

Insert and cutting parameters selection for Turning

Insert grade, chip breaker geometry, cutting parameters

Every cutting tool manufacturer has cutting parameters tables in tool catalogs. To decide cutting parameters for an insert, we just refer to the catalog of the manufacturer of the insert. This chart, for example, is for Taegutec's catalog, for turning inserts. The catalog will have charts like this for various workpiece materials.

Insert style **N** : Negative inserts **P** : Positive inserts

Application **F** : Finishing **M** : Medium **R** : Roughing

Depth of cut (mm)

Workpiece, stability and machine condition
 -Best: no scale, no interruption, good rigidity
 -Normal: a little scale, a little interruption, good rigidity
 -Poor: heavy scale, severe interruptions, poor rigidity

First and second choice grade, chipbreaker, cutting speed & feed rate in mm/rev

		Workpiece Material													
		0.15% Carbon Steel (150 ≈ BHN)				0.45% Carbon Steel (180 - 200BHN)				0.55% Carbon Steel (200 - 220BHN)					
F	- 1.0	Best	1	PV3010	FC	475	0.12	PV3010	FG	355	0.15	PV3010	FG	330	0.15
			2	CT3000	FC	430	0.12	TT8115	FG	340	0.15	TT8115	FG	315	0.15
M	1.0 - 2.5	Best	1	TT5100	ML	330	0.20	TT8115	MP	330	0.30	TT8115	MP	305	0.30
			2	TT8125	ML	420	0.20	TT8125	MP	300	0.30	TT8125	MP	280	0.30
		Normal	1	TT5100	MP	315	0.24	TT8115	PC	310	0.30	TT8115	PC	290	0.30
			2	TT8125	MP	400	0.24	TT8125	PC	280	0.30	TT8125	PC	260	0.30
		Poor	1	TT8020	MT	235	0.24	TT8135	RT	190	0.32	TT8135	RT	180	0.32
			2												
N	2.5 - 4.0	Best	1	TT5100	PC	300	0.28	TT8115	PC	310	0.35	TT8115	MP	290	0.35
			2	TT8125	PC	385	0.28	TT8125	PC	280	0.35	TT8125	PC	260	0.35
		Normal	1	TT5100	MT	285	0.28	TT8125	PC	280	0.35	TT8125	MT	260	0.35
			2	TT8125	MT	370	0.28	TT8125	MT	265	0.40	TT8125	MG-	245	0.40
		Poor	1	TT8020	MT	215	0.24	TT8135	RT	180	0.36	TT8135	RT	180	0.36
			2												
R	4.0 - 7.0	Normal	1	TT5100	RT	230	0.45	TT8125	RT	260	0.56	TT8125	RT	240	0.56
			2	TT8125	RT	320	0.45	TT8115	RT	290	0.56	TT8135	RT	270	0.56
		Poor	1	TT8020	RT	180	0.36	TT8135	RT	180	0.45	TT8135	RT	160	0.45
			2												
	7.0 -	Normal	1	TT5100	RH	210	0.57	TT8125	RH	245	0.71	TT8125	RH	225	0.71
			2												

To select the insert grade, chip breaker geometry and cutting parameters:

1. **Decide whether you want to use a Negative or Positive insert.** A negative insert is stronger, but also results in higher cutting forces. So you would use it when taking heavy or interrupted cuts, on a part that is held firmly and can withstand high cutting forces. A positive insert is weaker, sharper, and results in lower cutting forces. So you would use it to cut a part that is not held firmly, is likely to bend, or has thin walls. Based on Negative or Positive, look at the N or P rows in the first column.

2. **Decide on the application - Roughing, Medium machining, or Finishing.** Medium machining is merely roughing with a smaller depth of cut. Based on this, look at the F, M or R row in the second column. Now decide the depth of cut that you want to use, and look at the corresponding row. If you want to use a depth of cut of 3 mm, for example, you would look at the row 2.5 to 4.
3. **Decide in which category your work piece stability and machine condition falls** – Best, Normal or Poor. Best would be if the machine is in good condition, and the workpiece is held rigidly, is not likely to bend, has no scales, sand inclusions or blow holes. Poor would be if the machine is in bad shape, vibrates a lot, and the workpiece is not held rigidly, is very long or has thin walls, or is a casting or forging with scales and sand inclusions, or has a non-circular cross section that involves interrupted cutting. Normal would be between Best and Poor.
4. **You now get the grade and chipbreaker geometry** that you must use for the operation, **and the corresponding cutting speed and feed rate.** In the fifth column, row number 1 is the first choice and row number 2 is the second choice. Ideally use the first choice, or if you do not have it on your shop floor, use the second choice. The highlighted rectangle on the chart shows an example for cutting 0.45 % plain Carbon steel. We want to use a Negative insert, do medium machining with a depth of cut of 3 mm, and the machine and workpiece stability are Normal. We therefore use an insert of TT8125 grade with a PC chip breaker, a cutting speed of 280 m/min and feed rate of 0.35 mm/rev.
5. **Modify the cutting speed based on the required tool life.** The cutting parameters in the catalog are for a tool life of 15 minutes. This means that with these parameters the insert edge will have to be changed after 15 minutes of contact with the workpiece. If you want a different tool life, multiply the cutting speed by the appropriate factor in this table:

Tool life, minutes	10	15	20	25	30	45	60	90
Correction factor	1.11	1	0.93	0.88	0.84	0.75	0.7	0.6

Example:

If the cutting speed recommended in the catalog is 200 m/min and you want a tool life of 45 minutes, the multiplication factor is 0.75. You must therefore use a cutting speed of 150 m/min. For a life of 1 hour use a cutting speed of 140 m/min.