

3.10 Milling circular pockets - POCKET4

The cycle POCKET4 is available with Software Version 4.

**Programming**

POCKET4 (_RTP, _RFP, _SDIS, _DP, _PRAD, _PA, _PO, _MID, _FAL, _FALD, _FFP1, _FFD, _CDIR, _VARI, _MIDA, _AP1, _AD, _RAD1, _DP1)

**Parameters**

The following input parameters are always required:

_RTP	real	Retraction plane (absolute)
_RFP	real	Reference plane (absolute)
_SDIS	real	Safety clearance (to be added to the reference plane, enter without sign)
_DP	real	Pocket depth (absolute)
_PRAD	real	Pocket radius
_PA	real	Pocket center point, abscissa (absolute)
_PO	real	Pocket center point, ordinate (absolute)
_MID	real	Maximum infeed depth (enter without sign)
_FAL	real	Final machining allowance on pocket edge (enter without sign)
_FALD	real	Final allowance at base (enter without sign)
_FFP1	real	Feedrate for surface machining
_FFD	real	Feedrate for depth infeed
_CDIR	int	Milling direction: (enter without sign) Value: 0...Climb milling (as spindle rotation) 1...Opposed milling 2...With G2 (independent of spindle direction) 3...With G3
_VARI	int	Type of machining: (enter without sign) UNITS DIGIT: Value: 1...Roughing 2...Finishing TENS DIGIT: Value: 0...Perpendicular to the pocket center with G0 1...Perpendicular to the pocket center with G1 2...Along a helix

The other parameters can be selected as options. They define the insertion strategy and overlapping for solid machining: (enter without sign)

_MIDA	real	Maximum infeed width during solid machining in the plane
_AP1	real	Basic size pocket radius
_AD	real	Basic pocket depth from reference plane
_RAD1	real	Radius of the helical path during insertion related to the tool center point path)
_DP1	real	Insertion depth per 360° revolution on insertion along helical path



Function

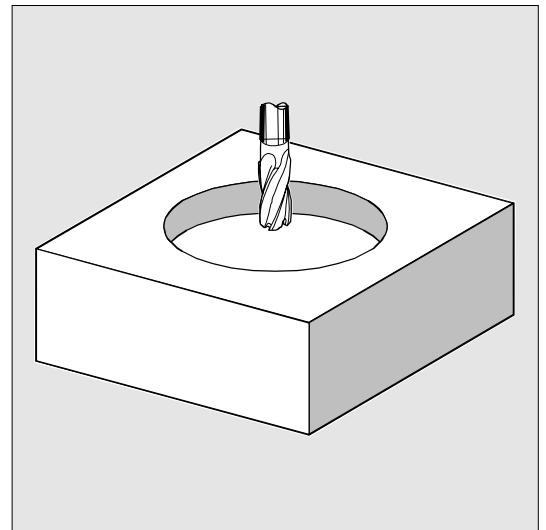
With this cycle you can machine circular pockets in the machining plane.

For finishing, a face cutter is needed.

The depth infeed will always start at the pocket center point and be performed vertically from there; thus predrill can be suitably performed in this position.

New functions compared to POCKET2:

- The milling direction can be defined with a G instruction (G2/G3) or climb milling or opposed from the spindle direction
- For solid machining, the maximum infeed width in the plane is programmable
- Finishing allowance for the pocket base
- Two different insertion strategies:
 - Vertically from the pocket center point
 - Along a helical path around the pocket center
- Shorter approach paths in the plane for finishing
- Consideration of a blank contour in the plane and a basic size at the base (optimum processing of pre-formed pockets possible)
- _MIDA is recalculated when machining the edge.





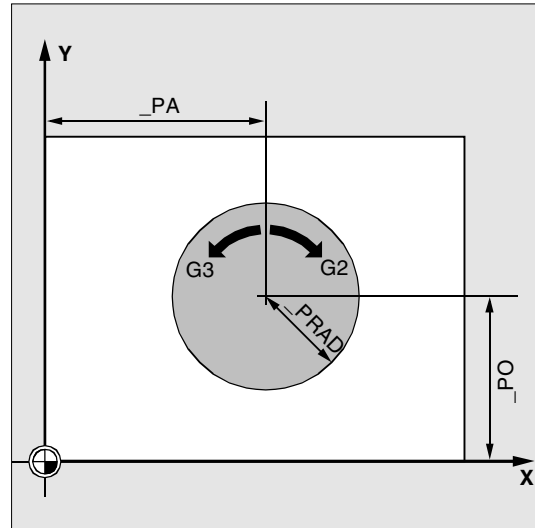
Sequence of operations

Position reached prior to cycle start:

This can be any position from which the starting position on the center point of the pocket at the retraction plane level can be approached without collision.

Motion sequence when roughing (VARI=X1):

With G0, the pocket center point is approached at the retraction plane level and then, from this position, with G0 the reference plane brought forward by the safety clearance is approached. Pocket machining is then performed according to the selected insertion strategy and considering the programmed blank dimensions.



Insertion strategies:

see Section 3.9 (POCKET3)

Accounting for blank dimensions

During solid machining, it is possible to take blank dimensions (for example, in the machining of precast workpieces) into account.

For circular pockets, the basic size `_AP1` at the edge is also circular (with a smaller radius than the pocket radius).



