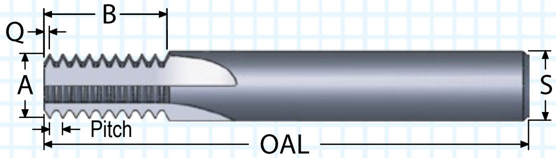
# THREAD MILLS - METRIC
## STRAIGHT FLUTE - CARBIDE
### FULL PROFILE




- Short length-of-cut for ideal length-to-diameter ratio
- Polished flute face for optimum performance
- Made with premium submicron grade carbide
- Internal crest cutting design for strongest possible tool

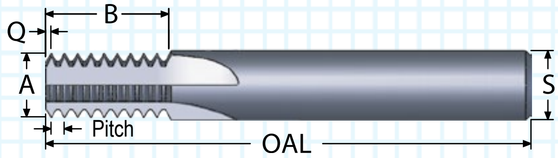
## 3 FLUTE

| MIN ID THREAD/ PITCH* | "A" TOOL DIA. | "B" LENGTH OF CUT | "Q" LENGTH | "S" SHANK DIA. | OAL | ORDER # | |
|---|---|---|---|---|---|---|---|
| | | | | | | UNCOATED | ALTiN+ |
| | | | | | | INTERNAL THREADS ONLY | |
| M3-.5 | 0.090 | 0.264 | 0.009 | 0.250 | 2.50 | TM3-.5MM | TM3-.5MM-A |
| M3-.5 | 0.090 | 0.185 | 0.009 | 0.250 | 2.50 | TM3-.5MM-S | TM3-.5MM-SA |
| M3.5-.6 | 0.090 | 0.269 | 0.011 | 0.250 | 2.50 | TM3.5-.6MM | TM3.5-.6MM-A |
| M3.5-.6 | 0.090 | 0.175 | 0.011 | 0.250 | 2.50 | TM3.5-.6MM-S | TM3.5-.6MM-SA |
| M4-.5 | 0.110 | 0.323 | 0.009 | 0.250 | 2.50 | TM4-.5MM | TM4-.5MM-A |
| M4-.5 | 0.110 | 0.224 | 0.009 | 0.250 | 2.50 | TM4-.5MM-S | TM4-.5MM-SA |
| M4-.7 | 0.110 | 0.342 | 0.012 | 0.250 | 2.50 | TM4-.7MM | TM4-.7MM-A |
| M4-.7 | 0.110 | 0.231 | 0.012 | 0.250 | 2.50 | TM4-.7MM-S | TM4-.7MM-SA |
| M4.5-.75 | 0.125 | 0.337 | 0.013 | 0.250 | 2.50 | TM4.5-.75MM | TM4.5-.75MM-A |
| M4.5-.75 | 0.125 | 0.219 | 0.013 | 0.250 | 2.50 | TM4.5-.75MM-S | TM4.5-.75MM-SA |
| M5-.7 | 0.140 | 0.397 | 0.012 | 0.250 | 2.50 | TM5-.7MM | TM5-.7MM-A |
| M5-.7 | 0.140 | 0.259 | 0.012 | 0.250 | 2.50 | TM5-.7MM-S | TM5-.7MM-SA |
| M5-.8 | 0.140 | 0.391 | 0.014 | 0.250 | 2.50 | TM5-.8MM | TM5-.8MM-A |
| M5-.8 | 0.140 | 0.265 | 0.014 | 0.250 | 2.50 | TM5-.8MM-S | TM5-.8MM-SA |
| M6-.5 | 0.170 | 0.520 | 0.009 | 0.250 | 2.50 | TM6-.5MM | TM6-.5MM-A |
| M6-.5 | 0.170 | 0.382 | 0.009 | 0.250 | 2.50 | TM6-.5MM-S | TM6-.5MM-SA |
| M6-.75 | 0.170 | 0.543 | 0.013 | 0.250 | 2.50 | TM6-.75MM | TM6-.75MM-A |
| M6-.75 | 0.170 | 0.366 | 0.013 | 0.250 | 2.50 | TM6-.75MM-S | TM6-.75MM-SA |
| M6-1 | 0.170 | 0.528 | 0.018 | 0.250 | 2.50 | TM6-1MM | TM6-1MM-A |
| M6-1 | 0.170 | 0.370 | 0.018 | 0.250 | 2.50 | TM6-1MM-S | TM6-1MM-SA |
| M6-1.25 | 0.170 | 0.561 | 0.022 | 0.250 | 2.50 | TM6-1.25MM | TM6-1.25MM-A |
| M6-1.25 | 0.170 | 0.364 | 0.022 | 0.250 | 2.50 | TM6-1.25MM-S | TM6-1.25MM-SA |
| M8-.75 | 0.235 | 0.662 | 0.013 | 0.250 | 2.50 | TM8-.75MM | TM8-.75MM-A |
| M8-1 | 0.235 | 0.685 | 0.018 | 0.250 | 2.50 | TM8-1MM | TM8-1MM-A |
| M8-1.25 | 0.235 | 0.660 | 0.022 | 0.250 | 2.50 | TM8-1.25MM | TM8-1.25MM-A |

