SIEMENS

SINUMERIK 840D/840Di
SINUMERIK 810D/FM-NC

Short Guide **10.2000 Edition**

Programming

User Documentation

www.rgbautomatyka.pl

SIEMENS

SINUMERIK 840D/840Di
SINUMERIK 810D/FM-NC

Short Guide
Programming

Valid for

Control	*Software version*
SINUMERIK 840D	6
SINUMERIK 840DE (Export version)	6
SINUMERIK 840Di	1
SINUMERIK 840DiE (Export version)	1
SINUMEbcK 810D	4
SINUMERIK 810DE (Export version)	4
SINUMERIK FM-NC	3

10.00 Edition

SINUMERIK® documentation

Printing history

Brief details of this edition and previous editions are listed below.

The status of each edition is shown by the code in the "Remarks" column.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint with new Order No.
C .... Revised edition with new status.
If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header of that page.

Edition	Order No.	Remark
11.94	6FC5298-0AB30-0BP0	A
04.95	6FC5298-2AB30-0BP0	C
03.96	6FC5298-3AB30-0BP0	C
08.97	6FC5298-4AB30-0BP0	C
12.98	6FC5298-5AB30-0BP0	C
10.00	6FC5298-6AB30-0BP0	C

This manual is included in the documentation available on CD-ROM **(DOCONC**D)

Edition	Order No.	Remark
10.00	**6FC5298-6CA00-0BG0**	**C**

Further information is available on the Internet under:
http://www.ad.siemens.de/sinumerik

This publication was produced with WinWord V 7.0 and Designer V 4.0

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

Subject to technical change without prior notice.

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design are reserved.

© Siemens AG 1994 - 2000. All Rights Reserved.

Introduction

How to use this booklet

This booklet is a programming guide describing all the main programming steps.

The aim is to provide the operator with some quick help and a memory aid especially for commands that are used infrequently or to offer a quick reference guide on various parameters.

The guide therefore has little text and is easily comprehensible.

First familiarize yourself with the symbols below so that you understand them better whenever they occur on the following pages.

The symbols

Gives you a tip or background information.

Points out hazards, sources of error or general problems.

© Siemens AG 1998. All rights reserved.

Structure of the descriptions

The system used in these descriptions is based on the following scheme:

Programming of the function

Meaning of the parameters

Explanatory illustration with example workpiece

© Siemens AG 2000. All rights reserved.

List of sections

1. What is at the Beginning of the Program 1-9
Absolute and incremental dimensions, G90, G91 1-10
Zero offset, G54 to G57 1-12
Selection of working plane G17 to G19 1-14

2. Programming Motion Commands 2-17
Rapid traverse, G0 2-18
Linear interpolation, G1 2-20
Circular interpolation, G2/G3 2-22
Radius programming, G2/G3 2-24
Circular interpolation through intermediate point, CIP 2-25
Thread cutting, G33 2-27
Rigid tapping, G331/G332 2-28
Tapping with floating tapholder, G63 2-29
Polar coordinates G110, G111, G112 2-30

3. Tool Offsets and Compensations 3-31
Tool call 3-32
Cutter radius path compensation, G41/G42 3-33
Tool nose radius compensation, G41/G42 3-34
Activating/deactivating the collision detection 3-35
Approach and exit the contour, NORM/KONT 3-36
Programmable contour travel mode, G450/G451 3-37

4. Programming Aids Assist the Programmer 4-39
Overview of frame concept 4-40
Translating/rotating the coordinate system, TRANS/ROT 4-42
Mirroring on the coordinate axes, MIRROR 4-44
Increasing/reducing size of contour, SCALE 4-45

5. Programming Preparatory Functions 5-47
Feedrate programming, G93 to G97 5-48
Exact stop, G9/G60 5-49
Feedrate in continuous path mode, G64, G641 5-50
Programming the spindle motion 5-51

6. Subroutine Technique and Cycles 6-53
Subroutine technique 6-55
The use of cycles 6-57
Explanation of parameters 6-58
Drilling, centering, CYCLE81 6-62
Drilling, counterboring, CYCLE82 6-63
Deep hole drilling, CYCLE83 6-64
Rigid tapping, CYCLE84 6-65
Tapping with floating tapholder, CYCLE840 6-66
Bore 1, CYCLE85 6-67
Bore 2, CYCLE86 6-68
Bore 3, CYCLE87 6-69

© Siemens AG 2000. All rights reserved.

Bore 4, CYCLE88 6-70
Bore 5, CYCLE89 6-71
Row of holes, HOLES1 6-72
Hole circle, HOLES2 6-73
Elongated holes on a circle, LONGHOLE 6-74
Slots on a circle, SLOT1 6-75
Circumferential slot, SLOT2 6-76
Rectangular pocket milling, POCKET1 6-77
Circular pocket milling, POCKET2 6-78
Thread cutting, CYCLE90 6-79
Recessing cycle, CYCLE93 6-80
Undercut cycle, CYCLE94 6-81
Stock removal cycle, CYCLE95 6-82
Thread undercut, CYCLE96 6-83
Thread cutting, CYCLE97 6-84
Chaining of threads, CYCLE98 6-85

7. Predefined Switching and Auxiliary Functions 7-87
List of M commands 7-88
Forms 7-89

8. Program Key 8-91
List of G functions 8-92

© Siemens AG 2000. All rights reserved.

1. What is at the Beginning of the Program

Absolute and incremental dimensions, G90, G91 1-10

Zero offset, G54 to G57 1-12

Selection of working plane G17 to G19 1-14

© Siemens AG 2000. All rights reserved.