For additional explanations see Section 3.9 (POCKET3)

Motion sequence when finishing (VARI=X2):

Finishing is performed in sequence from the edge until reaching the finishing allowance on the base, then the base is finished. If one of the finishing allowances is equal to zero, this part of the finishing process is skipped.

- Finishing on the edge

While finishing on the edge, the pocket is only machined once.

For finishing on the edge the path includes one fourth of circle which reaches the pocket radius. The radius of this path is less or equal to 2 mm or, if "less room" is available, equals the difference between the pocket radius and the mill radius.

The depth infeed is performed with G0 in the open towards the pocket center and the starting point of the approach path is also reached with G0.

- Finishing on the base

During finishing on the base, the machine performs G0 towards the pocket center until reaching a distance equal to pocket depth + finishing allowance + safety clearance. From this point onwards, the tool is always fed in **vertically** at the depth infeed feedrate (since a tool with a front cutting edge is used for base finishing).

The base surface of the pocket is machined once.



Description of parameters



See Section 2.1.2. (Drilling, Centering – CYCLE81) for a description of parameters `_RTP`, `_RFP`, `_SDIS`

See Section 3.7 (POCKET1) for a description of parameter `_DP`.



See Section 3.9 (POCKET3) for a description of parameters `_MID`, `_FAL`, `_FALD`, `_FFP1`, `_FFD`, `_CDIR`, `_MIDA`, `_AP1`, `_AD`, `_RAD1`, `_DP1`.

See Section 3.2 for cycle setting data `_ZSD[1]`.

`_PRAD` (pocket radius)

The shape of the circular pocket is determined by the radius only.

If the radius is less than the tool radius of the active tool, the cycle is aborted after alarm

61105 "Milling cutter radius too large" is output.

`_PA`, `_PO` (pocket center point)

With parameters `_PA` and `_PO` you define the center point of the pocket. Circular pockets are always measured from the center.

`_VARI` (machining mode)

You can define the type of machining with parameter `_VARI`.

Possible values are:

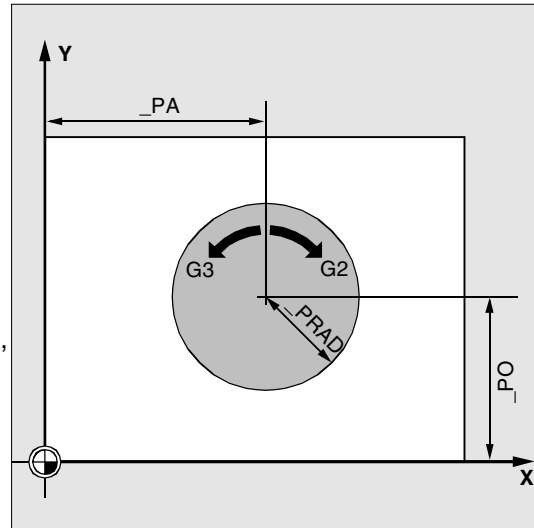
Units digit:

- 1= Roughing
- 2= Finishing

Tens digit (infeed):

- 0= Perpendicular to the pocket center with G0
- 1= Perpendicular to the pocket center with G1
- 2= Along an helical path

If another value has been programmed for parameter `_VARI`, the cycle is aborted after alarm 61002 "Machining type incorrectly defined" is output.





Further notes

A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 "No tool offset active" is output.

A new workpiece coordinate system that influences the actual value display is used in the cycle. The zero point of this coordinate system lies on the pocket center point.

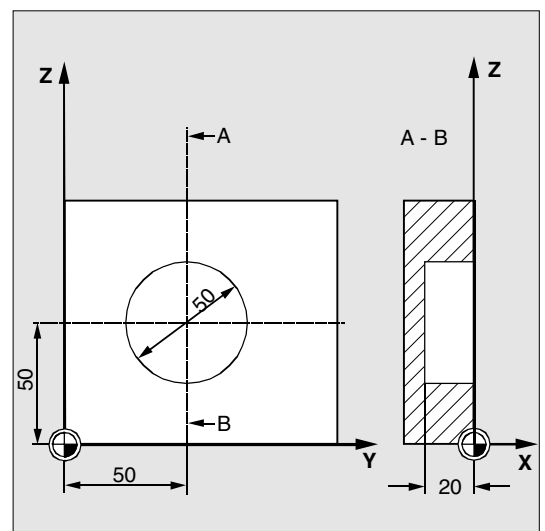
The original coordinate system becomes active again after the end of the cycle.



Programming example

Circular pocket

With this program you can machine a circular pocket in the YZ plane. The center point is defined by Y50 Z50. The infeed axis for the depth infeed is the X axis. Neither a final machining allowance nor a safety clearance is defined. The pocket will be machined using opposed milling. Infeed occurs along an helical path.



N10 G19 G90 G0 S650 M3 T20 D20	Specification of technology values
N20 Y50 Z50	Approach starting position
N30 POCKET4 (3, 0, 0, -20, 25, 50, 50, ->	Cycle call
-> 6, 0, 0, 200, 100, 1, 21, 0, 0, 0, ->	Parameters FAL and VARI are omitted
-> 2, 3)	
N40 M30	End of program

-> Must be programmed in a single block