***Thread mills can cut any larger size internal thread of the same pitch**


Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295

# THREAD MILLS - METRIC
## STRAIGHT FLUTE - CARBIDE
### FULL PROFILE



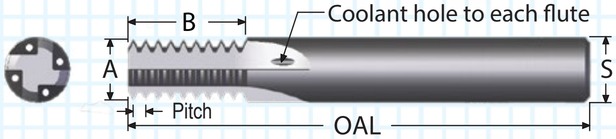

- Polished flute face for optimum performance
- Made with premium submicron grade carbide
- Internal crest cutting design for strongest possible tool

## 4 FLUTE

| MIN ID THREAD / PITCH* | "A" TOOL DIA. | "B" LENGTH OF CUT | "Q" LENGTH | "S" SHANK DIA. | OAL | ORDER # | |
|---|---|---|---|---|---|---|---|
| | | | | | | UNCOATED | ALTiN+ |
| | | | | | | INTERNAL THREADS ONLY | |
| M10-1 | 0.290 | 0.803 | 0.018 | 0.3125 | 3.50 | TM10-1MM | TM10-1MM-A |
| M10-1.5 | 0.290 | 0.792 | 0.027 | 0.3125 | 3.50 | TM10-1.5MM | TM10-1.5MM-A |
| M12-1.25 | 0.345 | 0.807 | 0.022 | 0.375 | 3.50 | TM12-1.25MM | TM12-1.25MM-A |
| M12-1.5 | 0.345 | 0.792 | 0.027 | 0.375 | 3.50 | TM12-1.5MM | TM12-1.5MM-A |
| M12-1.75 | 0.345 | 0.787 | 0.031 | 0.375 | 3.50 | TM12-1.75MM | TM12-1.75MM-A |
| M12-1 | 0.400 | 1.079 | 0.018 | 0.500 | 3.50 | TM12-1MM | TM12-1MM-A |
| M14-1.25 | 0.450 | 1.103 | 0.022 | 0.500 | 3.50 | TM14-1.25MM | TM14-1.25MM-A |
| M14-1.5 | 0.450 | 1.087 | 0.027 | 0.500 | 3.50 | TM14-1.5MM | TM14-1.5MM-A |
| M14-1.75 | 0.450 | 1.134 | 0.031 | 0.500 | 3.50 | TM14-1.75MM | TM14-1.75MM-A |
| M14-2 | 0.450 | 1.134 | 0.035 | 0.500 | 3.50 | TM14-2MM | TM14-2MM-A |
| M16-2.5 | 0.450 | 1.122 | 0.044 | 0.500 | 3.50 | TM16-2.5MM | TM16-2.5MM-A |

