5. Cycles

5.1 G69 measuring cycles

General

The measuring cycle is a NC program with the term Z69. This cycle must like all other cycles into the NC memory be loaded. The measuring cycle is called with G69.

Measuring cycles use the parameters P140 to P160 for programming. These parameters contain two types of information:

- Data, e.g.. P141 safety margin

- Parameter numbers, which indicate, where the data are stored,
  e.g.. P149: 500 (P149 is a pointer on another parameter).

Measuring cycles use the parameters P200 to P299 for internal calculations.
Parts it safely that only free parameters are used for the addressing of data fields.
Measuring cycles activate G40 (tool radius path correction out).

5.1 G69 measuring cycles (continued)

Measuring preparation

- Connecting it the sensor with the measuring input on the AAZ module (15pol. HD Sub Socket). If the measuring axes are distributed on several modules, all interconnect you Measuring inputs together (MT+ / MT-).

- Checking it the measuring logic for each measuring axis (P12045: $xxxx1010...).

- Checking it the tool data of the used sensor. G69 requires tool length and tool radius of the sensor.

5.1 G69 measuring cycles (continued)

Compilation of the cycle parameters

Parameter		meaning
P140		cycle selection
P141	[mm]	safety margin
P143	[mm/min]	measuring speed
P144	[n]	number of measurements at the same point (> 0)
P147		pointer on the measuring tolerance data
(P147)+4	[mm]	confidence range
P148		measuring axis in plane system (1... 3)
P149		pointer on the data of point of trigger
(P149)+0	[mm]	XN, point of trigger in negative direction 1. axis
(P149)+1	[mm]	XP, point of trigger in positive direction 1. axis
(P149)+2	[mm]	YN, point of trigger in negative direction 2. axis
(P149)+3	[mm]	YP, point of trigger in positive direction 2. axis
(P149)+4	[mm]	ZN, point of trigger in negative direction 3. axis
(P149)+5	[mm]	ZP, point of trigger in positive direction 3. axis
P150		pointer on the calibration drilling data
(P150)+0	[mm]	position of the calibration drilling focal point 1. axis (I)
(P150)+1	[mm]	position of the calibration drilling focal point 2. axis (J)
(P150)+2	[mm]	position of the calibration drilling focal point 3. axis (K)
(P150)+3	[mm]	calibration drilling diameter
P152	[mm]	expected drilling diameter
P154		pointer on the measuring data
(P154)+0	[mm]	measuring position 1. axis
(P154)+1	[mm]	measuring position 2. axis
(P154)+2	[mm]	measuring position 3. axis

5.1 G69 measuring cycles (continued)

Cycle	Parameter	Calibrate of the sensor in drilling	Calibrate of the sensor at area	Determining of centre a drilling	Determining of positions at area
Cycle selection	P140	1	2	60	40
Safety margin	P141	x	x	x	x
Measuring speed	P143	x	x	x	x
Number of measurem.	P144	x	x	x	x
Measuring tolerance	P147	x	x	x	x
Confidence range	(P147)+4	x	x	x	x
Measuring axis in plane	P148				x
Data of point of trigger	P149	x	x	x	x
Point of trigger XN	(P149)+0	o		x	x
Point of trigger XP	(P149)+1	o		x	x
Point of trigger YN	(P149)+2	o		x	x
Point of trigger YP	(P149)+3	o		x	x
Point of trigger ZN	(P149)+4				
Point of trigger ZP	(P149)+5		o		x
Calibration drilling data	P150	x			
Centre position 1.axis I	(P150)+0	x**	x*		
Centre position 1.axis J	(P150)+1	x**	x*		
Centre position 1.axis K	(P150)+2	x**	x*		
Drilling diameter	(P150)+3	x			
Expected diameter	P152			x	
Output data of the pos.	P154			x	x
Measuring pos. 1. axis	(P154)+0			o**	o*
Measuring pos. 2. axis	(P154)+1			o**	o*
Measuring pos. 3. axis	(P154)+2			o**	o*

x: Parameter must be programmed
o: cycle edition
* : one of these
**: two of these

5.1 G69 measuring cycles (continued)

Message	Meaning
8130	uppersize
8131	undersize
8132	permissible measure difference exceeded
8133	confidence range exceeded
8134	reference drilling smaller than sensor
8135	boring axis is measuring axis
8136	sensor cannot be initiated
8137	P140 incorrectly programs
8138	P141 incorrectly programs
8139	P142 incorrectly programs
8140	P143 incorrectly programs
8141	P144 incorrectly programs
8142	P145/147/149/150 incorrectly programs
8143	empirical value incorrectly programs
8144	P146 incorrectly programs
8145	measuring tolerance 1 incorrectly programs
8146	measuring tolerance 2 incorrectly programs
8147	measuring tolerance 3 incorrectly programs
8148	measuring tolerance 4 incorrectly programs
8149	measuring tolerance 5 incorrectly programs
8150	P148 incorrectly programs
8151	point of trigger XN incorrectly programs
8152	point of trigger XP incorrectly programs
8153	point of trigger YN incorrectly programs
8154	point of trigger YP incorrectly programs
8155	point of trigger ZN incorrectly programs
8156	point of trigger ZP incorrectly programs

5.1 G69 measuring cycles (continued)

Message	Meaning
8157	point of drilling I incorrectly programs
8158	point of drilling J incorrectly programs
8159	point of drilling K incorrectly programs
8160	point of drilling R incorrectly programs
8161	P151 incorrectly programs
8162	P152 incorrectly programs
8163	P153 incorrectly programs
8164	tool not called
8165	G53 not actively
8166	G54 to G59 not actively
8167	sensor actively outside of the safety area
8168	P154 incorrectly programs

5.1 G69 measuring cycles (continued)

Calibration of the sensor in drilling G69 P140:1

The cycle requires a calibration drilling and a sensor. Measuring is executed with the two axes of the selected interpolation plane.

The sensor must be pre-positioned first on the drilling centre point in the selected interpolation plane and the sensor ball within the drilling.

If for example G17 is defined, the centre point of the calibration drilling must in (P150)+0 and (P150)+1 and the diameter in (P150)+3 to be defined.

The cycle positions first the 1. and 2. axis in the interpolation plane to the programmed centre point of the drilling. Now the cycle moves the 1. axis in positive direction to the drilling edge. The axis starts with the feed of the NC program and reduces the feed to F:p143 at the distance of P141 (safety margin) before the expected drilling edge. If a measuring signal before the safety margin occurs, a message is output (M8167).

From the safety margin the cycle moves the axis to the expected drilling edge plus confidence interval. If there is still no signal, a message (M8136) appears.

If there is a signal of the sensor, immediately the axis movement stops and the cycle calculates the point of trigger of the sensor and writes it in (P149)+n (axis movement positively: Point of trigger negatively).

If P144 is > 1, the cycle returns to the safety margin. From here the cycle starts a new measuring movement to the same drilling edge. Afterwards the cycle looks the other side up of the drilling. Become subsequently, the movements with the 2. Axis executed.

If the cycle is finished, then is the data field (P149)+n with the points of trigger of the sensor described. Following measuring cycles can operate with these data. The position of the moved axes now is in the center of the drilling.

Parts it safely,
- that the tool radius of the sensor is correct (P8160).
- to operate in the null point, in which the parameter data (P150) apply.

5.1 G69 measuring cycles (continued)

Example: The calibration of the sensor in drilling G69 P140:1

```
N20     T1   M16
N30     G17 G53  G0   X600.000  Y300.000  Z280.000  A0   B0
N40     G1   Z270     F3000

N100                                                  {Cycle G69 / 1 initialize }
N110    P140:1                                        {Cycle type}
N120    P141:3                                        {Safety margin }
N130    P143:100                                      {Measuring speed }
N140    P144:2                                        {Number of the measurements }
N150    P147:500                                      {Pointer on tolerance data }
N160    P504:5                                        {Confidence range }
N170    P149:510                                      {Pointer on trigger data }
N180    P150:520                                      {Pointer on measuring data }
N182    P520:624.96   P521:324.3      P523:54         {I, J, D }
N230    G69                                           {Cycle call }

N240    G0   Z280     M30
```

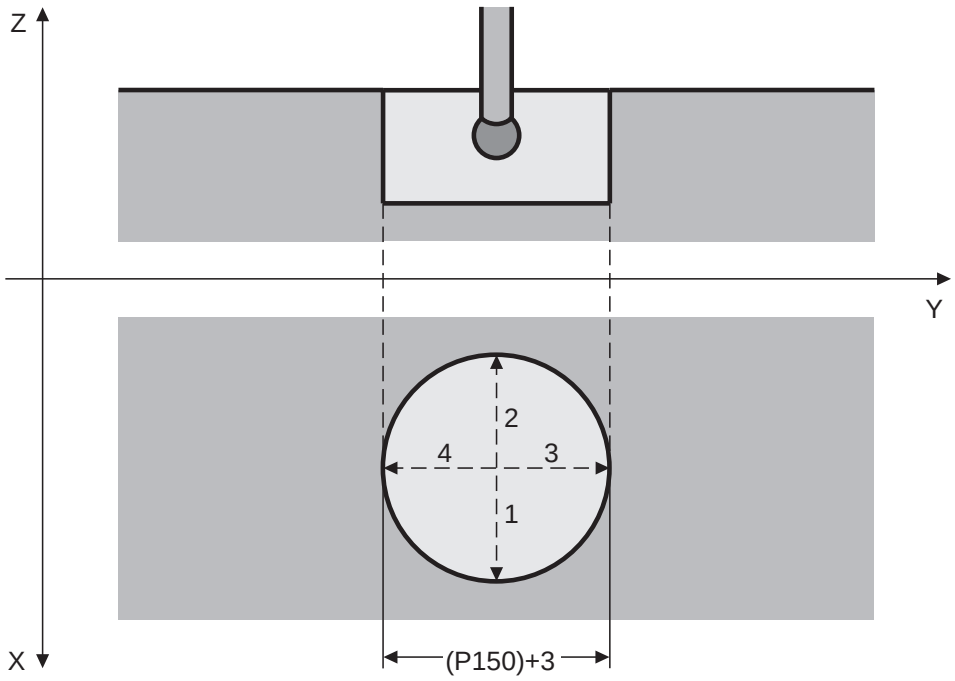

Figure 5-1

5.1 G69 measuring cycles (continued)

Calibration of the sensor at area G69 P140:2

The cycle requires a calibrationing area and a sensor. Measuring becomes with the 3. axis of the selected interpolation plane executed (definition of G17...).
The sensor must be pre-positioned on the calibrationing area.

If for example G17 is defined, the position on the calibrationing area must in (P150)+2 to be defined.

The cycle moves now 3. axis toward the calibrationing area. The axis starts with the feed of the NC program and reduces the feed to F:p143 at the distance of P141 (safety margin) before the expected area. If a measuring signal before the safety margin occurs, a message is output (M8167).

From the safety margin the cycle moves the axis to the expected area plus confidence interval. If there is still no signal, a message (M8136) appears.

If there is a signal of the sensor, immediately the axis movement stops and the cycle calculates the point of trigger of the sensor and writes it in (P149)+n (axis movement positively: Point of trigger negatively).

If P144 is > 1, the cycle turns back to the safety margin. From here the cycle starts a new measuring movement to the same area.

If the cycle is finished, then is the data field (P149)+n with the points of trigger of the sensor described. Following measuring cycles can operate with these data. The position of the moved axis now is in the safety margin.

Parts it safely,
- that the tool radius of the sensor is correct (P8160, P8162).
- to operate in the null point, in which the parameter data (P150) apply.

5.1 G69 measuring cycles (continued)

Example: The calibration of the sensor in drilling G69 P140:2

```
N20     T1   M16
N30     G17 G53  G0   X600.000  Y300.000  Z280.000  A0   B0
N40     G1   Z270      F3000

N100                                          {Cycle G69 / 2 initialize }
N110    P140:2                                {Cycle type}
N120    P141:3                                {Safety margin }
N130    P143:100                              {Measuring speed }
N140    P144:2                                {Number of the measurements }
N150    P147:500                              {Pointer on tolerance data }
N160    P504:5                                {Confidence range }
N170    P149:510                              {Pointer on trigger data }
N180    P150:520                              {Pointer on measuring data }
N182    P522:260.20                           {K }
N230    G69                                   {Cycle call }

N240    G0   Z280      M30
```

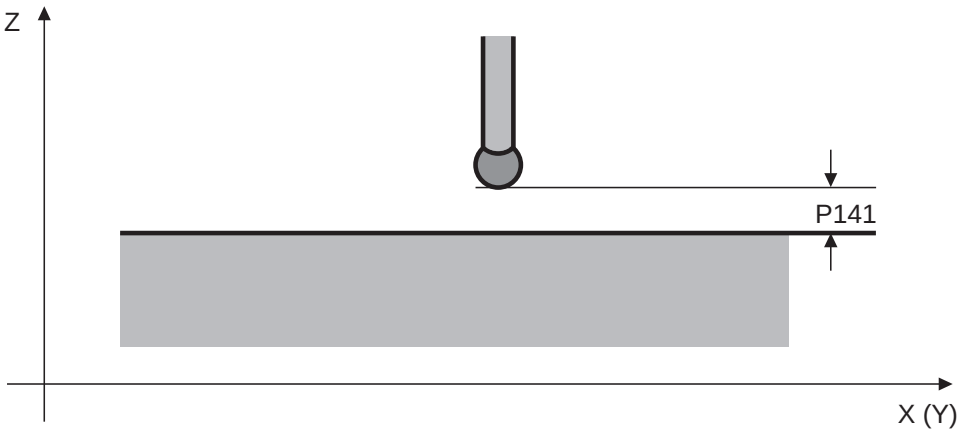

Figure 5-2

5.1 G69 measuring cycles (continued)

Determining the center of a drilling G69 P140:60

The measurement is executed with both axes of the selected interpolation plane (see G17...). The sensor should be at the beginning approximately on the drilling centre point in the selected interpolation plane and the sensor ball within the drilling pre-positioned.

The cycle moves the 1. axis in positive direction to the drilling edge. The axis starts with the feed of the NC program and reduces the feed to F:p143 at the distance
(diameters / 2 - safety margin) = (P152 / 2 - P141).

If a measuring signal before this point occurs, a message is output (M8167).

From here the cycle moves the axis to the expected drilling edge plus confidence interval. If there is still no signal, a message (M8136) appears.

If there is a signal of the sensor, immediately the axis movement stops and the cycle calculates the edge position.

If P144 is > 1, the cycle turns back to the safety margin. From here the cycle starts a new measuring movement to the same drilling edge. Afterwards the cycle looks the other side up of the drilling. Now become the movements with the 2. axis made.

If the cycle is finished, then is the data field (P154)+n with the drilling centre point described. The position of the moved axes now is in the center of the drilling.

Parts it safely that the tool radius of the sensor is correct (P8161, P8163).

