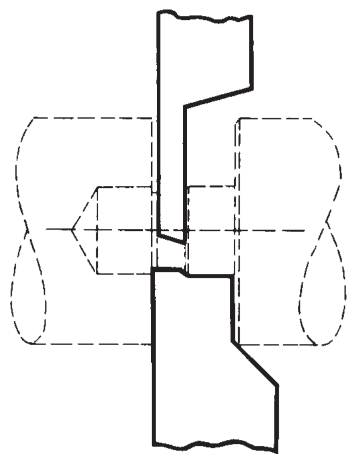


Figure #15

NARROWER THAN CUTOFF:

For single spindle machines, the marker should be .005/.010" narrower than the cutoff width since the cutoff tool can have a longer feed and, therefore, it does not have to dive in.

ANGLE ON MARKER:

The marker should be tapered away from center at an angle equal to that of the cutoff tool. This will make the part more rigid, especially if the part is close to the three to one length to diameter ratio. It is also a good idea to specify a .005/.010" flat on the high point of the angle to reduce wear on the point, especially on carbide tools.

Note: When roll threading or knurling, the marker angle should be 0 degree if it interferes with the overhang of the thread roll or knurl. See Figures 32 and 33.

CLEARING ROUND STOCK DIAMETER

CHAMFER CORNERS — 45 DEGREE — RADIUS:

If the tool extends vertically past a stock diameter, that is it does not remove any material from that stock diameter, it will most likely throw up a burr on the outer corner. To remove this burr, a step should be added to the tool that will chamfer this outside corner with either an angle or a radius. If no chamfer is specified, corner should be chamfered at .005” x 45 degree. If the tool is only cutting a form on one side of the stock, a 45 degree shoulder should be added to the tool starting at a diameter .010” smaller than the stock diameter and extending for a width of approximately .030”. See Figure #16.

However, if the tool is cutting a form on both ends of the stock leaving an uncut stock size in between, we would add an angle to both ends that starts at a diameter that is .010” smaller than the stock diameter and ending at a diameter that is .020” larger than the stock diameter and connecting these with a horizontal surface. This surface should clear the stock diameter by .010” on the side. See Figure #17.

For parts 1” diameter and over, these dimensions can be increased to starting at .020” under and ending at .030” over the stock diameter.

If a radius chamfer is preferred and the tool is only forming on one side of the stock, a shoulder approximately .040” wide x 10 degree from the horizontal should be added to the tool with a .010” radius in the corner and located so that the tangent point between the .010” radius and the 10 degree angle is located on the outer diameter of the stock. See Figure #18.

If the tool is cutting a form on both ends of the stock leaving an uncut stock size in between, we should run a horizontal surface between the two formed areas at a diameter that is .020” larger than the stock diameter and have it taper up on both ends at 10 degrees to intersect with the vertical surfaces at the stock diameter and with a .010” radius in each corner. See Figure #19.

For parts 1” diameter and over, the radius can be increased to .015” or greater.

The 10 degree runoff angle is added so that there will be no distinct line on the part where the radius ends if the tool were to be fed in too deep. Also, it will eliminate dig ins on one side of the part due to normal runout of the collet.

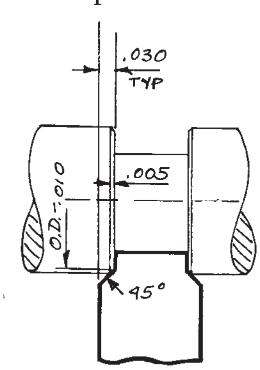


Figure #16

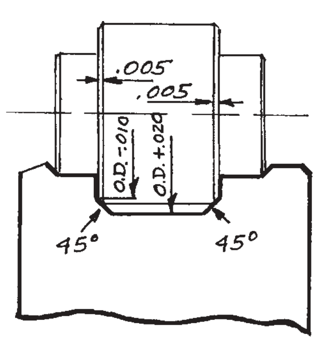


Figure #17

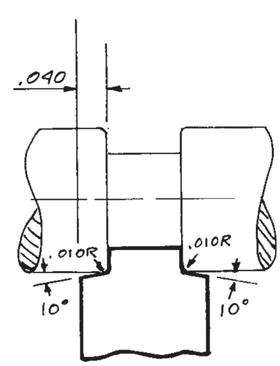


Figure #18

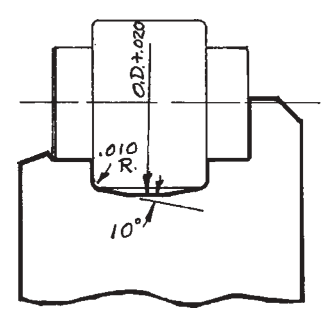


Figure #19

CLEARING HEXAGON, SQUARE, PINION OR OTHER SHAPE STOCK

CHAMFER CORNERS:

If a chamfer is shown but no particular chamfer diameter is specified, the chamfer should start at a diameter equal to about 95% of the flat dimension of the hex or square or of the root diameter of the pinion. If an angle is shown but not specified, it can usually be made at 15 to 45 degrees from the vertical depending on the application or the appearance required. See Figure #20.

If the tool is forming the angle on only one side of the stock, the angle should continue out to a width .020” (for small parts) to .050” (large parts) wider than the actual width of the chamfer on the part. See Figure #20.

If tool is straddling the stock and chamfering both ends of the stock, the angles should be continued out to a diameter that is at least .020” greater than the cross corner dimension of the stock and joined with a horizontal surface so as to clear the corners of the stock when it is rotating. See Figure #21.

Note: If no chamfers are specified, it is always desirable to put a 1 to 2 degree positive side clearance on any vertical surfaces adjacent to the stock to minimize burrs.

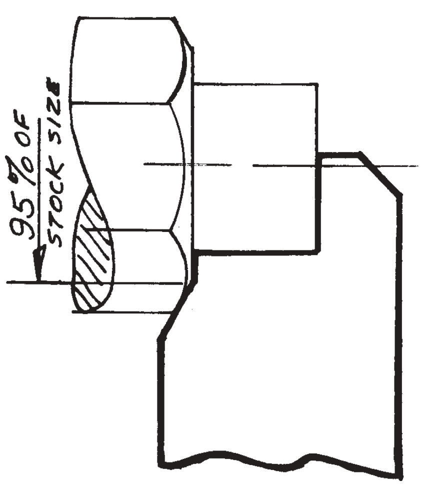


Figure #20

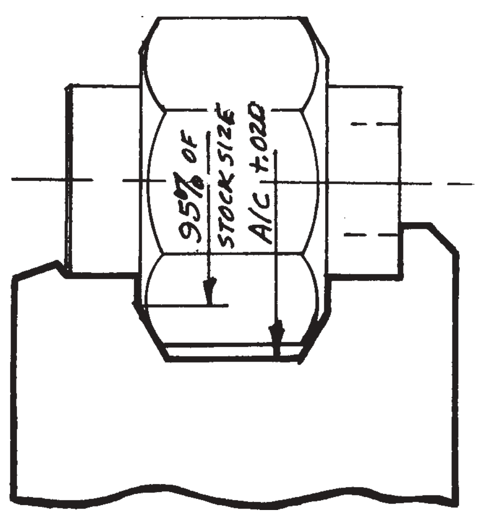


Figure #21

SIDE CLEARANCE

POSITIVE (FRONT TO BACK) SIDE CLEARANCE:

Side clearance (also called “undercut”) of 1 to 2 degrees is recommended for all vertical surfaces .090” or deeper for all tools whether flat or circular to eliminate side rubbing and wear of the tool.

CHAMFER OUTER WORKPIECE CORNER — ANGLE — RADIUS:

If step is deep or clearance is 2 degrees or greater, the normal round of the grinding wheel will be buried in the clearance causing the outer corner of the part to be sharp and unchamfered. To avoid the possibility of developing a burr on this corner, an angle is specified at the bottom of the clearance that extends outside of the undercut .005” x 45 degrees (or whatever size and angle that is specified). See Figure #22.

If a radius is required on the outer corner, that radius is specified so that the outer edge of the radius is in line with the leading edge of the undercut and a tangent angle of 5 to 10 degrees runs off it extending into the undercut and intersecting with the clearance angle. See Figure #23.

NEGATIVE CLEARANCE:

If the part application or tolerances permit it, a negative side clearance (taper) of 1 to 5 degrees on all vertical surfaces will eliminate side rubbing, improve part finish and increase tool life. See Figure #24.

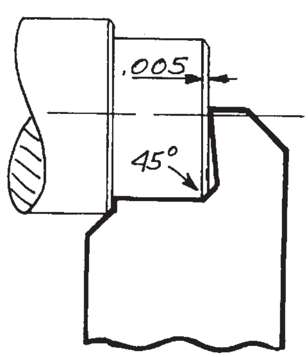


Figure #22

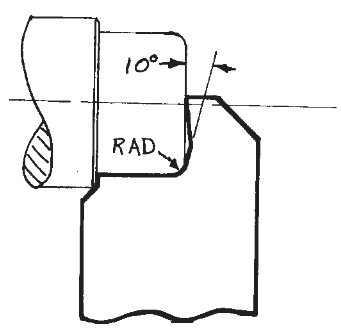


Figure #23

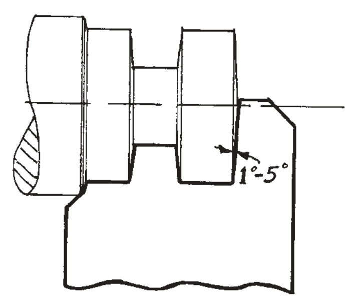


Figure #24

TOP TO BOTTOM TAPERED CLEARANCE:

Top to bottom tapered side clearance can be helpful on flat type tools when side rubbing is a severe problem. See Wire EDMing for top to bottom clearance on page 39. The usual rule is to specify a taper on all side surfaces that is equal to the allowable tolerance on the part. Ex: If you are cutting a groove .100 +/-.003", an equal taper would be ground on both sides of the rib of the tool so that it would be .1025" at the top of the cutting edge and taper down to .0975" at the smallest usable point of the tool after resharpenings. This way as the tool is resharpened, it will be within the allowable tolerances at all times. The same principle applies if forming a rib on the part, except in reverse, that is the opening in the tool would be .0975" at the top and .1025" at the smallest usable point. See Figure #25.

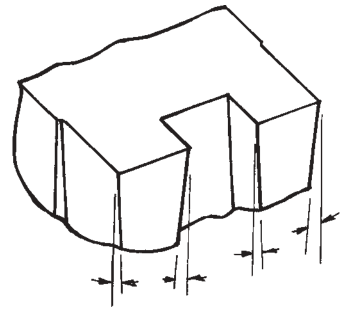


Figure #25

Note: We do not recommend this type tool since it makes the tool much more expensive to manufacture, reduces usable tool life and produces parts with inconsistent width sizes. Excessive side rubbing may be caused by the fact that the tool post is not in perfect perpendicular alignment with the spindle of the machine. This is especially true if rubbing is worse on one side than the other.

PREPARING PART FOR TURRET TURNING TOOL

FRONT END CHAMFER:

The front end of a part that is to be turned with a turning tool from the turret should always be chamfered with an angle (preferably 45 degrees) starting at a smaller diameter than the turned diameter and extending past the next larger diameter. If the chamfer does not extend to the next diameter and leaves a vertical face on the part, it is very likely that when the turning tool hits this face it will waver slightly and produce a dig in, visible ring or a noticeable difference in finish and dimension of the turned portion. Furthermore, by extending the chamfer below the diameter to be turned, it allows the turning tool to center and stabilize itself on the part thereby eliminating the waver that would be caused by the tool coming in contact with a vertical blunt surface. See Figure #26.

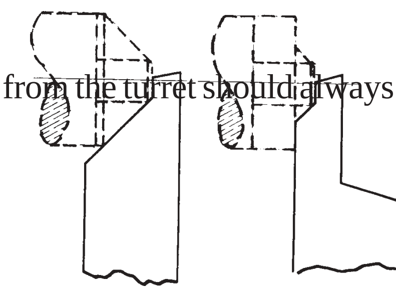


Figure #26

If a radius is specified at the start of the turned diameter, a tangent angle of 10 degrees should be run off the radius for .030" to .060" past the centerline of the radius and then run off with a 45 degree angle to the next diameter as explained for angular chamfer. See Figure #27.

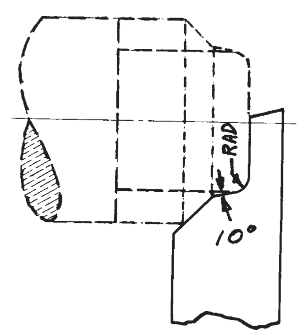


Figure #27

BLENDING WITH SURFACE TURNED BY ANOTHER TOOL

BLEND WITH TURNING TOOL OR HORIZONTAL SURFACE:

Add a shoulder to the tool that forms down to the diameter already turned by the turning tool or another tool. This shoulder should be approximately twice the width of the cutoff tool being used and should be half flat and the outboard half should be at 10 degrees. See Figure #28.

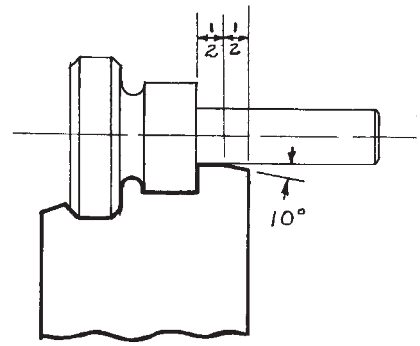


Figure #28

BLEND WITH ANGULAR SURFACE:

Add a shoulder as previously described except that half of it should follow the angle of the part and the balance should be at an angle that is 10 degrees greater.

FORMING BLANK FOR MALE CUT THREAD

If thread blank dimensions are not specified, the following recommendations will produce good threads and eliminate setup and tool adjustment problems.

THREAD DIAMETER:

Form to mean of thread O.D. diameter tolerance as per ASA standards for class thread desired.

THREAD CHAMFER:

Chamfer front end of thread at 45 degree angle to a depth that is 20-25% deeper than the single depth of the thread. This will make it easy for the die to start on and assure that there are no burrs or flags at the start of the thread. See Figure #29.

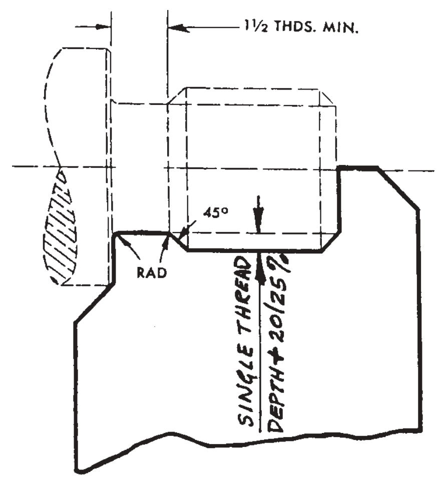


Figure #29

BACK END UNDERCUT:

If back end of the thread is undercut but dimensions are not specified, tool should be designed with an undercut that is 1-1/2 threads wide minimum and to a diameter that is 20-25% deeper than the single depth of the thread. This undercut should run up to the thread O.D. at 45 degrees towards the front end of the part and the bottom corners of the undercut should be rounded with a radius that is about 20-25% of the thread width. This will make it easy to set up the threading die since the 1-1/2 thread width plus the 45 degree angle gives the operator enough leeway to stop and reverse the threading die (which usually has about a 1 to 1-1/2 thread lead) and still get a full thread for the entire length of the thread. Also the radii will strengthen the thread when it is in use. See Figure #29.

