Tools Used in CNC Milling and Lathe Machines

The tools used in CNC machines are made of cemented carbide, High Speed Steel, Tungsten Alloys, Ceramics, and many other hard materials. The shapes and sizes of tools used in Milling machines and Lathe machines are different from each other. These tools are discussed next.

Milling Tools

There are various type of milling tools for different applications. These tools are discussed next.

END MILL

End mills are used for producing precision shapes and holes on a Milling or Turning machine. The correct selection and use of end milling cutters is paramount with either machining centers or lathes. End mills are available in a variety of design styles and materials; refer to Figure-32.

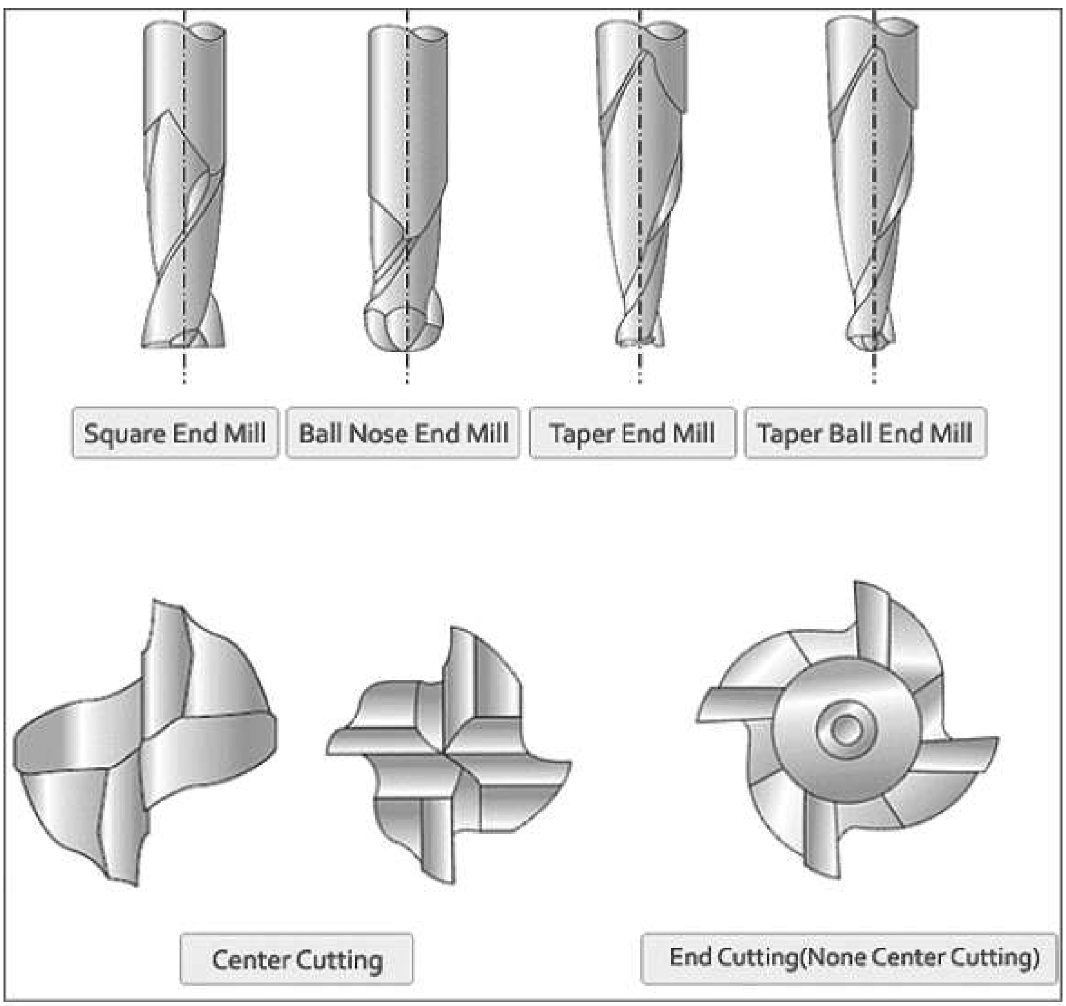


Figure-32. End Mill tool types

Titanium coated end mills are available for extended tool life requirements. The successful application of end milling depends on how well the tool is held (supported) by the tool holder. To achieve best results, an end mill must be mounted concentric in a tool holder. The end mill can be selected for the following basic processes:

FACE MILLING - For small face areas, of relatively shallow depth of cut. The surface finish produced can be 'scratchy".

KEYWAY PRODUCTION - Normally two separate end mills are required to produce a quality keyway.

WOODRUFF KEYWAYS - Normally produced with a single cutter, in a straight plunge operation.

SPECIALTY CUTTING - Includes milling of tapered surfaces, "T" shaped slots & dovetail production.

FINISH PROFILING - To finish the inside/outside shape on a part with a parallel side wall.

CAVITY DIE WORK - Generally involves plunging and finish cutting of pockets in die steel. Cavity work requires the production of three dimensional shapes. A Ball type end mill is used for the finishing cutter with this application.