5.1 G69 measuring cycles (continued)

Example: The calibration of the sensor in drilling G69 P140:60

```
N20     T1   M16
N30     G17 G53  G0   X600.000  Y300.000  Z280.000  A0   B0
N40     G1   Z270      F3000

N100                                          {Cycle G69 / 60 initialize }
N110    P140:60                               {Cycle type}
N120    P141:3                                {Safety margin }
N130    P143:100                              {Measuring speed }
N140    P144:2                                {Number of the measurements }
N150    P147:500                              {Pointer on tolerance data }
N160    P504:5                                {Confidence range }
N170    P149:510                              {Pointer on trigger data }
N180    P150:520                              {Pointer on measuring data }
N182    P154:530                              {Pointer on output }
N230    G69                                   {Cycle call }

N240    G0   Z280      M30
```

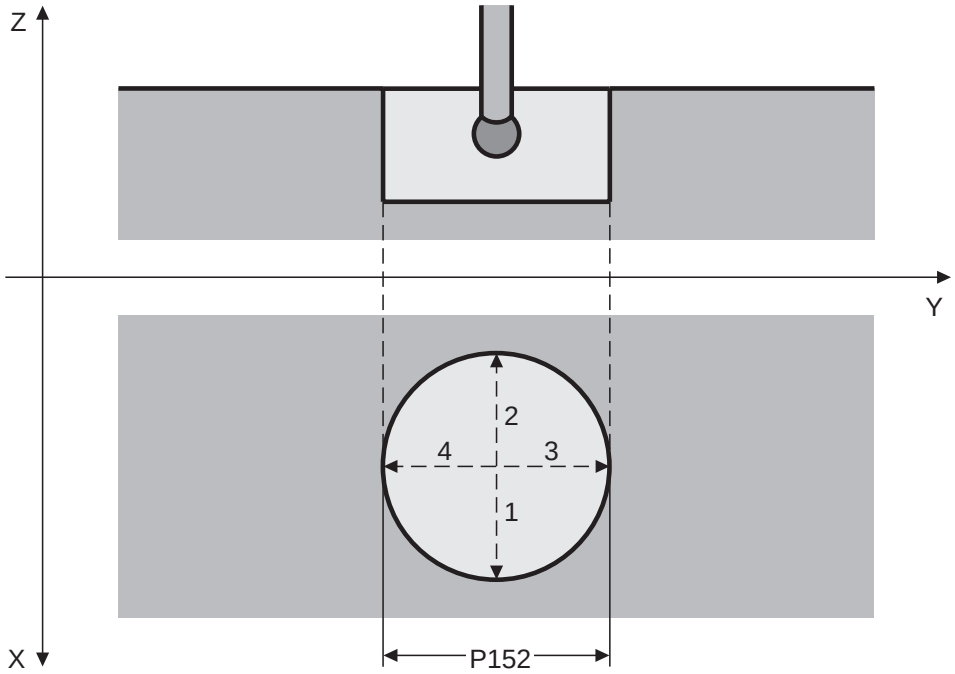

Figure 5-3

5.1 G69 measuring cycles (continued)

determining an individual item in the area G69 P140:40

The sensor must be pre-positioned at the beginning over the contact surface.
The measurement is executed with the axis defined in P148.

The cycle starts into negative direction with measuring speed (P143)
If there is still no signal of the sensor, the axes in the position (initial position - confidence interval (P147)+4) stop and a message (M8136) appears.

If there is a signal of the sensor, immediately the axis movement stops and the cycle calculates the position on the area and writes the data in (P154)+n.

Example G17	with P148:3	writes after cycle	P(154)+2
	with P148:2	writes after cycle	P(154)+1
	with P148:1	writes after cycle	P(154)+0

If P144 is > 1, the cycle turns back to the safety margin. From here the cycle starts a new measuring movement to the same area.

The position of the moved axes is in the safety margin.

Parts it safely,
- that the tool length of the sensor is correct (P8161, P8163), if 3. axis is selected
- that the tool radius of the sensor is correct (P8160, P8162), if 1. or 2. axis is selected

5.1 G69 measuring cycles (continued)

Example: The calibration of the sensor in drilling G69 P140:40

```
N20   T1   M16
N30   G17 G53  G0   X600.000  Y300.000  Z280.000  A0   B0
N40   G1   Z270      F3000

N100                                          {Cycle G69 / 40 initialize }
N110  P140:40                                 {Cycle type}
N120  P141:3                                  {Safety margin }
N130  P143:100                                {Measuring speed }
N140  P144:2                                  {Number of the measurements }
N150  P147:500                                {Pointer on tolerance data }
N160  P504:5                                  {Confidence range }
N170  P144:3                                  {Measuring axes }
N180  P149:510                                {Pointer on trigger data }
N182  P150:520                                {Pointer on measuring data }
N230  G69                                     {Cycle call }

N240  G0   Z280      M30
```

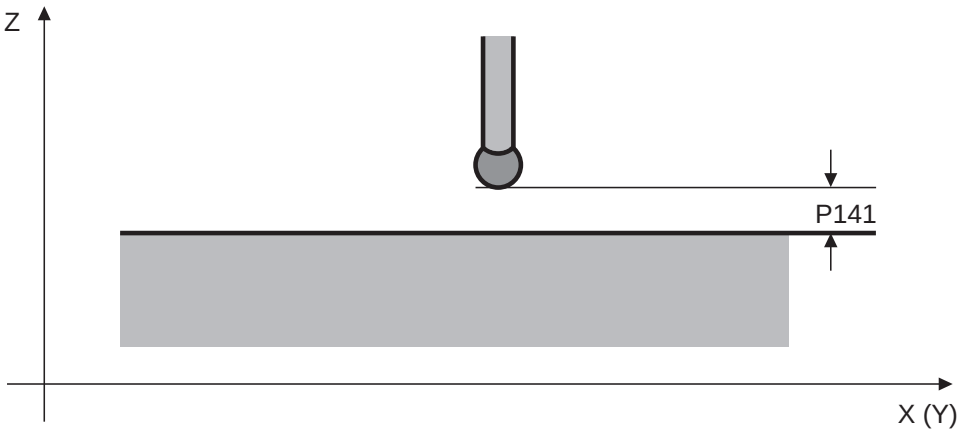

Figure 5-4

5.2 G71 / G72 / G73 / G74 / G75 Milling cycles

List of canned milling cycles

G71 Rectangular pocket roughing, conventional
G72 Rectungular pocket roughing, conventional and climb
G73 Rectangular pocket roughing and finishing
G74 Circular pocket roughing
G75 Circular pocket roughing and finishing

The millinging cycles are block by block effective.
The parameter inputs remain against it modal effectively.

In order to avoid over regulation, all parameters are to be reset, with a programming type not to be used.

Messages

The numbers for cycle messages start with 8xxx.
A list can be found in section messages

5.2.1 G71 Rectangular pocket roughing, conventional

Starting point definition

Before starting any machining cycles, the milling cutter should be above the workpiece by the safety margin and at the centre of the pocket.

Whether the starting point S1 or S2 is homed into depends on the sign of the programmed in-feed P16.
Positive sign: starting point S1
Negative sign: starting point S2

The milling direction is established by the CNC and always in the direction of the longer pocket side. If the pocket sides are the same, milling takes place in the X direction.

Parameter input

All parameters except for P16 should have positive sign when inputting.

Parameter	Significance
P11	Pocket dimension in X direction
P12	Pocket dimension in Y direction
P13	Pocket depth in Z direction
P14	Corner radius
P15	Contour allowance in X and Y directions
P16	In-feed dimension in X or Y directions (prefix + or -)
P17	In-feed dimension in Z direction
P18	Pocket depth allowance in Z direction
P19	Safety margin in Z direction
P21	Feed rate in Z direction (when plunging into material)

During cycle processing a possibly programmed correction becomes G41 / G42 switched off, since the tool radius correction in the cycle is considered.

Feed in z-direction (only when immersing into the material) is programmable with P21.
Is P21 programmed, does not become immersing into the material with under F programmed feed executed.

5.2.1 G71 Rectangular pocket roughing, conventional (continued)

Machining sequence

- Home in on starting point S1 or S2 (depending on prefix of P16)
- In-feed in Z direction at feed rate P21 (if programmed)
- Conventional milling to pocket dimension less allowance P15 in X and Y directions
- Retraction of milling cutter by 2 mm in Z direction
- Retraction in X or Y direction
- In-feed in Z direction
- In-feed in X or Y direction
- In-feed repetition until pocket dimension less allowance P15 is reached
- Milling over burrs
- Retraction in Z direction to safety margin
- Retraction to starting position in XY plane
- Repetition of in-feeds until pocket depth less allowance P18 is reached
- Retraction to starting position (centre of pocket)

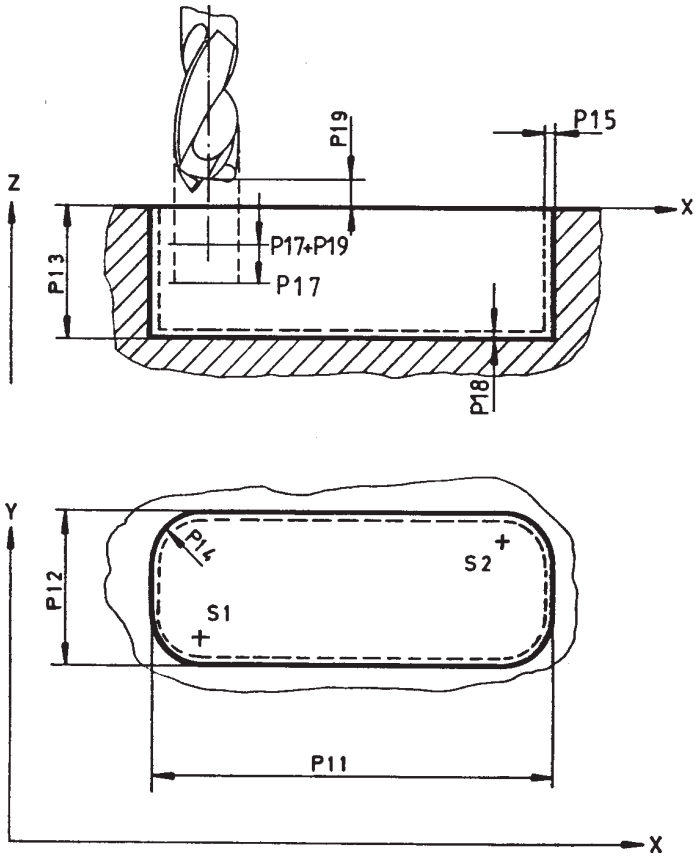

Figure 5-5

5.2.1 G71 Rectangular pocket roughing, conventional (continued)

G 71 Example 1: Pocket dimensions X=71mm, Y=41mm, Z=5mm; Tool-radius=3mm

N10	G00	G54	G90	F2500	S900	T02	M06	M03	M07
N20	G71 P14:--	X55,5 P15:0	Y40,5 P16:4	Z2 P17:3	P11:71 P19:2	P12:41 P21:100	P13:5		M30

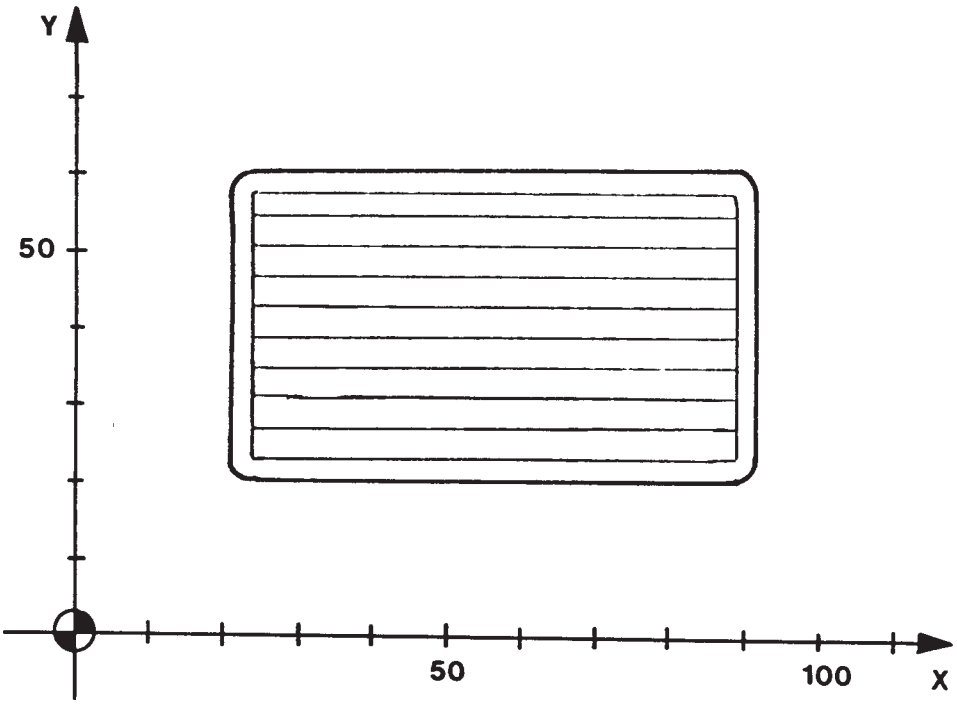

Figure 5-6

Parameters:

P11, P12, P13	Pocket dimensions in X, Y, Z direction
P16, P17	In-feed in Y, Z direction
P19	Safety margin in Z direction
P21	Feed rate in Z direction

Run: Block 10 Zero point, absolute dimension, feed rate, spindle, tool
Block 20 Cycle call-up, positioning on centre of pocket X, Y, Z, pocket dimensions P11, P12, P13, in-feed dimensions P16, P17, safety margin P19

Remarks: Since the corner radius P14 is not programmed, it becomes with the programmed tool radius equated.

5.2.1 G71 Rectangular pocket roughing, conventional (continued)

G71 Example 2: Pocket dimensions X=46mm, Y=81mm, Z=30mm;
Tool-radius=3mm, corner radius=13mm

N10	G00	G54	G90	F1000	S750	T02	M06	M03	M07
N20	G71 P14:13	X43 P16:-4	Y60,5 P17:7	Z1 P19:1	P11:46 P21:400	P12:81 M30		P13:30	

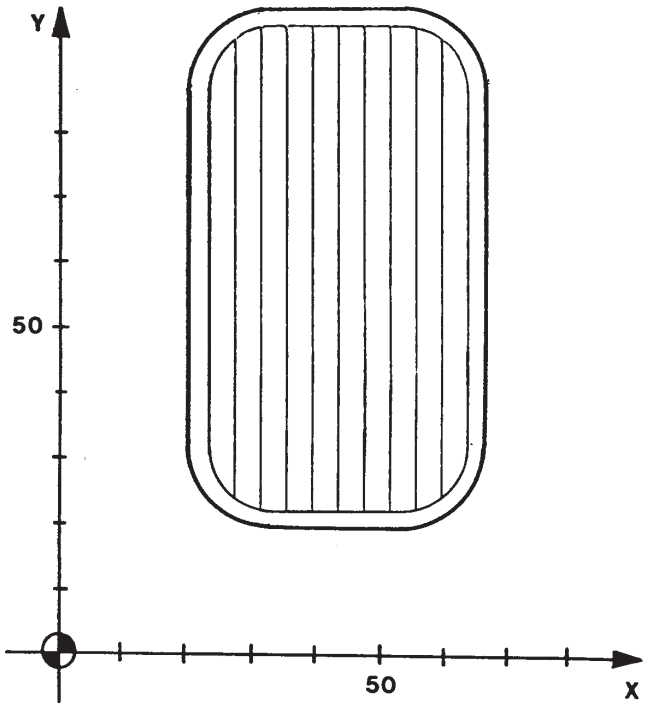

Figure 5-7

Parameters:

P11, P12, P13	Pocket dimensions in X, Y, Z direction
P14	Corner radius
P16, P17	In-feed in X, Z direction
P19	Safety margin in Z direction
P21	Feed rate in Z direction

Run:	Block 10	Zero point, absolute dimension, feed rate, spindle, tool
	Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z; pocket dimensions P11, P12, P13, in-feed dimensions P16, P17, corner radius P14, safety margin P19

Remarks: If P14 = 0 is or smaller than the tool radius, then becomes after that start clearing the program execution stopped and message 8001 indicated.

5.2.2 G72 Rectangular pocket roughing, climb and conventional

Starting point definition

Before the start of the machining cycles the milling cutter should be above the workpiece by the safety margin and in the middle of the pocket.

Whether the starting point S1 or S2 is homed into depends on the sign of the programmed in-feed P16.
Sign, positive: starting point S1
Sign, negative: starting point S2

The determination of the milling direction takes place in the CNC and always in the direction of the longer pocket side. If the pocket sides are the same, milling takes place along the X direction.

Parameter input

All parameters except for P16 should have a positive sign during inputting.

