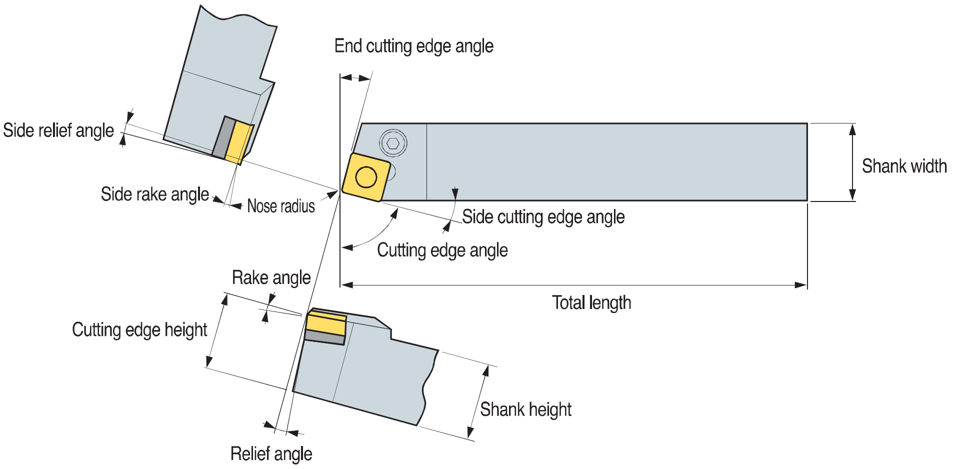
Insert shape and terminology

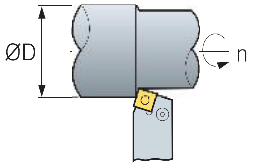


Relating angles between tool and workpiece

Cutting edge inclination	Terminology	Function	Effect
Rake angle	Side rake angle Rake angle	• Cutting force, Cutting heat, The effects of chip control on tool life	• (+) : Excellent machine-ability (reducing cutting force, weakening cutting edge strength) • (+) : When machining excellent machine-ability or thin workpiece. • (-) : When strong cutting edge is needed at interrupted condition or mill scale.
Relief angle	Relief angle Side relief angle	• Only cutting edge contact with cutting face	• (-) : Cutting edge is strong but has short tool life to make bad influence on flank wear.
Cutting edge angle	Cutting edge angle	• Affects chip control and cutting force direction	• (+) : Improved chip control because chip thickness is big.
	Side cutting edge angle	• Affects chip control and cutting force direction	• (+) : Strong cutting edge due to distributed cutting force but chip control is bad by thin chip thickness • (-) : Improved chip performance.
	End cutting edge angle	• Prevent friction between cutting edge and cutting face	• (-) : Cutting edge is strong but has short tool life to make bad influence on flank wear.

Calculation formulas for machining

Cutting speed



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

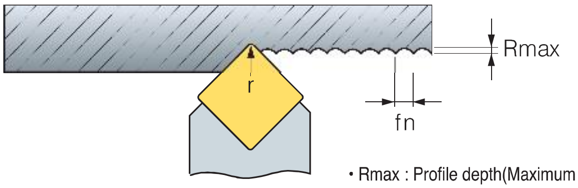
- vc : Cutting speed (m/min)
- D : Diameter (mm)
- n : Revolution per minute (min^{-1})
- π : Circular constant(3.14)

Feed

$$fn = \frac{vf}{n} \text{ (mm/rev)}$$

- fn : Feed per revolution(mm/rev)
- vf : Table feed (mm/min)
- n : Revolution per minute (min^{-1})

Surface finish



- Rmax : Profile depth(Maximum height roughness) (μ)
- fn : feed (mm/rev)
- r : nose radius

- Theoretical surface roughness

$$Rmax = \frac{fn^2}{8r} 1000 (\mu m)$$

- Practical surface roughness

Steel : Rmax × (1.5~3)
Cast iron : Rmax × (3~5)

