Product Information Report
End Mills

Overview

An end mill is a cutting tool used in industrial machining applications. An end mill bit can generally cut in all directions, though some cannot cut axially. End mills are used in milling applications such as profile milling, tracer milling, face milling and plunging.

What are the Different Types of End Mills?

A broad category of end mills exists, such as single end, double end, center-cutting, non center-cutting, ball nose and roughing.

Each category may be further divided by application and special geometry. End mills are sold in both imperial and metric sizes, depending on the country of origin.

2-Flute

4-Flute

4-Flute Ball Nose

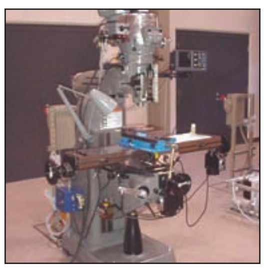
4-Flute Rougher

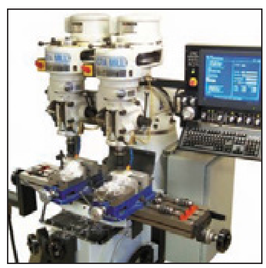
2-Flute Double End

3-Flute Double End

Applications

End mills are used in machining applications such as profile milling and face milling. Depending on the material being machined, different tool types and geometries may be used.





Remember

- Always wear eye protection when using any cutting tool

- Use a good quality cutting fluid

- Store end mills properly when not in use

What are End Mills Made From?

Lawson carries end mills made from two different materials: M42 cobalt and carbide. Cobalt end mills are ideal for any milling application. Carbide end mills offer higher speed and feed rates, but should only be used on stable, high-end CNC machines.

M42 Cobalt

2-Flute Single End Center-Cutting
4-Flute Single End Center-Cutting
4-Flute Single End Ball Nose
4-Flute Single End Coarse Rougher
2-Flute Double End Center-Cutting
Supertanium® II Triple-Cut (3-Flute) Center-Cutting

Carbide

2-Flute Single End Center-Cutting
4-Flute Single End Center-Cutting
4-Flute Single End Ball Nose
2-Flute Single End Aluminum Cutting

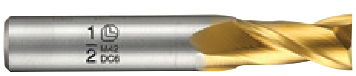
Coatings

- **TiN – Titanium Nitride (a yellowish coating)**
  The ideal coating for a wide range of applications, materials and cutting conditions where tool life and higher feeds and speeds are operating objectives. The high lubricity of TiN facilitates chip flow, prevents buildup and reduces cutting forces and temperatures. TiN reduces wear and improves chip formation to extend tool life. It is a good choice for machining iron-based materials, die molds, and for components requiring resistance to abrasive and adhesive wear. This coating is available on Lawson's 2- and 4-flute M42 Cobalt End Mills.
- **TiCN – Titanium Carbonitride (a bluish-grey coating)**
  This is a coating with exceptionally high hardness and low coefficient of friction which provides excellent wear resistance. TiCN performs well when cutting alloy steels, stainless steels, and in high-speed cutting where moderate temperatures are generated at the cutting edge. TiCN coating provides an excellent surface quality on machined components. It is also excellent for applications which require high feed and speed rates. This coating is available on Lawson's 2-Flute Carbide End Mills for aluminum cutting.
- **Nitride/Gold Oxide (a black and gold coating)**
  Combines the advantages of the lubricity of oxide with the abrasion resistance of nitriding. Recommended for abrasive ferrous applications. Not recommended for soft materials such as aluminum, magnesium or similar non-ferrous applications. This coating is available on Lawson's Supertanium® II Triple-Cut (3-Flute) Center-Cutting End Mills.

End Mill Selection

Utilize the shortest available tool possible for the application with the largest diameter permissible and the shortest flute length that depth of cut allows. The point style of the end mill is also important in determining what end mill should be used.

Flutes



2-Flute End Mills allow maximum chip volume and are used for plunge milling, roughing of slots, or peripheral milling. These multipurpose tools allow high feed rates where part finish and dimensional accuracy are not critical. When plunge cutting, it is recommended to use approximately 25% to 50% of the feed per tooth.

