THE NEW
AMERICAN MACHINIST'S HANDBOOK

Edited by **RUPERT LE GRAND**
Senior Associate Editor, American Machinist

Based upon earlier editions of "American Machinists' Handbook"

edited by **FRED H. COLVIN** *and* **FRANK A. STANLEY**

Preface

No handbook serving metalworking can be considered up to date unless it has been completely rewritten and revised since 1950.

This is true because of the exceptionally rapid advancement since the last war in the methods for machining, forming, and joining of metals; the improvement in cutting materials; the new constructional steels and tool steels; and the refinements in heat-treatment and finishing. Further, there have been many important developments in gearing, splines, threads, fastening devices, drafting practice, machine-tool components, and power-transmission equipment. In short, the technology, standards, and practices of metalworking have grown and broadened in keeping with the industry's expansion to the number 1 position in the American economy.

These considerations led to the development of "The New American Machinist's Handbook," as successor to that great work, "The American Machinists' Handbook," or "*Colvin & Stanley's*," as it was popularly known. Founded in 1908 by Fred H. Colvin and carried on by him through eight editions, with assistance by Frank A. Stanley, the original "American Machinists' Handbook" was considered a bible by hundreds of thousands of shop men and engineers all over the world.

Like its predecessor, the guiding principle in writing "The New American Machinist's Handbook" has been practicality of subject matter. Information has been drawn from hundreds, perhaps thousands, of sources, then condensed and rewritten into a logical and coherent whole. The purpose has been to create an encyclopedic treatment that would serve both as a reference work and as a text to broaden the knowledge of the reader in the many branches of metalworking activity by means of self-study.

An up-to-date handbook compresses into small space the essentials of important articles, technical papers, engineering and product standards, and proved shop and engineering practice. Needed information is conveniently available, in contrast to the task of hunting through a mass of books, papers, or back issues of magazines, only to find that the wanted material has been mislaid or thrown out. Then, too, the professional handbook writer has many sources of information unknown to the average reader.

In addition to being up-to-date, practical, and comprehensive, a handbook must be written to serve a broad audience. In this case, "The New American Machinist's Handbook" is intended to serve as a supplementary instruction manual for the vocational student and apprentice, and particularly as a reference work and text for machinists, toolmakers, machine-repair men, inspectors, foremen, superintendents, managers, estimators, process engineers, production and manufacturing engineers, tool engineers, product designers, machine designers, draftsmen, purchasing agents, and general executives in the metalworking industry.

A modern format has been adopted for "The New American Machinist's Handbook." Subject matter is divided into 45 sections grouped in 11 parts. For example, Part 1, Machining Methods, consists of 614 pages on 13 machining subjects: broaching; drilling; files and burrs; gears and gear cutting, splines and serrations; grinding processes; milling; planing and shaping; reaming; sawing; threading and thread systems; tapping; turning and boring; and screw-machine work. Part 2, Metal-forming Methods, logically groups spinning; pressworking and cold-roll forming; forging, upsetting, and cold heading; cold working of metals; die casting; and babbitting of bearings. The remaining 26 sections are likewise appropriately grouped.

An important feature of this book is the care with which all information on a single subject is combined in one section. For example, if you are interested in how to drill a specific material, it will be found in Section 2, Drills and Drilling, along with drill types, grinds, and selection, and not scattered through the sections on materials, which are reserved for properties, analyses, and forms.

Acknowledgements. Compilation of a handbook can be achieved only with the active support of many scores of individuals and organizations. In this book I have had the active help and wise counsel of Fred H. Colvin. He prepared certain material and made the index. The *American Machinist* and *Product Engineering* were the sources of most of the material derived from magazine articles. Then, too, much help was given by the AISI, ASA, ASME, SAE, AWS, the Metal Cutting Tool Institute, the Fasteners Institute, and many other associations, private companies, and individuals.

