MILLSTAR

Choosing Cutting Parameters/Calculating Cutting Speed and Feed – INCH For Ball Nose Inserts

Table 1 - Cutting Conditions for Using Steel Shank Holders

Working Material	Hardness	Grade	SFM	Feed fn (inch/Rev) Insert Diameter (inch) 0.250	0.312	0.375	0.500	0.625	0.750	1.000	1.250	Ap Max	Ae Max
Low Alloy Steel (1.7225)	200-280HB	TLN, HSN	450-1200	0.008	0.012	0.016	0.016	0.020	0.020	0.020	0.020	.10 x D	.40 x D
Alloy & Die Steel (1.2311, P20, DME2/3/5)	32-42HRC	TLN, HSN	300-800	0.006	0.010	0.012	0.016	0.016	0.016	0.020	0.020	.8 x D	.35 x D
Tool Steel (1.2344, 1.2379)	42-52HRC	TLN, HSN	300-700	0.006	0.010	0.012	0.016	0.020	0.020	0.020	0.020	.6 x D	.30 x D
Stainless Steel (1.4301, 1.4401)	200-350HB	XRN, TLN, HSN	250-400	0.006	0.010	0.012	0.016	0.016	0.016	0.020	0.020	.10 x D	.40 x D
Gray Cast Iron (GG25-GG30)	160-260HB	TLN, HSN	600-1200	0.008	0.012	0.016	0.020	0.023	0.023	0.028	0.028	.10 x D	.50 x D
Nodular Cast Iron (GGG60-GGG70)	180-300HB	TLN, HSN	450-1200	0.008	0.012	0.016	0.020	0.023	0.023	0.028	0.028	.15 x D	.50 x D
Copper Alloy	80-150HB	XRN	450-1500	0.010	0.016	0.020	0.023	0.028	0.028	0.028	0.028	.10 x D	.40 x D
Aluminum Alloys	30-120HB	XRN	1000-3000	0.010	0.016	0.020	0.023	0.028	0.028	0.028	0.028	.10 x D	.50 x D
Graphite		TLN, HSN	600-2000	0.012	0.020	0.023	0.028	0.030	0.030	0.030	0.030	.20 x D	.50 x D
Ni & Co Based Alloy	250-320HB	XRN, HSN	100-300	0.006	0.008	0.012	0.016	0.016	0.020	0.020	0.020	.8 x D	.50 x D
Titanium Alloy (Annealed)	<350HB	XRN, HSN	150-400	0.006	0.008	0.010	0.012	0.012	0.016	0.020	0.020	.8 x D	.33 x D
Titanium Alloy (Sol. Treated/Aged)	<380HB	XRN, HSN	120-300	0.004	0.006	0.008	0.012	0.012	0.012	0.016	0.016	.8 x D	.35 x D

Table 2 - Cutting Conditions for Using Carbide Shank Holders

Working Material	Hardness	Grade	SFM	Feed fn (inch/Rev) Insert Diameter (inch) 0.250	0.312	0.375	0.500	0.625	0.750	1.000	1.250	Ap Max	Ae Max
Low Alloy Steel (1.7225)	200-280HB	TLN, HSN	450-1200	0.012	0.016	0.016	0.02	0.023	0.023	0.028	0.028	.10 x D	.50 x D
Alloy & Die Steel (1.2311, P20, DME2/3/5)	32-42HRC	TLN, HSN	300-1200	0.012	0.012	0.012	0.016	0.02	0.02	0.023	0.023	.8 x D	.40 x D
Tool Steel (1.2344, 1.2379)	42-52HRC	TLN, HSN	300-800	0.012	0.012	0.012	0.016	0.02	0.02	0.023	0.023	.6 x D	.35 x D
Stainless Steel (1.4301, 1.4401)	200-350HB	XRN, TLN, HSN	250-400	0.012	0.012	0.016	0.02	0.023	0.028	0.028	0.03	.10 x D	.50 x D
Gray Cast Iron (GG25-GG30)	160-260HB	TLN, HSN	600-1200	0.012	0.016	0.02	0.02	0.023	0.028	0,8	0.04	.10 x D	.40 x D
Nodular Cast Iron (GGG60-GGG70)	180-300HB	TLN, HSN	450-1200	0.012	0.016	0.016	0.02	0.023	0.023	0.028	0.03	.10 x D	.40 x D
Copper Alloy	80-150HB	XRN	450-1500	0.012	0.016	0.016	0.02	0.023	0.023	0.028	0.028	.10 x D	.40 x D
Aluminum Alloys	30-120HB	XRN	1000-3000	0.012	0.016	0.02	0.023	0.028	0.028	0.03	0.03	.10 x D	.40 x D
Graphite		TLN, HSN	600-2000	0.012	0.02	0.023	0.028	0.03	0.03	0.03	0.03	.20 x D	.40 x D
Ni & Co Based Alloy	250-320HB	XRN, HSN	100-300	0.012	0.012	0.016	0.016	0.02	0.023	0.023	0.028	.8 x D	.50 x D
Titanium Alloy (Annealed)	<350HB	XRN, HSN	150-400	0.006	0.008	0.012	0.012	0.012	0.016	0.02	0.02	.8 x D	.50 x D
Titanium Alloy (Sol. Treated/Aged)	<380HB	XRN, HSN	120-300	0.004	0.006	0.008	0.012	0.012	0.016	0.016	0.02	.8 x D	.50 x D
Harden Steel (1.2344, 1.2379)	45-55HRC	TLN, HSN	300-1200	0.008	0.01	0.012	0.016	0.02	0.02	0.023	0.023	.8 x D	.35 x D