Power requirement

$$P_{kw} = \frac{Q \times kc}{60 \times 102 \times \eta} \qquad P_{HP} = \frac{P_{KW}}{0.75} \qquad Q = \frac{vc \times fn \times ap}{1000}$$

- PKW : Power requirement [kW]
- PHP : Power requirement (horse power) [HP]
- vc : Cutting speed [m/min]
- ap : Depth of cut [mm]
- fn : Feed per revolution [mm/rev]
- kc : Specific cutting resistance [kg/mm²]
- η : Machine efficiency rate (0.7~0.8)

Rough Kc	
Mild steel	190
Medium carbon steel	210
High carbon steel	240
Low alloy steel	190
High alloy steel	245
Cast iron	93
Malleable cast iron	120
Bronze, Brass	70

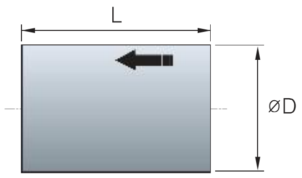
Material removal rate

$$Q = \frac{vc \times fn \times ap}{1000}$$

- Q : Material removal rate [㎤/min]
- ap : Depth of cut [mm]
- vc : Cutting speed [m/min]
- fn : Feed per revolution [mm/rev]

Machining time

External face machining 1



Constant Revolution per minute

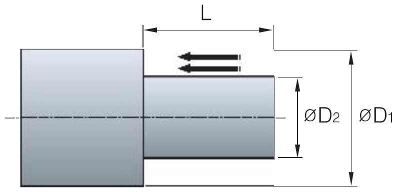
$$T = \frac{60 \times L}{fn \times n}$$

Constant cutting speed

$$T = \frac{60 \times \pi \times L \times D}{1000 \times fn \times vc}$$

T : Machining time [sec]
L : Cutting length [mm]
fn : Feed per revolution [mm/rev]
n : Revolution per minute [min]
D : Diameter of workpiece [mm]
vc : Cutting speed [m/min]

External face machining 2



Constant Revolution per minute

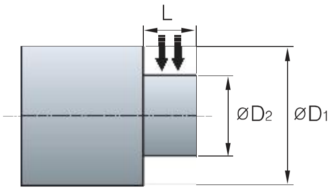
$$T = \frac{60 \times L}{fn \times n} \times N$$

Constant cutting speed

$$T = \frac{60 \times \pi \times L \times (D_1 + D_2)}{2 \times 1000 \times fn \times vc} \times N$$

T : Machining time [sec]
L : Cutting length [mm]
fn : Feed per revolution [mm/rev]
n : Revolution per minute [min]
D1 : Maximum diameter of workpiece [mm]
D2 : Minimum diameter of workpiece [mm]
vc : Cutting speed [m/min]
N : The number of pass = (D1-D2)/d/2

Facing



Constant Revolution per minute

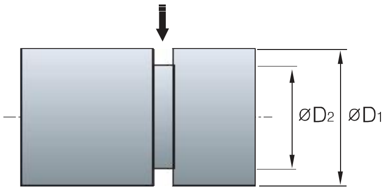
$$T = \frac{60 \times (D_1 - D_2)}{2 \times fn \times n} \times N$$

Constant cutting speed

$$T_1 = \frac{60 \times \pi \times (D_1 + D_2) \times (D_1 - D_2)}{4000 \times fn \times vc} \times N$$

T : Machining time [sec]
T1 : Machining time before the maximum rpm[sec]
L : Width of machining [mm]
fn : Feed per revolution [mm/rev]
n : Revolution per minute [min-1]
D1 : Maximum diameter of workpiece [mm]
D2 : Minimum diameter of workpiece [mm]
vc : Cutting speed [m/min]
N : The number of pass = (D1-D2)/d/2

