15 Thread Cutting Cycles

By programming a single point tool to feed axially over the same point again and again a thread will be cut. Three thread cutting cycles are provided: G32, G92 and G76. When these are used, each tool path will start out at the same point. Threading must be done in G97 mode only.

G32- Each axial pass requires the input of four blocks of data.

G92- Each axial pass requires the input of one block of data.

G76- One block of data is required to cut the whole thread, automatic in-feed and compound cutting are provided.

15.1 Thread Cutting Limitations

Due to the response delay in the servo system there is a limit to how fast the threading tool can be programmed to move. This limit is on the maximum allowable RPM with respect to the pitch of the thread.

The following formula will apply:

\[ \text{RPM} \times \text{PITCH} = <160 \]

The above means that the feed rate must not exceed 160 inches per minute.

The maximum allowable RPM in threading can be calculated as follows:

\[ 160 \text{ IPM} / \text{PITCH} \]
15.2 Imperfect Thread Calculation

When threading it is important to take into consideration the distance needed for the acceleration and deceleration of the cutting tool while it is in the work piece.

The cutting tool should be positioned far enough in front of the start of the thread to allow enough distance for acceleration before it enters the material (\(d_2\)). At the end of the thread, there should be enough distance allowed for deceleration of the tool (\(d_1\)).

\[
\begin{align*}
  d_2 &= \frac{\text{RPM} \times \text{Pitch}}{1800} \\
  d_1 &= d_2 \times 3.605
\end{align*}
\]

Notes on threading:
1) Do not use G96, use G97
2) Do not exceed the maximum allowable RPM in threading
3) Calculate the Imperfect Thread Portion and position tool accordingly.
4) Feed rate override is held at 100% during the threading operation.
15.3 G76 Thread Cutting, Multiple Repetitive

The G76 command is a two-line call out the same as all multiple repetitive cycles. However by setting certain parameters you can eliminate the first line. However the program will then not dictate these values the parameters will. If you need to change these values for each part you are better served to put them in the program. The program overrides the parameters.

The first line is as follows:

**G76 P (m) (r) (a) Q (d min) R (d)**

P is a six digit character, two digits each for m, r, and a
(m) = number of finishing passes
(r) = chamfering amount
(a) = included angle of the tool tip
(d min) = sets the minimum cutting depth
(d) = finishing allowance.

Standard values for each modifier m, r, a, dmin, and d are normally preset, their parameters and initial values are as follows:

<table>
<thead>
<tr>
<th>Modifier</th>
<th>Parameter #</th>
<th>Standard Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>5142</td>
<td>1</td>
</tr>
<tr>
<td>R</td>
<td>5130</td>
<td>10</td>
</tr>
<tr>
<td>A</td>
<td>5143</td>
<td>60</td>
</tr>
<tr>
<td>Dmin</td>
<td>5140</td>
<td>20</td>
</tr>
<tr>
<td>D</td>
<td>5141</td>
<td>10</td>
</tr>
</tbody>
</table>

The above parameters are for the 16/18/21 control and are to be set as desired for the thread/machining conditions on hand
The correct syntax for the second line is as follows:

**G76 X Z P Q F**

X = for OD threads, the minor diameter, for ID threads, the major diameter.
Z = the endpoint of the thread in the Z-axis.
P = height of a complete thread, radial, no decimal point (note 1)
Q = depth of first cutting pass, no decimal point (note 1)
F = feed rate of tool, this is the same as the thread pitch

Calculation of either the number of passes or the depth of the first pass is possible by applying one of the formulas as shown below:

Where D = the depth of the first pass.
P = the radial height of a single thread.
N = Number of passes (minus spring passes).

\[ N = \frac{P}{D} \]

\[ D = P \sqrt{N} \]

Note 1: .001 = 10, .01 = 100, .015 = 150
Tool-path of G76 cycle, OD thread:

1- Rapid to major diameter, minus 2*D
2- Thread to Z axis dimension, first pass
3- Rapid out to start point diameter
4- Rapid to start point Z
5- Rapid to diameter for second pass
6- Thread to Z-axis dimension.
7- Sequence is repeated until thread reaches it's programmed depth (minor diameter)
8- Repeat one pass without any additional in feed, this is a spring pass
9- Return to start point of cycle in X & Z.
Example of two line G76