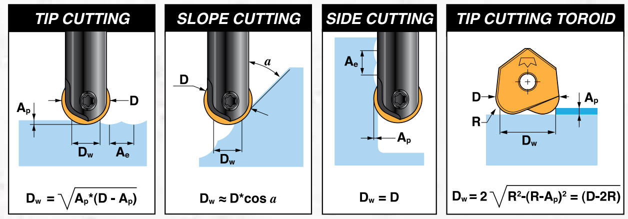
Choosing Cutting Parameters/Calculating Cutting Speed and Feed – INCH

1. Find the Cutting Speed & Feed fn

Find SFM and fn range in Table 1 or Table 2 at left. Choose the average value for SFM and the lower value for feed in the range.

2. Compute the Dw

In order to compute the RPM value of the spindle it is necessary to determine the Dw which is the effective engaged tool diameter. The Dw depends on the geometry of the inserts (ball nose or toroid) and the relative position of the tool against the working piece surface. Example calculation is of Dw is presented to the right.



3. Calculate Spindle Speed

Use the formula: N = SFM x 3.82 (÷Dw)

Table 3 - Working Diameter For Ball Nose Tools

ØD	0.250	0.312	0.375	0.500	0.625	0.750	1.000	1.250
Depth of cut	Dw Working Diameter (inch) Actual effective cutting diameter							
0.020	0.135	0.153	0.169	0.196	0.220	0.242	0.280	0.314
0.050	0.200	0.229	0.255	0.300	0.339	0.374	0.436	0.490
0.075	0.229	0.267	0.300	0.357	0.406	0.450	0.527	0.594
0.100	0.245	0.292	0.332	0.400	0.458	0.510	0.600	0.678
0.125	0.250	0.306	0.345	0.433	0.500	0.559	0.661	0.750
0.156		0.312	0.370	0.464	0.541	0.609	0.726	0.827
0.188			0.375	0.484	0.573	0.650	0.781	0.893
0.250				0.500	0.612	0.707	0.866	1.000
0.312					0.625	0.739	0.927	1.082
0.375						0.750	0.968	1.146
0.500							1.000	1.225
0.625								1.250

Table 4 - Working Diameter For Toroid Tools

Insert Diameter "D"	0.375	0.500	0.625	0.750	1.000	1.250
Depth of cut	Dw Working Diameter (inch) Actual cutting diameter of toroid inserts					
0.020	0.260	0.385	0.465	0.544	0.696	0.845
0.050	0.325	0.450	0.541	0.630	0.800	0.964
0.075	0.354	0.479	0.579	0.675	0.867	1.031
0.100	0.370	0.495	0.604	0.707	0.900	1.083
0.125	0.375	0.500	0.618	0.720	0.933	1.125
0.156			0.625	0.745	0.964	1.166
0.188				0.750	0.984	1.198
0.250					1.000	1.237
0.312						1.250

4. Calculate the Table Feed V_f (m/min)

Use the formula: $V_f = N * f_n * K_f$. K_f is the feed rate multiplier coefficient taking into consideration that chip load is less than theoretical value. Take the value of K_f from Table 5 or Table 6.