Grooving



Constant Revolution per minute

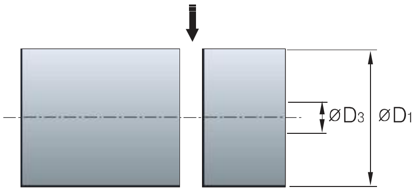
$$T = \frac{60 \times (D_1 - D_2)}{2 \times fn \times n}$$

Constant cutting speed

$$T_1 = \frac{60 \times \pi \times (D_1 + D_2) \times (D_1 - D_2)}{4000 \times fn \times vc}$$

T : Machining time [sec]
T1 : Machining time before the maximum rpm[sec]
L : Width of machining [mm]
fn : Feed per revolution [mm/rev]
n : Revolution per minute [min-1]
D1 : Maximum diameter of workpiece [mm]
D2 : Minimum diameter of workpiece [mm]
vc : Cutting speed [m/min]

Parting



Constant Revolution per minute

$$T = \frac{60 \times D_1}{2 \times fn \times n}$$

Constant cutting speed

$$T_1 = \frac{60 \times \pi \times (D_1 + D_3)\ (D_1 - D_3)}{4000 \times fn \times vc}$$

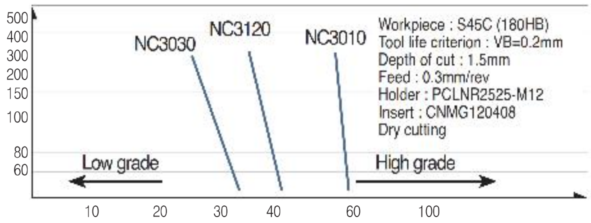
$$T_3 = T_1 + \frac{60 \times D_3}{2 \times fn \times n_{max}}$$

T : Machining time [sec]
T1 : Machining time before the maximum rpm[sec]
T3 : Machining time till maximum RPM[sec]
fn : Feed per revolution [mm/rev]
n : Revolution per minute [min-1]
nmax : Maximum revolution per minute [min-1]
D1 : Maximum diameter of workpiece [mm]
D3 : Maximum diameter at maximum RPM [mm]
vc : Cutting speed [m/min]

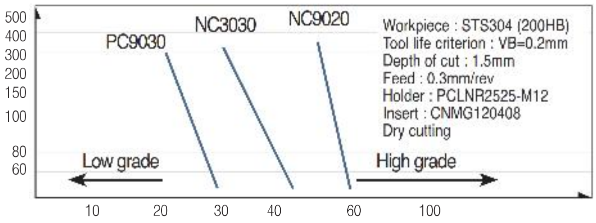
The affects of cutting condition

- The most desirable machining means short machining time, long tool life and good precision. This is the reason that proper cutting condition for each tools should be selected according to material's properties, hardness, shapes, the efficiency of machine.

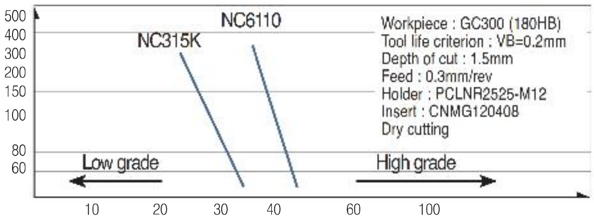
Cutting speed



The tool life feature of P grade



The tool life feature of M grade



The tool life feature of K grade

Cutting Speed's effects

- When the cutting speed increases up to 20% in an application, the tool life respectively decreases down 50%. Although inversely, if the cutting speed increases up to 50% the tool life decreases 20%. On the other hand if cutting speed is too low (20-40m/min) Tool life shortens due to vibration.

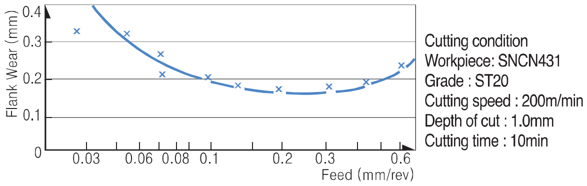
Feed

- The feed rate in turning means the progressed interval of a distance in a work piece within 1 revolution. The feed rate in a milling application means the table feed divided by number of teeth of cutter (feed rate per tooth).