Rupert Le Grand

Contents

Preface v

Part 1 **Machining Methods**

Section 1 Broaches and Broaching 1-1
Section 2 Drills and Drilling 2-1
Section 3 Files and Burrs 3-1
Section 4 Gears, Splines, and Serrations 4-1
Section 5 Grinding, Honing, Lapping, and Superfinishing 5-1
Section 6 Milling and Milling Cutters 6-1
Section 7 Planing and Shaping 7-1
Section 8 Reamers and Reaming 8-1
Section 9 Saws and Sawing 9-1
Section 10 Threading Processes and Thread Systems 10-1
Section 11 Taps and Tapping 11-1
Section 12 Turning and Boring 12-1
Section 13 Screw-machine Work 13-1

Part 2 **Metal-forming Methods**

Section 14 Metal Spinning 14-1
Section 15 Pressworking and Cold-roll Forming 15-1
Section 16 Forging, Upsetting, and Cold Heading 16-1
Section 17 Cold Working of Metals 17-1
Section 18 Die Casting 18-1
Section 19 Babbitting of Bearings 19-1

Part 3 **Assembly Methods**

Section 20 Arc Welding and Surfacing 20-1
Section 21 Gas Welding and Oxygen Cutting 21-1
Section 22 Resistance Welding 22-1
Section 23 Brazing 23-1
Section 24 Soldering and Soft Solders 24-1

Part 4 **Materials**

Section 25 Steels and Irons 25-1
Section 26 Nonferrous Metals 26-1

Section 27 Heat-treatment of Metals 27-1
Section 28 Gages, Tolerances, and Weights of Metal Products 28-1

Part 5 **Finishing of Metals**
Section 29 Metal-cleaning Processes 29-1
Section 30 Plating and Metal Coating 30-1
Section 31 Paint and Painting 31-1
Section 32 Electrolytic and Chemical Finishes for Metals 32-1

Part 6 **Inspection**
Section 33 Limits, Fits, and Tolerances 33-1
Section 34 Measuring and Gaging Equipment 34-1
Section 35 Tapers and Dovetails 35-1

Part 7 **Fastening Devices**
Section 36 Bolts, Screws, Rivets, and Washers 36-1
Section 37 Keys and Pins 37-1

Part 8 **Tool Engineering and Drafting Practice**
Section 38 Jig and Fixture Details 38-1
Section 39 Drafting Practice and Engineering 39-1
Section 40 Surface Finish 40-1

Part 9 **Machine-tool Standards**
Section 41 Standard Machine-tool Elements 41-1
Section 42 Machine-tool Inspection 42-1

Part 10 **Power-transmission Equipment**
Section 43 Chain, V-belts, and Motors 43-1

Part 11 **Mathematics and Tables**
Section 44 Mathematics 44-1
Section 45 Reference Tables 45-1

Index

Part 1

MACHINING METHODS

Section 1

BROACHES AND BROACHING

Consultant: C. MORAWSKI, Chief Tool Designer,
Detroit Broach Company

Definition 1-2
Applications 1-2
Kinds of Broaches 1-2
Broach Design 1-3
Broach Materials 1-3
Pitch of Teeth 1-3
Differential Tooth Spacing 1-3
Gullet Dimensions 1-4
Cut per Tooth 1-4
Face and Back-off Angles 1-5
Land 1-5
Shear Angle 1-5
Chip Breakers 1-6
Burnishing Bars 1-6
Estimating Production 1-7
Sharpening Broaches 1-7
Points in Broach Sharpening 1-7
Trouble-shooting Broaching Troubles 1-8
Cutting Fluids 1-9
Galling and Pickup 1-10

SECTION 1

BROACHES AND BROACHING

Definition. Broaching is a generating process, whereby metal is removed with a multiple-point tool, usually a bar, with tooth height increasing from the starting end. When the broach is pulled or pushed through or over the work, each tooth removes a chip of uniform thickness, in contrast to a milling cutter tooth, which removes a wedge-shaped chip. The chip thickness normally ranges from 0.007 to 0.001 in., depending on whether cutting is being done by the roughing, semifinishing, or finishing teeth. Usually one pass completes the hole or surface.

Applications. Broaching is firmly established as a mass-production process but is used also for short-run jobs. Some broaching machines are specially built to machine one product, like a cylinder block; others permit rapid changes of broaches and work-holding fixtures for a variety of pieces in small lots. And, of course, there are simple machines for occasional keyway-cutting jobs and the like.

Broaching is economical because only a single cut is usually required, and subsequent finishing operations are not needed. The process is also economical because of the number of elements of a surface, external or internal, that can be cut simultaneously. Only limitations are: all elements of a broached surface must be parallel with the broach axis; there must be no obstructions in the plane of the broached surface; and depth of cut is governed by the stroke and tonnage of the machine.