***Thread mills can cut any larger size internal thread of the same pitch**

# METRIC THREAD MILLS
## COOLANT THROUGH - SOLID CARBIDE
### FULL PROFILE

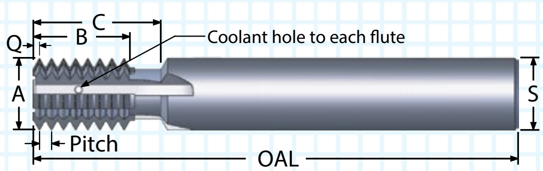



- ALTiN+ coating for higher cutting speed
- Coolant to each flute
- Made with premium submicron grade carbide
- Internal Threads Only

| MIN IN THREAD/ PITCH* | "A" TOOL DIA. | "B" LENGTH OF CUT | "Q" LENGTH | "S" SHANK DIA. | OAL | FLUTE | ORDER # | |
|---|---|---|---|---|---|---|---|---|
| | | | | | | | UNCOATED | COATED |
| | | | | | | | INTERNAL THREADS ONLY | |
| M3-.5 | 0.090 | 0.264 | 0.009 | 0.250 | 2.50 | 3 | TMC3-.5MM | TMC3-.5MM-A |
| M4-.5 | 0.110 | 0.323 | 0.009 | 0.250 | 2.50 | 3 | TMC4-.5MM | TMC4-.5MM-A |
| M4-.7 | 0.110 | 0.342 | 0.012 | 0.250 | 2.50 | 3 | TMC4-.7MM | TMC4-.7MM-A |
| M4.5-.75 | 0.125 | 0.337 | 0.013 | 0.250 | 2.50 | 3 | TMC4.5-.75MM | TMC4.5-.75MM-A |
| M5-.8 | 0.140 | 0.391 | 0.014 | 0.250 | 2.50 | 3 | TMC5-.8MM | TMC5-.8MM-A |
| M6-.5 | 0.170 | 0.520 | 0.009 | 0.250 | 2.50 | 3 | TMC6-.5MM | TMC6-.5MM-A |
| M6-1 | 0.170 | 0.528 | 0.018 | 0.250 | 2.50 | 3 | TMC6-1MM | TMC6-1MM-A |
| M8-1 | 0.235 | 0.685 | 0.018 | 0.250 | 2.50 | 3 | TMC8-1MM | TMC8-1MM-A |
| M8-1.25 | 0.235 | 0.660 | 0.022 | 0.250 | 2.50 | 3 | TMC8-1.25MM | TMC8-1.25MM-A |
| M10-1 | 0.290 | 0.803 | 0.018 | 0.3125 | 3.50 | 4 | TMC10-1MM | TMC10-1MM-A |
| M10-1.5 | 0.290 | 0.792 | 0.027 | 0.3125 | 3.50 | 4 | TMC10-1.5MM | TMC10-1.5MM-A |
| M12-1.25 | 0.345 | 0.807 | 0.022 | 0.375 | 3.50 | 4 | TMC12-1.25MM | TMC12-1.25MM-A |
| M14-1.5 | 0.450 | 1.087 | 0.027 | 0.500 | 3.50 | 4 | TMC14-1.5MM | TMC14-1.5MM-A |
| M14-2 | 0.450 | 1.134 | 0.035 | 0.500 | 3.50 | 4 | TMC14-2MM | TMC14-2MM-A |

***Thread mills can cut any larger size internal thread of the same pitch**

# METRIC THREAD MILL
## COOLANT THROUGH - CARBIDE TIPPED




- Non-crest cutting on the internal thread allows maximum flexibility for plated and non-standard threads

