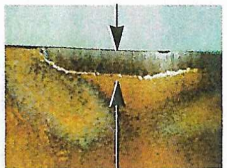
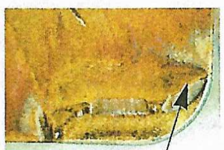
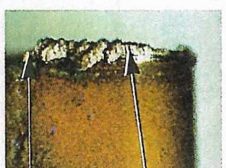
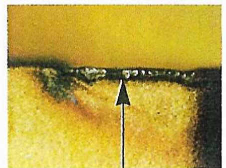
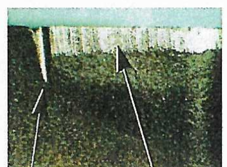
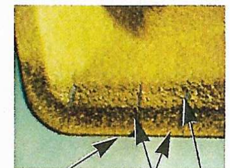
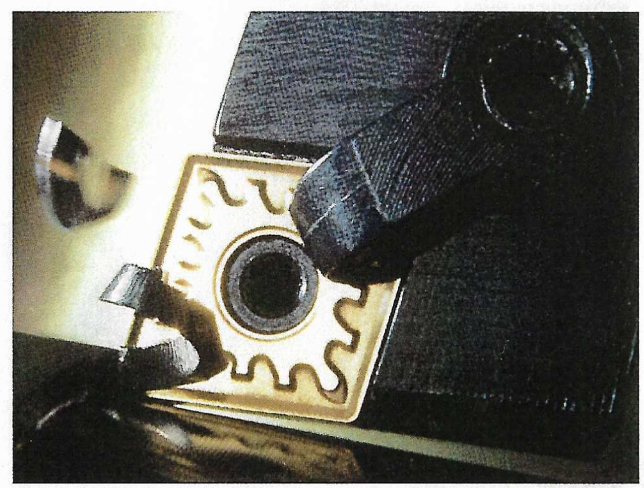
Note that 'G' and 'M' codes (including canned cycles) will vary from machine to machine. For a specific list of codes that would apply to you, refer to your machine's operator manual.

Failure Mode Analysis

VALENITE®

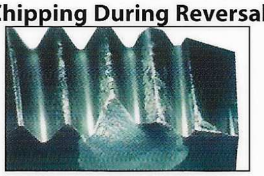
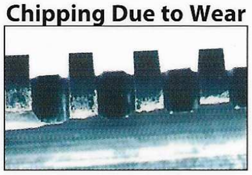
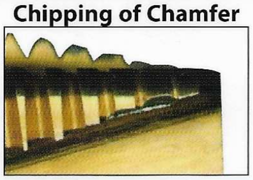
SpectraTurn™

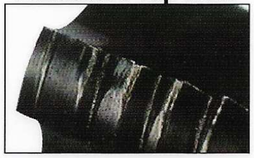
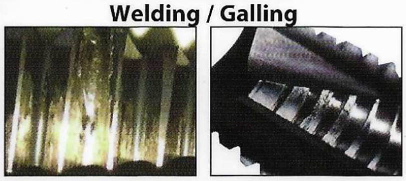
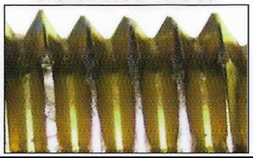
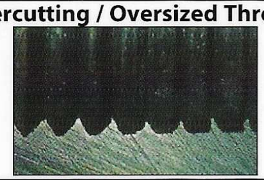
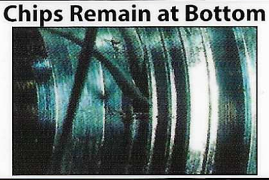
PROBLEM/FAILURE MODE	CAUSE	CONTROL ACTION/REMEDY
Rapid Flank Wear  Flank Wear Width	• Excessive Cutting Speed • Work Material Microstructure Contains Carbides	• Reduce Cutting Speed • Use Harder Grade • Select More Positive Rake Chipbreaker • Flood Cutting Zone w/Coolant
Crater  Localized Crater At Trailing Edge	• Excessive Cutting Speed • Ineffective Use of Coolant	• Reduce Cutting Speed & Feed • Select Harder Grade w/Oxide Coating • Select More Positive Rake Chipbreaker • Flood Cutting Zone w/Coolant
Built-Up Edge, Torn Finish, Chip Welding  Chip At DOC Line Build-Up	• Low Cutting Speed • High Feed Rate • Poor Shearing Action	• Increase Cutting Speed and/or Decreased Feed • Select More Positive Rake Chipbreaker • Select Tougher Grade (Use PVD Coated Insert) • Flood Cutting Zone w/Coolant
Edge Chipping  Chips On Flank	• Excessive Feed Rate • Interrupted Cut	• Reduce Feed Rate • Select Tougher Grade • Select Stronger Chipbreaker • Increase Lead Angle
Excessive Depth-Of-Cut Notching  Notch at DOC Line Flank Wear	• Scale Part • High Work Hardening Materials	• Increase Lead Angle • Increase Cutting Speed • Select Tougher Grade • Select Stronger Chipbreaker • Vary Depth of Cut if Possible
Fracture 	• Improper Selection of Grade/Chip-breaker and/or Cutting Conditions	• Reduce Feed Rate • Select Tougher Grade • Select Stronger Chipbreaker • Make Sure Set-Up is as Rigid as Possible
Thermal Cracking  Small Chip Perpendicular Thermal Cracks Spalling	• Extreme Variation in Cutting Temperatures • Interrupted Cut	• Reduce Feed Rate • Increase Cutting Speed • Select Stronger Chipbreaker
Unacceptable Chip Control (Low Carbon Steel) 	• Low Feed Rate • Large Nose Radius	• Increase Feed Rate • Select Smaller Nose Radius • Decrease Lead Angle



Problems, Causes and Solutions

Tap Troubleshooting

PROBLEM	CAUSE	SOLUTION
Chip Packing (Back Threaded Portion) 	Inappropriate spindle speed	Adjust RPM (lower or higher) for proper chip form
	Helix angle too large	Decrease helix angle or choose tap with low helix angle
	Chips not coiling / breaking properly	Use alternate coating
Chip Packing (Single Thread) 	*Occurs predominantly in horizontal applications*	
	Weak rake angle (positive)	Decrease rake angle
	Chips not evacuating properly	Use a POT style tap or a LHH / RHF
	Chips not coiling / breaking properly	Use alternate coating
Chipping During Reversal 	Chips left behind in flute during tap reversal	Improve wear resistance of tap
		Improve / add surface treatment / coating
	Material shrinkage	Increase coolant volume / concentration to control heat
Chipping Due to Wear 	Tap substrate not suitable for work material	Improve wear resistance of tap
		Improve / add surface treatment / coating
	Cutting action work hardened material	Shorten chamfer length
Chipping of Land Edge 	Occurs when tap either hits bottom or entrance of hole	Avoid hitting the bottom of the hole, check stroke length, alignment and hole size
Chipping of Land Axially 	Occurs when tap either hits bottom or entrance of hole	Avoid hitting the bottom of the hole, check stroke length, alignment and hole size
Chipping of Chamfer 	Tap substrate not suitable for work material	Improve wear resistance of tap
	Inappropriate pre-drill size	Select suitable pre-drill size

PROBLEM	CAUSE	SOLUTION
Premature Tap Wear 	Inappropriate spindle speed	Reduce spindle speed
	Possible work hardening of pre-drilled hole	Prevent work hardening of pre-drilled hole
	Inappropriate thread relief	Use proper thread relief
	Inappropriate chamfer length	Adjust chamfer length
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Welding / Galling 	Inappropriate spindle speed	Reduce spindle speed
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Deformed Lobes 	Possible work hardening of pre-drilled hole	Prevent work hardening of pre-drilled hole
	Inappropriate spindle speed	Reduce spindle speed
	Inappropriate pre-drill size	Increase pre-drill hole size as much as possible
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
	Tap substrate not suitable for material	Improve wear resistance of tap
Tap Breakage 	Possible chip packing	Avoid chip packing
	Inappropriate pre-drill size	Increase pre-drill hole size as much as possible
	Inappropriate spindle speed	Reduce spindle speed
	Possible runout or tapered hole	Reduce runout and assure hole is straight
	Too high of torque generated	Use tap holder with torque adjustment / limiting feature
	Possible tap collision with bottom of hole	Avoid hitting the bottom of the hole, check stroke length, alignment and hole size
Overcutting / Oversized Threads 	Inconsistent feed of spiral fluted style tap	Use compensating tension / compression tap holder
		Adjust feed rate appropriately
		Check CNC program
	Inconsistent feed of spiral pointed style tap	Use compensating tension / compression tap holder
		Adjust feed rate appropriately
		Check CNC program
Tearing on Flanks 	Inappropriate thread relief / rake angle	Use sharper / freer cutting relief and angle
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Extremely Torn Threads 	Possible welding / galling	Select appropriate cutting conditions
	Possible chip packing	Select appropriate cutting conditions
	Inappropriate thread relief	Use sharper thread relief
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Chips Remain at Bottom 	Inappropriate geometry of tap	Reduce chamfer relief angle
		Use thinner land width
		Reduce chamfer length
		Reduce cutting angle

ILLINOIS (HEADQUARTERS)
676 East Fullerton Avenue
Glendale Heights, IL 60139 USA
Toll Free: 800-837-2223
Fax: 800-837-3334

OHIO
3611 Socialville Foster Rd. Ste 102
Mason, OH 45040 USA
Phone: 513-755-3360
Fax: 513-755-3362

CANADA
538 King Forest Court
Burlington, ON L7P 5C1 Canada
Toll Free: 800-263-4861
Fax: 905-632-8466

CALIFORNIA
1921 Miraloma Ave. Suite B
Placentia, CA 92870 USA
Toll Free: 800-837-2223
Fax: 714-528-9209

GEORGIA
5324 Highway 85 Ste 100
Forest Park, GA 30297
Toll Free: 800-837-2223
Fax: 800-837-3334

MEXICO
Avenida Central No. 186
Col. Nueva Industrial Vallejo
07700 Ciudad de Mexico, D.F., Mexico
Phone: (52) 55-51-19-3363
Fax: (52) 55-51-19-3370

G codes (Mills)

Commonly used G codes

G0 · rapid motion
G1 · linear interpolation motion
G2 · CW interpolation motion
G3 · CCW interpolation motion
G4 · dwell
G10 · programmable data input
G17 · XY plane selection
G18 · ZX plane selection
G19 · YZ plane selection
G20 · programming in inches
G21 · programming in millimeters
G28 · return to reference point
G30 · return to secondary reference point
G40 · cutter compensation cancel
G41 · cutter compensation left
G42 · cutter compensation right
G43 · tool length compensation
G54-G59 · work coordinates
G54.1 P_ · extended work coordinates
G65 · macro subroutine call
G80 · canned cycle cancel
G90 · absolute programming
G91 · incremental programming
G98 · initial Z point return
G99 · R plane return

M codes (Mills)

Commonly used M codes

M0 · program stop
M1 · optional stop
M2 · end of program
M3 · spindle CW
M4 · spindle CCW
M5 · spindle stop
M6 · tool change
M8 · coolant on
M9 · coolant off
M19 · orient spindle
M30 · end of program (go to start)
M31 · chip conveyor forward
M33 · chip conveyor stop
M98 · call up sub·program
M99 · end of sub·program

Canned cycles (Mills)

Commonly used canned cycles

G73 · high · speed peck drill cycle
G76 · fine boring cycle
G81 · simple drilling cycle
G82 · drilling cycle with dwell
G83 · peck drilling cycle
G84 · righthand thread tapping cycle
G85 · boring cycle

- all canned cycles are cancelled by using G8Ø

Canned cycles (tool/spindle behavior)

