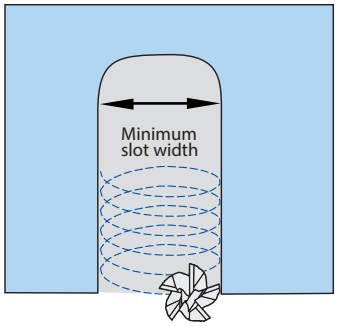
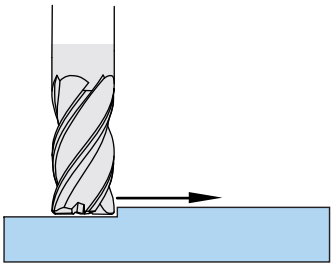


HEM Slotting Guides

The width of the desired slot will determine the number of flutes and endmill diameter that should be selected. The following guide shows the minimum slot width for each series of endmill.

Tool Series	Min. Slot Width	Max Slot Depth
XHP 7 Flute Finishers - Pg 86-90	2 x end mill diameter	Full length of cut
XHP 9 Flute Finishers - Pg 91-93	2 x end mill diameter	Full length of cut
XHP 11 Flute Finishers - Pg 94-96	2.25 x end mill diameter	Full length of cut
XHP 13 Flute Finishers Pg 98-100	2.5 x end mill diameter	Full length of cut
XHP 5 Flute Aluminum & Non Ferrous DLC Pg 26-32	1.75 x end mill diameter	Full length of cut
XHP 5 Flute Variable Index - Pg 59-78	1.75 x end mill diameter	.8 x length of cut
HP 7 Flute Finishers - Pg 84-85	2 x end mill diameter	.8 x length of cut

Speed and feed parameters for HEM slotting can be found marked as “Peripheral-HEM” in the speed and feed charts for those series.

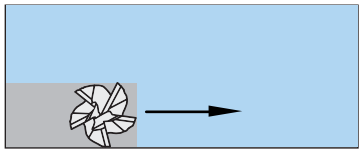


Facing

When facing, an endmill with a corner radius is suggested for the best finish. Apply the adjustments below to the Peripheral Rough values from the application guide below by tool category.

RDOC Formula

Step-over = (D - (2 x corner radius)) x .75



Tool Selection Guide:

Category 1: XHP 7, 9, 11 and 13 Flute Finishers

Category 2: XHP 5 Flute Aluminum & Non Ferrous DLC
XHP 5 Flute Variable Index
XHP 5 & 6 Flute Finishers

	Category 1					
	Rough Facing			Finish Facing		
Projection Length	SFM	IPT	ADOC	SFM	IPT	ADOC
0 to 3 x D	1.0 x chart value	.80 x chart value	.25 x D Max	1.0 x chart value	.70 x chart value	.07 x D Max
> 3 to 4 x D	1.0 x chart value	.80 x chart value	.25 x D Max	1.0 x chart value	.70 x chart value	.07 x D Max
> 4 to 5 x D	1.0 x chart value	.80 x chart value	.20 x D Max	1.0 x chart value	.70 x chart value	.05 x D Max
> 5 to 6 x D	1.0 x chart value	.80 x chart value	.20 x D Max	1.0 x chart value	.70 x chart value	.05 x D Max

	Category 2					
	Rough Facing			Finish Facing		
Projection Length	SFM	IPT	ADOC	SFM	IPT	ADOC
0 to 3 x D	1.2 x chart value	.85 x chart value	.25 x D Max	1.2 x chart value	.75 x chart value	.07 x D Max
> 3 to 4 x D	1.1 x chart value	.75 x chart value	.25 x D Max	1.1 x chart value	.65 x chart value	.07 x D Max
> 4 to 5 x D	1.0 x chart value	.65 x chart value	.25 x D Max	1.0 x chart value	.55 x chart value	.06 x D Max
> 5 to 6 x D	.9 x chart value	.55 x chart value	.25 x D Max	.9 x chart value	.45 x chart value	.05 x D Max

D = Tool Diameter

Helical Ramp to Create an Entry Hole

Using a helical ramp move to generate an entry hole is a preferred method to enter the middle of a part. The creation of the entry hole can be either a one-step or a two-step process depending on the number of flutes on the end mill. Tools with seven or fewer flutes only require one step; tools with more than seven flutes require two steps.