<table>
<thead>
<tr>
<th>Thread = 2 – 18</th>
<th>Pitch = .0555</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major Diameter = 2.0</td>
<td>Number of Finish Passes = 3</td>
</tr>
<tr>
<td>Minor Diameter = 1.9302</td>
<td>Chamfer Amount = 0 (pull straight out)</td>
</tr>
<tr>
<td>Radial Height of Thread = .0349</td>
<td></td>
</tr>
</tbody>
</table>

G0 X2.1 Z.2 rapid to clear stock
G76 P030060
G76 X1.903 Z-.8 P349 Q120 F.0555
G0 X9. X5. T500

Notes on using G76 threading cycle:

<table>
<thead>
<tr>
<th>Rapid move to start point</th>
<th>OD Threading</th>
<th>ID Threading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X axis approx. .1&quot; larger than major thread diameter</td>
<td>X axis approx. .1&quot; smaller than minor thread diameter</td>
</tr>
<tr>
<td></td>
<td>Z axis, in front of thread by at least 2 threads</td>
<td></td>
</tr>
<tr>
<td>G76 Threading Cycle</td>
<td>Order point is smaller than start point</td>
<td>Order point is larger than start point</td>
</tr>
</tbody>
</table>

Example: 2 1/2 – 18 TPI
Major Diameter: 2.0"
Minor Diameter: 1.9303
Radial Thread Depth: .0349"
Pitch:.0555 .

```
G0 X21 Z.2
G76 X1.9303 Z-.825 P349 Q120 F.0555
G0 G40 X10. Z5. T500
```
15.4 **G76 Thread Cutting, Multiple Repetitive, Taper**

By adding address R to the standard G76 line along with a numerical value taper threads can be generated.

\[
\text{G76 } \text{X Z R P Q F}
\]

\(R\) is the difference in diameter from the Start point of the tool at the beginning of the tapered thread to the end point of the tool at the end of the tapered thread. It is preferred to start at least .2 off the beginning of the thread and where possible to continue passed the end of the thread. The distance at the beginning and at the end must be part of the calculation. This value is expressed radial. The value is signed depending on the direction of the taper.

All other addresses relative to G76 are in effect.
15.5 G76 – THREADING CYCLE – TWO LINE FORMAT (FS 0,16,18,21T-FORMAT)
(Applicable with Fanuc Controls, T series, systems 0, 16, 18, 21. Also: Mitsubishi 500L, 50, 64)

FIRST COMMAND LINE: G76 P021060 Q05 R10 (See detailed explanation, below)

P 02 10 60 Specify “P”, followed by a six digit number. Translation is shown, here:

02 =Number of finish passes at the bottom of the thread (02 means 2 Passes) (Sets PAR 5142– see note 1)
10 =Chamfer-width or pullout-width at the “Z” end position of the thread. Chamfer size is expressed in1/10th fractions of the lead. 10: means the chamfer-width equals one lead. 05: means the chamfer width equals ½ of lead. 00: means no chamfer. See note 3, below. (Sets PAR 5130)
60 =The included angle between the thread flanks. ½ of the angle as specified is applied for the in-feed angle. Normally, 60° is used for standard threads. Other angles, such as: 80°, 60°, 55°, 30°, 29° or 0° can be specified (Sets PAR 5143)

Q05 =Minimum cutting depth (depth of cut becomes progressively smaller, after each pass. Remainder, smaller than “Q” is discarded (Sets PAR 5140).

R10 =Material allowance for finishing passes at bottom of thread (Sets PAR 5141)

SECOND COMMAND LINE: G76 X__ Z__ P100 Q200 R300 F0.038461 (See detailed explanation, below)

X =Diameter of the thread. For an external Thread – specify the minor diameter.
For an internal Thread – specify the major diameter. In case of taper threads, specify the diameter at the opposite end from the start point.

Z =End position of the thread.

P =Height of the thread. Calculation: Major diameter minus minor diameter, divided by 2 (Radius value, without decimal point) See note 2, below.

Q =Depth of the first cut. If “P” and “Q” are the same, then the thread is `cut in a single pass. (Radius value without decimal point) See note 2, below

R =Taper. (Radial height difference of taper slope, per side) Specify a negative value for OD taper thread. Specify a positive value for ID taper thread).