Parameter	Significance
P11	Pocket dimension in X direction
P12	Pocket dimension in Y direction
P13	Pocket depth in Z direction
P14	Corner radius
P15	Contour allowance in X and Y directions
P16	In-feed dimension in X or Y directions (sign +/-)
P17	In-feed dimension in Z direction
P18	Pocket depth allowance in Z direction
P19	Safety margin in Z direction
P21	Feed rate in Z direction (when plunging into material)

5.2.2 G72 Rectangular pocket roughing, climb and conventional (continued)

Machining sequence

- Home in on starting point S1 or S2 (each according to sign of P16)
- In-feed in Z direction at feed rate P21 (if programmed)
- Conventional milling to pocket dimension less the allowance P15 in X and Y direction
- In-feed in X or Y direction
- Milling in conventional direction
- In-feed in X or Y direction
- In-feed repetition until pocket dimension less allowance P15 is reached
- Milling over burrs along edge
- Retraction in Z direction to safety margin
- Retraction to starting position in XY plane
- In-feed in Z direction to previous dimension
- Milling over burrs along other edge
- Retraction in Z direction to safety margin
- Retraction to starting position in XY plane
- Repetition of in-feeds until pocket depth less allowance P18 is reached
- Retraction to starting position (centre of pocket)

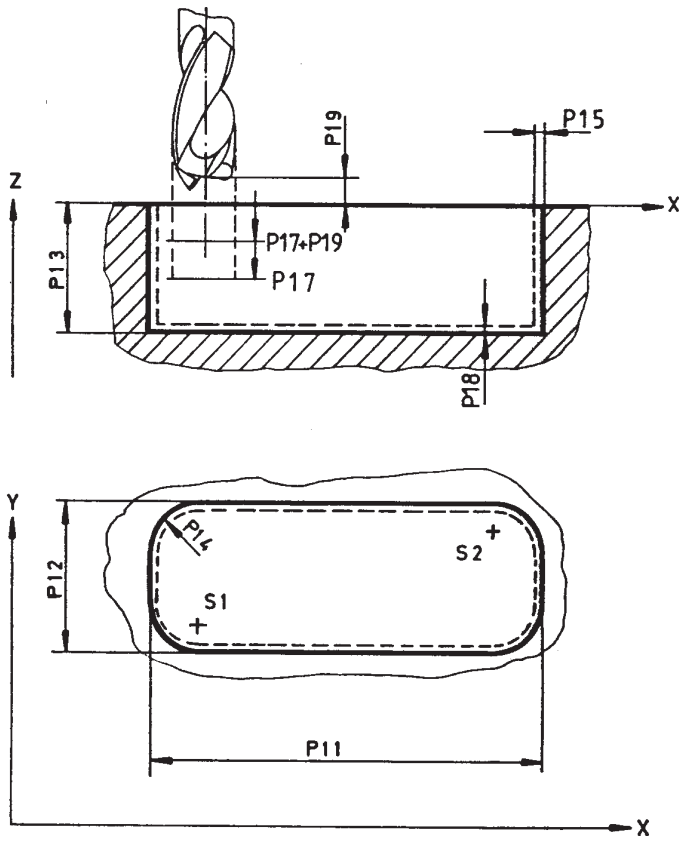

Figure 5-8

5.2.2 G72 Rectangular pocket roughing, climb and conventional milling (cont.)

G72 Example 1: Pocket dimensions X=66mm, Y=31mm, Z=5mm; Tool-radius=3mm

```
N10  G00  G54  G90      F2000     S850      T02       M06  M03  M07

N20  G72  X53  Y35,5    Z1        P11:66    P12:31    P13:5     P14:--     P15:0
     P16:5,1   P17:3    P19:1     M30
```

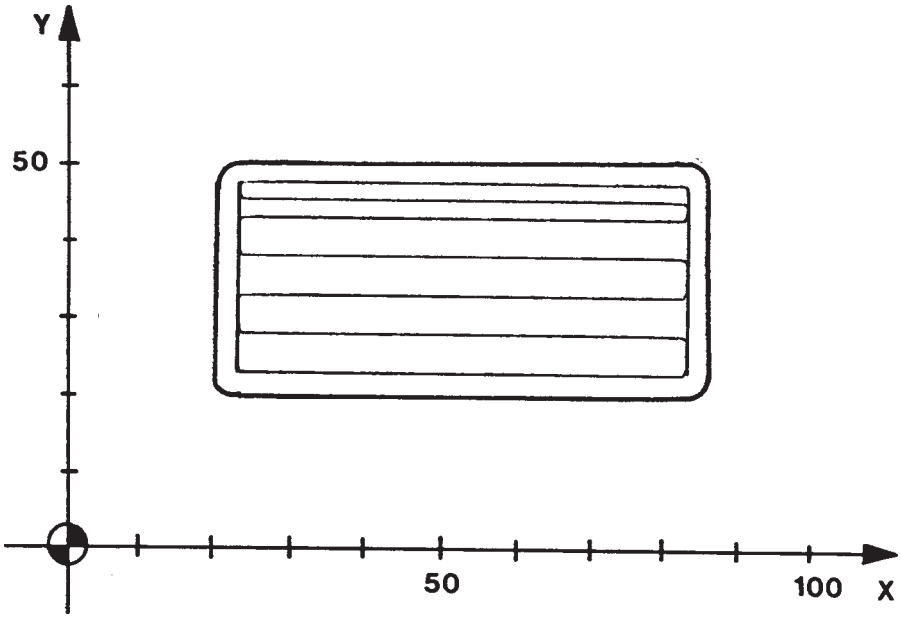

Figure 5-9

Parameters:

P11, P12, P13	Pocket dimensions in X, Y, Z direction
P16, P17	In-feed in Y, Z direction
P19	Safety margin in Z direction

Run:	Block 10	Zero point, absolute dimension, feed rate, spindle, tool
	Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z; pocket dimensions P11, P12, P13, in-feed dimensions P16, P17, safety margin P19

Remarks: As the corner radius P14 is not programmed it is identified along with the programmed tool radius.

5.2.2 G72 Rectangular pocket roughing, climb and conventional milling (cont.)

G72 Example 2: Pocket dimensions X=44mm, Y=69mm, Z=3mm; Tool radius=3mm, corner radius=15mm

```
N10  G00  G54  G90      F2000     S950      T02       M06       M03       M07

N20  G72  X42  Y54,5    Z1        P11:44    P12:69    P13:3     P14:15    P15=0
     P16:5,5  P17:3     P19:1     P21:100   M30
```

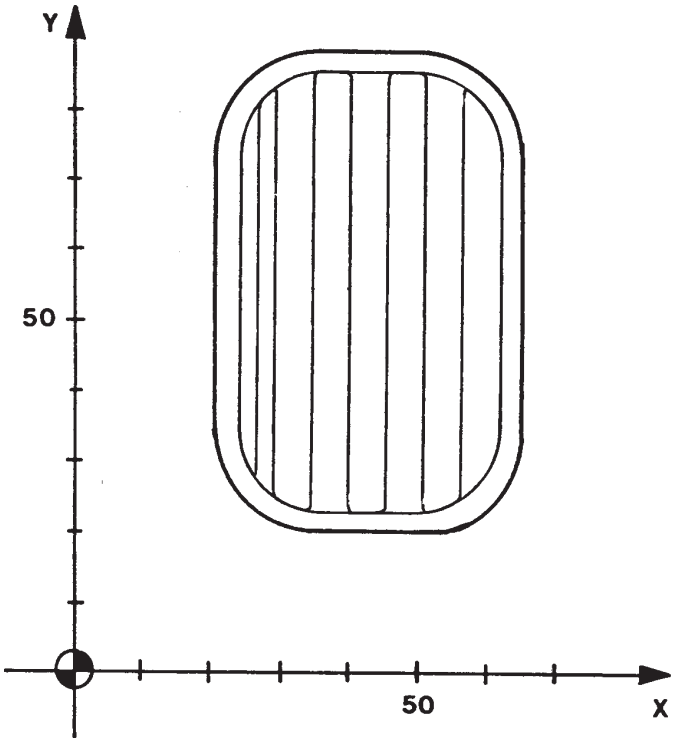

Figure 5-110

Parameters:

P11, P12, P13	Pocket dimensions in X, Y, Z direction
P14	Corner radius
P16, P17	In-feed in X, Z direction
P19	Safety margin in Z direction
P21	Feed rate in Z direction

Run:	Block 10	Zero point, absolute dimension, feed rate, spindle, tool
	Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z; pocket dimensions P11, P12, P13, in-feed dimensions P16, P17, corner radius P14, safety margin P19

Remarks: If P14=0 or is smaller than the tool radius the program run is shut down after the start and fault feed-back 1301 (geometry fault) is displayed.

5.2.3 G73 Rectangular pocket roughing and finishing

Starting point definition

Before the start of the machining cycles, the milling cutter should be above the workpiece at the safety margin and in the middle of the pocket.

Whether the starting point S1 or S2 is homed into depends on the sign of the programmed in-feed P16.
Sign, positive: starting point S1
Sign, negative: starting point S2

The milling direction is established in the CNC and is always in the direction of the longer pocket side. If the pocket sides are the same, milling takes place along the X direction.

Parameter input

All parameters except for P16 should have a positive sign when inputting.

Parameter	Significance
P11	Pocket dimension in X direction
P12	Pocket dimension in Y direction
P13	Pocket depth in Z direction
P14	Corner radius
P15	Contour allowance in X and Y direction
P16	In-feed dimension in X or Y direction (sign +/-)
P17	In-feed dimension in Z direction
P18	Pocket depth allowance in Z direction
P19	Safety margin in Z direction
P20	In-feed dimension when finishing, to remove allowance P15; In-feed direction as for P16
P21	Feed rate in Z direction (when plunging into material)

5.2.3 G73 Rectangular pocket roughing and finishing (continued)

Machining

- Home in on starting point S1 or S2 (each according to sign of P16)
- In-feed in Z direction at feed rate P21 (if programmed)
- Milling pocket contour (less allowance P15) to starting point
- In-feed repetition until pocket depth P13 less allowance P18 is reached
- Retraction to starting plane
- In-feed in X or Y direction
- In-feed in Z direction at feed rate P21 (if programmed)
- Conventional and climb milling of remaining material
- Retraction in Z direction and to starting position and in-feed
- In-feed repetition until pocket dimension less allowance P18 is reached.
- Retraction in Z direction to starting plane
- In-feed for finishing
- In-feed in Z direction at 0.5 of the programmed feed rate F to pocket depth less P18
- Finishing contour to starting point at 0.5 of the programmed feed rate
- In-feed repetition until finished contour is reached
- Retraction to starting position at programmed feed rate

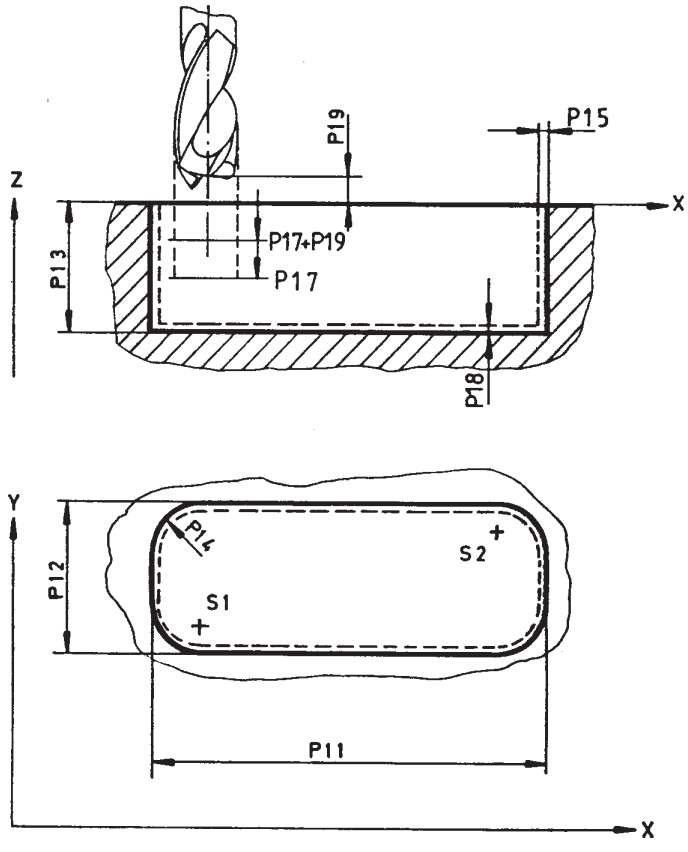

Figure 5-11

5.2.3 G73 Rectangular pocket roughing and finishing (continued)

G73 Example 1: Pocket dimensions X=64mm, Y=35mm, Z=5mm; Tool radius=3mm

```
N10  G00  G54      G90      F1500     S950     T02      M06      M03      M07

N20  G73  X52,5    Y37,5    Z1        P11:64   P12:35   P13:5    P14=0
     P15:1         P16:4,5        P17:2     P19:1    P20:1    M30
```

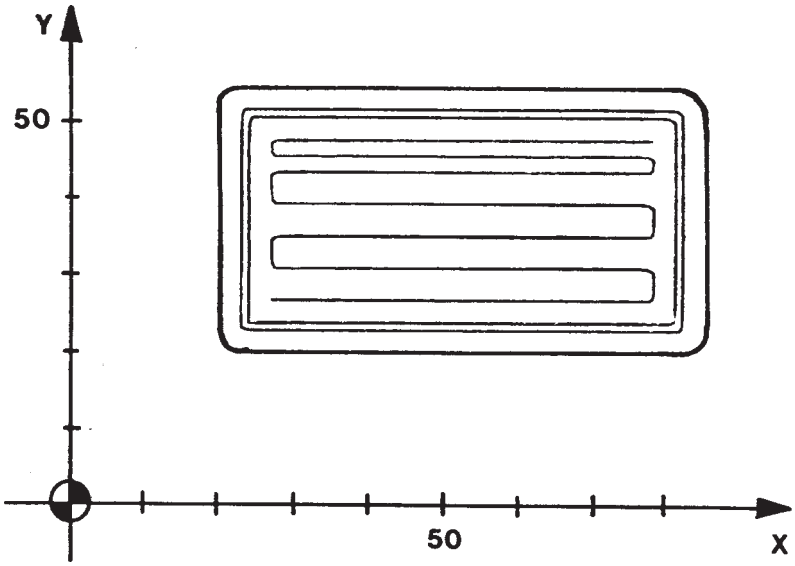

Figure 5-12

Parameters:

P11, P12, P13	Pocket dimensions in X, Y, Z direction
P15	Allowance in X and Y direction
P16, P17	In-feed in Y, Z direction
P19	Safety margin in Z direction
P20	Allowance during finishing

Run:

Block 10	Zero point, absolute dimension, feed rate, spindle, tool
Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z; pocket dimensions P11, P12, P13, in-feed dimensions P16, P17, allowance P15, safety margin P19, in-feed dimension when finishing P20

Remarks: As the corner radius P14 is not programmed it is made equal to the programmed tool radius.

5.2.3 G73 Rectangular pocket roughing and finishing (continued)

G73 Example 2: Pocket dimensions X=65mm, Y=35mm, Z=5mm; Tool radius=3mm, corner radius=11mm

N10	G00	G54	G90	F1500	S950	T02	M06	M03	M07
N20	G73	X52,5	Y37,5	Z1	P11:65	P12:35	P13:5	P14:11	
	P15:1,5		P16:4,2	P17:2	P19:1	P20:1	M30		

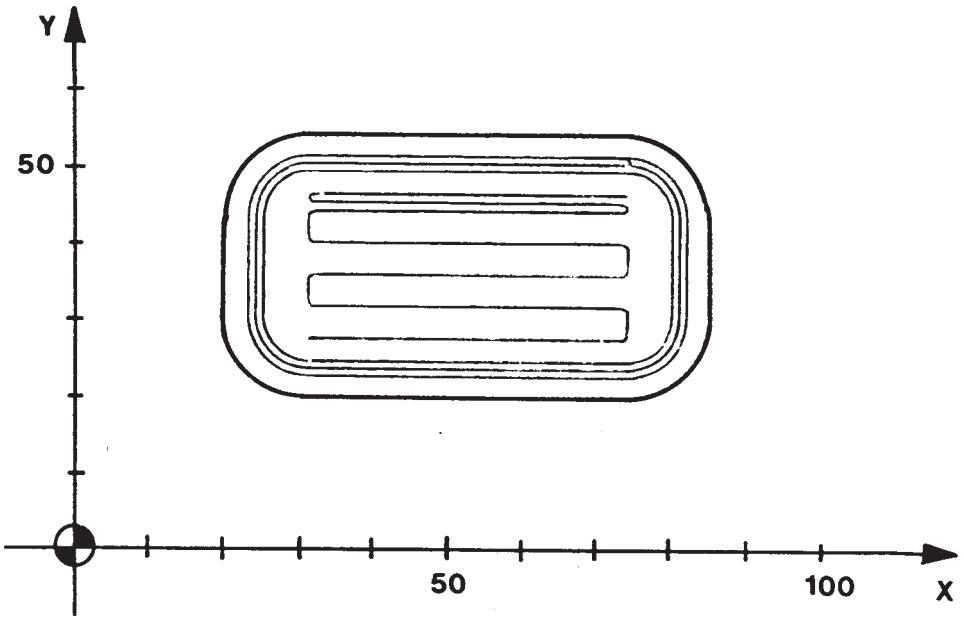

Figure 5-13

Parameters:

P11, P12, P13	Pocket dimensions in X, Y, Z direction
P14	Corner radius
P15	Allowance in X and Y direction
P16, P17	In-feed in Y, Z direction
P19	Safety margin in Z direction
P20	Allowance during finishing

Run:

Block 10	Zero point, absolute dimension, feed rate, spindle, tool
Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z; pocket dimensions P11, P12, P13, in-feed dimensions P16, P17, corner radius P14, safety margin P19, allowance P15, infeed dimension during finishing P20

Remarks: If P14=0 or is smaller than the tool radius the program run is shut down after the start and message 1301 (geometry fault) is displayed.