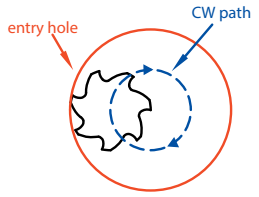
Step 1: Create helical ramp entry hole

The diameter of the starting hole will be: (tool diameter x 2) - (corner radius x 2)

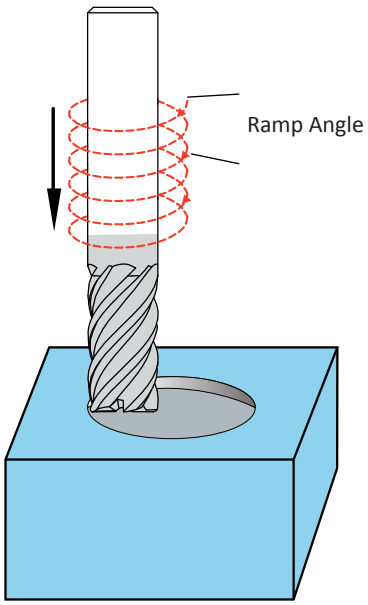
Use the following guide for speed, feed and ramp angle parameters. Note that the terms “Same as chart,” “Slotting speed in chart,” “Slotting feed in chart,” and IPT reference the data that is shown in the speed and feed charts located in each tool series section.

Tool Series	Speed	Feed Adjustment – with high-pressure coolant	Feed Adjustment – with standard flood coolant	Ramp Angle
XHP 2 Flute Aluminum and Non-Ferrous	Tools on page 16, use Slotting Speeds & Feeds on page 185.			3° - 5°
XHP 3 Flute Alum & Non-Ferrous ZRN	Tools on pages 18-22 and 24-25, use Slotting Speeds & Feeds page 186.			3° - 5°
XHP 3 Flute Roughers Alum & Non-Ferrous ZRN	Tools on page 23, use Slotting Speeds and Feeds page 187.			3° - 5°
XHP 5 Flute Alum & Non-Ferrous DLC	Tools on pages 26-32	IPT x 1.6	IPT x 1.25	3°
XHP 4 Flute Variable Index	Tools on pages 43-53 and 57, use Slotting Speeds & Feeds pages 192-193.			1° - 2.5°
XHP 5 Flute Variable Index	Tools on pages 62-78, use Slotting Speeds & Feeds pages 198-199.			1° - 2.5°
XHP 5 & 6 Flute Finishers	Tools on pages 81-83, use Slotting Speeds & Feeds page 201.			1° - 2.5°
XHP 6 Flute Finisher for Inconel	Tools on pages 79-80, use Slotting Speeds & Feeds page 202.			1° - 2.5°
HP 7 Flute Finishers	Tools on pages 84-85, use Slotting Speeds & Feeds page 201.			1° - 2.5°
XHP 7 Flute Finishers	Tools on pages 86-90	IPT x 1.6	IPT x 1.25	0.5°
XHP 9 Flute Finishers	Tools on pages 91-93	IPT x 1.6	IPT x 1.25	0.5°
XHP 11 Flute Finishers	Tools on pages 94-96	IPT x 1.6	IPT x 1.25	0.5°
XHP 13 Flute Finishers	Tools on pages 98-100	IPT x 1.6	IPT x 1.25	0.5°

IPT = Inch per tooth from the speed and feed charts
Speed = Surface feet per minute (SFM)



CW = Clockwise



Step 2: There are two common methods to enlarge an existing hole.

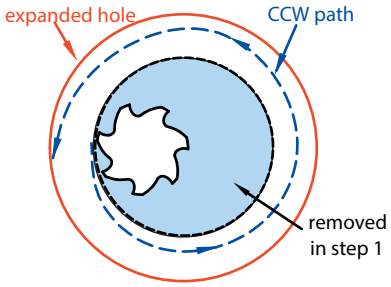
Method #1 – Expand the entry hole from inside out.

After reaching the desired entry hole depth in Step 1, and with the end mill still at depth, expand the hole outwards using the feed rate adjustment found in the chart below. Continue until the entry hole is enlarged to the expanded diameter shown at right.