F = Lead: distance between two threads. (1 divided by the pitch), six digits allowed after the decimal point.

NOTES:
1.) Upon execution of the G76-cycle all data contained on the first G76-command line is automatically stored in the parameter tables.

2.) Values for “P” and “Q” to be specified without decimal point for all Fanuc Controls.
   For example:
   0.0001”=1,  0.001”=10   0.01”=100   0.1”=1000   1.0”=10000

3.) Specifying a chamfer (pullout distance) reduces possible damage to the last thread lead near the Z-end position.
G76 – THREADING CYCLE – TWO LINE FORMAT (FS 0.16, 18, 21T-FORMAT)
(Applicable with Fanuc Controls, T series, systems 0, 16, 18, 21. Also: Mitsubishi 500L, 50, 64)

**FIRST LINE:**

G76  P02  10  60  Q05  R10

- Material allowance for finishing passes at bottom of thread. (PAR 5141)
- Minimum cutting depth (depth of cut becomes progressively smaller, after each pass. Fractions smaller than “Q” is discarded (PAR 5140).
- Included angle of thread in degrees (60 for std. thread). (PAR 5143)
- Chamfer amount at pull out end of thread, expressed by lead times 10. (10 = chamfer size equal to one lead, 05 = chamfer size equal to ½ of lead). (No chamfer = 00). (PAR 5130)
- Number of finish passes at bottom of thread (02 = 2 Passes). (PAR 5142)

**SECOND LINE:**

G76 X-- Z-- P100 Q200 R300 F0.038451

- Lead (1 divided by pitch), six digits allowed after the decimal point.
- Taper (Radial height difference per side) Specify a negative value for OD taper thread. Specify a positive value for ID taper thread.
- Depth of the first cut, (Radius value).
- Height of the thread, (Radius value).
- Z - end position of the threaded (W) In case of incremental command

**NOTES:**
1.) Upon execution the data of the first G76-command line, data is automatically stored in the parameter tables as indicated.
2.) Values for “P” and “Q” to be specified without decimal point for all Fanuc Controls. (For example, 0.0001=1 , 0.001=10 , 0.01=100 , 0.1=1000)
This format can also be used with Fanuc Controls, T series, system 0, 16, 18 and 21, when the tape format setting option is available. In this case, please display the “SETTING PAGE”, then check the “TAPE-F” setting. When “TAPE-F” is set = 0, the two-line format is valid (see previous page). When it is set = 1, the single-line format is valid. This setting will affect all G70-series canned cycles, not just the threading.

**Cycle Format:** G76 X__ Z__ I__ K__ D__ F__ A__ P__ Q__

- **X** = Diameter of the thread. For an external Thread – specify the minor diameter. For an internal Thread – specify the major diameter. In case of taper threads, specify the diameter at the opposite end from the cutting start point.

- **Z** = End position of the thread.

- **I** = Taper: Radial height difference of taper slope. Calculate the height difference of the slope using the entire Z-axis moving distance, including the Z-clearance at the start of the thread. Specify a negative value for OD taper thread. Specify a positive value for ID taper thread.

- **K** = Height of the thread, radius value. Calculation: Major diameter minus minor diameter, divided by 2.

- **D** = Depth of the first cut (Radius value). If “K” and “D” are the same, then the tread is cut in a single pass.

- **F** = Lead: distance between two threads. (1 divided by the pitch), six digits allowed after the decimal point.

- **A** = Tool nose angle or angle between thread flanks (Range: 0 to 120 degrees, in 1-degree increments) If “A” is omitted it is regarded as 0, straight in-feed is applied (In case of a V-shaped tool, both edges will cut at the same time)

- **P** = Cutting method:
  - P1=constant chip load, single edge cutting
  - P2=constant chip load, zigzag in-feed, alternating cutting edges
  - P3=constant cut depth, single edge cutting
  - P4=constant cut depth, zigzag in-feed

- **Q** = Spindle rotation shift angle. Data range is from 0 to plus or minus 360000 (360 degrees = 360000, without decimal point) This function is used for cutting of multiple-Lead threads. For example: in case of a 3-start thread the shift angle is 120 degrees between each thread. Hence, the first thread lead is cut, using Q=0, the second at Q=120000 and the third at Q=240000, where the Z-axis start position remains the same for each thread.
### 15.6 Programming Examples, using the G76-Thread Cutting Cycle

