WICKMAN

MACHINE TROUBLESHOOTING.

Wickman Coventry Limited.
Unit A, Three Spires industrial Estate, Ibstock Road, Coventry CV6 6JR
Telephone +44 (0) 1203 644511 Spares +44 (0) 1203 644055 Facsimile +44 (0) 1203 644130/644069

FRESH EYES

When anyone has fought trouble about so long, he gets to the point where he couldn't see the solution if it were written on the wall. The only cure is to walk away, do something else long enough to regain perspective, or get someone else to take a look.

In either case, come back to the job as if it had never been seen before. Take nothing for granted. Run a group of consecutive parts and check them against the print, writing down the findings. Then, define in writing exactly what is wrong. Just this simple re-evaluation and restatement of the problem often points to an answer which has been overlooked. If so, it probably turns out to be some mistake or misunderstanding which needed only fresh eyes to be seen. In any event, we need written record of how the job behaved for comparison with results after changes have been made.

The problem in every case will fall into one of four categories:

1. TOOL LIFE
2. VARIATION
3. CONCENTRICITY
4. SURFACE FINISH

Of course, more than one trouble may be present, but the cause will be one or more of these seven variables:

1. SPINDLE SPEED
2. FEED
3. COOLANT
4. TOOLING
5. MACHINE
6. PART
7. HUMAN

In Table VIII-1 the four troubles are listed from left to right in the order in which they are most likely to occur. Beneath each trouble, in declining order of probability, are the seven possible causes. This table is the result of the writer's findings in more than 10 years of screw machine troubleshooting experience. It is suggested that once the trouble is known, the variables be examined in the order shown. Note the priority of variables differs from one problem to another.

TABLE VIII-1

Tool Life	Variation	Concentricity	Surface Finish
Spindle Speed	Machine	Machine	Machine
Feed	Feed	Tooling	Feed
Coolant	Tooling	Feed	Spindle Speed
Tooling	Part	Part	Tooling
Machine	Spindle Speed	Spindle Speed	Coolant
Part	Coolant	Coolant	Part
Human	Human	Human	Human

ARE YOU SURE?

Before we move into systematic analysis of cause, however, we need to know what conditions exist on the machine. By the time troubleshooting begins in earnest, one or several persons most likely have made a stab at fixing the condition, and in all probability no one is absolutely sure what's been done. So, immediately after pinpointing the problem, double check all seven variables, taking no one's word (least of all your own) for anything.

Determine personally, and note in writing that:

1. Spindle speed is in fact what it is thought to be, and that s.f.m. is reasonable for the material and the operation.

2. Feed is as called for, and motion is smooth and uniform throughout the stroke.

3. Coolant appears clean, is being applied properly, and is of a type recommended for the work.

4. The machine has no apparent defects needing attention.

5. Proper tool grades are being used, and tools have been properly ground to specified geometries.

6. The part print is fully understood, and its tolerances are within the capability of the equipment as set up.

7. The operator has a positive attitude, and adequate skill for the job.

Like the first step, this is a fact-gathering function, but here again things which have been overlooked or taken for granted may point to the answer. Although in this article we assume the task is to solve trouble without changing work material, we should during this initial study make sure the specified grade is in fact being used.

Note that to this point the method ignores what's-been-done, concentrating instead on what's-wrong, and where-we-are. This is important because if the job has been cobbled.

juggled and fooled-with, as is probably the case, we need to have written record of prior conditions and effect in order to judge results as changes are made. It is entirely possible that as one variable is brought into line error will show up elsewhere. Don't be discouraged by that. It is an indication counterbalancing is being eliminated.

CLUES

The effect of conditions always shows up on the three elements of a metal cutting operation: The part, the chip and the tool. An advantage of experience is that with it one recognizes the results which should be obtained with a given material cut at given s.f.m. and feed with a given shape and type of tool with a given coolant. That's why old-time troubleshooters look in the chip pan for answers to problems on a machine. If you can make good chips, chances are you can make acceptable parts, always assuming the machine is capable of holding the necessary tolerances.

Each type of steel has a characteristic chip. The free-machining grades should produce short, bright broken ones. Ductile grades yield long, stringy chips. The higher alloy, and work-hardening materials, such as 1137, 1141, and 1144, may give off one or the other of the above kinds of chips, but the difference is that there may be scabby adhesions on the cut side. All of this should be true if the job is tooled correctly and running at reasonable rate. The old-timer's look at the chips is just one form of analysis by exception; he's looking for conditions which do not fit the pattern.

But in our analysis we shall look at the other two elements of the operation as well: The tool and the workpiece. On the tool we check for discoloration, cratering, built-up edge (BUE) and point-of-wear. We look at the workpiece's cut surface under a glass to determine if the tool rubbed, if work-hardened scabs are present, or if the chip came away cleanly. In this study we're concerned primarily with BUE.

BUE

It should be kept in mind that a BUE is work metal, welded to the tool by heat, and/or pressure and has the same work hardening characteristics as the metal being cut. One of the factors which make leaded and resulfurized low carbon steels "free-machining" is that these additives reduce welding. Hence, in free-machining grades little if any BUE should be found. Tools used on those grades generally wear by rounding off. If craters are found, they're the result of hot chips rubbing across the face of the tool. This hot-rub effect can be seen when the tools are examined under magnification.

In contrast the high manganese and high carbon grades have excellent work hardening qualities, and since these are generally cut at lower speeds (hence lower tool temperature) BUE's tend to form. This is not always bad, because when we have a moderately sized BUE which stays on tool point it may actually prolong the life of the edge.

What we don't want, however, is a BUE which builds and breaks away carrying hardened bits of metal across the top rake or between the tool and the part. If this is occurring, the tool's crater will have a pitted appearance, since a bit of the tool is torn off each time the BUE is lost.

Such a condition will yield sporadic change in work finish and short tool life. It is an indication a different coolant, and possibly changed tool geometries, should be tried. Coolants with active sulfur or chlorine give the operation the anti-weld features not had in the material. Lapped edges, and surface treated tools will also help.

In carbides, BUE is most likely to occur at too-slow s.f.m. The solution is obvious, though not always easy to obtain when other tools limit the speed range. A tip here is when s.f.m. cannot be increased, use a mineral base coolant instead of a water soluble.

With that background, let us consider what is involved in curing trouble in each of the four conditions shown in Table VIII-1, beginning with Tool Life.

TOOL LIFE TROUBLE

"Spindle Speed" is listed as the most likely cause, since speed creates heat, and heat is the most destructive force acting on tools. Indications speed is too high for conditions are when corners of forms, turning tools or cutoffs burn, or when the peripheries of drills, taps or reamers discolor.

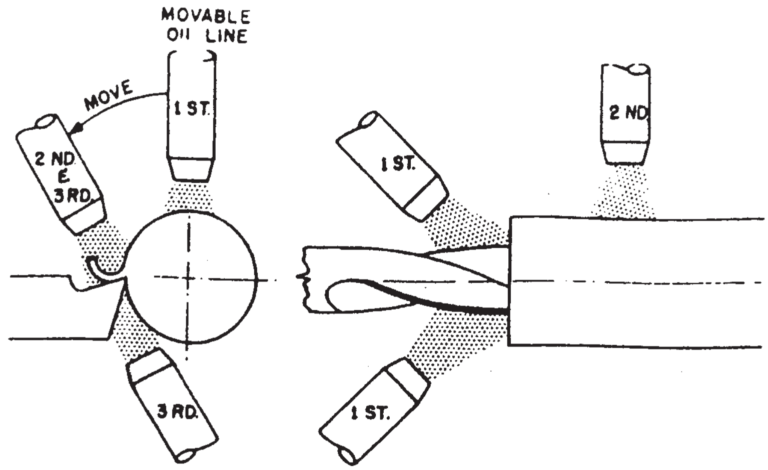
In short, the effect is seen where s.f.m. is greatest, or where the tool has least mass, hence least heat-carrying ability. If speed is too slow, unstable BUE's will be seen and wear will show up either as a rubbed condition on tool flanks, or cratering. Note that wear can result from too-slow speed as easily as from excessive speed.

Next is "Feed." If drills split down the middle, or edges chip without discoloring, or large unstable BUE's are seen, feed per revolution should be lowered. An exception to this is when a work hardening material is being machined and the equipment is somewhat loose. In this case, feed should be increased to keep the tool from rubbing and work hardening the surface. The tool must not rub in work hardening grades. Keep it cutting by whatever means is required.

Then we check on "Coolant." A large BUE indicates need for change to a fluid with greater anti-weld properties. If flank wear is seen, the coolant may be overly fortified. The answer: Switch to a thinner fluid with greater oiling action.

Let's pause here to consider what the coolant does and how it should be applied. Coolant (1) carries off heat from the tools and the part, (2) provides an oiling action to the cut and at times puts an anti-weld barrier between tool and material, and (3) flushes away chips. As shown in Figure VIII-2, these actions are enhanced by placement of the oil lines. Quite frequently an extra line will reduce BUE which is causing a tool to chatter. This is particularly helpful in wide forming operations from the cross slide.

FIG. VIII-2



When drilling, aim two lines up the hole, and another over the outside to carry off heat. The thinner the part's wall, the more important the outside line will be, and in deep, thin-wall work where there is little chance of getting much fluid to the drill point, copious amounts of water solubles flowed onto the part's O.D. often are helpful.

Next we investigate "Tooling." Here we shall assume the initial check on variable showed a proper grade of tool is being used. Cratering is most apt to be seen when cutting soft gummy materials, or high strength grades, for the reason mentioned earlier. In either case, smoother grinds and higher rake angles reduce cratering to manageable limits.

If heat craters are seen, it may be worthwhile to consider a shift to T-5 or T-15 H.S.S. which has greater hot hardness and abrasion resistance. In free-machining steels, little or no BUE should be evident if rake is proper for conditions. Remember, we don't want to eliminate BUE altogether, since it protects the cutting edge, but we do want it to be small and stable.

At times BUE's form and wipe off at the end of the cut. By using a 10X glass or better, and seeing how far back the crater extends, you can make a good guess at how much buildup is taking place.

A certain amount of flank wear is normal. Excessive wear on flanks is usually caused by too-little clearance. Increase clearance. If the condition persists, consider a switch to a tool having higher vanadium. When sides of form tools wear excessively, try to eliminate sharp corners. A 1/32 by 45 degree break will strengthen the edge of forms used on work hardening steels; in others, a small radius may work best, and this, of course, applies to all tools where it can be used.

In evaluating tools for sources of trouble, one must be careful to differentiate between actual wear and breakdown of the edge due to fatigue. The latter is microscopic chipping which can sometimes be noted under magnification. The solution is to first strengthen the edge by reducing rake and clearance. If that doesn't work, switch to a less brittle grade of tool.

Hard-to-cut steels require greater side clearances since they tend to spring back behind the edge. The tipoff on this is when sides of cleared tools gall or bug.

A few miscellaneous tips: Try not to follow any cut with another tool of identical shape, especially on work hardening materials. Make absolutely certain tools are not burned in grinding. Tool life often can be doubled by polishing and honing all edges. A high shine on the part generally indicates tools are rubbing, and the most frequent cause is a cam with too much dwell at the top of the rise. Worn cams and work hardening steels do not go together. Also make sure cross slide stops are not set overtight, since this can cause tool dwell also. Assuming the tool and the part are adequately supported, the most frequent causes of chatter are too much clearance or too great radius on the tool's edge.

The next variable to be studied is "Machine." Note it ranks fifth as a cause of poor tool life, but first in the three other types of trouble. The reason is that the machine contributes here by default . . . . . failure to provide sufficient rigidity or smoothness of motion which allows tools to rub, or chatter. This will be noted most in brittle tools, such as carbide or ceramics. The effect of worn cams on work hardening steels has been mentioned; loose pins, linkages, slides or spindles will give similar result. If possible, snug up the machine; if not, increase feed and lower speed to keep the tool constantly under pressure.

Let us now consider "Part" as a factor in tool life. The shape of the workpiece, and its tolerances, dictate the type of tools required. If trouble exists, check with Engineering to find out if slight angles are allowable on deeply cut straight sides, or if sharp corners are necessary.

Necks less than three threads wide almost always will reduce chaser or tap life by making shorter lead chamfers necessary. My rule for side cutting angles on forms is: In free-machining steels, 1/2 to three degrees positive; in ductile steels, five degrees, and in work hardening grades, eight to 10 degrees. It may not be possible to change part shape, but it's an area worth exploring.

Finally, we look at the "Human" factor, which ranks last in all four types of trouble. The reason is that if the man on the job is grossly unable or unwilling to handle the work, this should have shown up in the preliminary check on the seven possible variables. Nevertheless, spending time with the operator and explaining the reasons why the job must be done a certain way may give him a better grasp of the function, let him use his experience to further the desired result, and will give him a more positive attitude toward the job.

Somewhere within the definition of the problem, the preliminary doublecheck on conditions, or the more detailed analysis of the seven causes, the answer to tool life problems will be found.

As shown in Table VIII-1, the causes of dimension-to-dimension, or piece-to-piece changes in product are, in order of probability, Machine, Feed, Tooling, Part, Spindle Speed, Coolant, and Human.

"Machine" is the most likely troublemaker because, quite obviously, if tools were carried a like distance on each stroke and forced into the cut with equal pressure, parts should be alike. Something within the machine probably is allowing this sameness of motion to be disturbed. On a multiple-spindle automatic, variations in length, part-to-part, indicate end play in spindles, or erratic stock feeding; possibly "bounce" off the stock stop. On single-spindle models, length variation can also be caused by the foregoing, along with worn lead cam rollers and pins.

On either type, part-to-part changes in formed dimensions generally indicate bad spindles, faulty cross slide systems, or improperly set stop screws. This type of trouble is spotted on multiples by recording the dimensions of the parts from each spindle. Of course, if the problem is that simple, it should have been noted in the initial problem-finding stage. Even so, a graphed picture of a larger group of parts is a help, and may show relationships between dimensional variances which point to the cause in the most sticky types of trouble, the cases where a part is out of tolerance "just once in a while."

When large diameter stock is run in an older machine, the weight of 12-foot bars may cause deflection. To determine if this is the cause, cut bars in half or check the job when bars have been partially used up.

Anything which encourages vibration will affect tolerances. Snug up slide gibs, replace rollers, check end play, stop screw tensions, and make sure tools are solidly mounted and cams are tight.

The next most likely cause of variation is "Feed." Its chief effect is to increase endwise and sidewise pressures, hence if the machine is known to be somewhat sloppy, lower feed may give best results. Remember, though, as pointed out previously, that with work hardening steels, tools must have sufficient feed so they do not rub. We can also decrease feed per revolution by running at higher r.p.m. In other words, the pushing effect of feed can be minimized without lengthening cycle time in many cases. As a general policy, however, very close dimensional limits usually require feeds to be rather light for finishing cuts.

Next we turn to the "Tooling" variable. This is a factor in dimensional variation whenever tools are dull, chipped or incorrectly set or cleared. The effect is much the same as too-heavy feed: Cutting pressures are exerted which may be greater than the machine can take without deflecting. As was mentioned, watch for BUE's which build up and wipe off. Also make sure tools are not burned in grinding, since burned tools dull faster.

Tools which are grouped closely together are a likely source of the problem when variation occurs "just once in awhile." If there is not enough room for chips to fall free, they may pack and hold back a slide, or prevent oil from reaching tool points. The cure is to rearrange tools, or to give them chip-breakers so packing does not occur.

In considering "Part" as a variable, it must be recognized that if tolerances were wide open there would be no variation problem. This is just another way of saying the closer the limits the more chances of producing parts which fall outside them. As previously discussed, work with Engineering to get too-tight limits opened up, if possible, or tricky configurations changed to less troublesome ones.

While "Spindle Speed" ranked first as a cause of poor tool life, it stands fifth as a probable source of product variation. If the part is long, and must extend a great distance from the collet, lower speeds may cut down vibration and whip. It is more likely, though, that if this condition exists to a hindering degree, the part should be given extra outboard support.

"Coolant" comes sixth. By rearranging lines or adding high velocity lines aimed near the point of cutting, but not on it, it may be possible to flush away chips which would otherwise pack and cause variation. Of course, by keeping tools sharp longer a good coolant helps produce more uniform work.

Finally, the "Human" factor. No one is in better position to spot trouble in its early stages than the man at the machine. He should be instructed to report abnormal conditions, set tools and slides properly, and in general know if his equipment "acts right."

CONCENTRICITY

For trouble of this sort we investigate variables in the following order: Machine, Tooling, Feed, Part, Spindle Speed, Coolant and Human. The first four are by far the most important; the remaining three incidental.

"Machine" comes first because concentricity . . . . . or rather, the lack of it . . . . . almost always involves misalignment of tools to workpieces. You simply cannot drill, ream or bore a hole concentric to an outside diameter if the spindle is sloppy. On multiples, check for spindle carrier lockup.

In every case go over the slides, the spindle, and every element which affects tool-work rotational relationship. When a machine which has been on one job a long time is changed over to one of different length, a sudden show-up of eccentric conditions may be due to the slide having worn in one spot.

There is, of course, the possibility that hole eccentricity originates in a wobbling drill. However, if the problem is greater, check the machine with a dial indicator.

"Tooling" as a cause of eccentric conditions is most apt to show up in the hole. Here it should be said that while the initial definition of the problem should have made clear which dimensions were out of olerance, it is not always easy to determine whether an O.D. is eccentric to a hole or the hole is eccentric to the outside of the part.

As an example, consider a simple collar, formed all over to one size, and having a straight through hole. Assuming both I.D. and O.D. are found to be round, which is eccentric? Nine times out of 10 it will be the hole, and the tipoff is that O.D. chamfers are true, while I.D. chamfers are not. As good a way as any to tell what is wrong is to shut off the coolant and watch a part finished but not yet cut from the bar as it rotates. In this way hole wobble or O.D. runout can be seen at once. The best way to make this decision is, of course, to use the indicator on the part before it is parted from the bar. If a hole is off center, check the drill point first. It may be chipped, ground off center, or improperly thinned. Regardless of what anyone says, drills used in steel must be thinned to about 1/10th of the drill diameter for drills one inch or larger, 1/15th for 1/2-inch drills, and 1/20th for 1/8-inch drills.

Then there's the matter of lip clearance. In harder steels, clear about seven degrees: in free-machining grades, try 10 to 12 degree clearances. In harder steels, make the included angle of the point blunter. The drill should be choked as short as possible to avoid runout. Watch, too, for small side tools, such as knee turners, facers or hole chamferers which may be pushing a drill off center, or allowing chips to pack between the two. Many times it will be found that side forming, knurling or thread rolling tools are pushing the part away from its axis, causing drill or reamer eccentricity.

When drills follow each other, eccentricity will be least if each succeeding drill point has a slightly larger included angle. This lets the drill center itself properly instead of getting a bad start by having the point start first at the preceding drill's web section.

It has been my finding that more often hole concentricity is related to how freely a drill cuts, and how well it is centered in the holder.

"Feed" affects concentricity only when i.p.r. exceeds the rate which the tool can handle without crowding. "Part" is a troublemaking variable in concentricity when tolerances begin to approach the inherent eccentricity of the spindle-collet-workpiece package.

As mentioned at the start, the other three variables, Spindle Speed, Coolant and Human have little effect on concentricity, except to the extent that they may occasionally interact with one of the other four. We move to the fourth and final trouble, Surface Finish.

SURFACE FINISH

As shown in Table VIII 1, when the problem is unacceptable finish on the part, the variables rank in different order as potential trouble sources. In this case: Machine, Feed, Spindle Speed, Tooling, Coolant, Part and Human, respectively.