G73 high speed peck drilling: peck feed to depth, optional dwell, rapid out

G74 left hand tapping: feed to depth CCW, reverse motor, feed out CW

G76 fine boring: feed to depth, dwell, stop the spindle, orient, shift over, rapid out

G81 drilling: feed to depth, rapid out

G82 spot drilling: feed to depth, dwell, rapid out

G83 peck drilling: peck feed to depth, optional dwell, rapid out

G84 tapping: feed to depth CW, reverse motor, feed out CCW

G85 boring cycle: feed to depth, feed out

CNC mill letter codes

A: rotational/indexing axis (rotates around the 'X' axis)
B: rotational/indexing axis (rotates around the 'Y' axis)
D: tool radius offset number
F: feed rate
H: tool height/length offset number
I: arc center modification (X axis)
J: arc center modification (Y axis)
K: arc center modification (Z axis)
L: repetition count for sub-programs and fixed cycles
N: block number
O: program number
P: sub-program number, dwell time
Q: depth of peck, shift amount
R: rapid plane in canned cycles
S: spindle speed
T: tool number
X: X axis location
Y: Y axis location
Z: Z axis location

Speed and feed calculations terms

RPM (revolutions per minute): represents, spindle speed. Tells you how many full rotations your spindle will have completed in a minute's time

IPR (inches per revolution): Tells you how far your tool will travel every time it makes a complete rotation/revolution

IPT (inches per tooth): commonly known as chip load. Tells you how much material will be removed by each tooth every revolution

SFM (surface feet per minute): Tells you how many square feet your cutting tool would cover in material, in a minute's time

IPM (inches per minute): how far your tool will go through/along in a minute's time

MRR (material removal rate): how many cubic inches of material your tool removes during the time it spends actually cutting material

Milling

Common formulas

$$RPM = \frac{12 \times SFM}{3.14 \times \text{tool dia.}}$$

$$RPM = SFM \times 3.82 \div \text{tool dia.}$$

$$RPM = SFM \div \text{tool dia.} \div .262$$

$$SFM = \frac{\text{tool dia.} \times 3.14 \times RPM}{12}$$

$$SFM = RPM \times \text{tool dia.} \times .262$$

$$IPM = IPT \times RPM \times \text{number of teeth/flutes}$$

$$IPR = IPM \div RPM$$

$$IPT = \frac{IPM}{RPM \times \text{number of teeth/flutes}}$$

IPM = inches per minute
IPR = inches per revolution
IPT = inches per tooth (chip load)
RPM = revolutions per minute
SFM = surface feet per minute

Milling

SFM starting points for HSS

Aluminum · 200 - 600 SFM
Brass, Bronze · 100 - 200 SFM
Cast iron (soft) · 100 - 130 SFM
Cast iron (medium) · 60 - 90 SFM
Cast iron (ductile) · 55 - 85 SFM
Cast iron (malleable) · 100 - 200 SFM
Copper · 60 - 80 SFM
Magnesium · 125 - 250 SFM
Nickel · based alloys · 15 - 20 SFM
Plastics · 150 - 450 SFM
Stainless steel · 20 - 80 SFM
Steel (low carbon) · 75 - 100 SFM
Steel (medium carbon) · 55 - 85 SFM
Titanium · 40 - 60 SFM
Tool steel · 40 - 60 SFM

- for carbide tools, it's usually safe to double the SFM

Milling

SFM starting points for hard milling

Up to 45 HRC · 600-1000 SFM
45-58 HRC · 400-600 SFM
60+ HRC · 200-400 SFM

IPT starting points for hard milling

Up to 45 HRC · 3%-4% of tool dia.
45-58 HRC · 2%-3% of tool dia.
60+ HRC · 1%-2% of tool dia.

RDOC starting points for hard milling

Up to 45 HRC · 50% of tool dia.
45-58 HRC · 45% of tool dia.
60+ HRC · 45% of tool dia.

ADOC starting points for hard milling

Up to 45 HRC · 10% of tool dia.
45-58 HRC · 7% of tool dia.
60+ HRC · 5% of tool dia.

Carbide vs. HSS tooling

Carbide pros

- typically has longer tool life
- can usually be pushed a lot harder and faster
- retains hardness at higher temperatures
- usually improves surface finish
- more likely to hold its size for longer periods of time
- holds up better against highly abrasive material

Carbide cons

- much more expensive in terms of up-front cost
- extremely hard to sharpen
- doesn't handle interrupted cuts very well
- can't handle very much chatter
- much lower tensile strength
- more brittle, which tends to lead to more chipping and breaking
- not as good on soft material since it's usually not as sharp

Carbide vs. HSS tooling

HSS pros

- much less expensive up front cost
- is capable of holding a much sharper cutting edge
- quite easy to sharpen, or grind a tool down to size
- more flexible and durable in terms of chatter, interrupted cuts, machines with limited RPM and torque, and in handling drill walk

HSS cons

- tends to wear out much faster
- can only be sharpened a handful of times before it affects the tool's stability
- isn't usable on alloys beyond a certain hardness
- usually won't leave as good of a surface finish
- can't push too hard or fast, typically leading to longer cycle times
- won't usually hold its size quite as long

Cutting tools

Common cutting tool materials

carbide tipped · cheaper tough · hardened tool steel for body and shank, and a higher quality carbide cutting edge that's brazed to the body. Good for ferrous and nonferrous cast iron, steel, and steel alloys

Cobalt (M-42 : 8% Cobalt) · better toughness and wear resistance than HSS. A bit more expensive, but can typically be pushed about 10% faster. Good for cast iron, steel, and titanium alloys

high speed steel (HSS) · general purpose, less costly hardened tool steel.

Polycrystalline diamond (PCD) · synthetic shock/wear resistant diamond. Good for high speeds on plastics, nonferrous material, and alloys that have low machinability ratings

powdered metal (PM) · tougher and less brittle than solide carbide. Good for roughing material below 30RC

solid carbide · better heat resistance and rigidity than HSS. Good for high speeds on cast iron, plastics, nonferrous material, and materials rated with low machinability

Vanadium high speed steel (HSSE) · combination of HSS, carbon, vanadium carbide, and some other alloys. Decent wear resistance and toughness. Good for stainless steels and high · silicon aluminum

Cutting tools

Common cutting tool coatings

Aluminum Titanium Nitride (AlTiN)- one of the hardest and abrasion-resistant coatings. Good for carbon steels, cast irons, nickel alloys, stainless steels, titanium, and aerospace/aircraft materials

Titanium Aluminum Nitride (TiAlN)- high hardness and oxidation temperature. Speeds about 75%-100% faster than uncoated. Good for high-alloy carbon steels, stainless steels, titanium alloys, and high-temperature nickel-based alloys.

Titanium Carbonitride (TiCN)- high wear resistance and hardness. Speeds around 75% to 100% faster than uncoated. Likes higher spindle speeds. Good for aluminum alloys, cast irons, and stainless steels

Titanium Nitride (TiN)- general purpose coating. Has decent hardness and heat resistance. Speeds about 25%-30% faster than uncoated.

uncoated- general purpose. Good for non-ferrous material

Zirconium Nitride (ZrN)- high oxidation temperature. Resists sticking and fights against edge build-up. Good for aluminum, brass, copper, and titanium

ISO carbide grade classification

Color codes

blue · steel applications
green · non-ferrous applications
grey · hardened metal applications
orange · exotic material, high-temp alloys, super alloys
red · cast iron applications
yellow · stainless steel applications

Letter codes

H · hardened material
K · cast iron
M · stainless steel
N · non-ferrous material
P · steel
S · exotic material, high-temp alloys, super alloys

Numeric codes

0-50 · wear/shock resistance
the closer to 0, the more wear resistance. The closer to 50, the more shock resistance

Drilling

Common formulas

$$RPM = \frac{12 \times SFM}{3.14 \times \text{tool dia.}}$$

$$RPM = SFM \times 3.82 \div \text{tool dia.}$$

$$RPM = SFM \div \text{tool dia.} \div .262$$

$$SFM = \frac{\text{tool dia.} \times 3.14 \times RPM}{12}$$

$$SFM = RPM \times \text{tool dia.} \times .262$$

$$IPM = RPM \times IPR$$

$$IPR = IPM \div RPM$$

$$MRR = \text{tool dia.} \times IPR \times SFM \times 3$$

IPM = inches per minute
IPR = inches per revolution
MRR = material removal rate (in^3/min.)
RPM = revolutions per minute
SFM = surface feet per minute

Drilling

SFM starting points for HSS

Aluminum · 200 - 400 SFM
Brass, Bronze · 150 - 300 SFM
Bronze (high tensile) · 70 - 150 SFM
Cast iron (soft) · 75 - 125 SFM
Cast iron (medium) · 50 - 100 SFM
Cast iron (hard chilled) · 10 - 20 SFM
Cast iron (malleable) · 80 - 90 SFM
Copper · 60 - 175 SFM
Die casting (Zinc · base) · 300 - 400 SFM
Magnesium · 250 - 400 SFM
Nickel · based alloys · 30 - 50 SFM
Plastics · 100 - 300 SFM
Stainless steel · 20 - 50 SFM
Steel (forgings) · 40 - 50 SFM
Steel (low carbon) · 80 - 110 SFM
Steel (medium carbon) · 70 - 80 SFM
Steel (high tensile 35-40Rc) · 30 - 50 SFM
Steel (high tensile 40-45Rc) · 15 - 25 SFM
Titanium · 20 - 30 SFM

- for carbide drills, it's usually safe to double the SFM

Drilling

Inches per revolution

.001" - .002" (smaller than 1/8")
.002" - .004" (1/8" - 1/4")
.004" - .007" (1/4" - 1/2")
.007" - .015" (1/2" - 1")
.015" - .025" (larger than 1")

Drill point length

60° · .866 × drill dia.
82° · .575 × drill dia.
90° · .500 × drill dia.
118° · .300 × drill dia.
120° · .288 × drill dia.
135° · .207 × drill dia.

Fractional drills
Decimal equivalents

1/64" : .0156"
1/32" : .0313"
3/64" : .0469"
1/16" : .0625"
5/64" : .0781"
3/32" : .0938"
7/64" : .1094"
1/8" : .125"
9/64" : .1406"
5/32" : .1563"
11/64" : .1719"
3/16" : .1875"
13/64" : .2031"
7/32" : .2188"
15/64" : .2344"
1/4" : .250"
17/64" : .2656"
9/32" : .2813"
19/64" : .2969"
5/16" : .3125"
21/64" : .3281"
11/32" : .3438"
23/64" : .3594"
3/8" : .375"
25/64" : .3906"
13/32" : .4063"

Fractional drills
Decimal equivalents

27/64" · .4219"
7/16" · .4375"
29/64" · .4531"
15/32" · .4688"
31/64" · .4844"
1/2" · .500"
33/64" · .5156"
17/32" · .5313"
35/64" · .5469"
9/16" · .5625"
37/64" · .5781"
19/32" · .5938"
39/64" · .6094"
5/8" · .625"
41/64" · .6406"
21/32" · .6563"
43/64" · .6719"
11/16" · .6875"
45/64" · .7031"
23/32" · .7188"
47/64" · .7344"
3/4" · .750"
49/64" · .7656"
25/32" · .7813"
51/64" · .7969"
13/16" · .8125"

Fractional drills

Decimal equivalents

53/64": .8281"
27/32": .8438"
55/64": .8594"
7/8": .875"
57/64": .8906"
29/32": .9063"
59/64": .9219"
15/16": .9375"
61/64": .9531"
31/32": .9688"
63/64": .9844"

Lettered drills
Decimal equivalents

A .234" (15/64")
B .238"
C .242"
D .246"
E .250" (1/4")
F .257"
G .261"
H .266" (17/64")
I .272"
J .277"
K .281" (9/32")
L .290"
M .295"
N .302"
O .316"
P .323"
Q .332"
R .339"
S .348"
T .358"
U .368"
V .377"
W .386"
X .397"
Y .404"
Z .413"