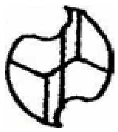
3-Flute End Mills are more rigid and have less cut interruption than 2-flute designs. They have a higher chip volume area than 4-flute designs for higher metal removal rates. 3-flute end mills have all the machining capabilities of 2-flute end mills, and are ideal for slotting applications. Improved part finish and dimensional accuracy can be achieved in a wider range of materials than with 2-flute types.

4-Flute End Mills are stronger than either 2- or 3-flute designs. The added rigidity allows higher metal removal rates with minimum deflection. Improved workpiece finishes and dimensional accuracy can be achieved. Limited chip volume area restricts stock removal rates and deep plunge cutting is not recommended. The 4-flute design is commonly used for finishing operations for the best surface finish.

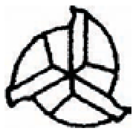
End Mill Selection (cont.)

Point Styles

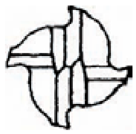
When choosing an end mill, the point style is just as important as the number of flutes or the tool length. Lawson offers 2-, 3- and 4-Flute Center-Cutting, 4-Flute Non Center-Cutting and 4-Flute Ball Nose end mill styles.



2-Flute



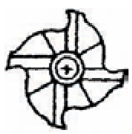
3-Flute



4-Flute

Center-Cutting

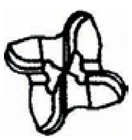
At least one of the end teeth extends to the center of the end mill. This allows the end mill to plunge directly into the material because material is removed from the entire diameter.



4-Flute

Non Center-Cutting

None of the end teeth extend to the center of the end mill. There is a female center on the tip of the end mill where no material would be removed if it was used in plunge milling applications. These end mills are designed primarily for peripheral milling.



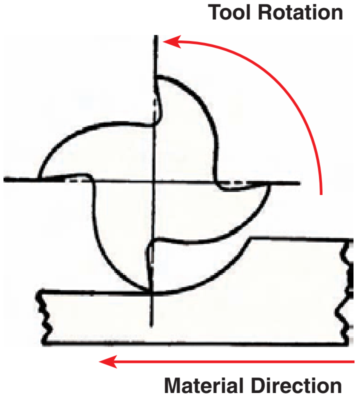
4-Flute

Ball Nose

Ball nose end mills have a rounded point for radiused cutting applications. They are commonly used to machine the rounded grooves for o-rings. Ball nose end mills can be used to plunge cut, but the cut must be combined with the action of a slotting cut. It can not be plunged straight down into the material.

Types of Milling

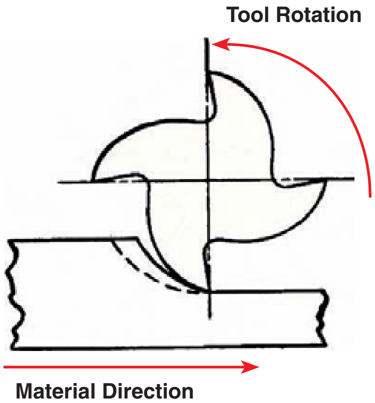
Conventional Milling



The end mill revolves opposite to the direction of the table feed. The width of the chip starts at zero and increases to a maximum at the end of the cut. This type of milling can lead to accelerated wear.

Note: Make sure workpiece is secure before beginning milling operation.

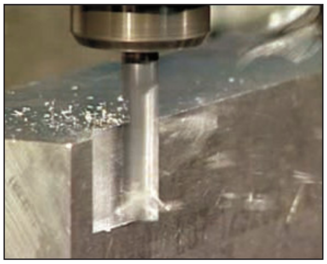
Climb Milling



The end mill revolves in the same direction as the table feed. The tooth meets the workpiece at the top of the cut, producing the thickest part of the chip first.

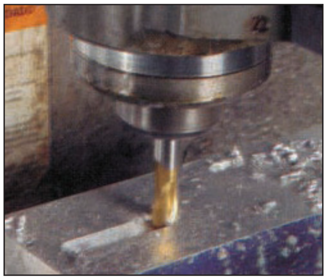
Note: Make sure workpiece is secure before beginning milling operation.

Peripheral Milling



The surface is milled parallel to the end mill axis. Peripheral milling can be either conventional or climb milling.

