

2.1.5 Rigid tapping – CYCLE84



Programming

CYCLE84 (RTP, RFP, SDIS, DP, DPR, DTB, SDAC, MPIT, PIT, POSS, SST, SST1)



Parameters

RTP	real	Retraction plane (absolute)
RFP	real	Reference plane (absolute)
SDIS	real	Safety clearance (enter without sign)
DP	real	Final drilling depth (absolute)
DPR	real	Final drilling depth relative to reference plane (enter without sign)
DTB	real	Dwell time at thread depth (chip breaking)
SDAC	int	Direction of rotation after end of cycle Values: 3, 4 or 5
MPIT	real	Pitch as thread size (with sign) Value range: 3 (for M3) ... 48 (for M48), the sign determines the direction of rotation in the thread
PIT	real	Pitch as value (with sign) Value range: 0.001 ... 2000.000 mm), the sign determines the direction of rotation in the thread
POSS	real	Spindle position for oriented spindle stop in the cycle (in degrees)
SST	real	Speed for tapping
SST1	real	Speed for retraction



Function

The tool drills at the programmed spindle speed and feedrate to the programmed thread depth.

With cycle CYCLE84 you can perform rigid tapping operations.



Cycle CYCLE84 can be used if the spindle to be used for the boring operation is technically able to go into position-controlled spindle operation.



A separate cycle CYCLE840 exists for tapping with compensating chuck (see Section 2.1.6).



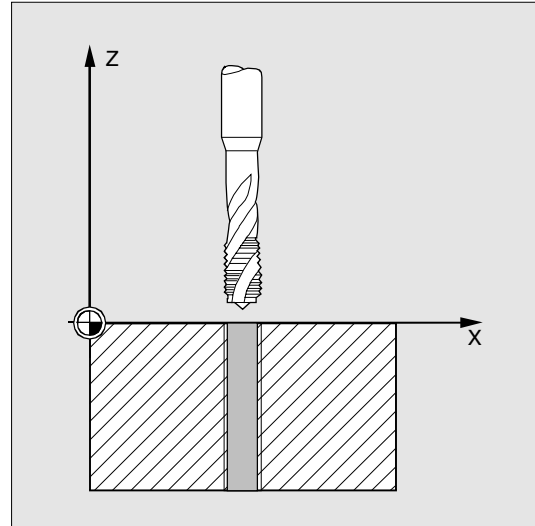
Sequence of operations

Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

The cycle implements the following motion sequence:

- Approach of the reference plane brought forward by the safety clearance with G0
- Oriented spindle stop with SPOS (value in parameter POSS) and conversion of spindle to axis mode
- Tapping to final drilling depth with G331 and speed SST
- Dwell time at thread depth (parameter DTB)
- Retraction to the reference plane brought forward by the safety clearance with G332, spindle speed SST1 and reversal of direction of rotation
- Retraction to the retraction plane with G0, spindle mode is reintroduced by reprogramming the spindle speed active before the cycle was called and the direction of rotation programmed under SDAC.



Description of parameters

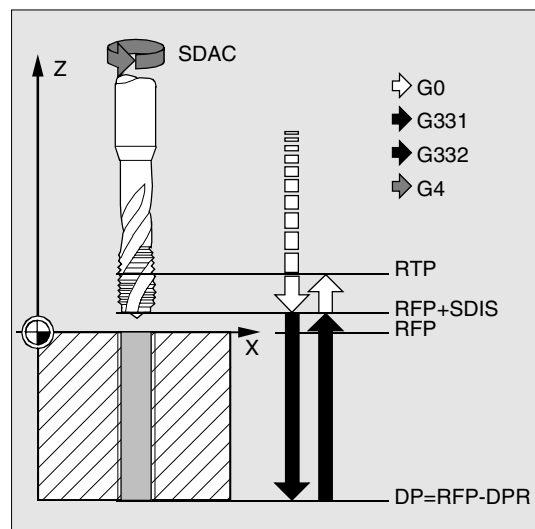
See Section 2.1.2. (Drilling, Centering – CYCLE81) for a description of parameters RTP, RFP, SDIS, DP, DPR

DTB (dwell time)

You program the dwell time in seconds. It is recommended that the dwell time is omitted for the tapping of blind holes.

SDAC (direction of rotation after end of cycle)

Under SDAC you program the direction of rotation after completion of the cycle. For tapping, the direction is changed automatically by the cycle.



MPIT and PIT (as thread size and as value)

The value for the thread pitch can either be defined as the thread size (for metric threads between M3 and M48 only) or as a value (distance from one thread turn to the next as a numerical value). The parameter not required in each case is omitted from the call or assigned the value zero.

Right or left threads are specified by the sign of the pitch parameter:

- Positive value → right (like M3)
- Negative value → left (like M4)

If the two thread pitch parameters have conflicting values, alarm 61001 "Thread pitch wrong" is generated by the cycle and cycle execution is aborted.

POSS (spindle position)

Before tapping starts in the cycle, oriented spindle stop is performed with command SPOS and the spindle is brought into position control.

You program the spindle position for this spindle stop under POSS.

SST (speed)

Parameter SST contains the spindle speed for the tapping block with G331.

SST1 (retraction speed)

Under SST1 you program the speed for the retraction out of the thread hole in the hole with G332.

If this parameter is assigned the value zero, the retraction movement is performed with the speed programmed under SST.

**Further notes**

The direction of rotation is always reversed automatically for tapping in cycle.

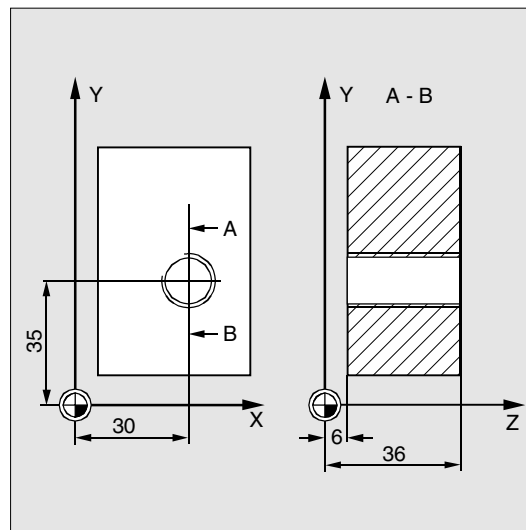
2.1 Drilling cycles



Programming example

Rigid tapping

A thread is tapped without a compensating chuck at position X30 and Y35 in the XY plane, the tapping axis is the Z axis. No dwell time is programmed. The depth is programmed as a relative value. The parameters for the direction of rotation and the pitch must be assigned values. A metric thread M5 is tapped.



```
N10 G0 G90 T4 D4
```

Specification of technology values

```
N20 G17 X30 Y35 Z40
```

Traverse to drilling position

```
N30 CYCLE84 (40, 36, 2, , 30, , 3, 5, ->
->, 90, 200, 500)
```

Cycle call, parameter PIT has been omitted, no value is entered for the absolute depth or the dwell time. Spindle stop at 90 degrees, speed for tapping is 200, speed for retraction is 500

```
N40 M30
```

End of program

-> Must be programmed in a single block