Example 1: Cutting a 1”-10 UNS -external thread:

<table>
<thead>
<tr>
<th>Action</th>
<th>Program</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Enter modal commands</td>
<td>G0 G18 G40 G97 G99</td>
</tr>
<tr>
<td>2. Enter the tool and tool offset command</td>
<td>T0101</td>
</tr>
<tr>
<td>3. Enter the Spindle command</td>
<td>G97 S100 M3 (M4)</td>
</tr>
<tr>
<td>(Always use G97, NEVER G96)</td>
<td></td>
</tr>
<tr>
<td>4. Turn ON the coolant</td>
<td>M8</td>
</tr>
<tr>
<td>5. Move the tool to the start position of</td>
<td>G0 Z0.125</td>
</tr>
<tr>
<td>the thread</td>
<td>X1.075</td>
</tr>
<tr>
<td>For “Z”, allow 125% of the Lead for</td>
<td></td>
</tr>
<tr>
<td>start-up clearance away from the</td>
<td></td>
</tr>
<tr>
<td>thread</td>
<td></td>
</tr>
<tr>
<td>Move “Z” fist, then “X”.</td>
<td></td>
</tr>
<tr>
<td>For “X”, allow 0.05” ~ 0.1”</td>
<td></td>
</tr>
<tr>
<td>diametrical clearance above the major</td>
<td></td>
</tr>
<tr>
<td>diameter (OD)</td>
<td></td>
</tr>
<tr>
<td>6. Enter the thread cutting cycle</td>
<td>G76 P020560 Q05 R0</td>
</tr>
<tr>
<td></td>
<td>G76 X0.875 Z-1.0 P625 Q250 F0.1</td>
</tr>
<tr>
<td>7. Return the tool to the tool exchange</td>
<td>G0 X</td>
</tr>
<tr>
<td>point</td>
<td>G0 Z___</td>
</tr>
<tr>
<td>Move the “X”-axis first, then “Z”</td>
<td>M1</td>
</tr>
<tr>
<td>Optional stop</td>
<td></td>
</tr>
</tbody>
</table>
Example 2: Cutting a 1”-10 UNS -internal thread:

<table>
<thead>
<tr>
<th>Action</th>
<th>Program</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Enter modal commands</td>
<td>G0 G18 G40 G97 G99</td>
</tr>
<tr>
<td>2. Enter the tool and tool offset command</td>
<td>T0101</td>
</tr>
<tr>
<td>3. Enter the Spindle command</td>
<td>G97 S100 M3 (M4)</td>
</tr>
<tr>
<td>(Always use G97, NEVER G96)</td>
<td></td>
</tr>
<tr>
<td>4. Turn ON the coolant</td>
<td>M8</td>
</tr>
<tr>
<td>5. Move the tool to the start position of the thread</td>
<td></td>
</tr>
<tr>
<td>For “Z”, allow 125% of the Lead for start-up clearance away from the</td>
<td></td>
</tr>
<tr>
<td>thread</td>
<td></td>
</tr>
<tr>
<td>Move “Z” first, then “X”.</td>
<td></td>
</tr>
<tr>
<td>For “X”, allow 0.05” ~ 0.1” diametrical clearance below the minor</td>
<td></td>
</tr>
<tr>
<td>diameter (I.D.)</td>
<td>G0 Z0.125 X0.800</td>
</tr>
<tr>
<td>6. Enter the thread cutting cycle</td>
<td>G76 P020560 Q05 R0</td>
</tr>
<tr>
<td></td>
<td>G76 X1.0 Z-1.0 P500 Q150 F0.1</td>
</tr>
<tr>
<td>7. Move the tool out of the bore, clearing the face</td>
<td>G0 Z___</td>
</tr>
<tr>
<td>8. Return the tool to the tool exchange point</td>
<td>G0 X___</td>
</tr>
<tr>
<td>Move the “X”-axis first, then “Z”</td>
<td>G0 Z___</td>
</tr>
<tr>
<td>Optional stop</td>
<td>M1</td>
</tr>
</tbody>
</table>

Note: Source for thread dimensions used in the thread cutting cycles shown above: “Machinery’s Handbook” (Twentieth edition).
15.7 G76 Thread Cutting, Multiple Repetitive, Multi Start

Multiple start threads are possible in the G76 mode, you have to shift the starting point for the extra threads by 1/n of the pitch.