| MIN ID THREAD / PITCH* | "A" TOOL DIA. | "B" LENGTH OF CUT | "C" TOOL REACH | "Q" LENGTH | "S" SHANK DIA. | OAL | FLUTES | ORDER # | |
|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | UNCOATED | ALTiN+ |
| | | | | | | | | INTERNAL OR EXTERNAL THREADS | |
| M24-1.5 | 0.740 | 1.058 | 1.370 | 0.027 | 0.750 | 6.00 | 4 | TMC24-1.5MM | TMC24-1.5MM-A |
| M24-2 | 0.740 | 1.100 | 1.370 | 0.036 | 0.750 | 6.00 | 4 | TMC24-2MM | TMC24-2MM-A |
| M24-2.5 | 0.740 | 1.076 | 1.370 | 0.045 | 0.750 | 6.00 | 4 | TMC24-2.5MM | TMC24-2.5MM-A |
| M24-3 | 0.740 | 1.058 | 1.370 | 0.054 | 0.750 | 6.00 | 4 | TMC24-3MM | TMC24-3MM-A |
| M36-4 | 0.990 | 1.095 | 2.000 | 0.071 | 1.000 | 6.00 | 6 | TMC36-4MM | TMC36-4MM-A |

***Thread mills can cut any larger size internal thread of the same pitch**


Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
SCT
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295

# THREAD MILL FEED AND SPEED CHART

| MATERIAL | HB/Rc | SPEED SFM* UNCOATED | SPEED SFM ALTiN+ | FEED ( INCHES PER TOOTH) TOOL DIAMETER | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | | | | .032 - .056 | .059 - .090 | .100 - .190 | .200 - .350 | .370 - .595 | .600+ |
| CAST IRON | 160 HB | 100-220 | 200-425 | .0004-.001 | .0004-.0008 | .0004-.0014 | .0004-.002 | .0004-.0035 | .0004-.006 |
| CARBON STEEL | 18 Rc | 100-200 | 190-425 | .0003-.001 | .0003-.0008 | .0003-.0014 | .0003-.002 | .0003-.005 | .0003-.006 |
| ALLOY STEEL | 20 Rc | 80-200 | 200-375 | .0003-.001 2 Passes | .0003-.0008 3 Passes | .0003-.0014 | .0003-.0024 | .0003-.005 | .0003-.006 |
| TOOL STEEL | 20 Rc | 80-175 | 175-250 | .0003-.0004 2 Passes | .0003-0.0005 3 Passes | .0003-.0005 | .0003-.0009 | .0003-.0026 | .0003-.004 |
| 300 STAINLESS STEEL | 150 HB | 90-120 | 120-255 | .0003-.0005 2 Passes | .0003-0.0006 3 Passes | .0003-.0007 | .0003-.002 | .0003-.0035 | .0003-.0045 |
| 400 STAINLESS STEEL | 195 HB | 90-150 | 140-375 | .0003-.0005 2 Passes | .0003-.0006 3 Passes | .0003-.0007 | .0003-.002 | .0003-.0026 | .0003-.0045 |
| HIGH TEMP ALLOY (Ni & Co BASE) | 20 Rc | 50-125 | 100-125 | .0003-.0004 3 Passes | .0003-.00045 3 Passes | .0003-.0005 2 Passes | .0003-.0009 | .0003-.0026 | .0003-.004 |
| TITANIUM | 25 Rc | 50-130 | 100-170 | .0003-.0004 3 Passes | .0003-.00045 3 Passes | .0003-.001 2 Passes | .0003-.0009 | .0003-.0015 | .0003-.003 |
| HEAT TREATED ALLOYS (38-45Rc) | 40 Rc | 50-90 | 90-150 | .0003-.0004 3 Passes | .0003-.00045 3 Passes | .0003-.0005 2 Passes | .0003-.0008 | .0003-.001 | .0003-.0025 |
| ALUMINUM | 100 HB | 100-800 | 100-1200 | .0005-.0015 | .0005-.002 | .0005-.0025 | .0005-.003 | .0005-.006 | .0005-.009 |
| BRASS, ZINC | 80 HB | 200-350 | 200-750 | .0005-.0015 | .0005-.002 | .0005-.0025 | .0005-.003 | .0005-.006 | .0005-.009 |

*SFM = Surface Feet per Minute

**Parameters are a starting point based on machinability rating at hardness listed.
Check machinability rating of the material to be machined and adjust accordingly.**


Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295

# THREAD MILL FEED AND SPEED APPLICATION



**It may be necessary to use more radial depth passes than shown on the chart (p.40) when cutting an unfavorable length-to-diameter ratio, coarse pitches, or hard materials.** When cutting a thread with two passes, cut approximately **65% of the thread on the first pass and 35 percent on the finish pass.** For three passes, use a **50/30/20** ratio. For four passes, use a **40/27/20/13** ratio. The idea is to equalize the side cutting pressure.