Plunge Milling/Slotting

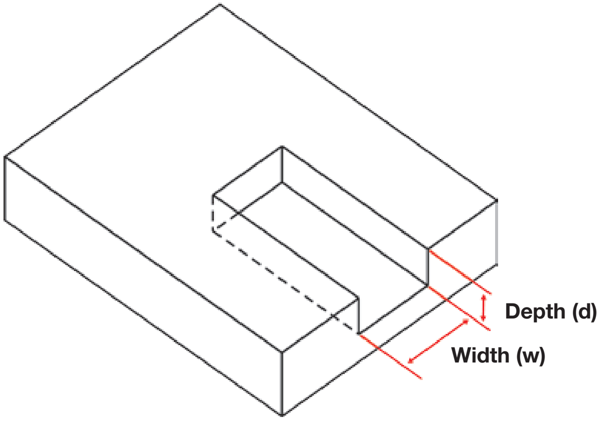


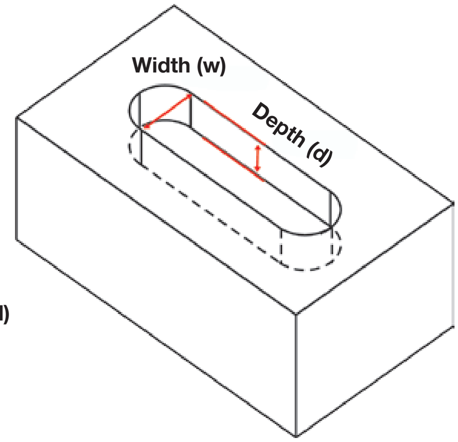
Plunge Milling: The direct movement between the workpiece and the center line of the end mill when the end mill sinks directly into the workpiece. The action of drilling a hole with a center-cutting end mill.

Slotting: All slotting applications are a combination of conventional milling and climb milling.

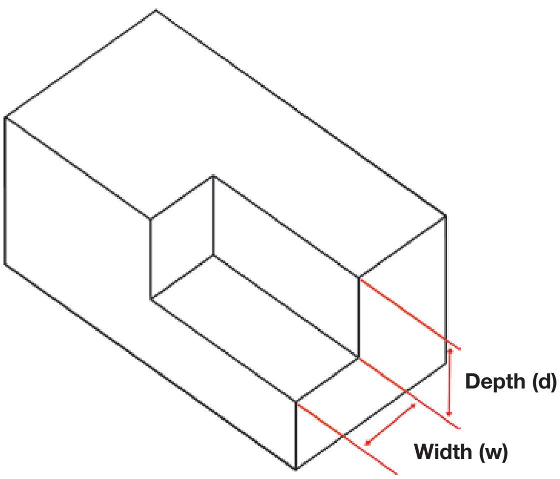
Types of Milling and End Mill Selection

Slotting

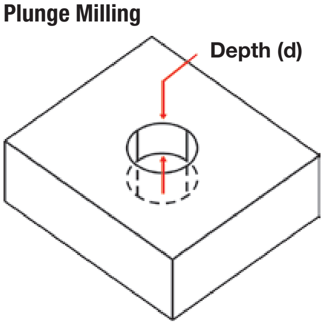




Peripheral Milling



Plunge Milling



Cutting Conditions	End Mill	2-Flute	3-Flute	4-Flute
Slotting	Free-Cutting Materials $d \leq 0.25D$	✔	☆	✔
	Free-Cutting Materials $d \leq 0.5D$	☆	☆	✘
	Free-Cutting Materials $d > 0.5D$	✔	✔	✘
	Tough Materials $d \leq 0.25D$	✔	☆	☆
	Tough Materials $d \leq 0.5D$	✔	☆	☆
	Tough Materials $d > 0.5D$	✔	✔	✔
Peripheral Milling	$w \times d \leq 0.15D_2$	✔	✔	☆
	$w \times d \leq 0.45D_2$	✔	✔	✘
	$w \times d > 0.45D_2$	Roughing End Mill		
Plunge Milling	$d \leq 0.25D$	✔	☆	✔
	$d \leq 0.5D$	☆	✔	✘
	$d \leq 1.0D$	☆	✔	✘
	$d > 1.0D$	Drill		

☆ *Best* ✔ *Useable* ✘ *Do Not Use*

Proper Speeds and Feeds

Before the machining operation can be performed, the proper speeds and feeds need to be selected. Using the proper speeds and feeds for the material being machined extends tool life and the appearance of the machined face.