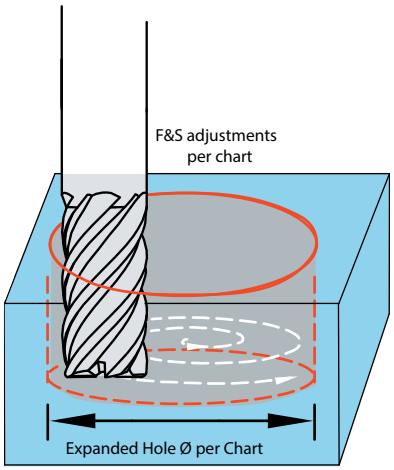
Once the expanded entry hole diameter is achieved, climb cut machining can begin at 100% of the Peripheral-HEM values in the feed and speed chart for the tool series you're using.

Tool Series	Expanded Hole ø	Feed Rate Adjustment	Step-Over Adjustment
XHP 9 Flute Finishers	3 x D	IPT x .75	RDOC x .5
XHP 11 Flute Finishers	3.75 x D	IPT x .75	RDOC x .5
XHP 13 Flute Finishers	3.75 x D	IPT x .75	RDOC x .5

D = Tool Diameter



CCW = Counter-Clockwise



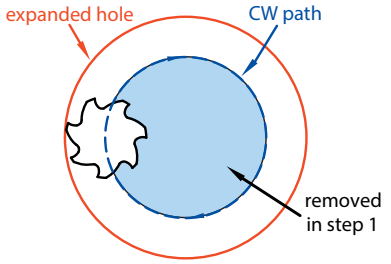
Method #2 – Expand the entry hole with a second helical ramp move.

Method B will expand the entry hole by doing a second helical ramp entry hole of a larger diameter than in Step 1. After completing Step 1, retract the end from the hole, and machine the second helical ramp entry hole using the same speed, feed and location as the first hole.

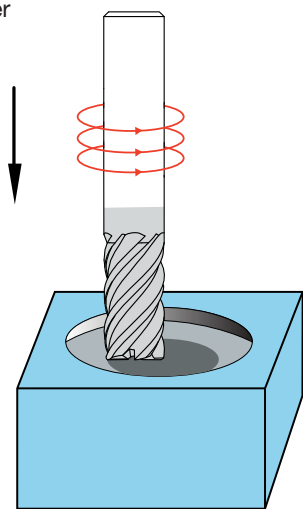
Once the expanded entry hole diameter is achieved, climb cut machining can begin at 100% of the Peripheral-HEM values in the feed and speed chart for the tool series you're using.

Tool Series	Expanded Hole ø	Feed Rate Adjustment	Ramp Angle
XHP 9 Flute Finishers	3 x D	IPT x 1.6	0.5°
XHP 11 Flute Finishers	3.75 x D	IPT x 1.6	0.5°
XHP 13 Flute Finishers	3.75 x D	IPT x 1.6	0.5°

D = Tool Diameter

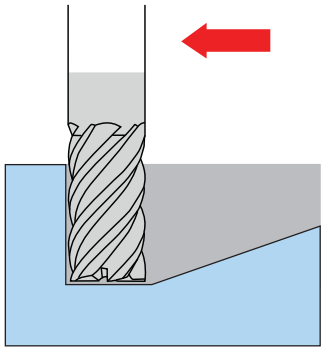


CW = Clockwise



Adjustments for Straight-Line Ramps

Straight-line ramp moves are an alternative method to enter the middle of a part. The following guide shows speed, feed and ramp angle data for different extra-high performance RedLine endmills.



Use the following guide for speed, feed and ramp angle parameters. Note that the terms “Same as chart,” “Slotting speed in chart,” and “Slotting IPT,” and “Helical ramp” reference the data shown in the speed and feed charts located in each tool series section. Not all tools are designed to allow the chip clearance required for straight-line ramping, as indicated in the guide.