Roughing End Mills, also known as ripping cutters or hoggers, are designed to remove large amounts of metal quickly and more efficiently than standard end mills; refer to Figure-33. Coarse tooth roughing end mills remove large chips for heavy cuts, deep slotting and rapid stock removal on low to medium carbon steel and alloy steel prior to a finishing application. Fine tooth roughing end mills remove less material but the pressure is distributed over many more teeth, for longer tool life and a smoother finish on high temperature alloys and stainless steel.

Figure-33. Roughing End Mill

BULL NOSE MILL

Bull nose mill look alike end mill but they have radius at the corners. Using this tool, you can cut round corners in the die or mold steels. Shape of bull nose mill tool is given in Figure-34.

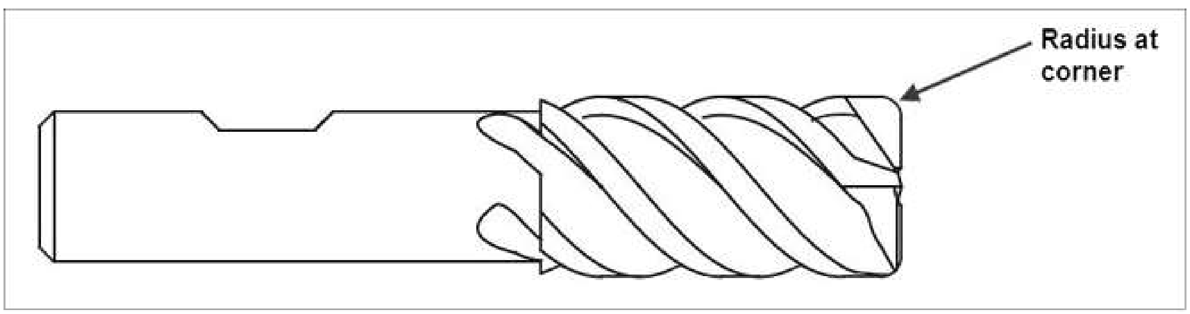


Figure-34. Bull Nose Mill cutter

BALL NOSE MILL

Ball nose cutters or ball end mills has the end shape hemispherical; refer to Figure-32. They are ideal for machining 3-dimensional contoured shapes in machining centres, for example in moulds and dies. They are sometimes called ball mills in shop-floor slang. They are also used to add a radius between perpendicular faces to reduce stress concentrations.

FACE MILL

The Face mill tool or face mill cutter is used to remove material from the face of workpiece and make it plane; refer to Figure-35.

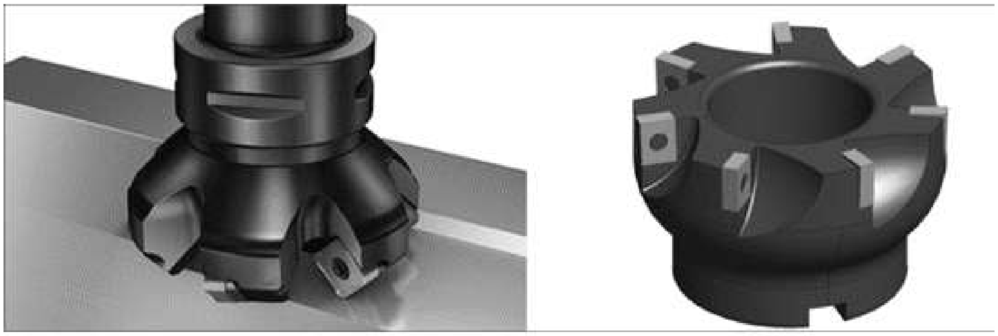


Figure-35. Face milling tool

RADIUS MILL AND CHAMFER MILL

The Radius mill tool is used to apply round (fillet) at the edges of the part. The Chamfer mill tool is used to apply chamfer at the edges of the part. Figure-36 shows the radius mill tool and chamfer mill tool.