Lawson's Cobalt and Carbide End Mills each have their own speed and feed charts. Roughing End Mills speed and feed chart is listed below.

Since all speed and feed charts give a range for the speed to use, here are some general guidelines to help determine the proper speed to use.

Use lower speed ranges for:
Hard Materials
Tough Materials
Abrasive Materials
Heavy Cuts
Minimum Tool Wear
Maximum Cutter Life

Use higher speed ranges for:
Softer Materials
Better Finishes
Smaller Diameter Cutters
Light Cuts
Frail Workpieces or Setups
Hand Feed Operations
Maximum Production Rates
Non-metallics

Reminder: When using a carbide end mill, the cutting fluid must either be continuously flowing over the end mill or no cutting fluid should be used. Intermittent cooling of a carbide tool can cause it to chip.

Roughing End Mills Speed and Feed Data

Material	SFM	Chip Load Per Tooth			
		1/8"	1/4"	1/2"	1"
Aluminum Alloys	125 – 250	0.0010	0.0020	0.0025	0.0030
Magnesium	125 – 250	0.0010	0.0020	0.0025	0.0030
Copper	75 – 100	0.0008	0.0015	0.0030	0.0060
Brass	85 – 110	0.0008	0.0015	0.0030	0.0060
Bronze	75 – 100	0.0008	0.0015	0.0030	0.0060
Cast Iron	100 – 125	0.0008	0.0015	0.0025	0.0050
Cast Steel	75 – 100	0.0008	0.0015	0.0025	0.0050
Malleable Iron	80 – 120	0.0008	0.0015	0.0025	0.0050
Stainless Steel					
Free Machining	75 – 90	0.0005	0.0007	0.0012	0.0020
Other	50 – 75	0.0005	0.0007	0.0012	0.0020
Steel					
Annealed	100 – 125	0.0010	0.0020	0.0040	0.0060
Rc 18-24	75 – 100	0.0070	0.0012	0.0030	0.0050
Rc 24-37	40 – 75	0.0005	0.0010	0.0020	0.0040
Titanium					
Up to Rc 30	40 – 75	0.0005	0.0012	0.0025	0.0050
Rc 30+	20 – 25	0.0005	0.0010	0.0020	0.0035
High-Temperature Alloys					
Austenitic	12 – 20	*	0.0007	0.0015	0.0030
Ferritic	50 – 75	0.0004	0.0007	0.0020	0.0050
Nickel Base	20 – 25	0.0004	0.0007	0.0015	0.0030
Cobalt Base	8 – 15	*	0.0007	0.0015	0.0030

All speeds and feeds are suggested starting points. They may be increased or decreased depending on machine condition, depth of cut, finish required, etc.

List of Symbols

F = Number of Flutes
D = Cutter Diameter
RPM = Revolutions per Minute
SFM = Surface Feet per Minute
IPM = Inches per Minute Feed Rate
IPR = Inches per Revolution Feed Rate

Machining Formulas

$$\text{SFM} = 0.262 \times \text{D} \times \text{RPM}$$

$$\text{RPM} = \frac{3.82 \times \text{SFM}}{\text{D}}$$

$$\text{IPR} = \frac{\text{IPM}}{\text{RPM}} \quad \text{or} \quad \text{Chip Load} \times \text{F}$$

$$\text{IPM} = \text{RPM} \times \text{IPR}$$

$$\text{Chip Load} = \frac{\text{IPM}}{\text{RPM} \times \text{F}} \quad \text{or} \quad \frac{\text{IPR}}{\text{F}}$$