Tool Series	Max Ramp Angle	SFM	Feed	Max Ramp Depth	Max Ramp Length
XHP 7 Flute Finishers	Not recommended	-	-	-	-
XHP 9 Flute Finishers	Not recommended	-	-	-	-
XHP 11 Flute Finishers	Not recommended	-	-	-	-
XHP 13 Flute Finishers	Not recommended	-	-	-	-
XHP 5 Flute Aluminum & Non Ferrous DLC	10°	Slotting speed	Slotting IPT x .65	75% of D	(.75 x D) / drop per inch
XHP 5 Flute Variable Index	2.5°	Slotting speed	Slotting IPT x .75	50% of D	(.5 x D) / drop per inch
HP 7 Flute Finishers	2.5°	Slotting speed	Slotting IPT x .75	50% of D	(.5 x D) / drop per inch
XHP 5 & 6 Flute Finishers	Not recommended	-	-	-	-
XHP 6 Flute Finisher for Inconel	Not recommended	-	-	-	-
XHP 4 Flute Variable Index	2.5°	Slotting speed	Slotting IPT x .75	50% of D	(.5 x D) / drop per inch
XHP 3 Flute Aluminum & Non Ferrous ZRN	Helical ramp x 5	Same as chart	Same as chart	100% of D	(.75 x D) / drop per inch
XHP 3 Flute Roughers Aluminum & Non Ferrous ZRN	Helical ramp x 5	Same as chart	Same as chart	100% of D	(.75 x D) / drop per inch
XHP 2 Flute Aluminum and Non Ferrous	15°	Slotting speed	Slotting IPT x .70	50% of D	(.5 x D) / drop per inch

D = Tool Diameter

Use this guide to determine the maximum ramp length. ▶

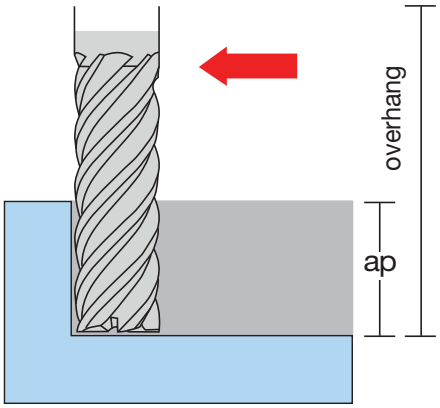
Ramp Angle	Drop (per inch)
0.5°	0.0088
1°	0.0175
2°	0.0375
2.5°	0.0438
3°	0.0525
5°	0.0875
10°	0.1750
15°	0.2625

Long Reach Application Adjustments

Adjustments must be made to reduce chatter and maximize tool life when using long length tools. The adjustments below are based on the total amount of tool projection and use the speed and feed data found in the application charts for each tool series.

Projection	SFM	Feed
> 1.25 to 3 x D	SFM x .95	IPT x .95
> 3 to 4 x D	SFM x .90	IPT x .90
> 4 to 5 x D	SFM x .80	IPT x .80
> 5 to 6 x D	SFM x .70	IPT x .70

D = Tool diameter
IPT = Inch per tooth
SFM = Surface feet per minute



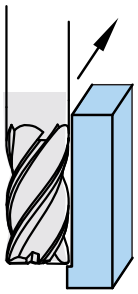
Tool Tip:
Eliminate Wall Taper When Finishing

STEP 1:
Run a climb cut finish pass using speed, feed and step-over values (RDOC) from the speed and feed charts. Adjust for tool projection if needed.

STEP 2:
Re-run the path using the same speeds and feeds but in a conventional cut direction. Simply retrace the prior finish pass; do not program to remove more stock. This skim pass, traveling in the opposite direction of the first pass, will help eliminate wall taper caused by tool deflection during the first pass.

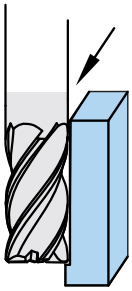
Step 1:

Climb cut
finish pass



Step 2:

Conventional cut
skim pass



Ball Nose Endmill Adjustments

Tools found on pages 16, 24-25, 56-57, 76-78.

The speeds and feeds of ball nose end mills must be adjusted to ensure proper tool life. Adjustments are based on the amount of tool engagement.