Numbered drills

Decimal equivalents

#1 · .228"
#2 · .221"
#3 · .213"
#4 · .209"
#5 · .2055"
#6 · .204"
#7 · .201"
#8 · .199"
#9 · .196"
#10 · .1935"
#11 · .191"
#12 · .189"
#13 · .185"
#14 · .182"
#15 · .180"
#16 · .177"
#17 · .173"
#18 · .1695"
#19 · .166"
#20 · .161"
#21 · .159"
#22 · .157"
#23 · .154"
#24 · .152"
#25 · .1495"
#26 · .147"

Numbered drills
Decimal equivalents

#27. .144"
#28. .1405"
#29. .136"
#30. .1285"
#31. .120"
#32. .116"
#33. .113"
#34. .111"
#35. .110"
#36. .1065"
#37. .104"
#38. .1015"
#39. .0995"
#40. .098"
#41. .096"
#42. .0935"
#43. .089"
#44. .086"
#45. .082"
#46. .081"
#47. .0785"
#48. .076"
#49. .073"
#50. .070"
#51. .067"
#52. .0635"

Numbered drills

Decimal equivalents

#53 · .0595"
#54 · .055"
#55 · .052"
#56 · .0465"
#57 · .043"
#58 · .042"
#59 · .041"
#60 · .040"
#61 · .039"
#62 · .038"
#63 · .037"
#64 · .036"
#65 · .035"
#66 · .033"
#67 · .032"
#68 · .031"
#69 · .0292"
#70 · .028"
#71 · .026"
#72 · .025"
#73 · .024"
#74 · .0225"
#75 · .021"
#76 · .020"
#77 .018"
#78 .016"
#79 · .0145"
#80 · .0135"

Reaming

SFM starting points for HSS

Aluminum · 90 - 175 SFM
Brass, Bronze · 90 - 175 SFM
Cast iron (soft) · 50 - 85 SFM
Cast iron (medium) · 25 - 50 SFM
Cast iron (ductile) · 50 - 70 SFM
Cast iron (malleable) · 50 - 70 SFM
Copper · 90 - 175 SFM
Magnesium · 90 - 175 SFM
Nickel · based alloys · 25 - 35 SFM
Plastics · 90 - 175 SFM
Stainless steel · 15 - 35 SFM
Steel (forgings) · 35 - 40 SFM
Steel (low carbon) · 50 - 70 SFM
Steel (medium carbon) · 40 - 50 SFM
Steel (high tensile 35 - 40 Rc) · 35 - 40 SFM
Steel (high tensile 40 - 45 Rc) · 20 - 30 SFM
Titanium · 20 - 30 SFM

- for carbide tools, it's usually safe to double the SFM

Reaming

Suggested stock removal

.001" - .002"	(smaller than 1/32")
.003" - .006"	(1/32" - 1/8")
.005" - .009"	(1/8" - 1/4")
.007" - .012"	(1/4" - 3/8")
.010" - .015"	(3/8" - 1/2")
.015" - .025"	(1/2" - 3/4")
.025" - .031"	(3/4" - 1")

- 2% - 3% of reamer diameter is usually fine for stock removal

Burraway Speeds & Feeds

100-160 SFM, .005-.008 IPR · HSS on aluminum
250-400 SFM, .008-.012 IPR · carbide on aluminum
80-130 SFM, .005-.008 IPR · HSS on steel
240-270 SFM, .005-.008 IPR · carbide on steel
40-50 SFM, .005-.008 IPR · HSS on tool steel
60-120 SFM, .005-.008 IPR · carbide on tool steel
40-50 SFM, .005-.008 IPR · HSS on steel forgings
60-120 SFM, .005-.008 IPR · carbide on steel forgings
80-90 SFM, .005-.008 IPR · HSS on malleable iron
80-180 SFM, .005-.008 IPR · carbide on malleable iron
30-50 SFM, .005-.008 IPR · HSS on monel metal
80-180 SFM, .005-.008 IPR · carbide on monel metal
30-50 SFM, .005-.008 IPR · HSS on stainless
80-250 SFM, .005-.008 IPR · carbide on stainless
25-45 SFM, .005-.008 IPR · HSS on titanium
80-250 SFM, .005-.008 IPR · carbide on titanium
40-60 SFM, .005-.008 IPR · HSS on cast iron
105-240 SFM, .008-.012 IPR · carbide on cast iron
100-300 SFM, .005-.01 IPR · HSS on brass & bronze
175-300 SFM, .006-.01 IPR · carbide on brass & bronze
100-300 SFM, .005-.01 IPR · HSS on plastic/nylatron
175-300 SFM, .006-.01 IPR · carbide on plastic/nylatron
-not recommended- HSS on composites
150-200 SFM, .001-.01 IPR · carbide on composites

E·Z Burr Speeds & Feeds

75-126 SFM, .003-.01 IPR · HSS free machining
230-260 SFM, .008-.015 IPR · carbide free machining
90-150 SFM, .003-.008 IPR · HSS on aluminum
230-260 SFM, .008-.015 IPR · carbide on aluminum
40-60 SFM, .003-.01 IPR · HSS on cast iron
230-260 SFM, .008-.015 IPR · carbide on cast iron
60-100 SFM, .004-.011 IPR · HSS on low carbon
230-260 SFM, .008-.015 IPR · carbide on low carbon
45-80 SFM, .003-.01 IPR · HSS on med. carbon
200-230 SFM, .006-.012 IPR · carbide on med. carbon
20-40 SFM, .003-.01 IPR · HSS on stainless
180-220 SFM, .004-.008 IPR · carbide on stainless
25-50 SFM, .003-.01 IPR · HSS on high alloy
180-220 SFM, .006-.012 IPR · carbide on high alloy

Formula

Standard · $RPM = \frac{SFM \times 3.82}{dia.}$

$IPM = RPM \times IPR$

$SFM = \frac{RPM \times dia.}{3.82}$

Metric · $RPM = \frac{M/min. \times 318.47}{dia.}$

$MM/Min = RPM \times MM/REV.$

$M/Min = \frac{RPM \times dia.}{318.47}$

Tapping

ISO thread tolerance callouts

- loose · 1A (external), 1B (internal)
- medium · 2A (external), 2B (internal)
- tight · 3A (external), 3B (internal)

Common chamfer lengths

- bottoming · 1-2 tapered threads
- plug · 3-5 tapered threads
- taper · 8-10 tapered threads

Common cutting angles

- straight flute · chips go wherever
- spiral flute · chips are forced upward
- spiral point · chips are forced downward

Common pitch diameter limits

- L1 · Basic to -.0005"
- H1 · Basic to +.0005"
- H2 · Basic +.0005" to +.001"
- H3 · Basic +.001" to +.0015"
- H4 · Basic +.0015" to +.002"
- H5 · Basic +.002" to +.0025"
- H6 · Basic +.0025" to +.003"
- H7 · Basic +.003" to +.0035"

Tapping

Common thread forms

- ACME-G · general purpose Acme thread
- NPT · American Standard taper pipe thread
- UN · Unified and American thread series (thread dia. w/constant pitch)
- UNC · Unified and American Coarse thread series (thread dia. w/coarse pitch)
- UNEF · Unified and American Extra-Fine thread series (thread dia. w/extra-fine pitch)
- UNF · Unified and American Fine thread series (thread dia. w/ fine pitch)

UNC/UNF taps

- $RPM = \dfrac{SFM \times 3.82}{\text{tap dia.}}$
- $IPR = \dfrac{1.00''}{\text{threads per inch}}$
- $IPM = \dfrac{RPM}{\text{threads per inch}}$

M/MF taps

- $RPM = \dfrac{SFM \times 97.028}{\text{tap dia. (mm)}}$
- $IPR = \text{pitch (mm)} \times .03937$
- $IPM = RPM \times \text{pitch (mm)} \times .03937$

Tapping

General tap drill size

- Cut taps

  tap drill size = tap basic major dia. − pitch

- Form taps

  $$\text{tap drill size} = \text{basic major dia.} - \frac{\text{pitch}}{2}$$

UNC/UNF special thread % reqs.

- Cut taps

  $$\text{drill size} = \text{basic major dia.} - \frac{.01299 \times \%\text{ of thread}}{\text{threads per inch}}$$

- Form taps

  $$\text{drill size} = \text{basic major dia} - \frac{.0068 \times \%\text{ of thread}}{\text{threads per inch}}$$

M/MF special thread % reqs.

- Cut taps

  $$\text{drill size (mm)} = \text{basic major dia.} - \frac{\%\text{ of thread} \times \text{pitch (mm)}}{76.98}$$

- Form taps

  $$\text{drill size (mm)} = \text{basic major dia.} - \frac{\%\text{ of thread} \times \text{pitch (mm)}}{147.06}$$

Tapping

SFM starting points

- Aluminum · cast alloys · 60 SFM
- Aluminum · diecast alloys · 70 SFM
- Aluminum · wrought alloys · 80 SFM
- Brass · 60-100 SFM
- Bronze · 30-40 SFM
- Copper · 60-80 SFM
- Gun metal · 50-60 SFM
- Cast iron · grey · 30-60 SFM
- Cast iron · ductile · 50 SFM
- Cast iron · alloys · 15-30 SFM
- Malleable alloys · 20-40 SFM
- Magnesium alloys · 50-70 SFM
- Nickel alloys · 10 SFM
- Nimonic alloys · 10-12 SFM
- Mild steel · 30-50 SFM
- Steel · medium carbon · 35 SFM
- Tool steels · 15-25 SFM
- Cast steel · 25 SFM
- Titanium alloys · 10 SFM
- Stainless steel · 300 series · 10-20 SFM
- Stainless steel · 400 series · 15 SFM
- Plastic · 50-70 SFM
- Zinc · diecasting · 80 SFM

Turning insert identification

- turning inserts typically have ten designations, excluding its grade

-

1	2	3	4	5	6	7	8	9		10
C	N	M	G	12	04	08	(E)	(N)	-	MP

- 1. insert shape
  - A. parallelogram 85°
  - B. parallelogram 82°
  - C. rhombic 80°
  - D. rhombic 55°
  - E. rhombic 75°
  - F. rhombic 50°
  - H. hexagonal
  - K. parallelogram 55°
  - L. rectangular
  - M. rhombic 86°
  - O. octagonal
  - P. pentagonal
  - R. round
  - S. square
  - T. triangular
  - V. rhombic 35°
  - W. trigon
  - X. special design

Turning insert identification

· 2· relief angle

A· 3°
B· 5°
C· 7°
D· 15°
E· 20°
F· 25°
G· 30°
N· 0°
P· 11°
O· other relief angle

· 3· tolerance class

	cutting point	thickness	inscribed circle
A	± .0002"	± .001"	± .001"
C	± .0005"	± .001"	± .001"
E	± .001"	± .001"	± .001"
F	± .0002"	± .001"	± .0005"
G	± .001"	± .005"	± .001"
H	± .0005"	± .001"	± .0005"
J	± .0002"	± .001"	± .002" - ± .006"
K	± .0005"	± .001"	± .002" - ± .006"
L	± .001"	± .001"	± .002" - ± .006"
M	± .003" - ± .008"	± .005"	± .002" - ± .006"
N	± .003" - ± .008"	± .001"	± .002" - ± .006"
U	± .005" - ± .015"	± .005"	± .003" - ± .01"

Turning insert identification

· 4. chipbreaker and clamping system

A	Y	cylindrical	N
B	Y	cylindrical	N
C	Y	cylindrical	N
F	N	N	double·sided
G	Y	cylindrical	double·sided
H	Y	one countersink 70·90°	one·sided
J	Y	×2 countersink 70·90°	double·sided
M	Y	cylindrical	one·sided
N	N	N	N
Q	Y	cylindrical	N
R	N	N	one·sided
T	Y	one countersink 40·60°	one·sided
U	Y	×2 countersink 40·60°	double·sided
W	Y	cylindrical	N
X	custom	custom	custom
	hole	hole configuration	chip breaker