Table 5 - Feed Rate Multiplier For Ball Nose Inserts

Insert Diameter "D"	0.250	0.312	0.375	0.500	0.625	0.750	1.000	1.250
Depth of cut	Feedrate Multiplier Factors (for Working Diameters Dw)							
0.020	1.850	2.040	2.220	2.550	2.840	3.000	3.750	4.000
0.050	1.250	1.360	1.470	1.670	1.840	2.000	2.290	2.550
0.075	1.090	1.170	1.250	1.400	1.540	1.670	1.900	2.100
0.100	1.020	1.070	1.130	1.250	1.370	1.470	1.670	1.840
0.125	1.000	1.020	1.060	1.150	1.250	1.340	1.510	1.660
0.156		1.000	1.010	1.080	1.160	1.230	1.380	1.510
0.188			1.000	1.030	1.090	1.150	1.280	1.400
0.250				1.000	1.020	1.060	1.150	1.250
0.312					1.000	1.020	1.080	1.150
0.375						1.000	1.030	1.090
0.500							1.000	1.020
0.625								1.000

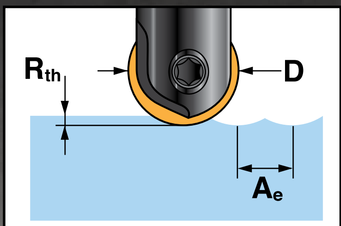
Table 6 - Feed Rate Multiplier For Toroid Tools

Insert Diameter "D"	0.375	0.500	0.625	0.750	1.000	1.250
Depth of cut	FEEDRATE MULTIPLIER FACTORS (inch) (for Toroid Working Diameters Dw)					
0.020	1.850	1.850	2.040	2.220	2.550	2.840
0.050	1.250	1.250	1.360	1.470	1.670	1.840
0.075	1.090	1.090	1.170	1.250	1.400	1.540
0.100	1.020	1.020	1.070	1.130	1.250	1.370
0.125	1.000	1.000	1.020	1.060	1.150	1.250
0.156			1.000	1.010	1.080	1.160
0.188				1.000	1.030	1.090
0.250					1.000	1.020
0.312						1.000

Verify Surface Roughness (R_{th})

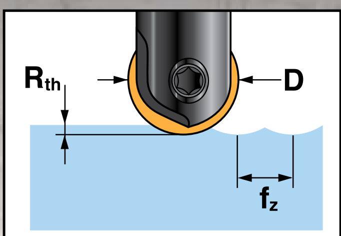
1. Decreasing the A_e and feed by half will improve surface roughness by 4 times.
2. Using $f_z = A_e$ in most cases is the best option.

Surface Roughness Step-Over



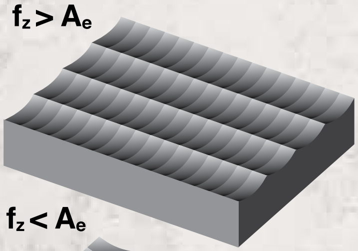
$$R_{th}, A_e = \frac{D}{2} - \sqrt{\frac{D^2 - A_e^2}{4}}$$

Surface Roughness Feed Dir

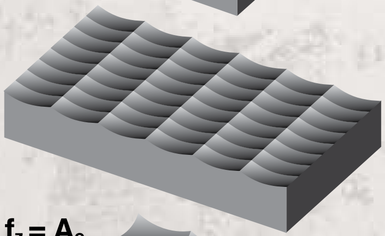


$$R_{th}, A_e = \frac{D}{2} - \sqrt{\frac{D^2 - f_z^2}{4}}$$

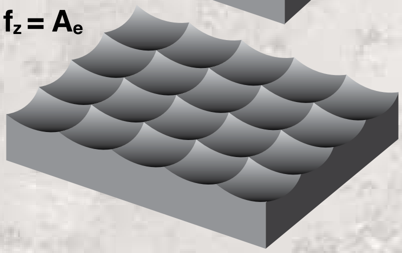
$f_z > A_e$



$f_z < A_e$



$f_z = A_e$



Inch

Feed & Speed Calculations

Nomenclature

- D = cutter/insert diameter
- Dw = effective cutter diameter
- z = number of teeth
- Vc = cutting speed
- n = number of revolutions per minute
- Vf = feed rate, or table feed
- f = feed per revolution
- fz = feed per tooth
- Fzcor = feed/tooth, chip thinning corrected
- CF = centerline feed, helical interpolation
- x = multiplier symbol (inch)
- • = multiplier symbol (metric)
- ap = axial depth of cut
- ae = width of cut, step-over
- R = insert radius
- SFM = surface feet per minute
- RPM = revolutions per minute
- IPM = inch per minute, feed rate
- π = 3.14159...circle, circumference:dia. ratio
- Rth,ae = theoretical surface roughness in step-over direction
- Rth,fz = theoretical surface roughness in feed direction
- B = bore dia., helical interpolation
- IC = inscribed circle (2 x R)
- Q = metal removal rate