Absolute and incremental dimensions, G90, G91

Programming

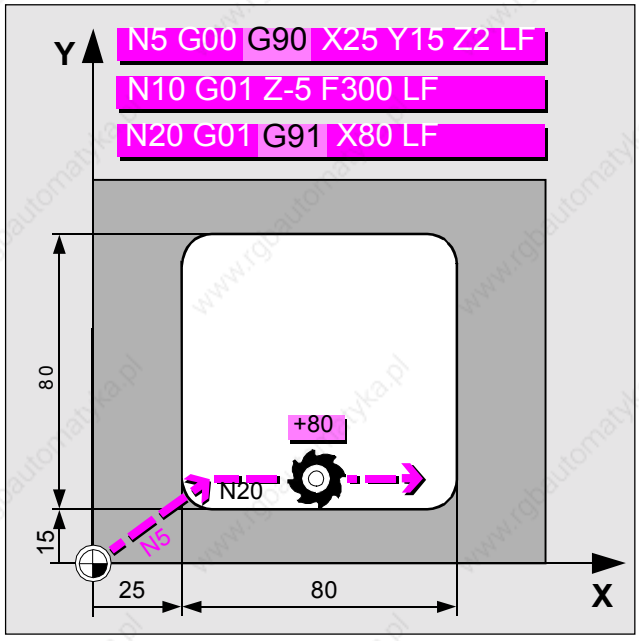
```
N5 G0  G90 X25 Y15 Z2 LF
N20 G1 G91 X80 F300 LF
```

G90	Absolute dimension input, all data refers to the actual workpiece zero.
G91	Incremental dimension input, each dimension refers to the contour point last input.

You can change over from block to block as often as you want between absolute and incremental data input.

Within a block, you can also change the type of input for individual axes by specifying AC for absolute coordinates or IC for incremental coordinates.
Example: X = AC (400)

Milling:



Change between absolute and incremental dimension programming

© Siemens AG 2000. All rights reserved.

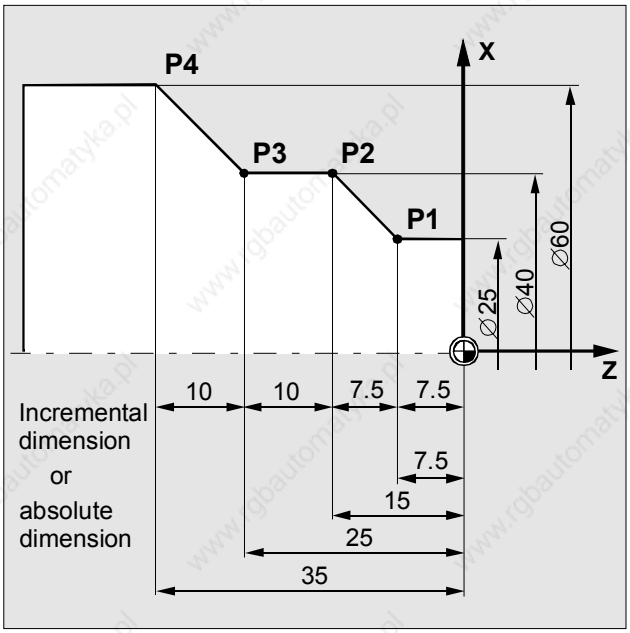
Absolute and incremental dimensions, G90, G91 (cont.)

Programming

```
N5 G0 G90 X25 Z1
N10 G1 Z-7.5 F0.2
N20 G1 X40 Z-15
N30 G1 G91 Z-10
N40 G1 G90 X60 Z-35
```

G90	Absolute dimension input, all data refers to the actual workpiece zero.
G91	Incremental dimension input, each dimension refers to the contour point last input.

Turning:



Dimensions: incremental or absolute dimensions

© Siemens AG 2000. All rights reserved.

Zero offset, G54 to G57

Programming

```
N30...LF
N40 G54 LF
N50 G0 X30 Y75LF
```

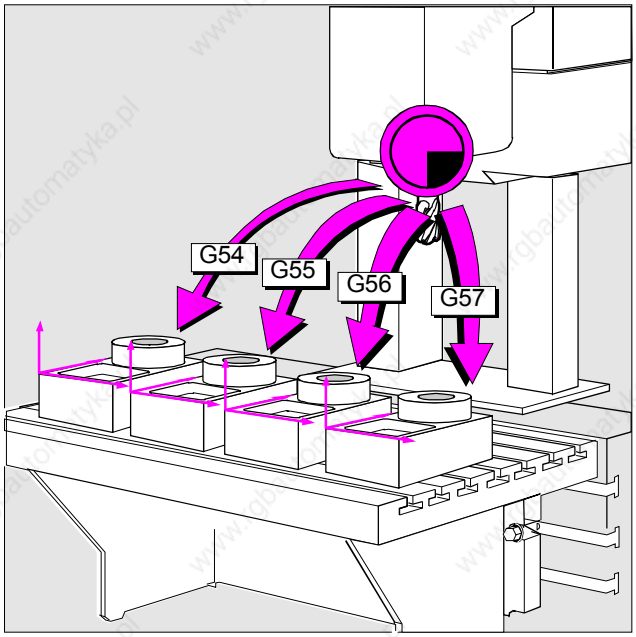
Other zero offsets: G55...G57, G505…G599

Parameters

X,Y,Z Coordinates of the zero offsets (definition of workpiece coordinate system). These must have been input via operator panel or universal interface into the control before programming.

With command G53, zero offsets can be suppressed block by block; deactivate with G500.

Milling:



Zero offsets make multiple machining operations possible

© Siemens AG 2000. All rights reserved.

Zero offset, G54 to G57 (cont.)

Programming

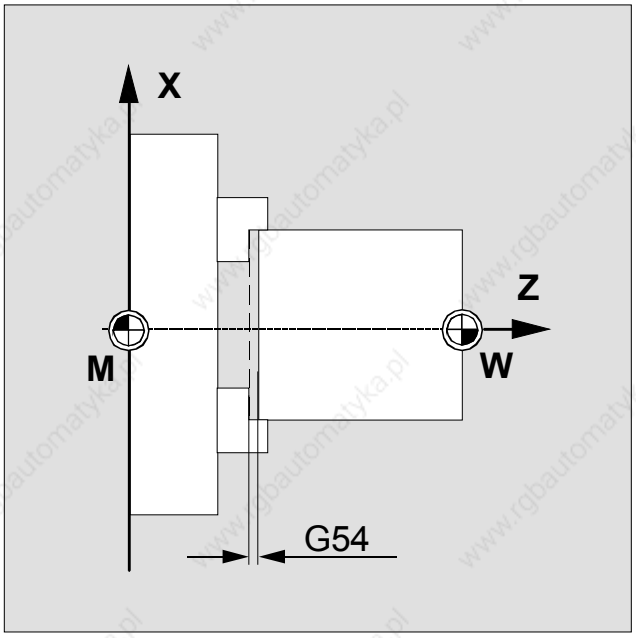
```
N10 G54
N20 G0 Z0.2
```

Parameter

Z	Coordinates of the zero offsets (definition of workpiece coordinate system). These must have been input via operator panel or universal interface into the control before programming.