5.2.4 G74 circular pocket roughing

Starting point definition

Before the start of the machining cycles the milling cutter should be above the workpiece at the safety margin and at the centre of the pocket.

Parameter input

All parameters except for P16 should have a positive sign when inputting. The sign of P16 determines the direction of milling, i.e.:
Milling direction G02, the sign of P16 is positive
Milling direction G03, the sign of P16 is negative

Parameter	Significance
P11	Internal radius (if core drilled)
P12	Depth of pocket
P14	Pocket radius
P15	Allowance at circumference
P16	In-feed dimension in X direction
P17	In-feed dimension in Z direction
P18	Pocket depth allowance in Z direction
P19	Safety margin
P21	Feed rate in Z direction (when plunging into material)

When specifying the internal radius P11, the first in-feed in the X direction is corrected by the appropriate amount: 1st in-feed = P11 + P16.

5.2.4 G74 circular pocket roughing (continued)

Machining

- In-feed in Z direction at feed rate P21 (if programmed)
- In-feed in X direction
- Spiral milling
- Milling circular pocket to finished size less allowance P15
- Retraction along the semicircle with R=2.5mm max.
- Retraction in Z direction by 2mm
- Retraction to centre of pocket
- In-feed repetition until pocket dimension less allowance P18 in Z direction is reached
- Retraction to starting position

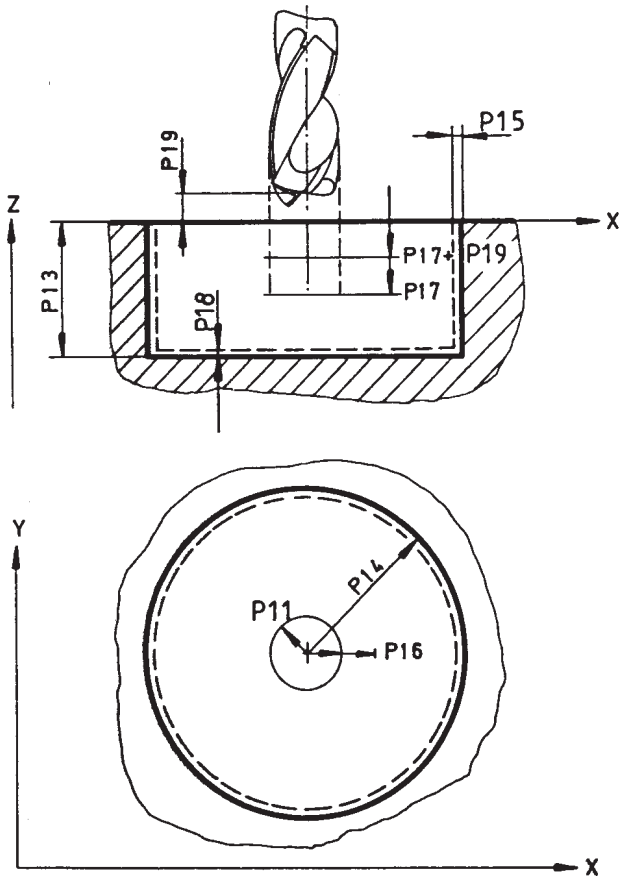

Figure 5-14

5.2.4 G74 Circular pocket roughing (continued)

G74 Example 1: Pocket radius R=33mm, pocket depth Z=27mm; Tool radius=3mm

```
N10  G00  G54  G90  F1500     S950     T02     M06     M03     M07
N20  G74  X53  Y53  Z1        P13:27   P14:33  P16:5,1 P17:5   P19:1   M30
```

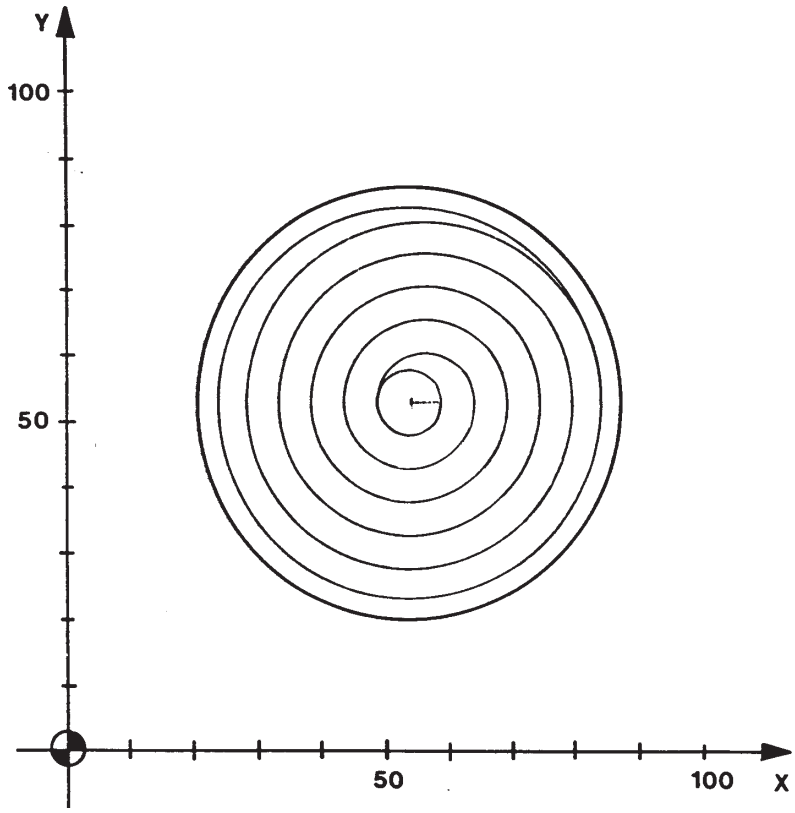

Figure 5-15

Parameters:

P13	Pocket depth in Z direction
P16, P17	In-feed in X, Z direction
P14	Pocket radius
P19	Safety margin in Z direction

Run:	Block 10	Zero point, absolute dimension, feed rate, spindle, tool
	Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z, pocket depth P13, in-feed dimensions P16, P17, pocket radius P14, safety margin P19

5.2.4 G74 circular pocket roughing (continued)

G74 Example 2: Pocket radius R=30mm, pocket depth Z=31mm; Tool radius=3mm, core bore drilling at 20mm dia.

```
N10  G00  G54  G90  F1500     S950     T02     M06     M03     M07

N20  G74  X50  Y50  Z1        P11:10   P13:31  P14:30  P16:4   P17:8
     P19:1      M30
```

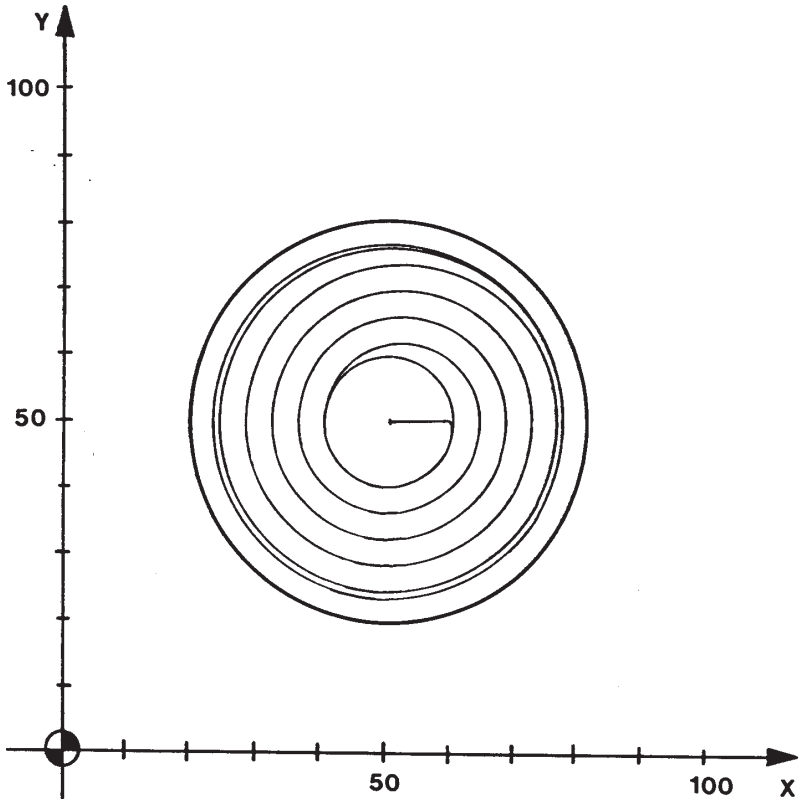

Figure 5-16

Parameters:

P11	Internal radius (if core drilled)
P13	Pocket depth in Z direction
P16, P17	In-feed in X, Z direction
P14	Pocket radius
P19	Safety margin in Z direction

Run:	Block 10	Zero point, absolute dimension, feed rate, spindle, tool
	Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z, pocket depth P13, in-feed dimensions P16, P17, pocket radius P14, safety margin P19

5.2.5 G75 circular pocket roughing and finishing

Starting point definition

Before the start of the machining cycles the milling cutter should be above the workpiece at the safety margin and at the centre of the pocket.

Parameter input

All parameters except for P16 should have a positive sign when inputting. The sign of P16 determines the direction of milling, i.e.:
Milling direction G02, the sign of P16 is positive
Milling direction G03, the sign of P16 is negative

Parameter	Significance
P11	Internal radius (if drilled)
P13	Depth of pocket
P14	Pocket radius
P15	Allowance at circumference
P16	In-feed dimension in X direction
P17	In-feed dimension in Z direction
P18	Pocket depth allowance in Z direction
P19	Safety margin
P20	In-feed dimension when finishing, to remove P15, in-feed direction as with P16
P21	Feed rate in Z direction (when plunging into material)

With specification of the internal radius P11 the first feed in x-direction becomes around the suitable amount corrects: 1. Feed = P11 + P16.

5.2.5 G75 circular pocket roughing and finishing (continued)

Machining

- In-feed in Z direction at feed rate P21 (if programmed)
- In-feed in X direction
- Spiral milling
- Milling circular pocket to finished size less allowance P15
- Retraction along the semicircle with R=2.5mm max.
- Retraction in Z direction
- Retraction to centre of pocket
- In-feed repetition until pocket dimension less allowance P18 in Z direction is reached
- Retraction along the semicircle with R=2.5mm max.
- In-feed at half feed rate for finishing along the semicircle
- Milling along the circle at half of the programmed feed rate
- Retraction along the semicircle
- In-feed repetition until finished dimension is reached
- Retraction along the semicircle
- Retraction to starting position at programmed feed rate

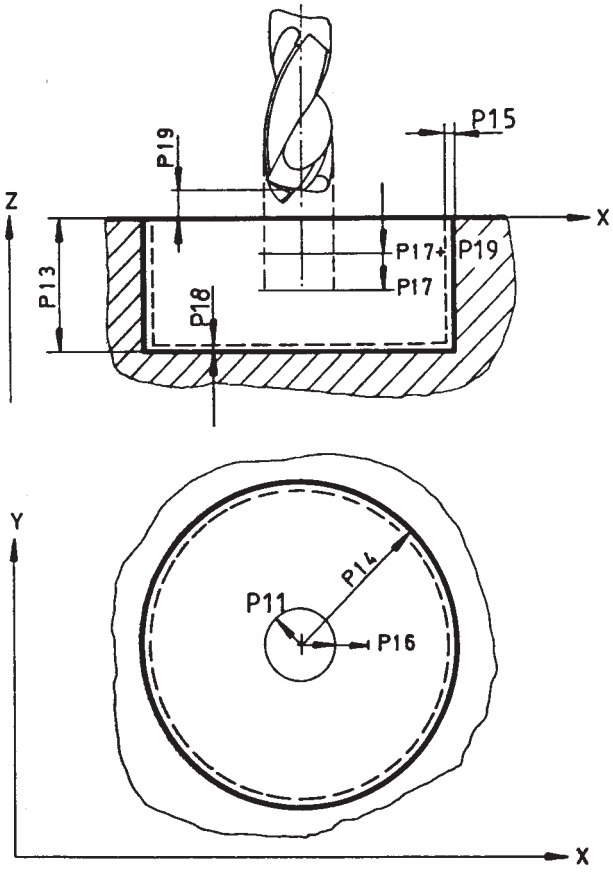

Figure 5-17

5.2.5 G75 circular pocket roughing and finishing (continued)

G75 Example 1: Pocket radius R=28mm, pocket depth Z=35mm; Tool radius=3mm

```
N10  G00  G54  G90  F1500      S950      T02       M06       M03       M07

N20  G75  X48  Y48  Z1         P13:35    P14:28    P15:2     P16:4,1   P17:8
     P19:1       P20:2       M30
```

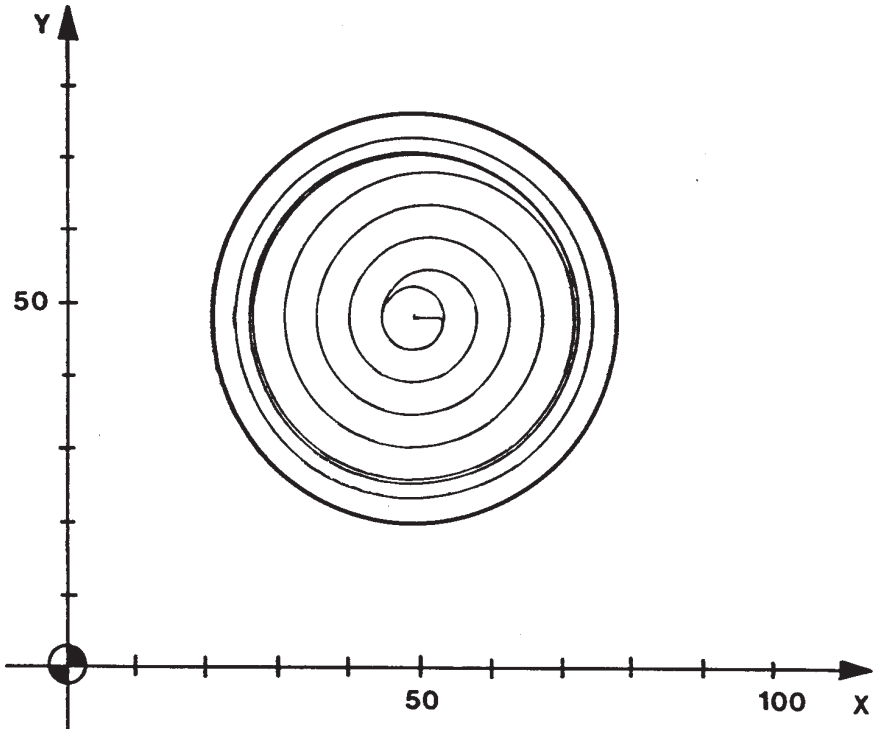

Figure 5-18

Parameters:

P13	Pocket depth in Z direction
P14	Pocket radius
P15	Allowance at circumference
P16, P17	In-feed in X, Z direction
P19	Safety margin in Z direction
P20	In-feed dimension during finishing

Run:	Block 10	Zero point, absolute dimension, feed rate, spindle, tool
	Block 20	Cycle call-up, positioning on centre of pocket X, Y, Z, pocket depth P13, in-feed dimensions P16, P17, pocket radius P14, safety margin P19, in-feed dimension during finishing P20

5.3 G80 / G81 / G83 / G84 / G85 Drilling cycles

the drilling cycle functions G81 / G83 / G84 / G85 operate modally and overwrite each other mutually. G80 clears all drilling cycles resulting in G00 (rapid traverse) and G64 (clearing feed rate 100%) becoming effective automatically.