Cutting Speed

To find the SFM of a cutter:

$$SFM = \frac{\varpi \times D \times RPM}{12} = 0.262 \times D \times RPM$$

Example:
To find the SFM of a 3/4" Ø cutter rotating at 6000 RPM:

$$.262 \times D \times RPM = 0.262 \times 0.75 \times 6000 = 1179\ SFM$$

Metric Formula:

$$v_c = \frac{n \bullet \pi \bullet Dw}{1000} = m / minute$$

Spindle Speed

To find the RPM of a cutter:

$$RPM = \frac{12 \times SFM}{\varpi \times D} = \frac{3.82 \times SFM}{D}$$

Example
To find the RPM of a 1/2" Ø cutter rotating at 800 SFM:

$$RPM = \frac{3.82 \times 800}{.500} = 6112\ RPM$$

Metric Formula:

$$n = \frac{v_c \bullet 1000}{\varpi \bullet Dw} = min^{-1}$$

Feed Per Tooth

To find the feed per tooth of a cutter:

$$FZ = \frac{IPM}{z \times RPM}$$

Example
To find the feed per tooth of a two flute cutter rotating at 10000 RPM with a table travel of 240 inches per minute:

$$FZ = \frac{240}{2 \times 10\,000} = 0.012\ FZ$$

Metric Formula:

$$f = \frac{v_f}{n \bullet z} = mm / tooth$$

Feed Rate or Table Feed

To find the feed (table feed) in inches per minute:

$$IPM = RPM \times f_z \times z$$

Example
To find the feed per tooth of a two flute cutter rotating at 5000 RPM with a feed per tooth of 0.006:

$$IPM = 5000 \times 0.006 \times 2 = 60\ IPM$$

Metric Formula:

$$v_f = n \bullet f_z \bullet z = mm / minute$$

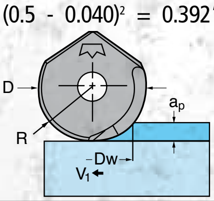
Effective Cutting Diameter, Ball Nose

To find the effective cutter Ø of a ball nose tool:

$$Dw = 2 \times \sqrt{R^2 - (R - a_p)^2}$$

Example
To find the effective cutter diameter engaged of a 1.0" Ø ball nose tool cutting at 0.040" depth of cut:

$$Dw = 2 \times \sqrt{0.5^2 - (0.5 - 0.040)^2} = 0.392"$$



Metric Formula:
use same formula in mm.

Effective Cutting Diameter, Toroid

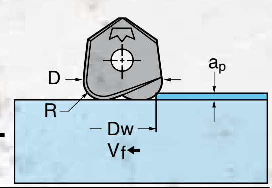
To find the effective cutter Ø of a Toroid bull nose tool:

$$Dw = 2 \times \sqrt{R^2 - (R - a_p)^2} + (D - 2R)$$

Example

$$Dw = 2 \times \sqrt{.1875^2 - (.1875 - .1)^2} + (.75 - (2 \times .1875)) = .707$$

To find the effective cutter diameter engaged of a 3/4" Ø Toroid bull nose tool cutting at 0.100" depth of cut:



Metric Formula: use same formula in mm.

Surface Roughness, Step-Over

To find the surface roughness in step-over direction:

$$R_{th,ae} = \frac{D}{2} - \sqrt{\frac{D^2 - a_e^2}{4}}$$

Example
To find the theoretical roughness of a 3/4" Ø ball nose tool in step-over direction of the cut (peak-to-valley or cusp height), with a 0.030 step-over value:

$$R_{th,ae} = \frac{.75}{2} - \sqrt{\frac{.75^2 - .03^2}{4}} = 0.0003"$$

Metric Formula: use same formula in mm.