Kinds of Broaches. A pull broach cuts when it is pulled through a hole or over a surface. A push broach cuts when it is pushed through a hole or over a surface.

Broach classification:

1. Method of operation—pull or push.
2. Type of operation—internal or external.
3. Construction—solid or built-up.
4. Function—keyway, surface, round hole, serration, combination round and spline, spline, helical spline, rifling, and burnishing. In helical spline cutting and rifling, the broach may be rotated by a special head, lead bar, and puller adapter.

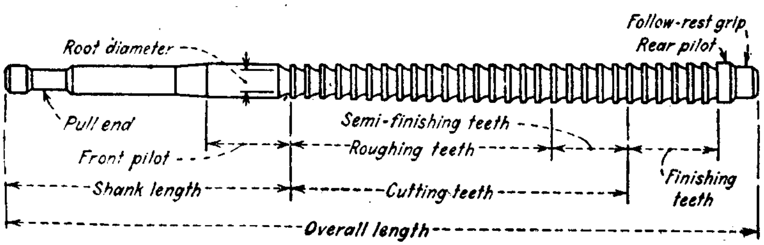


FIG. 1. Typical internal broach of the pull type.

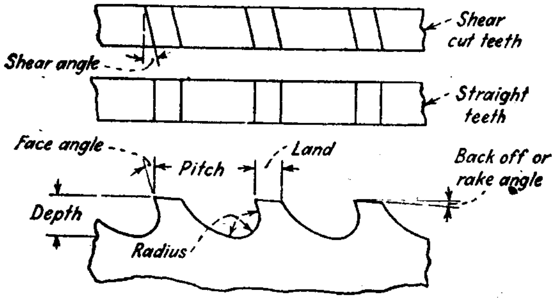


FIG. 2. Broach nomenclature.

BROACH DESIGN

The broach user is interested in broach design as an aid to select the proper tool for a job. Broaches are ordinarily designed by specialists because of problems in correct design for specific circumstances and difficulty of manufacture.

Broach Materials. Steels used in manufacture of broaches are tungsten and moly high-speed steel. Carbide cutting sections have been brazed or inserted in broaches. Some of the most modern broaches use a series of inserts like single-point tools, which can easily be removed, sharpened, and replaced.

Pitch of Teeth. Pitch, or spacing, of teeth (Fig. 2) affects chip space and strength of the tooth and controls the number of teeth in contact with the work and alignment while in the cut. Pitch is determined by length of cut, chip thickness, and material being broached. Cast iron does not require the chip space necessary for steel as cast-iron chips crumble while steel chips curl up and need more room.

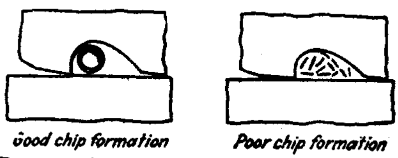


FIG. 3. Chip space depends on the material broached.

The Broaching Tool Institute suggest the formula

$$\text{Pitch} = 0.35\sqrt{\text{length of cut}}$$

EXAMPLE: With a cut 4 in. long, the pitch is $0.35 \times 2 = 0.70$ in. For a 9-in. cut in the same material, the pitch is $0.35\sqrt{9} = 1.05$ in. This gives more total room for chips but not in proportion to the length of the cut. Table 1 gives dimensions of round broaches for various cut lengths.

For successful broaching, at least two teeth should always be engaged with the work, and three is better. This holds true for both horizontal and vertical broaching. It is better to reduce the chip per tooth than to use too few teeth, where the power of the broaching machine is limited.

Differential Tooth Spacing. If a broach vibrates and leaves tooth marks on the work it may be advisable to vary the tooth spacing in groups of two or three teeth. This prevents chatter in internal broaching. On surface broaching, chatter may be caused by contact with more than one surface at once.

Gullet Dimensions. Gullets between teeth should be round at the bottom and have an area ten times the cross-sectional area of the chip for steel. For cast iron the gullet area should be five times that of the chip area. If a spiral chip, as from steel, curls into too small a chip space, added resistance dulls the broach, causes rough work surfaces, and may break teeth and overload machine. Good chip formation (Fig. 3) is essential.