G80 is automatically set at the program start.

List of drilling cycles

Drilling cycle according to DIN 66025	Programming in CNC900
G81 Drilling with rapid return	G81
G82 Drilling with rapid return and	G81, G04. Free cutting time free-cutting
G83 Deep hole drilling	G83
G84 Tapping	G84, G04. Spindle turnaround time
G85 Drilling with return at feed rate	G85
G86 Drilling with rapid return, free cutting and spindle „off“	G81, G04. Free cutting time, M05
G87 Drilling with manual return	-
G88 Drilling with manual return and free cutting	-
G89 Drilling with return at feed rate and	G85, G04. Free cutting time Further possibilities with the CNC900, e.g.: G83, G04. Free cutting time G83, G04. Free cutting time, M05

5.3 G80 / G81 / G83 / G84 / G85 Drilling cycles

Programming of characteristics

for the programming of the drill cycles only parameters can be used.

Parameter	Meaning		
P30	Free cutting time	Turn back time of the spindle after achieving the drilling depth	
P31	Compensating chuck	G84 P31 : 3 = thread cutting without compensating chuck right P31 : 4 = thread cutting without compensating chuck left P31 : 0 or - = G84 standard cycle (with compensating chuck)	
P32	Drilling feed		
P32	Thread pitch	bei G84	
P33	Drilling depth		absolute
P33	Thread depth	bei G84	absolute
P34	Anticipation plane	Safety margin of the processing upper edge	absolute
P35	Retreat plane	To run around of obstacles in the setting axis	absolute
P36	No. of strokes	G83, constant setting depth = (P33 - P34) / P36	
P36	Setting depth	G83, degressive setting depth	
P37	1st Stroke	G83, degressive setting depth	
P38	Safety margin	G83, presetting = 1mm	incremental
P39	Gradual decrease	G83, degressive setting depth P39 : 1 = on P39 : 0 or - = off	

Drilling depth P33, anticipation plane P34 and retreat plane P35 are dependant of the planes set over G17, G18 or G19. In the following examples, G17 is activated with the axis names X (1st axis), Y (2nd axis) and Z (3rd axis).

5.3 G80 / G81 / G83 / G84 / G85 Drilling cycles (continued)

Parameters for drilling cycles

Drilling cycle	Parameter									
	Free cutting time	Drilling feed	Drilling depth	Anticipation plane	Retreat plane	Number of strokes	1st stroke	Safety margin	Gradual decrease	Driling speed
	P30	P32	P33	P43	P35	P36	P37	P38	P39	S
G81	*	*	*	*	*	-	-	-	-	*
G83 constant	*	*	*	*	*	*	-	*	-	*
G83 degressiv	*	*	*	*	*	*	*	*	*	*
G84	*	*	*	*	*	-	-	-	-	*
G85	*	*	*	*	*	-	-	-	-	*

When calling up a drilling cycle, a direction of spindle rotation and spindle speed should be active. If these two values have already been set when the cycle is called up they are taken as the preset values for the drilling cycles.

Programming of a safety margin (P38) is optional. If this characteristic is not input a safety margin of 1 mm is preset.

When calling up a drilling cycle, the fields marked * should have been programmed in one way or another.

Messages

The numbers for cycle messages start with 8xxx.
A list can be found in section messages

Activation conditions

For a drilling cycle to be carried out, at least one of the X, Y or Z coordinates should be programmed per main block. This also applies to parameter P33 for the drilling depth Z.

5.3 G80 / G81 / G83 / G84 / G85 Drilling cycles (continued)

Positioning speed

The pre-positioning of the axes takes place at rapid traverse. When using polar coordinates programming with interpolation mode G02 or G03 set, positioning is at the programmed feed rate. It is therefore good policy to specify the feed rate with parameter P32.

Effect of G91 incremental dimensions

Using incremental dimensions, the positioning axes (XY with G17, ZX with G18, YZ with G19) are traversed in incremental mode. The specified coordinates are always taken as absolute for the characteristics (Z, R, K).

Effect of G66

A programmed G66 is effective throughout the complete drilling cycle.

Effect of G63/G64

The function, when set, is retained throughout the complete drilling cycle. The tapping cycle G84 automatically sets G63 for period of the in-feed traverse.

Effect of other additional functions

Other additonal functions are effective in a block with travel information (cycle is executed) before or after the travels that are defined in the cycle.

5.3.1 G81 Drilling with return at rapid travers

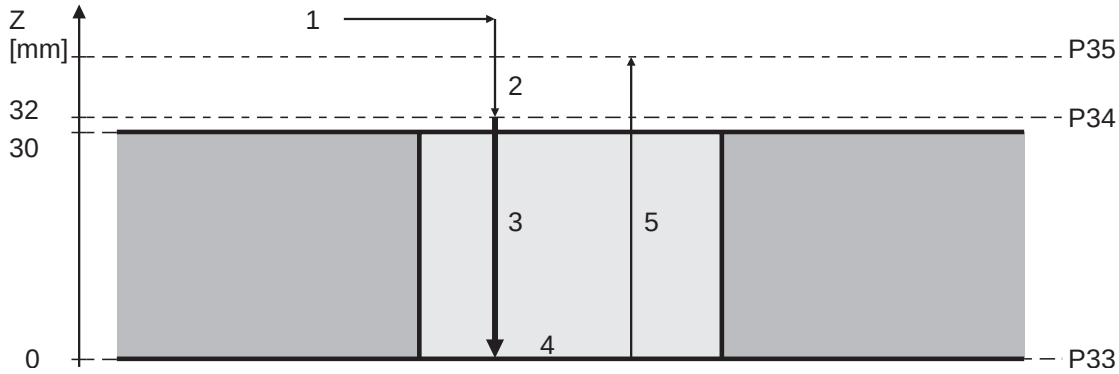

Figure 5-19

Meaning of the parameters

P30	Free cutting time	Turn back time of the spindle after achieving the drilling depth
P32	Drilling feed	
P33	Drilling depth	
P34	Anticipation plane	Safety margin of the processing upper edge
P35	Retreat plane	To go around of obstacles in the setting axis

5.3.1 G81 Drilling with return at rapid travers (continued)

Example

```
N20  G56  G00     X0   Y0   Z100     S1200    F1500    M03      T01  M16
N30  G81  Z80     P30:1     P32:800  P33:0    P34:32   P35:60
N40  M05  ...
```

Sequence plan

1 Switch on spindle (M03, M04) and position X and Y axis in rapid traverse.

2 After reaching the X and Y position, position the Z axis to the anticipation plane (P34) in rapid traverse.

3 After reaching the anticipation plane the Z axis is traversed at working feed rate (P32) to the drilling depth (P33).

4 After reaching the drilling depth any possibly programmed free cutting time (P30) is awaited.

5 After it the Z-axis in rapid traverse moves on the retreat plane (P35) and afterwards the spindle is switched off (M05).

5.3.2 G83 Deep-hole drilling with constant setting depth

If the parameter P39:0 or is not programmed, the programmed drilling depth (P33) by repeated constant setting depth of the active drilling depth is achieved.

The active drilling depth is (P33 - P34) * n / P36 , however n = 1, 2, .. (settin depth = stroke) is.

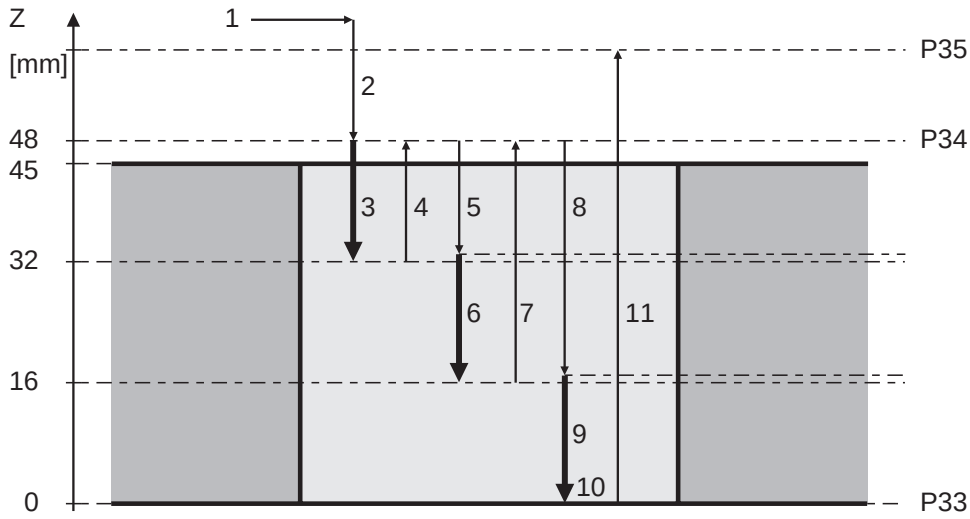

Figure 5-20

Meaning of the parameters

P30	Free cutting time	Turn back time of the spindle after achieving the drilling depth
P32	Drilling feed	
P33	Drilling depth	
P34	Anticipation plane	Safety margin of the processing upper edge
P35	Retreat plane	To go around of obstacles in the setting axis
P36	No. of strokes	constant setting depth = (P33 - P34) / P36
P38	Safety margin	presetting = 1mm
P39	Gradual decrease	0 or - = off

5.3.2 G83 Deep-hole drilling with constant setting depth (continued)

Example

N20 G56 G00 X0 Y0 Z100 S1200 F1500 M03 T01 M16

N30 **G83** Z80 P30:1 P32:800 **P33:0** **P34:48** P35:60 **P36:3** P38:1 P39:0

N40 M05 ...

Sequence plan

1 Switch spindle on (M03, M04) and X and Y axis in rapid traverse position.

2 After achieving the X and Y position the Z-axis runs in rapid traverse on those anticipation plane (P34).

3 After achieving the anticipation plane (P34) the Z axis with working feed moves (P32) to the 1. drilling depth (P34 + ((P33 - P34) * 1 / P36)).

4 After achieving the 1. drilling path the Z axis drives back in rapid traverse up to the anticipation plane.

5 From the anticipation plane the Z axis drives afterwards in rapid traverse up to the 1. drilling depth plus safety margin (P34 + ((P33 - P34) * 1 / P36)) + P38).

6 The Z axis induces itself with working feed (P32) to 2. drilling depth (P34 + ((P33 - P34) * 2 / P36)).

7 After achieving the 2. drilling depth the Z axis drives back in rapid traverse up to the anticipation plane.

8 From the anticipation plane the Z-axis drives afterwards in rapid traverse up to the 2. drilling depth plus safety margin (P34 + ((P33 - P34) * 2 / P36)) + P38).

9 The Z axis induces itself with working feed (P32) to the 3. drilling depth etc. to P33.

10 With the achieving of the drilling depth (P33) a programmed free cutting time (P30) becomes been waiting.

11 Afterwards the Z axis in rapid traverse moves back on the retreat plane (P35) and afterwards the spindle is switched off (M05).

5.3.3 G83 Deep-hole drilling with degressive setting depth

If the parameter P39:1 is programmed (the input value is not determining), the programmed drilling depth (P33) by repeated, reduced setting depth (gradual decrease) of the active drilling depth is achieved.

The active drilling depth is P37-(n-1)*P36, whereby n=1, 2,.. is (nth setting depth = stroke).
If P37-(n-1)*P36 ≤ P36, the active drilling depth = P36 is set.

Before each setting depth checked, whether two setting depths (drilling depth calculated + P36 (gradual decrease measure)) still feasible are. If not, then the new drilling depth calculated by remainder path / 2 and executed the final two setting depths with this value.

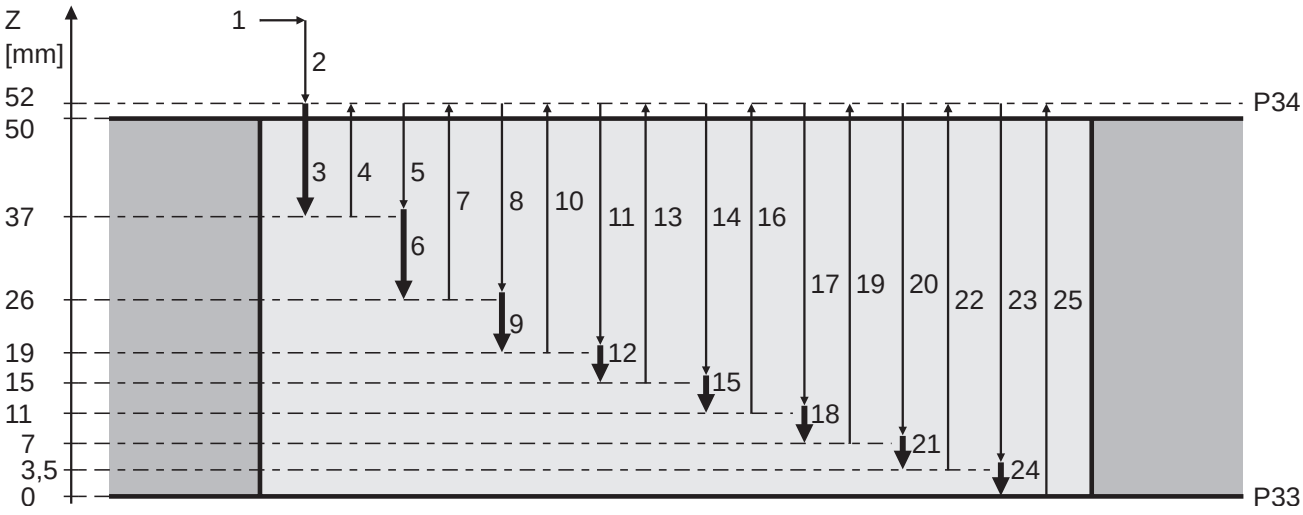

Figure 5-21

Meaning of the parameters

P32	drilling feed	
P33	drilling depth	
P34	anticipation plane	safety margin of the processing upper edge
P35	retreat plane	to the run around of obstacles in the setting axis
P36	setting depth	degressive setting depth
P37	1. stroke	degressive setting depth
P38	safety margin	presetting = 1mm
P39	gradual decrease	degressive setting depth 1 = on

5.3.3 G83 Deep-hole drilling with degressive setting depth (continued)

Example

N20 G56 G00 X0 Y0 Z100 S1200 F1500 M03 T01 M16

N30 **G83** Z80 P32:800 **P33:0** **P34:52** **P36:4** **P37:15** P38:1 **P39:1**

N40 M05 ...

Sequence plan

1 Switch spindle on (M03, M04) and X and Y axis in rapid traverse position.
2 After achieving the X and Y position the Z axis runs on the anticipation plane (P34).

3 After achieving the anticipation plane the Z axis with working feed induces itself to 1. drilling depth (P37) on the Z position: P34 - P37 = 52 - 15 = 37.
4 After achieving 1.drilling depth returns the Z axis in rapid traverse to the anticipation plane.

5 Of the anticipation plane in rapid traverse up to the 1. drilling depth plus safety margin (P38).
6 With working feed to 2. drilling depth on the Z position: 37 - 11 = 26 (11 = 15 - 4).
7 After achieving the 2. drilling depth to the anticipation plane back.