Surface Roughness, Feed Direction

To find the surface roughness in feed direction:

$$R_{th,fz} = \frac{D}{2} - \sqrt{\frac{D^2 - f_z^2}{4}}$$

Example
To find the theoretical roughness of a 3/4" Ø ball nose tool in feed direction of the cut (peak-to-valley or cusp height), with a 0.005 feed per tooth value:

$$R_{th,fz} = \frac{.75}{2} - \sqrt{\frac{.75^2 - .005^2}{4}} = 0.000008"$$

Metric Formula: use same formula in mm.

MILLSTAR

Choosing Cutting Parameters/Calculating Cutting Speed and Feed – METRIC For Ball Nose Inserts

Table 1 - Cutting Conditions for Using Steel Shank Holders

Working Material	Hardness	Grade	Vc m/min	Feed fn (mm/Rev) Insert Diameter (mm)									Ap Max	Ae Max
				6	8	10	12	16	20	25	30	32		
Low Alloy Steel(1.7225)	200-280HB	TLN, HSN	150-200	0,2	0,3	0,4	0,4	0,5	0,5	0,6	0,6	0,6	.15 x D	.15 x D
Alloy & Die Steel (1.2311, P20, DME2/3/5)	32-42HRC	TLN, HSN	100-150	0,15	0,25	0,3	0,4	0,4	0,4	0,5	0,5	0,5	.20 x D	.20 x D
Tool Steel (1.2344, 1.2379)	42-52HRC	TLN, HSN	120-160	0,15	0,25	0,3	0,4	0,5	0,5	0,6	0,6	0,6	.20 x D	.20 x D
Stainless Steel (1.4301, 1.4401)	200-350HB	XRN, TLN, HSN	90-120	0,15	0,25	0,3	0,4	0,4	0,4	0,5	0,5	0,5	.20 x D	.20 x D
Gray Cast Iron (GG25-GG30)	160-260HB	TLN, HSN	200-360	0,2	0,3	0,4	0,5	0,6	0,6	0,7	0,7	0,7	.10 x D	.10 x D
Nodular Cast Iron (GGG60-GGG70)	180-300HB	TLN, HSN, HSN	150-300	0,2	0,3	0,4	0,5	0,6	0,6	0,7	0,7	0,7	.15 x D	.15 x D
Copper Alloy	80-150HB	XRN	150-200	0,25	0,4	0,5	0,6	0,7	0,7	0,8	0,8	0,8	.10 x D	.10 x D
Aluminum Alloys	30-120HB	XRN	200-300	0,25	0,4	0,5	0,6	0,7	0,7	0,8	0,8	0,8	.6 x D	.6 x D
Graphite		TLN	200-400	0,3	0,5	0,6	0,7	0,8	0,8	0,9	0,9	0,9	.5 x D	.5 x D
Ni & Co Based Alloy	250-320HB	XRN, HSN	30-70	0,15	0,2	0,3	0,4	0,4	0,5	0,5	0,6	0,6	.30 x D	.30 x D
Titanium Alloy (Annealed)	<350HB	XRN, HSN	50-120	0,15	0,2	0,25	0,35	0,35	0,4	0,45	0,5	0,5	.33 x D	.33 x D
Titanium Alloy (Sol. Treated/Aged)	<380HB	XRN, HSN	40-90	0,1	0,15	0,2	0,3	0,3	0,35	0,4	0,45	0,45	.35 x D	.35 x D
Harden Steel (1.2344, 1.2379)	45-55HRC	TLN, HSN	70-90	0,15	0,25	0,3	0,4	0,5	0,5	0,6	0,6	0,6	.30 x D	.30 x D