ANGULAR OR RADIUS FACE MEETING THREAD:

If part has either an angular or radius face extending up to the threaded diameter, that angle or radius should only extend to the bottom diameter of the chamfer as determined per previous rule. This will make it easy for the die to start on and assure that there are no burrs or flags at the start of the thread. See Figure #30.

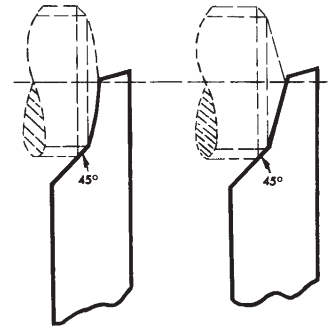


Figure #30

FORMING BLANK FOR ROLLED THREAD

If thread blank dimensions are not specified, the following recommendations will produce good threads and simplify setup.

THREAD DIAMETER:

Form thread O.D. to mean of the pitch diameter tolerance as per ASA standards for class of thread desired. Since the pitch diameter is half way down the thread form and since the roll displaces metal from the bottom of the thread and pushes it upward by the same amount, the thread O.D. will increase to its prescribed diameter. In actual practice, this blank diameter may have to be varied a few thousandths because of the way the material being threaded may react. This is usually not a problem unless the tool is forming other critical diameters that will be thrown out of tolerance by this variation. See Figure #31 .

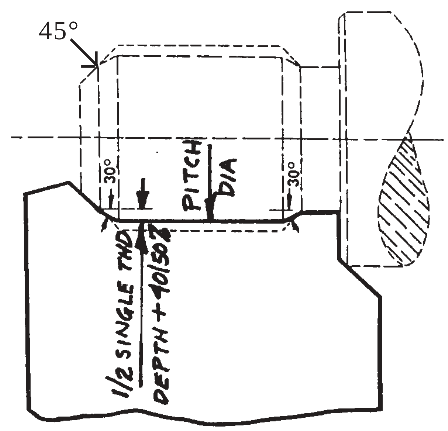


Figure #31

THREAD CHAMFER:

Chamfer starting end of thread blank at an angle of 30 degrees from horizontal to a depth that is 40-50% deeper than half the single depth of the thread. This angle will roll up to approximately 45 degrees and assure that there are no burrs or flags at the start of the thread. See Figure #31.

This rule also applies to the back end of the thread if an undercut is specified, except that the width of the undercut is not critical because, unlike a threading die, the thread roll can thread almost up against the shoulder.

BACK END NOT UNDERCUT:

If back end of a roll threaded part is unthreaded, the tool must be designed so that it forms the unthreaded portion to the length and diameter specified and then stepped down to the pitch diameter at an angle of 30 degrees from vertical (matching the flank of the thread). Thread rolls should be lined up as close as possible to this flank, thereby, producing a complete thread. See Figure #32.

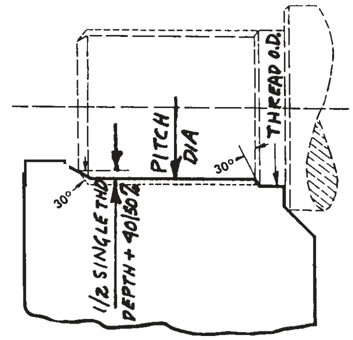


Figure #32

ANGULAR OR RADIUS FACE MEETING THREAD:

If part has either an angular face extending up to the threaded diameter, that angle or radius should only extend to the bottom diameter of the chamfer as determined per previous rule except angle should be 30 degrees from horizontal insead of 45 degrees. This will make it easy for the roll to start on and assure that there are no burrs or flags at the start of the the thread.

PROBLEMS ENCOUNTERED IN ROLL THREADING:

Elongation of the threaded portion of the part due to the flow of the metal caused by the rolling operation.

Closing in of holes on thin walled parts.

Variations in thread size caused by variations in the thread blank diameter due to tool wear or machine problems.

FORMING BLANK FOR A KNURLED SURFACE

BLANK DIAMETER:

If a diameter is specified over the knurl, the blank should be formed to the pitch diameter of the knurl or the diameter as specified by the knurl manufacturer, and chamfered at a 45 degree angle down to a depth that is 40-50% deeper than half the single depth of the knurl teeth. This will allow the knurl to start on easily and produce burr free parts. Since the knurl rolls displace metal from the bottom of the tooth and push it upward by the same amount, the knurl O.D. will increase to its prescribed diameter. See Figure #33.

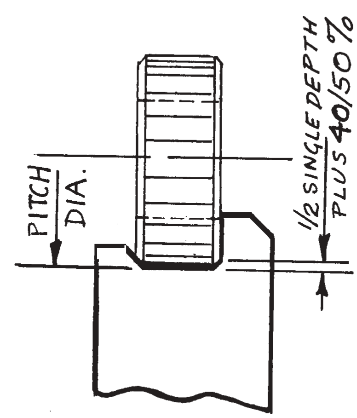


Figure #33

If no diameter is given over the knurl and it is being formed on the stock diameter or a previously turned diameter, it is still important to chamfer the corners as per previous instructions.

If a close tolerance is specified for the knurled diameter, it may be necessary to turn it with the form tool *after* the knurling operation.

FORMING BLANK FOR CENTERLESS GROUND PART

When one or more diameters of a part must be held to closer tolerances than can be attained on the screw machine, it is usually specified that those diameters be centerless ground to size in a secondary operation. The standard practice is to leave the diameter .010/.015" oversize, but in doing so it is important to design the tool so that all other dimensions of the part are to their finished dimensions and that any chamfers, angles or radii on the diameter being ground will be to size after the .010/.015" is removed in the grinding operation.

CHAMFERS AND ANGLES:

If a chamfer or angle is specified, it must be made large enough so that after the diameter is ground to size it will be reduced to its actual specified dimension. If no chamfer is specified a 45 degree angle that extends .005" on the side (.010" total) below the finished diameter should be specified so as to avoid burrs on the corners after grinding. See Figure #34.

CORNER RADII:

If a radius is specified on the diameter being ground, the centerline of the radius should be located so that the radius will be tangent to the finished diameter after the grinding operation and should run off with an angle of 10 degrees up to the pregrinding formed diameter. See Figure #34.

GROUND DIAMETER ADJACENT TO A SHOULDER:

If the surface being ground intersects with a vertical face, this inside corner should be undercut .030/.060" wide x .005/.010 deep to allow for rounding of the corner of the grinding wheel to assure that the corner will be sharp after grinding. See Figure #35.

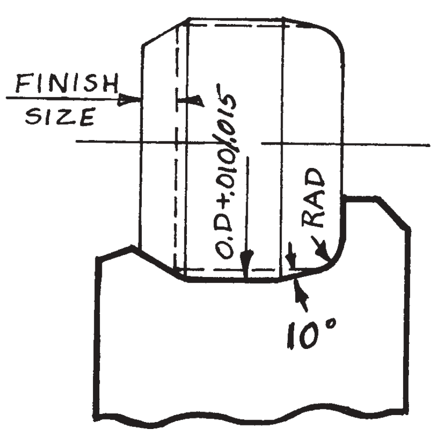


Figure #34

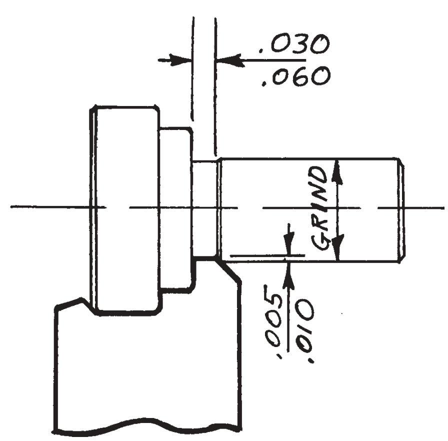


Figure #35

DESIGNING "V" GROOVES OR PROJECTIONS

"V" SHAPED PROJECTIONS ON THE PART:

"V" shaped projections on the part, made up of angles, radii, or a combination of both are very often specified. They are usually dimensioned to the theoretical sharp corner intersection points.

However, it is impossible to produce by conventional grinding or EDM machining an internal "V" form on the tool to this theoretical sharp corner. The sharpest corner that can be produced by these methods is a .005" - .007" radius.

Therefore, as shown in Figure 36 if all width, angle, or radius dimensions are held as specified, the diameter to the actual point of the part will be reduced by the amounts as shown. If this is not acceptable, it will be necessary to change one or more of the other dimensions that are critical to the function of the part.

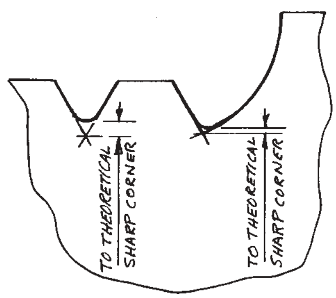


Figure #36

Furthermore, regardless of how any part is dimensioned, the tool must always be calculated to allow for this minimum corner radius so as to be certain that all dimensions of the tool will be attainable.

"V" SHAPED GROOVES INTO THE PART:

"V" shaped grooves into the part can be made to the theoretical sharp corner dimension, however, if the part tolerances permit, it is recommended that a flat of .003"-.005" be specified at the point. The advantage of this flat is that it will not wear out as quickly as a sharp corner and will, therefore, increase tool life and improve part appearance. See Figure 37.

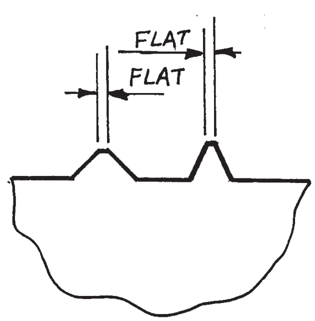


Figure #37

SECTION 2

DRAWING, CALCULATING AND DIMENSIONING RESHARPENABLE QUICK CHANGE FORM TOOL INSERTS

First review the part print in relation to Section 1. “Check List of Things to Determine Before Starting to Draw Tool”.

Next determine the mean size of all diameter and length intersection points of the portion of the part to be formed. If some diameter or length intersection points are not specified but are determined by an angle, a radius, or a combination of both, these points must be calculated before proceeding. Unknown dimensions of triangles and radii can be easily calculated with the aid of the “FTSC” computer program.

Mark all these diameters and individual widths either on the part print or make a rough sketch of part. Make sure that all widths add up to any total widths shown and that all diameters and widths are logical proportionally. Also, specify the dimensions and configuration of the marker, corner break, front chamfer, cutoff and faceoff portions of the tool, if any.

Predrawn "Insert Tool Design" sheets for the size insert being used are available from Somma. These sheets simplify tool design since they eliminate drawing the basic dimensions of the insert and only require filling in of the formed portion of the tool. They are based on an enlargement scale of 4” = 1”. (Part form is drawn at 4 times its actual size.) This enlargement is usually adequate to clearly be able to distinguish the outline of the form, however, if the part is very small or very intricate, it may be necessary to draw a detail of the form at a larger scale on a separate sheet. Tool should always be drawn to scale as accurately as possible since it will visually show any miscalculations that may have been made.

Next, we must determine the “Base Diameter” of the part. The “Base Diameter” is the smallest diameter being formed by the tool. Following are examples of various “Base Diameter” calculations:

Operation	Base Diameter
Forming past center	Minus 2 times the past center distance. See Figure 1.
Cutting off	Minus 2 times the total of the past center distance and 2 times the depth of the cutoff angle. See Figure 2.

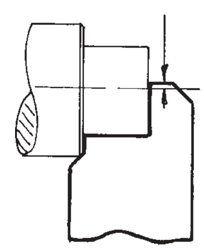


Figure #1

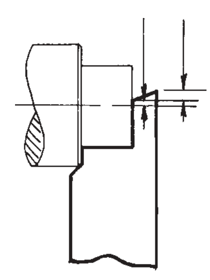


Figure #2

Operation	Base Diameter
Facing off into the hole	Hole diameter minus two times the distance going past the hole. See Figure 3.
Forming a diameter	Smallest diameter being formed. See Figure 4.
Cutoff marker on smallest diameter	Diameter minus 2 times depth of marker projection.See Figure 5.
Front end chamfer on smallest diameter	Diameter minus 2 times depth of chamfer projection. See Figure 6.

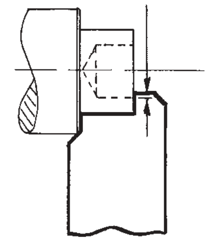


Figure #3

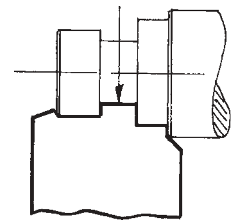


Figure #4

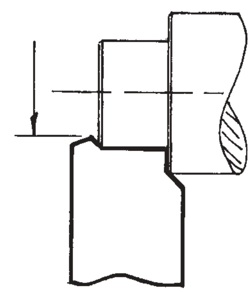


Figure #5

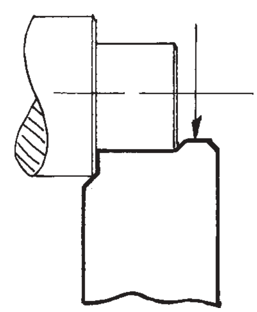
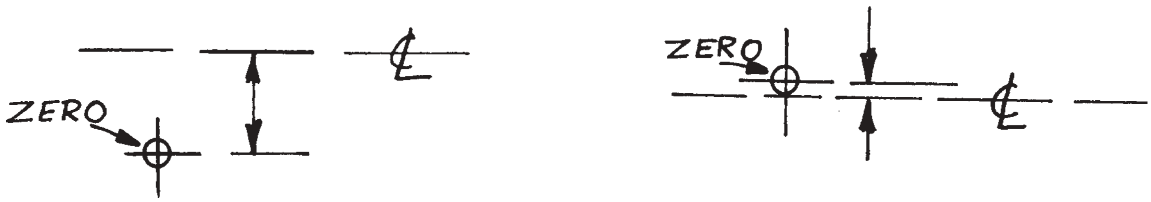


Figure #6

"RQC Style" inserts should be made on wire EDM machines to guarantee repeatability from insert to insert and from lot to lot. Therefore, to simplify EDM programming, all dimensions should be given from a "Zero" point in the forward left hand corner of the insert. See sample design on page 40.

Once we determine the base diameter, we draw the horizontal centerline of the part in relationship to the zero point of the insert. This distance to center will be drawn at 4x the base diameter divided by two (because it is a radius from the centerline). To simplify this calculation, merely draw it at two times its actual size.