8 Of the anticipation plane in rapid traverse up to 2. drilling depth plus safety margin (P38).
9 With working feed to the 3. drilling depth on the Z position: 26 - 7 = 19 (7 = 11 - 4).
10 After achieving the 3. drilling depth to the anticipation plane back.

11 Of the anticipation plane in rapid traverse up to the 3. drilling depth plus safety margin (P38).
12 With working feed to 4. drilling depth on the Z position: 19 - 4 = 15 (P36:4).
13 After achieving the 4. drilling depth to the anticipation plane back.

14 Of the anticipation plane in rapid traverse up to 4. drilling depth plus safety margin (P38).
15 With working feed to 5. drilling depth on the Z position: 15 - 4 = 11 (P36:4).
16 After achieving the 5. drilling depth to the anticipation plane back.

17 Of the anticipation plane in rapid traverse up to 5. drilling depth plus safety margin (P38).
18 With working feed to 6. drilling depth on the z-position: 11 - 4 = 7 (P36:4).
19 After achieving the 6. drilling depth to the anticipation plane back.

20 Of the anticipation plane in rapid traverse up to 6. drilling depth plus safety margin (P38).
21 With working feed to 7. drilling depth on the Z position: 7 / 2 = 3,5.
22 After achieving the 7. drilling depth to the anticipation plane back.

23 Of the anticipation plane in rapid traverse up to 7. drilling depth plus safety margin (P38).
24 With working feed to 8. drilling depth on the Z position: 3.5 - 3.5 = 0.
25 After achieving 8. drilling depth returns the Z axis in rapid traverse to the anticipation plane.

5.3.4 G84 Thread cutting

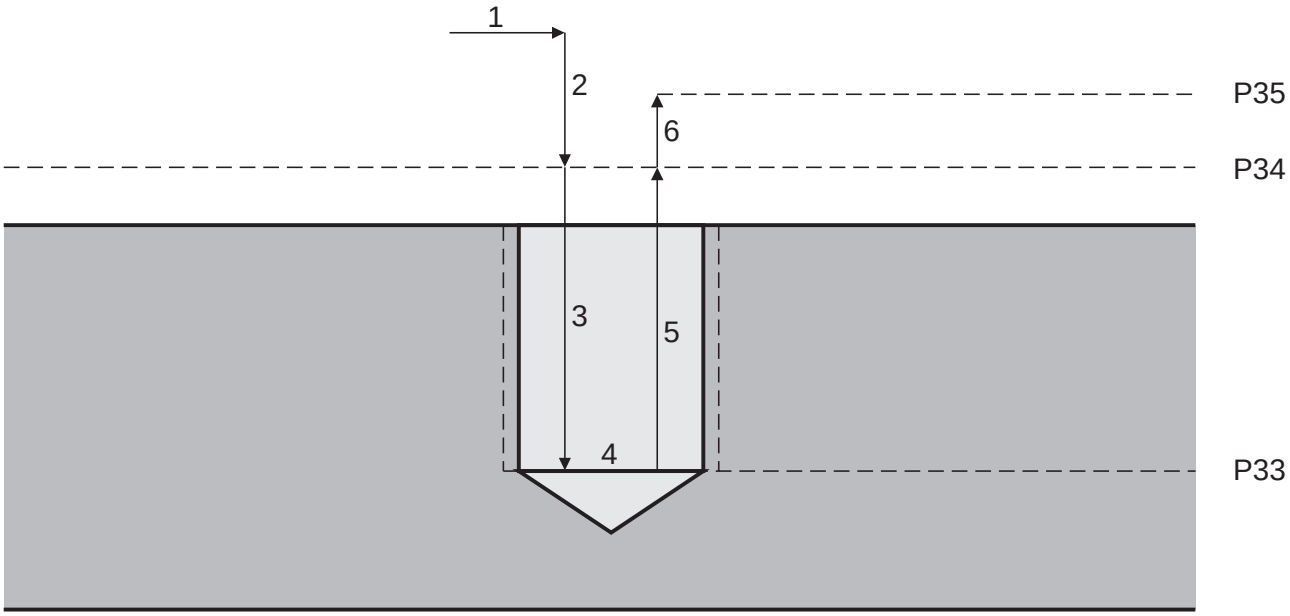

Figure 5-22

Meaning of the parameters

P31	balance fodder	P31: 1	= thread cutting without balance fodder
		P31: 0 or -	= G84 standard cycle (with balance fodder)
P32	thread pitch		
P33	thread depth		
P34	anticipation plane	safety margin of the processing upper edge	
P35	retreat plane	to running around obstacles in the setting axis	

5.3.4 G84 Thread cutting (continued)

Example

```
N10  ...
N20  G56  G00      X0   Y0   Z100      S1200     F1500     M03      T01  M16
N30  G84  Z80      P32:10    P33:20    P34:52    P35:60
N40  M05  ...
```

Sequence plan

1 Switch spindle on (M03, M04) and X and Y axis in rapid traverse position.

2 After achieving the X and Y positions the Z axis runs on the anticipation plane.

3 After achieving the anticipation plane the Z axis is joined with the spindle axis and the Z axis moves to the drilling depth (P33).

4 With the achieving of the drilling depth the spindle rotation is turned around and the spindle reversal time (P30) recalled.

5 At process of the retention time the Z axis returns to the anticipation plane (P34).

6 With the achieving of the anticipation plane the Z-axis in rapid traverse becomes on the retreat plane (P35) driven and afterwards the spindle switched off (M05).

5.3.5 G85 Drilling with retraction at feed rate

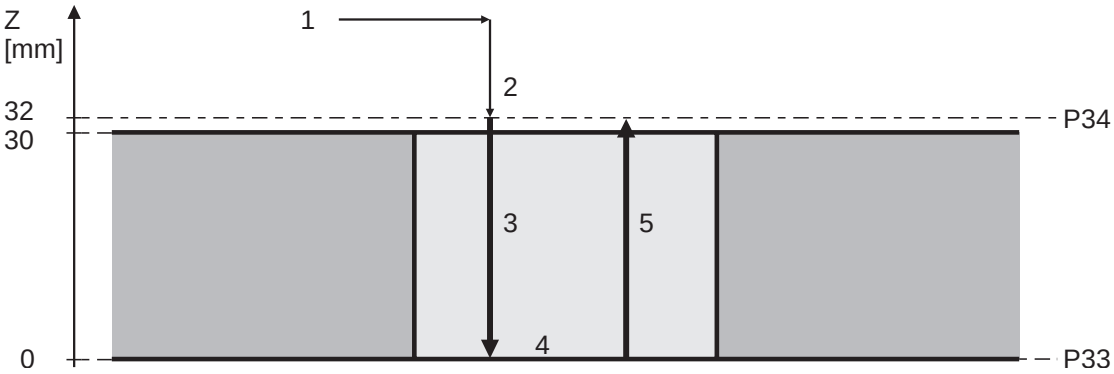

Figure 5-23

Meaning of the parameters

P30	free cutting time	
P32	drilling feed	
P33	drilling depth	
P34	anticipation plane	safety margin of the processing upper edge
P35	retreat plane	to running around obstacles in the setting axis

5.3.5 G85 Drilling with retraction at feed rate (continued)

Example

```
N10  ...
N20  G56  G00      X0   Y0   Z100     S1200    F1500    M03      T01  M16
N30  G85  Z80      P30:1     P32:800  P33:0    P34:32
N40  M05  ...
```

Sequence plan

1 Switch spindle on (M03, M04) and X and Y axis in rapid traverse position.

2 After achieving the X and Y positions the Z axis runs in rapid traverse on those anticipation plane (P34).

3 After achieving the anticipation plane the Z axis with working feed becomes the drilling depth (P33) moves.

4 With the achieving of the drilling depth a programmed free cutting time (P30) is waited for.

5 Afterwards retreat in the working feed to anticipation plane and becomes following spindle switched off (M05).

5.4 G86 / G87 / G88 / G89 cycle patterns

The cycle patterns serve for the repeated version of prozess cycles (boring and milling cycles). Over the programming of certain parameters the points of positioning are described, in which the prozess cycles are to be executed.

The parameters cover the definition of a target as well as the number of points of positioning. For definition of the target and the points of positioning different programming types can be used.

The partitioning of the cycle patterns takes place on the basis of geometrical criteria:

G86	vector type processing
G87	parallelogram type processing
G88	grid type processing
G89	circle type processing

The cycle patterns are block by block effective.

The parameter inputs remain against it modal effectively.

In order to avoid over regulation, all parameters are to be reset, which are not used with a programming type.

A compilation of the parameters is listed on the next side.

5.4 G86 / G87 / G88 / G89 cycle patterns (continued)

Parameter for G86 vector type processing

P100	coordinate of the 1. axis (X)
P101	coordinate of the 2. axis (Y)
P103	vector bracket related to the1. axis (X)
P104	vector length
P105	vector splitting
P106	number of points of positioning

Parameter for G87 periphery processing of a parallelogram

Vector 1	Vector2	
P100	P110	coordinate of the 1. axis (X)
P101	P111	coordinate of the 2. axis (Y)
P103	P113	vector bracket related to the 1. axis (X)
P104	P114	vector length
P105	P115	vector splitting
P106	P116	number of points of positioning

Parameter for G88 grid processing

Vector 1	Vector 2	
P100	P110	coordinate of the 1. Axis @@@(x)
P101	P111	coordinate of the 2. Axis @@@(y)
P103	P113	vector bracket related to the @@@1.Achse (x)
P104	P114	vector length
P105	P115	vector splitting
P106	P116	number of points of positioning

Parameter for G89 circle processing

P120	coordinate of the 1. axis (X) of the pitch diameter focal point
P121	coordinate of the 2. axis (Y) of the pitch diameter focal point
P122	circle diameter
P123	start bracket
P124	travel bracket
P125	vector part
P126	number of points of positioning

5.4 G86 / G87 / G88 G89 cycle patterns (continued)

Messages

Messages to the cycles have numbers, which start with 8xxx.
A listing is to be found in the paragraph ‘ 9,9 messages of cycles ‘.

Messages appear, if

- the target with a programming type by programming of additional parameters multiple is defined and no agreement is achieved.

- the target is not sufficiently determined.

In these cases the program is interrupted.

Therefore all parameters are to be reset, which are not used with a programming type for the definition of the target.

5.4 G86 / G87 / G88 G89 cycle patterns (continued)

Call of prozess cycles with cycle patterns

Together with a cycle pattern call a process cycle (boring or milling cycle) can be activated, which is then called after each positioning step in the cycle pattern.

Effectiveness of M functions

Pre path M functions are executed in the starting point A, after path M functions in the target B.

M functions, which are effective within cycles (e.g. M03, M05,..), become for each cycle call pre path or after path actively.

Start the points of positioning

Starting the respective points of positioning effected with G00. Contains the theorem a cycle call, in each point of positioning the cycle is executed. Otherwise an accurate stop (G08) is activated

5.4.1 Vector type processing

Start point and target point with G86

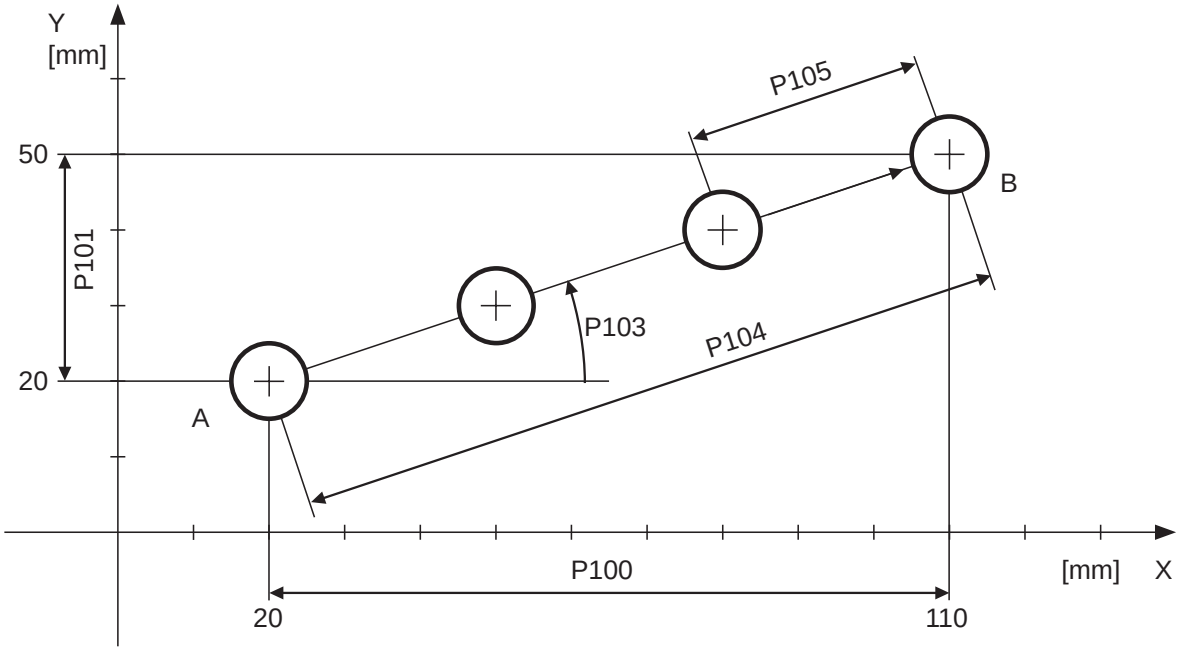

Figure 5-24 A = Start point, B = Target point

Meaning of the parameters

P100 AB coordinate of the 1. axis (X)
P101 AB coordinate of the 2. axis (Y)
P103 vector bracket related to the 1. axis (X)
P104 vector length
P105 vector splitting
P106 number of positioning points

5.4.1 Vector type processing (continued)

Example 1: Programming with AB coordinates P100 and P101 and number of points of positioning

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	**P100:90**	**P101:30**	P103:–	P104:–	P105:–	**P106:4**		
N30	P32:800	P33:0	P34:32					
N40	**G86**	G81						

Example 2: Programming with vector brackets P103 and vector length P104 and number of points of positioning

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	P100:–	P101:–	**P103:20**	**P104:96**	P105:–	**P106:4**		
N30	P32:800	P33:0	P34:32					
N40	**G86**	G81						

Example 3: Programming with vector brackets P103 and vector splitting P105 and number of points of positioning

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	P100:–	P101:–	**P103:20**	P104:–	**P105:32**	**P106:4**		
N30	P32:800	P33:0	P34:32					
N40	**G86**	G81						

5.4.1 Vector type processing (continued)

Process

N10	programming of the start point with X and Y
N20	programming of the target point with the parameters P100 and P101 or P103 and P104 or P103 and P105
N30	programming of the parameters of the process cycle
N40	calls of the cycle pattern and the process cycle

Sequence of the operation: position first and then process.

Note: not used parameters are to be reset.

5.4.1 Vector type processing (continued)

Programming the start point A

The start point is established by programming the X, Y.. coordinates. If these coordinates have not been programmed the machine stop location coordinates are used as the starting coordinates.

Programming the target point B

Programming mode 1

The target point B is established by programming the X (AB) P100 and Y (AB) P101 coordinates.

Bx = Ax + X (AB)
By = Ay + Y (AB)

Programming mode 2

The target point B is established by programming the angle P103 and the length of vector P104.

X (AB) = P104 * cosP103
Y (AB) = P104 * sinP103

Programming mode 3

The target point B is established by programming the angle P103, the vector part P105 and the number of positioning points P106.

For P106 equal to or greater than 2 the following apllies:

X (AB) = P105 * (P106-1) * cosP103
Y (AB) = P105 * (P106-1) * sinP103

For P106=0 and P106=1:

X (AB) = P105 * P106 * cosP103
Y (AB) = P105 * P106 * sinP103

5.4.1 Vector type processing (continued)

Data analysis

Angel P103

≥ 0°: angle in positive mathematical sense (left-handed rotation)
referred on the positive vector of the 1. axis (X)

< 0°: angle in negative mathematical sense (right-handed rotation)
referred on the positive vector of the 1. axis (X)

≥ 360°: reduction of the angle on smaller 360°

Influence of the planes that are switched on

The sizes X and Y and the angle P103 are plane-oriented.