In turning machines, a zero offset is generally advisable in the Z direction only.

Turning:



Zero offsets in the Z direction

© Siemens AG 2000. All rights reserved.

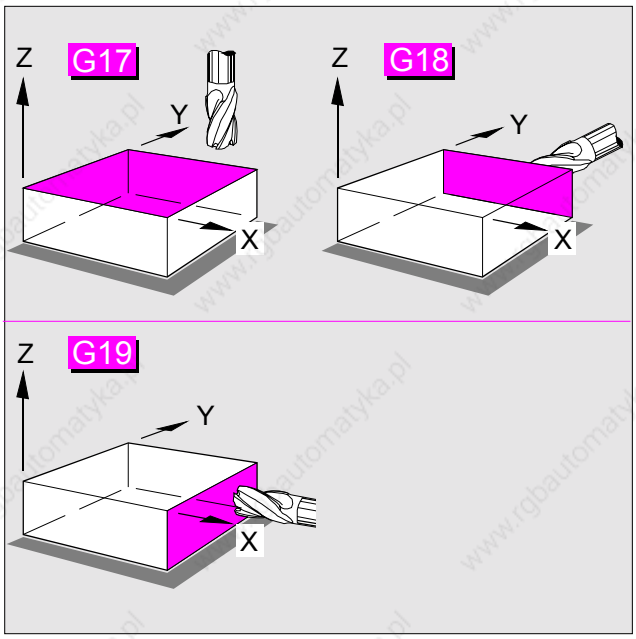
Selection of working plane G17 to G19

Programming N10 G0 X50 Z50 G17 D1 F1000 LF

Command	Working plane	Infeed axis
G17	X/Y	Z
G18	Z/X	Y
G19	Y/Z	X

Programming of the working plane is needed for computation of the tool offset data.
It is not possible to change the working plane when G41/G42 is active.

Milling:



Selection of working planes for horizontal and vertical milling operations

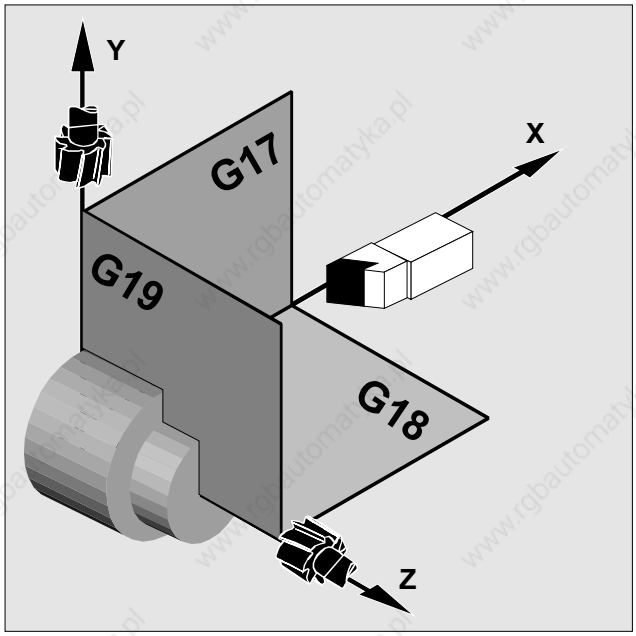
© Siemens AG 2000. All rights reserved.

Selection of working plane, G17 to G19 (cont.)

Programming N10 G0 X10 Z20 G18 D1 F200

In the basic setting, G17 is preset for milling (X/Y plane) and G18 for turning (Z/X plane).

Turning:



Selection of working planes for horizontal and vertical turning operations

© Siemens AG 2000. All rights reserved.

© Siemens AG 2000. All rights reserved.

2. Programming Motion Commands

Rapid traverse, G0	2-18
Linear interpolation, G1	2-20
Circular interpolation, G2/G3	2-22
Radius programming, G2/G3	2-24
Circular interpolation through intermediate point, CIP	2-25
Thread cutting, G33	2-27
Rigid tapping, G331/G332	2-28
Tapping with floating tapholder, G63	2-29
Polar coordinates G110, G111, G112	2-30

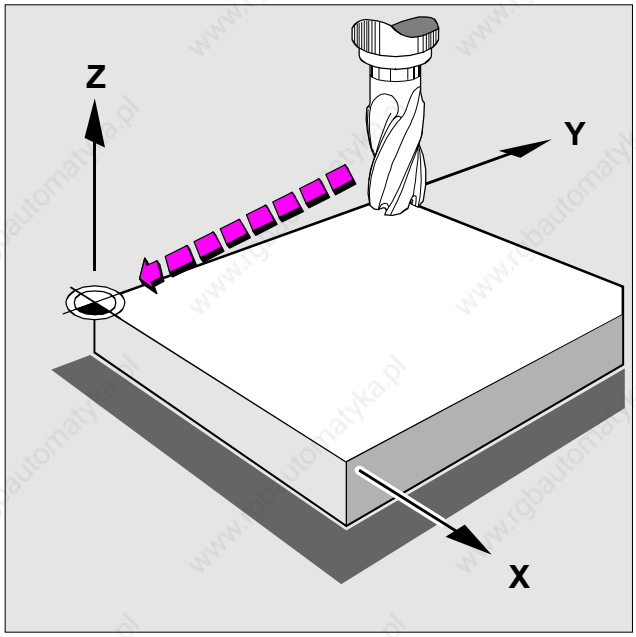
© Siemens AG 2000. All rights reserved.