The centerline will therefore be drawn at 2x the base diameter above the zero point if it is a positive number, or 2x the base diameter below the zero point if it is a minus number. Draw this line lightly since it is for reference only. See Figure #7.



POSITIVE NUMBER NEGATIVE NUMBER

Figure #7

Next, starting at the left end of the insert (zero point), mark off along this centerline all the width points that we calculated previously, at a scale of 4x their actual width. Then draw vertical lines lightly down from each of these points for an inch or two.

Next, draw all the intersecting diameter points on the appropriate vertical lines. These points are drawn at 2x part diameter from the centerline as explained previously. Next, connect all these points with horizontal lines, vertical lines, radii or angles as required to develop the outline of the tool form. When finished darken the lines that outline the form and erase those portions of the original construction lines that are no longer needed.

Next, draw horizontal and vertical dimension lines for each width and diameter as shown on sample tool design. *Note that all length dimensions are from the zero point and that they are cumulative and not individual. Also note that widths dimensions never change and are exactly as determined on our original part print or sketch.*

However, the vertical step differentials of form tools are not the same on the tool as the step differentials of the part. This is due to the effects of the geometry of the front clearance angle and top rake angle on the length of the steps. See Section 6 “Form Tool Geometry”. These step differentials must be corrected and are easily calculated with the “FTSC’-Form Tool Step Correction” program. (Pg. 43)

To simplify these calculations and to have a permanent record of diameters used, write in the base diameter and part diameters in small figures and in parenthesis below the vertical dimension lines. See sample tool design. Now using the “FTSC” program, calculate the corrected step dimensions for every one of the diameter intersection points and fill them in on the corresponding vertical dimension lines. Although these corrected step differentials will affect any angles on the part, it is not necessary to specify the corrected angles since they will come out right on the part when the intersection points are joined in the wire EDM operation.

If part has radii specified, they usually do not have to be corrected unless they are held to a close tolerance (+/-.001”). If they are only partial chords, however, the radius has to be recalculated to meet the corrected intersection points. See section on “Correcting Angles and Radii” (Pg. 41).

If part has a contour consisting of a combination of two or more radii or radii and angles see section on “Calculating Complex Angle and Radius Contours” (Pg. 42).

If form does not take up full width of insert, clear the tool away on an angle away from the part so that is does not interfere with other tools in the setup. *Note that cleared surface should never go any deeper than the maximum depth of cut specified for the size insert being used*. See sample tool design.

DESIGNING INSERTS FOR A “FAMILY” OF PARTS ("DATUM POINT" TOOLING)

One of the advantages of “RQC” resharpenable quick change inserts is that when making a “family” of parts, the inserts can be designed so that when changing from job to job, diameter and lengthwise dimensions will be maintained by merely changing the insert, without moving the holder or the slide. This is known as "Datum Point" tooling.

When designing the insert, the “Datum Point” can be either the face of the part left in the collet after cutting off (Figure 7A) or the outermost face of the completed part before it is cut off (Figure 7B). Also, all length dimensions must be calculated in relation to these points.

Since usually end working tools have to be moved because of differences in the configuration of the operations being performed, as shown in Figure 7A, it is best to use the face of the part left in the collet after cutting off as the “Datum Point”. This eliminates the necessity of moving the cutoff tool.

However, if all the end working operations are of the same configuration and dimensions, as in Figure 7B, it is better to use the outboard face of the completed part as the “Datum Point.” In this case, the only tool that has to be moved is the cutoff tool.

The inserts must be calculated so that the diameter “Datum Point” from which all diameters are calculated is the smallest diameter being formed on any of the parts in the family. This diameter must be used as the “base diameter” when calculating the inserts for all of the parts in the family. It is important that center height is established with the insert that is forming this smallest part diameter.

Note that in all cases where the smallest diameter of each part is different, only one insert is the master length and the other inserts are shorter corresponding to this difference.

In the examples shown in Figure 7A, the front end operations are different on each of the three parts. Therefore, since the end working tools will have to be changed, the face of the next part is used at the Datum Point so that the cutoff tool will not have to be moved.

In the examples shown in Figure 7B, the front end operations are the same on all the parts. In this case, it is best to use the outboard face of the piece as the “Datum Point” thereby only necessitating moving the cutoff tool. Since this job was being done on a multiple spindle machine, a third “Datum Point” tool was able to be added that produced a partial cutoff and marker groove against which the cutoff tool could easily be aligned.

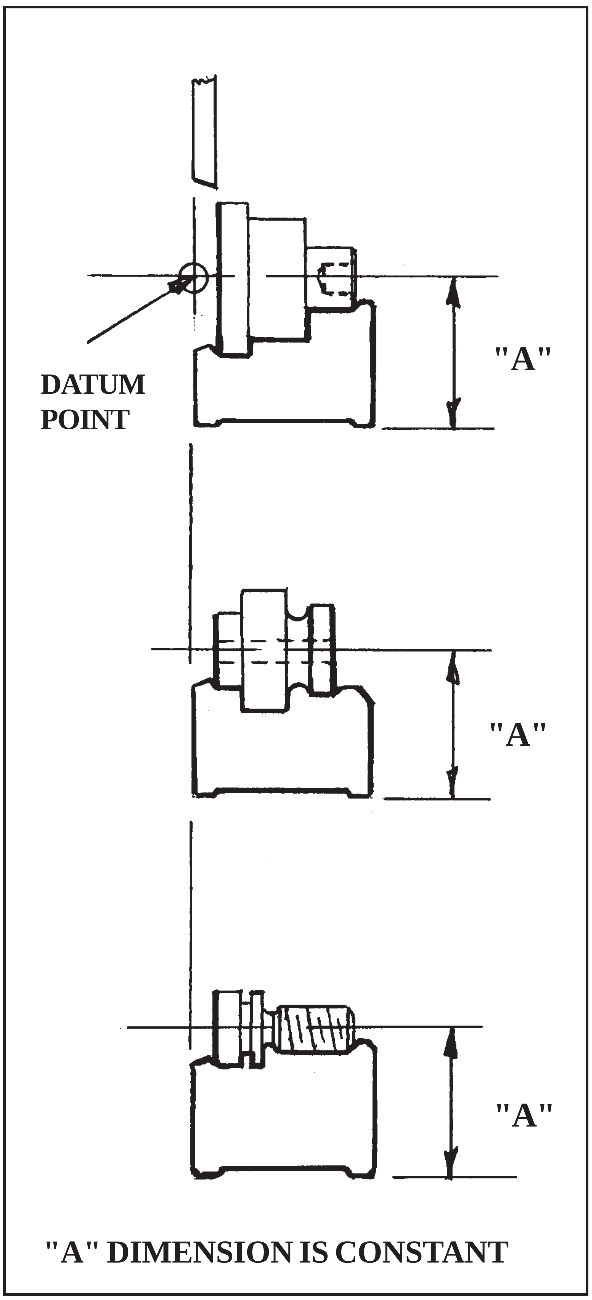


Figure 7A

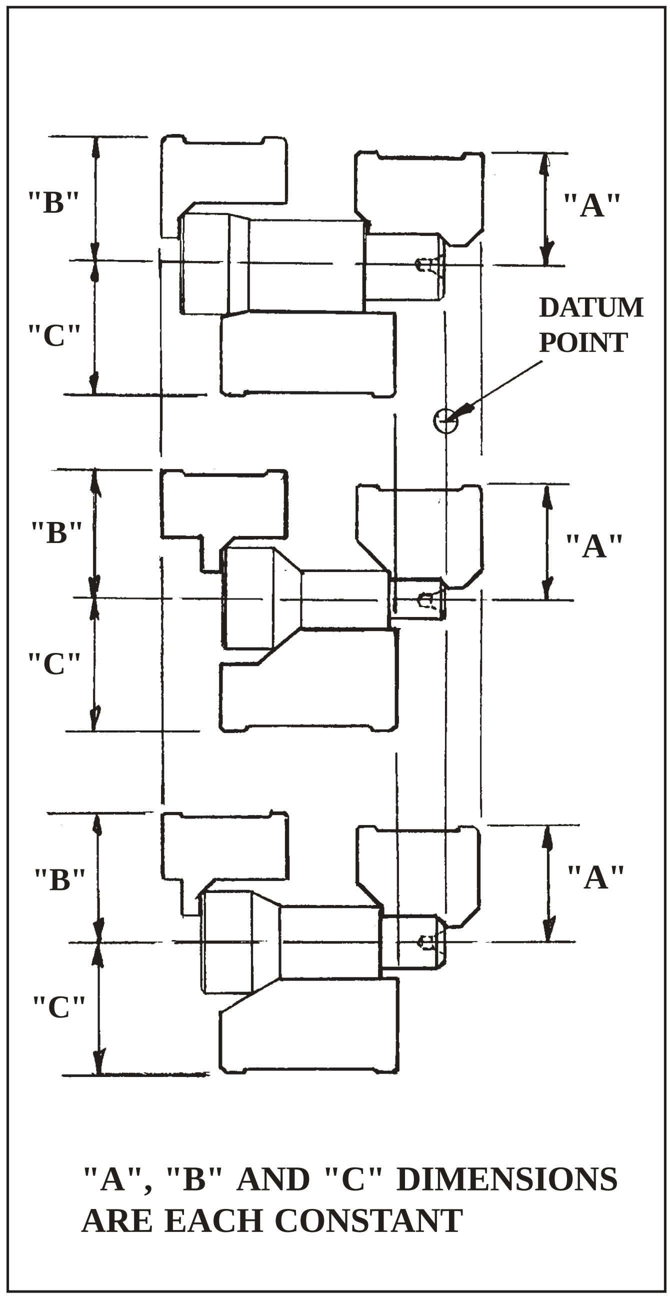


Figure 7B

DRAWING, CALCULATING AND DIMENSIONING CIRCULAR FORM TOOLS

First review the part print in relation to Section 1, “Check List of Things to Determine Before Starting to Draw Tool”.

Next, determine the mean size of all diameter and length intersection points of the portion of the part to be formed. If some diameter or length intersection points are not specified, but are determined by an angle, a radius, or a combination of both, these points must be calculated before proceeding. Unknown dimensions of triangles and radii can be easily calculated with the aid of the “FTSC” computer program.

Mark all these diameters and individual widths either on the part print or make a rough sketch of part. Make sure that all widths add up to any total widths shown and that all diameters are logical proportionally. Also specify the width and configuration of the marker, corner break, front chamfer, cutoff and faceoff portions of the tool, if any.

If you do not have a prescribed design sheet, “Circular Form Tool Design" sheets are available from Somma. These sheets simplify tool designing since the basic design of the circular blank is predrawn and only requires filling in of the blank dimensions and the formed portion of the tool. The design of the form portion of the tool can usually be drawn at an enlargement scale of 4” = 1”. (Form is drawn at 4 times its actual size.) This enlargement is usually adequate to clearly be able to distinguish the outline of the form, however, if the part is very small or very intricate it may be necessary to draw a detail of the tool at a larger scale on a separate sheet. Tool should always be drawn to scale as accurately as possible since it will visually show any miscalculations that may have been made.

Next, we must determine the “Base Diameter” of the part. The “Base Diameter” is the smallest diameter being formed by the tool. See pages 27 & 28 for examples of various “Base Diameter” calculations:

Once we determine the base diameter, we draw the horizontal centerline of the part in relationship to the major diameter of the tool at 4” = 1” scale. This distance to center will be drawn at 4x the base diameter divided by 2 (because it is a radius from the centerline.) To simplify this calculation merely draw it at two times its actual size.

The centerline will therefore be drawn at 2x the base diameter above the major tool diameter if it is a positive number or 2x below the major tool diameter if it is a minus number. We should draw this line lightly since it is for reference only. See Figure #7.

POSITIVE NUMBER **Figure #7** NEGATIVE NUMBER

Next, starting at the left end of the tool, mark off along this centerline all the width points that we calculated previously at a scale of 4x their actual width. Then draw vertical lines lightly down from each of these points for an inch or two. Next draw all the intersecting diameter points on the appropriate vertical lines. These points are drawn at 2x actual diameter from the centerline as explained previously. Next connect all these points with horizontal lines, vertical lines, radii or angles as required to develop the outline of the tool form. When finished, darken the lines that outline the form and erase those portions of the original construction lines that are no longer needed.

Next, draw horizontal and vertical dimension lines for each width and diameter as shown on sample tool design. *Note that all length dimensions should be shown individually. Also note that width dimensions never change and are exactly as determined on our original part print or sketch.*

All vertical step differentials should be shown as tool diameters since circular tools are easiest checked with micrometers. However, the diameter differentials are not the same on the tool as the diameter differentials of the part. This is due to the effects of the offset of the cutting edge of the tool from center and the top rake angle. See Section 6 “Form Tool Geometry”. These corrected tool diameters are easily calculated with our “FTSC-Form Tool Step Correction” program. (Pg. 43).

To simplify these calculations and to have a permanent record of diameters used, write in the base diameter and the part diameters in small figures and in parentheses below the tool diameter dimension lines. See sample tool design. Now using the “FTSC” program, calculate the corrected tool diameters for every one of the diameter intersection points and fill them in on the corresponding tool diameter dimension lines. Since these corrected step differentials will affect any angles on the part, the angles will have to be recalculated and specified accordingly.

If part has radii specified, they usually do not have to be corrected unless they are held to a very close tolerance (+/-.001”). If they are only partial chords, however, the radius has to be recalculated to meet the corrected intersection points. See section on “Correcting Angles and Radii.” (Pg. 41).

If part has a contour consisting of a combination of two or more radii and angles, see section on “Calculating Complex Angle and Radius Contours” (Pg. 42).

If form does not take up full width of the tool blank, clear the tool on an angle away from the part so that it does not interfere with other tools in the setup.

DRAWING, CALCULATING AND DIMENSIONING DOVETAIL FORM TOOLS

First review the part print in relation to Section 1, “Check List of Things to Determine Before Starting to Draw Tool”.

Next, determine the mean size of all diameter and length intersection points of the portion of the part to be formed. If some diameter or length intersection points are not specified but are determined by an angle, a radius, or a combination of both, these points must be calculated before proceeding. Unknown dimensions of triangles and radii can be easily calculated with the aid of the “FTSC” computer program.

Mark all these diameters and individual widths either on the part print or make a rough sketch of part. Make sure that all widths add up to any total widths shown and that all diameters are logical proportionally. Also specify the dimensions and configuration of marker, corner break, front chamfer, cutoff and faceoff portions of the tool, if any.

If you do not have a prescribed design sheet to use, “Dovetail Tool Design" sheets are available from Somma. These sheets simplify tool designing since the basic design of the dovetail blank is predrawn and only require filling in of the dovetail dimensions and the formed portion of the tool. The design of the form portion of the tool can usually be drawn at an enlargement scale of 4” = 1”. (form is drawn at 4 times its actual size.) This enlargement is usually adequate to clearly be able to distinguish the outline of the form, however, if the part is very small or very intricate it may be necessary to draw a detail of the tool at a larger scale on a separate sheet. Tool should always be drawn to scale as accurately as possible since it will visually show any miscalculations that may have been made.