"Machine" is first because of the need for rigidity. Look and listen for signs and sounds of chatter. Do not change any other variables in an effort to improve finish until absolutely sure the machine has bee snugged up as well as possible.

"Feed" has an effect mainly because a turned surface is for all practical purposes a thread whose pitch is equivalent to feed per revolution, and if that thread is to be blurred into a smooth-looking finish, its "pitch" must be very short.

"Spindle Speed," when increased, will help us obtain a smoother finish mainly by reducing the BUE. In general, smoother finish calls for running faster while taking light cuts at low i.p.r. feed rates. There's a limit to this, of course: At the point where tool edges start to burn, or wear rapidly, the advantages of higher s.f.m. are gone.

"Tooling" affects finish in a number of ways, through rigidity or the lack of it, through type of tool and manner of tool grinding, and through the use of adequately cleared yet well-supported cutting edges. The sequence is also important. When trouble develops, see what can be done to take the load off the finishing tool, and if possible, give it a position all its own. This is particularly worth considering when longitudinal turning or boring cuts are taken because side pressure during these operations almost certainly will cause poorer finish.

When a shaving tool creates the finish, strive for .0015 to .0025-inch of material on a side when r.m.s. is critical. More than that will lead to BUE, which in turn may cause tearing, or chatter.

In drilling and reaming, check widths of lands. In work hardening grades, leave as little margin as possible. For instance, on a 1/2 to one-inch reamer, no more than .010-inch uncleared margin is best if galling is to be avoided. We know that reducing chip thickness . . . . . in effect reducing feed . . . . . gives better finish. This can be accomplished on a reamer by putting a secondary chamfer of five to 15 degrees behind the standard chamfer of 38 to 45-degrees; i.e., 10 to 15 degrees for the work hardening steels, and under 10 degrees for other steels.

"Coolant" is a factor in avoiding rubbing, and in reducing BUE. Some oil men go into much detail on the mechanics of improving finish with cutting fluids. It is our finding that if the situation is sufficiently critical to require a change of coolant, an oil supplier should be called on for recommendations and perhaps in-shop assistance.

Finally, the "Human" element. Some men take great pride in finish: others couldn't care less. Look for the man who has a feel for properly ground and correctly set tools, and put him on the jobs where low r.m.s. must be achieved.

SUMMARY

As we close, let's recap the analyzing method described in this article. Step 1 is to find out, first hand, precisely what the problem is. Step 2 is to doublecheck all seven variables for obvious mistakes or misunderstandings and to make sure we have all the facts.

In both cases, findings should be written out for future reference. Step 3 is to check the variables in the order given under each type of problem in Table VIII-1. As we check, records should be kept.

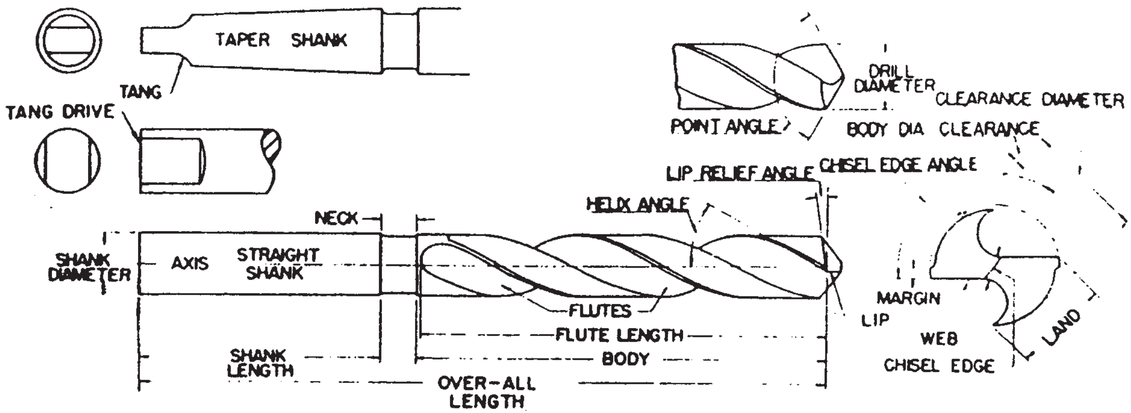
Somewhere along the way, the problem will begin to come apart. As this occurs, the fault of the variable is corrected before proceeding.

Aim always for the true cause, rather than just trying to adjust what seems to be wrong.

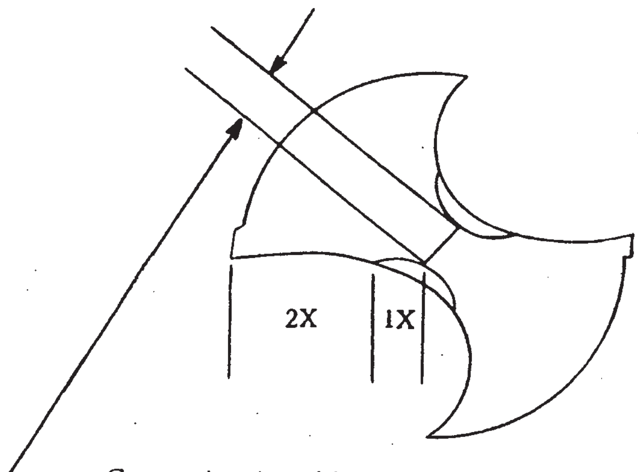
As mentioned, when you're stumped, when you've tried everything, stop. Get away from the problem for a while so you can come back with a fresh outlook. Perhaps you've just worked on it so long you've begun to miss the same step every time.

An analysis of this sort can't be done in minutes, nor is it necessary in the easier cases. But when trouble is chronic, or when tolerances are so tight everything has to be right, a systematic approach is better, and in the long run cheaper, than running the risk of introducing new error to offset what's really wrong with the job.

FIG. VIII-3: Terms applying to twist drills

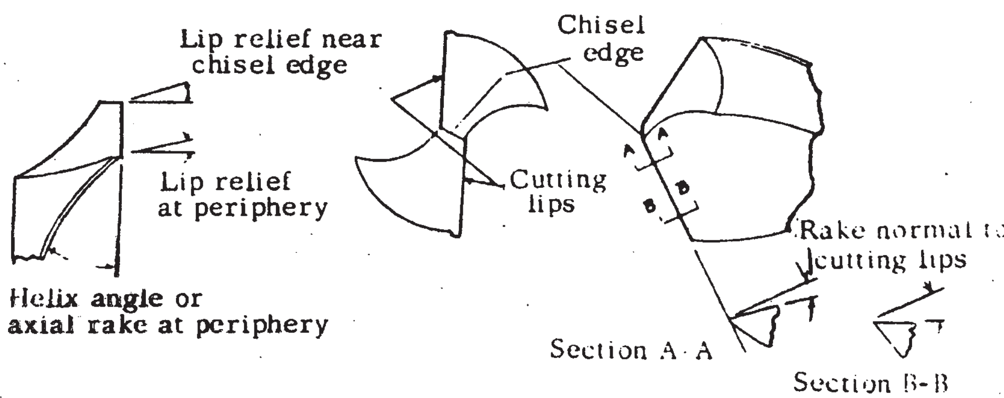


(Illustration courtesy of Metal Cutting Tool Institute, New York, N.Y.)



Drill Diameter	Ratio
1/8	20:1
1/2	15:1
1	10:1

General point-thinning grind for steel.
Width is in relation to drill diameter.



(Illustration courtesy of Metal Cutting Tool Institute, New York, N.Y.)

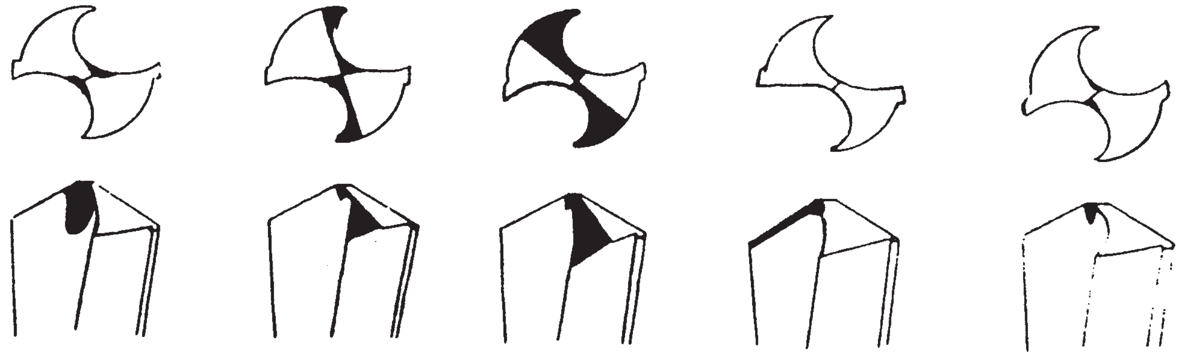
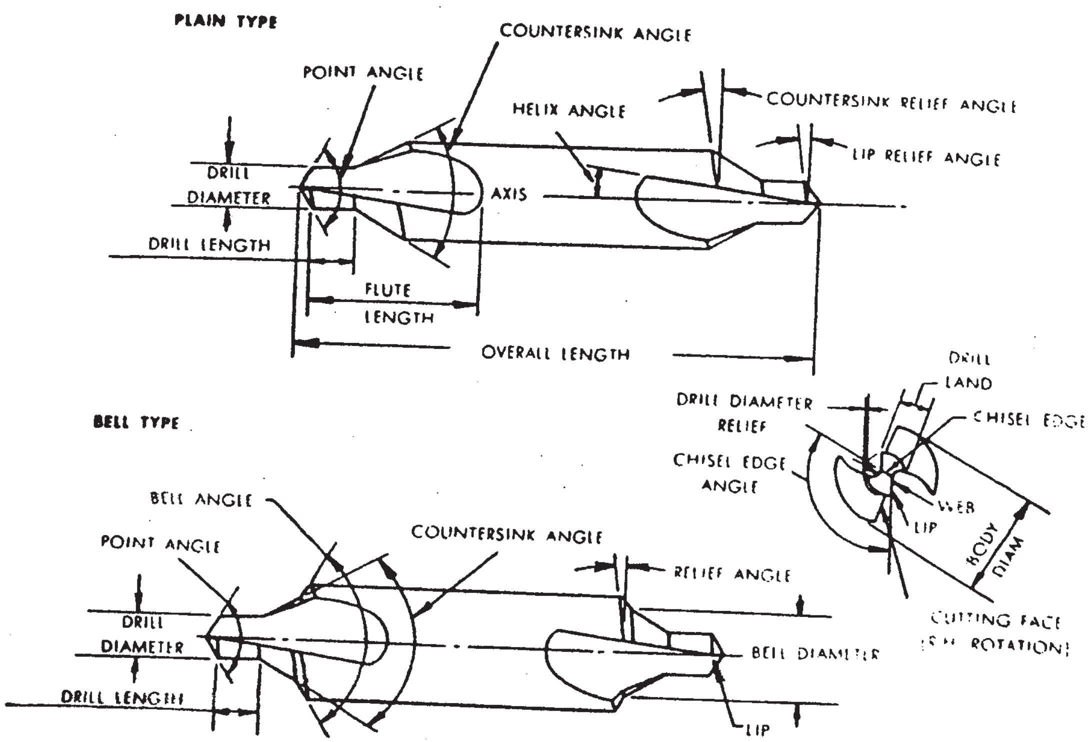


FIG. VIII-4: Terms applying to combined drills and countersinks



(Illustration courtesy of Metal Cutting Tool Institute, New York, N.Y.)

FIG. VIII-5: Checking the lip angle

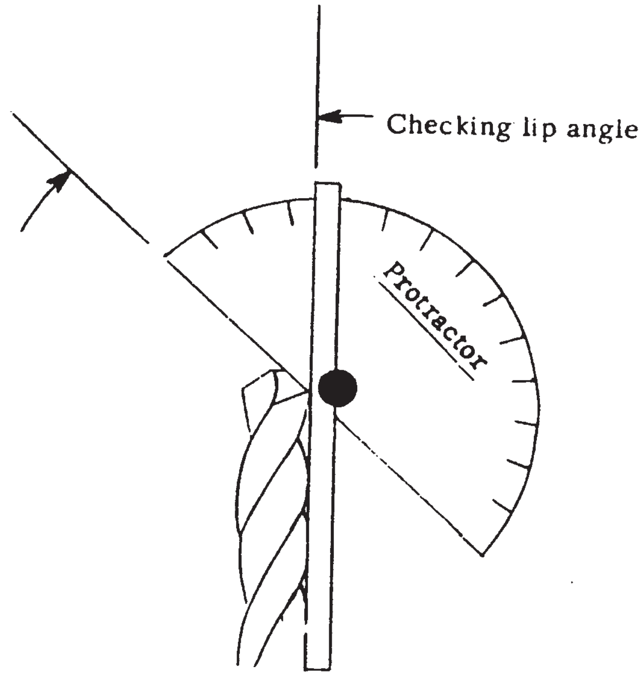
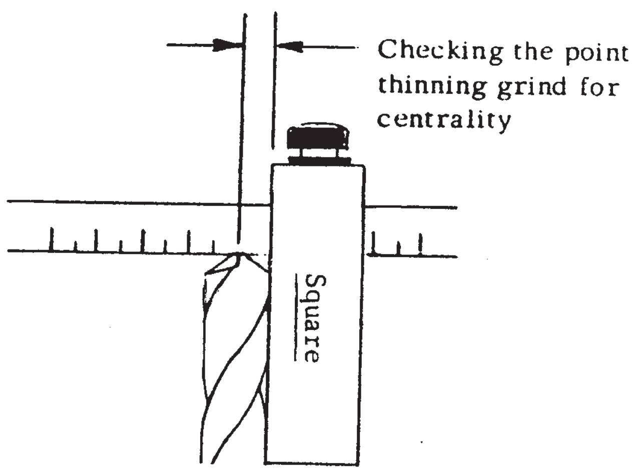
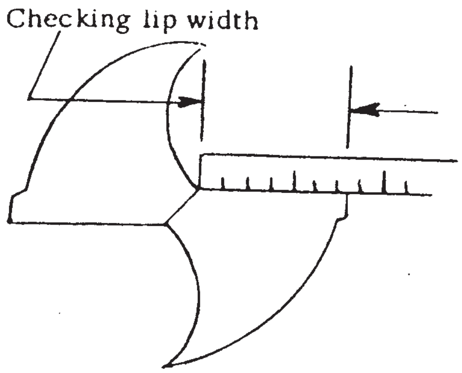


FIG. VIII-6: Checking the point thinning grind for centrality





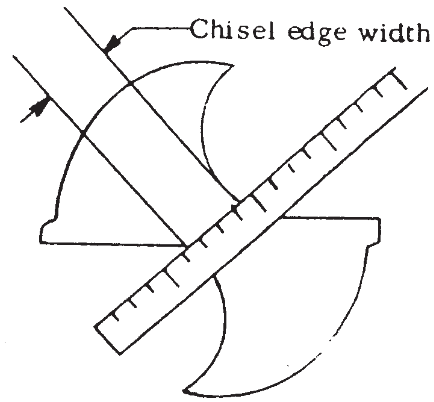
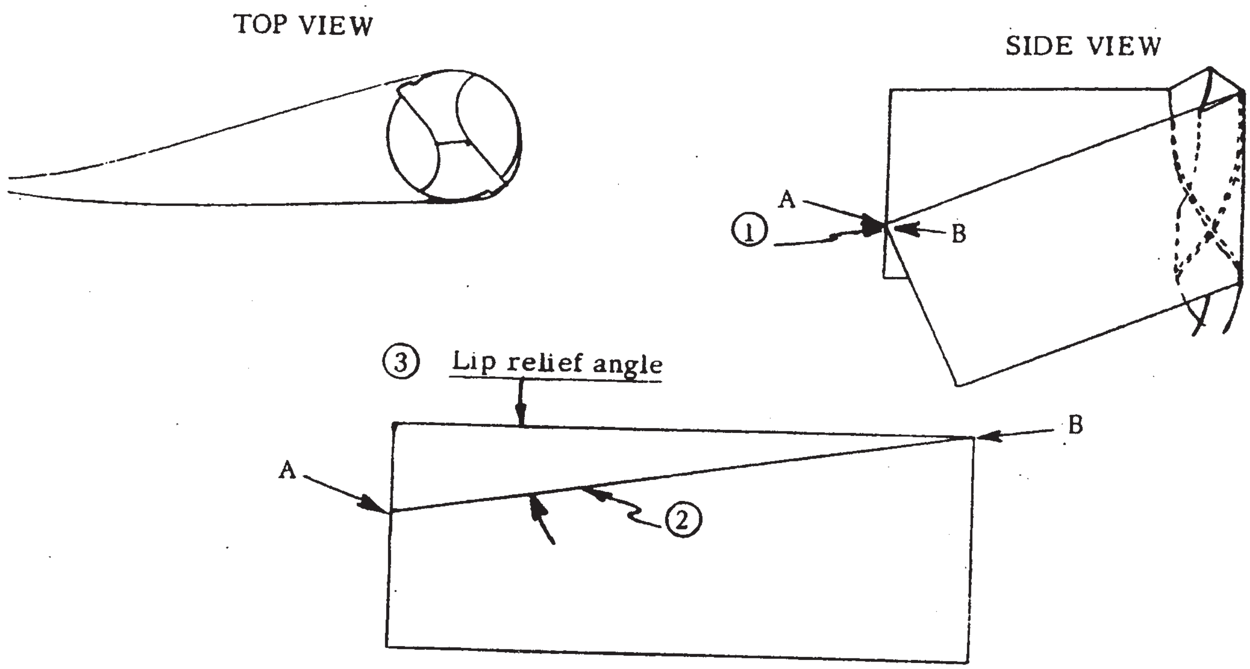


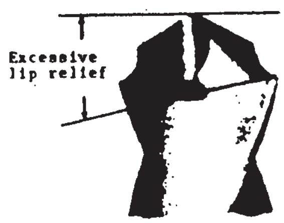
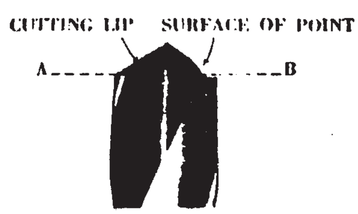
FIG. VIII-8: Checking lip relief with paper



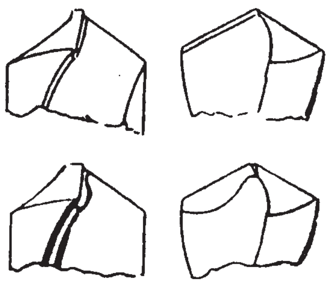
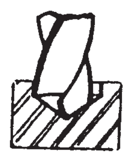
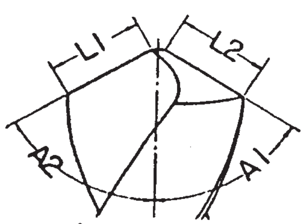
① Mark where point of top paper (B) touches edge of bottom paper (A).

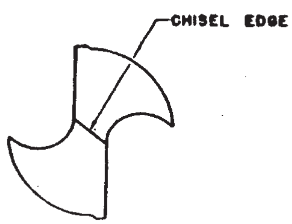
② Open paper, lay flat, and draw a diagonal line from A to B.

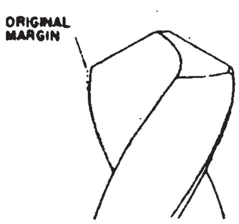
③ Read the lip relief angle as indicated.