The effects of feed

- When the feed rate decreases the flank wear is increased. When the feed rate is too low, the tool life shortens radically.
- When the feed rate increases, the flank wear increases due to high temperatures, however the feed rates effects tool life less than the cutting speed. And higher feed rates improve machining efficiency.

Relationship between feed and flank wear in steel turning



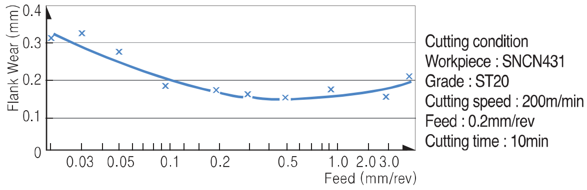
Depth of cut

- Determined by the required allowances in machining a material and the capacity the machine can tolerate.
  There are cutting limits according to the different shapes and sizes of the insert.

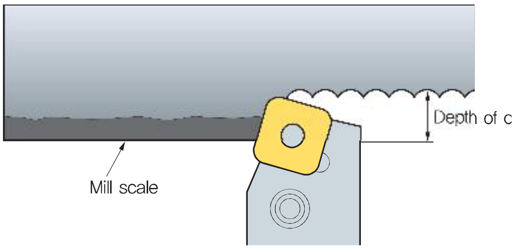
The effect of a depth of cut

- The depth of cut does not have a big influence on tool life.
- When the depth of cut is small the work piece is not cut but rather rubbed. In these cases, machine off the work hardened parts that decrease tool life.
- When machining a cast skin or milling scale smaller depth of cuts usually cause chipping and abnormal wear because of hard impurities in the surface of the work piece.

Relationship between depth of cut and flank wear in steel turning



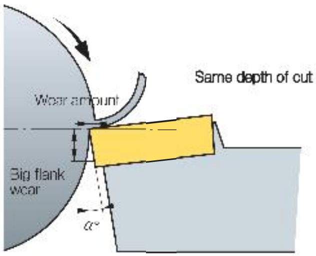
Surface parts including mill scale Roughing



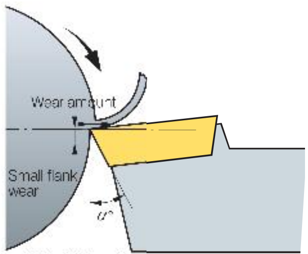
Relief angle

Relief angle avoids the friction between workpiece and relief face and makes cutting edge move along workpiece easily.

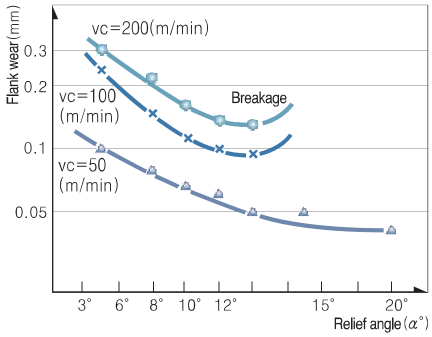
Relationship between various relief angle and flank wear



Low relief angle



High relief angle



- Workpiece : SNCM431(HB200)
- Grade : P20
- ap : 1mm
- fn : 0.32mm/rev
- T : 20min

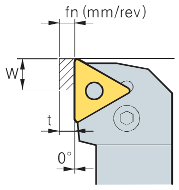
- Affects
  1. If relief angle is big Flank wear decreases.
  2. If relief angle is big Cutting edge strength weakens.
  3. If relief angle is small Chattering occurs .
- Selection system
  1. Hard workpiece / When strong cutting edge is needed - Low relief angle
  2. Soft workpiece / Workpiece turning to work hardening easily - High relief angle

---

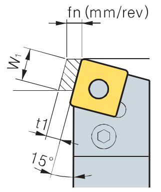
Side cutting edge angle

Side cutting edge angle has big influence on chip flow and cutting force therefore proper Side cutting edge angle is very important.