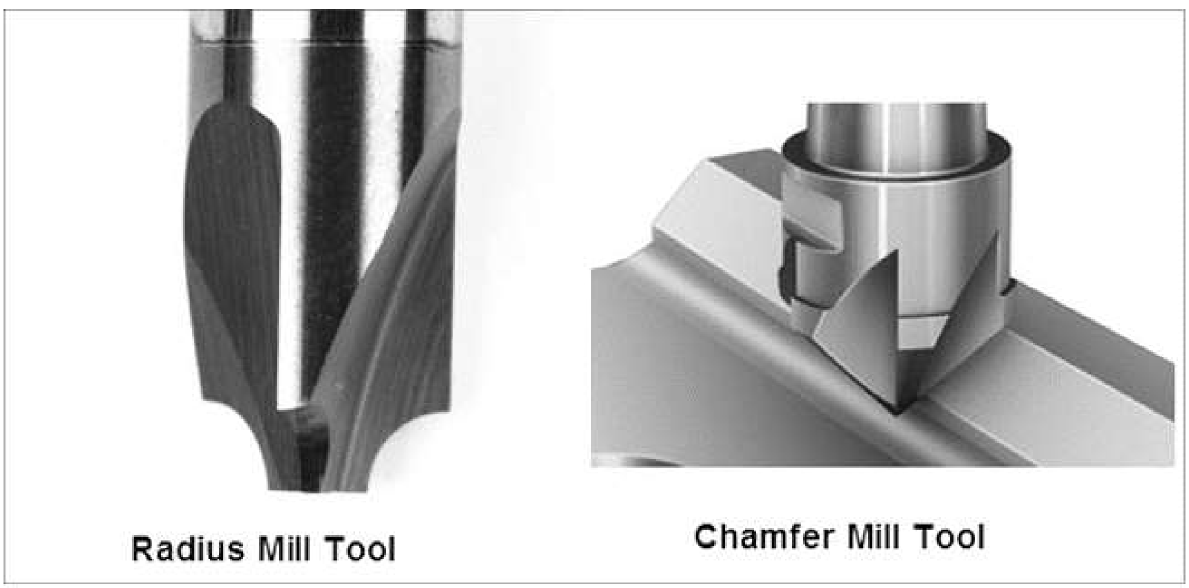


Figure-36. Radius mill and Chamfer mill tool

SLOT MILL

The Slot mill tool is used to create slot or groove in the part metal. Figure-37 shows the shape of slot mill tool.

Figure-37. Slot mill tool

TAPER MILL

In CNC machining, taper end mills are used in many industries for a large number of applications, such as walls with draft or clearance angle, tool and die work, mold work, even for reaming holes to make them conical. There are mainly two types of taper mills, Taper End Mill and Taper Ball Mill; refer to Figure-32.

DOVE MILL

Dove mill or Dovetail cutters are designed for cutting dovetails in a wide variety of materials. Dovetail cutters can also be used for chamfering or milling angles on the bottom surface of a part. Dovetail cutters are available in a wide variety of diameters and in 45 degree or 60 degree angles; refer to Figure-38.

Figure-38. Dovetail milling cutters

LOLLIPOP MILL

The Lollipop mill tool is used to cut round slot or undercuts in workpiece. Some tool suppliers use a name Undercut mill tool in place of Lollipop mill in their catalog. The shape of lollipop mill tool is given in Figure-39.

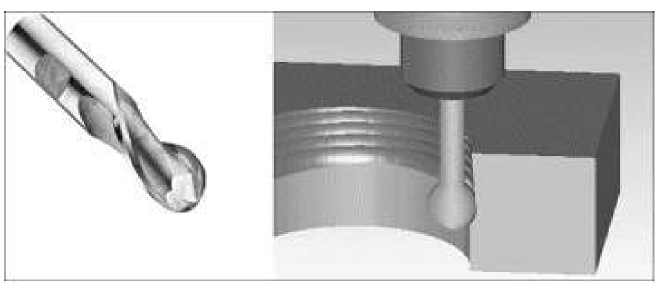


Figure-39. Lollipop mill tool

ENGRAVE MILL

The Engrave mill tool is used to perform engraving on the surface of workpiece. Engraving has always been an art and it is also true for CNC machinist. You can find various shapes of engraving tool that are single flute or multi-flute; refer to Figure-40. You can use ball mill/end mill for engraving or you can use specialized engrave mill tool for engraving. This all depends on your requirement. If you want

to perform engraving on softer materials or plastics then it is better to use ball end mill but if you want an artistic shade on the surface then use the respective engrave mill tool. Keep a note of maximum depth and spindle speed mentioned by your engrave mill tool supplier.