PROBLEM	CAUSE	CORRECTION
Drill chips on cutting lip edge Showing results of giving a drill too great lip-relief – the edges of the cutting lips have broken down because of insufficient support.  (Metal Cutting Tool Inst.)	Cutting lip weakened by excess lip relief angle	Reduce lip relief angle
	Feed too heavy	Reduce feed
	Unequal grind of lip width or web (drill point)	Regrind
	Drill too far out of holder causing vibration	Reset drill further back in holder
	Rake angle too small	Increase rake by grinding or use a faster helix drill
Drill chips on outer corners of cutting lips at margin (A)  (Metal Cutting Tool Inst.)	Spindle speed (r. p. m.) too high	Reduce speed
	Insufficient rigidity	Snug up drill. holder. machine, etc.
Drill breaks  A drill point without any lip clearance. Note that corners of cutting lip A and of heel B are in the same plane (Cleveland Twist Drill Co.)	Lip relief angle too small	Increase lip relief angle
	Dull drill	Regrind
	Spindle speed (r. p. m.) low in proportion to feed	Increase speed or decrease feed
	Drill seizing	When using several drills to attain hole depth, each succeeding drill should be .001/.002" smaller in diameter

(continued)

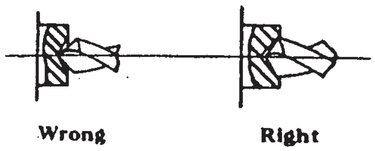
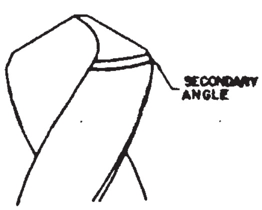
PROBLEM	CAUSE	CORRECTION
Drill breaks (contd.)  Two types of chip breakers (Metal Cutting Tool Institute)	Chips clog flutes	Increase feed; decrease rake angle; widen flute by grinding; grind chip breaker on lip; use chip breaker drill; change point angle; use oil hole drill; increase coolant pressure; change position of coolant line lines
	Side pressure caused by another tool working in conjunction with drill	Change location or sequence of this tool; add a support to work
 Incorrect Point. Lips of unequal length but at equal angles	Drill plunging into work	Broken or loose brake spring; cam rise too short; check, reset low speed dog or index dog
(Metal Cutting Tool Inst.)	Point grind not equal causing excess load on one lip or drill to walk off center	Regrind
	Drill protruding too far out of holder	Reset drill further back in holder
Angles and lengths of cutting lips must be equal	Spindle loose	Repair or adjust spindle
	(Illustration courtesy of Cleveland Twist Drill Company)	
Drill splits up center 	Chisel edge or web too wide	Reduce thickness
	Feed too heavy	Reduce feed
(Metal Cutting Tool Inst.)	Lip relief angle too small	Increase lip relief angle

PROBLEM	CAUSE	CORRECTION
Extreme outer corners of drill wear rapidly	Spindle speed (r.p.m.) too high	Reduce speed
 The Indication of Too Great Speed. The outer corners of the drill have worn away rapidly because excessive speed has drawn the temper.	Rake angle too great	Reduce rake or use slower helix drill
	Insufficient coolant	Assist heat removal by adding an extra line on drill or on workpiece
(Metal Cutting Tool Institute)	Drill burned when sharpened	Regrind making sure to remove all burned portions
Drill fails to bite into work	Lip relief angle too small	Increase lip relief angle
 Properway to grind lip-clearance. The angle indicated is the angle at the circumference of the drill	Work hardening of material or surface by previous tool/tools	Regrind previous tool, check, reset dwell to a minimum
	Drill dull	Regrind
 Point of drill after drill has been cut back in size and repointed	Web or chisel edge too wide	Reduce web or chisel edge width
	Feed per revolution too low	Increase feed
	End play in spindle	Check, adjust spindle
	(Illustrations courtesy of Cleveland Twist Drill Company)	
Drill has a tendency to pull ahead during cut	Rake or helix angle too high	Reduce rake by grinding flat along drill lip: use slower helix drill
	End play in spindle	Check and tighten up spindle

PROBLEM	CAUSE	CORRECTION
Drill requires heavy feeding pressure throughout cut  The section on the left was cut from a drill near the point while the section on the right was cut near the shank. The difference in the thickness of the web at these two points is shown by the length of the white lines between the two sections in the illustration.	Chisel edge or web too wide	Decrease width of chisel edge or web
	Included point angle too large	Use less included point angle
	Lip relief angle too small	Increase lip relief angle
Drill cuts tapered hole  Incorrect Point. Lips of unequal length but at equal angles	Misalignment of drill	Realign drill
	Bent drill	Replace drill
	Drilled hole diameter reduced by subsequent (usually cross slide) operation	Check and reset subsequent operation tools that are causing the problem
	Lips not equal	Regrind
	Web not central	Regrind
Drill cuts undersize 	Worn margin on drill	Remove worn area or change drill
	Insufficient or improper coolant	Increase amount of coolant and use correct grade
	Drill O.D. small	Replace drill; 'mike' O.D.

(Cleveland Twist Drill Co.)

PROBLEM	CAUSE	CORRECTION
Drill cuts oversize	Lips not equal	Regrind
 Incorrect Point. Lips of unequal length and at unequal angles (Cleveland Twist Drill Co.)	Chisel edge not central	Regrind and check 90° from the lip
	Drill out of line	Realign drill
	Machine spindles loose	Adjust spindles
	Drill clamped on margin causing axial misalignment	Clamp on shank; use screw machine stub drills
	Dull or incorrectly set tool working in conjunction with drill causing side pressure	Check, regrind or reset problem tool - change sequence of operation
	Drill oversize	Change to proper size; "mike" O.D.
Drill cutting eccentric to axis of part	Drill lips unequal	Regrind
	Web or chisel edge not central	Regrind
	Drill bent	Replace
	Drill not aligned properly	Realign drill; if problem persists replace drill holder, collet or bushing and do not clamp on margin
(continued)	Side pressure caused by another tool	Relocate other tool or lower feeds and increase rake on other tool; add a support

PROBLEM	CAUSE	CORRECTION
Drill cutting eccentric to axis of part (contd.)	Spindle loose or bearings worn	Tighten up spindle if possible
 (Brown & Sharpe Mfg. Co.)	Drill's included point angle is smaller than previous drill or countersink	Regrind drill making sure its included point angle is at least 2° more than the previous drill
Rough finish on hole	Drill is dull on cutting lips or margin	Regrind and remove all worn areas or replace
	Incorrect or insufficient coolant	Use proper coolant in copious amounts; use oil hole drill
	Welding on margin	Reduce width of margin
	Drill not in alignment, causing rubbing on margin	Realign drill; replace bushing; check holder alignment with spindle
	Chips clogging flutes	Use chip breaker grind; use oil hole drill: increase feed to obtain better chip shape
	Incorrect grind	Regrind and try a small 10° angle .010 / .020" wide on the outside corners at the margin of the drill
(Cleveland Twist Drill Co.)	Drill margin dragging along a previously cut hole of same diameter	When using several drills to attain hole depth, each succeeding drill should be about .001/.002" smaller in diameter.

NOTE: Check with drill manufacturers' recommendations and handbooks.

FIG. VIII-9: Terms appl ng to reamers

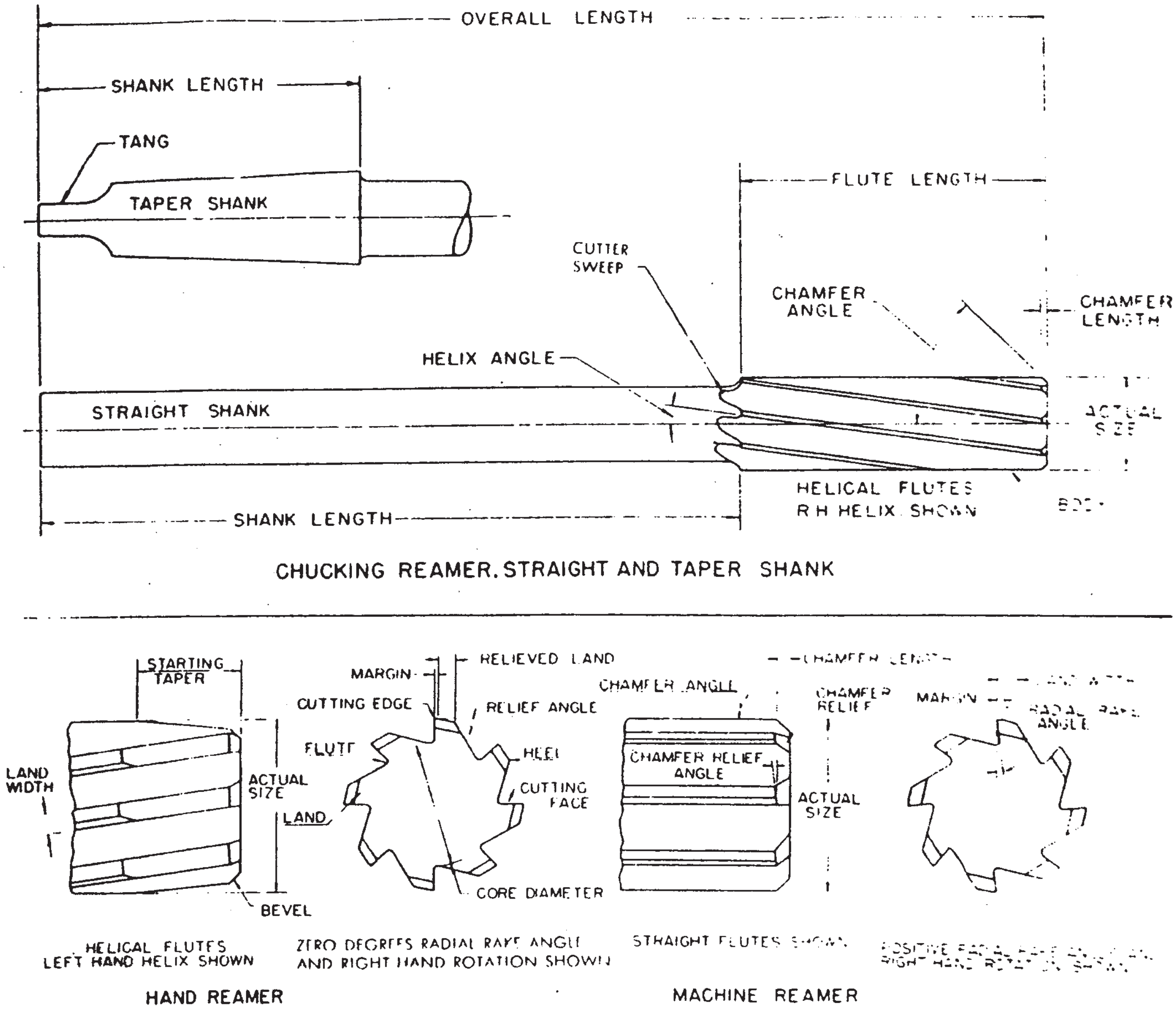


FIG. VIII-10: Stub screw machine reamer with straight shank and cross pin hole for pin drive floating holders

(Illustrations courtesy of Metal Cutting Tool Institute, New York, N. Y.)

FIG. VIII-11: Two methods of grinding the chamfer on a reamer for producing finish

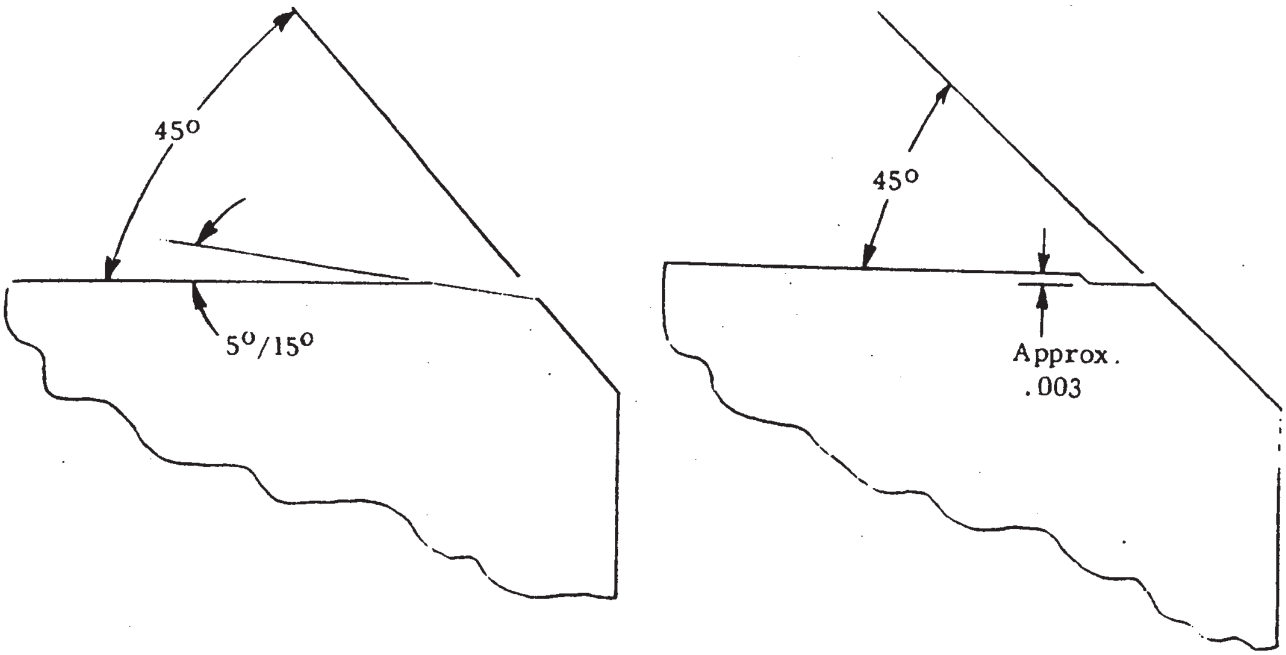


FIG. VIII-11a: Types of reamers

Straight flute, right hand cut 

Rose, straight flute, right hand cut 

Left hand helical flute, right hand cut 

Right hand helical flute, right hand cut 

Taper pipe, spiral flute 

Shell Reamers - High Speed

Straight flute 

Helical flute 

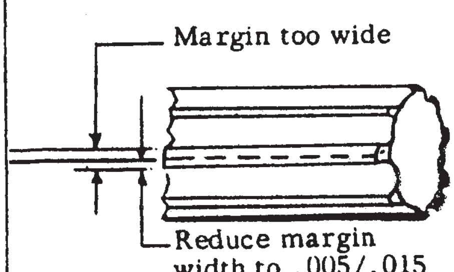
Arbors for Shell Reamers

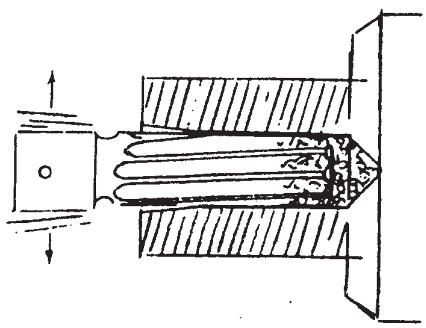
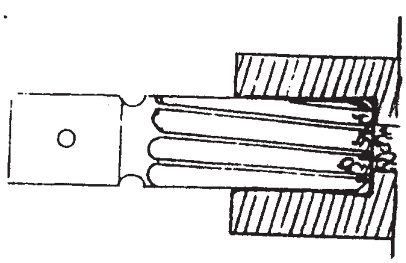
 Straight shank

 Taper shank

(Illustrations courtesy of The Cleveland Twist Drill Co., Cleveland, Ohio)

PROBLEM	CAUSE	CORRECTION
Chipping of cutting edge	Insufficient rigidity (vibration)	Use less float
	Feed per revolution heavy	Reduce feed
Chatter	Spindle speed too high	Reduce spindle speed
	Reamer extended too far out of holder	Use shorter reamer or move reamer back in holder to reduce overhang
	Poor start because of insufficient or no chamfer	Chamfer front of hole to "lead" reamer
	Vibration	Too much float in holder: adjust or change holder
	Straight fluted reamer being used	Change to a spiral type, preferably right - hand cut - left-hand spiral or left-hand cut - right-hand spiral
	Drilled hole too close to reamer size	Use smaller drill
	Excessive chamfer relief	Reduce chamfer relief
	Removing too much material in proportion to feed	Use larger drill or slower feed
	Unequal grind of chamfer	Regrind to correct
	Margin too wide	Reduce width of margin to .005/.015"



PROBLEM	CAUSE	CORRECTION
Hole not round	Straight fluted reamer being used and cutting a hole smaller than the reamer diameter	Change to a spiral fluted reamer or a reamer that has diametrically unequal cutting edges
Bell mouth holes	Incorrect alignment	Realign reamer; replace bushing; check holder alignment with spindle
	Reamer hitting bottom of hole	Move back into holder
	Poor start	Check concentricity of starting chamfer or add a chamfer if there is none
	Drilled hole too small causing excess removal of material	Increase drill or bore size
	Badly drilled hole	Resharpen and/or realign drill
	Reamer chamfer ground unequally	Regrind to correct
Bell bottom holes	Reamer striking bottom	Move back into holder
	Misalignment of reamer or holder	Realign reamer
	Drill cutting large at bottom of hole	Regrind and reset drill
	Chips fouling at bottom of hole	Use oil hole reamer; change grind; increase drill size to cut down on chips; use spiral reamer