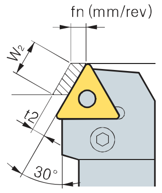
Side cutting edge angle and Chip thickness



① Approach angle 0˚



② Approach angle 15˚

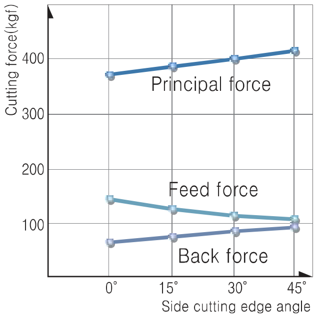


③ Approach angle 30˚

As side cutting edge angle is getting bigger chips are getting thinner and wider(refer to left picture).
At the same feed and depth of cut with approach angle 0˚
Chip thickness is the same as feed(t=fn) and chip width is equal to depth of cut (W=ap).

t1 = 0.97t, W1 = 1.04W
t2 = 0.87t, W2 = 1.15W

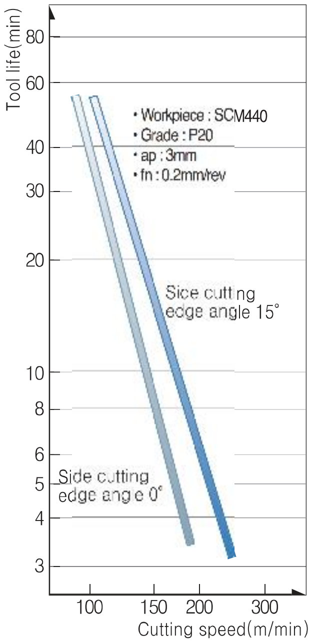
Side cutting edge angle and 3 cutting forces



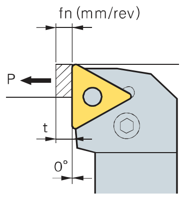
- Workpiece : SCM440(HB250)
- Grade : TNGA220412
- vc= 100m/min
- ap= 4mm
- fn= 0.45mm/rev

- Affects
  1. Big side cutting edge angle with the same feed makes chip attaching length longer and chip thickness thinner. So that cutting forces scatter to long cutting edge therefore tool life gets longer.
  2. Big side cutting edge angle for machining long bars can cause bending.
- Selection system
  1. Deep depth of cut finishing / Long thin workpiece / Low machine rigidity - Side cutting edge angle
  2. Hard and high calorific power workpiece / Roughing big workpiece / High machine rigidity - Side cutting edge angle

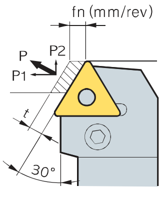
Side cutting edge angle and Tool life



Side cutting edge angle and Cutting load



① "Force P is loaded."



② "Force P is scattered to P1, P2."

As approach angle gets bigger Back force gets bigger and feed force gets smaller.

Side cutting edge angle and Cutting performance

Specification	Low ← Approach angle → High	
Wear rate	High ←‑‑‑‑→	Low
Workpiece	Easy to cut material ←‑‑‑‑→	Difficult to cut material
Machining power	Small ←‑‑‑‑→	Big
Chatter	Hard to occur ←‑‑‑‑→	Easy to occur
How to machine	Finishing ←‑‑‑‑→	Roughing
Workpiece rigidity	Long thin workpiece ←‑‑‑‑→	Thick workpiece
Machine rigidity	In case of low rigidity ←‑‑‑‑→	In case of high rigidity

End cutting edge angle

It affects machined surface to prevent interference between surface of workpiece and insert.