Table 2 - Cutting Conditions for Using Carbide Shank Holders

Working Material	Hardness	Grade	Vc m/min	Feed fn (mm/Rev) Insert Diameter (mm)									Ap Max	Ae Max
				6	8	10	12	16	20	25	30	32		
Low Alloy Steel (1.7225)	200-280HB	TLN, HSN	260-380	0,3	0,4	0,4	0,5	0,6	0,6	0,7	0,7	0,7	.15 x D	.50 x D
Alloy & Die Steel (1.2311, P20, DME2/3/5)	32-42HRC	TLN, HSN	250-330	0,25	0,3	0,3	0,4	0,5	0,5	0,6	0,6	0,6	.20 x D	.50 x D
Tool Steel (1.2344, 1.2379)	42-52HRC	TLN, HSN	240-320	0,25	0,3	0,3	0,4	0,5	0,5	0,6	0,6	0,6	.20 x D	.50 x D
Stainless Steel (1.4301, 1.4401)	200-350HB	XRN, TLN, HSN	200-260	0,25	0,3	0,4	0,5	0,6	0,65	0,7	0,8	0,8	.20 x D	.50 x D
Gray Cast Iron (GG25-GG30)	160-260HB	TLN, HSN	360-450	0,35	0,45	0,5	0,5	0,6	0,7	0,8	1,0	1,0	.10 x D	.40 x D
Nodular Cast Iron (GGG60-GGG70)	180-300HB	TLN, HSN	300-400	0,3	0,4	0,4	0,5	0,6	0,6	0,7	0,8	0,8	.15 x D	.15 x D
Copper Alloy	80-150HB	XRN	300-400	0,3	0,4	0,4	0,5	0,6	0,6	0,7	0,7	0,7	.10 x D	.40 x D
Aluminum Alloys	30-120HB	XRN	400-500	0,3	0,4	0,5	0,6	0,7	0,7	0,8	0,8	0,8	.6 x D	.40 x D
Graphite		TLN, HSN	600-800	0,3	0,5	0,6	0,7	0,8	0,8	0,9	0,9	0,9	.5 x D	.40 x D
Ni & Co Based Alloy	250-320HB	XRN, HSN	80-110	0,25	0,3	0,4	0,4	0,5	0,6	0,6	0,7	0,7	.30 x D	.50 x D
Titanium Alloy (Annealed)	<350HB	XRN, HSN	150-230	0,15	0,2	0,25	0,35	0,35	0,4	0,45	0,5	0,5	.33 x D	.50 x D
Titanium Alloy (Sol. Treated/Aged)	<380HB	XRN, HSN	110-220	0,1	0,15	0,2	0,3	0,3	0,35	0,4	0,45	0,45	.35 x D	.50 x D
Harden Steel (1.2344, 1.2379)	45-55HRC	TLN, HSN	120-220	0,2	0,25	0,3	0,4	0,5	0,5	0,6	0,6	0,6	.30 x D	.30 x D

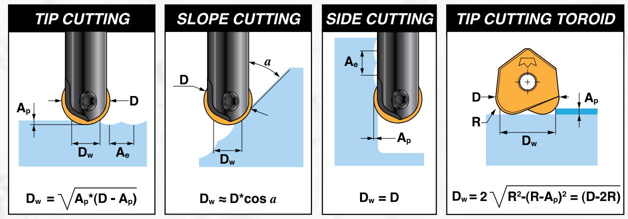
Choosing Cutting Parameters/Calculating Cutting Speed and Feed – METRIC

1. Find the Cutting Speed V_c (m/min) & Feed f_n (mm/n^{-1})

Find Vc and fn range in Table 1 or Table 2 above. Choose the average value for Vc and the lower value for feed in the range.

2. Compute the Dw

In order to compute the RPM value of the spindle it is necessary to determine the Dw which is the effective engaged tool diameter. The Dw depends on the geometry of the inserts (ball nose or toroid) and the relative position of the tool against the working piece surface. Example calculation is of Dw is presented to the right.



3. Calculate Spindle Speed N(n/min)

Use the formula: $N = (V_c * 1{,}000)/\prod * Dw$

Table 3 - Working Diameter For Ball Nose Tools (tip cutting)

ØD	Ap																		
	0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1	1,5	2	2,5	3	3,5	4	5	6	7
6	1,5	2,2	2,6	3	3,3	3,6	3,9	4,1	4,3	4,5	5,2	5,7	5,9	6,0					
8	1,8	2,5	3	3,5	3,9	4,2	4,5	4,8	5,1	5,3	6,2	6,9	7,4	7,7					
10	2	2,8	3,4	3,9	4,4	4,7	5,1	5,4	5,7	6,0	7,1	8,0	8,7	9,2	9,5				
12	2,2	3,1	3,7	4,3	4,8	5,2	5,6	6,0	6,3	6,6	7,9	8,9	9,7	10,4	10,9	11,3	11,8		
14	2,4	3,3	4,1	4,7	5,2	5,7	6,1	6,5	6,9	7,2	8,7	9,8	10,7	11,5	12,1	12,6	13,4	13,9	
16	2,5	3,6	4,3	5	5,6	6,1	6,5	7,0	7,4	7,7	9,3	10,6	11,6	12,5	13,2	13,9	14,8	15,5	15,9
20	2,8	4	4,9	5,6	6,2	6,8	7,4	7,8	8,3	8,7	10,5	12,0	13,2	14,3	15,2	16,0	17,3	18,3	19,1
25		4,5	5,4	6,3	7,0	7,7	8,2	8,8	9,3	9,8	11,9	13,6	15,0	16,2	17,3	18,3	20,0	21,4	22,4
30			6	6,9	7,7	8,4	9,1	9,7	10,2	10,8	13,1	15,0	16,6	18,0	19,3	20,4	22,4	24,0	25,4
32				7,1	7,9	8,7	9,4	10,0	10,6	11,1	13,5	15,5	17,2	18,7	20	21,2	23,2	25,0	26,5