Cut per Tooth. The size of chip removed per tooth depends upon stage of the operation—whether roughing, semifinishing, or finishing. Roughing teeth on broaches for steel should be designed to remove narrow thick chips by use of chip breakers. Such chips are easily removed.

STEELS. In general, the chips removed in roughing steel should be from 0.005 to 0.010 in. thick. The semifinishing section removes chips about 0.005 in. thick; the finishing section takes chips about 0.0005 to 0.002 in. thick.

For free cutting steel use a cut per tooth of 0.004 to 0.006 in. on dia for splines and 0.0015 to 0.003 in. on dia for round holes. For keyway and surface broaching use 0.003 to 0.006 in. per tooth.

NICKEL-ALLOY STEELS. For broaching spline holes, cut per tooth can be 0.004 in. on dia for splines, 0.002 in. on dia for round broaches, 0.004 in. for keyway and surface broaches. Shear angles may be 5 to 20°.

Face angle varies between 8 and 20°, decreasing with hardness. Back-off or rake angle varies between ½ and 2° for internal broaches, up to 3½° for surface broaches.

NITRIDING STEELS. If treated to obtain correct machinability, these steels may be broached with tools used for other alloy steels, but broaching speed may have to be reduced to improve finish and increase broach life. For internal broaching, a cut per tooth of 0.004 to 0.005 in. is recommended. On surface broaching the cut may vary from 0.0025 to 0.0035 in., depending on length, shape, and size of part.

TABLE 1. DIMENSIONS OF ROUND AND SPLINE BROACHES FOR STEEL

Cut Length	Pitch	Land	Depth	Radius
3/16	1/8	3/64	3/64	1/32
1/4	3/16	1/16	1/16	1/32
3/8	7/32	1/16	5/64	3/64
1/2	1/4	1/16	3/32	1/16
3/4	5/16	3/32	1/8	5/64
1	11/32	3/32	1/8	5/64
1 1/8	3/8	7/64	9/64	3/32
1 1/4 to 1 5/8	7/16	1/8	5/32	7/64
1 3/4 to 2	1/2	1/8	3/16	1/8
2 1/4 to 2 1/2	9/16	9/64	7/32	9/64
2 3/4 to 3 1/4	5/8	5/32	15/64	5/32
3 1/2 to 4	11/16	5/32	9/32	3/16
4 1/4 to 4 3/4	3/4	5/32	19/64	3/16
5 to 5 1/2	13/16	5/32	5/16	13/64
5 3/4 to 6	7/8	3/16	11/32	7/32
6 1/2 to 7	15/16	3/16	23/64	7/32
7 1/2 to 8	1	7/32	3/8	1/4
8 1/2 to 10 1/2	1 1/8	1/4	25/64	1/4
11 to 12	1 1/4	9/32	13/32	7/64

FIG. 4.

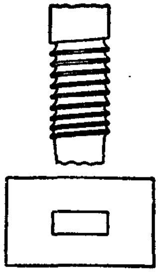


FIG. 5.

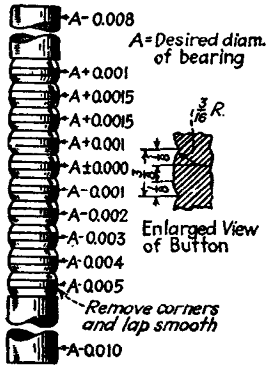


FIG. 6.

FIG. 4. Broach for cutting keyways or slots. FIG. 5. A shear angle may be used on a square-hole broach to avoid vibration. FIG. 6. Burnishing bars are frequently used after broaching of holes. Size of steps depends on the work material.

STAINLESS STEELS. Cut per tooth, 0.001 to 0.005 in. per tooth for round broaches. Speeds, between 8 and 20 fpm. Face angle, 12 to 18°. Back-off, a minimum, or about 2°. Use chip breakers.

NICKEL ALLOYS. For spline broaches the step per tooth should not exceed 0.003 in. for monel and inconel. Depth of cut for round broaches used on material harder than Rockwell 14 C should not exceed 0.0015 in., and for spline broaches 0.002 in. Chip breakers are recommended.

CAST IRONS. A greater cut per tooth can be used than for free-cutting steel. Less chip room is required.