Figure-40. Engrave mill tools

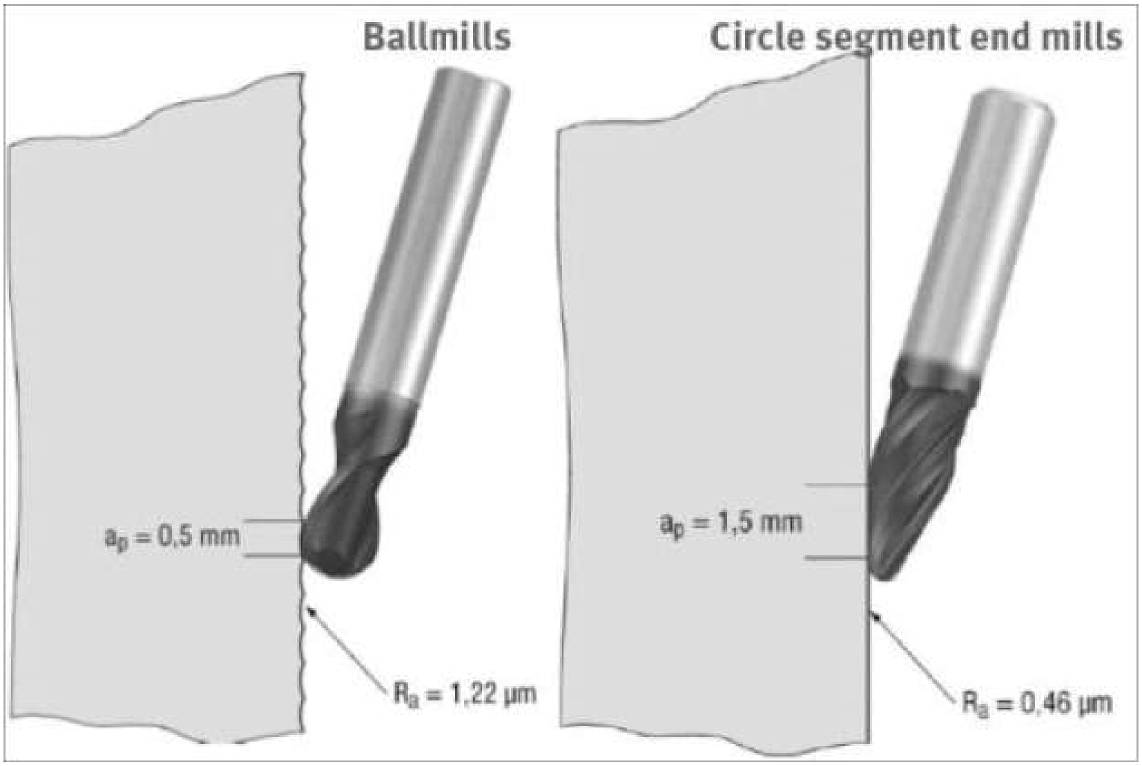
THREAD MILL

The Thread mill tool is used to generate internal or external threads in the workpiece. The most common question here is if we have Taps to create thread then why is there need of Thread mill tool. The answer is less machining time on CNC, tool cost saving, more parts per tool, and better thread finish. Now, you will ask why to use tapping. The answer is low machine cost. Figure-41 shows thread mill tools.

Figure-41. Thread Mill

BARREL MILL

Barrel Mill tool is the tool recently being highly used in machining turbine/impeller blades and other 5-axis milling operations. Barrel Mill has conical shape with radius at its end; refer to Figure-42. Note that earlier Ball mill tools were used for irregular surface contouring but Barrel Mill tools give much better surface finish so they are highly in demand for 5-axis milling now a days.



Figure–42. Barrel Mill versus Ball Mill Tool

DRILL BIT

Drill bit is used to make a hole in the workpiece. The hole shape depends on the shape of drill bit. Drill bits for various purposes are shown in Figure-43. Note that drill is the machine or holder in which drill bit is installed to make cylindrical holes. There are mainly four categories of drill bit; Twist drill bit, Step drill bit, Unibit (or conical bit), and Hole Saw bit (Refer to Figure-44). Twist drill bits are used for drilling holes in wood, metal, plastic, and other materials. For soft materials the point angle is 90 degree, for hard materials the point angle is 150 degree, and general purpose twist drill bits have angle of 150 degree at end point. The Step drill bits are used to make counter bore or countersunk holes. The Unibits are generally used for drilling holes in sheetmetal but they can also be used for drilling plastic, plywood, aluminium, and thin steel sheets. One unibit can give holes of different sizes. The Hole saw bit is used to cut a large hole from the workpiece. They remove material only from the edge of the hole, cutting out an intact disc of material, unlike many drills which remove all material in the interior of the hole. They can be used to make large holes in wood, sheet metal, and other materials.