Next, we must determine the “Base Diameter” of the part. The “Base Diameter” is the smallest diameter being formed by the tool. See pages 27 & 28 for examples of various “Base Diameter” calculations.

If the form is being cut on a wire EDM machine, all dimensions should be given from a “Zero” point in the forward left hand corner of the tool design to simplify programming. This “Zero” point should be specified in relation to the theoretical sharp outer corner of the dovetail angle at the base of the blank on the side closest to the spindle.

Once we determine the base diameter, we draw the horizontal centerline of the part in relationship to the zero point of the tool. This distance to center will be drawn at 4x the base diameter divided by two (because it is a radius from the centerline). To simplify this calculation, merely draw it at two times its actual size.

The centerline will therefore be drawn at 2x the base diameter above the zero point if it is a positive number, or 2x the base diameter below the zero point if it is a minus number. Draw this line lightly since it is for reference only. See Figure #7.

POSITIVE NUMBER **Figure #7** NEGATIVE NUMBER

Next, starting at the left end of the tool (zero point), mark off along this centerline all the width points that we calculated previously at a scale of 4x their actual width. Then draw vertical lines lightly down from each of these points for an inch or two.

Next, draw all the intersecting diameter points on the appropriate vertical lines. These points are drawn at 2x part diameter from the centerline as explained previously. Next connect all these points with horizontal lines, vertical lines, radii or angles as required to develop the outline of the tool form. When finished, darken the lines that outline the form and erase those portions of the original construction lines that are no longer needed.

Next, draw horizontal and vertical dimension lines for each width and diameter as shown on sample tool design. *Note that all length dimensions are from the zero point and that they are cumulative and not individual. Also note, that width dimensions never change and are exactly as determined on our original part print or sketch.*

However, the vertical step differentials are not the same on the tool as the step differentials of the part. This is due to the effects of the geometry of the front clearance angle and top rake angle on the length of the steps. See Section 6 “Form Tool Geometry”. These step differentials must be corrected and are easily calculated with the “FTSC-Form Tool Step Correction” program. (See Pg. 43).

To simplify these calculations and to have a permanent record of diameters used, write in the base diameter and part diameters in small figures and in parenthesis below the vertical dimension lines. See sample tool design. Now using the “FTSC” program calculate the corrected step dimensions for every one of the diameter intersection points and fill them in on the corresponding vertical dimension lines. Although these corrected step differentials will affect any angles on the part, it is not necessary to specify the corrected angles since they will come out right on the part when the intersection points are joined in the wire EDM operation.

If part has radii specified, they usually do not have to be corrected unless they are held to a close tolerance (+/-.001”). If they are only partial chords, however, the radius has to be recalculated to meet the corrected intersection points. See section on “Correcting Angles and Radii.” (Pg. 41).

If part has a contour consisting of a combination of two or more radii or radii and angles, see section on “Calculating Complex Angle and Radius Contours” (Pg. 42).

If form does not take up full width of the tool blank, clear the tool on an angle away from the part so that it does not interfere with other tools in the setup.

DESIGNING SHAVE TOOLS

The form on all shave tools, whether they are dovetail, circular or insert type has to be located in relation to the centerline of the holder it is being mounted in. This is because the centerline of the holder is on the same centerline as the workpiece and this distance is a "constant". This means that as the diameter of the workpiece gets larger, the height (or the diameter) of the shave tool gets smaller.

Note: No correction is required for part diameter differentials on insert or dovetail type shave tools.

SOMMA INSERT TYPE HOLDERS: All step dimensions must be maintained in relation to the base diameters and master insert height and calculated as follows:

CAT. NO.	MASTER HEIGHT	BASE DIAMETER	TOP RAKE
RQCDS-A RQCDS-SL RQCDS-DV RQCMS-0	.875	.000	0 Deg.
RQCMS-1	.875	.125	0 Deg.
RQCMS-2	1.125	.125	0 Deg.
RQCMS-3	1.125	.500	0 Deg.

DOVETAIL TYPE SHAVE TOOL HOLDERS:

The height of the tool is determined by subtracting one half the diameter being formed from the constant, which is the distance from the base of the dovetail to the centerline of the shank. See Figure 8A showing a holder with a constant dimension of 1.125". ***Note: Constant varies according to the holder size.***

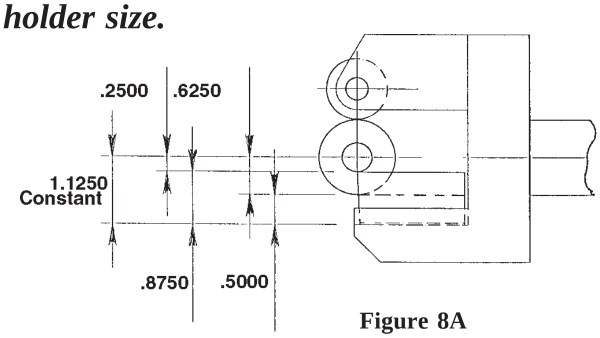


Figure 8A

DAVENPORT CIRCULAR SHAVE TOOLS: All diameters are based on constant "A" and must be corrected using the following specifications:

	MASTER DIAMETER	OFFSET	TOP RAKE	BASE DIAMETER	CONSTANTS
REGULAR HOLDER					
Shave Tool	1.3012	.062	0 Deg.	.075	"A" = 11/16"
Roll	.925	.000	0 Deg.	.075	"B" = 1/2"
OVERSIZE HOLDER					
Shave Tool	1.3012	.062	0 Deg.	.262	"A" = 25/32"
Roll	.925	.000	0 Deg.	.262	"B" = 19/32"

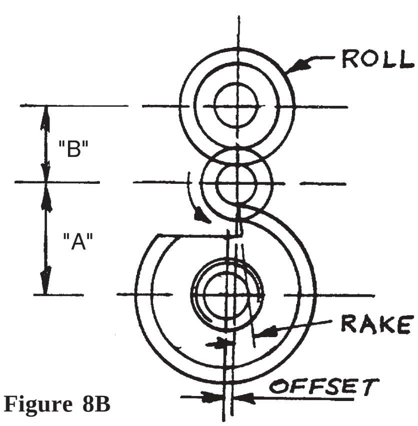
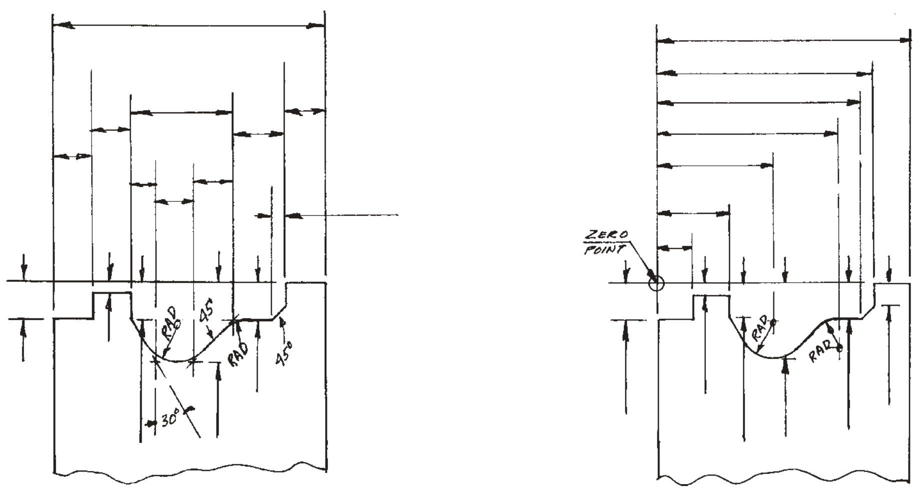


Figure 8B

Note: All roll diameters are based on constant "B" and must be calculated using the above specifications. (See Figure 8B).

DIMENSIONING TOOLS FOR CONVENTIONAL GRINDING

Follow the procedures as previously explained for wire EDM machining of inserts except that width dimensions are given individually instead of cumulatively, and intersection points of angles and radii must be specified. Also, because step differential corrections will affect any angles on the part, angles will have to be recalculated and specified accordingly. Figure #8 illustrates this difference in dimensioning of tool designs for conventional grinding and wire EDM machining.



CONVENTIONAL GRINDING **Figure #8** **WIRE "EDM" MACHINING**

CAD-CAM DESIGNING OF FORM TOOLS

Design the complete workpiece on the computer orienting it in the direction it is being fed out of the machine with the spindle on the left. Enter all dimensions, (including angles and radii) in the same format as the print except using the mean of the tolerance for all dimensions.

Important: Make a copy of this design and store original for future reference.

To Design Form Tool:

1. Delete all dimensions from copy of workpiece design.
2. Change color and design form tool outline along portion of workpiece being formed, adding marker, faceoff, cutoff, chamfers, side clearance, etc. as needed.
3. Delete original workpiece outline leaving only form tool outline.
4. Design and dimension balance of form tool configuration, such as insert, dovetail, circular.
5. For wire EDMing, using the farthest point to the left as the zero point, enter by computer the width dimensions to all intersecting points in relation to this zero. (See Figure 8).
6. For conventional grinding, enter by computer all width dimensions individually instead of cumulatively. (See Figure 8.) All intersection points of angles and radii must be specified and corrected for step differential.
7. Enter all step depths in relation to the highest point of the form. Depths must be corrected in accordance with the geometry of the type tool being designed.

If your Cad-Cam does not have a Form Tool Step Correction program, step depths can be calculated by projecting the workpiece to the front end view, superimposing the form tool geometry configuration over it in its correct relation to the workpiece and then measuring by computer the step distances to the intersection of the workpiece and the cutting edge of the form tool. See Figures 8C with sample dimensions.

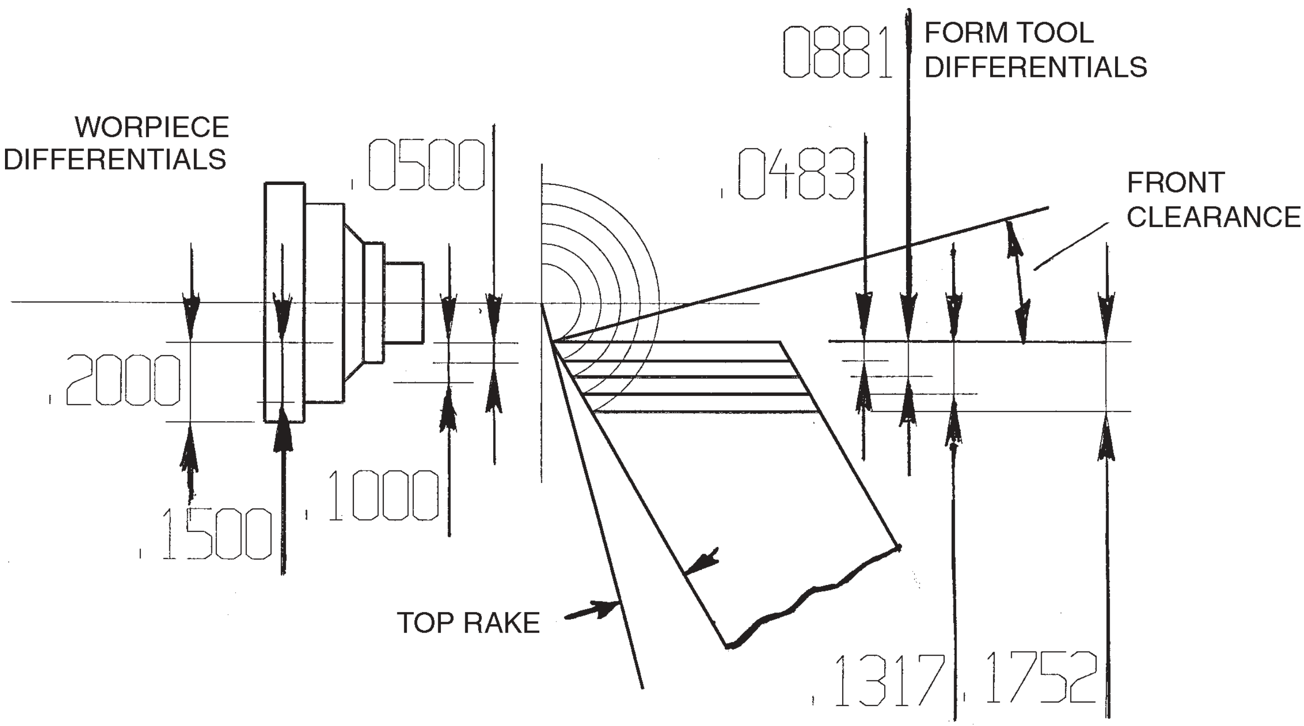


Figure #8C

Step corrections can also be calculated either manually or with the Somma "FTSC" Computer Form Tool Step Correction Program. (Page 43).

Note: *Shave tools, whether insert type or dovetail, do not require correction of step depths. this also applies to skive tools.*

For Davenport Circular Shave Tools corrections, see Page 36.

DESIGNING RQC INSERTS FOR WIRE EDM MACHINING OF TOP TO BOTTOM CLEARANCE

Design insert as per previous instructions except instead of using mean size for all width dimensions, use the largest dimension allowable within the specified tolerance for all groove widths on the workpiece and the smallest dimension allowable within the specified tolerance for all rib widths on the workpiece.

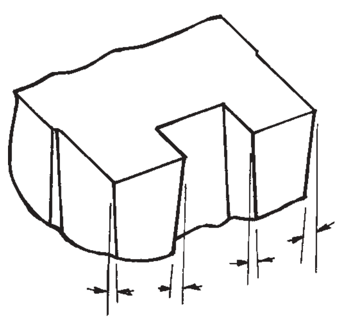
In order to get the maximum resharpenable tool life from the insert, the top to bottom clearance should be such that as the insert is resharpened to it's point of maximum usability, it will always be within the allowed tolerance range.

The degree that the wire is offset is determined by the minimum tolerance on any of the widths being formed. The following chart shows the degree that the wire should be offset to attain the required tolerance on the workpiece.

Important: The EDM program should be such that the wire is only offset when cutting vertical surfaces and should be at 0 degree for all horizontal and angular or radius surfaces.

