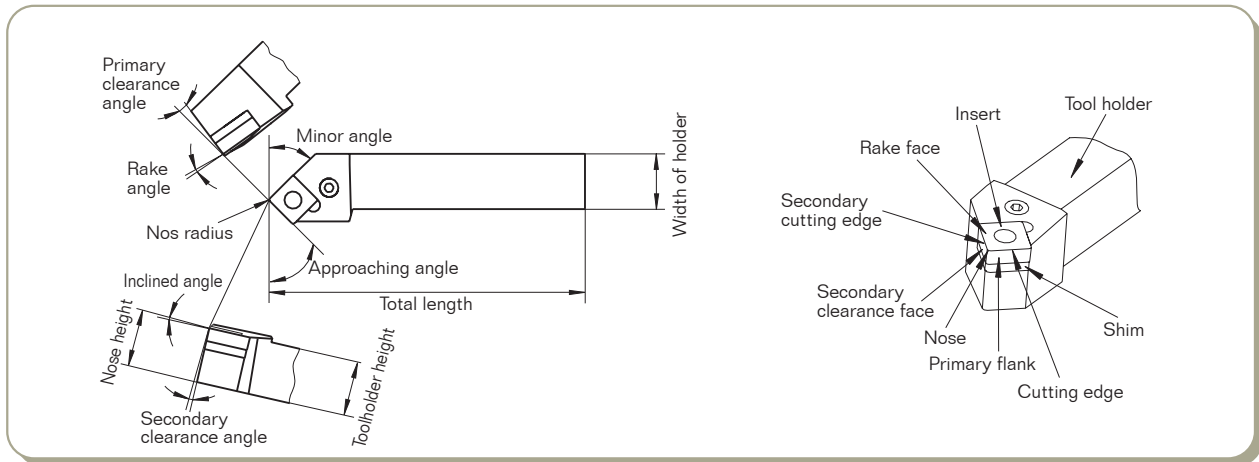


The functions of each part of turning tools

**1 The names of each part of turning tools**



**2 Effects of rake angle**

Larger rake angle makes cutting edge sharper, reduces resistant forces of chip flow, diminishes friction and prevent deformation, leading to smaller cutting forces and cutting power, lower cutting temperature, less abrasion and higher surface quality. However, too large rake angle would reduce the rigidity and strength of tool. Heat can't be diffused easily. Serious breakage and abrasion on tool would occur, reducing tool life. Please choose rake angle according to machining conditions.

Value selection	Situations
Small rake angle	<ul style="list-style-type: none"> <li>●When machining brittle and hard materials</li> <li>●When roughing and intermittent cutting</li> </ul>
Big rake angle	<ul style="list-style-type: none"> <li>●When machining plastic or soft materials</li> <li>●When finishing</li> </ul>

**3 Effects of clearance angle**

The main function of clearance angle is to reduce the friction between the clearance face of tool and the surface of workpiece. When the rake angle is fixed, larger clearance angle can increase the sharpness of cutting edge, reduce cutting forces and friction, and then achieve higher surface quality. However, if clearance angle is too large, the strength of cutting edge would decrease. Also, heat can't be diffused easily and serious abrasion would occur, reducing tool life.

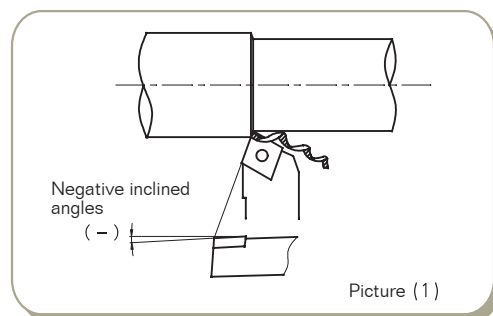
The principle of choosing clearance angle: Choose small clearance angle if friction is not serious.

Value Selection	Situations
Small clearance angle	<ul style="list-style-type: none"> <li>●In order to increase nose strength when roughing</li> <li>●When machining brittle and hard materials</li> </ul>
Large clearance angle	<ul style="list-style-type: none"> <li>●In order to reduce friction when finishing</li> <li>●When machining materials easy to be hardened</li> </ul>

**4 Effects of inclined angle**

Positive or negative inclined angle determines the direction of chip flow, and also affects the strength and impact resistance of insert nose.