Cobalt HSS End Mill Speed and Feed Data – Applications in Various Materials

Material	Heat-Resistant Cobalt Base Alloys, High Tensile Steels (50-55 C)		Heat-Resistant Austenitic Alloys, High Tensile Steels (46-50 C)		Heat-Resistant Nickel Base Alloys, High Strength Stainless Steels, High Strength Titanium Alloys		High Strength Stainless Steels, High Tensile Steels (40-60 C) Medium Strength Titanium Alloys		Heat-Resistant Ferritic Base Alloys Medium Strength Stainless Steels Unalloyed Titanium Tool Steels (30-40 C)		Machine Steel, Hard Brass and Bronze, Electrolytic Copper Mild Steel Forgings (20-30 C)		Cast Iron, Mild Steel, Half-Hard Brass and Bronze		Brass, Bronze, Alloyed Aluminum, Abrasive Plastics		Aluminum, Plastics, Wood	
	Speed 5-10 SFM	Feed	Speed 10-15 SFM	Feed	Speed 15-20S FM	Feed	Speed 20-40 SFM	Feed	Speed 40-60 SFM	Feed	Speed 60-80 SFM	Feed	Speed 80-100 SFM	Feed	Speed 100-200 SFM	Feed	Speed 200-600 SFM	Feed
End Mill Dia.	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth	RPM	Chip Load Per Tooth
1/16	–	–	–	–	–	–	1,222-2,444	.0002-.0005	2,444-3,667	.0002-.005	3,667-4,888	.0002-.0005	4,888-6,111	.0002-.0005	6,111-12,222	.0002-.0005	12,222 Up	.0002-.0005
3/32	–	–	–	–	611-815	.0002-.0005	815-1,629	.0002-.0005	1,629-2,750	.0002-.005	2,750-3,259	.0002-.0005	3,259-4,073	.0002-.0005	4,073-8,146	.0002-.0005	8,146 Up	.0002-.0005
1/8	–	–	–	–	456-611	.0002-.0005	611-1,222	.0002-.0005	1,222-1,833	.0002-.005	1,833-2,440	.0002-.001	2,440-3,056	.0002-.001	3,056-6,112	.0002-.001	6,112 Up	.0002-.001
3/16	–	–	204-306	.0002-.0005	306-407	.0002-.0005	407-815	.0002-.0005	815-1,222	.0002-.005	1,222-1,625	.0002-.001	1,625-2,037	.0002-.001	2,037-4,074	.0002-.001	4,074-12,222	.0002-.001
1/4	76-153	.0002-.001	153-230	.0002-.001	229-306	.0002-.001	306-611	.0002-.001	611-917	.0002-.001	917-1,222	.0005-.002	1,222-1,528	.0005-.002	1,528-3,056	.0005-.002	3,056-9,168	.0005-.002
5/16	61-122	.0002-.001	122-183	.0002-.001	183-244	.0002-.001	244-489	.0002-.001	489-733	.0002-.001	733-978	.0005-.002	978-1,222	.0005-.002	1,222-2,444	.0005-.002	2,444-7,332	.0005-.002
3/8	51-102	.0002-.001	102-153	.0002-.001	153-203	.0002-.001	203-407	.0005-.002	406-611	.0005-.002	611-815	.001-.003	815-1,019	.001-.003	1,019-2,038	.0005-.003	2,038-6,114	.0005-.002
7/16	44-88	.0005-.001	88-132	.0005-.001	131-175	.0005-.002	175-349	.0005-.002	349-524	.0005-.002	524-698	.001-.003	698-873	.001-.003	873-1,746	.0005-.003	1,746-5,238	.0005-.002
1/2	38-76	.0005-.001	76-115	.0005-.001	115-153	.0005-.002	153-306	.0005-.003	306-458	.001-.003	458-611	.001-.003	611-764	.001-.003	764-1,528	.0005-.003	1,528-4,584	.0005-.002
9/16	34-68	.0005-.002	68-104	.0005-.002	104-136	.0005-.002	136-272	.0005-.003	272-412	.001-.003	412-543	.001-.004	543-678	.001-.004	678-1,356	.0005-.004	1,356-4,071	.0005-.003
5/8	31-61	.0005-.002	61-92	.0005-.002	92-122	.0005-.002	122-244	.001-.004	244-367	.001-.004	367-489	.001-.004	489-611	.001-.004	611-1,222	.0005-.004	1,222-3,666	.0005-.003
11/16	28-56	.0005-.002	56-84	.0005-.002	84-111	.0005-.002	111-222	.001-.004	222-337	.001-.004	337-444	.001-.004	444-555	.001-.004	555-1,110	.0005-.004	1,110-3,330	.0005-.003