![Diagram of thread cutting](image)

Example

Cut 5" - 4TPI, 3 start

G0 X5.1 Z.15 rapid to start of thread
G76 X4.9633 Z-1.4 P1534 Q250 F.25 cut first thread
G0 W.0833 shift by 1/n of pitch
G76 X4.9633 Z-1.4 P1534 Q250 F.25 cut second thread
G0 W.0833 shift by 1/n of pitch
G76 X4.9633 Z-1.4 P1534 Q250 F.25 cut third thread
G0 X9. Z5. T900

Procedure:

1- Cut first thread starting in front of thread by at least d₁
2- Shift starting point by 1/n of pitch (this is the starting point of the second thread)
3- Cut second thread
4- Shift starting point by 1/n of pitch (this is the starting point of the third thread)

The shift amount (W) is determined as follows:

\[ W = (1 \text{ inch} / \# \text{ of starts}) \times \text{Pitch} \]

In this example \( W = (1/3) \times .25 \)
\[ W = .0833 \]
15.8 G92 Thread Cutting

The G92 command will drive the cutting tool in a "box" pattern.

Straight threads can be cut using the following command:

G92 X Z F Q

X = the diameter that you are cutting the pass at
Z = the endpoint of the threading pass
F = the feed rate (pitch) of the thread
Q = the

The tool must first be positioned to the start point of the cutting cycle. The G92 command will then specify the diameter (order point) to thread at and the pitch.

Tool path of G92 cycle, OD thread:

1 - Tool rapids to cutting diameter (order point)
2 - Thread to Z axis dimension (order point)
3 - Rapid out to start point diameter
4 - Rapid to start point Z
Notes on using G92 threading cycle

<table>
<thead>
<tr>
<th>Rapid move to start point</th>
<th>OD Threading</th>
<th>ID Threading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X axis approx. 0.1&quot; larger</td>
<td>X axis approx. 0.1&quot; smaller</td>
</tr>
<tr>
<td></td>
<td>than major thread diameter</td>
<td>than minor thread diameter</td>
</tr>
<tr>
<td></td>
<td>Z axis, in front of thread by at</td>
<td></td>
</tr>
<tr>
<td></td>
<td>least 2 threads</td>
<td></td>
</tr>
<tr>
<td>G92 Threading Cycle</td>
<td>Order point is smaller than start</td>
<td>Order point is larger than start</td>
</tr>
<tr>
<td></td>
<td>point</td>
<td>point</td>
</tr>
</tbody>
</table>

Example of G92

<table>
<thead>
<tr>
<th>Thread = 2 – 18</th>
<th>Pitch = 0.0555</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major Diameter = 2.0</td>
<td>Minor Diameter = 1.9302</td>
</tr>
</tbody>
</table>

| Minor Diameter = 1.9302 | Radial Thread Height = 0.0349 |

G00 X2.1 Z.2  
G92 X1.976 Z-.8 F.0555  
X1.966  
X1.9584  
X1.952  
X1.9464  
X1.9412  
X1.9366  
X1.9322  
X1.9303  
X1.9303  
G0 X9. Z5. T500  

rapid to stock  
1st pass  
2nd pass  
3rd pass  
4th pass  
5th pass  
6th pass  
7th pass  
8th pass  
9th pass  
spring pass  
tool change position, cancel G92 & offset  

G92 is modal, it is not necessary to repeat the command for each block, just change the X-axis order point. When you are finished with this cycle cancel it out with the G0 command.

If the thread does not run into a relief groove it is suggested that chamfering pullout be applied. This will be done at a 45-degree angle and is controlled by parameter #5130. The chamfering distance is set from 0.1 - 12.7 of pitch. 1 = 0.1 pitch of chamfer, 10 = 1 pitch of chamfer.
15.9 G92 Thread Cutting, Taper

By adding address R to the standard G92 line along with a numerical value taper threads can be generated.

G92 X Z R F

\( R \) = is the difference in diameter from the beginning of the tapered thread to the end of the tapered thread. This is expressed radial. The value is signed depending on the direction of the taper.
All other addresses relative to G92 are in effect.