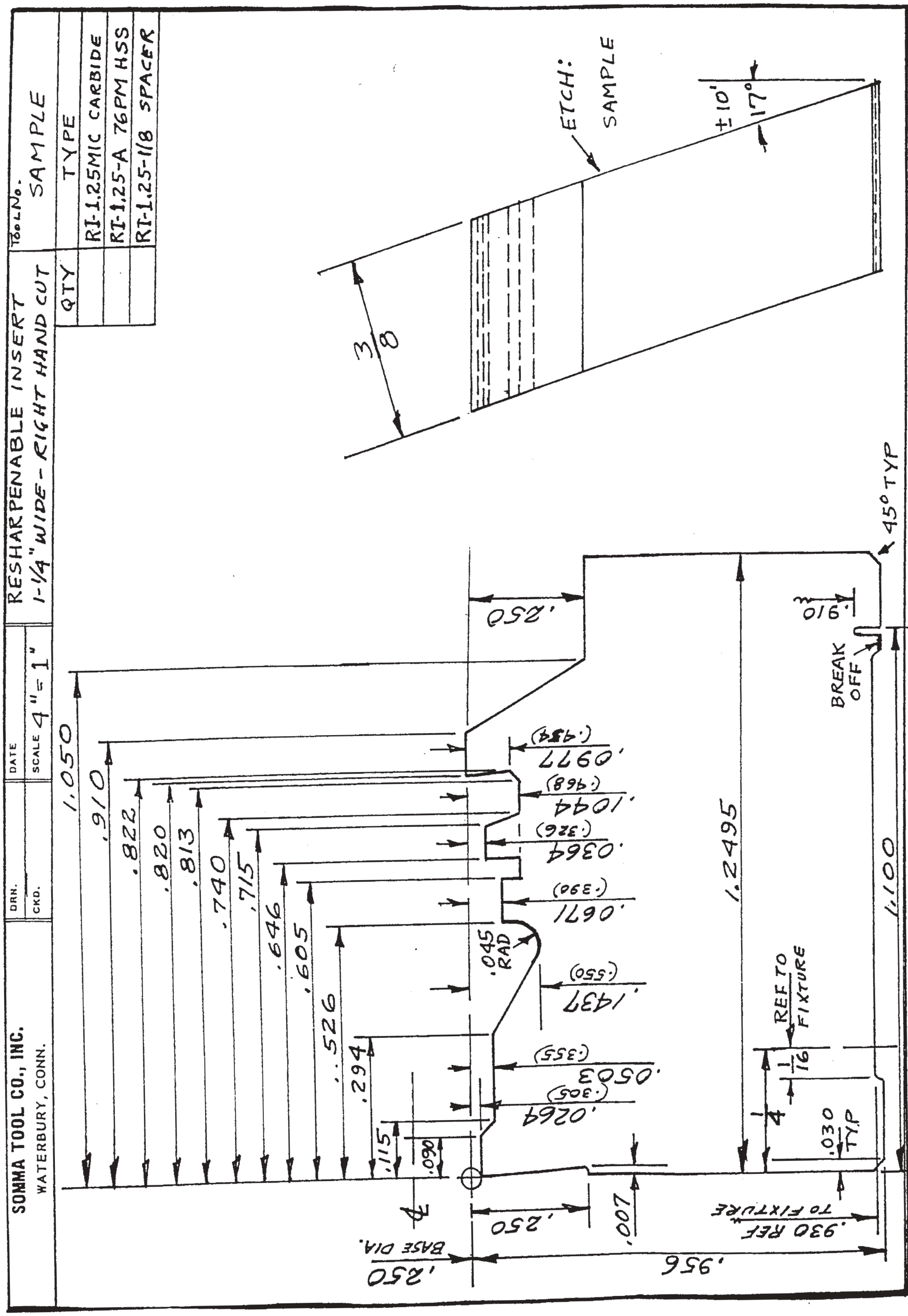
The pivot point of the wire must be in line with the intersection of the 17 degree offset and the longest length of the insert. (.956" for the 3/8" thick inserts and .630" for 5/16" thick inserts.)

Because the insert is mounted at 17 degrees in relation to the pivot point, a slight front to back clearance will be generated on all vertical surfaces. This does not do any harm and in fact is usually advantageous.



TOP TO BOTTOM CLEARANCE CALCULATIONS BASED ON .250" OF RESHARPENABILITY WITH 17 DEGREE EDM FIXTURE

TAPER PER SIDE @ .250" DEPTH	.001"	.0015	.002"	.003"	.004"	.005"	.006"
OFFSET DEGREE	0.23	0.35	0.46	0.69	0.92	1.15	1.37


SOMMA TOOL CO., INC.
WATERBURY, CONN.
DRN.
CKD.
DATE
SCALE 4" = 1"
RESHARPENABLE INSERT
1-1/4" WIDE - RIGHT HAND CUT
QTY
TYPE
TOOL NO.
SAMPLE
RI-1.25MIC CARBIDE
RI-1.25-A 76PM HSS
RI-1.25-1/8 SPACER
ETCH:
SAMPLE
±10'
17°
3/8
45° TYP
.250
.910
BREAK OFF
1.2495
1.100
REF TO FIXTURE
1/16
1/4
.030 TYP
.930 REF TO FIXTURE
.956
.250 BASE DIA.
.250
.007
1.050
.910
.822
.820
.813
.740
.715
.646
.605
.526
.294
.115
.090
.0977 (.434)
.1044 (.968)
.0364 (.326)
.0671 (.390)
.045 RAD
.1437 (.550)
.0503 (.355)
.0264 (.305)

CORRECTING ANGLES AND RADII FOR EFFECTS OF CHANGE CAUSED BY STEP DIFFERENTIAL CORRECTIONS.

Whenever an angle is specified, the height width and angle must be calculated to arrive at the dimensions as they must be on the finished part. However, the height will be different on the tool than on the part because of the effects of the step differential corrections.

Therefore, the angle must be recalculated so that it will meet the corrected height dimension. Triangles are easily calculated with the "FTSC" program. See Figure 9.

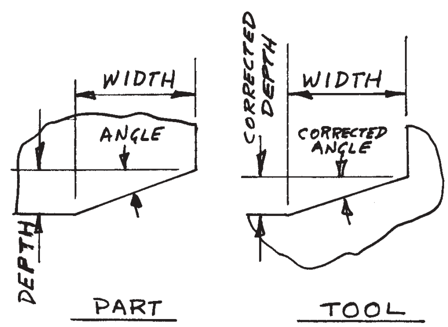


Figure #9

Whenever only a chord of a radius is specified, the height, width and radius of the chord must be calculated to arrive at the dimensions as they must be on the finished part. However, either the height or the width will be different on the tool than on the part because of the effects of the step differential corrections.

Therefore, the chord must be recalculated and the radius changed so that it will meet the corrected height or width dimension. Chords are easily calculated with the "FTSC" program. See Figure 10.

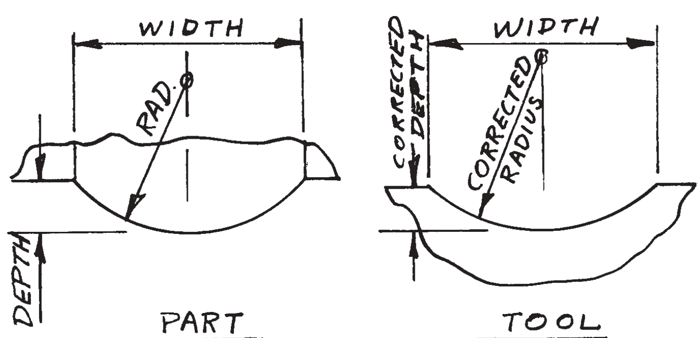


Figure #10

Note: It is almost impossible to produce a perfectly spherical radius (+/-.001") on the part with a form tool because of the distortions caused by the step differential corrections inherent in a form tool. If a radius must be held to close tolerances, it is best to form it with a shave tool which cuts tangentially, and therefore does not require step corrections.

CALCULATING COMPLEX ANGLE AND RADIUS CONTOURS

When a radius and angle are tangent to each other, it will help the toolmaker to know the location of the tangent point of the radius and the angle and also the theoretical point at which the angle meets the intersecting surface.

As seen in Figure 11, the angle from the center of the radius to the intersecting point is 1/2 of the angle to the tangent point. Therefore, finding “A” is merely a matter of solving this triangle.

To find the tangent point also merely requires solving of the triangle that intersects tangent point.

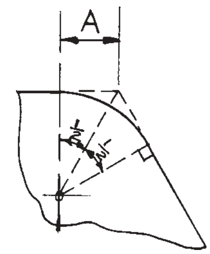


Figure #11

The following diagrams illustrate many of the contours that may have to be calculated in designing form tools if you do not have Cad-Cam available.

All of these angles and radii can be calculated easily if they are broken down into right angle triangles or chords, that can be solved with the “FTSC” program.

The diagrams (Figure 12) show the right angle triangles that comprise the contours and that have to be solved. Depending on which dimensions are known, it is easy enough to find the unknown sides, angle or radii. Once these dimensions are calculated, they can be used in conjunction with the known dimensions to find all the dimensions required for the toolmaker to produce the tool.

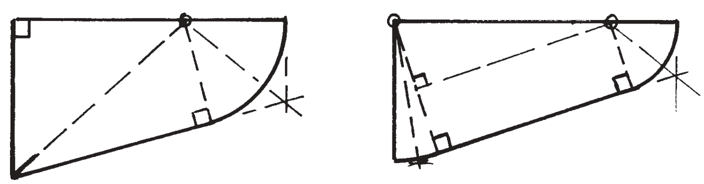


Figure #12

The secret for solving the complex contours shown in Figures 13 is that angles "A1" and "B1" are identical to angles "A2" and "B2".

Therefore, for example, if radius "R1" is two times the size of radius "R2", the other sides will also be in the same ratio of 2:1. Since distance "XY" can be readily calculated, distance "ZY" will be 2/3 of "XY" and distance "XZ" will be 1/3 of "XY".

With "XZ" and "ZY" now known it is easy to solve for angles "A1"/"A2" which are identical and "B1"/"B2" which are also identical.

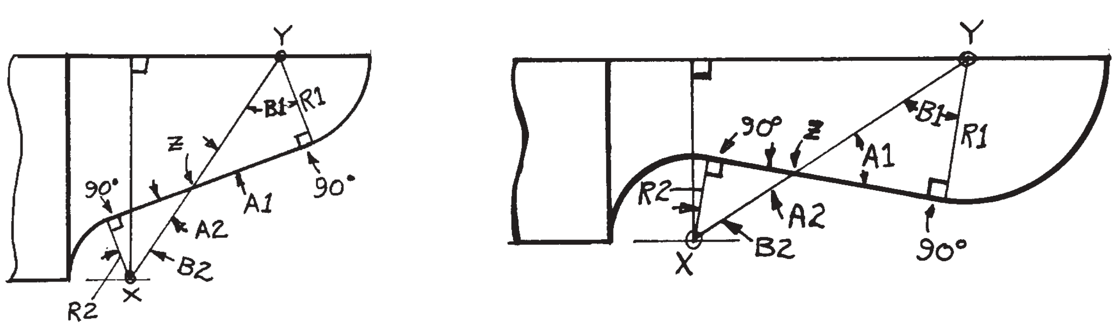


Figure #13

"FTSC" FORM TOOL DESIGN CALCULATION PROGRAMS

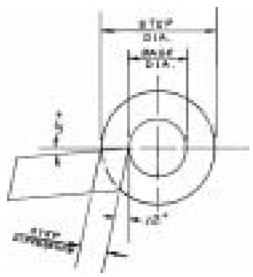
SOMMA'S EXCLUSIVE IBM COMPATIBLE P.C. DISC PERFORMS ALL YOUR FORM TOOL CALCULATIONS

FOR RESHARPENABLE QUICK CHANGE FORM TOOL INSERTS

All you need do is enter:

- Front Clearance Angle = 12º
- Top Rake Angle = 5º
- Base Diameter of Part =

Then as you enter each part diameter, the program calculates the corrected step difference.

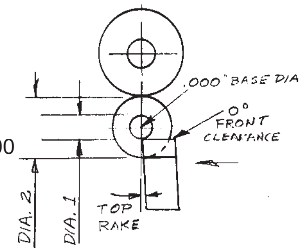


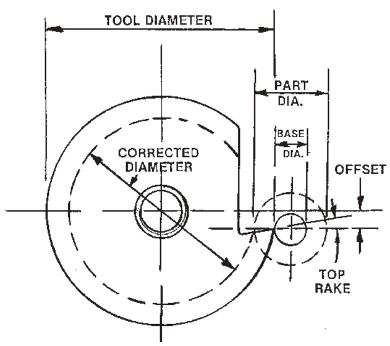
FOR RESHARPENABLE QUICK CHANGE SHAVE TOOL INSERTS

All you need do is enter:

- Front Clearance Angle = 0°
- Top Rake Angle = 0°
- Base Diameter of Part = .000", 125 or .500

Then as you enter each part diameter, the program calculates the step difference.



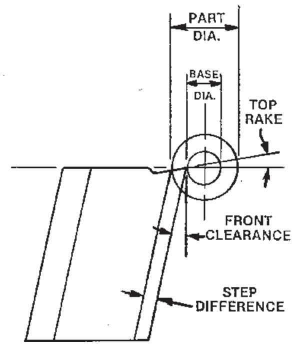


FOR CIRCULAR FORM TOOLS

All you need do is enter:

* Tool Diameter---
* Tool Offset Distance---
* Top Rake Angle---
* Base Diameter of Part---

Then as you enter each part diameter, the program calculates tool diameter and step difference.



FOR FLAT TYPE FORM TOOLS

All you need do is enter:

* Front Clearance Angle---
* Top Rake Angle---
* Base Diameter Of Part---

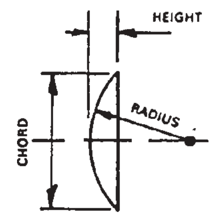
Then as you enter each part diameter, the program calculates the corrected step difference.

FOR RADIUS PROBLEMS

All you need do is:

* Enter any two of the three dimensions of a chord---

The program will calculate the third dimension.

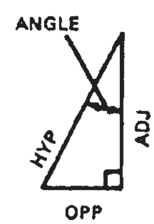


FOR TRIANGLE PROBLEMS

All you need do is:

* Enter any two of the four dimensions of a right angle triangle---

The program will calculate the remaining two dimensions.



FEATURES:

* Leads you through problems.
* Eliminates long, complicated manual calculations.
* Eliminates corrected diameter tables, charts, graphs, trig tables, and square root calculations.
* Calculates any Circular Form Tool - regardless of Diameter, Offset or Top Rake.
* Calculates any Flat Type Form Tool - regardless of Front Clearance Angle or Top Rake.
* Can print out answers for a permanent record.
* Available in DOS or WINDOWS version.

Cat. No.	Operating System
FTSC-3.5	DOS
FTSCW	WINDOWS
FTSC-2	CALCULATOR

SECTION 3

HELPFUL IDEAS

REDUCING CUTOFF THICKNESS

The narrowness of the cut-off is limited by two conditions; first, the relation of the width of the blade to its length, or depth of penetration, so that it will not walk, vibrate or break off, and secondly, its ability to dissipate the heat generated in cutting so that the blade will not burn out in usage.

With these conditions in mind, it is obvious that if for any reason the blade is not penetrating the full diameter of the stock, such as when cutting off into a hole, or when cutting off and forming the front end of the next piece, the blade width can be narrowed considerably.

Although you can theoretically cut off a part with a blade thickness equal to that required for cutting off the diameter of the part adjacent to the cut-off blade rather than that required for the full diameter of the stock, in practice it is best to make the blade width the mean size between the two. This is due to the fact that since the blade is actually starting to cut at the stock O.D., it does need a little extra width to dissipate the extra heat thus generated.