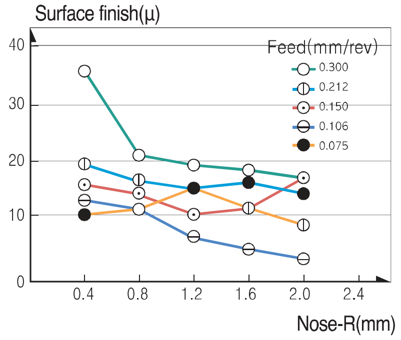
Affects

1. If end cutting edge angle reduces cutting edge get stronger but cutting heat generated by machining increases.
2. Small end cutting edge angle can cause chattering due to the increases cutting force.

Nose-R

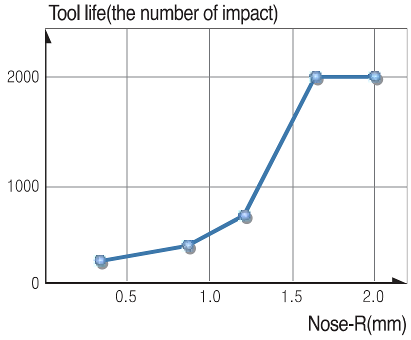
1. Nose-R affects not only surface roughness but strength of cutting edge.
2. In general, It's desirable that Nose-R is 2~3 times bigger than feed.

Nose R and surface finish



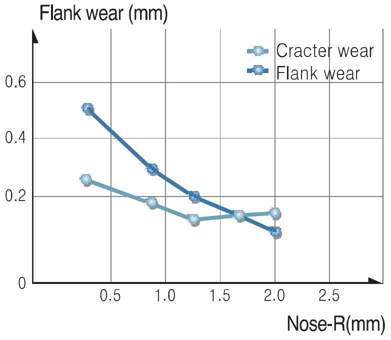
- Workpiece : SNCM439, HB200
- Grade : P20
- vc = 120m/min, ap = 0.5mm

Nose R and tool life



- Workpiece : SCM440, HB280
- Grade : P10
- vc = 100m/min, ap = 0.5mm
- fn = 0.3mm/rev

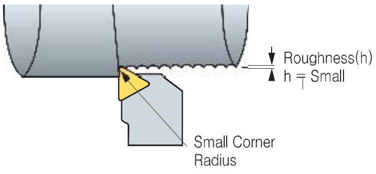
Nose R and wear of tool



- Workpiece : SNCM439, HB200
- Grade : P10
- vc = 140m/min, ap = 2mm
- fn = 0.2mm/rev, T = 10min

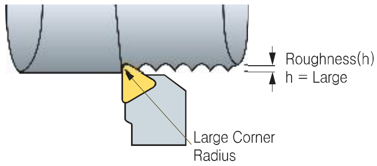
Affects of Nose-R

1. Big Nose-R improves surface finish.
2. Big Nose-R improves cutting edge strength.
3. Big Nose-R reduces flank wear and crater wear.
4. Too big Nose-R causes chattering due to increased cutting force.

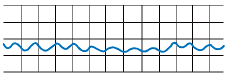
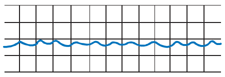
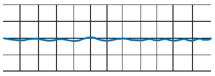
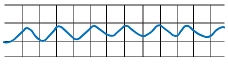
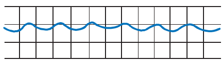
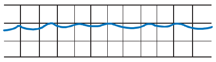
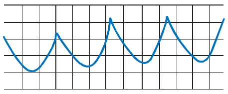
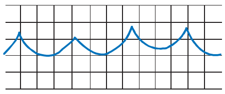
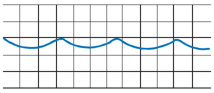


Selection system

1. For finishing with small depth of cut / long and thin workpiece / When machine power is low - Small Nose-R
2. For applications that need strong cutting edge such as intermittent and machining mill scale / For roughing of big workpiece / When the machine power is strong eough - Big Nose-R



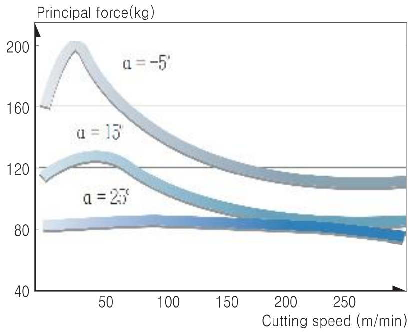
Relationship between nose radius, feed and various surface roughness.