Rapid traverse, G0

Programming N10 G0 X0 Y0Z3 LF

Parameters X, Y, Z Coordinates of the target point

Milling:



Fast tool positioning in rapid traverse for milling

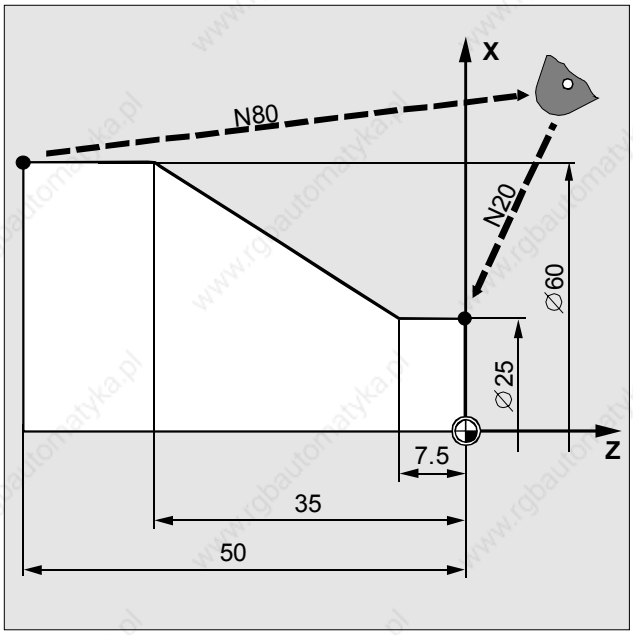
© Siemens AG 2000. All rights reserved.

Rapid traverse, G0 (cont.)

Programming N20 G0 X25 Z1

Parameters X, Z Coordinates of the target point

Turning:



Fast tool positioning in rapid traverse for turning

© Siemens AG 2000. All rights reserved.

Linear interpolation, G1

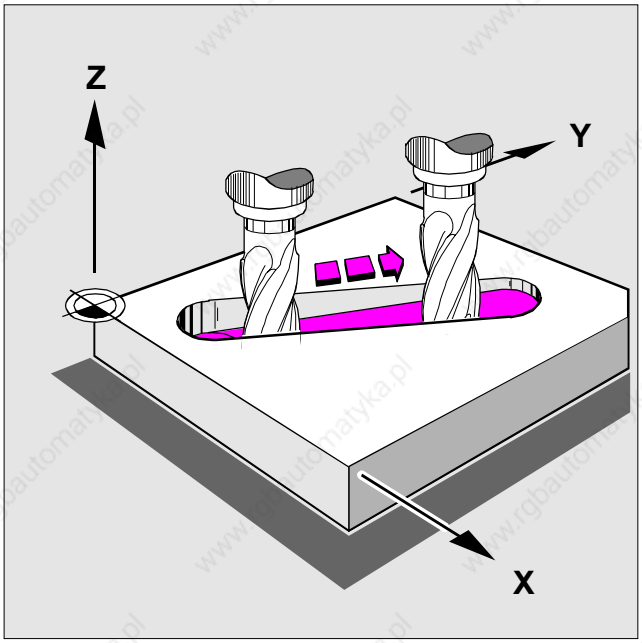
Programming

```
N10 G0 G90 X10 Y10 Z1 S800 M3 LF
N20  G1    Z-12 F500 LF
N30 X30 Y35 Z-3 F700 LF
```

Parameters

X, Y, Z	Coordinates of the target point
F	Feedrate

Milling:



Making an inclined slot

© Siemens AG 2000. All rights reserved.

Linear interpolation, G1 (cont.)

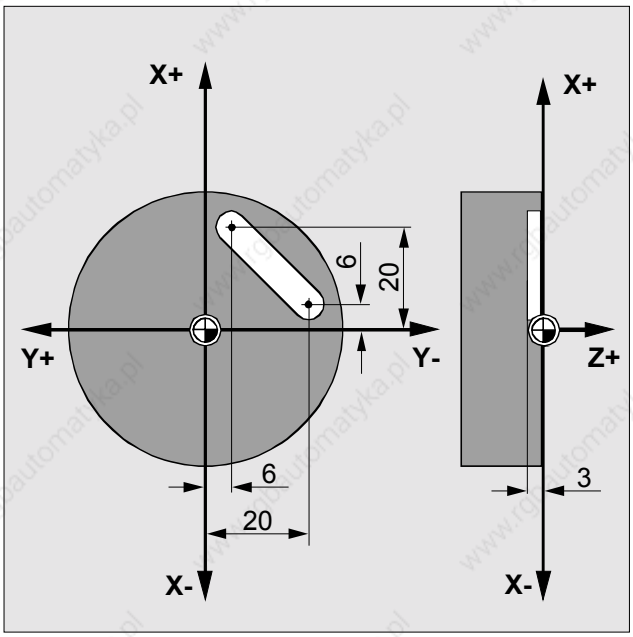
Programming

```
N10 G17 S400 M3
N20  G0  X40 Y-6 Z2
N30  G1  Z-3 F40
N40 X12 Y-20
```

Parameters

X, Y, Z	Coordinates of the target point
F	Feedrate

Turning/milling:



Making a slot

© Siemens AG 2000. All rights reserved.

Circular interpolation, G2/G3

Programming

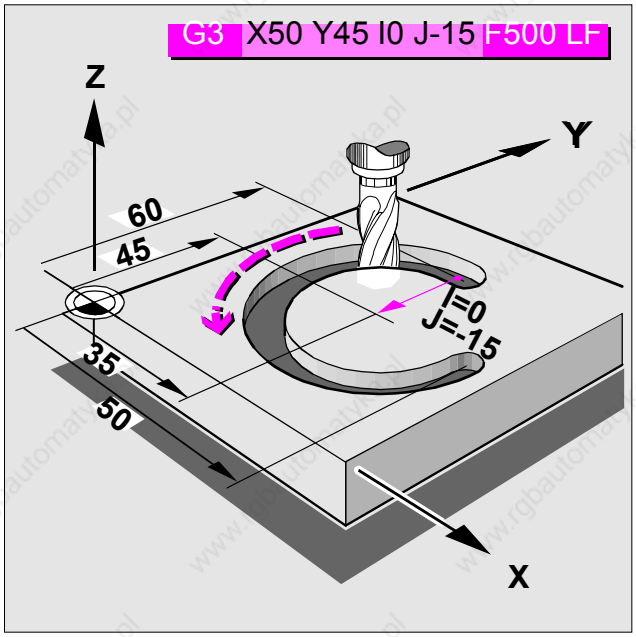
```
N5 G0 G90 X35 Y60 LF
N10 G3 X50 Y45 I0 J-15 F500 LF
```

Parameters

X, Y, Z	Coordinates of the circle end point
I, J, K	Interpolation parameters (directions: I in X, J in Y, K in Z) for determining the circle center point
AR	Opening angle

With G2 the tool travels clockwise, with G3 counterclockwise. Viewing direction along the third coordinate axis.
When programming with the opening angle the circle center point or circle end point must also be specified.