Turning insert identification

· 5· insert size (inscribed circle)

1.2 · 5/32" (.1563")
1.5 · 3/16" (.1875")
1.8 · 7/32" (.2188")
.236 · 6mm (.2362")
2 · 1/4" (.250")
2.5 · 5/16" (.3125")
.315 · 8mm (.315")
3 · 3/8" (.375")
394 · 10mm (.3937")
3.5 · 7/16" (.4375")
472 · 12mm (.4724")
4 · 1/2" (.500")
4.5 · 9/16" (.5625")
5 · 5/8" (.625")
.630 · 16mm (.6299")
5.5 · 11/16" (.6875")
6 · 3/4" (.750")
.787 · 20mm (.7874")
7 · 7/8" (.875")
.984 · 25mm (.9843")
8 · 1"
10 · 1 1/4" (1.250")
1.250 · 32mm (1.2598")

Turning insert identification

·6· insert thickness

.5 · 1/32" (.0313")
.6 · .040"
1 · 1/16" (.0625")
1.2 · 5/64" (.0781")
1.5 · 3/32" (.0938")
2 · 1/8" (.125")
2.5 · 5/32" (.1563")
3 · 3/16" (.1875")
3.5 · 7/32" (.2188")
4 · 1/4" (.250")
5 · 5/16" (.3125")
6 · 3/8" (.375")
7 · 7/16" (.4375")
8 · 1/2" (.500")

Turning insert identification

· 7· corner radius

X0· .0015"

0· .004"

.5· .008"

1· 1/64" (.0156")

2· 1/32" (.0313")

3· 3/64" (.0469")

4· 1/16" (.0625")

5· 5/64" (.0781")

6· 3/32" (.0938")

7· 7/64" (.1094")

8· 1/8" (.125")

00· round insert (inch)

M0· round insert (metric)

A· square w/45° chamfer

D· square w/30° chamfer

E· square w/15° chamfer

K· square w/15° double chamfer

N· truncated triangle insert

P· flattened corner triangle

Turning insert identification

- 8. cutting edge condition
  - E: round cutting edges
  - F: sharp cutting edges
  - S: chamfered and rounded cutting edges
  - T: chamfered cutting edges

- 9. cutting direction
  - L: left handed
  - N: neutral
  - R: right handed

- 10. chip breaker
  - chip breaker labels vary from company to company. For example, Kennametal usually labels their medium machining chip breakers as MN, whereas Sandvik uses the callout PM usually

Dwelling

- dwell commands are called out with a GØ4
- dwell time is usually called out by a 'P' value in the same block as your GØ4
- excluding a decimal point in your 'P' value will tell your machine how many milliseconds to dwell
- adding a decimal point after the 'P' value will convert that 'P' value to seconds
- one second = 1,ØØØ milliseconds

Examples

GØ4 P3 = three milliseconds
GØ4 P3. = three seconds
GØ4 P1ØØØ = one thousand milliseconds (1 sec.)
GØ4 P1ØØØ. = one thousand seconds

Fanuc controllers

GØ4 X___ = seconds
GØ4 U___ = seconds
GØ4 P___ = milliseconds

Misc. controllers

GØ4 F___ = seconds
GØ4 S___ = spindle revolutions/rotations

- (Dwell revs.) × (6Ø,ØØØ ÷ RPM) = P value
- seconds to dwell for one spindle revolution = 6Ø/RPM

Tapping formulas

$$\text{standard RPM} = \frac{\text{SFM} \times 3.82}{\text{tap dia.}}$$

$$\text{standard IPR} = \frac{1''}{\text{TPI}}$$ (threads per inch)

$$\text{standard IPM} = \frac{\text{RPM}}{\text{TPI}}$$

$$\text{Metric RPM} = \frac{\text{SFM} \times 97.028}{\text{tap dia. (in MM)}}$$

$$\text{Metric IPR} = \text{pitch (in MM)} \times .03937$$

$$\text{Metric IPM} = \text{RPM} \times \text{pitch (in MM)} \times .03937$$

Hardness

HRB/HRC · Rockwell hardness
BHN/HB · Brinell hardness

HRC to HB conversion

21-30 HRC · $HB = (HRC \times 5.970) + 104.7$
31-40 HRC · $HB = (HRC \times 8.570) + 27.6$
41-50 HRC · $HB = (HRC \times 11.158) - 79.6$
51-60 HRC · $HB = (HRC \times 17.515) - 401$

HRB to HB conversion

55-69 HRB · $HB = (HRB \times 1.646) + 8.7$
70-79 HRB · $HB = (HRB \times 2.394) - 42.7$
80-89 HRB · $HB = (HRB \times 3.297) - 114$
90-100 HRB · $HB = (HRB \times 5.582) - 319$

True position

True position calculation

$$\text{true position} = 2 \times \sqrt{XVAR^2 + YVAR^2}$$

XVAR: a feature's locational deviation from the basic dimension in 'X'
YVAR: a feature's locational deviation from the basic dimension in 'Y'

Example
Let's say you have a .01" true position tolerance on a feature. That feature is off .002" in 'X' and .003" in 'Y' from nominal

$$\text{true position} = 2 \times \sqrt{XVAR^2 + YVAR^2}$$

$$= 2 \times \sqrt{.002^2 + .003^2}$$

$$= 2 \times \sqrt{4E\text{-}6 + 9E\text{-}6}$$

$$= 2 \times \sqrt{.000004 + .000009}$$

$$= 2 \times \sqrt{1.3E\text{-}5}\ (.000013)$$

$$= 2 \times .0036$$

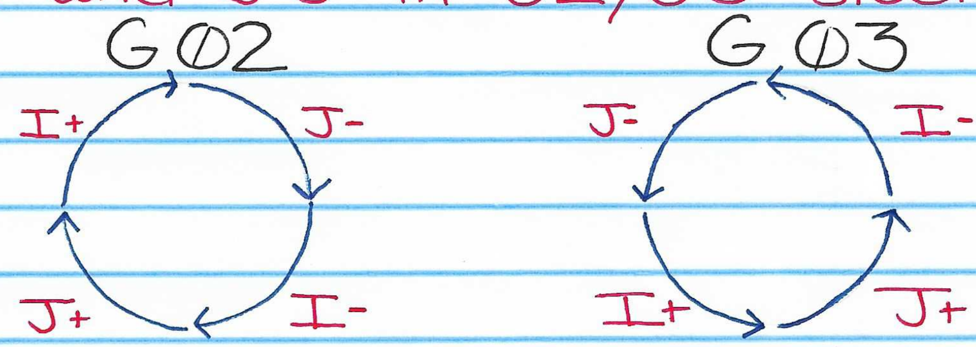
$$= .0072"$$ (you're in tolerance!)

GØ2's and GØ3's

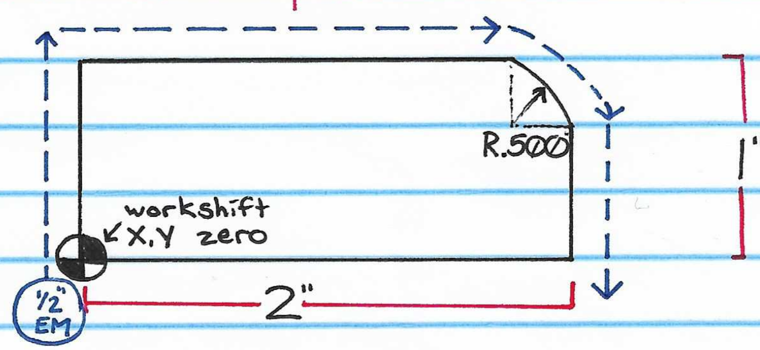
G2: CW circular interpolation
G3: CCW circular interpolation

- 'X' and 'Y' values in your G2/G3 line mark the end point of the arc about to be swung
- 'I' and 'J' mark the distance between the starting point of the arc to its center

I's and J's in G2/G3 blocks



Program example



GØ1Y1.25Ø F3Ø.
X1.5ØØ
GØ2X2.25ØY.5ØØ
GØ1Y-.25Ø

Blueprint reading
Common callouts

- ◎ · concentricity · basically a 3·D version of roundness. Tells how closely a feature/object comes to being a perfect cylinder. Not only does it need to be round, but it also needs to be straight along its axis.
- ⌴ counterbore · indicates a counter-bore. Should be accompanied by an O.D. and depth callout
- ⌵ countersink · indicates a counter-sink. Should be accompanied by a depth and degree callout
- ⌭ cylindricity · states that when revolving, all points of a particular surface should be equally distant from a common axis
- ↧ depth · indicates that a dimension applies to the depth of that feature
- ⌀ diameter · indicates that a dimension applies to the full width of a circular feature
- ▱ flatness · indicates that a surface should have all elements one a single plane

Blueprint reading

Common callouts

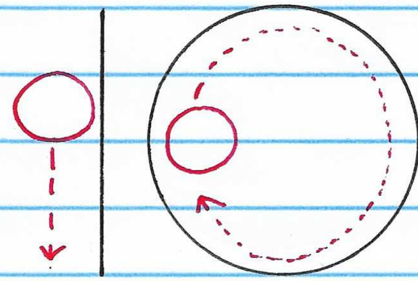
- // parallelism - indicates that all parts of a surface, line, or axis are equally distant from a datum plane or axis
- ⊥ perpendicularity - indicates that the condition of a surface, line, or axis should be 90° from a datum plane or axis
- ⌖ position tolerance - indicates a zone in which the axis or center plane of a given feature is allowed to deviate from true position
- ○ roundness - a 2-D tolerance that controls how closely a cross-section of a cone, cylinder, or sphere is to being a perfect circle
- — straightness - indicates that an element of a surface or axis should reside on a straight line
- √ surface finish - how smooth or rough a surface can be

Cutter compensation

- Initiated with a G41 or G42
- Cancelled with a G40
- Must have a specified diameter offset
- G41 (cutter comp. left) will move the cutter to the left when you add a positive amount to the designated radial wear, and will move it to the right when adding a negative amount
- G42 (cutter comp. right) will move the cutter to the right with a positive number, and to the left with a negative number
- to get an idea of how your cutter is going to move, picture yourself walking behind the cutter as it travels through the material
- the amount you add to your radial wear will be taken off both sides of certain features; such as circles, and slots that take two passes to and fro to create

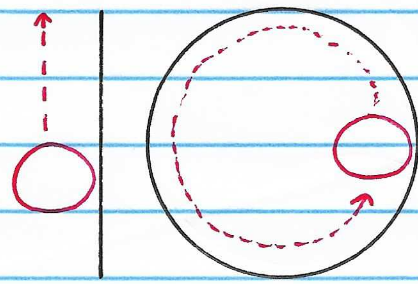
Cutter compensation

G41



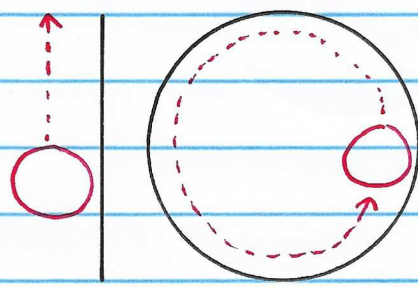
- a positive number will move the cutter left, removing more material
- a negative number will take less material off

G41



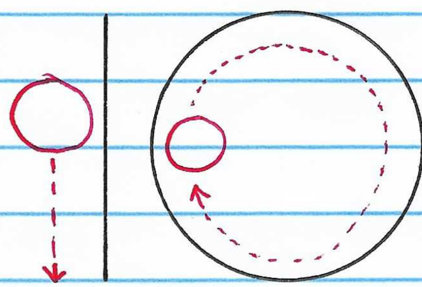
- \+ take less material
- − take more material

G42



- \+ take more material
- − take less material

G42



- \+ takes less material
- − takes more material

Surface finish

U.S. Ra

2,000Ra · either very good rough cast surfaces, or very rough machining surfaces

1,000Ra · will have very obvious tool marks

500Ra · will have obvious tool marks

250Ra - 320Ra · will have visible tool marks

125Ra - 200Ra · tool marks won't be obvious, but visible nonetheless

63Ra - 100Ra · blurry tool marks, but tool travel direction's obvious

32Ra - 50Ra · tool travel direction is blurry, but still visible

16Ra - 25Ra · tool travel direction is blurry

8Ra - 12.5Ra · tool travel direction is no longer visible

4Ra · dark, glossy finish

Machine shop language
Commonly used terms

Blind hole · a hole put into a part that doesn't go completely through it
Burr · a rough and/or sharp edge left behind on a feature after machining
Chamfer · an angle cut into the edge/corner of a feature
Chatter · vibrations between your tool and part
Concentric · how close two or more features are to sharing the same center
Deburring · removing sharp edges
Feed · how quickly a tool goes into/across a part
Flute · the groove in a cutting tool that lets chips get out, and lets coolant get to the cutting edges while it runs
Kerf · the width of cut made by a saw
I.D. · refers to the inner diameter of a feature
O.D. · refers to the outside diameter of a feature
Tapping · the process of adding threads to a part/feature

Misc.