BRASSES AND BRONZES. A somewhat greater cut per tooth than for steel is permissible, and a chip breaker is valuable on roughing teeth.

ALUMINUM AND MAGNESIUM. Larger chip spaces than provided for other metals are desirable. Standard broaches can be used. To overcome trouble in maintaining tolerances, increase the finishing cut to 0.002 in. per tooth.

Face and Back-off Angles. See Table 2. The back-off angle is frequently varied from a maximum at the starting end of the broach to ½° at the finishing end.

Land. Strength of a broach tooth depends on the land, which must be greater for heavy than for light cuts. The teeth are backed off clear to the edge for roughing cuts, varying from ½ to 3° for internal broaches and up to 3½° for surface broaches.

Maintenance of size is helped by not backing off the entire length of the land on the finishing teeth, increasing the straight land from the first finishing tooth. This helps to give longer life to the broach. Too much straight land increases friction, which may expand the work and gall, or abrade, the finished surface.

Shear Angle. A shear angle can be used on a surface broach to give better finish and eliminate vibration. For cast iron, use a shear angle of 20°, for steel forgings

Table 2. Face and Back-off Angles
(Cylindrical and Surface Broaches)

Material Broached	*Degrees*
Face, or Hook, Angle	
Cast iron	6 to 8
Hard steel	8 to 12
Soft steel	15 to 20
Aluminum	10, or more
Brass and bronze	0 to 10, or more
Brittle brass	−5 to +5
Back-off Angle	
Cast iron	2 to 5
Steel:	½ to 2*
Roughing teeth	1½
Finishing teeth	½ to 1†
Brass and bronze:	
Roughing teeth	2
Semifinishing teeth	1
Finishing teeth	½†
Spline broaches:	
Roughing teeth	3
Finishing teeth	1½†
Surface broaches	Up to 3½

* Back-off angle may vary from 2° at the beginning end of broach to ½ at the finishing end. Holding back-off angle to a minimum reduces size loss when the broach is sharpened.

† Part of the land of finishing teeth may be straight and may be graduated from first to last finishing tooth. Size of land determines number of resharpenings possible before the broach is ground under size. Too much land increases cutting friction, causing expansion and galling of broached surfaces on some material.

about 10 to 15°. In slotting, however, a shear cut forces the chips against one side to roughen the surface.

Chip Breakers. Tools that broach tough material and form wide chips should have chip breakers. These are nicks on the roughing-section teeth but are seldom used on the semifinishing section and never on the finishing section of the broach. Chip breakers produce grooves that must be removed by succeeding teeth. As a rule, chip breakers are not used in broaching cast iron, except on extra heavy roughing cuts.

Broaches are generally made of 18-4-1 or 18-4-2 tungsten high-speed steels or from molybdenum high-speed steels, but carbides are also used. For round-hole and rifling broaches, the carbide is shaped in the form of rings and brazed to a bar. But surface broaches are being made of a series of toolholders incorporating removable carbide inserts especially for heavy cuts on cast irons.

BURNISHING BARS

Bearings and bushings are frequently burnished after broaching by a bar with polished buttons (Fig. 6). This tool has ten buttons which increase the diameter by a total of only 0.001 in. The lower or entering end is 0.010 in. below size, the button diameters increasing by 0.001 in. until 0.001 in. oversize is reached. The eighth and ninth buttons are 0.0015 in. oversize, whereas the last is 0.001 in. and the upper end has a clearance of 0.002 in. The diameter of the final button insures a bearing clearance of 0.001 in., even though the metal may close in after the 0.0015-in.

buttons pass. Burnishing buttons are sometimes included in the broach following the cutting teeth.

ESTIMATING PRODUCTION

Normal broaching speed for many types of steel has been set at 30 fpm for the usual hydraulic broaching machine up to 20 tons capacity. Small parts have been broached at more than 40 fpm, but the hydraulic equipment must be increased beyond the economical limit. The higher the tonnage capacity of the machine the slower its economical speed. The range is usually from 4 to 30 fpm, with 18 to 24 fpm usual for average work.

Production depends on the speed of cutting and return, starting and stopping, and the handling of the work in and out of fixtures. Starting and stopping is usually figured at 2 sec and loading at 5 sec. An efficiency of 85% is considered good.