◆As diagram (1) shows, when the inclined angle is negative, namely nose is in the lowest point as apposed to the bottom of tool, chips flow to the machined surface of workpiece.



General turning

Parting and grooving

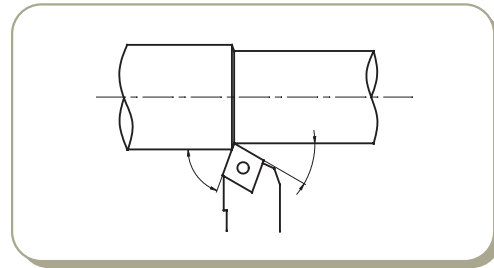
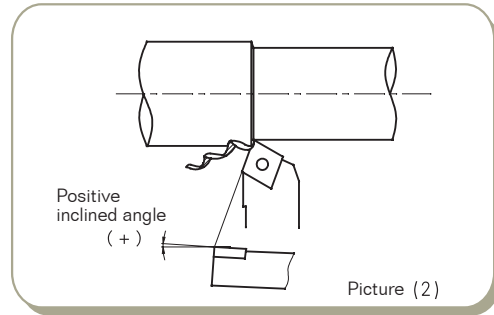
Threading

General technical information for turning

### General technical information for turning

◆ As diagram (2) shows, when inclined angle is positive, namely the nose is in the highest point as apposed to the bottom of the tool, chips flow to the areas of workpiece surface that haven't been machined.

◆ The change of inclined angle also affects insert nose strength and impact resistance. When the inclined angle is negative, the nose is in the lowest point of cutting edge. When the cutting edge enters the workpiece, the contacting point is on the cutting edge or rake face, protecting the nose from impact and increase the strength of the nose. Normally, negative inclined angle should be chosen for tools with big rake angle. This can not only increase nose strength, but also prevent the impact of entry.



## 5 Effects of approach angle

Reduced approaching angle increases the strength of tools and enable heat to diffuse easily, improving surface quality. This is because when the approach angle is small, cutting edge width is large, and then the unit width of cutting edge bears less cutting force. Meanwhile, tool life can be improved.

Normally, select 90° approach angle for turning of slender and step shaft; select 45° approach angle for external turning, end surface machining and chamfering. When approach angle is larger, radial force is reduced, cutting is stable, cutting thickness is increased, and chip breaking is excellent.

Value selection	Situations
Small approach angle	For those materials with high intensity, high hardness and hardened layer on the surface
Big approach angle	When rigidity of the machine is not enough

## 6 Effects of minor angle

Minor angle is the main angle that can affect surface quality, and it can also affect tool strength. If the approach angle is too small, the friction between the secondary flank and machined surface of workpiece will increase, causing vibration.

The principle of selecting minor angle: Select small minor angle when roughing or when the friction is unaffected and there is no vibration. Select large minor angle when finishing.

## 7 Nose radius

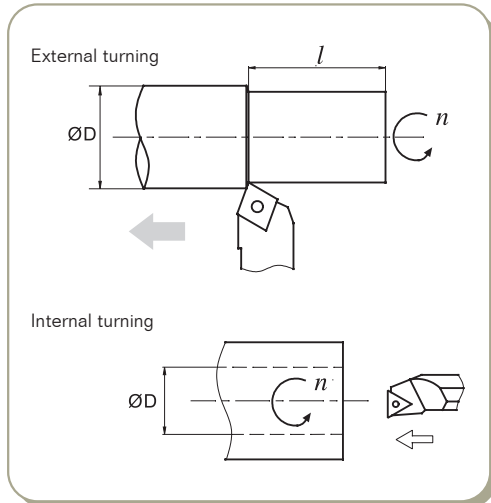
Nose radius significantly affects nose strength and surface quality.

Large nose radius means higher cutting edge strength, and the abrasion on the rake face and clearance face can be reduced to some extent. However, if the nose radius is too large, radial force will increase, and vibration is easy to occur, affecting machining precision and surface quality.