Table 4 - Working Diameter For Toroid Tools (tip cutting)

Insert Diameter "D"	10	12	16	20	25	30	32
Depth of Cut	Dw Working Diameter (metric) Actual cutting diameter of toroid inserts						
0,5	7,3	9,3	11,9	14,3	17,8	20,4	21,6
1,0	8,5	10,5	13,3	16,0	19,6	22,5	23,8
2,0	9,7	11,7	14,9	18,0	22,0	25,2	26,6
3,0	10,0	12,0	15,8	19,2	23,4	27,0	28,5
4,0			16,0	19,8	24,3	28,3	29,9
5,0				20,0	24,9	29,2	30,8
6,0					25,0	29,7	31,5
8,.0						30,0	32,0

4. Calculate the Table Feed V_f (m/min)

Use the formula: $V_f = N * f_n * K_f$. K_f is the feed rate multiplier coefficient taking into consideration that chip load is less than theoretical value. Take the value of K_f from Table 5 or Table 6.

Table 5 - Feed Rate Multiplier For Ball Nose Inserts

Insert Diameter "D"	6	8	10	12	16	20	25	30	32
Depth of Cut	Feedrate Multiplier Factors (for working diameters Dw)								
0,5	1,8	2,0	2,2	2,5	2,8	3,2	3,5	3,8	4,0
1,0	1,2	1,5	1,6	1,8	2,0	2,2	2,5	2,6	2,8
2,0	1,0	1,1	1,2	1,3	1,5	1,6	1,8	1,9	2,0
3,0	0,0	1,0	1,1	1,1	1,2	1,4	1,5	1,6	1,7
4,0		1,0	1,0	1,1	1,2	1,2	1,3	1,4	1,5
5,0			1,0	1,0	1,1	1,1	1,2	1,3	1,4
6,0				1,0	1,0	1,1	1,2	1,2	1,3
8,0					1,0	1,0	1,1	1,1	1,2
10,0						1.,0	1,0	1,1	1,1
12,5							1,0	1,0	1,0
16,0								1,0	1,0

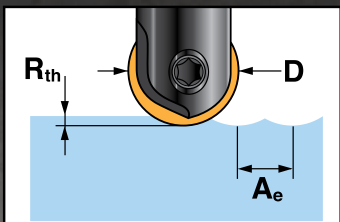
Table 6 - Feed Rate Multiplier For Toroid Tools

Insert Diameter "D"	10	12	16	20	25	30	32
Depth of Cut	Feedrate Multiplier Factors (for Toroid working diameters Dw)						
0,5	1,8	1,8	2,0	2,2	2,5	2,6	2,8
1,0	1,2	1,2	1,5	1,6	1,8	1,9	2,0
2,0	1,0	1,0	1,1	1,2	1,3	1,4	1,5
3,0	1,0	1,0	1,0	1,1	1,1	1,2	1,2
4,0			1,0	1,0	1,1	1,2	1,2
5,0				1,0	1,0	1,1	1,1
6,0					1,0	1,0	1,0
8,0						1,0	1,0

Verify Surface Roughness (R_{th})

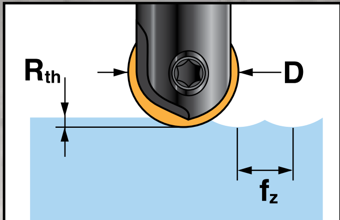
1. Decreasing the A_e and feed by half will improve surface roughness by 4 times.
2. Using $f_z = A_e$ in most cases is the best option.