EXAMPLE:

Cutting speed = 24 fpm = 288 ipm
Return speed = 34 fpm = 408 ipm
Stroke = 40 in.

Starting and stopping time = 2 sec
Loading time = 5 sec

$$\text{Cutting time} = \frac{40 \times 60}{288} = 8.33 \text{ sec}$$

$$\text{Return time} = \frac{40 \times 60}{408} = 5.9 \text{ sec}$$

$$\text{Starting and stopping} = 2.0 \text{ sec}$$

$$\text{Loading time} = \underline{5.0 \text{ sec}}$$

$$\text{Complete cycle} = 21.23 \text{ sec}$$

$$\text{Predicted output} = \frac{60 \times 60 \times 85\%}{21.23} = 144 \text{ pcs per hr}$$

SHARPENING BROACHES

It is not necessary to grind all finishing teeth each time a broach is sharpened. Grinding the first one or two teeth is usually sufficient until they have worn under size. Suggestions for sharpening internal and external broaches are given in Fig. 7. The grinding cut on the face of the tooth should blend into the radius as shown. Round broaches can be ground as shown in Fig. 8.

POINTS IN BROACH SHARPENING

1. Maintain original tooth form because design characteristics affect operating efficiency (Fig. 9).

2. Maintain original chip space to permit smooth chip flow.

3. Remove just enough stock to sharpen tooth. Grinding away more material shortens broach life.

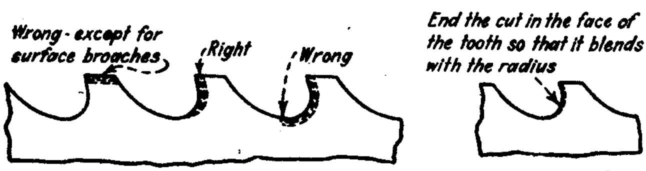


FIG. 7. Incorrect and correct methods of sharpening broach teeth. The gullet must be a smooth curve.

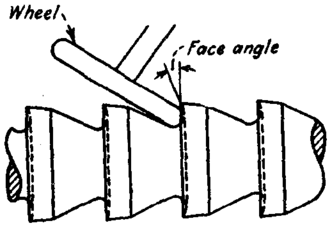


FIG. 8. Wheel plane is set a greater angle than the face angle of the broach tooth, in order to avoid reducing the face angle when sharpening.

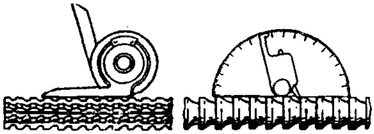


FIG. 9. Face angle of a new broach should be measured before use and checked after sharpening. A combination hook and radius gage, right, determines face angle and proper curvature of chip space.

4. Use blueprint as guide to face angle, back-off angle, tooth depth, radius, and land width.

5. Remove galls and nicks on OD and also on spline sides before sharpening.

6. If you have no broach sharpener, mount a high-speed grinder on the cross-slide of a lathe for sharpening cylindrical broaches on centers. Wheel-head angle = face angle × wheel dia ÷ broach root dia.

7. Do not throw away a broach with a broken tooth. Remove tooth and restep three or four following teeth to distribute load.

8. Back off cylindrical broach teeth only when absolutely necessary because of extremely poor condition. Use a broach sharpener or a good cylindrical grinder to increase back-off angle without changing tooth diameter at cutting edge.

9. Use a steadyrest on a long broach to prevent sag, vibration, and chatter.

10. Measure tooth height carefully when grinding surface-broach lands.

11. Demagnetize the broach if sharpened on a magnetic chuck.

12. Be sure, when setting up broach inserts in a holder, that all contacting surfaces are clean and free from chips, that the teeth blend to give proper chip space where two sections meet, that inserts are not tightened too hard as they are likely to break.

13. Grind dry or wet. In dry grinding, be careful to avoid burning the cutting edges and letting the wheel "spark out."

14. Do not let broach teeth strike any metal surface, as teeth are extremely hard and easily damaged. Store them in wood or lined racks with individual compartments.

15. Use the right type of wheel—Recommended: face grinding—vitrified aluminum oxide disk wheel of 46 to 80 grit with soft or medium bond for roughing and 100 grit for finishing; backing off—vitrified aluminum oxide cup wheel of 60 grit with medium bond. For extremely smooth finish, use finer grain up to 400.