Value selection	Situations
Small nose radius	<ul style="list-style-type: none"> <li>● Finishing at small cutting depth</li> <li>● Machining parts such as slender shaft</li> <li>● When the rigidity of the machine is not enough</li> </ul>
Large nose radius	<ul style="list-style-type: none"> <li>● When roughing</li> <li>● When machining hard materials (intermittent cutting)</li> <li>● When the rigidity of the machine is not enough</li> </ul>

Calculation method of turning parameters

**1** Calculation of cutting speed



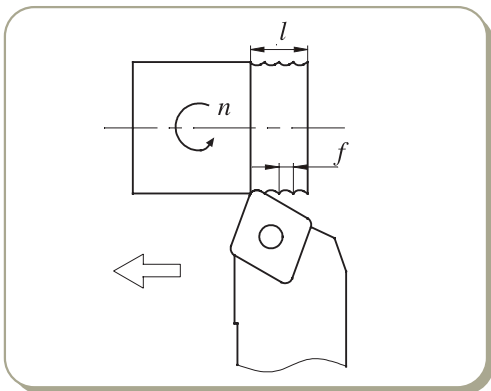
$$V_c = \frac{\pi \times D \times n}{1000} \text{ (m/min)}$$

In the formula:  $V_c$ : Cutting speed (m/min)  
 $n$ : Rotating speed of main axle (rev/min)  
 $D$ : Diameter of workpiece (mm)

For example: When the rotating speed is 280rev/min and the diameter of workpiece is 150mm, the cutting speed should be:

$$V_c = \frac{\pi \times D \times n}{1000} \text{ (m/min)} = 132 \text{ (m/min)}$$

**2** Calculation of feed rate



$$f = \frac{l}{n} \text{ (mm/rev)}$$

In the formula:  $f$ : Feed rate per rotation (mm/rev)  
 $l$ : Cutting length per minute (mm/min)  
 $n$ : Rotating speed of main axle (rev/min)

For example: When the rotating speed of main axle is 500rev/min, and the cutting length per minute is 100mm/min, the feed rate per rotation should be:

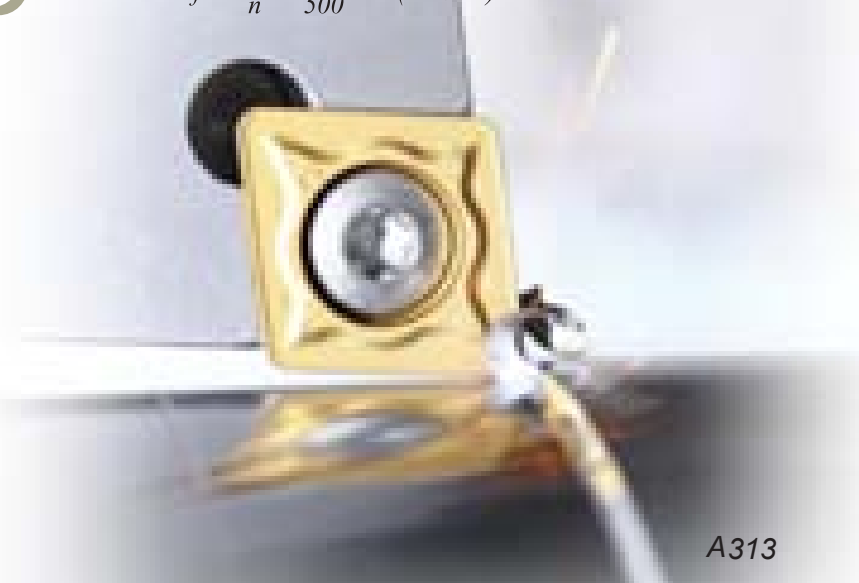
$$f = \frac{l}{n} = \frac{100}{500} = 0.2 \text{ (mm/rev)}$$