Thread mills can sometimes be used to cut multiple start threads. Call engineering for assistance.

Thread mills can be cut off for shorter thread depths or necked back for deeper thread depths. Call for price and delivery.

In order to apply the Feed and Speed chart appropriately, it is necessary to understand that machining centers will apply the feed rate at the centerline of the spindle. It is correct to use a normal calculation and the following Feed & Speed Chart when cutting in a straight line; however, it is incorrect when cutting an internal thread. Therefore, the feed rate must be recalculated.

***The following is an example of how to apply the feed rate correctly:***

The tool is a TM290-24A cutting a 3/8-24 thread in stainless steel.
The outside diameter of the tool is 0.290.
The surface foot per minute (SFM) is 150.
The chip per tooth ia 0.001. The tool has four flutes.
The revolutions per minute (RPM) equal the SFM x 3.82 divided by the outside diameter of the tool.

In this example: **(150 x 3.82) / 0.290**, which equals 1975 RPM.
The RPM x feed (chip per tooth) x the number of flutes equals the Non-Adjusted Feed Rate or NAFR.
In this example: **1975 x 0.001 x 4 = 7.9 NAFR**
The major diameter of the thread is 0.375. We will call this D.
The outside diameter of the tool is 0.290. We will call this d.
We will call the Adjusted Feed Rate the AFR.
The formula for the AFR for internal interpolation is **AFR = NAFR x (D-d) ÷ D**

In this example: **AFR = 7.9 x (0.375 - 0.290) ÷ 0.375**

Therefore, the Adjusted Feed Rate equals 1.79. This is the feed rate that will equal 0.001 chip per tooth in the above example. This is the feed rate that must be used in the CNC program.

# THREAD MILL TROUBLESHOOTING

| PROBLEM | CAUSE | SOLUTION |
|---|---|---|
| TAPERED THREADED HOLE | TOOL PRESSURE | Reduce the chip load and/or make more radial passes. |
| NO-GO GAGE GOES & GO GAGE DOES NOT GO | THREAD OVERCUTTING | Use a tool of smaller diameter with correct pitch. Make sure helical "ramp in" is used. |
| TEETH ARE CHIPPING | TOOL PRESSURE | Reduce feed rate per tooth. |
| | BUILT-UP EDGE | Use a coated tool to help reduce built-up edge. |
| RAPID WEAR | TOOL RUBBING NOT CUTTING | Increase chip load per tooth. |
| TEETH ARE BURNING | TOO MUCH HEAT | Reduce speed. Use a coated tool. Increase coolant. |
| TOOL BREAKS | TOO MUCH TOOL PRESSURE | Helical "arc in" must be used. Reduce feed rate and/or use more radial passes. Adjusted Feed Rate (AFR) must be used. (See Thread Mill Feed and Speed Chart) |

Thread milling tools form a thread using a motion referred to as "helical interpolation." This process involves the machine simultaneously moving all three axes. The resulting motions are circular and axial. The "X" and "Y" axes move in a circular manner and the "Z" axis in an axial direction per 360° at a distance equal to the pitch of the thread being machined. The tool should "ramp in" over 90° in order to avoid breakage. This must be a helical move. Move "Z" axially by pitch ÷ 4 since 90° is 360° ÷ 4.

Bottom-to-top climb cutting machining is recommended when machining a right-hand thread. This will avoid re-cutting any chips. For left hand threading, a top-to-bottom machining with a right-hand helical tool is the preferred method. Refer to troubleshooting chart above for solutions to potential thread milling problems.


Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295