Circles

circumference = radius × 2 × π
= diameter × π

diameter = radius × 2
= circumference ÷ π

radius = diameter ÷ 2
= circumference ÷ (π × 2)

Metrics

1M (meter) = 39.37" (inches)

1CM (centimeter) = .3937"

1MM (millimeter) = .03937"

25.4MM = 1"

MM = inches × 25.4

inches = MM × .03937

CM = inches × 2.54

inches = CM × .3937

M = inches × .0254

inches = M × 39.37

M = feet × .3048

feet = M × 3.2808

U.S. standard

1' (foot) = 12" (inches)

1" = $.08\overline{3}$'

inches = feet × 12

feet = inches × $.08\overline{3}$

Pre-loading Offsets

Tools

G1ØG9ØL__ PØØ__ R__

- L1Ø = height geometry
- L11 = height wear
- L12 = radius geometry
- L13 = radius wear

- P value = tool/offset number

- R value = offset amount

Workshifts

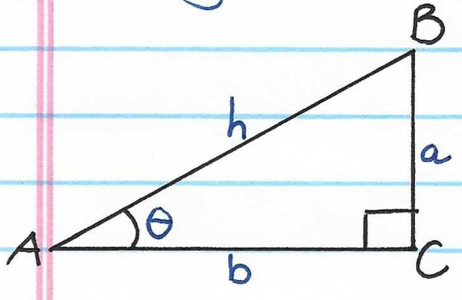
G1ØG9ØL__ P__ X__ Y__ Z___

- L2 = standard workshifts
- L2Ø = G54.1 P__ workshifts

- L2PØ = common workshift
- L2P1 = G54
- L2P2 = G55
- L2P3 = G56
- L2P4 = G57
- L2P5 = G58
- L2P6 = G59

- L2ØP1 = G54.1 P1
- L2ØP2 = G54.1 P2
- L2ØP3 = G54.1 P3

Trigonometry



Pythagorean theorem

$a^2 + b^2 = h^2$ (c^2)

a = opposite side
b = adjacent side
h = hypotenuse (commonly labelled 'c')
COS = cosine
COT = cotangent (uncommon)
CSC = cosecant (uncommon)
TAN = tangent
SEC = secant (uncommon)
SIN = sine

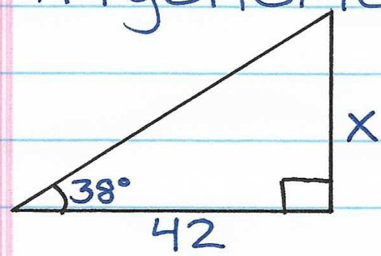
$$\text{SIN } \theta = \frac{\text{opposite side}}{\text{hypotenuse}}$$

$$\text{COS } \theta = \frac{\text{adjacent side}}{\text{hypotenuse}}$$

$$\text{TAN } \theta = \frac{\text{opposite side}}{\text{adjacent side}}$$

- note that the sum of a triangle's three angles will always equal 180°

Trigonometry example



- $TAN\,\Theta = \dfrac{\text{opposite}}{\text{adjacent}}$

- $TAN(38°) = \dfrac{X}{42}$

- to get X alone, we need to multiply both sides of the equation by 42

- $42 \times TAN(38°) = X$

- $42 \times .7813 = X$

- $32.8146 = X$

- calculator needs to be in degree mode to do this calculation

Tangents

01°	.0175
02°	.0349
03°	.0524
04°	.0699
05°	.0875
06°	.1051
07°	.1228
08°	.1405
09°	.1584
10°	.1763
11°	.1944
12°	.2126
13°	.2309
14°	.2493
15°	.2679
16°	.2867
17°	.3057
18°	.3249
19°	.3443
20°	.3640
21°	.3839
22°	.4040
23°	.4245
24°	.4452
25°	.4663
26°	.4877
27°	.5095
28°	.5317
29°	.5543
30°	.5774
31°	.6009
32°	.6249
33°	.6494
34°	.6745
35°	.7002
36°	.7265
37°	.7536
38°	.7813
39°	.8098
40°	.8391
41°	.8693
42°	.9004
43°	.9325
44°	.9657
45°	1.0000
46°	1.0355
47°	1.0724
48°	1.1106
49°	1.1504
50°	1.1918
51°	1.2349
52°	1.2799
53°	1.3270
54°	1.3764

Tangents

55°	1.4281
56°	1.4826
57°	1.5399
58°	1.6003
59°	1.6643
60°	1.7321
61°	1.8040
62°	1.8807
63°	1.9626
64°	2.0503
65°	2.1445
66°	2.2460
67°	2.3559
68°	2.4751
69°	2.6051
70°	2.7475
71°	2.9042
72°	3.0777
73°	3.2709
74°	3.4874
75°	3.7321
76°	4.0108
77°	4.3315
78°	4.7046
79°	5.1446
80°	5.6713
81°	6.3138
82°	7.1154
83°	8.1443
84°	9.5144
85°	11.4301
86°	14.3007
87°	19.0811
88°	28.6363
89°	57.2900

Sines

01°	.0175	28°	.4695
02°	.0349	29°	.4848
03°	.0523	30°	.5000
04°	.0698	31°	.5150
05°	.0872	32°	.5299
06°	.1045	33°	.5446
07°	.1219	34°	.5592
08°	.1392	35°	.5736
09°	.1564	36°	.5878
10°	.1736	37°	.6018
11°	.1908	38°	.6157
12°	.2079	39°	.6293
13°	.2250	40°	.6428
14°	.2419	41°	.6561
15°	.2588	42°	.6691
16°	.2756	43°	.6820
17°	.2924	44°	.6947
18°	.3090	45°	.7071
19°	.3256	46°	.7193
20°	.3420	47°	.7314
21°	.3584	48°	.7431
22°	.3746	49°	.7547
23°	.3907	50°	.7660
24°	.4067	51°	.7771
25°	.4226	52°	.7880
26°	.4384	53°	.7986
27°	.4540	54°	.8090

Sines

55°	.8192
56°	.8290
57°	.8387
58°	.8480
59°	.8572
60°	.8660
61°	.8746
62°	.8829
63°	.8910
64°	.8988
65°	.9063
66°	.9135
67°	.9205
68°	.9272
69°	.9336
70°	.9397
71°	.9455
72°	.9511
73°	.9563
74°	.9613
75°	.9659
76°	.9703
77°	.9744
78°	.9781
79°	.9816
80°	.9848
81°	.9877
82°	.9903
83°	.9925
84°	.9945
85°	.9962
86°	.9976
87°	.9986
88°	.9994
89°	.9998

Cosines

01°	.9998	28°	.8829
02°	.9994	29°	.8746
03°	.9986	30°	.8660
04°	.9976	31°	.8572
05°	.9962	32°	.8480
06°	.9945	33°	.8387
07°	.9925	34°	.8290
08°	.9903	35°	.8192
09°	.9877	36°	.8090
10°	.9848	37°	.7986
11°	.9816	38°	.7880
12°	.9781	39°	.7771
13°	.9744	40°	.7660
14°	.9703	41°	.7547
15°	.9659	42°	.7431
16°	.9613	43°	.7314
17°	.9563	44°	.7193
18°	.9511	45°	.7071
19°	.9455	46°	.6947
20°	.9397	47°	.6820
21°	.9336	48°	.6691
22°	.9272	49°	.6561
23°	.9205	50°	.6428
24°	.9135	51°	.6293
25°	.9063	52°	.6157
26°	.8988	53°	.6018
27°	.8910	54°	.5878

Cosines

55°	.5736
56°	.5592
57°	.5446
58°	.5299
59°	.5150
60°	.5000
61°	.4848
62°	.4695
63°	.4540
64°	.4384
65°	.4226
66°	.4067
67°	.3907
68°	.3746
69°	.3584
70°	.3420
71°	.3256
72°	.3090
73°	.2924
74°	.2756
75°	.2588
76°	.2419
77°	.2250
78°	.2079
79°	.1908
80°	.1736
81°	.1564
82°	.1392
83°	.1219
84°	.1045
85°	.0872
86°	.0698
87°	.0523
88°	.0349
89°	.0175

Changing Tool Rotation on SL-25

1.) Press "Custom" key
2.) Enter "Rotary Tool" menu by pressing '3' then "input"
3.) Navigate to desired tool
4.) Press 'Ø' or '1' accordingly, then press "input"

- Mill 'Z' tools require a 'Ø'
- Mill 'X' tools require a '1'

CNC lathe letter codes

- A · rotational axis (rotates around the X axis)
- B · rotational axis (rotates around the Y axis)
- C · rotational axis (rotates around the Z axis)
- D · canned cycle data (depth of cut)
- E · precision feed rate (useful for threading)
- F · feed rate (typically inches per revolution)
- I · arc center in X axis for G2 or G3 commands. Also canned cycle data
- J · arc center in Y axis for G2 or G3 commands. Also canned cycle data.
- K · arc center in Z axis for G2 or G3 commands. Also canned cycle data.
- N · block number
- O · program name
- P · G and M code parameter. Also canned cycle data
- Q · peck increment in canned cycles. Also canned cycle data.
- R · retract height. Also can define size of arc radius.

CNC lathe letter codes

- S: speed (represents RPM in G97 mode, SFM in G96 mode)
- T: tool selection
- U: incremental X axis moves
- V: incremental Y axis moves
- W: incremental Z axis moves
- X: position of X axis
- Y: position of Y axis
- Z: position of Z axis

G codes (lathes)

Commonly used G codes

G0· rapid motion
G1· linear interpolation motion
G2· CW interpolation motion
G3· CCW interpolation motion
G4· dwell
G10· programmable data input
G20· programming in inches
G21· programming in millimeters
G28· return to reference point
G30· return to secondary reference point
G32· thread cutting
G70· finishing machining cycle
G71· turning canned cycle
G72· facing canned cycle
G96· constant surface speed
G97· constant surface speed cancel
G98· feed rate per minute
G99· feed rate per revolution

M codes (lathes)

Commonly used M codes

M0 · program stop
M1 · optional stop
M2 · end of program
M3 · spindle CW
M4 · spindle CCW
M5 · spindle stop
M8 · coolant on
M9 · coolant off
M29 · rigid tap mode
M30 · end of program (go to start)
M40 · spindle gear at middle
M41 · low gear select
M42 · high gear select
M68 · hydraulic chuck close
M69 · hydraulic chuck open
M78 · tailstock advance
M79 · tailstock reverse
M98 · sub·program call
M99 · end of sub·program

Turning canned cycles

G71 O.D./I.D. stock removal

G71 U__ R__ ;

G71 F_ I_ K_ P_ Q_ S_ T_ U_ W__ ;

first block

U= depth of cut each pass in 'X'

R= retract distance after each pass

second block

F= feed rate throughout cycle

I= amount of material left in 'X' for the last roughing pass (should be a negative number for I.D. turning, positive for O.D.)