Milling:



Making a circular slot

© Siemens AG 2000. All rights reserved.

Circular interpolation, G2/G3 (cont.)

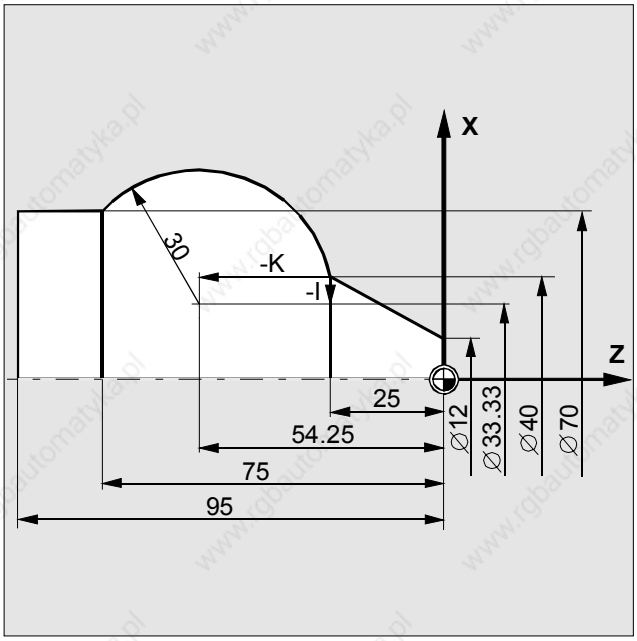
Programming

```
N10 G0 X12 Z0
N20 G1 X40 Z-25 F0.2
N30 G3 X70 Z-75 I-3.335 K-29.25
```

Parameters

X, Y, Z	Coordinates of the circle end point
I, K	Interpolation parameters (directions: I in X, K in Z) for determining the circle center point

Turning:



Making a ball end bolt

© Siemens AG 2000. All rights reserved.

Radius programming, G2/G3

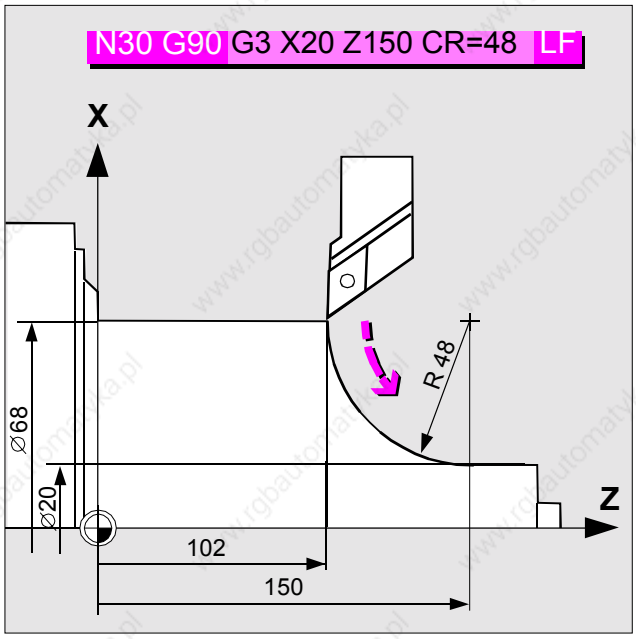
Programming

```
N20 G90 G0 X68 Z102
N30 G90 G3 X20 Z150 CR=48 F300 LF
```

Parameters

CR	Circle radius
CR+	Traversed angle ⇔ 180°
CR-	Traversed angle > 180°
X... Z...	Definition of end point

Radius programming is not allowed if the traversed angle is 360°.



Radius programming from drawing

© Siemens AG 2000. All rights reserved.

Circular interpolation through intermediate point, CIP

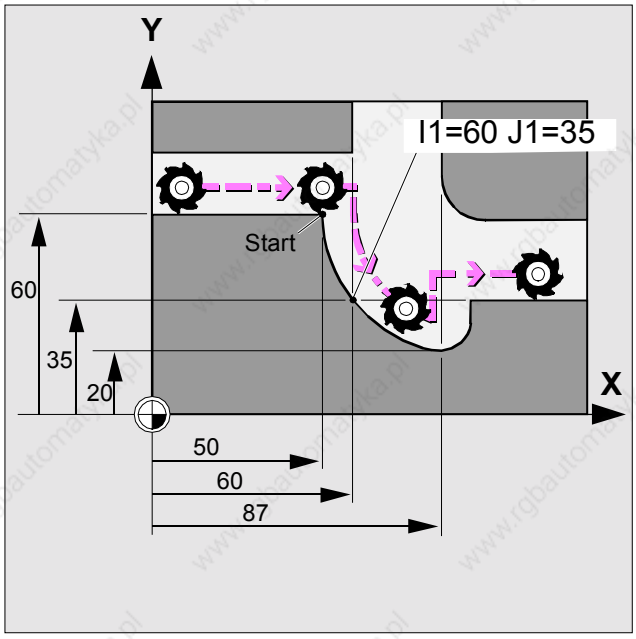
Programming N10 CIP X87 Y20 I1=60 J1=35 LF

Parameters

X, Y, Z	Coordinates of the circle end point
I1, J1, K1	Interpolation parameters for determining the intermediate point

If the circle parameter point is not given in the production drawing, you can program circular interpolations with CIP without additional calculations.
You can also use this function to program circles in space.

Milling:



Circular interpolation through intermediate point

© Siemens AG 2000. All rights reserved.

Circular interpol. through interm. point, CIP (cont.)