16. Stone cutting edges lightly to remove burrs and gain smoother surfaces, but do not remove enough material to form a negative land.

TROUBLE-SHOOTING BROACHING TROUBLES

Broken Teeth. Packing of chips due to improper grinding may be one cause. On surface broaching a large error in alignment can throw too heavy a load on teeth, causing breakage. Always check holder for straight travel before a part is actually broached. Check steps of inserts with dial indicator.

Spoiled Work or Broken Insert. Check insert assembly in holder to see if screws are too long or too short. If the insert is loose, the screws are too long. If screws are too short and are pulled up with force, the screw hole becomes weak and eventually pulls out.

POOR FINISH AND VARIATION IN SIZE. Look for loose clamps. Check loading fixture and seating of pieces. Improper loading and chip accumulation are causes.

BREAKAGE OF INTERNAL BROACH. Check alignment. See that direction of pull is at right angles to faceplate. Check center axis of broach with axis of faceplate.

DRIFTING. Check the center of the starting hole. It probably is not centralized with broach center.

ROUND OR SPLINE BROACHES CUT OFF CENTER. This is caused by "drifting." On round holes one side does not clean up. On spline broaches the splines will be eccentric. See above recommendation to eliminate drifting.

EXCESSIVE WEAR AND DULLING OF TEETH. Again, this is usually the result of drifting. Also check lubricant. If too rich in sulfur, cut back with paraffin oil.

CHATTER. Inserts may have featheredge and require stoning. Parts not held tight enough. Part vibrates from forcing the cut. Chatter can also develop from using too light a machine. Check hydraulic system.

PARTS WILL NOT HOLD SIZE. Look for something loose while broach is cutting. Part may be springing due to cutting force. Check clamps. Are they strong enough? Check for deflection in machine.

TEARING AND/OR HEAVY BURRS. Dead soft steel is draggy and can be the cause of this condition. Material should be about 28 to 36 Rockwell.

TABLE 3. CUTTING FLUIDS FOR BROACHING VARIOUS MATERIALS

Material Group*	Cutting Fluids†	Material Group*	Cutting Fluids†
1	90 K + 10 M 70 K + 50 M SM-SML	5	10–20 W + 1 SO SM-SML M + (10–15) L LM
2	20–25 W + 1 SO LM	6	10–20 W + 1 SO SM-SML M + (10–15) L LM
3	5–15 W + 1 SO SM-SML M + (10–20) L	7	10–15 W + 1 SO SM-SML M + (10–20) L LM
4	5–15 W + 1 SO SM-SML	8	5–10 W + 1 SO SM-SML M + (10–20) L LM

* Material groups:
1. Aluminum and alloys; al. and zinc die castings
2. Brass
3. Bronze
4. Copper, Everdur, inconel, monel, nickel
5. Wrought and malleable iron
6. Low-carbon and free-cutting steels
7. Medium-carbon and tough low-alloy steel
8. High-carbon high-alloy steels, including stainless

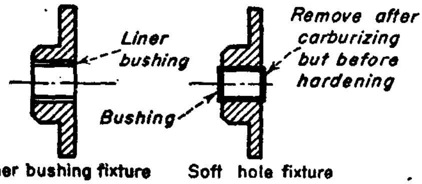
† Cutting fluids:
W = water
K = kerosene
M = mineral oil
LM = straight mineral oil
SM = sulfurized mineral oil
SML = sulfurized mineral lard oil

REASONS FOR PART GALLING AND PICKUP

1. Broach Teeth Damaged

Repair mutilated teeth by: (1) heavy face grind, or (2) send to supplier for OD grind and restepping. Check handling and setup practices. Use follower supports if needed.

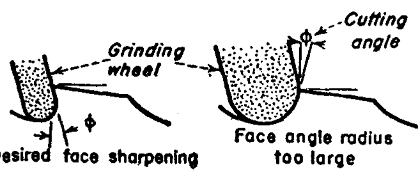
2. Hard Fixtures or Liner Bushings



Use soft (Rockwell C 30-35) liner bushings or soft-hole fixtures. On a carburized and hardened bushing or fixture, the hole can be bored after carburizing to remove the hardening agent before hardening the piece, and thus produce the desired "soft hole" in the hardened part.