K= amount of material left in 'Z' for the last roughing pass (usually positive)

P= first line of repeated sub·routine

Q= last line of repeated sub·routine

S= spindle speed throughout cycle

T= tool and offset number to use throughout the cycle

U= amount of material left in 'X' for the finish pass (- for I.D., + for O.D.)

W= amount of material left in 'Z' for the finish pass (usually +)

- a good chunk of the second block letter callouts are optional. Will vary a little bit from machine to machine

Turning canned cycles

G72 facing canned cycle

```
G00 X2. Z.2 ;
G72 W.04 R.05;
G72 P100 Q102 U0. W.005 F.01;
N100 G00 Z0. ;
G1 X-.064;
N102 Z.2;
```

first block

W= depth of cut each pass in 'Z'
R= retract distance after each pass

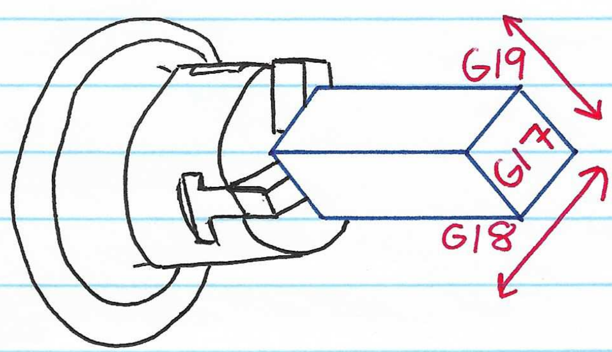
second block

P= first block of repeated sub·routine
Q= last block of repeated sub·routine
U= amount of material left in 'X' for the finish pass (should be zero in facing cycles)
W= amount of material left in 'Z' for the finish pass (usually a positive number)
F= feed rate throughout cycle

CNC lathe live tooling

Commonly used M codes

M14 - main spindle clamp
M15 - main spindle un-clamp
M19 - orient spindle
M133 - live tool drive forward
M134 - live tool drive reverse
M135 - live tool drive stop
M154 - C-axis engage
M155 - C-axis disengage



- In M133 and M134 blocks, RPM is represented by a P value, rather than a typical S

CNC lathe live tooling

Cross working cycles

G75 - peck drill cycle (G19, G98)
G195 - radial rigid tap cycle (G19, G99)
G196 - radial rigid tap left (G19, G99)
G241 - drill cycle (G19, G98)
G242 - drill cycle w/ dwell (G19, G98)
G243 - peck drill cycle (G19, G98)

Face working cycles

G81 - drill cycle (G18, G98)
G82 - drill cycle w/dwell (G18, G98)
G83 - peck drill cycle (G18, G98)
G95 - rigid tap cycle (G18, G99)
G186 - rigid tap left cycle (G18, G99)

X-C axis milling manual slots (G18, G98)
X-Y axis milling manual radius (G17, G98)

G17 - X-Y plane
G18 - X-Z plane
G19 - Y-Z plane
G98 - feed rate per minute
G99 - feed rate per revolution

Turning

Common formulas

RPM = SFM × 3.82 ÷ machining diameter

RPM = SFM ÷ machining diameter ÷ .262

SFM = RPM × machining diameter × .262

MRR = D.O.C. × feed × SFM × 12

RPM = revolutions per minute
SFM = surface feet per minute
MRR = material removal rate (in^3/min.)
D.O.C. = depth of cut

Turning

SFM starting points for HSS

Aluminum · 200 - 300 SFM
Brass, Bronze · 100 - 200 SFM
Bronze (high tensile) · 40 - 60 SFM
Cast iron · 50 - 80 SFM
Copper · 60 - 80 SFM
Stainless steel · 40 - 50 SFM
Steel (low carbon) · 80 - 110 SFM
Steel (medium carbon) · 60 - 80 SFM
Steel (high carbon) · 35 - 40 SFM

- for carbide tooling, it's usually safe to double the SFM

Threading

SFM starting points for HSS

Aluminum · 50 - 60 SFM
Brass, Bronze · 40 - 50 SFM
Bronze (high tensile) · 20 - 25 SFM
Cast iron · 20 - 25 SFM
Copper · 20 - 25 SFM
Stainless steel · 15 - 20 SFM
Steel (low carbon) · 35 - 40 SFM
Steel (medium carbon) · 25 - 30 SFM
Steel (high carbon) · 15 - 20 SFM

- for carbide tooling, it's usually safe to double the SFM

SFM starting points

120-250 SFM · HSS drills on aluminum
80-120 SFM · HSS drills on standard steels
500-700 SFM · carb. drills on aluminum
200-300 SFM · carb. drills on standard steels
200-400 SFM · HSS general milling on aluminum
80-180 SFM · HSS general milling on standard steels
400-600 SFM · carb. general milling on aluminum
160-300 SFM · carb. general milling on standard steels
150-200 SFM · .002" IPT · c-boring aluminum w/ HSS endmill
70-100 SFM · .0015" IPT · c-boring standard steels w/ HSS endmill
250-400 SFM · .002" IPT · c-boring aluminum w/ carbide endmill
100-200 SFM · .0015" IPT · c-boring standard steels w/ carbide endmill

Misc.

- Date and time change on VM3 (Haas)
  1.) Press CRNT COMDS
  2.) Page up/down until date and time appear
  3.) Press Emergency Stop
  4.) Type in current date (MM-DD-YYYY) or time (HH:MM)
  5.) Press WRITE/ENTER
  6.) Reset Emergency Stop

- G82- Spot drill canned cycle
  - Requires dwell time

- Dwelling (P)
  - Ex: P3ØØ = 1/3 of a sec.
    P1ØØØ = 1 sec.
    P2ØØØ = 2 sec.
  - adding a decimal before the number will convert to milliseconds
  - adding a decimal after the number will cause the machine to dwell for that many seconds (Ex.: P1ØØØ. = 1,ØØØ sec.)

- G84- Tapping canned cycle
  - Insert a J to change the retracting speed
  - Ex: J2 will retract twice the plunge feed rate
  - A one or zero won't affect anything

Text Engraving (G47)PØ (Haas)

Ex:
```
T2
M6
GØG17G2ØG4ØG8ØG9ØXØYØG54
G43H2.Z1.1M3S55ØØ
G47PØ(Jeremy)X2.Y-4.IØJ.5R.1Z-.ØØ5F15.E5.
GØG8ØZ2.
M5
G28G91ZØHØ
M3Ø
```

E = Plunge feed rate
F = Engraving feed rate
I = Angle of rotation (default is zero)
J = Height of text (default is 1in. Min = .ØØ1)
P = Ø for literal string engraving
1 for sequential serial number engraving
32-126 for ASCII characters
R = Return plane
X = 'X' start of engraving
Y = 'Y' start of engraving
Z = Depth of cut

use any: A-Z character
a-z character
Ø-9
`~!@#$%^&*-_=+?.\,:[]/<>;{}

Text Engraving (cont.) (G47)P32-126 (Haas)

- Engraving special characters
- P values for specific characters

32. "space"
33. !
34. "
35. #
36. $
37. %
38. &
39. '
40. (
41.)
42. *
43. +
44. ,
45. -
46. .
47. /
48-57. 0-9
58. :
59. ;
60. <
61. =
62. >
63. ?
64. @
65-90. A-Z
91. [
92. \
93.]
94. ^
95. _
96. `
97-122. a-z
123. {
124. |
125. }
126. ~

Ex: To engrave $2.00, you need two lines of code. First would use 'P36' to engrave the '$', and the second would use P0(2.00). The 'X' and 'Y' start point for second line needs to be shifted over to create a space.

Misc. (Haas)

- Running a program in graphics mode:
  1.) Select the program
  2.) Enter 'MEM' or 'MDI' mode
  3.) Press 'SETNG/GRAPH' twice
  4.) Press 'Cycle Start'

- Duplicating a program:
  1.) Press 'LIST PROG'
  2.) Select the program
  3.) Type in new program number
  4.) Press 'F1'
  5.) Select "duplicate program/file"

- Changing background color (Haas)
  1.) Press 'SETNG/GRAPH'
  2.) Go to option 198
  3.) Choose a number between Ø and 254

- Homing out one axis
  1.) Enter letter of axis
  2.) Press 'HOME G28'

  or

  1.) Go into 'MDI' mode
  2.) Type in "G28(axis letter)ØHØ"
  3.) EOB, Insert, Cycle Start