Figure-43. Drill Bits for different purposes

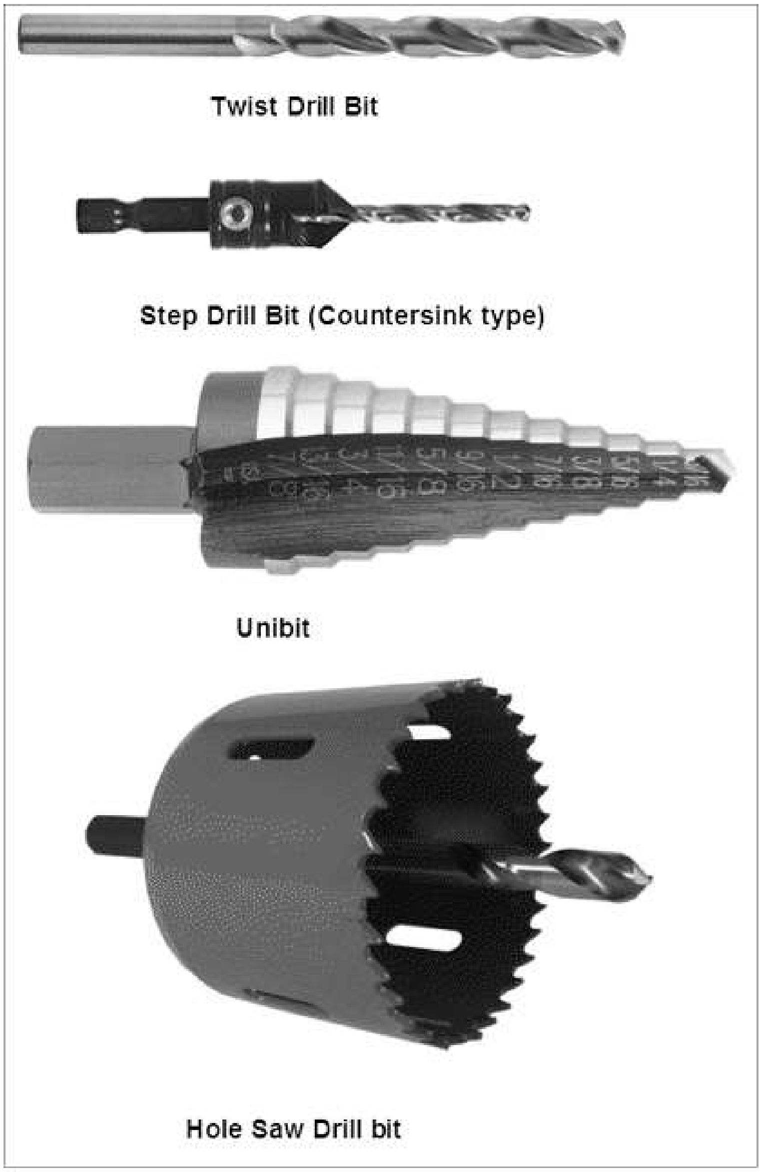


Figure-44. Types of drill bits

REAMER

Reamer is a tool similar to drill bit but its purpose is to finish the hole or increase the size of hole precisely. Figure-45 shows the shape of a reamer.

Figure–45. Reamer tool

BORE BAR

Bore Bar or Boring Bar is used to increase the size of hole; refer to Figure-46. One common question is why to use bore bar if we can perform reaming or why to perform reaming when we have bore bar. The answer is accuracy. A reamer does not give tight tolerance in location but gives good finish in hole diameter. A bore bar gives tight tolerance in location but takes more time to machine hole as compared to reamer. The decision to choose the process is on machinist. If you need a highly accurate hole then perform drilling, then boring and then reaming to get best result.

Figure-46. Boring Bar

Lathe Tools or Turning Tools

The tools used in CNC lathe machines use a different nomenclature. In CNC lathe machines, we use insert for cutting material. The Insert Holder and Inserts have a special nomenclature scheme to define their shapes. First we will discuss the nomenclature of Insert holder and then we will discuss the nomenclature of Inserts.

INSERT HOLDERS

Turning holder names follow an ISO nomenclature standard. If you are working on a CNC shop floor with lathes, knowing the ISO nomenclature is a must. The name looks complicated, but is actually very easy to interpret; refer to Figure-47.

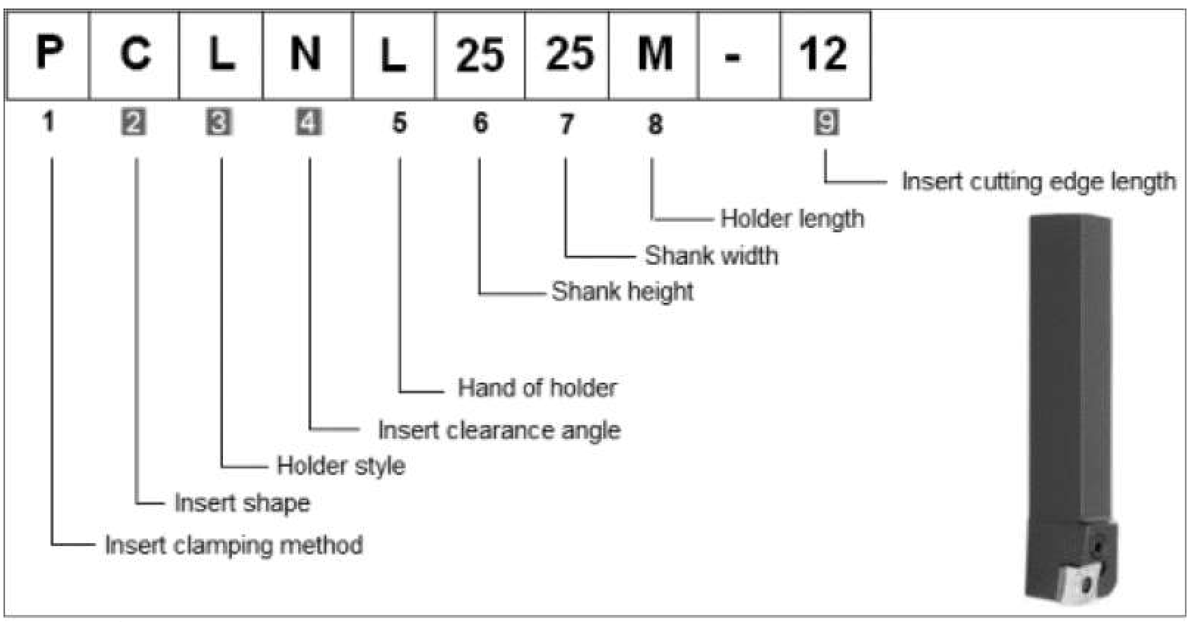


Figure-47. CNC Lathe Insert Holder nomenclature

When selecting a holder for an application, you mainly have to concentrate on the numbers marked in red in the above nomenclature. The others are decided automatically (e.g., the shank width and height are decided by the machine), or require less effort. In Figure-48, the rows with the question mark indicate the parameters that require the decision by machinist based on job.

	Parameter		How is this decided ?
1	Insert clamping method		Select based on cutting forces. Top clamping is the most sturdy, screw clamping the least.
2	Insert shape	?	Decided by the contour that you want to turn.
3	Holder style	?	Decided by the contour that you want to turn.
4	Insert clearance angle	?	Positive / Negative, based on application.
5	Hand of holder		Decided based on whether you want to cut towards the chuck or away from the chuck, and on turret position - turret front / rear
6	Shank height		Decided by holder size.
7	Shank width		Decided by machine.
8	Holder length		Decided by machine.
9	Insert cutting edge length	?	Decide based on depth of cut you want to use.