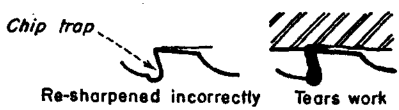
3. Improper Face Angle

4. Negative Rake



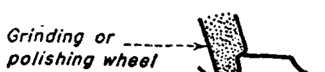
Check sharpening practice, broach print, and broach design. Check face-angle radius on grinding wheel.

5. Deep or Shallow Face-angle Radius



Check sharpening practice and broach print.

6. Rounded Cutting Edges



Check grinding-wheel dressing method. Use bottom of chip space instead of broach OD for steadyrest support.

Section 2

DRILLS AND DRILLING

Twist Drills 2-2
Drill Sizes 2-3
Nomenclature for Twist Drills 2-3
Drills for Reamed Holes 2-5
Drills for Bolt-clearance Holes 2-10
Drills for Dowel-pin Holes 2-10
Drills for Tapped Holes 2-11
Drills for Taper-pin Holes 2-16
Tap Drills for Plastics 2-17
Special Twist Drills and Their Uses 2-17
Deep-hole and Gun Drills 2-18
Carbide Drills 2-19
Carbide-drill Troubles, Causes, and Remedies 2-21
Small-hole Drills 2-22
Rules for Drilling of Microscopic Holes 2-22
Drill Grinding 2-24
Drill-point Angles 2-24
Lip Clearance 2-25
Lip Shape 2-25
Web Thinning 2-25
Hand vs. Machine Grinding 2-26
Hollow Grinding 2-27
Approximate Grinding Method 2-28
Estimating Speeds and Feeds 2-29
Notes on Drilling Various Materials 2-29
Aluminum 2-29
Magnesium 2-29
Copper Alloys 2-30
Cast and Malleable Iron 2-31
Nickel Alloys 2-33
Armor Plate 2-34
Hardened Steel 2-34
Carbon-steel Sheet 2-34
Nickel Steels 2-35
Nitriding Steels 2-39
Stainless Steels 2-36
Plastics 2-36
Titanium 2-36

SECTION 2

DRILLS AND DRILLING

Drills are probably more widely employed than any other tool in the shop, but their selection, grinding, and conditions of use often receive the least care. Cost per hole is frequently higher than it should be, because a drilled hole is considered either a clearance hole for fastening purposes or the starting point for tapping or production of an accurate hole by second operations like reaming or boring.

If accuracy and smoothness are not important in a drilled hole, at least the cost is. Too often a drill is selected merely for size and length. Much drilling can be done satisfactorily with the standard point angle of 118°. Where quantities of holes must be produced, however, refinements in practice are desirable. Factors that bear watching are: (1) machine sharpening that achieves good geometry of the drill point, such as cutting edges of the same length and at equal angles with the drill axis; (2) point angle, lip clearance, and web thickness related to the material being drilled; (3) proper support in a close-fitting drill bushing of suitable length; and (4) machine, chuck or driver, and fixture in good condition. Sometimes special drills are justified.

TWIST DRILLS

The broad definition for a twist drill is "an end-cutting tool having one or more cutting edges, and having helical or straight flutes for the passage of chips." In general practice, a standard twist drill (Fig. 2) has two cutting edges, two helical

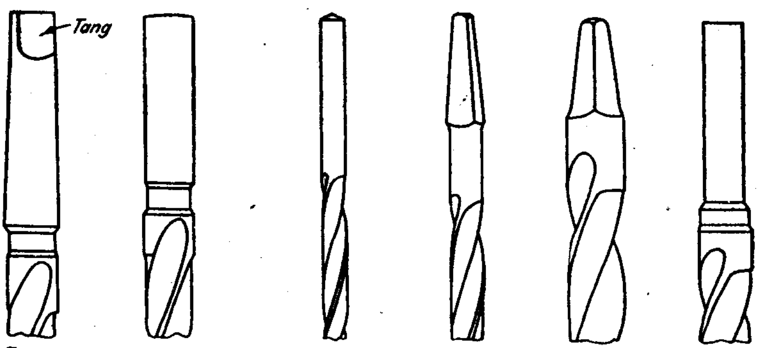


Fig. 1. Conventional shanks for twist drills.