3/4	26-51	.0005-.002	51-76	.0005-.002	76-102	.001-.004	102-203	.001-.004	203-306	.001-.004	306-407	.001-.004	407-509	.002-.006	509-1,018	.001-.006	1,018-3,054	.001-.004
13/16	24-47	.001-.003	47-71	.001-.003	71-94	.001-.004	94-189	.001-.004	189-284	.001-.004	284-379	.002-.006	379-469	.002-.006	469-938	.001-.006	938-2,814	.001-.004
7/8	22-44	.001-.003	44-65	.001-.003	65-87	.001-.004	87-175	.001-.004	175-262	.002-.006	262-349	.002-.006	349-436	.002-.006	436-872	.001-.006	872-2,616	.001-.004
15/16	20-40	.001-.003	40-62	.001-.003	62-81	.001-.004	81-163	.001-.004	163-246	.002-.006	246-326	.002-.006	326-407	.002-.006	407-814	.001-.006	514-2,442	.001-.004
1	19-38	.001-.003	38-58	.001-.003	58-76	.001-.004	76-153	.002-.006	153-229	.002-.006	229-306	.002-.006	306-382	.002-.006	382-764	.002 Up	764-2,292	.002 Up
1-1/8	34	.0015-.004	34-51	.0015-.004	51-68	.0015-.005	68-136	.002-.006	136-204	.002-.006	204-272	.002-.006	272-340	.003 Up	340-680	.002 Up	680-2,040	.002 Up
1-1/4	31	.0015-.004	31-46	.0015-.004	46-61	.0015-.005	61-122	.002-.006	122-183	.002-.006	183-244	.003 Up	244-306	.003 Up	306-612	.002 Up	612-1,836	.002 Up
1-3/8	28	.0015-.004	28-42	.0015-.004	42-55	.0015-.005	55-111	.002-.006	111-167	.003 Up	167-222	.003 Up	222-278	.003 Up	278-556	.002 Up	556-1,668	.002 Up
1-1/2	26	.0015-.004	26-38	.0015-.004	38-51	.002 Up	51-102	.003 Up	102-153	.003 Up	153-204	.003 Up	204-255	.003 Up	255-510	.003 Up	510-1,530	.002 Up
1-5/8	24	.002 Up	35	.002 Up	35-47	.002 Up	47-94	.003 Up	94-141	.003 Up	141-188	.003 Up	188-235	.003 Up	235-470	.003 Up	470-1,410	.002 Up
1-3/4	22	.002 Up	32	.002 Up	32-43	.002 Up	43-87	.003 Up	87-131	.003 Up	131-175	.003 Up	175-218	.003 Up	218-436	.003 Up	436-1,308	.002 Up
1-7/8	20	.002 Up	30	.002 Up	30-40	.003 Up	40-81	.003 Up	81-122	.003 Up	122-163	.003 Up	163-204	.003 Up	204-408	.003 Up	408-1,224	.003 Up
2	19	.002 Up	29	.003 Up	29-38	.003 Up	38-76	.003 Up	76-115	.003 Up	115-153	.003 Up	153-191	.003 Up	191-382	.003 Up	382-1,146	.003 Up
2-1/8	18	.003 Up	28	.003 Up	36	.003 Up	36-72	.003 Up	72-108	.003 Up	108-144	.003 Up	144-179	.003 Up	179-358	.003 Up	358-1,074	.003 Up
2-1/4	17	.003 Up	26	.003 Up	34	.003 Up	34-68	.003 Up	68-102	.003 Up	103-136	.003 Up	136-170	.003 Up	170-340	.003 Up	340-1,020	.003 Up
2-3/8	16	.003 Up	25	.003 Up	32	.003 Up	32-64	.003 Up	64-97	.003 Up	97-128	.003 Up	128-161	.003 Up	161-322	.003 Up	322-966	.003 Up
2-1/2	15	.003 Up	23	.003 Up	30	.003 Up	30-61	.003 Up	61-92	.003 Up	92-122	.003 Up	122-153	.003 Up	153-306	.003 Up	306-918	.003 Up
2-5/8	15	.003 Up	22	.003 Up	29	.003 Up	29-58	.003 Up	58-88	.003 Up	88-116	.003 Up	116-145	.003 Up	145-290	.003 Up	290-870	.003 Up
2-3/4	14	.003 Up	21	.003 Up	28	.003 Up	28-56	.003 Up	56-83	.003 Up	83-111	.003 Up	111-139	.003 Up	139-278	.003 Up	278-834	.003 Up
2-7/8	14	.003 Up	20	.003 Up	27	.003 Up	27-53	.003 Up	53-80	.003 Up	80-106	.003 Up	106-132	.003 Up	132-264	.003 Up	264-792	.003 Up
3	13	.003 Up	19	.003 Up	26	.003 Up	26-51	.003 Up	51-76	.003 Up	76-102	.003 Up	102-127	.003 Up	127-154	.003 Up	254-762	.003 Up