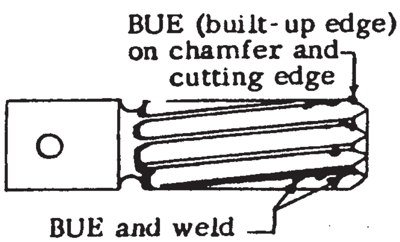
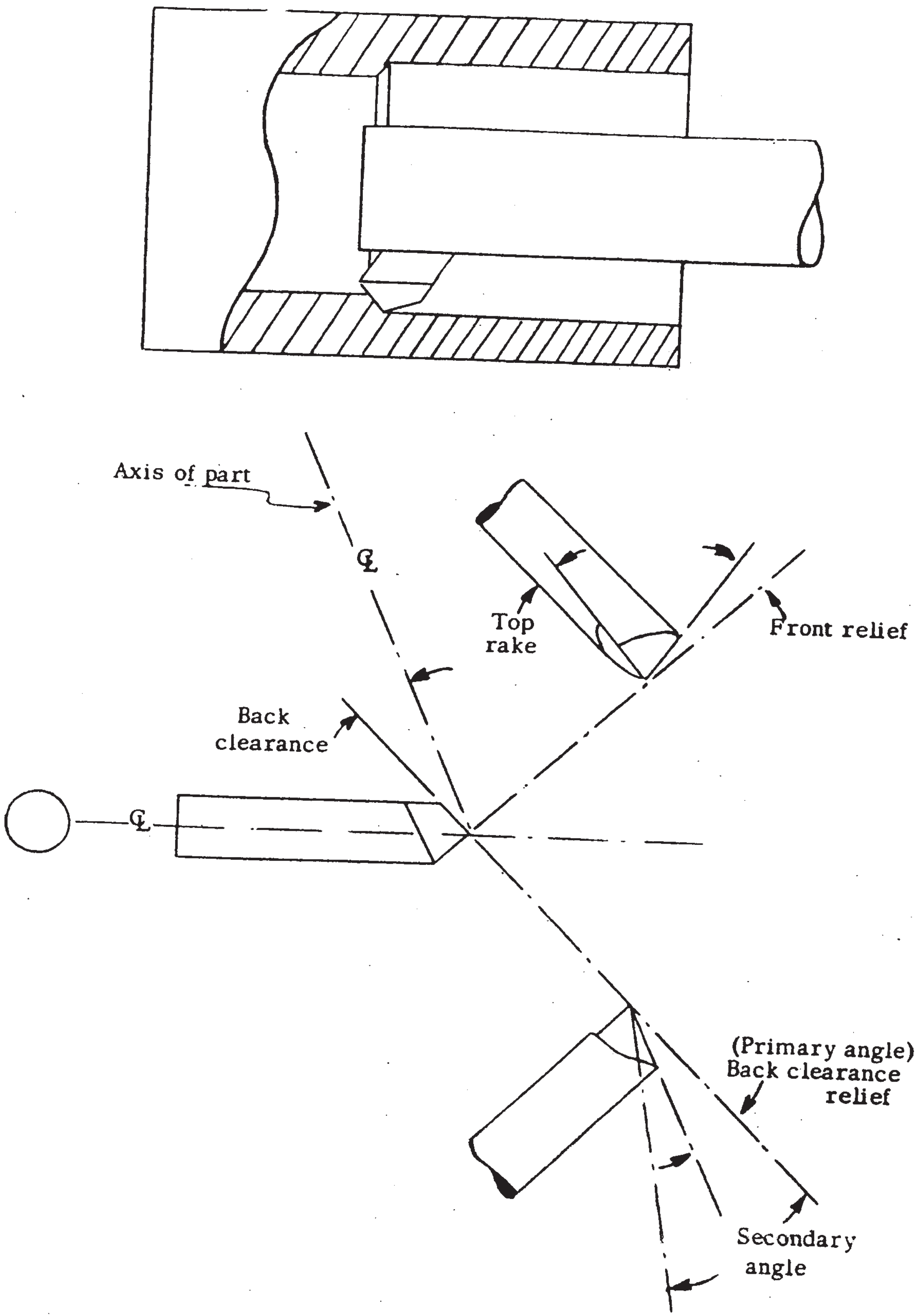
PROBLEM	CAUSE	CORRECTION
Tapered hole	Reamer out of alignment	Adjust holder to realign reamer; check for bent reamer or holder
	Hole squeezed by subsequent operation	Check subsequent operation
	Reamer worn	Replace reamer or remove worn portion and regrind chamfer
Rough hole	Removing too much material	Increase drill or bore size
	Too much or not enough float in holder	Readjust float
	Welding of material on margin	Reduce width of margin
	Built-up edge on cutting chamfer	Increase angle of chamfer; increase rake
	Improper coolants	Use proper coolant for material from a finish standpoint
	Not enough coolant	Add an extra coolant line; use an oil hole reamer
	Chips packing in flute	Feed too light; drill size too small; not enough coolant; use oil hole reamer
	Built-up edge along flute side of margin	Realign reamer; increase radial rake
	Reamer bent or out of alignment	Replace or realign
	Burrs or nicks on reamer	Remove nicks or replace reamer

FIG. VIII-12


Axis of part
℄
Top
rake
Front relief
Back
clearance
℄
(Primary angle)
Back clearance
relief
Secondary
angle

PROBLEM	CAUSE	CORRECTION
Tool breaks	Excessive clearance	Reduce clearance of primary angle
	Feed per revolution high	Reduce feed or increase speed if possible
Tool wears	Speed too high	Reduce speed
	Inadequate coolant	Use increased heat removing coolant; adjust volume of flow and alignment of coolant line
Tool chips	Vibration	Snug up machine; choke up tool holder
	Excessive clearance	Reduce clearance of primary angle
	Tool material too brittle	Change to tougher tool material
Tapered hole	Heavy feed	Reduce feed
	Tool dull	Resharpen
	Machine misalignment	Check spindle and boring bar slide alignment
Hole size varies	Worn tool	Sharpen
	Boring bar overhangs farther than necessary	Reduce overhang
	Machine not completely indexing	Repair indexing and locking mechanism

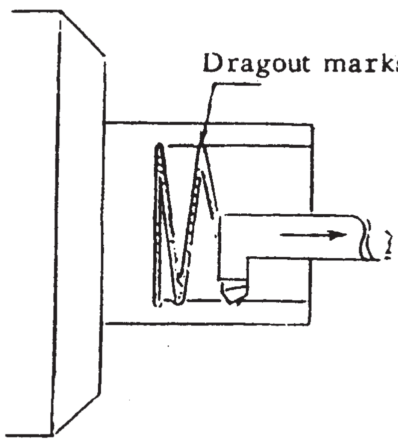
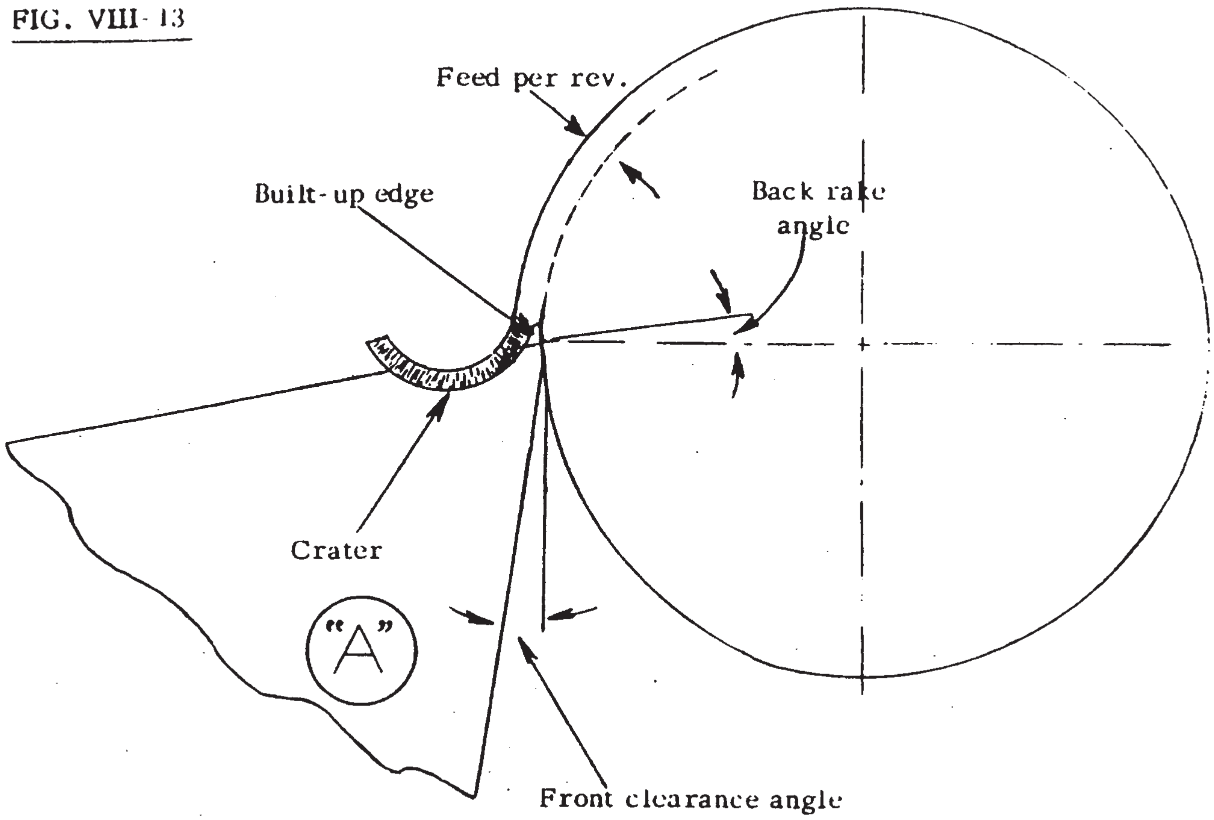
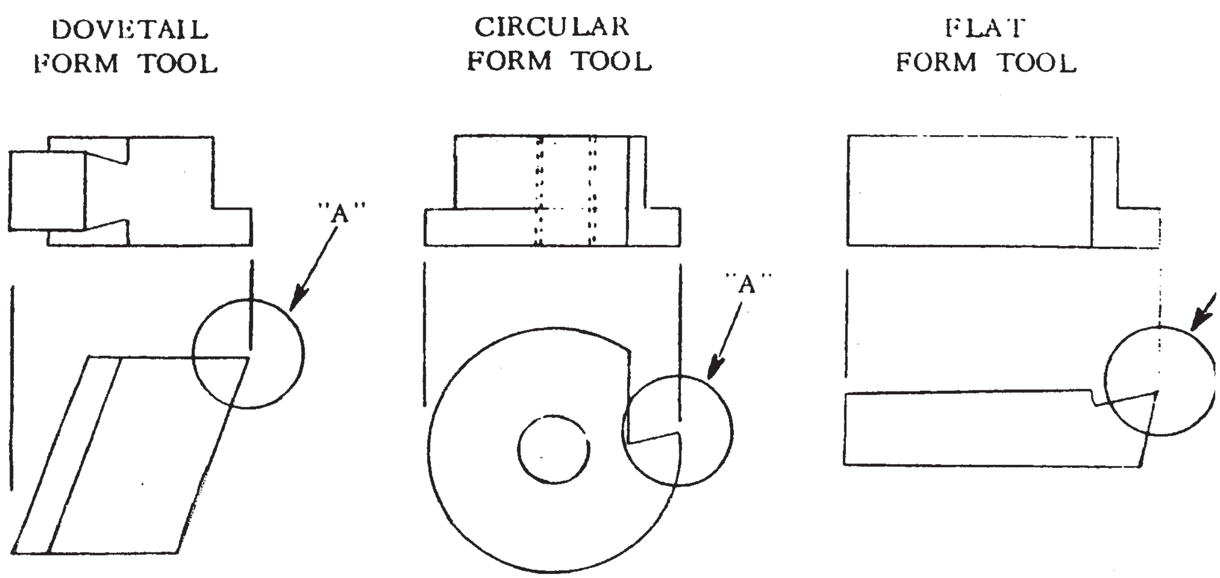
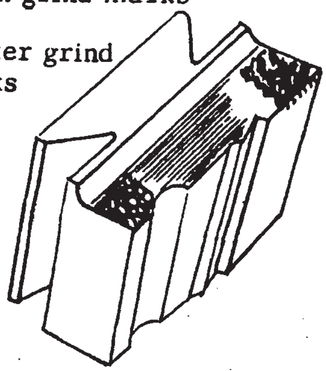
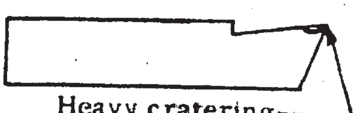
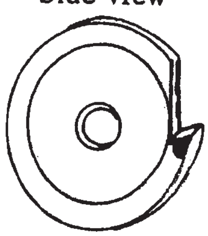
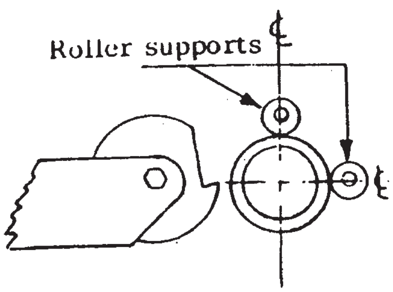
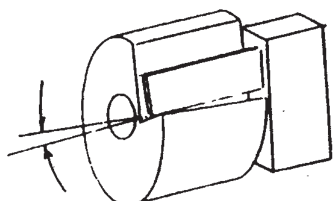
PROBLEM	CAUSE	CORRECTION
Rough finish	Heavy feed	Reduce feed
	Tool dull	Resharpen
	Speed too low relative to tool cutting angles causing a larger built-up edge	Increase speed: increase tooltop rake; resharpen smoother
	Improper or inadequate coolant	Use coolant that keeps built-up edge low and provides lubricity
	Drag out mark	Reduce primary clearance angle; reduce feed; rotate tool and holder into different position
	Tool above or below center	Recenter tool

FIG. VIII-13

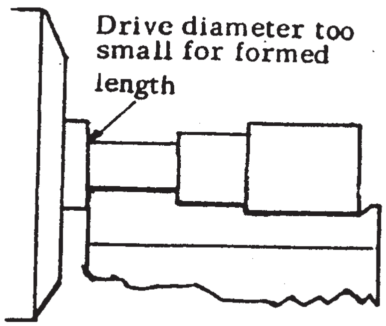
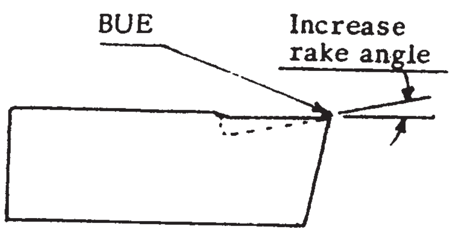

Feed per rev.
Built-up edge
Back rake angle
Crater
"A"
Front clearance angle


DOVETAIL FORM TOOL
CIRCULAR FORM TOOL
FLAT FORM TOOL
"A"
"A"

PROBLEM	CAUSE	CORRECTION
Tool breaking	Diving into cut	Increase throw
	Collets loose (bar slipping)	Check and reset collets to proper tension
	Chips between tool and holder	Clean this surface before installing tool
	Tool dull	Resharpen
Tool chips Rough grind marks Chatter grind marks  Too much heat from grinding	Lack of rigidity	Add more or better support; reduce speed: check and tighten collets
	Tool heated too much when sharpening, causing microcracks	With high speed tools, keep cool with coolant when grinding; with cast alloy and carbide use less feed and clean grinding wheel
	Tool not ground smooth	Resharpen with proper wheel to a smoother surface
Tool wears  Heavy cratering weakens edge Side view 	Flank wear, not enough clearance	Increase clearance
	Side wear from rubbing	Increase clearance (2° plus is good): change to a positive side cutting angle on tool; add a corner break on tool
	Heavy cratering causing weakening of cutting edge	Increase back rake: grind rake surface smoother: use better coolant: change to better tool: decrease speed

PROBLEM	CAUSE	CORRECTION
Tool does not hold size	Improper tension on stops	Reset tension
	Tool dull	Resharpen
	Tool above or below center	Reset tool on center
 Relationship of roller support and center line for proper support	Part not properly supported	Reset support while part is revolving; variation in size of diameter being supported
	Loose spindles	Check and tighten spindle bearings
	Feed per revolution too heavy	Reduce feed
	Machine not locating properly on index	Check and correct condition
Taper on finished part	Tool not ground straight across rake surface	Resharpen and check with square
 Tool ground out of square	Tool and holder not square with work	Remove, clean and reset
	Work springing away from tool	Add a support
Tool chatter	Tool below center	Reset to center
	Excess overhang	Reduce overhang
	Spindle vibration	Check and correct
	Excess back rake	Decrease back rake

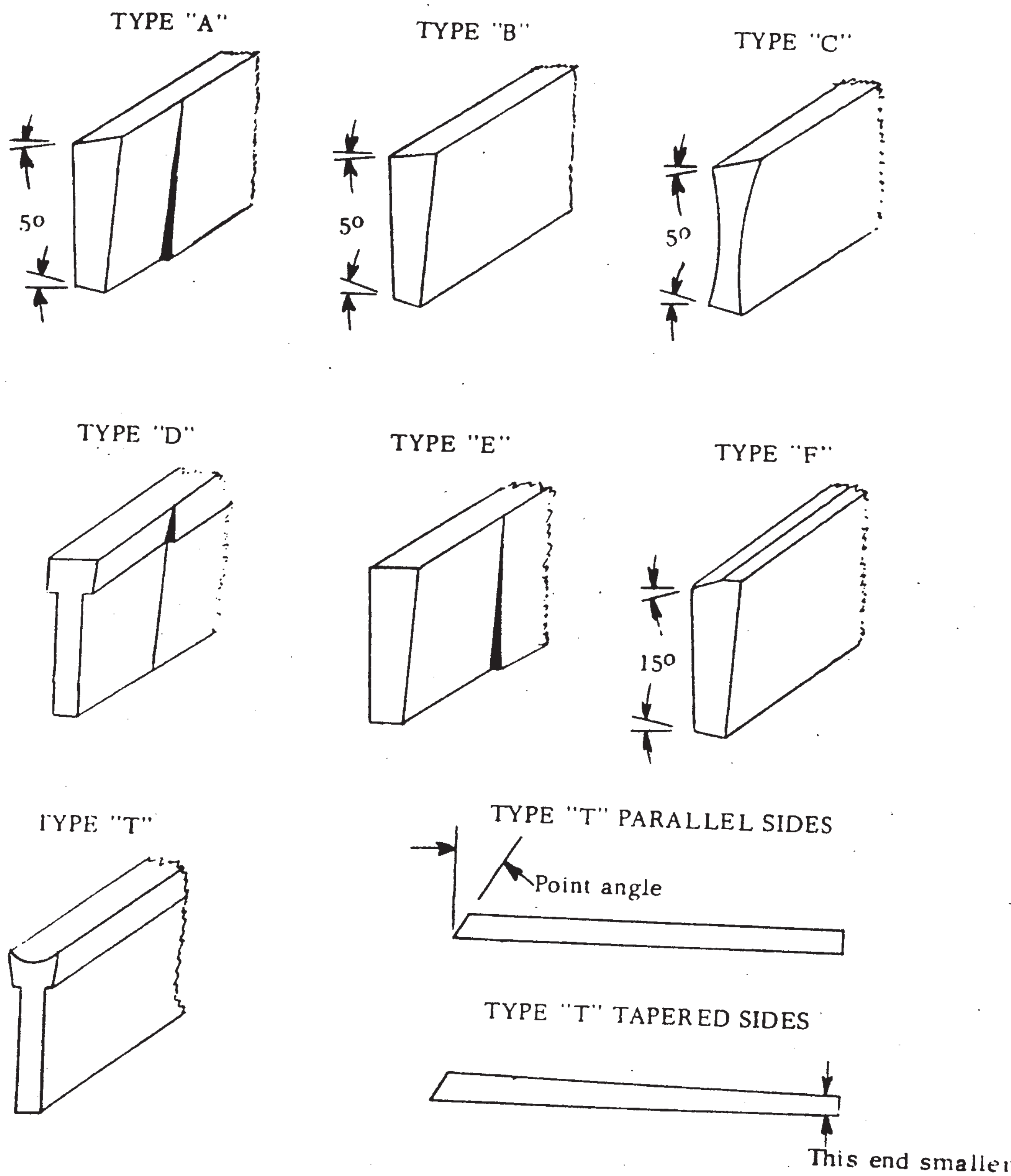
(continued)

PROBLEM	CAUSE	CORRECTION
Tool chatter (contd.) 	Ratio of part formed length to smallest formed diameter (drive diameter) too great	Add a support
	Feed too low	Increase feed
	Collet loose	Check and reset collet tension
	Nicks or burrs on tool holders	Check and correct
Rough finish 	Built-up edge too high	Increase back rake; decrease feed; use active sulfur mineral oil coolant; grind back rake smoother; increase speed
	Tool not on center	Reset to center
	Feed per revolution too high	Reduce feed
	Dull tool	Resharpen
	Cutting area loaded with chips keeping coolant away from cutting edge	Relocate tool; add an extra high pressure oil line to blow away chips; grind chip breaker
	Tool burned when manufactured or when sharpened	Regrind and check tool surfaces (See section on Tool Grinding)