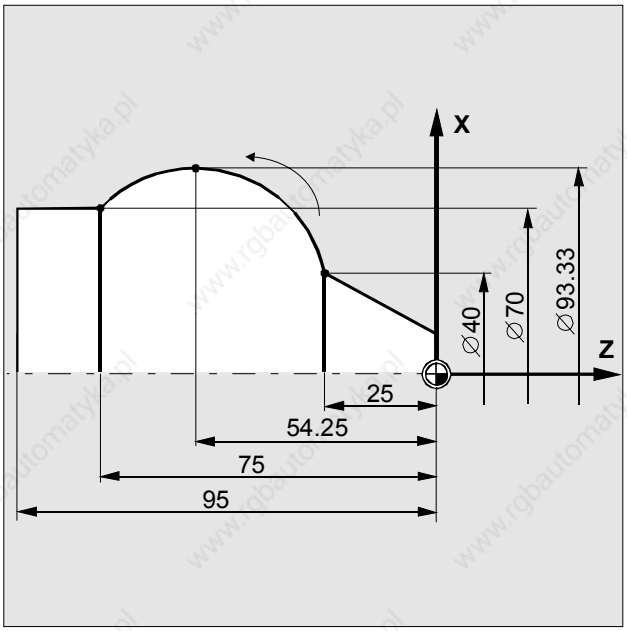
Programming

```
N90 G1 X40 Z-25
N100 CIP X70 Z-75 I1=93.33 K1=-54.25
```

Parameters

X, Z	Coordinates of the circle end point
I1, K1	Interpolation parameters for determining the intermediate point

Turning:



Circular interpolation through intermediate point

© Siemens AG 2000. All rights reserved.

Thread cutting, G33

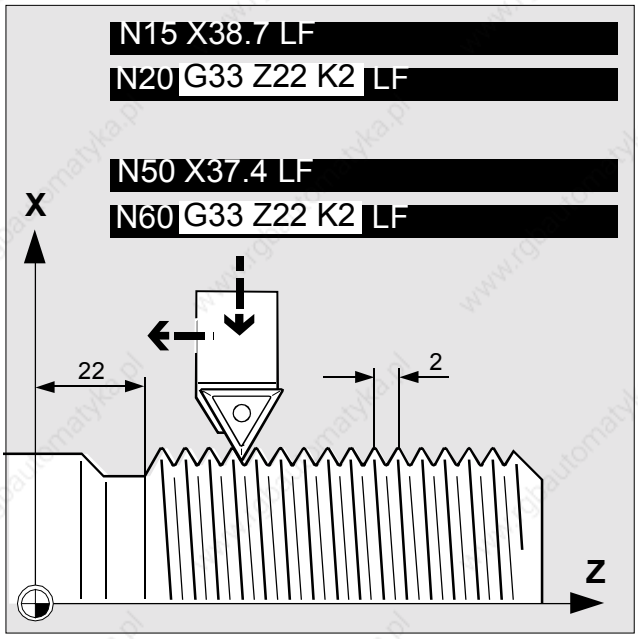
Programming N20 G33 Z22 K2 LF

Parameters

Z, X	Thread end point
K	Thread lead for cylindrical thread
I	Thread lead for face thread
I	Thread lead for taper thread (angle of taper > 45°)
K	Thread lead for taper thread (angle of taper < 45°)
SF	Start point offset in degrees

Right-hand or left-hand threads are programmed by specifying the direction of spindle rotation M3/M4. Spindle rotation and speed must be programmed in the block before G33.

For programming taper threads, enter the X and Z coordinates with G33. Multiple threads can be programmed with offset start points (SF=…).



Making a longitudinal thread

© Siemens AG 2000. All rights reserved.

Rigid tapping, G331/G332

Programming

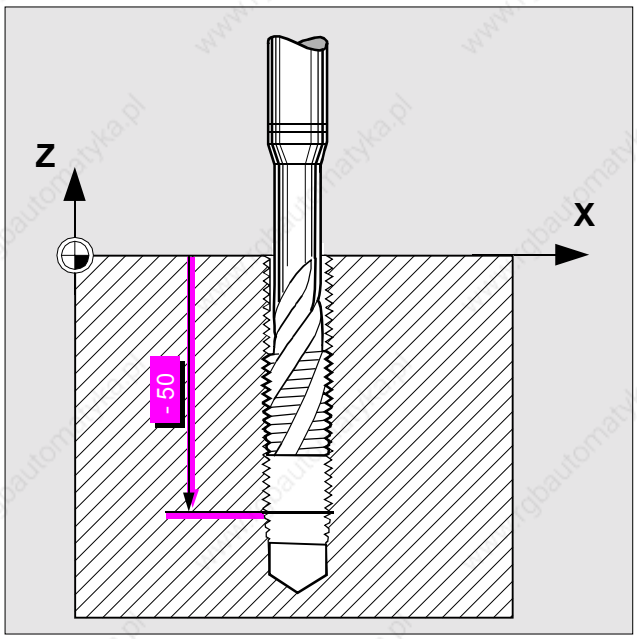
```
N40 SPOS=0 LF
N50 G331 Z-50 K2 S500 LF
N60 G332 Z5 K2 LF
```

SPOS=0	Change spindle to position control and put into position
G331	Tapping
G332	Tapping with retraction. The spindle changes direction of rotation automatically

Parameters

X, Y, Z	Thread end point
I, J, K	Thread lead. Positive lead (e.g. K4) right-hand thread, negative lead (e.g. K-4) left-hand thread.

For this function, the spindle must be equipped with a pulse generator.



Tapping

© Siemens AG 2000. All rights reserved.

Tapping with floating tapholder, G63

Programming

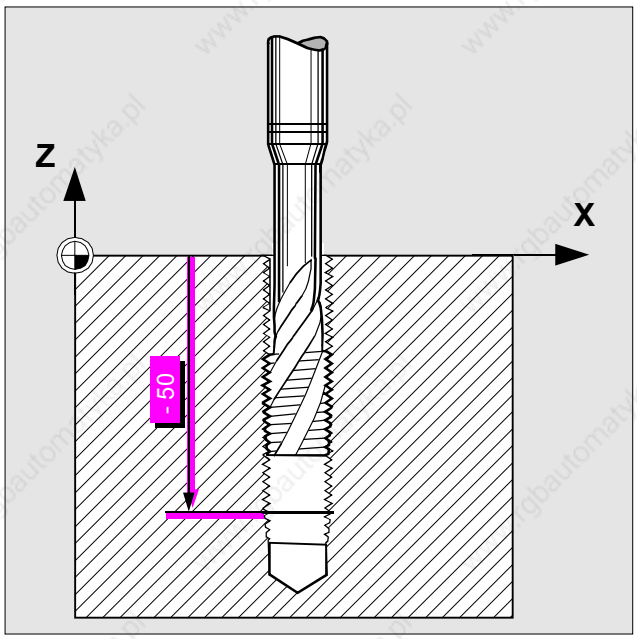
N10	G63 Z-50 M3	S...F...LF
N20	G63 Z4 M4	F...LF

G63 For the retraction movement, you program another block with G63 and the relevant direction of spindle rotation.