Surface Roughness Step-Over



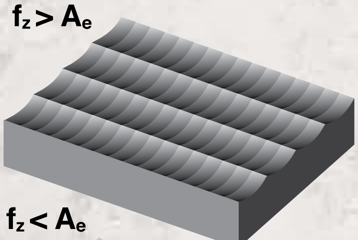
$$R_{th}, A_e = \frac{D}{2} - \sqrt{\frac{D^2 - A_e^2}{4}}$$

Surface Roughness Feed Dir

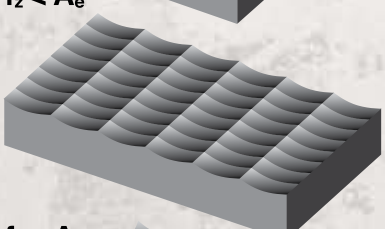


$$R_{th}, A_e = \frac{D}{2} - \sqrt{\frac{D^2 - f_z^2}{4}}$$

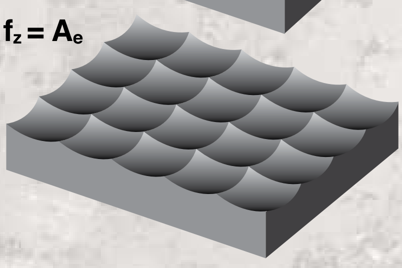
$f_z > A_e$



$f_z < A_e$



$f_z = A_e$



Metric

Normal Cutting Parameter Recommendations for BDS, BD and FB inserts

For typical mold steels (1.2311, 1.2344, 1.2711, 1.2714, etc.)

The following parameters are ONLY if cutting a flat surface with the bottom of the tool. For finish machining using only the side of the tool, use the cutting parameters for Ball Inserts.

1. Spindle speed: n(s) (min^{-1})

FB or BD Ø==>	6	8	10	12	16	20	25	32
Material hardness	Spindle speed n(s)							
< 40 HRc	8000	5600	4600	3600	2800	2250	1800	1500
40-54 HRc	7000	4900	3850	2800	2400	1960	1540	1350
55-64 HRc	6000	4200	3100	2200	1800	1500	1350	1200

2. Feed per tooth: fz (mm/tooth)

FB or BD Ø ==>	6	8	10	12	16	20	25	32
Material hardness	Feed per tooth f_z							
< 40 HRc	0.15	0.15	0.2	0.25	0.3	0.35	0.35	0.4
40-54 HRc	0.15	0.15	0.2	0.25	0.3	0.35	0.35	0.4
55-64 HRc	0.1	0.12	0.15	0.20	0.25	0.25	0.25	.25

3. Cutting depth: ap (mm)

FB or BD Ø ==>	6	8	10	12	16	20	25	32
Material hardness	Maximum Cutting depth a_p							
< 40 HRc	0.3	0.35	0.5	0.6	0.8	1.0	1.25	1.6
40-54 HRc	0.2	0.25	0.5	0.6	0.8	1.0	1.25	1.6
55-64 HRc	0.1	0.12	0.4	0.45	0.65	0.8	1.0	1.25

4. Maximum Cutting width / step-over: ae (mm)

FB or BD Ø ==>	6	8	10	12	16	20	25	32
Material hardness	Cutting width a_e							
< 40 HRc	4	6	8	9	13	17	20	26
40-54 HRc	4	6	8	9	13	17	20	26
55-64 HRc	4	6	8	9	13	17	20	26

Additional recommendations and conditions which make it necessary to modify normal cutting parameters

1. Always use climb cutting in roughing operation.
2. Enter the material with the cutter by straight ramping or helical interpolation ramping. A 2° ramp angle will achieve best results.
3. When roughing a cavity level by level (Z-level) it is best to start in the center and work outward in a square, rectangular or round spiral depending on the shape of the work piece. Use climb cutting.
4. Long tool body extension from the spindle or tool adapter will make it necessary to decrease the recommended parameters above:
   - If the tool body extension is 3 times the ball insert diameter or less, use the recommendations on page 1.
   - If the tool body extension is 4 times the ball insert diameter, multiply the cutting parameters by 0.9 (90 %).
   - If the tool body extension is 5 times the ball insert diameter, multiply the cutting parameters by 0.75 (75 %). Use a tapered tool body for additional strength.
   - If the tool body extension is 6 times the ball insert diameter, multiply the cutting parameters by 0.6 (60 %). Use a tapered tool body for additional strength.
   - If the tool extension is greater than 6 times the ball insert diameter, it is recommended to use a carbide tool body instead of a steel body for additional rigidity.
5. If the spindle speed recommended is higher than the spindle speed available on the machine, use the highest spindle speed available. You may use the same recommended feed per tooth, cutting depth and cutting width as shown above. We do not recommend reducing the feed per tooth.