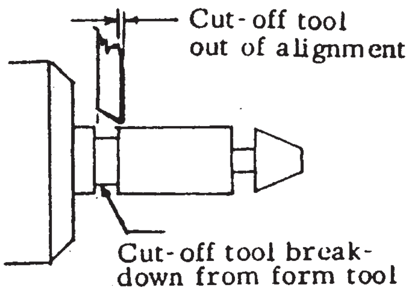
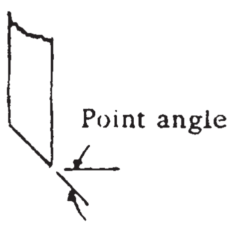
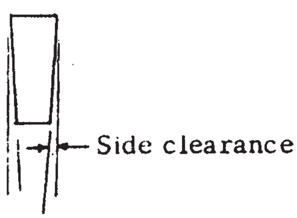
PROBLEM	CAUSE	CORRECTION
Part breaking while forming	Job set too far out from collet	Reset closer
	Insufficient cam rise	Increase cam rise
	Excessive feed	Reduce feed
	Dull tool	Resharpen
	Tool above center	Reset tool to center
	Inadequate support	Add a support
	Loose slide	Check and adjust slide

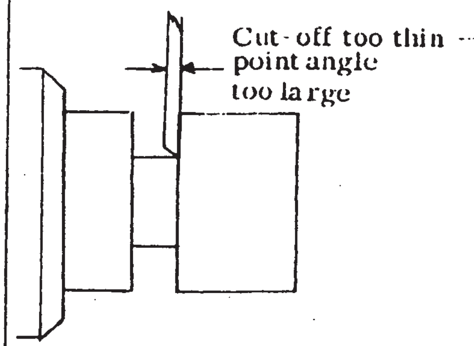
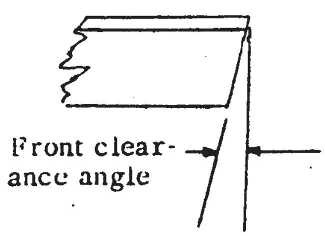
CUT - OFF TOOL DATA

FIG. VIII-15

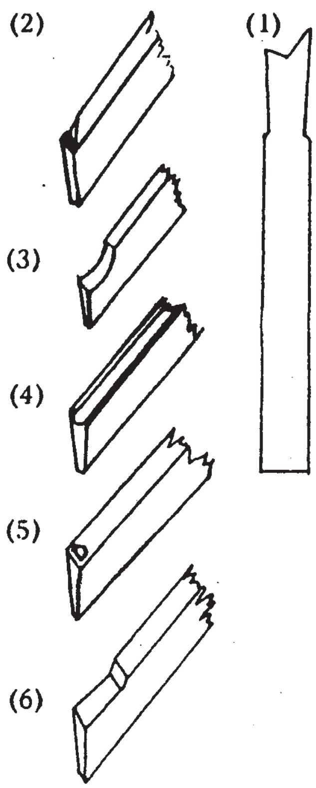


Side clearance - 2°/4°
Front clearance - 8°

PROBLEM	CAUSE	CORRECTION
Cut-off breaks	Heavy feed	Reduce feed
	Too much overhang	Reduce by moving tool back into holder
	Out of alignment with prior cut	Realign
	Tool dull causing binding	Resharpen
	Slide loose causing uneven cutting and diving	Tighten slide and check slide rollers and pins
	Insufficient cam rise	Increase cam rise
Cut-off chips	Lack of rigidity	Check and tighten spindle, slide and tool holder
	Not enough cam travel or rise	Increase rise
	Too large point angle	Decrease point angle
	Tool not on center	Check and reset
Cut-off wears	Spindle speed high	Reduce speed
	Cut-off too narrow for depth of cut	Use wider tool
	Lack of clearance on tool sides	Check, regrind side clearance and make sure cut-off tool is straight in holder
	Inadequate amount of coolant causing tool burning	Relocate coolant line

PROBLEM	CAUSE	CORRECTION
Variations in part length	Loose tool slide	Check and tighten
	Cut-off too thin	Use wider, sturdier tool
	Piece pushing back during previous operation	Check collet tension; tools may be dull
	End play in spindle	Check and reset end play
	Dull cut-off tool	Resharpen
	Excessive feed	Reduce feed
	Point angle too large	Reduce angle
Chatter	Tool below center	Reset
	Not enough or too much clearance	Regrind to proper tool geometry
	Too much overhang	Reduce by moving tool back into holder
	Tool loose	Tighten holder
	Bad spindles or end play	Check and correct condition

PROBLEM	CAUSE	CORRECTION
Rough finish	Lack of coolant	Relocate and increase coolant
	Dull tool	Resharpen
	Tool rubbing on side	Increase side clearance: remove foreign matter from between tool and holder, which causes loss of side clearance
	Chips loading in cutting area	Grind chip breaker
	Tool not on center	Reset to center
	Not enough back rake	Increase back rake by changing tool holder or grinding in more rake on tool ($5^{o}/10^{o}$ is generally used)
	Excessive feed	Decrease feed
	Spindle speed (rpm) low	Increase rpm



(1) Special cut-off tool for tubing (fish-tail)
(2) V-type
(3) Radial hook
(4) Radius type
(5) Chip groove
(6) Straight hook

FIG. VIII-16

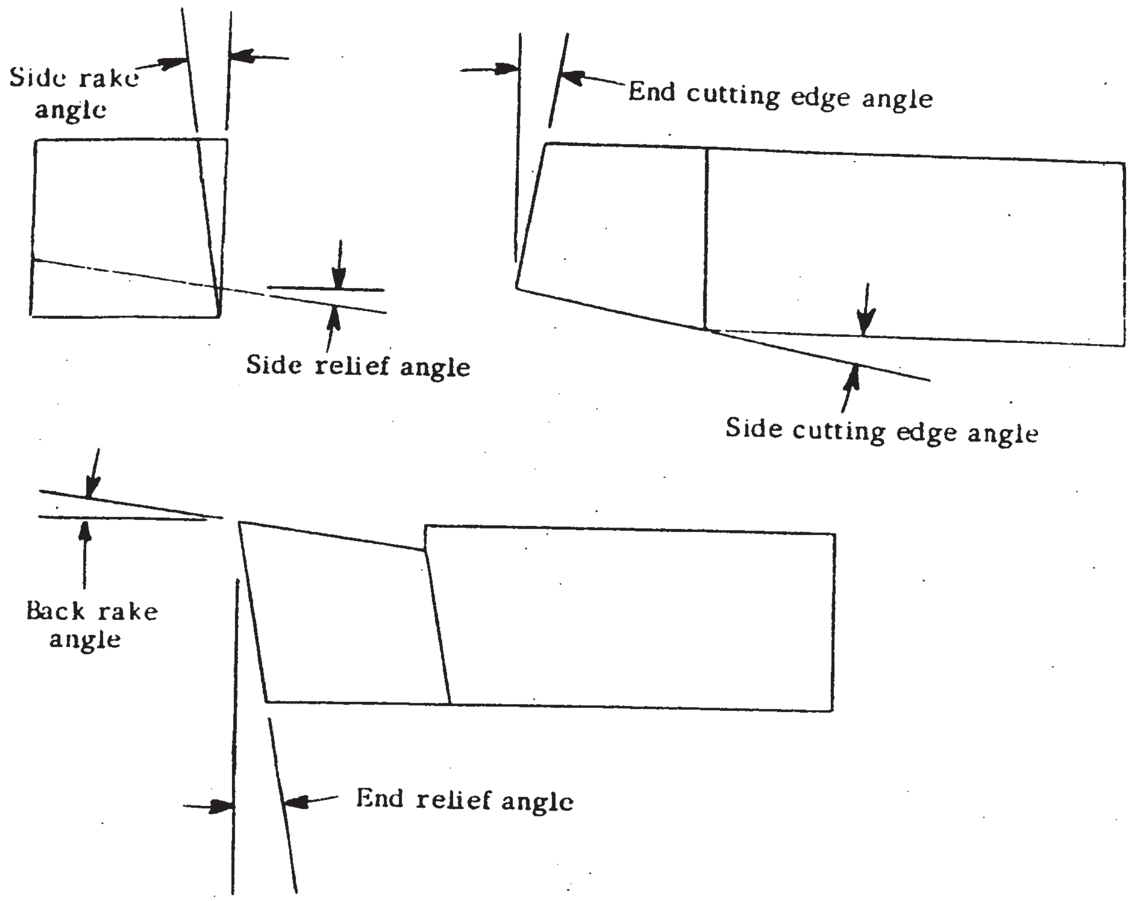
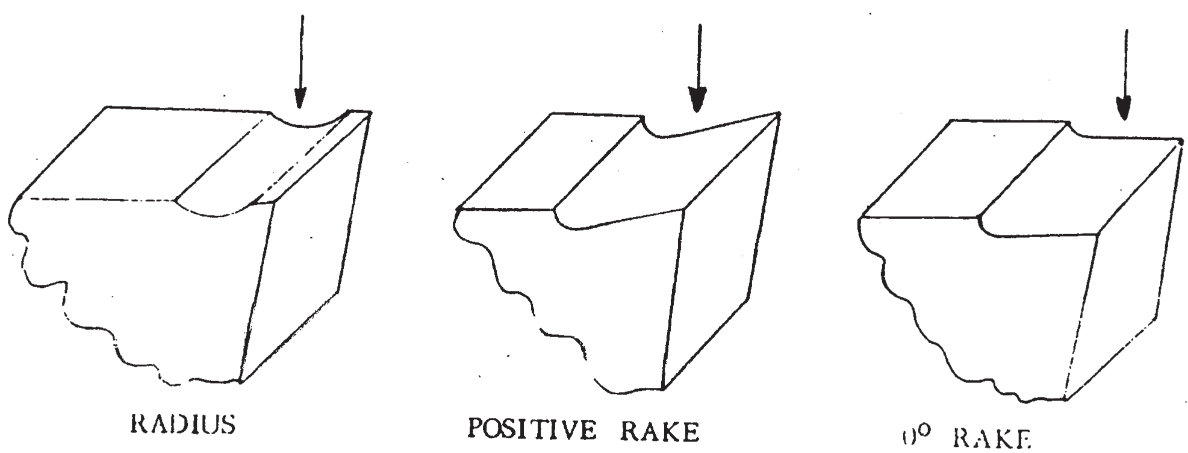
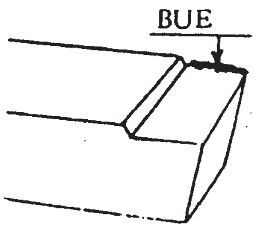
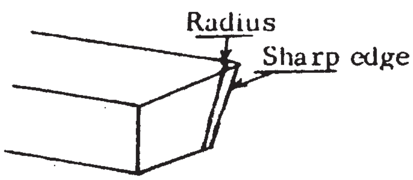
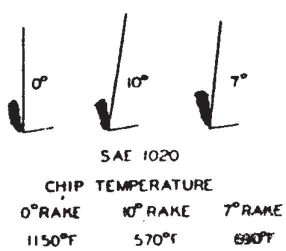


FIG. VIII-17: Three general-use chipbreakers



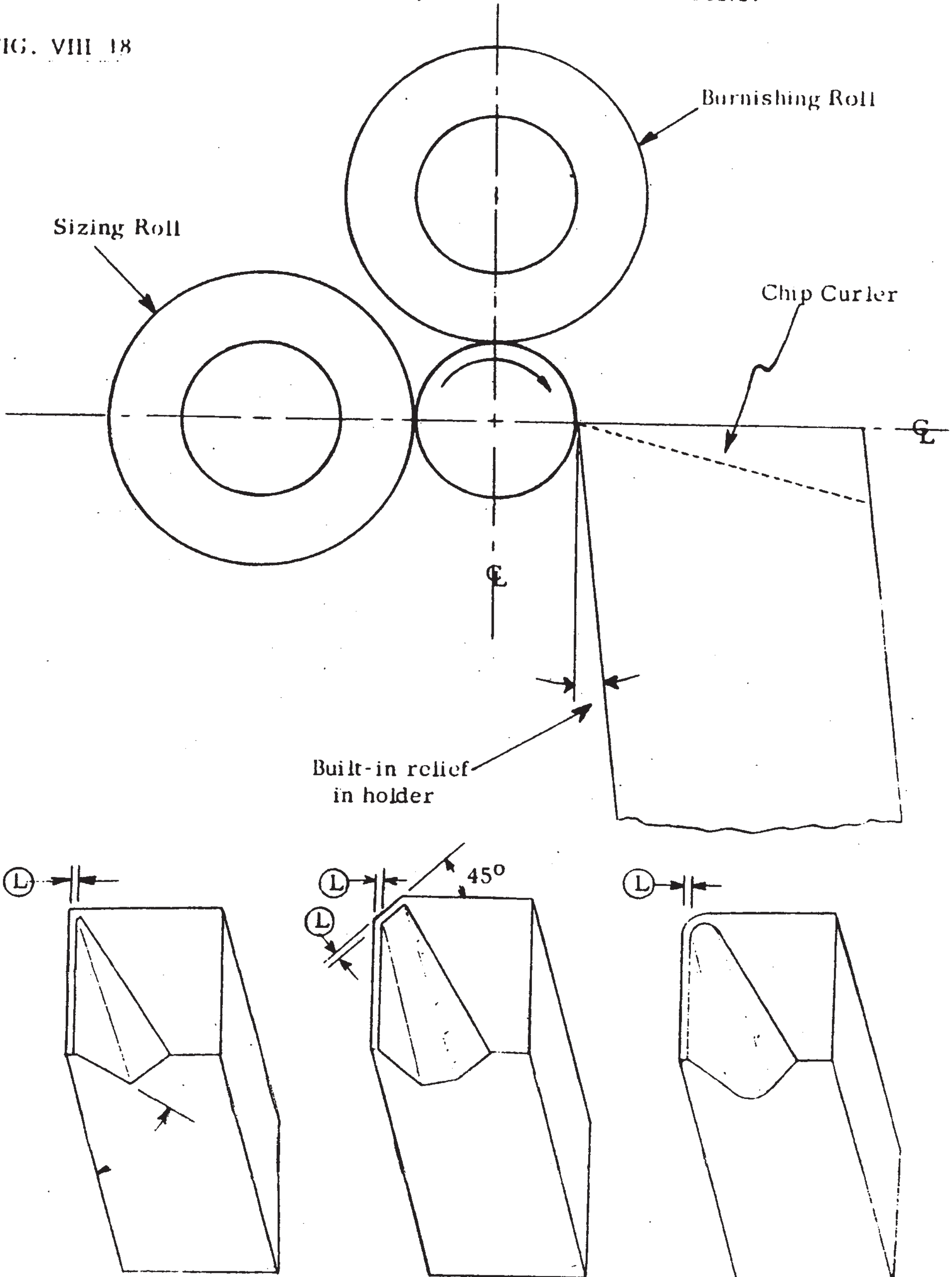
PROBLEM	CAUSE	CORRECTION
Tool breaks	Lack of rigidity	Check spindles, tool holder and reduce overhang of tool: add support
	Burned tool causing cracks	Regrind or change tool bit
Tool wears 	Excess speed causing burning of edge	Reduce speed
	Not enough relief	Increase relief
	Tool not on center	Reset tool bit
	Large built-up edge	Use active sulfur coolant: increase rake: grind smoother: reduce feed
Tool chips	Excess relief	Reduce relief
	Tool too hard or brittle	Change to a tougher tool steel or carbide
Rough finish 	Improper or not enough coolant	Use active sulfur coolant in large amounts at the cutting edge
	Tool corner sharp	Grind a radius or a chamfer on tool corner
	Feed per revolution high	Decrease feed
	Vibration	Snug up machine and spindle; add support
	Not enough rake	Increase rake
	Dull tool	Resharpen

(R-O Mfg. Co.)

BOX TURNING DATA
(Box Mill, Box Tool)

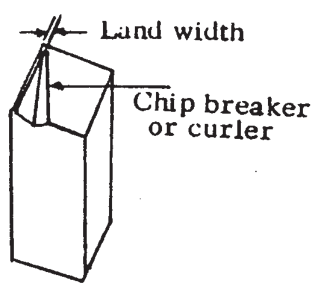
This tool holder may utilize rollers or "V" rests.

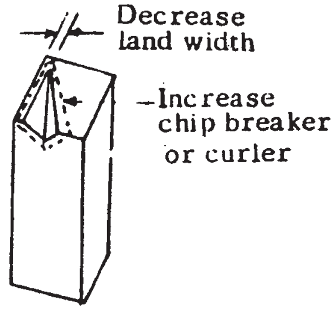
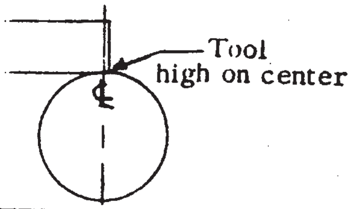
FIG. VIII 18



Ⓛ (Land) = 1 to 1-1/2 times feed per revolution.

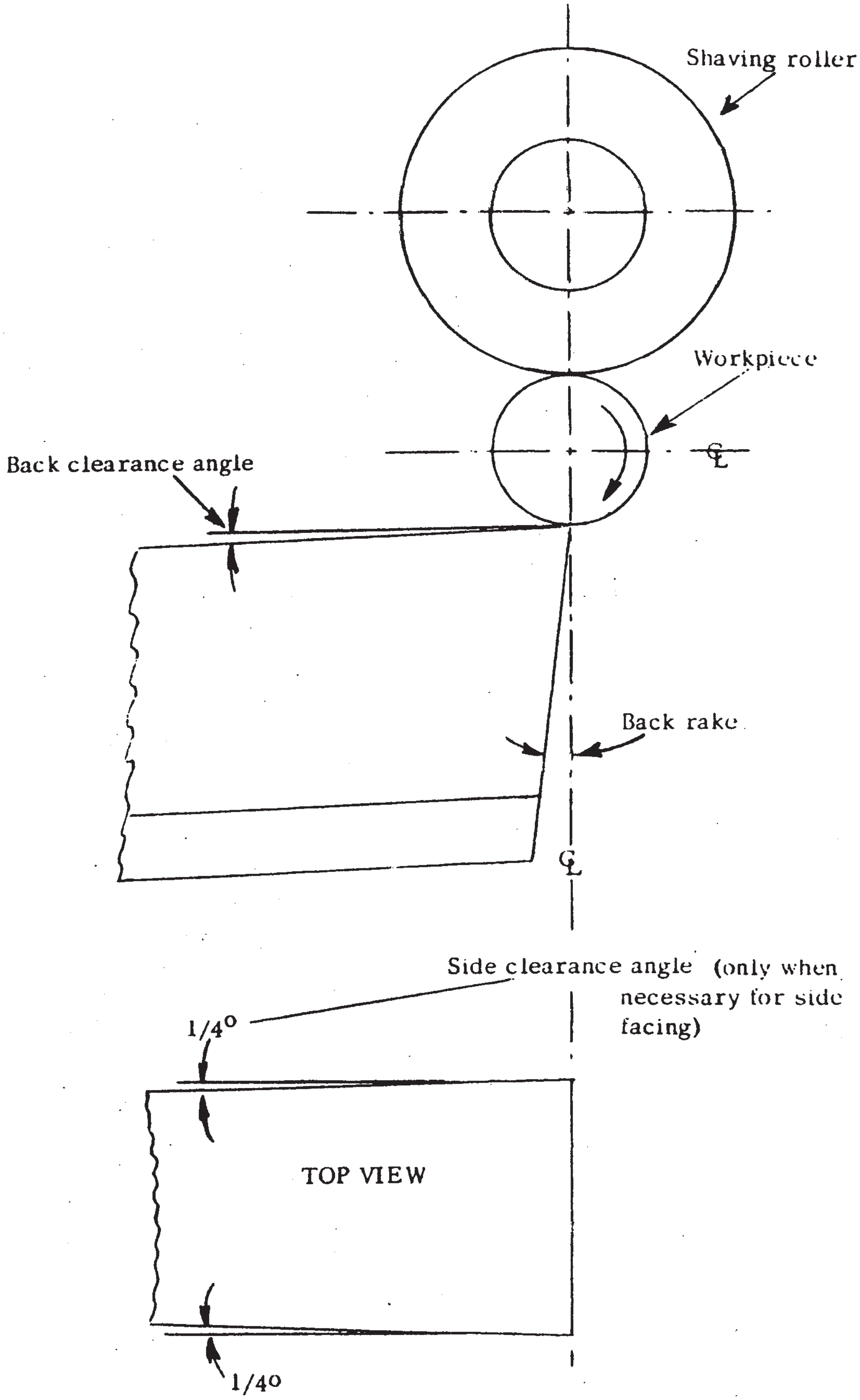
BOX TURNING TOOLS

PROBLEM	CAUSE	CORRECTION
Tool breaks	Rollers or V-rest not set right	Reset to proper pressure
	Tool jamming in	Check and tighten brake; check cam travel or rise
	Tool burned when sharpening	Resharpen
	Cast alloy or carbide tool cooled in water when sharpened, causing cracks	Regrind to remove cracks or replace tool
	Feed too heavy	Reduce feed
Tool wears	Excess speed	Reduce speed
	Tool material not abrasion resistant or hard enough	Check into tool material and change
	Tool not on center	Recenter tool
	Large built-up edge	Increase rake; decrease width of land along cutting edge to equal approximately the feed per revolution; change coolant
Flaking on surface	Roller or V pressure too high	Reset pressure less
	Tool bit above center	Reset tool lower