General turning

Parting and grooving

Threading

General technical information for turning



### General technical information for turning

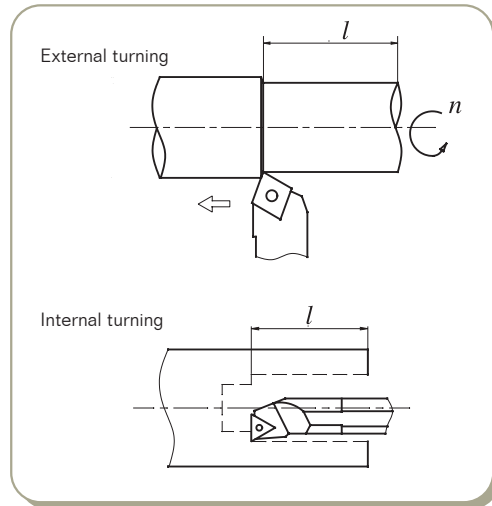
#### 3 Cutting time calculation of external and internal turning

$$T = \frac{l}{f \times n} \text{ (min)}$$

In the formula: T: Cutting time (min)  
 l: Length of machined areas (mm)  
 f: Feed rate (mm/rev)  
 n: Rotating speed of main axle (rev/min)

For example: When the rotating speed of main axle is 250rev/min, and the feed rate is 0.2mm/rev, the time needed for a cutting length of 150mm should be:

$$T = \frac{l}{f \times n} = \frac{150}{0.2 \times 250} = 3 \text{ (min)}$$

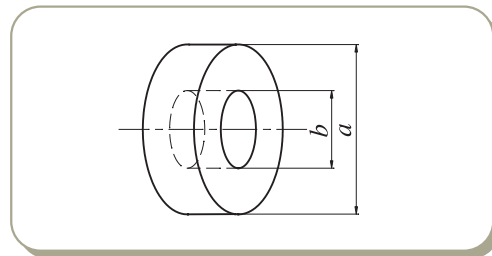


#### 4 Time calculation for end surface turning (constant linear speed)

$$T = \frac{\pi \times (a^2 - b^2)}{4000 \times V_c \times f} \text{ (min)}$$

In the formula: T: Cutting time (min)  
 V<sub>c</sub>: Cutting speed (m/min)  
 f: Feed rate (mm/rev)

For end surface without hole, b=0, the formula is still valid.



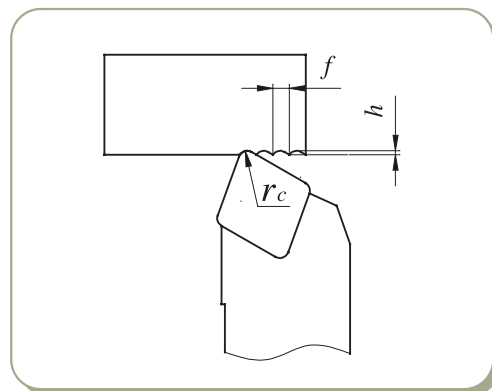
#### 5 Theoretical value calculation of machined surface roughness

$$R = \frac{f^2}{8r_c} \times 1000 \text{ (}\mu\text{m)}$$

In the formula: R: Theoretical roughness value of machined surface  
 f: Feed rate (mm/rev)  
 r<sub>c</sub>: Nose radius (mm)

For example: When the feed rate is 0.2mm/rev, and the nose radius is 0.4mm, the theoretical roughness value of machined surface should be:

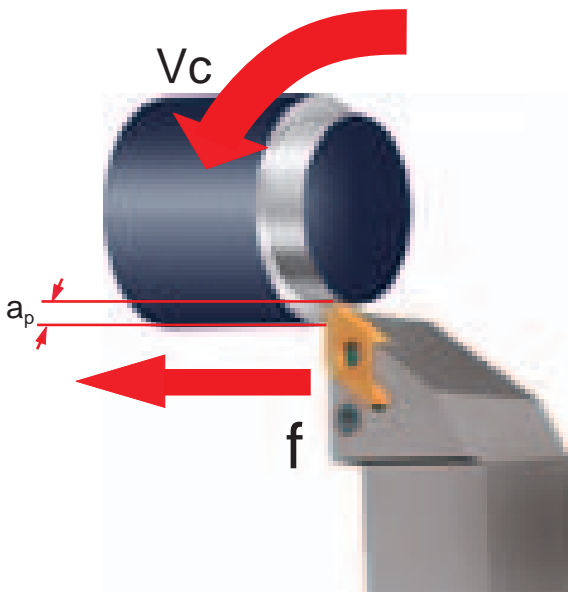
$$R = \frac{f^2}{8r_c} \times 1000 = \frac{0.2^2}{8 \times 0.4} \times 1000 = 12.5 \text{ (}\mu\text{m)}$$



Effect of three main turning parameters on machining

**Effects of three main parameters**

Normally, short machining time, long tool life and high machining precision are expected in machining, so the material quality, hardness, and shape of the workpiece, and properties of machine should be fully considered, and then we can select suitable tools and adopt high-efficiency cutting parameters, namely three parameters.