Plane	Axis	P103 referred on the positive vector of the axis
G17 (XY)	X Y	X
G18 (ZX)	Z X	Z
G19 (YZ)	Y Z	Y

Vector length P104 and vector part P105

Negative values are changed into positive values without indication.

5.4.1 Vector type processing (continued)

Number of positioning points P106

Negative values are changed into positive values.
Non-integer values are round down to the next smaller integer value.

Number of positioning points P106=0
The target point B (from A to B) is approached directly; eventually activated cycles are not executed.

Number of positioning points P106=1
The target point B (from A to B) is approached directly and an eventually activated cycle is executed in the target point.

Number of positioning points P106 and vector part P105

Further identifications for the programming modes 1 and 2 could be the number of positioning points P106 or the vector part P105.

When indicating the vector part P105, the number of positioning points is determined:

N = (AB / P105) + 1

After this, the vector part is calculated:

T’ = AB / (P106 - 1)

If the calculated number N is an integer number:

P105' = P105

If the calculated number N is not an integer number:

P105' unequal to P105

The positioning points is executed with the value T’.

5.4.2 G87 Parallelogram type processing

Start and target point with G87

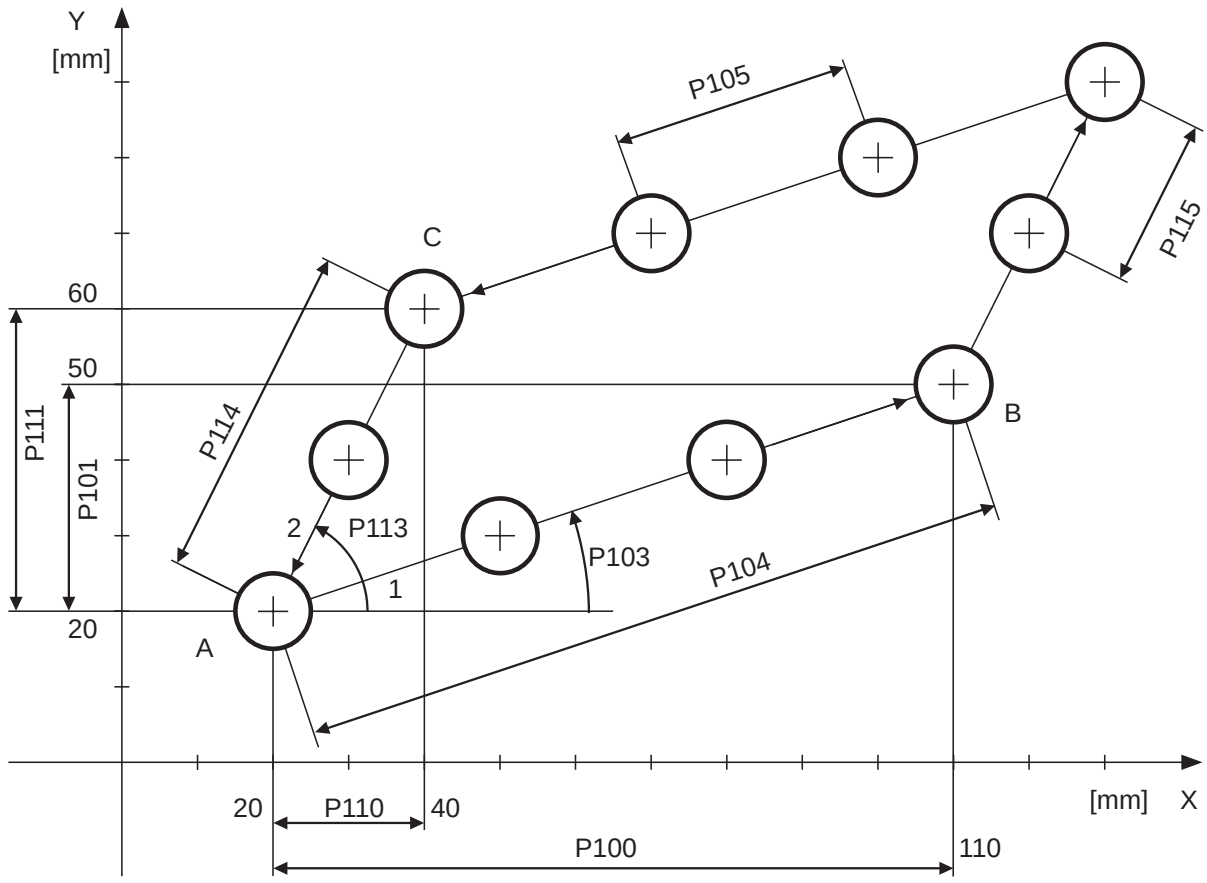

Figure 5-25

Meaning of the parameters

Vector 1	Vector 2	
P100	P110	coordinate of the 1. axis (X)
P101	P111	coordinate of the 2. axis (Y)
P103	P113	ector bracket related to the 1. axis (X)
P104	P114	vector length
P105	P115	vector part
P106	P116	number of positioning points

5.4.2 G87 Parallelogram type processing (continued)

Example 1: Programming with AB coordinates P100, P101 and P110, P111 and number of positioning points

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	**P100:90** P114:–	**P101:30** P105:–	**P110:20** P115:–	**P111:40** **P106:4**	P103:– **P116:3**	P113:–	P104:–	
N30	P32:800	P33:0	P34:32					
N40	**G87**	G81						

Example 2: Programming with vector brackets P103, P113 and vector length P104, P114 and number of positioning points

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	P100:– **P114:46**	P101:– P105:–	P110:– P115:–	P111:– **P106:4**	**P103:20** **P116:3**	**P113:60**	**P104:96**	
N30	P32:800	P33:0	P34:32					
N40	**G87**	G81						

Example 3: Programming with vector brackets P103, P113 and vector splitting P105, P115 and number of positioning points

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	P100:– P114:–	P101:– **P105:32**	P110:– **P115:23**	P111:– **P106:4**	**P103:20** **P116:3**	**P113:60**	P104:–	
N30	P32:800	P33:0	P34:32					
N40	**G87**	G81						

5.4.2 G87 Parallelogram type processing (continued)

Process

N10 programming of the start point with X and Y

N20 programming of the target point with the parameters
P100, P110 and P101, P111 or P103, P113 and P104, P114 or P103, P113 and P105, P115

N30 programming of the parameters of the process cycle

N40 calls of the cycle pattern and the process cycle

Sequence of the operation: first position and then process.

Sequence of processing: for and return trip always
first vector 1 and then vector 2

Note: not used parameters are to be reset

5.4.2 G87 Parallelogram type processing (continued)

Programming of the start point A

The start point A is determined by programming the coordinates X, Y..
If the coordinates are not programmed, the coordinates of the machine location are used as start coordinates.

Programming of the target points B and C

Programming mode 1

The target points B and C are determined by programming the coordinates X (AB) P100 / Y (AB) P101 and X (AC) P110 / Y (AC) P111.

Target point B	Bx = Ax + X(AB) By = Ay + Y(AB)	Target point C	Cx = Ax + X(AC) Cy = Ay + Y(AC)

Programming mode 2

The target points B and C are determined by programming the angles P103 / P113 and vector length P104 / P114.

Target point B	X(AB) = P104 * cosP103 Y(AB) = P104 * sinP103	Target point C	X(AC) = P114 * cosP113 Y(AC) = P114 * sinP113

Programming mode 3

The target points B and C are determined by programming the angles P103 / P113, the vector parts P105 / P115 and number of positioning points P106 / P116.

For P106 / P116 >= 2:

Point B	X(AB) = P105 * (P106-1) * cosP103 Y(AB) = P105 * (P106-1) * sinP103	Point C	X(AC) = P115 * (P116-1) * cosP113 Y(AC) = P115 * (P116-1) * sinP113

For P106 / P116 = 0 and P106 / P116 = 1 :

Point B	X(AB) = P105 * P106 * cosP103 Y(AB) = P105 * P106 * sinP103	Point C	X(AC) = P115 * P116 * cosP113 Y(AC) = P115 * P116 * sinP113

5.4.2 G87 Parallelogram type processing (continued)

Data evaluation

Angle P103, P113

≥ 0°: angle in positive mathematical sense (left-handed rotation)
referred on the positive vector of the x-axis

< 0°: angle in negative mathematical sense (right-handed rotation)
referred on the positive vector of the x-axis

≥ 360°: reduction of the angle on smaller 360°

Influence of the plane that is switched on:

The sizes X and Y and the angle P103, P113 are plane-oriented.

Plane	Axis	P103, P113 referred on the positive vector of the axis
G17 (XY)	X Y	X
G18 (ZX)	Z X	Z
G19 (YZ)	Y Z	Y

Vector length P104, P114 and vector part P105, P115

Negative values are changed into positive values without indication.

5.4.2 G87 Parallelogram type processing (continued)

Number of positioning points P106, P116

Negative values are changed into positive values.
Non-integer values are round down to the next smaller integer value.

Number of positioning points P106, P116 = 0
The target point B / C (from A to B / C) are approached directly; eventually activated cycles are not executed.

Number of positioning points P106, P116 = 1
The target points B / C (from A to B / C) are approached directly and an eventually activated cycle is executed in the target point.

Number of positioning points P106, P116 and vector part P105, P115

Further identifications for the programming modes 1 and 2 could be the number of drilling holes P106, P116 or the vector part P105, P115.

When indicating the vector part P105, P115, the number of drilling holes is determined:

P106, P116 = (AB / P105, P115) + 1

After this, the vector part is calculated:

P105', P115' = AB / (P106, P116 - 1)

If the calculated number P106, P116 is an integer number:

P105', P115' = P105, P115

If the calculated number P106, P116 is not an integer number:

P105', P115' unequal to P105, P115

The drilling pattern is executed with the value P105', P115'.

5.4.3 G88 Grid type processing

Start and target point at G88

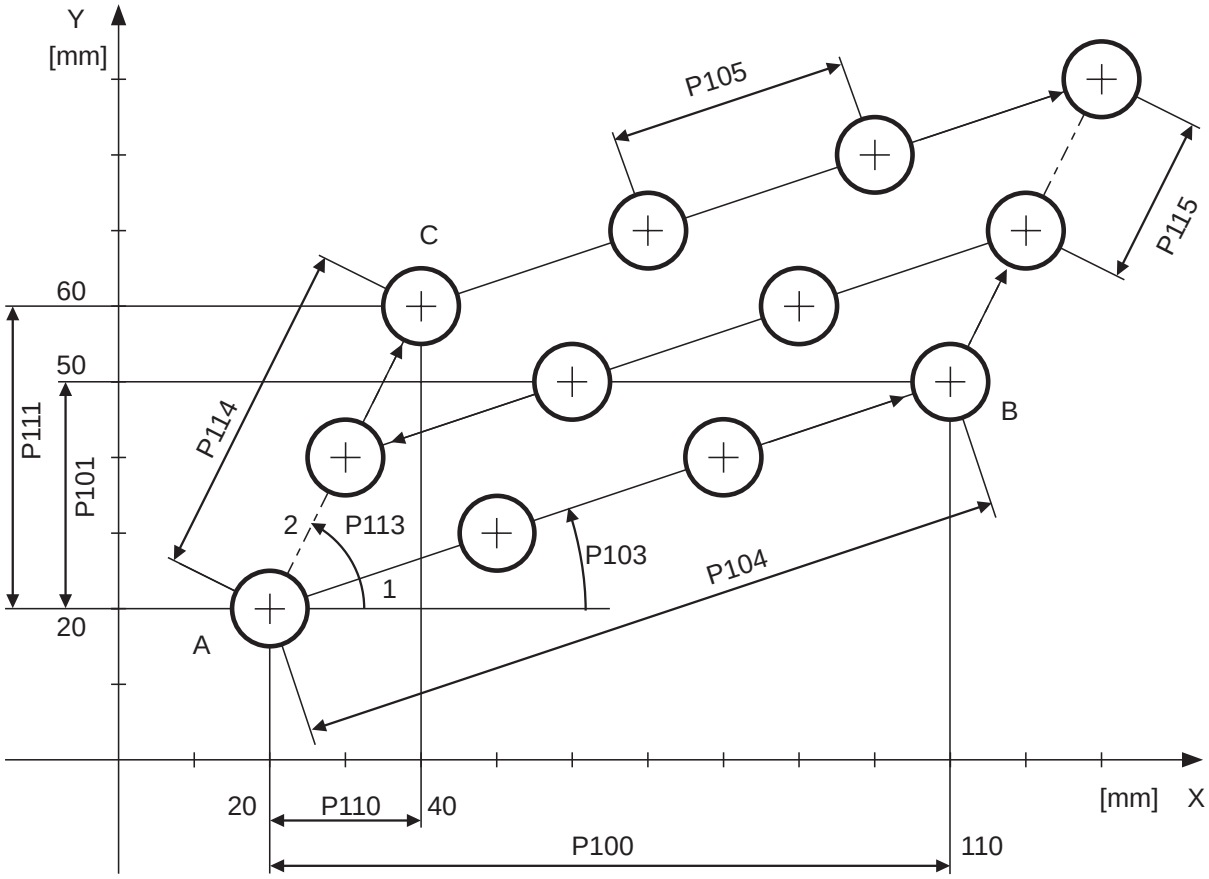

Figure 5-26

Meaning of the parameters

Vector 1	Vector 2	
P100	P110	coordinate of the 1. axis (X)
P101	P111	coordinate of the 2. axis (Y)
P103	P113	ector bracket related to the 1. axis (X)
P104	P114	vector length
P105	P115	vector part
P106	P116	number of positioning points

5.4.3 G88 Grid type processing (continued)

Example 1: Programming with AB coordinates P100, P110 und P101, P111 and number of positioning points

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	**P100:90** P114:–	**P101:30** P105:–	**P110:20** P115:–	**P111:40** **P106:4**	P103:– **P116:3**	P113:–	P104:–	
N30	P32:800	P33:0	P34:32					
N40	**G88**	G81						

Example 2: Programming with vector brackets P103, P113 and vector length P104, P114 and number of positioning points

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	P100:– **P114:46**	P101:– P105:–	P110:– P115:–	P111:– **P106:4**	**P103:20** **P116:3**	**P113:60**	**P104:96**	
N30	P32:800	P33:0	P34:32					
N40	**G88**	G81						

Example 3: Programming with vector brackets P103, P113 and vector splitting P105, P115 and number of positioning points

N10	G00	**X20**	**Y20**	Z100	S500	M03	T01	M16
N20	P100:– P114:–	P101:– **P105:32**	P110:– **P115:23**	P111:– **P106:4**	**P103:20** **P116:3**	**P113:60**	P104:–	
N30	P32:800	P33:0	P34:32					
N40	**G88**	G81						

5.4.3 G88 Grid type processing (continued)

Process

N10 programming of the start point with X and Y

N20 programming of the target point with the parameters
P100, P110 and P101, P111 or P103, P113 and P104, P114 or P103, P113 and P105, P115

N30 programming of the parameters of the process cycle

N40 calls of the cycle pattern and the process cycle

Sequence of the operation: first position and then process.

Sequence of processing: first with journey there first vector 1 completely
and afterwards the first position on vector 2,

then with return trip vector 1 completely
and afterwards the next position on vector 2,

then again with journey there vector 1 completely etc.
until the final position on vector 2 achieves
and the processing of vector 1 is executed

thereafter the initial position is started.

Note: not used parameters are to be reset

5.4.3 G88 Grid type processing (continued)

Programming the start point A

The start point A is determined by programming the coordinates X, Y...
If the coordinates are not programmed, the coordinates of the machine location are used as starting coordinates.

Programming of the target points B and C

Programming mode 1

The target points B and C are determined by programming the coordinates X (AB) P100, Y (AB) P101 and X (AC) P110, Y (AC) P111.

Target point B	Bx = Ax + X(AB) By = Ay + Y(AB)	Target point C	Cx = Ax + X(AC) Cy = Ay + Y(AC)

Programming mode 2

The target points B and C are determined by programming the angles P103, P113 and vector length P104, P114.