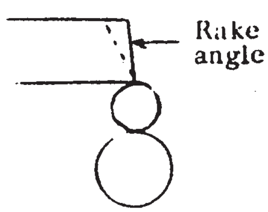
PROBLEM	CAUSE	CORRECTION
Spiral on surface	Roller or V pressure uneven	Adjust rolls or V blocks
	Tool not on center	Recenter tool bit
	Tool chip breaker not large enough	Increase width of V type chip breaker (curler) and possibly increase rake
	Land too wide along cutting edge and at point	Reduce to proper width
	Large built-up edge	Change to proper coolant for finish; reduce land width; decrease feed
General rough finish	Rolls not round (hole or O.D.)	Check and replace
	Feed too high	Reduce feed
	Improper or not enough coolant	Change coolant; increase flow; change coolant direction; add another line
	Chips packing	Use larger holder to help clear chips; change chip breaker; change or add an oil line
Tapered surface	Excessive roll pressure	Reset and reduce pressure
	Rolls not round	Replace
	Tool high on center	Lower tool to center

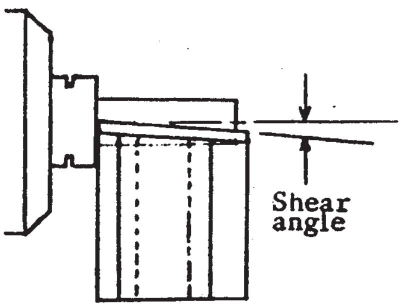
SHAVING TOOL DATA

FIG. VIII-19

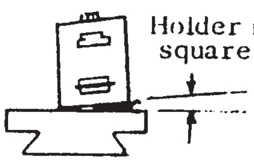


SHAVE TOOLS

PROBLEM	CAUSE	CORRECTION
Tool wears excessively	Spindle speed high	Reduce spindle speed
	Tool set ahead or behind roller	Reset to center line of roller
	Tool going past center	Reset throw of tool
	Tool material not correct	Select tool material for abrasion resistance qualities
	Feed too heavy	Reduce feed per revolution
Tool chips on edge	Spring in holder weak or broken	Replace
	Built-up edge high	Reduce feed: increase rake: use proper coolant; grind rake smoother or hone
Chatter	Feed per revolution heavy	Reduce feed
	Spindle speed high	Reduce spindle speed
	Roller and/or pin worn	Check and replace
	Shave holder worn	Check and replace or shim to snug up
(continued)	Not enough rake	Increase rake possibly up to 20°; rake generally used is 3°/5°

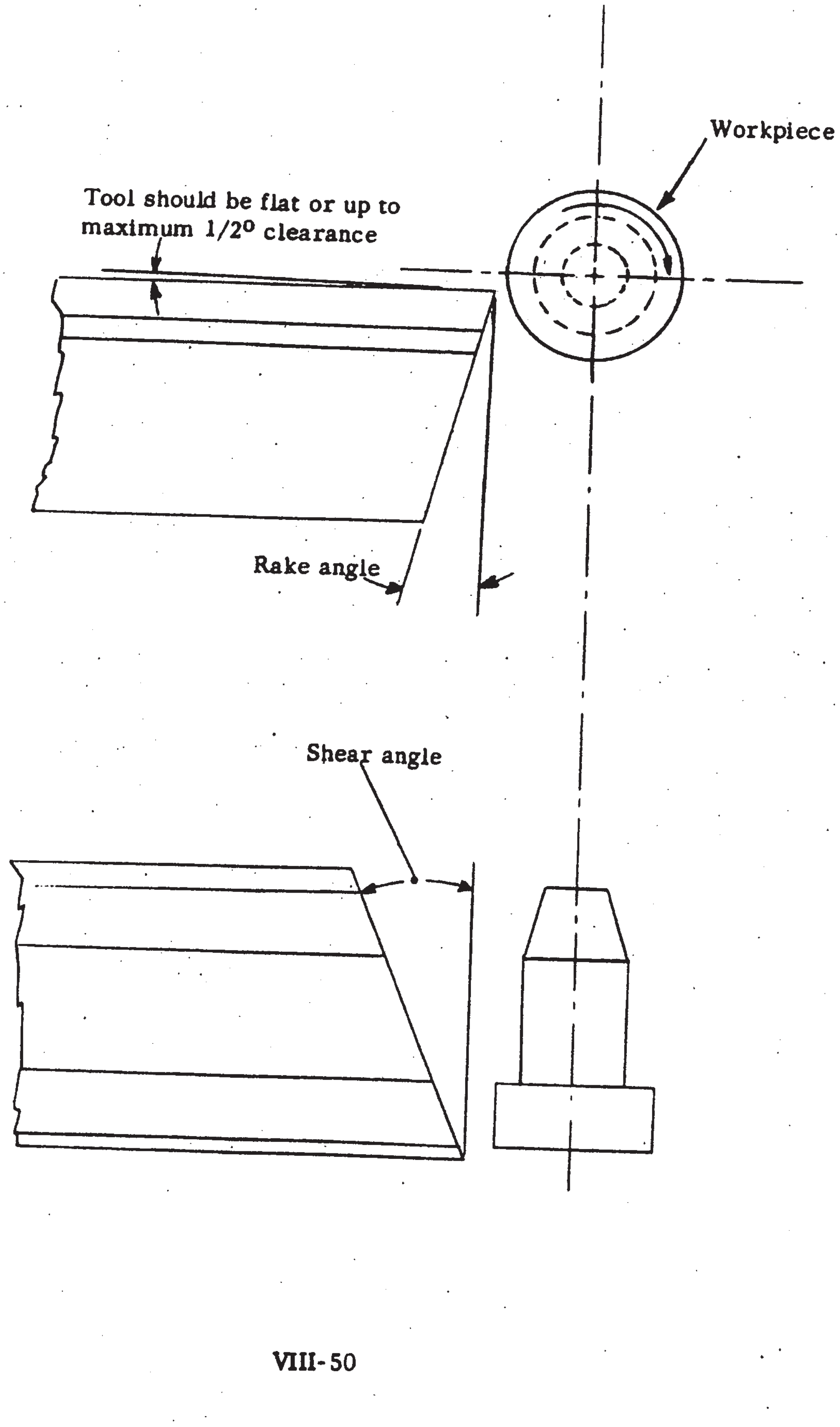
PROBLEM	CAUSE	CORRECTION
Chatter (contd.)  	Too much material being removed	Check and reset to a minimum; i.e., .0015 to .003 on a side
	Vibration	Grind 2° shear angle on front of tool: set tool ahead of roller center line; decrease amount of lift of tool holder: attach a large piece of lead to shave tool holder; use heavier springs in holder
	Machine spindles worn - index not locking	Check and correct condition
Variation of part diameter	Built-up edge heavy	Increase rake; decrease feed; change coolant: hone rake
	Tool dull	Resharpen
	Tool edge ahead or behind center line	Reset tool on center
	Blank tapered	Straighten out form tool
	Roller or pin has flats or is not round	Replace roller / pin
	Too much material being removed by shave tool	Reduce amount of material to be removed on a side: .0015/.003 is best to hold size and roundness

SHAVE TOOLS (contd.)

PROBLEM	CAUSE	CORRECTION
Workpiece breakage	Tool diving into piece	Increase cam rise and, or reposition tool to center line of part
	Piece set too far from collet	Reset job closer to collet
	Tool removing too much material	Reduce blank size diameter
	Tool ahead or behind roller center line causing excess pressure	Reset tool to roller center line
Shaved diameter tapered 	Tool not square with roller	Reset tool
	Tool not ground square	Resharpen
	Defective holder	Check and replace
	Holder not square	Adjust
Rough finish	Dull tool	Resharpen
	Tool form finish is rough	Remake tool or replace
	Blank diameter too large	Reduce blank diameter
	Improper or inadequate coolant	Use proper coolant in copious amounts: for steel use active sulfur oil
(continued)		

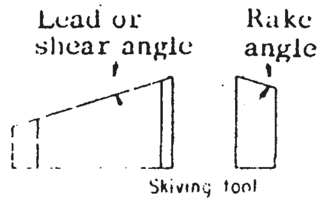
PROBLEM	CAUSE	CORRECTION
Rough finish (contd.)	Blank surface rough	Resharpen blank form tool and increase rake on this tool (use Dykem on blank prior to shave cut; if traces of color are visible after shave cut, blank is too rough)
	Tool not set on roller center line	Reset tool
	Built-up edge heavy	Increase rake angle; decrease feed; change coolant; increase speed; grind rake or hone extra smooth

FIG. VIII-20



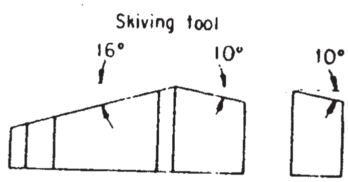
Workpiece
Tool should be flat or up to maximum 1/2° clearance
Rake angle
Shear angle

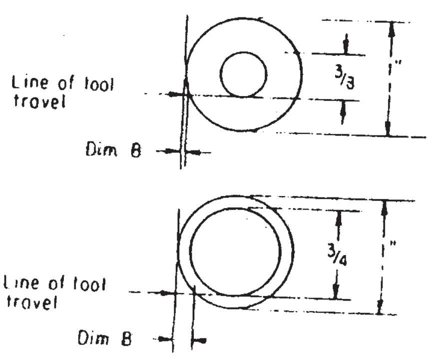
SKIVING TOOLS

PROBLEM	CAUSE	CORRECTION
Tool edge chipping	Rake angle too high	Reduce rake angle: recommended starting rake angles are: Brass 17°, Aluminum 2[illegible]°, Steel 25°
	Built-up edge	Grind better surface finish on cutting edge; use active sulfur coolant
	Vibration	Check tool holder; add a support
("American Machinist")		
Tool wears excessively	Spindle speed too high for material	Reduce spindle speed
	Feed per revolution too high	Reduce feed per revolution
	Tool burned when made or when sharpened	Check surface hardness of tool form and rake
Variation in part diameter	Tool dull	Resharpen
	Lack of rigidity	Use a support
	Excessive clearance	Check and reduce clearance; recommended clearance angle is 1-2°
	Too much overhang of tool	Reduce overhang
(continued)		

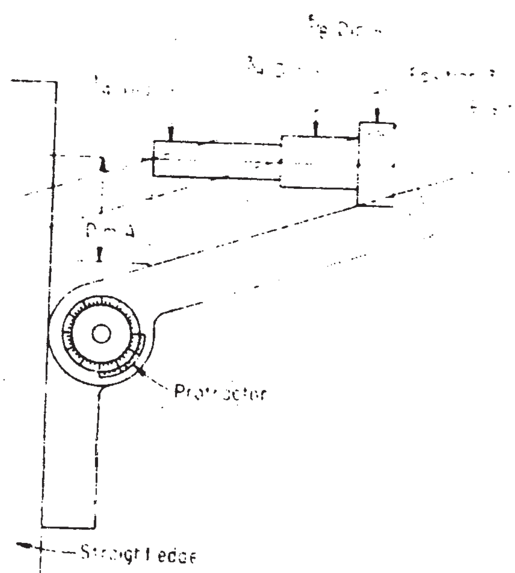
PROBLEM	CAUSE	CORRECTION
Variation in part diameter (contd.)	Excessive feed	Reduce feed per revolution
	Shear angle too small	Increase shear angle
	Tool not perpendicular to axis of part causing side rubbing	Check and reset tool
	Insufficient rake angle	Increase rake angle
Work piece breaking	Excessive feed	Reduce feed
	Lack of support	Use support
	Shear angle not large enough	Increase shear angle
	Tool or holder loose	Check and tighten
	Spindle loose	Check and repair
	Tool and part too far out of collet	Reset closer to collet
	Tool plunging into work	Increase cam rise or travel
Chatter	No support or improper support	Use best support possible
	Not enough rake	Increase rake
	Too much overhang of tool	Reduce overhang
	Lack of rigidity	Check and snug up spindle, slides and tool holder

PROBLEM	CAUSE	CORRECTION
Rough finish	Not enough rake	Increase rake angle
	Improper coolant	Correct coolant: for steel use an active sulfur oil
	Tool surface finish poor	Remake tool with improved surface finish
	Dull tool	Resharpen
	Spindle speed too low causing large built-up edge	Increase spindle speed
	Insufficient shear angle	Increase shear angle
	Insufficient coolant	Increase coolant volume not velocity



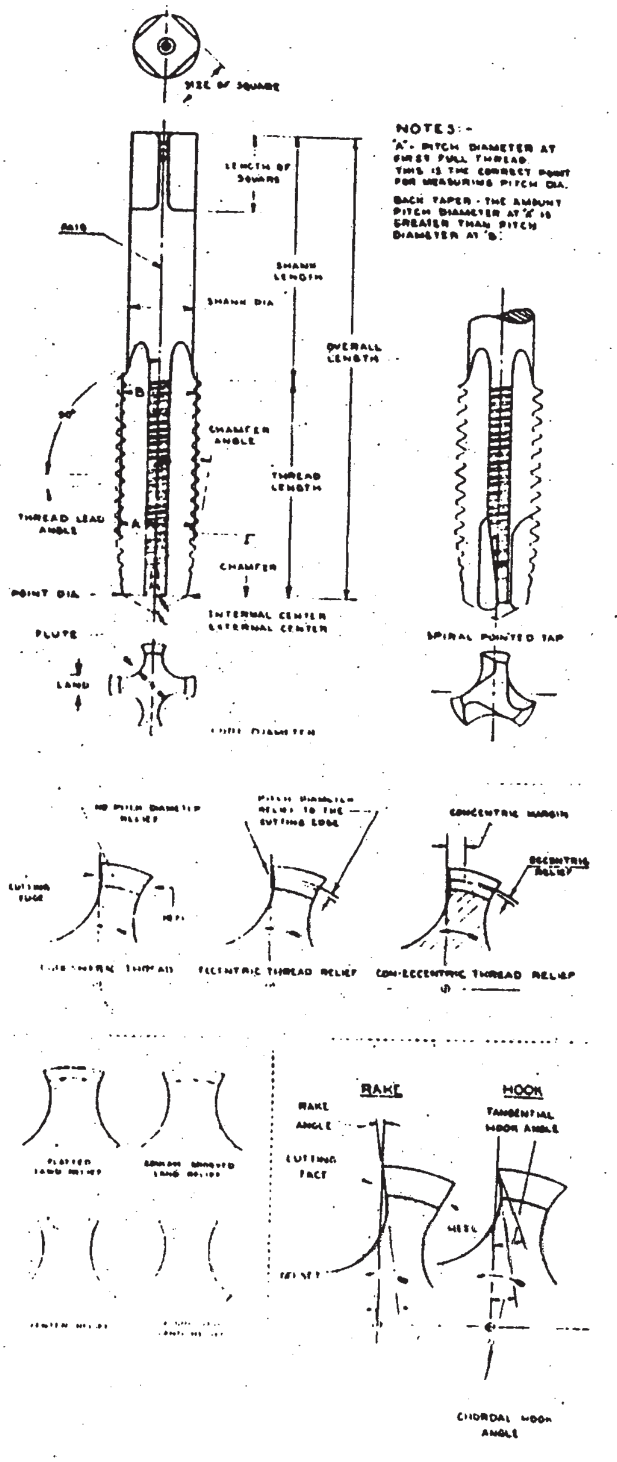


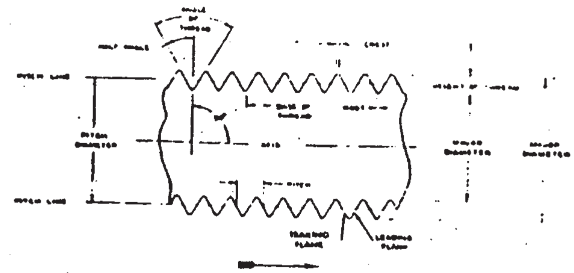
Simple graphic method pinpoints com rise



Illustrations from "Skiving on Automatics" - AMERICAN MACHINIST, February 2[illegible] 1966, courtesy of the McGraw-Hill Publishing Company.

FIG. VIII-21: Terms applyi..g to Taps





Helical fluted hand tap

Taper pipe tap

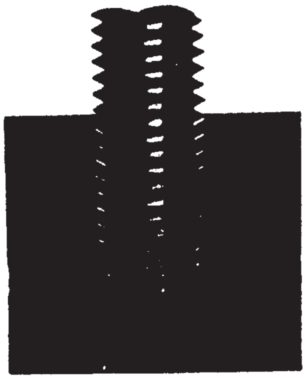
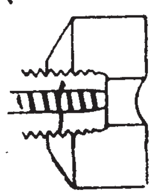
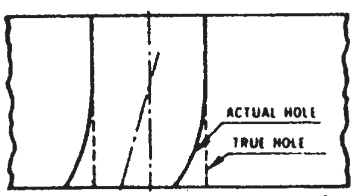
Bottoming tap

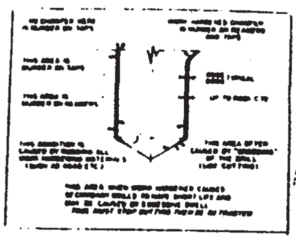
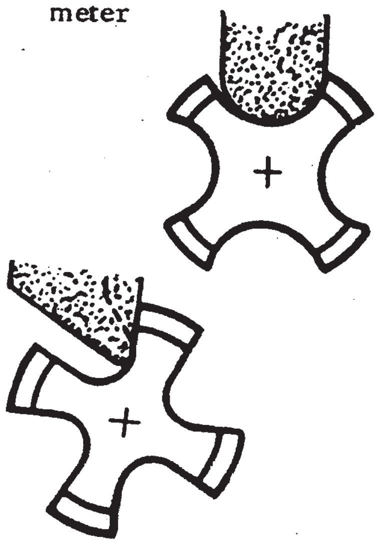
Plug tap

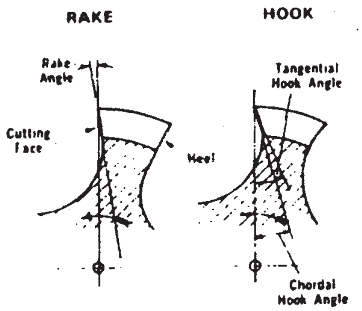
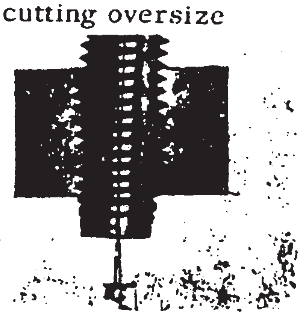
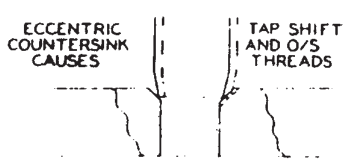
Taper tap

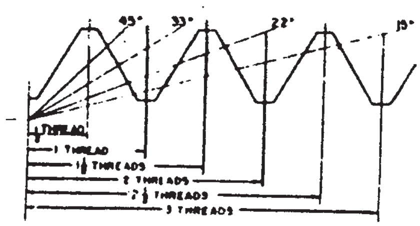
Interrupted thread taper pipe tap

(Illustrations courtesy of the Metal Cutting Tool Institute, New York, N.Y.)