**Cutting speed ( $V_c$ )**

When the workpiece is rotating on the machine, the number of its rotation per minute is defined as Rotating speed of main axle ( $n$ ). Because of its rotation, the cutting speed measured on the contacting point of diameter is defined as linear speed, m/min. Normally, linear speed is considered to measure the effect of cutting speed on machining.

**Effect of cutting speed**

Cutting speed has significant effect on tool life. When the cutting speed is increased, cutting temperature will increase and tool life will be shortened. Cutting speed varies according to the different types and hardness of workpiece. The below conclusions are reached after many cutting experiments:

(1) Normally tool life would be reduced to half when the cutting speed is increased by 20%. Tool life would be 20% of the original life if the cutting speed is raised by 50%.

(2) Low speed (20-40m/min) cutting could easily cause vibration and shorten tool life.

**Feed rate ( $f_n$ )**

Feed rate is defined as the moving distance of tool after workpiece rotates for one circle, measured by mm/rotation.

**Effect of feed rate**

Feed rate is a key factor that determines surface quality. Meanwhile it also affect the range of chip forming and the thickness of chips during machining.

In term of the effect on tool life, small feed rate leads to serious abrasion on clearance face, greatly reducing tool life.

**Cutting depth ( $a_p$ )**

Cutting depth is defined as the difference between machined surface and unmachined surface, measured by mm. It is half the difference value between the original diameter and machined diameter.

**Effect of cutting depth**

Cutting depth should be determined by the machining allowance and shape of workpiece, power and rigidity of machine, and tool rigidity.

The change of cutting depth has little effect on tool life. If the cutting depth is too low, the cutting nose only scrapes the hardened layer on the workpiece surface, reducing tool life. When there is hardened oxide layer on workpiece surface, higher cutting depth should be adopted within the possible range of machine's power to avoid cutting nose just cutting the hardened layer of workpiece.

## Comparison table for turning inserts chipbreaker

General turning

Parting and grooving

Threading

Comparison table for turning inserts chipbreaker

### Comparison table for turning insert chipbreaker

Negative inserts

ISO	Machining range	ZCC-CT	SANDVIK	KORLOY	TaeguTec	WALTER	SECO	MITSUBISHI	SUMITOMO	KENAMETAL	DIJET	HITACHI	TUNGALOY	KYOCERA	VALANTTE	
P	For extra finishing		QF	HU	FA, EA	NF3	FF1	PK※ FH, FY	FA, FL	UF, FF	F1, FA	FE	01※, TF, ZF	DP※, GP, VF XP, XP-T	F1	
	For finishing	DF	PF MF	HF	FG, SF	NS6	MF2	C, SA, SH	SU, LU, SX	LF, FN	PF, UR UA, UT	BE, CE	NS, 27 TS, AS	HQ, CQ	F2(2B), F5(5C)	
	For finishing (Soft steel)	SF		HF				SY					17	XQ, XS		
	For finishing (Wiper)	WGF	WP WF	HW		NF	W-MF2	SW	LUW	FW			AFW, ASW	WP, WQ		
	For semi-finishing	DM PM	PM QM SM	HA HC HM	MC MT MP	NM4 NM6	MF3 MR3 MF5	MV MA MH	GU UG UX	MG MN		PG UB	AB AY AE	NM ZM TM DM 37	CJ, GS PS, HS PT, CS	F3, F4(8A), M2(2C), M3 M4, M5(5B), M6, M7, 55, M8
	For semi-finishing (Wiper)	WGM	WM		WS, WT	NM	W-M3	MW	GUW	MW						
		DR (Double-side)	PR		MT, MG	NM9	MR7	GH	MU, MX	RN	RN	UD, GG	AR, RE	TH	GT, HT	
For light roughing	LR, DR (Single-side) HDR, HPR	QR PR HR	HR HI	HT, RH	NR6	R4, R6 R7, PR9	HZ, HX HV	MP, HG HP	MR, RM RH	MR, RM RH	UC	HX HE	57, 65, TU	HX	R3, R4, R6(9A) R7(9B), R9(9C)	