Figure 1 shows an example of cutting off into a hole. The dotted line illustrates the thickness required to cut-off solid stock. The solid line shows that since in actuality we are only cutting off the wall thickness and not extending as far into the part, we can reduce the cut-off thickness accordingly.

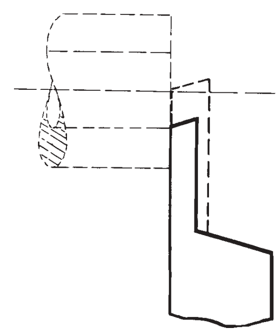


Figure #1

Figure 2 shows another manner in which cut-off thickness can be reduced. Here we have a part that is turned down on the front end by the cut-off tool. Therefore, our method for determining the cut-off width would be the mean between the cut-off thickness needed for the turned down diameter and that required for the stock O.D. In fact, the size calculated can be based on the diameter of the part at the base of the front chamfer.

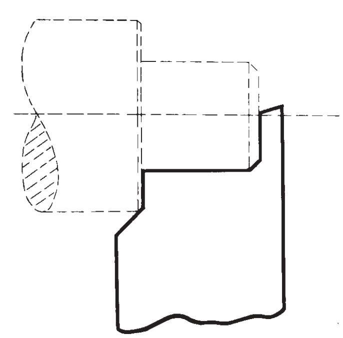


Figure #2

Figure 3 shows how the cut-off thickness can be reduced when cutting off with a radius, but also how this can vary depending upon the size of the radius. In the left hand diagram the radius is fuller, thereby giving a greater area in the cut-off portion of the tool as shown, therefore, the cut-off thickness can be made thinner. In the right hand diagram, the radius is very flat, thereby cutting down the area for heat dissipation and rigidity and necessitating the widening of the cutoff blade to compensate for this loss of thickness.

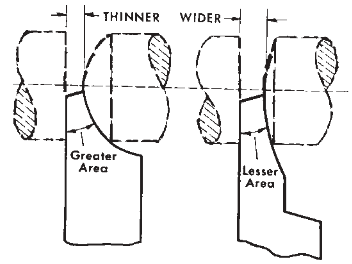


Figure #3

Figure 4 shows a case in which the cut-off thickness is strengthened by the fact that the tool is forming a taper on the front end of the part and also cutting off on a radius. This not only gives the blade extra rigidity, but also extra area to help dissipate the heat generated in cutting. Here again, a considerably narrower cut-off can be used than that required to cut off the stock diameter.

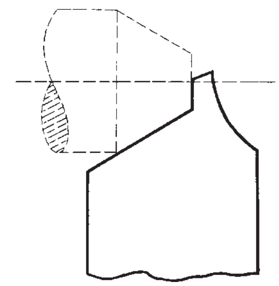


Figure #4

Figure 5 shows another means by which cut-off thicknesses can be reduced. Here the form tool is turning down a hub on the back end of the part. Making the width of the marker wider than the cut-off blade is the equivalent of actually only cutting off stock the diameter of the back shoulder since the cut-off tool does not come in contact with the stock size at any time. It is, however, necessary to face off the step left on the front of the part with the form tool as shown, or remove it with some subsequent tool.

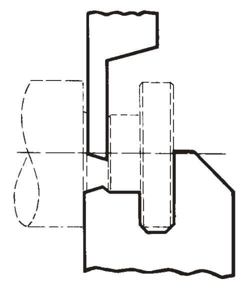


Figure #5

Still another means of narrowing the cutoff width is shown in Figure 6. In this instance, when a hole has to be drilled in the front of the part that is larger in diameter than the back end of the part, the cut-off blade can be narrower than the marker on the form tool, and need only be as wide as is needed to cut-off the back diameter. Again in this case, the blade does not come in contact with the full diameter of the stock at any time. Another advantage is that it is not necessary to face off the front of the part, since the step is removed by the subsequent drilling operation.

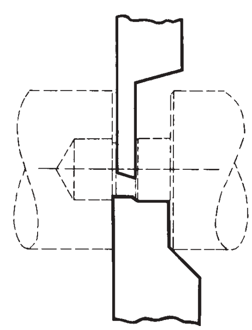


Figure #6

Figure 7 illustrates a cut-off blade that can be thinned down when the back end of the part is formed to a smaller size, thereby retaining the rigidity of the cut-off blade thickness required to face off the stock O.D., and yet thinning it out for the portion of the back end of the preceding piece that is formed down.

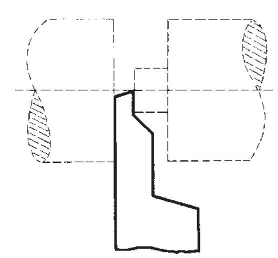


Figure #7

Figure 8, however, shows the case where the cutoff thickness must be maintained and sometimes even be made wider than that required for the stock diameter. This applies when the piece has to be tapped all the way through, in which case in order to allow for the lead on the tap and a certain amount of working tolerance, the cut-off blade should be the width of 2-1/2 threads minimum.

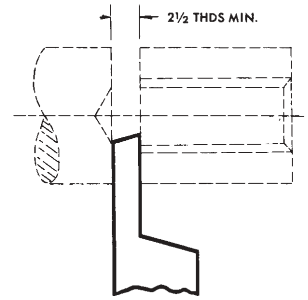


Figure #8

IMPROVING CUT-OFF FINISH

Where part tolerances permit it, a slight negative clearance on the cut-off blade, as shown in Figure 9, can do wonders for the finish on the cut-off end of the part, and also increase tool life. This taper can be slight, on small parts it need not exceed 1 degree to 2 degrees or on very large parts a taper of .005”-.010” is usually sufficient.

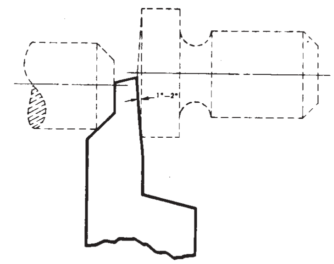


Figure #9

CONTROLLING CRITICAL DIMENSIONS

Figure 10 shows how to control a critical dimension by “nailing it down” in the form tool. When the overall length, or any intermediate lengths, are critical, it is best to incorporate a facing shoulder into the tool, (even though the part is turned from the turret or by another tool) thereby controlling the lengths consistently and eliminating the necessity for the operator to spend an undue amount of time in setting up and maintaining these lengths. This is especially true on older machines with inconsistent feed outs, but is true on many jobs where bounce back from the stop varies with the length of the bar, due to the gradual decrease in its weight as it is cut up into parts.

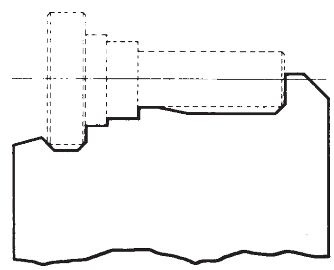


Figure #10

ELIMINATING RINGS ON THE WORKPIECE

Figure 11 illustrates another condition to avoid when roughing out, or whenever two different tools overlap in the forming of a part.

As you will notice, tool “A”, which forms the part prior to tool “B” runs off on an angle at the point of overlap. If this angle is not put on the tool and it ran off at right angles as shown by the broken line, there would be a line, ring, or difference in color on the part where tool “A” left off and tool “B” took over. This is due to the fact that tool “B” would be doing no cutting on the portion roughed out by tool “A” until it reached the finish diameter, whereas the balance of the tool would be removing stock all the way down from the O.D. This will cause faster breakdown of this area of the tool with the resulting size and finish difference.

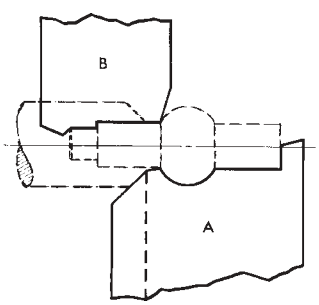


Figure #11

The addition of the angle causes this difference to be less pronounced, since there is no definite point at which tool “A” leaves off. Rather, there is a gradual sideways movement of this wear point as the tool takes the part down to size.

CUTTING OFF WITH A RADIUS

When cutting off and forming a radius on the back end of the part falling off, we run into many difficulties in trying to maintain the correct radius size. As shown in Figure 12, the center line of the radius on the tool must be slightly ahead of the point of the cut-off tool to allow for the fact that the part will break off at some diameter before it reaches center.

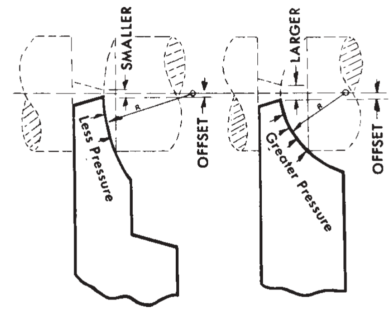


Figure #12

This offset varies with the size of the radius. If the radius is rather flat, as shown on the left hand drawing, the pressure exerted against the piece falling off is lesser, therefore, the piece will have a smaller break-off diameter. If the radius is fuller, as shown in the right hand diagram, the pressure is greater, tending to break the piece off sooner and thereby increasing the size of the offset.

The length and width of the part also affect the diameter at which the piece breaks off. A long heavy part will break off sooner than a short light part.

Also as the tool becomes dull it will not cut as freely and will exert more pressure against the piece being cut off thereby breaking it off sooner. This latter is the cause for variations in radii on parts in the same production run even though the same tools have been used throughout.

If this offset is estimated correctly, a practically perfect radius can be formed on the part as shown in Figure 13 in the left hand diagram. However, if this offset is not estimated correctly, the part breaks off sooner than anticipated.

The radius becomes flattened out as shown in the right hand diagram because it actually produces a flat on the center and two radii swung from points off the center of the piece. In this case, the radius of the cut-off tool and radius shown by the dotted line are exactly the same. The difference is that the radius is swung from the off-center point at which the part breaks off, creating the appearance of a flatter radius.

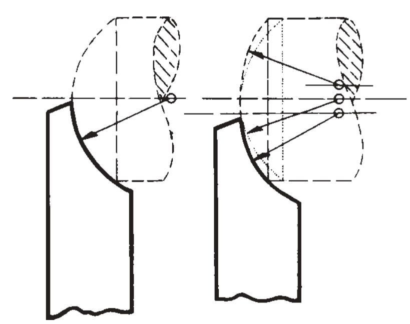


Figure #13

This flattening of the radius also tends to make it difficult to maintain the overall length of the part, especially when the radius cut-off tool is blending in with a radius that has been partially formed by the form tool.

This is one of the most common problems in cutting off with a radius. This condition can, however, be overcome by dubbing back the cut-off angle on the form tool until the radius falls in correctly. This can be done by hand, by trial and error, and then the tool can be mounted on an arbor and ground to this point all the way around. Because of this fact, it is a good idea to increase the past center distance on a radius cut-off tool when it is also forming the front end of the next part so that if it has to be ground back, the cut-off tool will still face past the center of the part and not necessitate a complete regrinding of the form tool. This is shown in Figure 14.

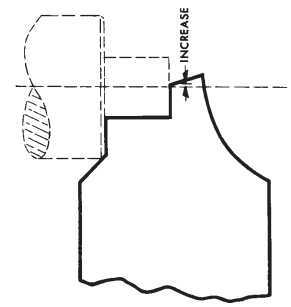


Figure #14

The break off diameter can also be reduced somewhat, when cutting off with a radius, by using a sharper cut-off angle. This angle can be increased to 20 degrees on steel and other hard materials, and to 25 degrees to 30 degrees on brass, and other free machining materials.

The other important factor in producing a good radius on the back end of the part being cut-off is to form as much of it as possible with the forming tool, leaving the minimum amount to be formed with the radius cut-off tool.

Since it is practically impossible to form a good radius with only the cut-off tool, it is essential that a rear radius be partially formed, as deep as possible without breaking off, before the radius cutting off operation.

If the part is not partially formed and the complete radius is attempted to be put on with the cut-off alone, the radius will usually end up with flags, chatters, undue roughness and incorrect radius size.

We have found that it is possible to form down safely to 1/3 the diameter or 1/3 the length of the part being formed, whichever is the greater, as shown in Figure 15.

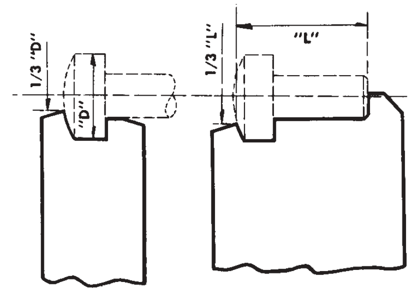


Figure #15

This rule, however, may have to be waived when forming a part with a thread cut on it, such as is shown in Figure 16. In this case, it is not advisable to go much below the root diameter of the thread since the pressure exerted by the die or chasers while cutting the thread may tend to twist the part off the bar.

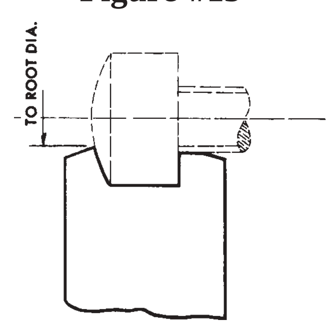


Figure #16

ROUGH IN WHERE POSSIBLE

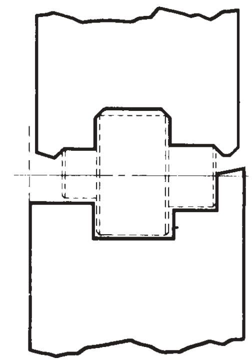


Figure #17

Screw machines that have more than two slides offer many opportunities for roughing out the part prior to finishing, however, even the older two slide machines can be used to advantage by combining roughing out with the cut-off tool, and then finishing from the other slide as shown in Figure 17.

The extra investment in a roughing tool is negligible in comparison to the increased production, reduced down time, better part tolerance and finish, and even reduced cycle time in some instances.

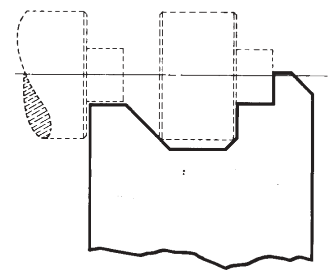


Figure #18

Therefore, we always recommend roughing in wherever feasible. This can be done in many ways; with a combination cut-off and form as shown previously; by extending the form tool to form the front of the next part, as shown in Figure 18, or even with a turning tool from the turret. The important thing to remember is that whenever a minimum of stock is left for the finishing tool, maximum production efficiency is obtained.

It is also important, however, to remember to not rough in too close to the finish dimensions. This can result in lost production because of the difficulty of lining up the rough and finish tool, or worse yet, if too much stock is removed by the rough tool, the finish tool does not come in contact with some areas of the part, causing it to be out of tolerance.

Another problem caused by roughing in too close to the finish sizes, is premature breakdown of the finish form tool because it does not have enough stock to bite into causing the tool to rub and gall and tear.