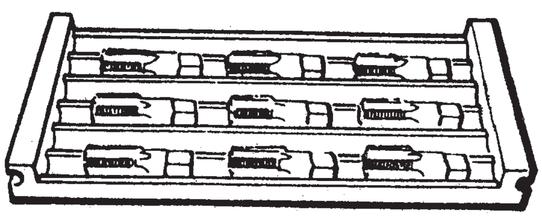
PROBLEM	CAUSE	CORRECTION
Tap breaking	Tap hitting bottom	Increase drill depth or reduce depth of tapping
	Chips jam when tap is pulling out	Reduce tap land; change grind to change chip shape
	Lack of coolant	Use oil hole tap or add coolant lines
	Chamfer angle short	Increase length of chamfer angle
Chip packing may be overcome by a shorter thread depth or a lower percentage of thread ("American Machinist")	Welding on tap surface	Reduce land; change coolant; reduce spindle speed
 (Greenfield Tap & Die)	Hole too small (thread percentage too high)	Check and use larger drill or reamer to produce less thread percentage: 60% if print allows
	Eccentric hole	Correct drilling problem
	Dull tap	Resharpen
	Spindle speed too slow	Increase spindle speed
(Bay State Tap and Die) *BENT OR CROOKED HOLES are those whose axes are not straight, generally produced by tools "leading off." Threads will tend to follow the hole.*	Misalignment	Realign tap
Tap chipping	Hole chamfer not concentric or large enough	Reset and resharpen spotting drill
	Excessive rake, hook angle or chamfer angle relief	Regrind or replace tap
(continued)		

PROBLEM	CAUSE	CORRECTION
Tap chipping (contd.)	Tap not in line with spindle	Realign tap
	Tap being crowded into cut	Decrease cam rise, change or use slower cycle time to decrease feed on tap, reduce spring pressure
	Unequal chamfer grind	Resharpen tap
(R-O Mfg. Co.)	Spindle speed too slow	Increase spindle speed
Variation in pitch diameter	Tap crowding	Check cam rise
	Dull tap	Resharpen
	Floating holder worn	Replace
	Hole diameter varies or out of round	Check and regrind drill, reamer or boring tool
	Excessive land on tap	Regrind in flute to lessen lands; use land relief tap
	Not enough rake or hook angle	Increase rake or hook angle
(Greenfield Tap & Die)	Flutes not large enough, chips packing	Grind flute or change tap
Tap "picking up"	Not enough rake or hook angle	Increase rake or hook angle
(continued)		

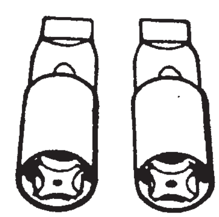
PROBLEM	CAUSE	CORRECTION
Tap "picking up" (contd.)  (Bay State Tap and Die)	Rake or hook not ground smooth	Regrind smoother and polish
	Land too wide	Grind in flute to reduce land; use land relief tap
	Insufficient or incorrect coolant	Change coolant and increase amount
	Tap burned when sharpened	Regrind or replace
	Dull tap	Resharpen or replace; tap may have been used too long without sharpening
	Chamfer relief insufficient	Increase chamfer relief
Tap cutting oversize	Misalignment	Realign tap
	Spindle speed too slow	Increase spindle speed
	Improperly ground tap	Regrind or replace
	"H" size of tap incorrect	Use lower size class of tap
	Improper coolant	Change coolant
("American Machinist")	Blank hole not to size or out of round	Check drill and reset or regrind
 	Starting chamfer in hole not large enough or not concentric	Increase hole chamfer size and reset spot drill
	Worn tap picking up on lands	Reduce speed; increase hook; realign tap; change coolant; grind longer chamfer
(R-O Mfg. Co.)		

PROBLEM	CAUSE	CORRECTION
Rough threads	Incorrect or insufficient coolant	Change coolant to an anti-weld type and increase amount
	Dull tap or burned when sharpened	Resharpen or replace
	Spindle speed high or too low	Reduce speed if edge is burned or decrease speed if built-up edge is high
	Misalignment of tap	Realign tap
(Bay State Tap and Die Co.)	Chips packing in flutes	Increase size of flutes; polish flutes; use spiral tap; use oil hole tap
	Blank hole small (thread percentage high)	Increase drilled hole size; 60% thread if print allows
(Geometric Tool Co.)	Chamfer angle not long enough (chip thickness heavy)	Increase chamfer angle; 3 thread angle or more is most desirable

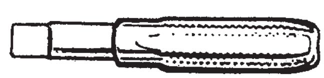
HANDLING AND STORAGE



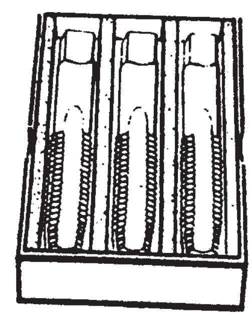
Wooden trays or drawers with dividers to isolate each tap.



Protective tubes



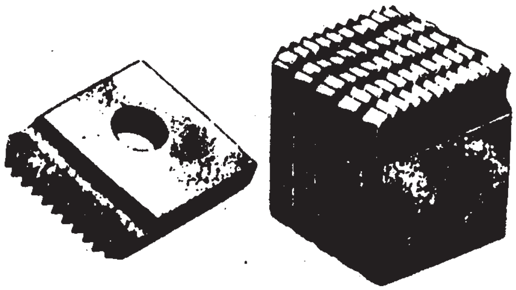
Plastic or similar protective coating.



The original package, if available

(Bay State Tap and Die Co.)

FIG. VIII-23: Insert chasers



H&G INSERT CHASER NOMENCLATURE

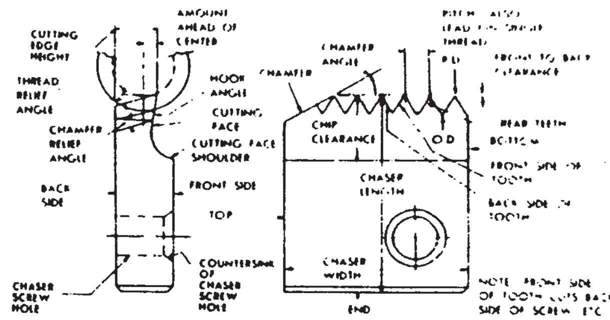


FIG. VIII-24: Die head chasers, blade type, tapped (R H shown)

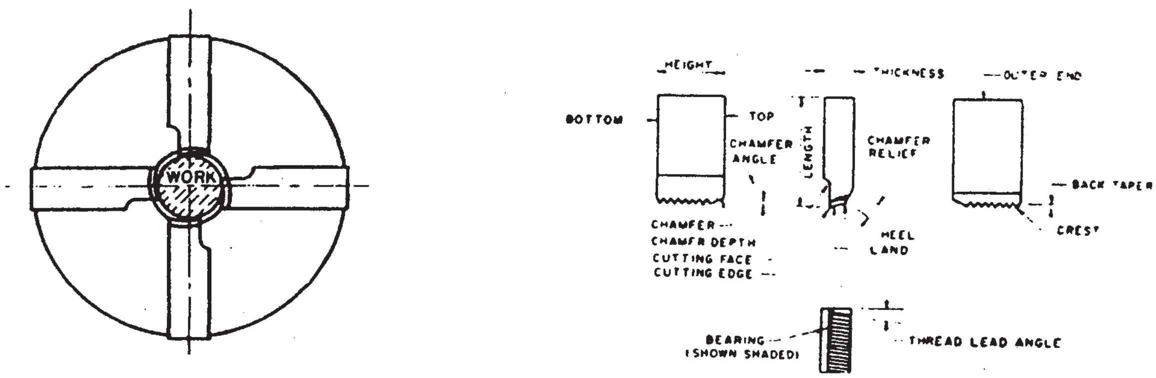
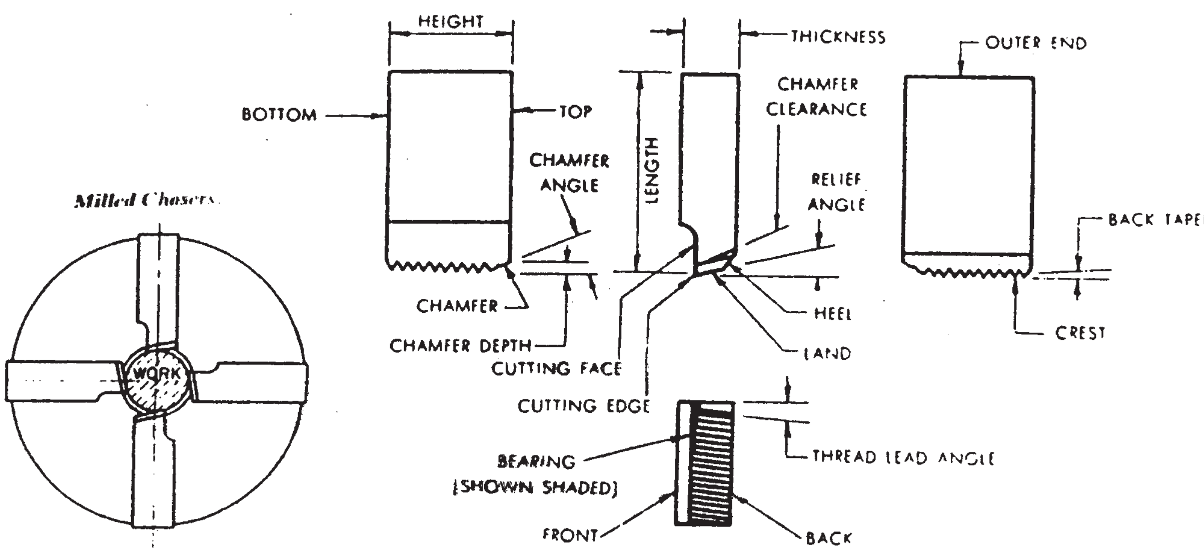


FIG. VIII-25: Die head chasers, blade type, milled (R H shown)



(Fig. 23 courtesy of H & G Division of The Cleveland Twist Drill Co., Cleveland, [illegible]
(Figs. 24 and 25 courtesy of The Metal Cutting Tool Institute, New York, N.Y.)

FIG. VIII-26: Tangent chasers

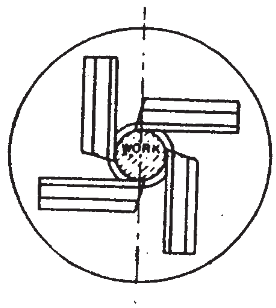


FIG. VIII-27: Die head chasers, tangent type (R H shown)

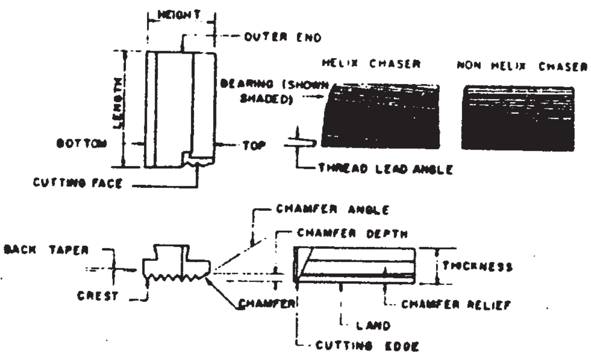


FIG. VIII-28: Circular chasers

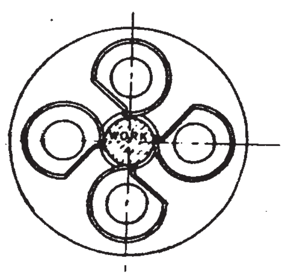


FIG. VIII-29: Die head chasers; circular type (R H shown)

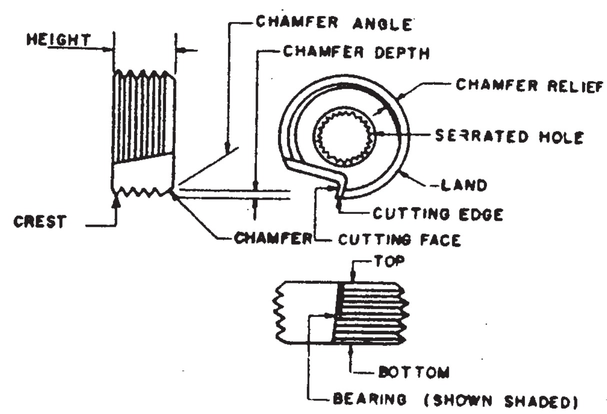
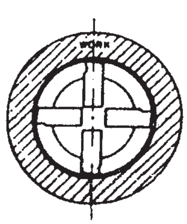
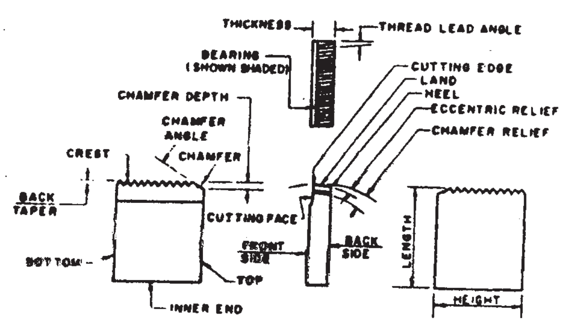


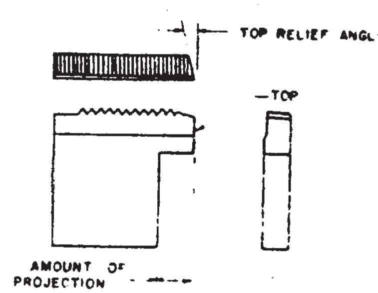
FIG. VIII-30: Tap chaser

FIG. VIII-31: Regular tap chasers, blade type (R H shown)

FIG. VIII-32: Projection tap chaser, blade type



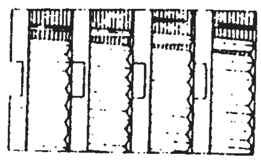




(Illustrations courtesy of the Metal Cutting Tool Institute, New York, New York)

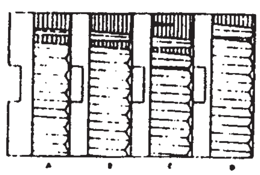
PROBLEM	CAUSE	CORRECTION
Chaser breakage	Chasers hit shoulders	Reset die head opening mechanism
	Blank out of round, over-size or not concentric	Check blank and reset tools
	Die head not aligned	Realign holder
	Dull or worn chasers	Resharpen chasers before they are too dull
	Die head sloppy	Repair die head
	Improper coolant causing large built-up edge to form	Change coolant to anti-weld type
	Chamfer grind not equal on all chasers	Regrind
Chaser wear excessive	Spindle speed high	Decrease spindle speed
	Incorrect coolant	Change coolant
	Chamfers not ground equally	Resharpen
	Chamfer not long enough	Increase length of chamfer
	Not enough rake or hook	Increase rake or hook
	Cam rise improper causing crowding or pulling of head	Use proper cam rise
	Chaser burned when made or sharpened	Check and resharpen
	Die head out of line with spindle	Realign die head

CORRECTLY GROUND SET



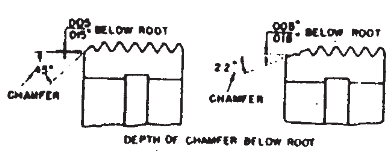
Note that chamfer is slightly different on each chaser of set.

INCORRECTLY GROUND SET



Chaser B is correctly ground. Chaser C has been ground with too long a chamfer. Chaser D has too short a chamfer. Also Chaser A is too thin and not of the same thickness as the other three chasers of the set.

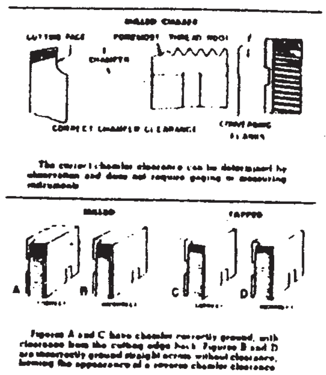
DEPTH OF CHAMFER



The chamfer should start .005" to .015" below the root of the chaser teeth. All chasers of a set should be ground at uniform angle and depth so that each chaser will start cutting at the same time.

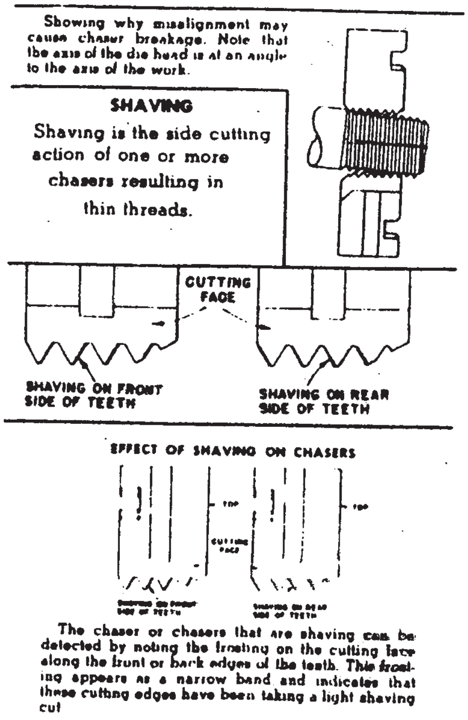
(Geometric Tool Co.)

PROBLEM	CAUSE	CORRECTION
Variation in pitch	Die head locking pin worn	Replace locking pin
	Die head springs weak or broken	Replace springs
	Variation in blank size	Check and reset tools cutting blank
	Cam rise incorrect causing crowding or drag	Check and change cam rise
	Spindle speed too high	Decrease spindle speed
	Dull chasers	Resharpen
	Chamfer relief inadequate	Regrind chamfer



(Geometric Tool Co.)

PROBLEM	CAUSE	CORRECTION
Tapered threads	Misalignment	Realign die head or spindle; check for bent shank on die head
	Chamfer unequal	Resharpen
	Worn die head locking pin	Replace locking pin
	Die head crowding on blank	Check camming and change
	Loose die head parts or chips inside head	Check, clean, and retighten
	Chasers or carriers sloppy in die head	Replace die head or chasers; remove plates, check for burrs and dirt
	Dirt under chasers or in die head	Clean out
	Chasers shave	Remove sharp edge on chasers
	Machine loose	Correct machine conditions
	Poor start of chasers on blank	Increase chamfer; make sure it's not work hardened; check cam and starting spring



(Geometric Tool Co.)