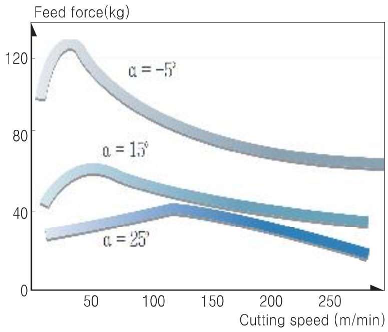
Nose "R" Feed(mm/rev)	0.4	0.8	1.2
0.15			
0.26			
0.46			

Cutting edge shape and the affects

Rake angle

Rake angle has big influence on cutting force, chip flow and tool life.





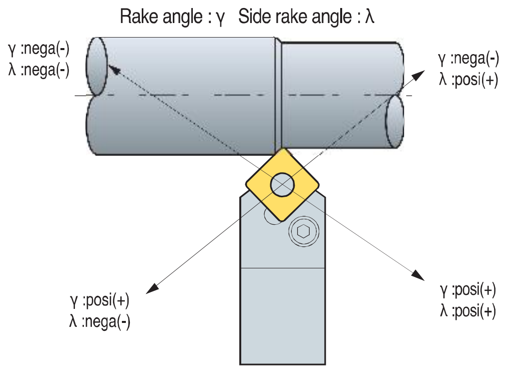
• Affects

1. High rake angle results in good suface finish.
2. As the rake angle increases by 1° Machining power decreases by 1%.
3. High rake angle weakens cutting edge.

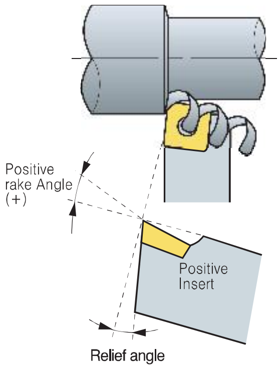
• Selection system

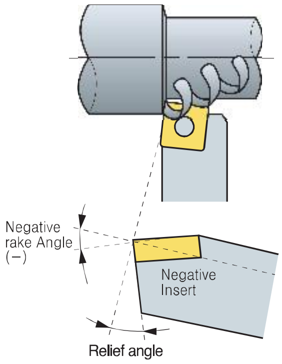
1. For hard workpiece / For applications that need strong cutting edge such as interrupted and machining mill scale
   - Low rake angle
2. For soft workpiece / Easy to cut material / When the rigidity of machine power and workpiece is low - High rake angle

Rake angle and the direction of chip flow



In order to prevent machined surface from damages Avoid nega, posi combination. γ :nega(-) λ :posi(+)





Selecting proper tools

Nowadays, It's very difficult to select the best tools in complicating tooling system and various cutting conditions.
However, It can be simplified by classifying basic factors below.

Selection of inserts and tool holder

Listed below is the basic factors and choose B according to A.

A : Basic factors

- Workpiece material
- Workpiece shape
- Workpiece size
- Hardness of workpiece
- Surface roughness of workpiece (before machining)
- Surface finish required
- Type of lathe machine
- Condition of lathe machine (rigidity, power etc)
- Horse power of machine
- Clamping method of workpiece

B : Selection system

① Select as big approach angle as possible.
② Select as big shank as possible.
③ Select as strong cutting edge of insert as possible
④ Select as big nose radius as possible
⑤ In finishing, Select the insert using many corners
⑥ Select as small insert as possible
⑦ Cutting speed should be determined carefully according to cutting conditions
⑧ Select as deep depth of cut as possible
⑨ Select as fast feed as possible
⑩ Cutting condition should be determined within chip breaker application ranges.