※ Periphery grinding type

Comparison table for turning inserts chipbreaker

Comparison table for turning insert chipbreaker

Negative inserts

ISO	Machining range	ZCC-CT	SANDVIK	KORLOY	TaeguTec	WALTER	SECO	MITSUBISHI	SUMITOMO	KENNAMETAL	DIJET	HITACHI	TUNGALOY	KYOCERA	VALANTTE
<b>M</b>	For finishing	EF	MF	HA	FG, SF	NF4		FS	SU	K, FP		SE	SS	GU	F1, F2(2B), F5(5C)
	For semi-finishing	EM	MM	HS	ML, MP	NM4		MS, ES	EX, UP	P, MP	SF, SG	DE	SA, SM, S	SU, HU, ST	F3, F4(8A), M2(2C), M3, M4, M5(5B), M6, M7, 55, M8
	For roughing	ER	MR	GS, HM	MT, RH	NR4	M5, MR7 56, R6	GH, HZ	MP	RP					R3, R4, R6(9A) R7(9B), R9(9C)
<b>K</b>	For finishing	PM	KF	Without chip-breaker	FG	MA		Through chip-breaker,	UZ	FN		Y	CM	Through chip-breaker, C	F2(2B)
	For Semi-Finishing	PM	KM	Through chip-breaker, HM	MC, MT, MG	MA, NM5		Through chip-breaker,	UX	Through chip-breaker, UN		V	33, through chip-breaker,	ZS, GC	M5(5B), M6, M8
	For roughing	Without chip-breaker	KR	GR, HR, GH	RT, RH	MA		Without chip-breaker						Without chip-breaker	R3, R4, R7(9B)
<b>S</b>	For finishing	NF			SF	NF4	MF1※	FJ※		FS※, K※					F5(5C), M2(2C)
	For semi-finishing	NM	NGP※, 23			NM4	M1	MJ※	SU※	NGP※			SA		M4, M5(5B), M7, 55
	For roughing		SR			NR4		GJ		MS					

※ Periphery grinding type

Comparison table for turning inserts chipbreaker

Threading

Parting and grooving

General turning

## Comparison table for turning inserts chipbreaker

General turning

Parting and grooving

Threading

Comparison table for turning inserts chipbreaker

### Comparison table for turning insert chipbreaker Positive inserts

ISO	Machining range	ZCC.CT	SANDVIK	KORLOY	TaeguTec	WALTER	SECO	MITSUBISHI	SUMITOMO	KENNAMETAL	DIJET	HITACHI	TUNGALOY	KYOCERA	VALANTE
<b>P</b>	For finishing	USF, SF, HF	UF, PF	HFP	FA, FG	PF4	FF1 F1	FV, SV	FP, LU SU, SK	11, UF LF		JQ	01※, PF FS	GP, XP VF	PF4 JQ, JZ
	For finishing (Wiper)		WK※, W WP			PF	W-F1	SW	LUW	FW					
	For semi-finishing	HM	UM PM	HMP C25	MT, CMX	PS5 PM5	F2	MV, through chip-breaker	MU	MF	FT	JE	PM 23, 24	HQ, XQ GK	PM2 PM4
For semi-finishing (Wiper)			WM		PM		MW			MW					
<b>M</b>	For finishing	EF	MF	HFP	FA, FG	PF4		SV					SS※		1A, 2A
	For semi-finishing	EM	MM	HMP C25	MT CMX	PS5 PM5		Through chip-breaker, MV							PM2 PM4
<b>K</b>	For semi-finishing	HM, HR without chip-breaker	KF KM KR	HMP C25	MT CMX	MW PS5 PM5		Without chip-breaker			FT		Without chip-breaker	Without chip-breaker※	PM2 PM4
	For finishing/ For semi-finishing					PF4 PS5 PM5		FJ※	SC※	LF※ HP※					PM2, 1A 2A
<b>N</b>	For general turning	LC, LH	AL	TAAK MA	FL	PM2			AG	HP	ALU ACB		PP	A3	1L, 1A 2A

※ Periphery grinding type