NOTE: All speed and feed data are suggested starting points. They may be increased or decreased depending on machine condition, depth of cut, finish required, coolant, etc. When using TiN-coated M42 cobalt cutters, increase speed rate by 25%.

Milling Problems	Possible Cause	Possible Solution
Chip Packing	Cutting too much material	Adjust feed or speed
	Not enough chip room	Use end mill with fewer flutes
	Not enough coolant	Apply more coolant, use air pressure
Rough Surface Finish	Feed too fast	Slow down to correct feed
	Slow speed	Use higher speed
	Too much wear	Regrind earlier stage
	Chip biting	Cut less amount per pass
	No tooth end concavity	Add margin (touch primary with oilstone)
Burr	Too much wear on primary relief	Regrind sooner
	Incorrect condition	Correct milling condition
	Improper cutting angle	Change to correct cutting angle
No Dimensional Accuracy	Condition too tough	Change to easier condition
	Lack of accuracy (machine and holder)	Repair machine or holder
	Not enough rigidity (machine and holder)	Change machine or holder condition
	Not sufficient number of flutes	Use end mill with greater number of flutes
No Perpendicular Side	Feed too fast	Slow down to correct feed
	Too great a cutting amount	Reduce cutting amount
	Too long a flute or overall length	Use proper tool length, hold shank deeper
	Insufficient number of flutes	Use end mill with greater number of flutes
Chipping	Feed too fast	Slow down to proper feed
	Feed too fast on first cut	Slow down on first bite
	Not enough rigidity of tool and holder	Change rigid machine tool or holder
	Loose holder	Tighten tool holder
	Loose holder (workpiece)	Tighten workpiece fixture
	Lack of rigidity (tool)	Use shortest end mill available, hold shank deeper, try down cut
	Teeth too sharp	Change to lower cutting angle, primary relief
Wear	Speed too fast	Slow down, use more coolant
	Hard material	Use higher grade tool material, add surface treatment
	Biting chips	Change feed speed to change chip size or clear chips with coolant or air pressure
	Improper feed speed (too slow)	Increase feed speed, try down cut
	Improper cutting angle	Change to correct cutting angle
	Primary relief angle too low	Change to larger relief angle
Breakage	Feed too fast	Slow down feed
	Cutting amount too large	Adjust to smaller cutting amounts per tooth
	Flute length or overall length too large	Hold shank deeper, use shorter end mill
	Too much wear	Regrind at earlier stage
Chattering	Feed and speed too fast	Correct speed and feed
	Not enough rigidity (machine and holder)	Use better machine tool or holder or change condition
	Relief angle too large	Change to smaller relief angle, add margin (tough primary with oil stone)
	Loose holder (workpiece)	Hold workpiece tighter
	Cutting too deep	Correct to smaller cutting depth
	Flute length or overall length too large	Hold shank deeper, use shorter end mill or try down cut
Short Tool Life (Dull Teeth)	Cutting friction too high	Regrind at earlier stage
	Tough work material	Select premium tool
	Improper cutting angle	Change cutting angle and primary