  Ex: G28ZØHØ; → Cycle Start

Vol 2

Tap Drill Sizes and Pitch Diameter Limits

Tapping Guide

Tap Size	Threads Per Inch			Minor Diameter		Tap Drill Diameter - Cut Taps				
	UNC	UNF	8-Pitch	Min. 2B	Max. 2B	80% Thread (in)	75% Thread (in)	70% Thread (in)	65% Thread (in)	60% Thread (in)
0	–	80	–	0.0465	0.0514	0.0470	0.0478	0.0486	0.0494	0.0503
1	64	–	–	0.0561	0.0623	0.0568	0.0578	0.0588	0.0598	0.0608
	–	72	–	0.0580	0.0635	0.0586	0.0595	0.0604	0.0613	0.0622
2	56	–	–	0.0667	0.0737	0.0674	0.0686	0.0698	0.0709	0.0721
	–	64	–	0.0691	0.0752	0.0698	0.0708	0.0718	0.0728	0.0738
3	48	–	–	0.0764	0.0845	0.0774	0.0787	0.0801	0.0814	0.0828
	–	56	–	0.0797	0.0865	0.0804	0.0816	0.0828	0.0839	0.0851
4	40	–	–	0.0849	0.0939	0.0860	0.0876	0.0893	0.0909	0.0925
	–	48	–	0.0894	0.0968	0.0904	0.0917	0.0931	0.0944	0.0958
5	40	–	–	0.0979	0.1062	0.0990	0.1006	0.1023	0.1039	0.1055
	–	44	–	0.1004	0.1079	0.1014	0.1029	0.1043	0.1058	0.1073
6	32	–	–	0.1040	0.1140	0.1055	0.1076	0.1096	0.1116	0.1136
	–	40	–	0.1110	0.1190	0.1120	0.1136	0.1153	0.1169	0.1185
8	32	–	–	0.1300	0.1390	0.1315	0.1336	0.1356	0.1376	0.1396
	–	36	–	0.1340	0.1420	0.1351	0.1369	0.1387	0.1405	0.1424
10	24	–	–	0.1450	0.1560	0.1467	0.1494	0.1521	0.1548	0.1575
	–	32	–	0.1560	0.1640	0.1575	0.1596	0.1616	0.1636	0.1656
12	24	–	–	0.1710	0.1810	0.1727	0.1754	0.1781	0.1808	0.1835
	–	28	–	0.1770	0.1860	0.1789	0.1812	0.1835	0.1858	0.1882
1/4	20	–	–	0.1960	0.2070	0.1980	0.2013	0.2045	0.2078	0.2110
	–	28	–	0.2110	0.2200	0.2129	0.2152	0.2175	0.2198	0.2222
5/16	18	–	–	0.2520	0.2650	0.2548	0.2584	0.2620	0.2656	0.2692
	–	24	–	0.2670	0.2770	0.2692	0.2719	0.2746	0.2773	0.2800
3/8	16	–	–	0.3070	0.3210	0.3101	0.3141	0.3182	0.3222	0.3263
	–	24	–	0.3300	0.3400	0.3317	0.3344	0.3371	0.3398	0.3425
7/16	14	–	–	0.3600	0.3760	0.3633	0.3679	0.3726	0.3772	0.3818
	–	20	–	0.3830	0.3950	0.3855	0.3888	0.3920	0.3953	0.3985
1/2	13	–	–	0.4170	0.4340	0.4201	0.4251	0.4301	0.4351	0.4400
	–	20	–	0.4460	0.4570	0.4480	0.4513	0.4545	0.4578	0.4610
9/16	12	–	–	0.4720	0.4900	0.4759	0.4813	0.4867	0.4921	0.4976
	–	18	–	0.5020	0.5150	0.5048	0.5084	0.5120	0.5156	0.5192
5/8	11	–	–	0.5270	0.5460	0.5305	0.5364	0.5423	0.5482	0.5541
	–	18	–	0.5650	0.5780	0.5673	0.5709	0.5745	0.5781	0.5817
3/4	10	–	–	0.6420	0.6630	0.6461	0.6526	0.6591	0.6656	0.6721
	–	16	–	0.6820	0.6960	0.6851	0.6891	0.6932	0.6972	0.7013
7/8	9	–	–	0.7550	0.7780	0.7595	0.7668	0.7740	0.7812	0.7884
	–	14	–	0.7980	0.8140	0.8008	0.8054	0.8101	0.8147	0.8193
1	8	–	–	0.8650	0.8900	0.8701	0.8782	0.8863	0.8945	0.9026
	–	12	–	0.9100	0.9280	0.9134	0.9188	0.9242	0.9296	0.9351
1-1/8	7	–	–	0.9700	0.9980	0.9765	0.9858	0.9951	1.0044	1.0137
	–	12	–	1.0350	1.0530	1.0384	1.0438	1.0492	1.0546	1.0601
	–	–	8	0.9900	1.0150	0.9951	1.0032	1.0113	1.0195	1.0276
1-1/4	7	–	–	1.0950	1.1230	1.1015	1.1108	1.1201	1.1294	1.1387
	–	12	–	1.1600	1.1780	1.1634	1.1688	1.1742	1.1796	1.1851
	–	–	8	1.1150	1.1400	1.1201	1.1282	1.1363	1.1445	1.1526
1-3/8	6	–	–	1.1950	1.2250	1.2018	1.2126	1.2235	1.2343	1.2451
	–	12	–	1.2850	1.3030	1.2884	1.2938	1.2992	1.3046	1.3101
	–	–	8	1.2400	1.2650	1.2451	1.2532	1.2613	1.2695	1.2776
1-1/2	6	–	–	1.3200	1.3500	1.3268	1.3376	1.3485	1.3593	1.3701
	–	12	–	1.4100	1.4280	1.4134	1.4188	1.4242	1.4296	1.4351
	–	–	8	1.3650	1.3900	1.3701	1.3782	1.3863	1.3945	1.4026
1-5/8	–	–	8	1.4900	1.5150	1.4951	1.5032	1.5113	1.5195	1.5276
1-3/4	5	–	–	1.5330	1.5670	1.5422	1.5551	1.5681	1.5811	1.5941
	–	–	8	1.6150	1.6400	1.6201	1.6282	1.6363	1.6445	1.6526
1-7/8	–	–	8	1.7400	1.7650	1.7451	1.7532	1.7613	1.7695	1.7776
2	4-1/2	–	–	1.7590	1.7950	1.7691	1.7835	1.7979	1.8124	1.8268
	–	–	8	1.8650	1.8900	1.8701	1.8782	1.8863	1.8945	1.9026
2-1/4	4-1/2	–	–	2.0090	2.0450	2.0191	2.0335	2.0479	2.0624	2.0768
	–	–	8	2.1150	2.1400	2.1201	2.1282	2.1363	2.1445	2.1526
2-1/2	4	–	–	2.2290	2.2670	2.2402	2.2564	2.2727	2.2889	2.3051
	–	–	8	2.3650	2.3900	2.3701	2.3782	2.3863	2.3945	2.4026

Tap Drill Sizes: Fractional Cut Taps

To minimize tapping problems and lengthen tool life, use the largest drill possible to produce a minor diameter that will result in the lowest percentage of full thread consistent with adequate strength. A minor diameter that provides a 55% to 65% thread is sufficient for most requirements, but in some cases a higher percentage of thread may be necessary to conform with the minor diameter limits of the thread class specified.

Suggested Percentage of Full Thread in Tapped Holes

Material		*Deep Hole Tapping	Average Commercial Work	Thin Sheet Stock or Stamping
Free Cutting	Aluminum, Brass, Bronze, Cast Iron, Copper, Mild Steel, Tool Steel	60% - 70%	65% - 70%	75% - 85%
Hard or Tough Cutting	Cast Steel, Drop Forging, Monel Metal, Nickel Steel, Stainless Steel	55% - 65%	60% - 70%	

* Generally, deeper than 1 1/2 times the hole diameter.

Formula: Tap Drill Size

$$\text{Drill Size} = \text{Tap Major Dia} - \frac{0.01299 \times \%\text{ of Full Thread}}{\#\text{ of Threads Per Inch}}$$

Example: Determine Drill Size for 2"—12N Tap, 70% Full Thread.

Basic Major Diameter of Tap = 2.0000"

0.01299 x 70 = 0.9093 ÷ 12 = 0.0758"

Drill Size = 1.9242"

Formula: Percentage of Full Thread

$$\%\text{ of Full Thread} = \text{Threads Per Inch} \times \frac{\text{Tap Major Dia} - \text{Drill Dia}}{0.01299}$$

Example: Determine the % of Full Thread for 2"—12N Tap, using 1.9242" Drill.

Threads Per Inch = 12

2.000-1.9242 = 0.0758 ÷ 0.01299 = 5.835

Percentage of Full Threads = 70%

Pitch Diameter Limit: Internal Screw Thread Classes and Tap Recommendations

Size	Threads Per Inch		Basic Pitch Diameter	Unified Classes of Thread				American National Classes of Thread			
				CLASS 2B For General Applications		CLASS 3B For Closer Fits		CLASS 2		CLASS 3	
	UNC	UNF/ UNS	All Classes Minimum	Pitch Diam. Limits Maximum	Rec. Taps	Pitch Diam. Limits Maximum	Rec. Taps	Pitch Diam. Limits Maximum	Rec. Taps	Pitch Diam. Limits Maximum	Rec. Taps
0	—	80	0.0519	0.0542	H2	0.0536	H1	0.0536	H1	0.0532	H1
1	64	—	0.0629	0.0655	H2	0.0648	H1	0.0648	H1	0.0643	H1
1	—	72	0.0640	0.0665	H2	0.0659	H1	0.0658	H1	0.0653	H1
2	56	—	0.0744	0.0772	H2	0.0765	H1	0.0764	H1	0.0759	H1
2	—	64	0.0759	0.0786	H2	0.0779	H1	0.0778	H1	0.0773	H1
3	48	—	0.0855	0.0885	H2	0.0877	H1	0.0877	H1	0.0871	H1
3	—	56	0.0874	0.0902	H2	0.0895	H1	0.0894	H1	0.0889	H1
4	40	—	0.0958	0.0991	H2	0.0982	H2	0.0982	H2	0.0975	H1
4	—	48	0.0985	0.1016	H2	0.1008	H1	0.1007	H1	0.1001	H1
5	40	—	0.1088	0.1121	H2	0.1113	H2	0.1112	H2	0.1105	H1
5	—	44	0.1102	0.1134	H2	0.1126	H1	0.1125	H1	0.1118	H1
6	32	—	0.1177	0.1214	H3	0.1204	H2	0.1204	H2	0.1196	H1
6	—	40	0.1218	0.1252	H2	0.1243	H2	0.1242	H2	0.1235	H1
8	32	—	0.1437	0.1475	H3	0.1465	H2	0.1464	H2	0.1456	H1
8	—	36	0.1460	0.1496	H2	0.1487	H2	0.1485	H2	0.1478	H1
10	24	—	0.1629	0.1672	H3	0.1661	H3	0.1662	H3	0.1653	H1
10	—	32	0.1697	0.1736	H3	0.1726	H2	0.1724	H2	0.1716	H1
12	24	—	0.1889	0.1933	H3	0.1922	H3	0.1922	H3	0.1913	H1
12	—	28	0.1928	0.1970	H3	0.1959	H3	0.1959	H3	0.1950	H1
1/4	20	—	0.2175	0.2224	H5	0.2211	H3	0.2211	H3	0.2201	H2
1/4	—	28	0.2268	0.2311	H4	0.2300	H3	0.2299	H3	0.2290	H1
5/16	18	—	0.2764	0.2817	H5	0.2803	H3	0.2805	H3	0.2794	H2
5/16	—	24	0.2854	0.2902	H4	0.2890	H3	0.2887	H3	0.2878	H1
3/8	16	—	0.3344	0.3401	H5	0.3387	H3	0.3389	H3	0.3376	H2
3/8	—	24	0.3479	0.3528	H4	0.3516	H3	0.3512	H3	0.3503	H1
7/16	14	—	0.3911	0.3972	H5	0.3957	H3	0.3960	H5	0.3947	H3
7/16	—	20	0.4050	0.4104	H5	0.4091	H3	0.4086	H3	0.4076	H1
1/2	13	—	0.4500	0.4565	H5	0.4548	H3	0.4552	H5	0.4537	H3
1/2	—	20	0.4675	0.4731	H5	0.4717	H3	0.4711	H3	0.4701	H1
9/16	12	—	0.5084	0.5152	H5	0.5135	H3	0.5140	H5	0.5124	H3
9/16	—	18	0.5264	0.5323	H5	0.5308	H3	0.5305	H3	0.5294	H2
5/8	11	—	0.5660	0.5732	H5	0.5714	H3	0.5719	H5	0.5702	H3
5/8	—	18	0.5889	0.5949	H5	0.5934	H3	0.5930	H3	0.5919	H2
3/4	10	—	0.6850	0.6927	H5	0.6907	H5	0.6914	H5	0.6895	H3
3/4	—	16	0.7094	0.7159	H5	0.7143	H3	0.7139	H3	0.7126	H2
7/8	9	—	0.8028	0.8110	H6	0.8089	H4	0.8098	H6	0.8077	H4
7/8	—	14	0.8286	0.8356	H6	0.8339	H4	0.8335	H4	0.8322	H2
1	8	—	0.9188	0.9276	H6	0.9254	H4	0.9264	H4	0.9242	H4
1	—	12	0.9459	0.9535	H6	0.9516	H4	0.9515	H4	0.9499	H4
1	—	14	0.9536	0.9609	H6	0.9590	H4	0.9585	H4	0.9572	H4
1-1/8	7	—	1.0322	1.0416	H8	1.0393	H4	1.0407	H4	1.0381	H4
1-1/8	—	12	1.0709	1.0787	H6	1.0768	H4	1.0765	H4	1.0749	H4
1-1/4	7	—	1.1572	1.1668	H8	1.1644	H4	1.1657	H4	1.1631	H4
1-1/4	—	12	1.1959	1.2039	H6	1.2019	H4	1.2015	H4	1.1999	H4
1-3/8	6	—	1.2667	1.2771	H8	1.2745	H4	1.2768	H4	1.2738	H4
1-3/8	—	12	1.3209	1.3291	H6	1.3270	H4	1.3265	H4	1.3249	H4
1-1/2	6	—	1.3917	1.4022	H8	1.3996	H4	1.4018	H4	1.3988	H4
1-1/2	—	12	1.4459	1.4542	H6	1.4522	H4	1.4515	H4	1.4499	H4
1-1/2	—	8	1.4188	1.4283	H7	1.4259	H5	1.4278	H7	1.4251	H5
1-5/8	—	8	1.5438	1.5535	H8	1.5510	H6	1.5531	H7	1.5503	H5
1-3/4	5	—	1.6201	1.6317	H9	1.6288	H7	1.6317	H9	1.6283	H7
1-3/4	—	8	1.6688	1.6786	H8	1.6762	H6	1.6785	H8	1.6756	H5
1-7/8	—	8	1.7938	1.8037	H8	1.8013	H6	1.8038	H8	1.8008	H6
2	4.5	—	1.8557	1.8681	H10	1.8650	H7	1.8684	H10	1.8646	H7
2	—	8	1.9188	1.9289	H8	1.9264	H6	1.9292	H8	1.9261	H6