"FISHTAIL" CUTOFF FOR TUBING

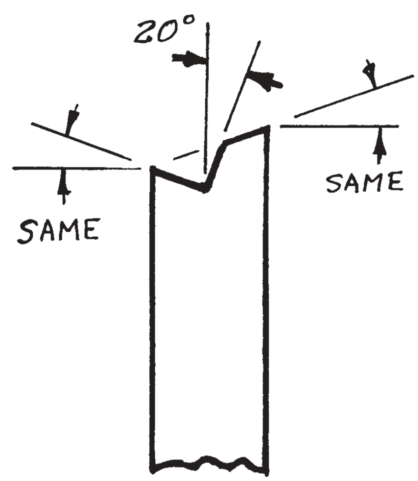


Figure #19

When cutting off tubing, the heel of the cutoff angle has a tendency to throw a burr into the hole of the piece still in the machine. To avoid this problem a double angle similar to a fishtail or a "V" should be ground into the cutoff blade.

An angle of the same degree as the cutoff angle but in the opposite direction should be specified. The high point of this angle should start at the low end of the original angle and should extend about one third of the way across the blade. From the low point of this angle an angle of 20 degrees from the vertical should extend upwards until it meets the original angle. See Figure 19.

The advantage of this double angle is that, because the high point of the angles are against the face of the part on both sides of the blade, the metal will cut off clean rather than being rolled back into the hole.

DOVETAIL TOOLS

As a manufacturer of dovetail tool blanks for the screw machine industry, we have over the years reviewed thousands of tool designs supplied by both large and small companies. This opportunity to observe tooling practices has convinced us that substantial savings can be had through standardization of dovetail holders and blank sizes, and through design and tolerancing of tool prints for simpler form grinding or EDM machining.

Even where only a few tools are used each week, it pays to modify or replace existing holders to reduce the number of different dovetail sizes to an absolute minimum, preferably to American Standard specifications.

The next step should be to standardize the heights, widths and lengths required to as few as possible, specifying 1/2-inch, increments on the height and width.

The great majority of jobs can be condensed into a minimum number of standards which can then be purchased in larger quantities for in-plant or outside finish grinding or EDM machining.

Height of blanks should be fractional and should conform with standards. For interchangeability, dimensions that determine the part diameter with the least tolerance should be controlled from the bottom of the dovetail, and should conform with standard blank height so as not to necessitate extra stock removal. All other diameter dimensions should be based from this first surface, rather than in relation to overall height of tool. See Figure 20.

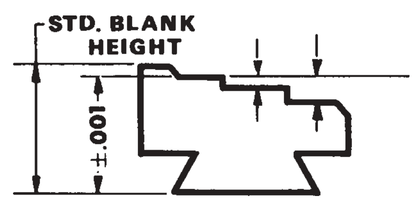


Figure #20

Width of blanks should be fractional and should conform with standards. For interchangeability, a dimension should be given from the side of the dovetail that bears against the stationary side of the tool holder, to the most critical length dimension of the part. All other length dimensions should be dimensioned in relation to this point, not in relation to sides of tool. Excess width should be left on the side of the tool away from the spindle. See Figure 21.

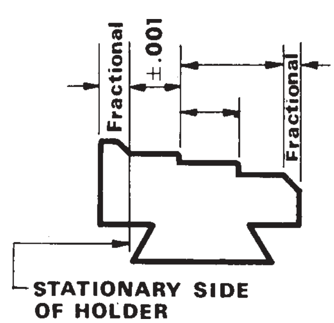


Figure #21

Dovetail width should be dimensioned to the theoretical sharp points with +/- .003 tolerance. Points of dovetail should be chamfered with "1/32 approx. flat". Dovetail depth size should be dimensioned as + 1/64/ -.000, corner radii should be shown as "1/32R or 1/16R approx." See Figure 22.

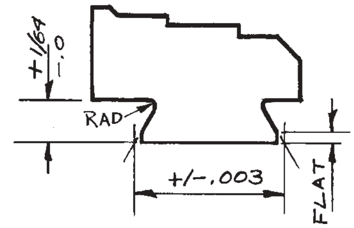


Figure #22

Dovetail blanks should be designed as parallelograms, that is, with same front and back end angles to reduce cost of manufacture, and also for ease of resharpening. This way blanks can be resharpened by laying on a magnetic chuck instead of necessitating use of angle vise or special fixtures. See Figure 23.

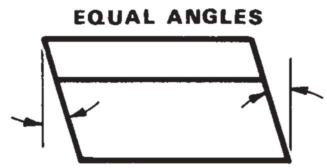


Figure #23

CHECKING DOVETAIL SIZE

Gages should be used to check width of dovetails to theoretical points, similar to those used in blank making operations. These gages are simpler to use and more accurate than checking with balls or rods. See Figure 24.

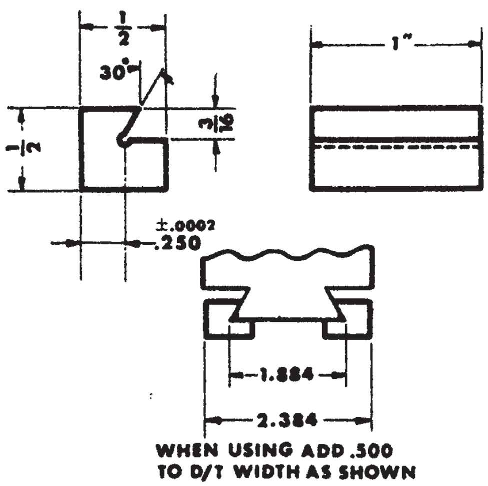


Figure #24

CARBIDE TIPPED DOVETAIL TOOLS

On carbide-tipped tools, length of carbide should be as long as possible, preferably one half the overall length of the tool. Width of carbide should never be much wider than actual cutting width, since this increases blank price and also grinding time to remove the excess. Specify standard carbide manufacturers' style 1000 or style 0000 blanks whenever possible.

The foregoing applies to dovetail shave tools as well as dovetail form tools, except that height of finished shave tools cannot always be standardized since it must vary according to the diameter being shaved.

THE FORGOTTEN ART OF SKIVING

Skiving is one of the oldest and most efficient methods for producing certain types of parts on screw machines: Parts that are long and slender, parts with close diameter tolerances and finishes, and parts requiring truly spherical radii. Yet most layout men, set-up men, and operators seem to go to any lengths to avoid skiving.

This is probably due to a lack of knowledge and familiarity with the principles of successful skiving, including applications, tool design and manufacture, and the unavailability (until now) of a tool holder with all the features necessary for simple, dependable operation. Skiving is a basically simple operation. Once a few principles are understood, any screw machine plant should be able to run skiving jobs routinely with few, if any, problems.

What is skiving?

Convention form tools are mounted so that the formed cutting edge of the tool is on the centerline of the part and cuts radially. Cutting action is determined by the combination of radial clearance angle and top rake angle, as shown in Figure 25. Diameters are controlled by advancing the forming tool towards the center of the part. But difficulty is encountered in forming long parts because the entire form contacts the work piece at one time. Therefore, the smallest diameter is formed to its finished size at the same time as all the other diameters are being formed, causing long parts to break off prematurely.

On the other hand, skive tools are mounted so that the formed cutting edge, which is ground for the full length of the tool, is advanced into the work piece below center and cuts tangentially. Cutting action is determined by the combination of "Shear" angle and "Lead" angle, as shown in Figure 26. Part diameters are controlled by raising the tool towards the center. The cutting edge is obtained by grinding the "Shear" angle on the front end of the tool and the "Lead" angle across the width of the form.

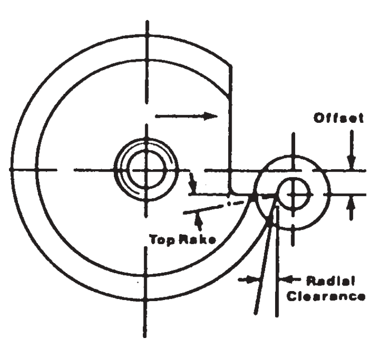


Figure #25

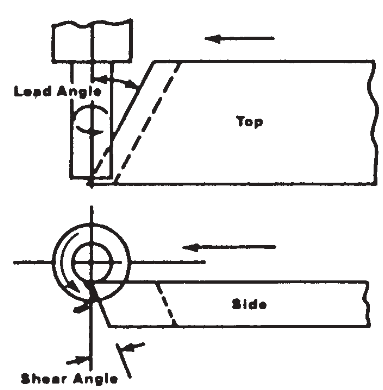


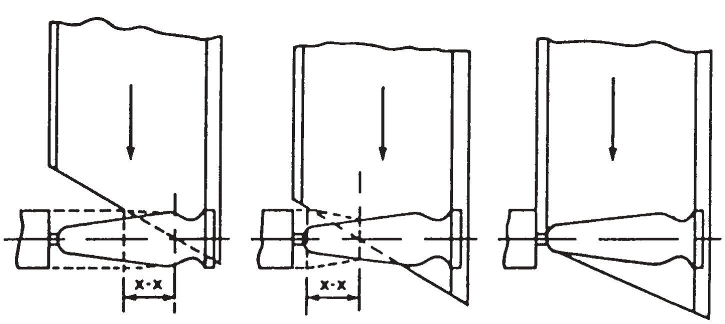
Figure #26

Since only that part of the tool that is in contact with the work piece at any given time is actually cutting, the part is not weakened until that portion of the tool which forms the smallest diameter of the part actually passes below the centerline. Furthermore, as any portion of the tool passes under the centerline, that diameter is completely formed to size and the tool exerts no further cutting pressure on that area of the part. This means that all cutting action takes place in the area from where the tool contacts the part until it passes under center. This area is shown as X-X in Figures 27 and 28. Figures 27A, 27B, and 27C show what happens as the tool is advanced past center.

The shear angle is critical to the cutting action. But, unlike the top rake on form tools, it does not affect the part diameter relationship. Therefore, this angle can be varied until the best possible cutting action is obtained. Too blunt an angle will tend to cut hard and deflect the part, causing dimensional errors and poor finishes. Too steep an angle lengthens the feed cycle and causes the cutting edge to burn out prematurely, again affecting diameters and finish.

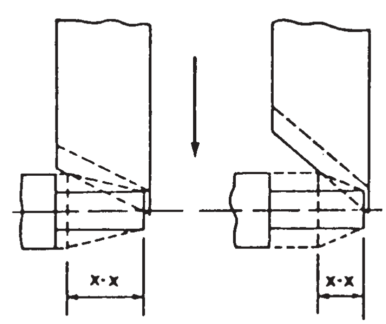
It has been our experience that approximately a 20 degree shear angle and a 20-30 degree lead angle are good starting points. A little experimentation can produce surprisingly different results in finishes, tolerances, cutting action, and cycle times and is well worth the effort.

Once these principles are understood, it is easy to see how the lead and shear angle can be varied to suit the configuration of the part so as not to weaken it prematurely. It's important to keep the lead angle as short as practical, since the steeper the angle, the more throw (and cycle time) is required. See figures 28A and 28B.



#27A **#27B** **#27C**

Figure #27



#28A **#28B**

Figure #28

Advantages of Skiving:

Since skiving is a freer cutting operation, cross slide feed rates can be increased approximately 3-6 times for brass, aluminum and easy to machine materials, and 2-3 times for ferrous and difficult-to-machine materials. This offsets the increased throw time required because of the lead angle. At times it is possible to grind a double angle to reduce throw as shown by dotted lines in Figure 29.

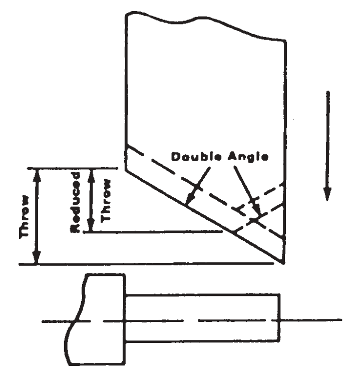


Figure #29

Another advantage of skiving is that, since skive tools cut tangentially instead of radially, step differentials remain unchanged and angles do not have to be corrected, simplifying tool design. Since diametral corrections are not required (as in conventional tools), skiving is about the only way a perfectly spherical radius can be formed on a work piece.

In addition to being used for long parts, skiving is used successfully for parts requiring close diameter tolerances or finishes. Much closer tolerances can be maintained because part diameters are controlled by raising or lowering the skive took, as in a shaving operation, instead of feeding the tool against a stop. For the same reason, tool wear does not affect diameter dimensions directly (except through poor cutting action).

Better finishes are also obtained with skiving. As the formed portion of the tool continued to advance under the center of the part, it produces a shaving or burnishing action on the part. Carbide tipped tools are particularly recommended where very good finished are required.

Although a certain amount of this burnishing action is desirable, too much drag can cause the part to spring or deflect. Therefore, most skive tool holders are built with a 1/4 degree maximum backtaper to prevent excess rubbing as the tool passes under the work piece. However, while eliminating drag, this backtaper causes tapers in the part. In combination with the lead angle, it causes the point at which the tool passes the center of the part to drop away from the center, thus increasing part diameter.

The amount of taper produced can be calculated as follows:

Width of skive tool x Tan of lead angle x Tan of backtaper angle (usually 0 deg. 15') = Taper per side.

These tapers have, in the past, been overcome by either packing up the holder by this calculated amount or by grinding an offsetting taper on the bottom of the skive tool itself. Both of these methods are hit or miss and a cause of many operating problems.

Tapers are also occasionally caused by deflection of the part, but this can be overcome by use of an end support on the part.

Another difficulty encountered has been that most holders are not designed to accommodate extra wide tools and, therefore, necessitate the grinding of “T” or “L” shape shanks on the tools. This shortens resharpenable tool life, increasing the cost of the tools. It also causes rigidity problems in many instances. A further difficulty has been that the depth of form in the tool is limited because of a too small distance between the center of the spindle and the lowest adjustment of the taper wedge in the tool holder.

With all these problems in mind, Somma developed a skive tool holder to fit the Brown & Sharpe and other machines. The unique feature of this tool is a rocker-type taper wedge that can be adjusted to offset part tapers.

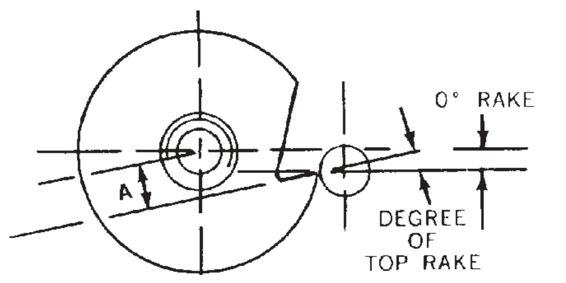
Other features include: Extra wide tool openings to eliminate cutting down of shanks of skive tools; greater distance from center of spindle to lowest adjustment point of taper wedge to accommodate larger part diameters; rugged construction to eliminate chattering; available for rear slide for forward rotation or front slide for jobs requiring left hand rotation; 1/4 degree maximum back taper built into the tools to prevent part deflection.