Parameters

S Spindle speed
F Feedrate
M3 Direction of rotation right
M4 Direction of rotation left
Calculation of feedrate:
F = Spindle speed x Thread lead

For this function, you need a tap in the floating tapholder. A pulse generator is not needed for the spindle.



Tapping with floating tapholder

© Siemens AG 2000. All rights reserved.

Polar coordinates G110, G111, G112

Programming

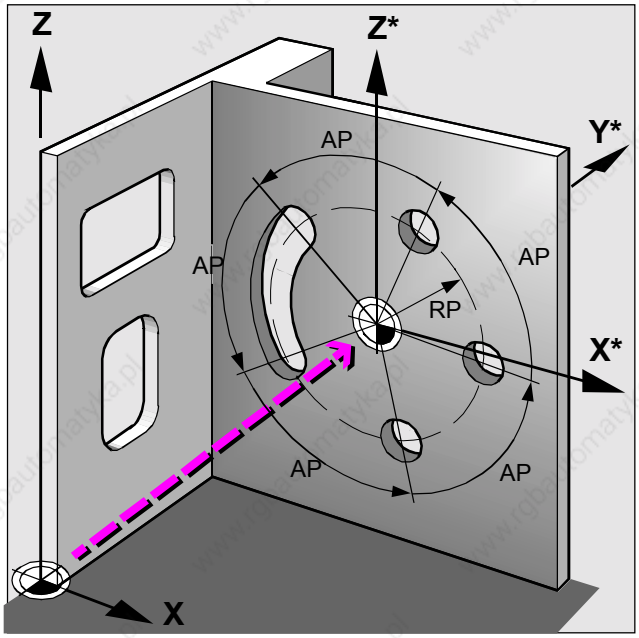
```
N30 G111 X40 Y35 Z40 LF
N40 G3 RP... AP...   LF
```

G110	Definition of pole, referred to tool position last programmed
G111	Definition of pole, absolute in the workpiece coordinate system
G112	Definition of pole, referred to pole last valid

Parameters

X, Y, Z	Coordinates of the pole
RP	Radius, distance between pole and target point
AP	Angle between path between pole and target point and the angle reference axis (pole axis first named)

The pole (center point) can be defined in rectangular or polar coordinates.
When programming the circle, the pole is in the circle center point and RP corresponds to the circle radius.



Description of travel paths using polar coordinates

© Siemens AG 2000. All rights reserved.

3. Tool Offsets and Compensations

Tool call	3-32
Cutter radius path compensation, G41/G42	3-33
Tool nose radius compensation, G41/G42	3-34
Activating/deactivating the collision detection	3-35
Approach and exit the contour, NORM/KONT	3-36
Programmable contour travel mode, G450/G451	3-37

© Siemens AG 2000. All rights reserved.

Tool call

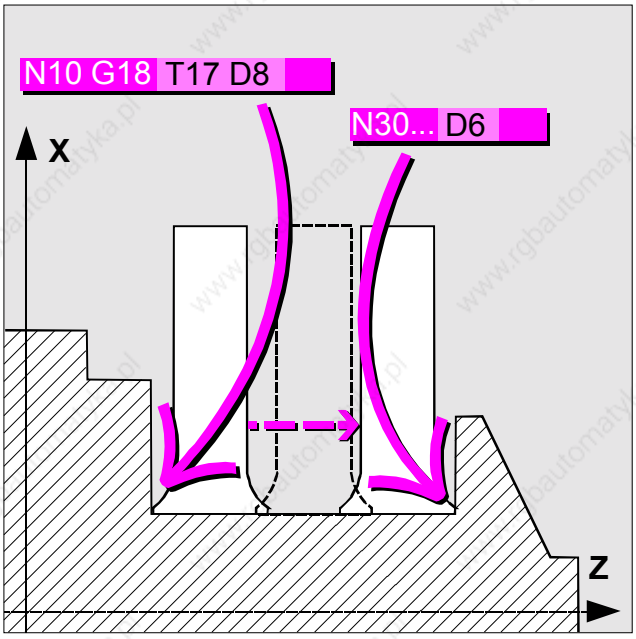
Programming N10 G18 T17 D8 LF

T... Call tool number
D... Call tool offset, activate tool length compensation

To make sure that the tool offset values are calculated correctly in the axes, the machining level must be selected before calling the tool.

Tool offset values can be exchanged in the course of the NC run. The machining plane does not have to be reprogrammed.

If no D number is to be input when the tool is called, a D number can be specified via machine data.



Offset values for left-hand and right-hand tool nose for recessing tool

© Siemens AG 2000. All rights reserved.

Cutter radius path compensation, G41/G42

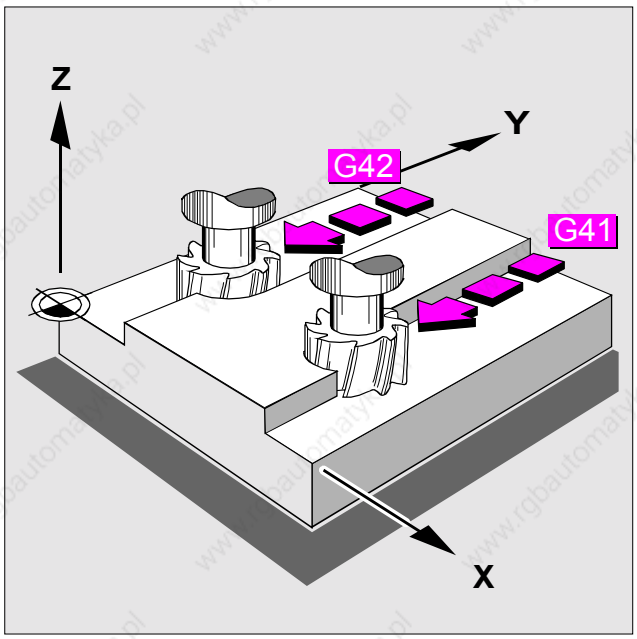
Programming N10 G1 G17 G41 D8 X... Y... Z... F500 LF

G41	Call for cutter path compensation, tool in traversing direction left of contour
G42	Call for cutter path compensation, tool in traversing direction right of contour
G40	Deselection of cutter path compensation

The tool length compensation acts automatically after tool offset D has been called.