Figure–48. CNC Lathe Insert Holder nomenclature parameters

Figure-49 and Figure-50 show the options available for each of the parameters.

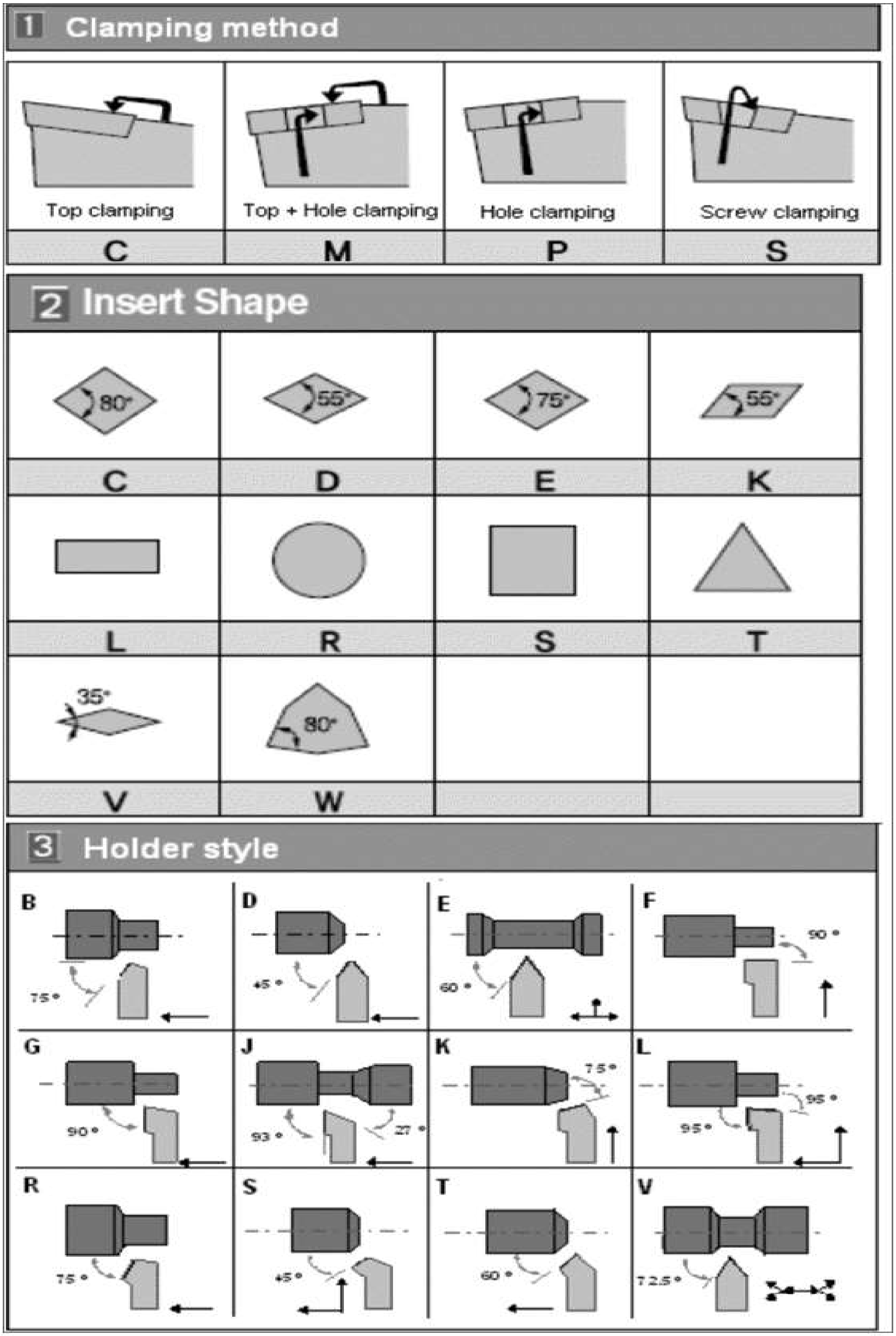
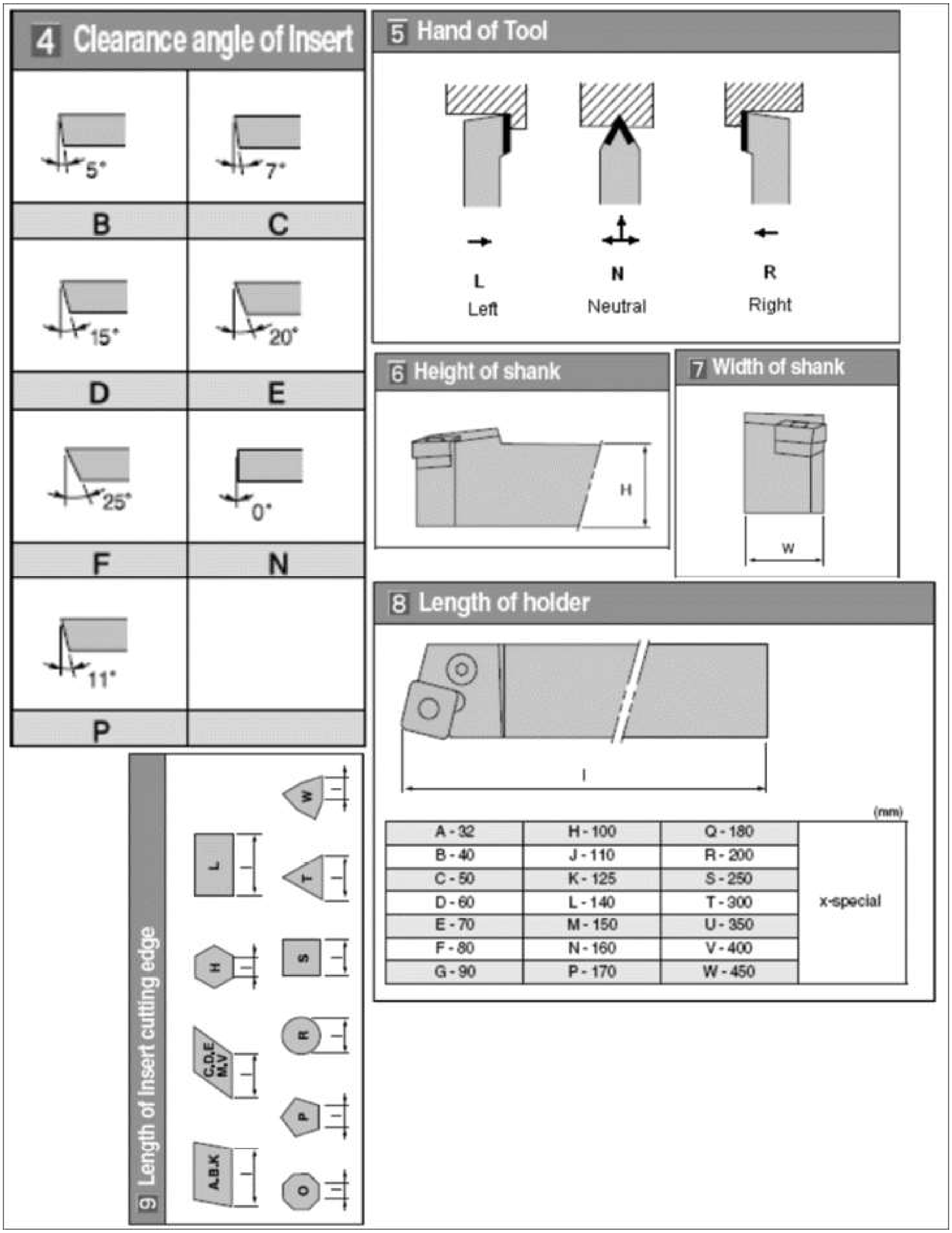