Target point B	X(AB) = P104 * cosP103 Y(AB) = P104 * sinP103	Target point C	X(AC) = P114 * cosP113 Y(AC) = P114 * sinP113

Programming mode 3

The target points B and C are determined by programming the angles E1 / E2, the vector parts T1 / T2 and number of drilling holes N1 / N2.

For N >= 2:

Point B	X(AB) = P105 * (P106-1) * cosP103 Y(AB) = P105 * (P106-1) * sinP103	Point C	X(AC) = P115 * (P116-1) * cosP113 Y(AC) = P115 * (P116-1) * sinP113

For P106, P116 = 0 and P106, P116 = 1:

Point B	X(AB) = P105 * P106 * cosP103 Y(AB) = P105 * P106 * sinP103	Point C	X(AC) = P115 * P116 * cosP113 Y(AC) = P115 * P116 * sinP113

5.4.3 G88 Grid type processing (continued)

Data evaluation

Angle P103, P113

≥ 0°: angle in positive mathematical sense (left-handed rotation) referred on the positive vector of the x-axis

< 0°: angle in negative mathematical sense (right-handed rotation) referred on the positive vector of the x-axis

≥ 360°: reduction of the angle on smaller 360°

Influence of the plane that is switched on:

The sizes X and Y and the angle P103, P113 are plane-oriented.

Plane	Axis	P103, P113 referred on the positive vector of the axis
G17 (XY)	X Y	X
G18 (ZX)	Z X	Z
G19 (YZ)	Y Z	Y

Vector length P104, P114 and vector part P105, P115

Negative values are changed into positive values without indication.

5.4.3 G88 Grid type processing (continued)

Number of drilling holes P106, P116

Negative values are changed into positive values.
Non-integer values are round down to the next smaller integer value.

Number of drilling holes P106, P116 = 0
The target points B / C (from A to B / C) are approached directly; eventually activated cycles are not executed.

Number of drilling holes P106, P116 = 1
The target points B / C (from A to B / C) are approached directly and an eventually activated cycle is executed in the target point.

Number of drilling holes P106, P116 and vector part P105, P115

Further identifications for the programming modes 1 and 2 could be the number of drilling holes P106, P116 or the vector part P105, P115.

When indicating the vector part P105, P115, the number of drilling holes is determined:

P106, P116 = (AB / P105, P115) + 1

After this, the vector part P105, P115 is calculated:

P105', P115' = AB / (P106, P116 - 1)

If the calculated number P106, P116 is an integer number:

P105', P115' = P105, P115

If the calculated number P106, P116 is not an integer number:

P105', P115' unequal to P105, P115

The drilling pattern is executed with the value P105', P115'.

5.4.4 G89 Circle type processing

Start and target point at G88

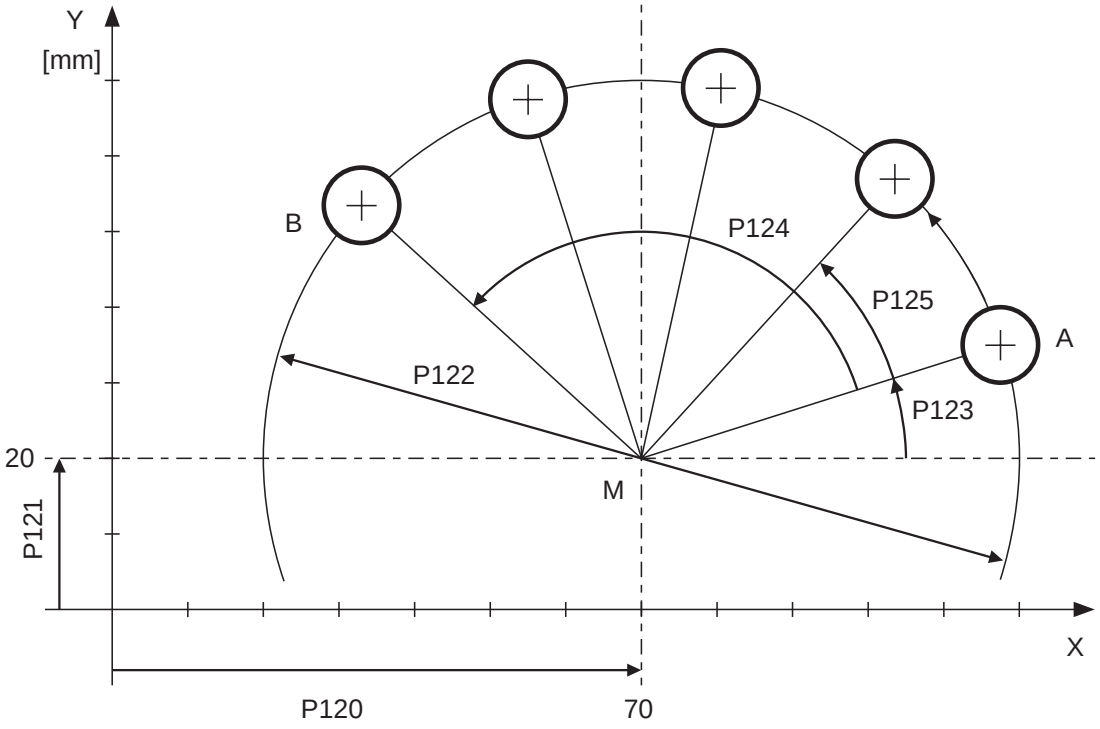

Figure 5-27

Meaning of the parameters

P120	Coordinate of the 1. axis (X) of the positioning points
P121	Coordinate of the 2. axis (Y) of the positioning points
P122	Pitch circle diameter
P123	Start angle
P124	Travel angle
P125	Vector part
P126	Number of positioning points

5.4.4 G89 Circle type processing (continued)

Example 1: **Programming with centre coordinates P120, P121, circle diameter P122, start brackets P123 and travel bracket P124 and number of positioning points**

N10	G00 M03	Z100 T01	S500 M16				
N20	**P120:70**	**P121:20**	**P122:100**	**P123:18**	**P124:120**	P125:--	**P126:5**
N30	P32:800	P33:0	P34:32				
N40	**G89**	G81					

Example 2: **Programming with centre coordinates P120, P121, circle diameter P122, start brackets P123 and vector part P125 and number of positioning points**

N10	G00 M03	Z100 T01	S500 M16				
N20	**P120:70**	**P121:20**	**P122:100**	**P123:18**	P124:--	**P125:30**	P126:--
N30	P32:800	P33:0	P34:32				
N40	**G89**	G81					

5.4.4 G89 Circle type processing (continued)

process

N10 programming of the start point with P120, P121, P122 and P123

N20 programming of the target point with the parameters
P120, P121, P122, P123 and P124 or P120, P121, P122, P123 and P125

N30 programming of the parameters of the process cycle

N40 calls of the cycle pattern and the process cycle

Sequence of the operation: first position and then process.

Sequence of processing: dependent on the programmed type of interpolation G00, G01 or G02, G03 become the particulars positions straight linear or circular interpolates started.

Note: not used parameters are to be reset

5.4.4 G89 Circle type processing (continued)

Programming the start point A

The starting point A always has to be determined by programming identifications at G89, i.e. the pitch circle centre point P120, P121, the pitch circle radius P122/2 and the starting angle P123.

Programming the target point B

Programming mode 1

The target point B is determined by programming the travel angle P124.
Bx = P120 + (P122/2) * cos(P123+P124)
By = P121 + (P122/2) * sin(P123+P124)

Programming mode 2

The target point is determined by programming the number of positioning points P126 and the vector part P125.
Bx = P120 + (P122/2) * cos(P123 + (P126-1) * P125)
By = P121 + (P122/2) * sin(P123 + (P126-1) * P125)

P124 = (P126-1) * P125

5.4.4 G89 Circle type processing (continued)

Data evaluation

Angle P123

≥ 0°: angle in positive mathematical sense (left-handed rotation) referred on the positive vector of the x-axis

< 0°: angle in negative mathematical sense (right-handed rotation) referred on the positive vector of the x-axis

≥ 360°: reduction of the angle on smaller 360°

Influence of the plane that is switched on:

The sizes X and Y and the angle P123 are plane-oriented.

Plane	Axis	P123 referred on the positive vector of the axis
G17 (XY)	X Y	 X
G18 (ZX)	Z X	 Z
G19 (YZ)	Y Z	 Y

Vector part P125

Negative values are changed into positive values without indication.

5.4.4 G89 Circle type processing (continued)

Number of positioning points P126

Negative values are changed into positive values.
Non-integer values are round down to the next smaller integer value.

Number of positioning points P126 = 0
The target point B (from A to B) is approached directly; eventually activated cycles are not executed.

Number of positioning points P126 = 1
The target point B (from A to B) is approached directly and an eventually activated cycle is executed in the target point.

Number of positioning points P126 and vector part P125

Further identifications for the programming modes 1 and 2 could be the number of positioning points P126 or the vector part P125.

When indicating the vector part P125, the number of positioning points P126 is determined:

P126 = (P124 / P125) + 1

After this, the vector part is calculated:

P125' = P124 / (P126-1)

If the calculated number P126 is an integer number:

P125' = P125

If the calculated number N is not an integer number:

P125' unequal to P125

The drilling pattern is executed with the value P125'.

5.4.4 G89 Circle type processing (continued)

Starting the points of positioning (G89)

Starting of the points of positioning depends on the programmed type of interpolation.

G00 or G01 actively

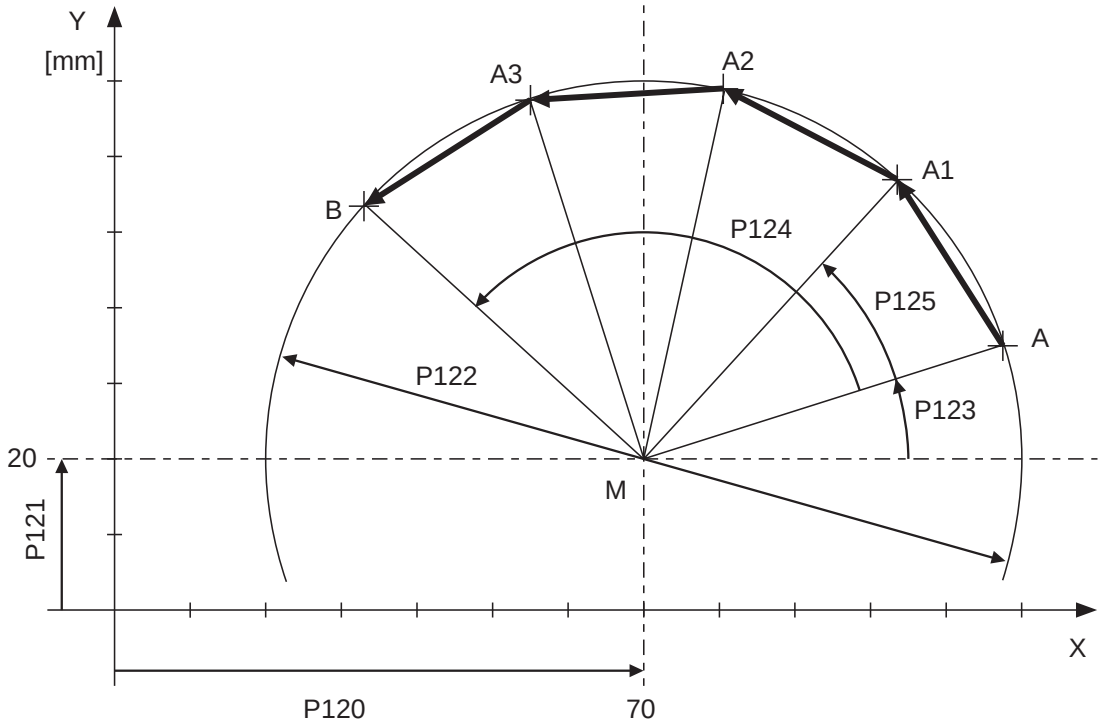

Figure 5-28

The positions (A1, A2, A3 and B) are started with G00.
The sign of the travel bracket P124 or, if this does not indicated, the sign of the vector part P125 determines, whether the circle is anti-clockwise rotation or is clockwise rotation processed:

Positive sign of the bracket:	anti-clockwise rotation,
Minus sign of the bracket:	clockwise rotation.

5.4.4 G89 Circle type processing (continued)

G02 or G03 actively

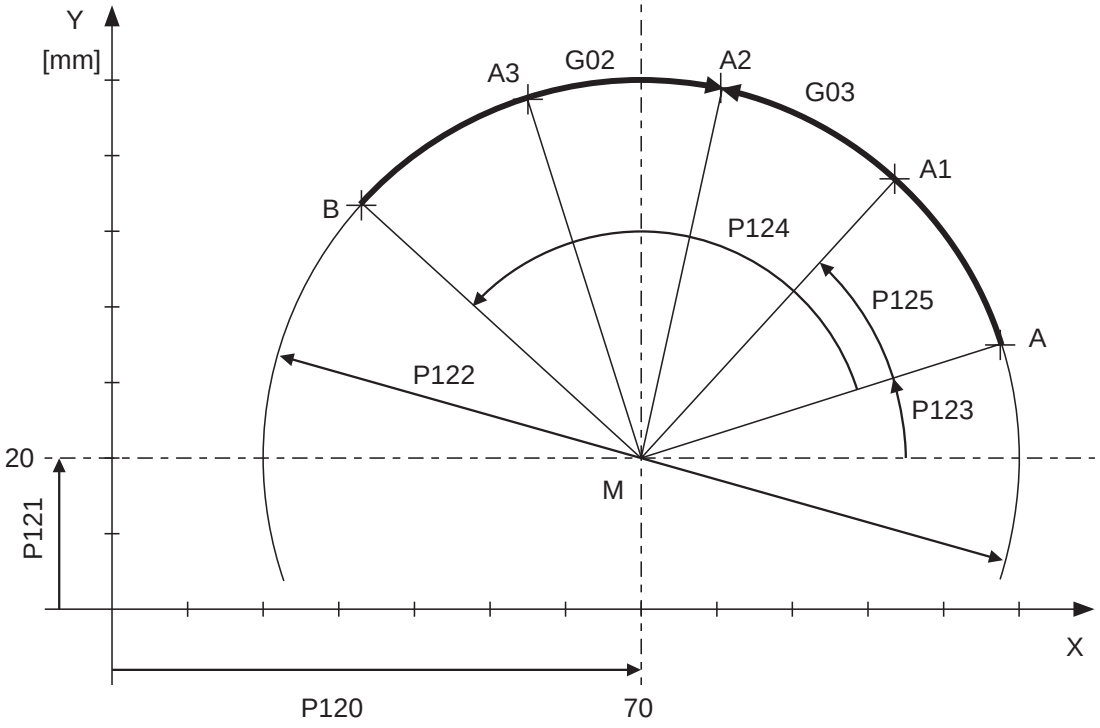

Figure 5-29

The positioning points are started with G02 or G03 and max. programmable feed speed.

Two cases are to be differentiated:

The travel bracket is positive (anti-clockwise rotating)

With G03 also the positioning direction is positive.
With G02 the positioning direction is negative (moving in opposite directions to the travel bracket).

The travel bracket is negative (clockwise rotatingly)

With G02 also the positioning direction is positive.
With G03 the positioning direction is negative (moving in opposite directions to the travel bracket).

5.5 Customer specific cycles

A cycle is a program that generates certain repeated sequences.

For user-specific cycles, these sequences are defined in a Z-program.

Example: Z1001
```
N10  ...
.
.
N40  P500 : P500+1
.
.
N60  M30
```

The defined cycle can be started then in a machining program with a G-function.
The G-function must have the same number as the Z-program.

Example: P2000

```
N10  ...
.
.
N60  P500 : 2    G1001
.
.
N90  M30
```

The G-function numbers can be selected arbitrarily except those numbers that are already reserved for other functions.

Input variables can be transmitted to the Z-program with parameters. The parameter numbers P500 to P4999 are free for the user. P0 to P499 is reserved for BWO-cycles.
If the BWO-cycles are not used, they are also free for the user.

Addtional functions to the operating system
Additional functions could be implemented with a DLL (Dynamic Link Library) that is written in programming language C.