In our many years of supplying tooling for the screw machine industry, we have seen all types of parts skived, such as ball point pencil tips, hypodermic needle hubs, ornamental lamp finials, and fireplace andiron parts, as well as ball-type fittings and parts requiring close tolerances or exceptional finishes. We hope that the preceding information will encourage you to join the ranks of the companies that are producing unusual parts efficiently and economically by skiving.

SECTION 4

USEFUL TABLES

The following table shows the distance the cutting edge must be ground below center to attain various degrees of top rake on B&S and Davenport circular form tools. If the tool is not sharpened to the exact top rake it was calculated for, the various diameters of the part will be incorrect.



Degree of Top Rake	No. 00 B&S 1-3/4" Dia. "A"	No. 0 B&S 2-1/4" Dia. "A"	No. 2 B&S 3" Dia. "A"	Davenport 2" Dia. "A"
0	.125	.156	.250	.125
3	.170	.214	.327	.177
5	.200	.252	.378	.211
7	.230	.291	.428	.245
10	.273	.347	.503	.295
12	.302	.384	.552	.328
15	.345	.439	.624	.378

SINGLE DEPTH OF THREADS					
TPI	S.D.	TPI	S.D.	TPI	S.D.
8	.081	30	.0215	62	.010
9	.072	32	.020	64	.010
10	.065	34	.019	66	.0095
11	.059	36	.018	68	.0095
12	.054	38	.017	70	.009
13	.050	40	.016	72	.009
14	.0465	42	.0155	74	.009
15	.0435	44	.015	76	.0085
16	.0405	46	.014	78	.008
18	.036	48	.0135	80	.008
20	.0325	50	.013	82	.0075
22	.0295	52	.0125	84	.0075
24	.027	54	.012	86	.0075
26	.025	56	.0115	88	.007
27	.024	58	.011	90	.007
28	.023	60	.0105		

AMERICAN STANDARD TAPER PIPE THREAD
TAPER = 1 degree 47'

Size	Major Dia. maximum	Length of effective thread	Single Depth
1/8-27	.409	.264	.030
1/4-18	.541	.402	.044
3/8-18	.678	.408	.044
1/2-14	.844	.534	.057
3/4-14	1.054	.546	.057
1"-11-1/2	1.318	.683	.070
1-1/4"-11-1/2	1.663	.707	.070
1-1/2"-11-1/2	1.902	.724	.100

DIAMETERS OF NUMBER SIZE THREADS	
SIZE	MAJOR DIA. MAX
0	.060
1	.073
2	.086
3	.099
4	.112
5	.125
6	.138
8	.164
10	.190
12	.216

WIDTH OF CUTOFF	DEPTH OF ANGLE		
	15 Deg.	20 Deg.	23 Deg.
.020	.0055	.007	.0085
.030	.008	.011	.013
.040	.011	.0145	.017
.050	.0135	.018	.021
.060	.016	.022	.0255
.070	.019	.0255	.030
.080	.0215	.029	.034
.090	.020	.033	.038
.100	.027	.0365	.0425
.120	.032	.0435	.051
.140	.0375	.051	.0595
.160	.043	.058	.068

TABLE OF ACROSS CORNERS DIMENSIONS OF HEXAGON AND SQUARE STOCK

FLATS	A/C HEX	A/C SQUARE	FLATS	A/C HEX	A/C SQUARE
.0625	.072	.088	.5937	.685	.839
.078	.090	.110	.625	.721	.884
.0937	.108	.132	.6562	.757	.928
.1093	.126	.154	.6875	.794	.972
.125	.144	.177	.7187	.830	1.016
.1406	.162	.199	.750	.866	1.060
.156	.180	.220	.7812	.902	1.105
.1718	.198	.243	.8125	.938	1.149
.1875	.216	.265	.8437	.973	1.193
.203	.234	.287	.875	1.012	1.237
.2187	.252	.309	.9062	1.046	1.281
.2343	.270	.331	.9375	1.082	1.326
.250	.288	.353	.9687	1.118	1.370
.2812	.325	.398	1.000	1.155	1.414
.3125	.361	.442	1.0625	1.226	1.502
.3437	.397	.486	1.125	1.299	1.591
.375	.433	.530	1.1875	1.371	1.679
.4062	.469	.574	1.250	1.443	1.767
.4375	.505	.619	1.3125	1.516	1.856
.4687	.541	.663	1.375	1.588	1.944
.500	.577	.707	1.4375	1.660	2.033
.5312	.613	.751	1.500	1.732	2.121
.5625	.649	.795	1.5625	1.804	2.209
			1.625	1.876	2.291

To find across corners of any hexagon: Multiply flat size by 1.1547

To find across corners of any square: Multiply flat size by 1.414

To find across corners of any octagon: Multiply flat size by 1.0824

SECTION 5.

SELECTING TOOL MATERIAL

There are three characteristics that affect the performance of any cutting material, they are:

TOUGHNESS *— The relative ability of the material to withstand deflection and interrupted cuts without chipping.*

WEAR RESISTANCE *— The relative ability of the material to withstand the abrasive action of difficult to machine materials.*

RED HARDNESS *— The relative ability of the material to withstand the heat generated when cutting hard materials without losing its hardness.*

Important!! If a high speed steel tool is not properly heat treated, these characteristics may be destroyed.

M2 H.S.S. (Rockwell C63-65) Has the highest toughness as well as good wear resistance and red hardness, and is also the least expensive. It can be used for:

Short run jobs - almost any material
Mild steel/Non ferrous materials
Tools with fragile cross sections of forms
Interrupted cuts - hex, square, pinion stock

M42 H.S.S. (Rockwell C65-68) Has high red hardness plus good wear resistance and toughness, and is medium priced. It can be used for:

Materials that generate excessive heat rather than abrasion.
Same applications as T15PM where quantity of parts or machinability does not justify cost of T15PM.

T15PM H.S.S. (Rockwell C64-66) Has high wear and abrasion resistance plus good red hardness and toughness, and is higher priced. It can be used for:

Long run jobs - any material
Size control and good finish
Abrasive materials
Aircraft steels
Hard to machine aluminums
Bronze/copper
Berrylium copper

76PM H.S.S. (Rockwell C65-68) Has the highest wear resistance and red hardness plus good toughness, and is the highest priced. It can be used when none of the above steels are satisfactory.

Cast Alloys The next step up from the high speed steels are the cast alloys. These materials, although they usually work out exceedingly well in a single point tool, do not work out quite as well in a form tool.

The reason for this is that by nature these materials are hardest close to the outer surfaces and softer and more porous towards the center. Therefore, unless the cast alloy is preformed for the individual job, you run the risk of grinding the form into the soft core of the material, with the resultant loss of performance, and also the possibility of running into blow holes which can ruin a complete grinding job very unexpectedly.

The extra cost of the preform, plus the fact that these tools must be constructed similar to a carbide tool, that is a turned and milled steel body with cast alloy tips brazed on (due to the fact that this material is only available in the tool bit form) makes them comparatively expensive tool for the small difference in performance.

Carbide By stepping up to carbide, we can experience probably the greatest measurable improvement when used under the right conditions as to equipment and handling.

Carbides are being used successfully on all kinds of applications on screw machines, from plastics to the toughest alloys, for cutting off to center and for interrupted cuts.

On a screw machine you usually are unable to run the job faster when changing over to carbide. This is because it is usually impractical to increase the speeds and feeds to take advantage of the full capabilities of carbide. However, the usage of carbide can be justified by greatly increased tool life between grinds (even at the same speeds and feeds) plus better surface finishes.

There are many grades of carbide, however, it has been our experience that the C2 grade works well on many materials, but that Micro Grain carbide gives the very best results on difficult to machine materials or long runs.

Circular carbide tools can be made with standard brazed on tips, or as a solid carbide circular ring brazed to a soft steel center hub for greatest long range economy.

WEAR RESISTANT COATINGS

Greatly improved cutting tool life can usually be attained by coating high speed steel and carbide cutting tools with a film of wear resistant metal compounds.

The two most widely used coatings are:

"TIN" titanium nitride coating that has been available for many years, and "TICN" titanium carbo-nitride coating, a relatively new process that has proven to outperform "TIN" for certain types of applications.

Much of the effectiveness of any coating is lost in the resharpening operation when the coating is ground off the cutting surface. However, it does still retain some of its effectiveness by eliminating friction on the side surfaces of the tool.

Therefore, as you can see from the preceding, selection of tool material is an important factor in every job. Each job should be considered individually, not merely taking any one material as being the cure all for every case but rather analyzing every job as to material being cut, production required, finish required, tool costs, equipment being used, and then selecting the material that will meet the most of your requirements. - But remember *profits* are not determined by the savings in tool cost but rather by the *number of parts* in the work pan at the end of the day.

SECTION 6

FORM TOOL GEOMETRY

The cutting action of form tools is determined by the front clearance angle and the top rake angle. This cutting edge geometry must be taken into consideration when calculating the step differentials of the form tool.

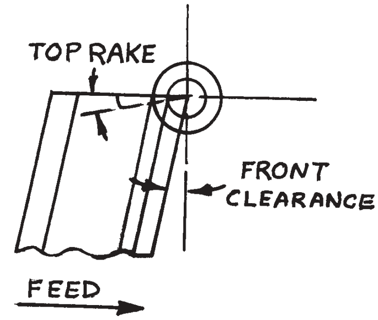


Figure #1

For dovetail forms tools, the front clearance angle is determined by the holder the tool is mounted in. The most common angle is 12 degrees. See Figure 1.

For circular form tools, the effective front clearance angle is determined by the major diameter of the tool and the distance the tool is located off center. Again, this is determined by the holder the tool is mounted in. For most holders, the effective clearance angle is between 7 to 11 degrees. See Figure 2.

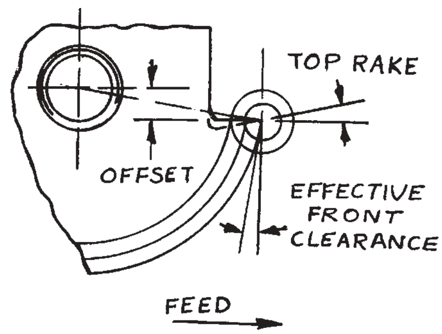


Figure #2

For circular and dovetail type form tools, the top rake is determined by the designer and can vary from 0 degrees to as much as 20 degrees for some very soft and gummy materials. The most common top rake angle is 5 degrees because it works well on most materials.

CIRCULAR FORM TOOLS

All circular tools are mounted so that the centerline of the tool is offset from the center of the machine spindle in order to produce an effective front clearance angle at the cutting edge. Figure 3 has been drawn with an exaggerated offset to show that in order to get the correct “X” part diameter difference, the tool diameters must be corrected, since “Y” is visibly shorter than “X”.

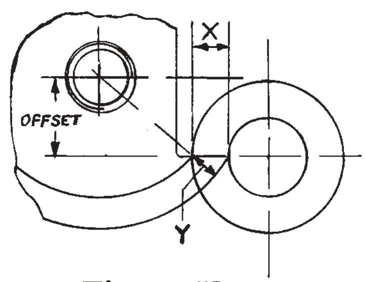


Figure #3

Figure #4 shows the further effect of adding a top rake to the cutting edge. As you can see, the tool diameters have to be corrected again, since if they are not, the diameters would come oversize as shown by the dotted lines. This is why it is very important that the tool be sharpened to the exact top rake that it was calculated for.

These step correction calculations are very complex, however, they are easily calculated today with the aid of computers.

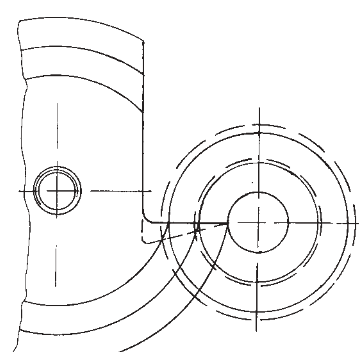


Figure #4

DOVETAIL FORM TOOLS

Figure 5 shows that in order to get the correct part diameter difference on dovetail tools, the step differential must be corrected, since "Y" is visibly shorter than "X". If the tool has zero top rake, the calculation is very simple, "Y" is equal to "X" times the Cosine of the front clearance angle.

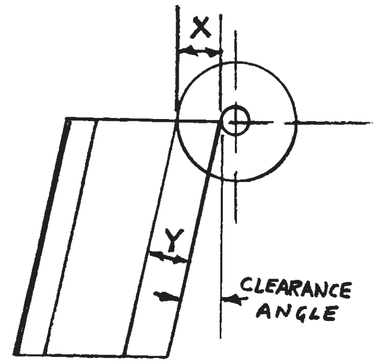


Figure #5

Figure 6 shows the effect of adding a top rake and also changing the smallest diameter that the tool is forming. As can be seen if two tools are making the same part and one tool is cutting to center and the other is cutting to a larger diameter, the tool step differentials will be different in order to maintain the same part diameters. As can be seen "A" and "B" are not the same as "A1" and "B1". Top rake step correction calculations are very complex, however, they are easily calculated today with the aid of computers. (See "FTSC Calculator")

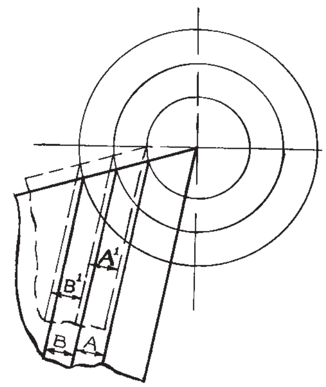


Figure #6

SHAVE TOOLS

For dovetail shave tools the front clearance angle is usually 0 degrees and the top rake angle is 0 to 5 degrees. These angles have very little effect on tool calculations so steps need not be corrected. See Figure 7.

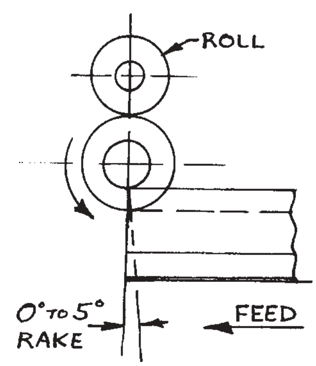


Figure #7

For Davenport circular shave tools, the cutting edge is offset from center and the specified top rake angle is about 5 degrees. However, we recommend 0 degree top rake because it has been our experience that 0 degree top rake produces more accurate workpiece diameters. Also, there is a substantial radial relief as the tool passes underneath the part because of the circular form of the tool. The cutting edge offset affects step differentials and, therefore, diameters must be corrected. See Figure 8.

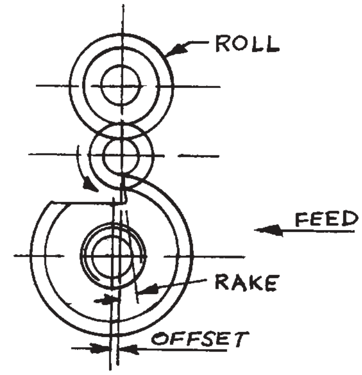


Figure #8