ISO Metric Class of Threads				
CLASS 6H For Commercial Threads				
Size mm	Pitch mm	Pitch Dia. Limits (Inch) Min.	Max.	Rec. Taps
M1.6	0.35	0.0541	0.0574	D3
M2	0.4	0.0686	0.0720	D3
M2.5	0.45	0.0870	0.0906	D3
M3	0.5	0.1054	0.1092	D3
M3.5	0.6	0.1225	0.1268	D4
M4	0.7	0.1396	0.1442	D4
M5	0.8	0.1764	0.1812	D4
M6	1.0	0.2107	0.2165	D5
M8	1.25	0.2830	0.2892	D5
M10	1.5	0.3554	0.3624	D6
M12	1.75	0.4277	0.4355	D6
M14	2.0	0.5001	0.5083	D7
M16	2.0	0.5788	0.5871	D7
M20	2.5	0.7235	0.7322	D7
M24	3.0	0.8682	0.8785	D8
M30	3.5	1.0917	1.1026	D9
M36	4.0	1.3151	1.3268	D9
M39	4.0	1.4331	1.4450	D9
M42	4.5	1.5385	1.5509	D10
M42	3.0	1.5768	1.5873	D8
M42	2.0	1.6024	1.6112	D7
M42	1.5	1.6152	1.6231	D6
M45	4.5	1.6566	1.6690	D10
M45	3.0	1.6949	1.7054	D8
M48	5.0	1.7619	1.7751	D10
M48	3.0	1.8130	1.8241	D9
M48	2.0	1.8386	1.8479	D7
M48	1.5	1.8514	1.8598	D6
M56	5.5	2.0641	2.0781	D11

D3 = Basic PD + 0.0009″ to Basic PD + 0.0015″
D4 = Basic PD + 0.0012″ to Basic PD + 0.0020″
D5 = Basic PD + 0.0015″ to Basic PD + 0.0025″
D6 = Basic PD + 0.0018″ to Basic PD + 0.0030″
D7 = Basic PD + 0.0019″ to Basic PD + 0.0035″
D8 = Basic PD + 0.0024″ to Basic PD + 0.0040″
D9 = Basic PD + 0.0025″ to Basic PD + 0.0045″

Sizes through 1″ Diameter
H1 = Basic PD to Basic PD + 0.0005″
H2 = Basic PD + 0.0005″ to Basic PD + 0.0010″
H3 = Basic PD + 0.0010″ to Basic PD + 0.0015″
H4 = Basic PD + 0.0015″ to Basic PD + 0.0020″
H5 = Basic PD + 0.0020″ to Basic PD + 0.0025
H6 = Basic PD + 0.0025″ to Basic PD + 0.0030″
Sizes larger than 1″ diameter through 1-1/2″ diameter
H4 = Basic PD + 0.0010″ to Basic PD + 0.0020″

TEXAS
(National Headquarters)
1945 West Walnut Hill Ln.
Irving, TX 75038, USA
Toll Free: 800-837-2223
Fax: 800-837-3334

ILLINOIS
676 East Fullerton Avenue
Glendale Heights, IL 60139, USA
Toll Free: 800-837-2223
Fax: 800-837-3334

CALIFORNIA
1921 Miraloma Ave. Suite B
Placentia, CA 92870, USA
Toll Free: 800-837-2223
Fax: 714-528-9209

OHIO
3611 Socialville Foster Rd.
Ste 102
Mason, OH 45040, USA
Phone: 513-755-3360
Fax: 513-755-3362

GEORGIA
5324 Highway 85 Ste 100
Forest Park, GA 30297, USA
Toll Free: 800-837-2223
Fax: 800-837-3334

CANADA
538 King Forest Court
Burlington, ON L7P 5C1, Canada
Toll Free: 800-263-4861
Fax: 905-632-8466

MEXICO
Avenida Central No. 186
Col. Nueva Industrial Vallejo
07700 Ciudad de Mexico, D.F.,
Mexico
Phone: (52) 55-51-19-3363
Fax: (52) 55-51-19-3370

Drill problems & potential fixes

Chipping on drill point
- reduce feed rate
- reduce or eliminate amount of pecks

Chipping on drill O.D.
- reduce feed rate
- check drill's concentricity
- make sure coolant supply is constant

Heavy wear on drill corners
- increase feed rate
- check drill's concentricity
- make sure coolant supply is constant

Long, stringy chips
- increase feed rate
- increase or include amount of pecks

Drilled hole under-sized
- increase feed rate
- verify drill's sharpness and diameter

Drilled hole over-sized
- reduce feed rate
- increase RPM

Material welding
- increase RPM

Material packing
- reduce feed rate
- increase RPM
- add a peck cycle

Endmill problems & potential fixes

Tool chipping

- check tool's concentricity
- reduce ramp angle
- reduce feed rate during approach

Tool breaking

- reduce feed rate
- reduce axial depth of cut
- try to reduce amount of tool stickout

Tool chattering

- increase radial depth of cut
- reduce axial depth of cut
- leave more material for finish pass
- check tool's concentricity

Tool leaving behind a bigger burr

- reduce feed rate
- increase RPM
- check tool's sharpness

Walls aren't straight

- reduce feed rate
- reduce axial depth of cut
- try to reduce amount of tool stickout

Reamer problems & potential fixes

Reamed hole under-sized
- reduce feed rate
- leave more material for reamer to cut
- check reamer's diameter

Reamed hole over-sized
- increase feed rate
- check reamer's concentricity
- check reamer's diameter

Reamed hole isn't straight
- check reamer's concentricity
- leave more material for reamer to cut
- make sure drilled hole is straight before reaming

K·Tool c·bore drill mills

Recommended SFM

- 1000-2500SFM · aluminum, brass, bronze
- 800-1200SFM · mild/low carbon steel
- 500-800SFM · high carbon/alloy steel
- 400-700SFM · low carbon tool steel
- 300-600SFM · tool steel
- 300-600SFM · stainless steel
- 600-1000SFM · iron
- 120-180SFM · nickel/titanium alloy

Recommended IPT

- .003"-.008"IPT · aluminum, brass, bronze
- .001"-.005"IPT · mild/low carbon steel
- .001"-.005"IPT · high carbon/alloy steel
- .001"-.005"IPT · low carbon tool steel
- .001"-.004"IPT · tool steel
- .001"-.004"IPT · stainless steel
- .002"-.006"IPT · iron
- .001"-.003"IPT · nickel/titanium alloy

Sumitomo MDW HGS solid carbide drills

SFM

steels	P0	300-650
	P1	225-625
	P2	175-525
	P3	225-575
	P4	175-525
	P5	190-525
	P6	125-350
stainless	M1	120-325
	M2	120-350
irons	K1	150-530
	K2	150-500
exotics	S1	65-160
	S2	40-120
non·ferrous	N1	500-1000
	N2	400-800
45·60Rc steel	H1	60-225

IPR

.002" - .01" (0 - .196")
.003" - .014" (.197" - .394")
.003" - .014" (.395" - .630")

Sumitomo MDW GS solid carbide drills

SFM

steels	P0	110-360
	P1	110-360
	P2	100-325
	P3	110-360
	P4	100-325
	P5	110-360
	P6	80-250
stainless	M1	50-210
	M2	60-225
irons	K1	115-415
	K2	115-375
exotics	S1	40-110
	S2	30-80
non-ferrous	N1	300-800
	N2	300-800
45-60Rc steel	H1	40-100

IPR

.002"-.01" (0-.196")

.003"-.014" (.197"-.394")

.003"-.014" (.395"-.630")

Kennametal universal solid carbide drills

SFM

steels	P0	260-520
	P1	230-460
	P2	300-460
	P3	200-330
	P4	160-330
	P5	160-260
	P6	130-230
stainless	M1	100-160
	M2	130-200
	M3	100-160
irons	K1	260-560
	K2	260-460
	K3	260-430
non-ferrous	N1	300-1030
	N2	300-890
	N3	300-890
	N4	300-590
exotics	S1	30-100
	S2	30-80
	S3	30-100
	S4	30-130

IPR

.001"-.007" (5/64"/.078")
.001"-.007" (1/8"/.125")
.001"-.009" (3/16"/.188")
.002"-.012" (1/4"/.250")
.002"-.015" (5/16"/.313")
.003"-.017" (3/8"/.375")
.003"-.019" (1/2"/.500")
.004"-.024" (5/8"/.625")
.004"-.029" (3/4"/.750")

Crownloc SD101-105

Recommended SFM

- 490-1150 SFM · aluminum
- 130-425 SFM · steel
- 100-260 SFM · hardened steel
- 165-295 SFM · stainless steel
- 230-525 SFM · iron

Recommended IPR

- .002"-.013" IPR (.374" - .472")
- .003"-.015" IPR (.472" - .551")
- .003"-.019" IPR (.551" - .630")
- .003"-.023" IPR (.630" - .708")
- .004"-.023" IPR (.708"-1")

Garr

General purpose carbide endmills

- 250-500SFM .001"-.015" IPT · aluminum
- 100-250SFM .0005"-.006" IPT · 10/11/13 steels
- 50-225SFM .0002"-.005" IPT · 4140/4340

High performance carbide endmills

- 300-600SFM .0015"-.01" IPT · aluminum
- 150-500SFM .0008"-.005" IPT · 10/11/13 steels
- 125-350SFM .0005"-.004" IPT · 4140/4340

General purpose drills

- 150-400SFM .002"-.014" IPR · aluminum
- 100-200SFM .001"-.01" IPR · 10/11/13 steels
- 80-175SFM .001"-.008" IPR · 4140/4340

High performance drills

- 250-500SFM .005"-.019" IPR · aluminum
- 175-300SFM .001"-.013" IPR · 10/11/13 steels
- 175-300SFM .001"-.011" IPR · 4140/4340

General purpose reamers

- 150-225SFM .0005"-.0018" IPT · aluminum
- 80-125SFM .0004"-.0017" IPT · 10/11/13 steels
- 80-125SFM .0003"-.0015" IPT · 4140/4340

Komet

KUB Pentron drills

- 490-1640 SFM · .003"-.009" IPR · aluminum
- 520-1210 SFM · .002"-.006" IPR · 10/11/13 steels

KUB Quatron drills

- 820-1970 SFM · .003"-.012" IPR · aluminum
- 820-980 SFM · .004"-.01" IPR · 10/11/13 steels

KUB Trigon drills

- 820-1970 SFM · .001"-.01" IPR · aluminum
- 820-980 SFM · .002"-.006" IPR · 10/11/13 steels

MicroKom fine boring bars

- 980-1640 SFM · .001"-.006" IPR · aluminum
- 820-980 SFM · .002"-.005" IPR · 10/11/13 steels

M03Speed boring heads

- 820-1310 SFM · .003"-.006" IPR · aluminum

B002 boring bars

- 980-1640 SFM · .002"-.006" IPR · aluminum
- 820-980 SFM · .002"-.005" IPR · 10/11/13 steels

DiHart reamers

- 360-1800 SFM · .002"-.012" IPT · aluminum