Figure-49. Clamping Method, Insert Shapes, and Holder Style



A - 32	H - 100	Q - 180	x-special
B - 40	J - 110	R - 200	
C - 50	K - 125	S - 250	
D - 60	L - 140	T - 300	
E - 70	M - 150	U - 350	
F - 80	N - 160	V - 400	
G - 90	P - 170	W - 450	

Figure-50. Insert Holder Parameters

General CNC Insert name is given as

C	N	M	G	12	04	08
1	2	3	4	5	6	7

Meaning of each box in nomenclature is given next.

1 = Turning Insert Shape

The first letter in general turning insert nomenclature tells us about the general turning insert shape, turning inserts shape codes are like C, D, K, R, S, T, V, W. Most of these codes surely express the turning insert shape like

C = C Shape Turning Insert

D = D Shape Turning Insert

K = K Shape Turning Insert

R = Round Turning Insert

S = Square Turning Insert

T = Triangle Turning Insert

V = V Shape Turning Insert

W = W Shape Turning Insert

Figure-51 shows the turning inserts shapes.

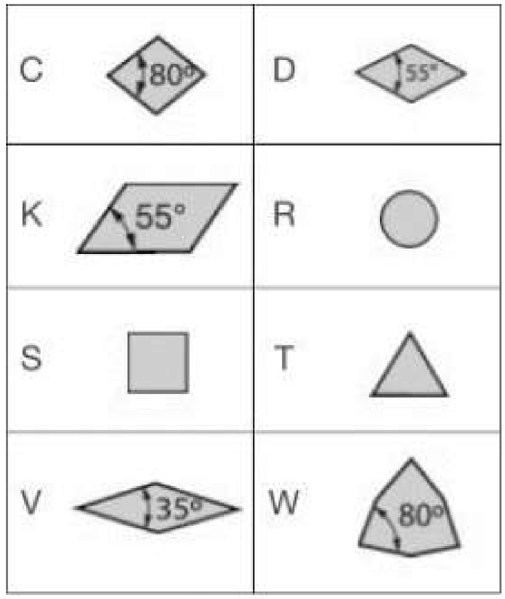


Figure-51. Turning Insert Shapes

The general turning insert shape play a very important role when we choose an insert for machining. Not every turning insert with one shape can be replaced with the other for a machining operation. As C, D, W type turning inserts are normally used for roughing or rough machining.