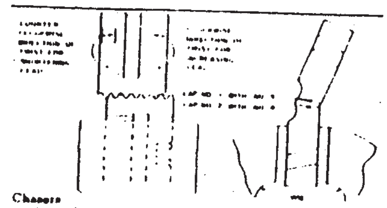
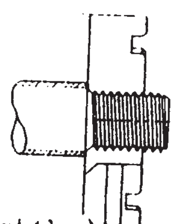
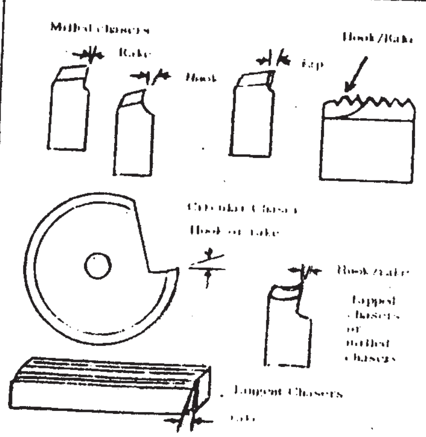
PROBLEM	CAUSE	CORRECTION
Lead error	Chasers shaving	Remove sharp chaser edge or burr; pull each chaser across a flat piece of lead with valve grinding compound on it. Remove dirt or chips on chaser holders – remove nicks or burrs from chasers. slot or holder
 Chasers are twisted in respect to each other when lapping for lead.	Die head pushing on blank	Check camming and change
	Die head out of line	Realign head
(Geometric Tool Co.)	Chamfers not equal	Resharpen
Rough threads	Spindle speed too high or low	Decrease if chasers are burned: increase if built-up edge is large
Attempting to remove too much material with chasers. 	Blank too large	Decrease blank size
	Rake or hook incorrect	Increase rake or hook angle
(Geometric Tool Co.)	Incorrect coolant	Change to anti - weld type coolant
	Die head out of line with spindle	Realign die head or threading spindle
	Die head pushing or pulling	Correct camming to acquire that the die head follows properly
	Chamfers unequal	Resharpen
	Burrs on chasers edge from grinding	Remove burrs carefully so that thread is not deformed
	Chasers dull	Resharpen
	Chasers nick shoulder	Adjust stop

FIG. VIII- : Nomenclat re

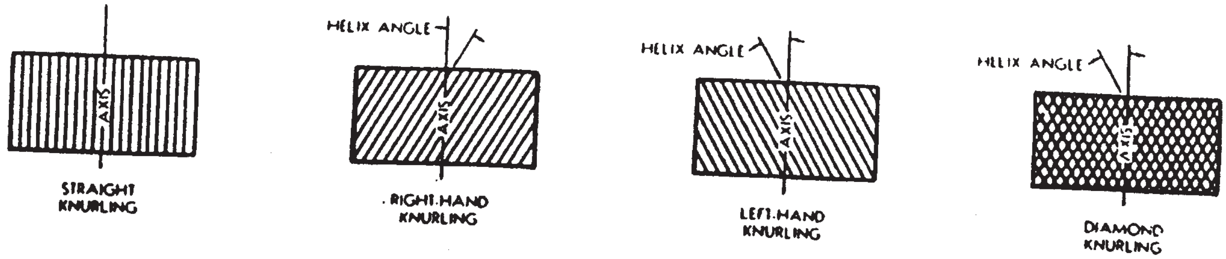
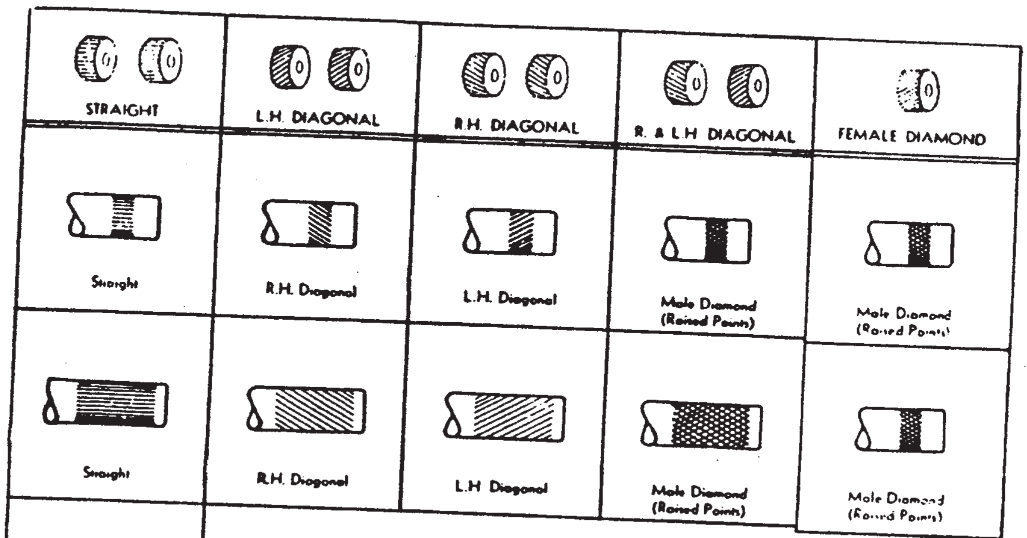
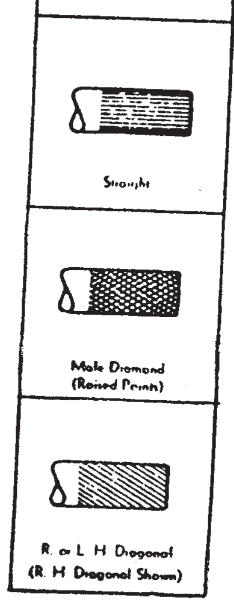


FIG. VIII- : Types of knurls





Approximate Increase of Blank Diameters

with Standard Reed Circular Pitch Knurls on Soft Steels

*Teeth per Inch	Straight and Diagonal** Knurling	**Diamond Knurling	
		Male (Raised Points)	Female (Depressed Points)
12	.034	.038	..
16	.025	..	..
20	.020	.023	.014
25	.016	.018	.011
30	.013	.015	.009
35	.011	..	..
40	.009	.010	
50	.009	.010	.007
80	.005	.006	

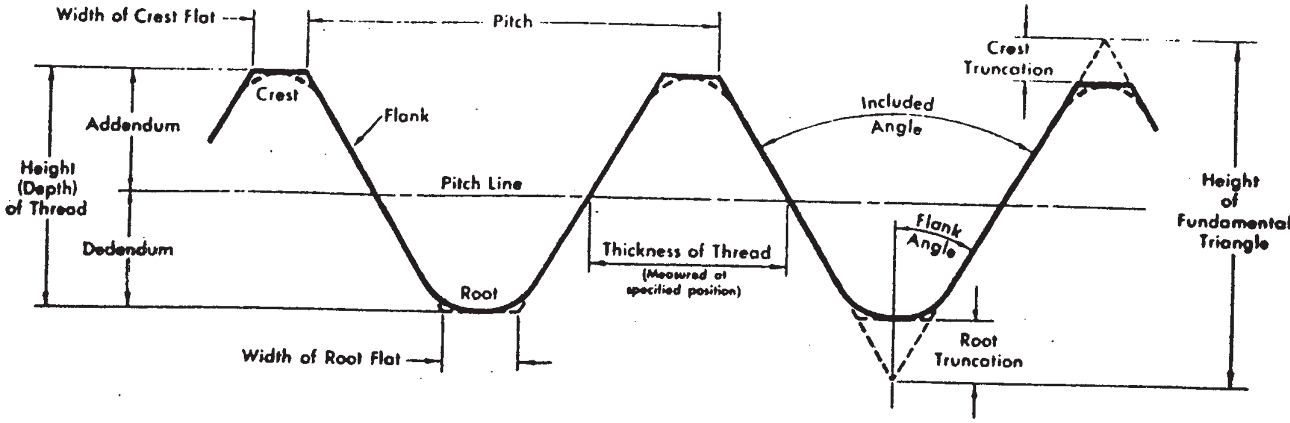
*Refers to normal teeth per inch on diagonal and diamond knurling.
**With 30° helix angle.

(Illustrations courtesy of the Reed Rolled Thread Die Co., Holden, Mass.)

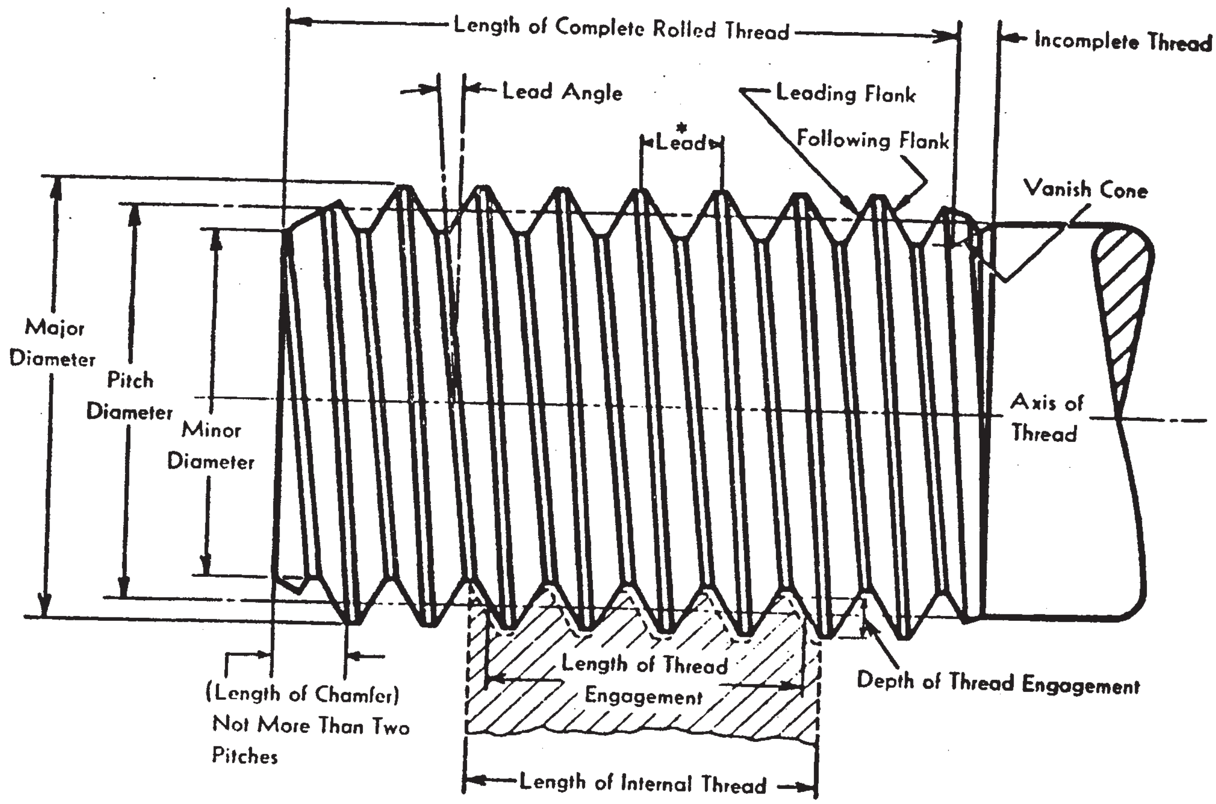
PROBLEM	CAUSE	CORRECTION
Knurl wearing	Spindle speed too high	Reduce speed
	Knurl feeding on too slowly (too many revolutions)	Increase feed per revolution
	Incorrect coolant	Change coolant
Knurl not tracking $\frac{\text{Teeth on knurled part}}{\text{Blank diameter}} = \text{Diametrical pitch}$ Example: $\frac{48}{.500} = 96$ 	Feed per revolution too low	Increase feed per revolution
	Pitch of knurl incorrect for blank	Check and change knurl or blank size
	Knurl sticking in holder	Check and free up
Oversize knurl	Blank large	Decrease blank size
	Excessive pressure on knurls	Reduce pressure
Undersize knurl	Blank diameter small	Increase blank size
	Insufficient pressure on knurls	Increase pressure
Straight knurl spiraling 	Knurl set at an angle to axis	Reset knurls

PROBLEM	CAUSE	CORRECTION
Knurl flaking	Too many revolutions used to knurl	Increase feed per revolution
	Blank too large (over knurling)	Decrease blank size
	Knurls not lined up with axis of part	Realign
Knurl variation in size or form	Variation in blank size	Sharpen form tool; check tension stops for too much or too little pressure
	Worn or broken knurl/knurls	Replace knurls
	Tapered blank	Reset form tool
	Uneven adjustment of twin knurls	Reset (Stop knurl at center of part. Allow several rotations without feed to "clear" knurl. BOTH knurls should be tight.)
	Knurl not on center	Reset

FIG. VIII- : Terms applied to Unified and American External Screw Threads



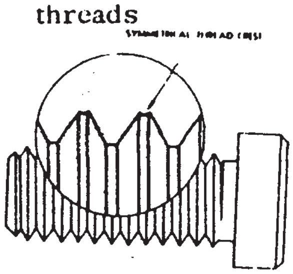
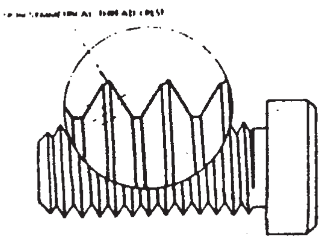
Thread Form



*On single threads, the lead and pitch are the same. On multiple threads (double, triple, quadruple, quintuple, sextuple) the lead is an integral multiple of the pitch.

Screw Thread

(Illustrations courtesy of the Reed Rolled Thread Die Co., Holden, Mass.)

TROUBLE	CAUSE	
	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
Slivers or flakes on threads  	Rolls not in match Center line of rolls not parallel with center line of work Loose or worn cross slide or adapter Overfilling rolls Material not adaptable to cold working Rough finish on blank Seamy stock Feed rate too slow, causing rolls to slip on work Incorrect roll diameter	Center line of roll not parallel with center line of work Loose or worn cross slide or holder Overfilling roll Material not adaptable to cold working Rough finish on blank Seamy stock Feed rate too slow, causing rolls to slip on work Incorrect roll diameter
Drunken threads	Rolls not in match Center line of rolls not parallel with center line of work Inaccurate rolls Work bending during rolling	Center line of roll not parallel with center line of work Inaccurate roll Work bending during rolling
Thread with expanded lead	Expanded lead in rolls Material extruding on short length of blank	Expanded lead in roll Material extruding on short length of blank
Thread with contracted lead	Contracted lead in rolls	Contracted lead in roll

TROUBLE	CAUSE	
	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
Offsize threads		
Pitch Diameter and Major Diameter both oversize	Oversize blanks	Oversize blanks
Pitch Diameter oversize, Major Diameter correct size	Oversize blanks; if finished thread is full, thread on roll is too shallow	Oversize blanks; if finished thread is full, thread on roll is too shallow
Pitch Diameter oversize, Major Diameter undersize	Insufficient squeeze on rolls; if finished thread is full, thread on roll is too shallow	Insufficient penetration of roll in work; if finished thread is full, thread on roll is too shallow
Pitch Diameter correct size, Major Diameter oversize	Blank too large; thread on roll deeper than necessary	Blank too large; thread on roll deeper than necessary
Pitch Diameter correct size, Major Diameter undersize	Blank too small; if finished thread is full, thread on roll is too shallow	Blank too small; if finished thread is full, thread on roll is too shallow
Pitch Diameter undersize, Major Diameter oversize	Too much squeeze; thread on roll deeper than necessary	Too much penetration of roll in work; thread on roll deeper than necessary
Pitch Diameter undersize, Major Diameter correct size	Blank too small; thread on roll deeper than necessary	Blank too small; thread on roll deeper than necessary
Pitch Diameter and Major Diameter both undersize	Blank too small	Blank too small

TROUBLE	CAUSE	
	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
Tapered threads		
Pitch Diameter straight Major Diameter tapered and not filled out on small end	Tapered blank	
Pitch Diameter and Major Diameter both tapered same way	Tapered blank and rolls set up with taper to match	Tapered blank and roll set up with taper to match
Pitch Diameter and Major Diameter tapered in opposite directions and thread not filled out on end with small Major Diameter	Rolls not squeezed tight enough on edge with large Pitch Diameter and small Major Diameter, or work bending during rolling	Work bending during rolling
Out-of-round thread	Out-of-round blank Center line of rolls not parallel with center line of work Feed rate too high Insufficient work revolutions Material not ductile enough for cold-working Not rolling to center line of work	Out-of-round blank Center line of roll not parallel with center line of work Feed rate too high Insufficient work revolutions Material not ductile enough for cold-working
Split thread-axially	Seamy stock Mark from shave tool or hollow mill	Seamy stock Mark from shave tool or hollow mill

TROUBLE	CAUSE	
	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
Poor thread form	Work bending during rolling Rolls not in match Too many work revolutions Excessive or restricted end travel of rolls Center line of rolls not parallel with center line of work Poor thread form in rolls	Work bending during rolling Too many work revolutions Excessive or restricted end travel of roll Center line of roll not parallel with center line of work Poor thread form in rolls
Poor finish on threads	Overfilling rolls Rolls not in match Material accumulated in threads on roll Material not ductile enough for cold-working Chips from other operations between rolls and work Correspondingly poor finish on rolls Rolls that are worn or broken	Overfilling roll Material accumulated in threads on roll Material not ductile enough for cold-working Chips from other operations between rolls and work Correspondingly poor finish on roll Roll that is worn out or broken
Thread filled out in center, but not towards ends, or vice versa	Roll with varying diameter from end to end Blank with varying diameter from end to end Center line of rolls not parallel with center line of work	Roll with varying diameter from end to end Blank with varying diameter from end to end Center line of rolls not parallel with center line of work

TROUBLE	CAUSE	
	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
*Crests not filled out	Blank too small Thread on roll too deep	Blank too small Thread on roll too deep

*Many users do not consider this a serious objection; and by allowing their threads to pass with crests not filled out, overloading of rolls is avoided and roll life is prolonged.

TROUBLE	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
Scuffed crests	Attachment not retracting fast enough Rolls and gear train binding Rolling beyond center line of work Material accumulated in threads on rolls	Holder not retracting fast enough Roll binding Material accumulated in threads on rolls
Hollow work, hole closes in	Machine hole after rolling Needs supporting mandrel Feed rate too high, causing too rapid penetration	Machine hole after rolling Needs supporting mandrel Feed rate too high, causing too rapid penetration
Hollow work, hole enlarged	Machine hole after rolling Supporting mandrel too tight Blank too large on Major Diameter Feed rate too high causing too rapid penetration	Machine hole after rolling Supporting mandrel too tight Blank too large on Major Diameter Feed rate too high causing too rapid penetration

TROUBLE	CAUSE	
	TWO-ROLL GEARED ATTACHMENT	SINGLE ROLL HOLDER
Hollow work, out of round	Machine hole after rolling Feed rate too high causing too rapid penetration Too few work revolutions	Machine hole after rolling Feed rate too high causing too rapid penetration Too few work revolutions
Hollow work, tapered threads due to uneven wall thickness or support from adjacent section	Machine hole after rolling Improper mandrel not giving support where needed Feed rate too high causing too rapid penetration Taper of rolls not great enough to compensate for tendency of work to taper Too thin wall thickness	Machine hole after rolling Improper mandrel not giving support where needed Feed rate too high causing too rapid penetration Taper of roll not great enough to compensate for tendency of work to taper Too thin wall thickness

The foregoing information on Thread Rolling has been reproduced from the Reed Rolled Thread Die Company handbook on thread and form rolling

Wickman
COVENTRY LIMITED

AUTOMATIC HOUSE, Discovery Way, Leofric Business Park, Binley, Coventry CV3 2TD
Telephone +44 (0) 24 7654 7900, Facsimile +44 (0) 24 7654 7950, Web Site: www.wickman.co.uk