If the depth of cut (ADOC) is <50% of the tool diameter:

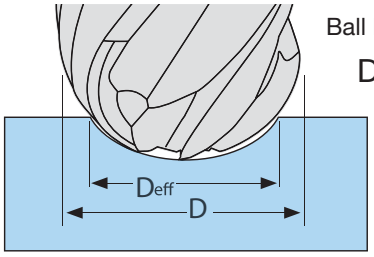
Adjustments must be made to determine the effective cutting diameter and to adjust for axial chip thinning. Follow these steps:

STEP 1:
Use speed and feed values for slot cuts from the speed and feed charts for the appropriate material and tool diameter.

Note: Make an additional adjustment using the chart below if the tool projection exceeds 2.5 x the tool diameter.

Projection	Speed Adj	Feed Adj
> 2.5 to 3 x D	SFM x .95	IPT x .95
> 3 to 4 x D	SFM x .90	IPT x .90
> 4 to 5 x D	SFM x .80	IPT x .80
> 5 to 6 x D	SFM x .70	IPT x .70

STEP 2:
Determine the effective cutting diameter (De) of the end mill based on the axial depth of cut. The effective cutting diameter will be used to make both speed and feed adjustments.



Use D when making shallow cuts with full radius

Ball Nose "Effective Diameter"

$$D = 2x \sqrt{R^2 - (R - ADOC)^2}$$

For easy reference, use the chart below.

Depth of Cut (ADOC)	1/8		1/4		3/8		1/2		3/4		1	
	Depth	De	Depth	De	Depth	De	Depth	De	Depth	De	Depth	De
10% of tool diameter	.013	.075	.025	.150	.038	.225	.050	.300	.075	.450	.100	.600
20% of tool diameter	.025	.100	.050	.200	.075	.300	.100	.400	.150	.600	.200	.800
30% of tool diameter	.038	.115	.075	.229	.113	.344	.150	.458	.225	.687	.300	.917
40% of tool diameter	.050	.123	.100	.245	.150	.367	.200	.490	.300	.73	.400	.980
50% of tool diameter	.063	.125	.125	.250	.186	.375	.250	.500	.375	.7500	.500	1.000

STEP 3:
Calculate speed based on using the effective cutting diameter. Use the standard SFM to RPM conversion formula. Substitute the effective cutting diameter (De) for the actual tool diameter (D).

RPM = (SFM x 3.82) / De

STEP 4:
Calculate the adjusted feed rate based on the effective cutting diameter and the axial chip thinning formula.

IPTadj = (D x IPT) / De

STEP 5: The new feed rate is calculated:

IPM = RPM x (Z x IPT adj)

Z = # of flutes
D = Actual tool diameter
De = Effective cutting diameter
IPM = Inches per minute
RPM = Rotations per minute
SFM = Surface feet per minute
IPT = Feed rate from chart for slot milling
IPT adj = Adjusted chip load per tooth fractional

If the axial depth of cut (ADOC) is ≥50% of the tool diameter:

- Use the speed and feed values shown for the slotting operation in the speed and feed charts for the series of end mill being used.

- If the tool projection exceeds 2.5 x the tool diameter, adjust the slotting speeds and feeds by the chart for long reach tool adjustments.

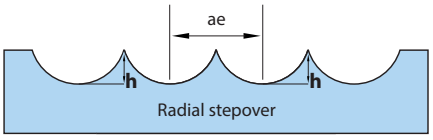
Surface Finish

Radial depth of cut (RDOC), or step-over, is based on the desired finish. The lighter the step-over, the lower the scallop height (material left uncut by the radius of the tool), and the better the finish.

This chart calculates approximate scallop height using the following formula:

Tool Diameter	Step-over % of OD	Step-over Actual	Approx Scallop Height
1/8	10%	.013	.0003
	20%	.025	.0013
	30%	.038	.0028
1/4	10%	.025	.0006
	20%	.050	.0025
	30%	.075	.0056
3/8	10%	.038	.0009
	20%	.075	.0038
	30%	.113	.0084
1/2	10%	.050	.0013
	20%	.100	.0050
	30%	.150	.0113
3/4	10%	.075	.0019
	20%	.150	.0075
	30%	.225	.0169
1	10%	.100	.0025
	20%	.200	.0100
	30%	.300	.0225

$h \sim (ae^2) / (8R)$



h = Scallop height
ae = Radial step-over
R = Radius of end mill
(tool diameter x .5)