2 = Turning Insert Clearance Angle

The second letter in general turning insert nomenclature tells us about the turning insert clearance angle.

The clearance angle for a turning insert is shown in Figure-52.

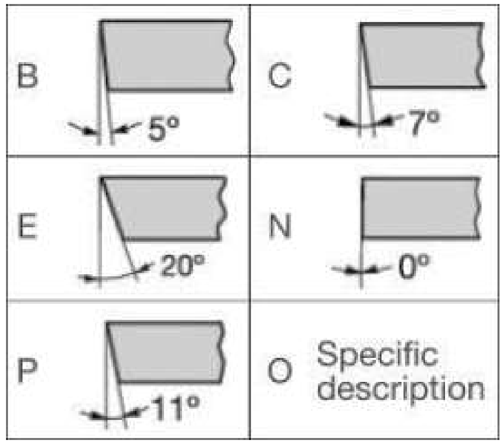


Figure-52. Turning insert clearance angle

Turning insert clearance angle plays a big role while choosing an insert for internal machining or boring small components, because if not properly chosen the insert bottom corner might rub with the component which will give poor machining. On the other hand, a turning insert with 0° clearance angle is mostly used for rough machining.

3 = Turning Insert Tolerances

The third letter of general turning insert nomenclature tells us about the turning insert tolerances. Figure-53 shows the tolerance chart.

Code Letter	Cornerpoint (inches)	Thickness (inches)	Inscribed Circle (in)	Cornerpoint (mm)	Thickness (mm)	Inscribed Circle (mm)
A	.0002"	.001"	.001"	.005mm	.025mm	.025mm
C	.0005"	.001"	.001"	.013mm	.025mm	.025mm
E	.001"	.001"	.001"	.025mm	.025mm	.025mm
F	.0002"	.001"	.0005"	.005mm	.025mm	.013mm
G	.001"	.005"	.001"	.025mm	.13mm	.025mm
H	.0005"	.001"	.0005"	.013mm	.025mm	.013mm
J	.002"	.001"	.002-.005"	.005mm	.025mm	.05-.13mm
K	.0005"	.001"	.002-.005"	.013mm	.025mm	.05-.13mm
L	.001"	.001"	.002-.005"	.025mm	.025mm	.05-.13mm
M	.002-.005"	.005"	.002-.005"	.05-.13mm	.13mm	.05-.15mm
U	.005-.012"	.005"	.005-.010"	.06-.25mm	.13mm	.08-.25mm

Figure-53. Insert tolerance chart

4 = Turning Insert Type

The fourth letter of general turning insert nomenclature tells us about the turning insert hole shape and chip breaker type; refer to Figure-54.

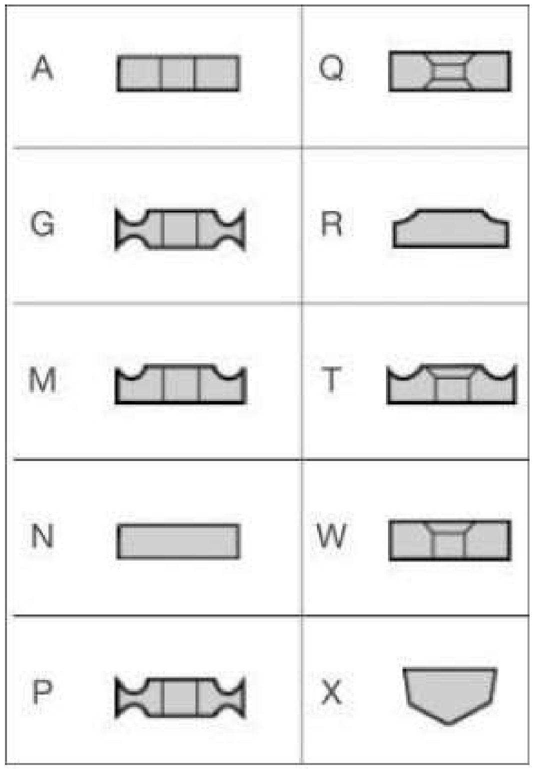


Figure-54. Turning Insert hole shape and chip breaker

5 = Turning Insert Size

This numeric value of general turning insert tells us the cutting edge length of the turning insert; refer to Figure-55.

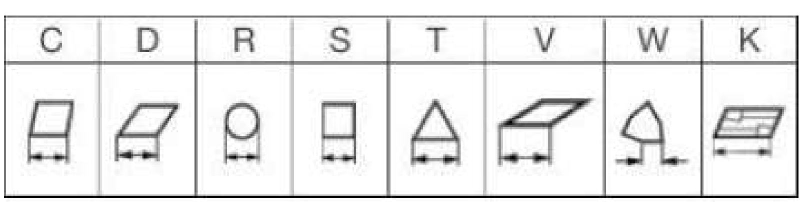


Figure-55. Turning Insert Cutting Edge Length

6 = Turning Insert Thickness

This numeric value of general turning insert tells us about the thickness of the turning insert.

7 = Turning Insert Nose Radius

This numeric value of general turning insert tells us about the nose radius of the turning insert.

Code = Radius Value

04 = 0.4

08 = 0.8

12 = 1.2

16 = 1.6

You can know more about tooling from your tool supplier manual.