In the NC block with G40/G41/G42 at least one axis of the selected working plane (G17 to G19) must be programmed.

Selection and deselection of the CRC must be made in a program block with G0 or G1.
The offset acts only in the programmed working plane (G17 to G19).



The control computes the tool path

© Siemens AG 2000. All rights reserved.

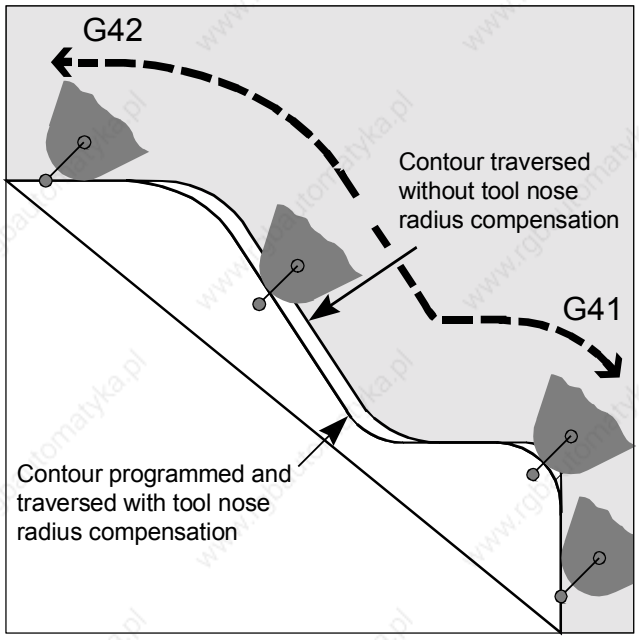
Tool nose radius compensation, G41/G42

Programming N5 G90 G0 G18 G41 D... X... Y... Z... LF

G41	Call for radius compensation, tool motion in traversing direction left of workpiece
G42	Call for radius compensation, tool motion in traversing direction right of workpiece
G40	Deselection of radius compensation

In the NC block with G40/G41/G42 at least one axis of the selected working plane (G17 to G19) must be programmed.

Selection and deselection of the compensation must be made in a program block with G0 or G1. The compensation acts only in the programmed working plane (G17 to G19).



Tool nose radius compensation for machining slopes and circular arcs

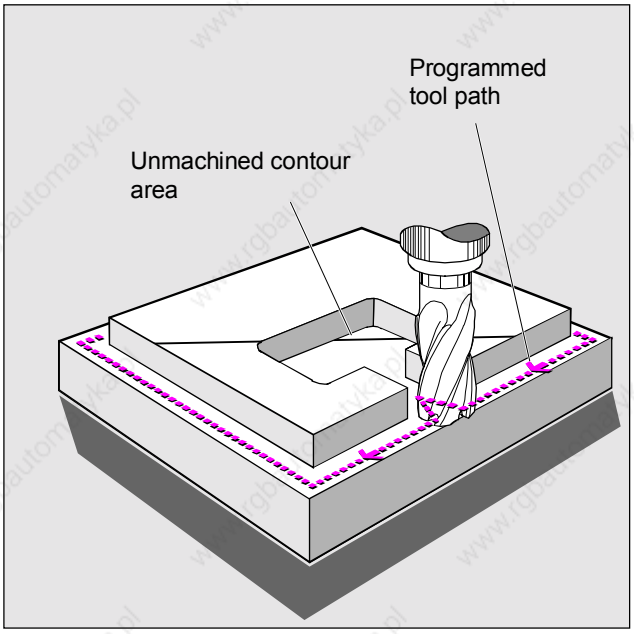
© Siemens AG 2000. All rights reserved.

Activating/deactivating the collision detection

Programming N10 G41 X...Y...Z... CDON LF

CDON Collision detection on
CDOF Collision detection off

The control detects and corrects critical machining situations.
Example: For machining the inside corner, a tool radius has been selected that is too high.



The control takes action in good time

© Siemens AG 2000. All rights reserved.

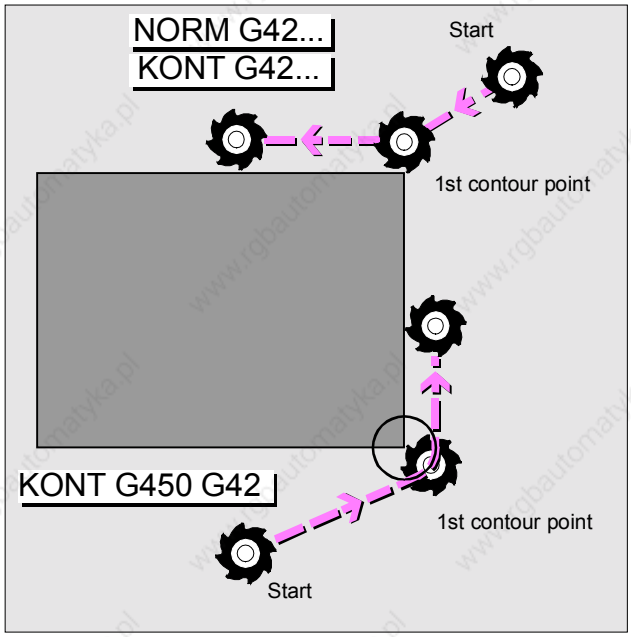
Approach and exit the contour, NORM/KONT

Programming KONT G41 G450 X... Y... Z... LF

NORM The tool travels directly along a straight line and is perpendicular to the contour point.

KONT The tool travels around the contour point in accordance with the programmed behavior at corners G450/G451.

For KONT: If start point and contour point are on one side of a workpiece, the contour point is approached as with NORM directly along a straight line.



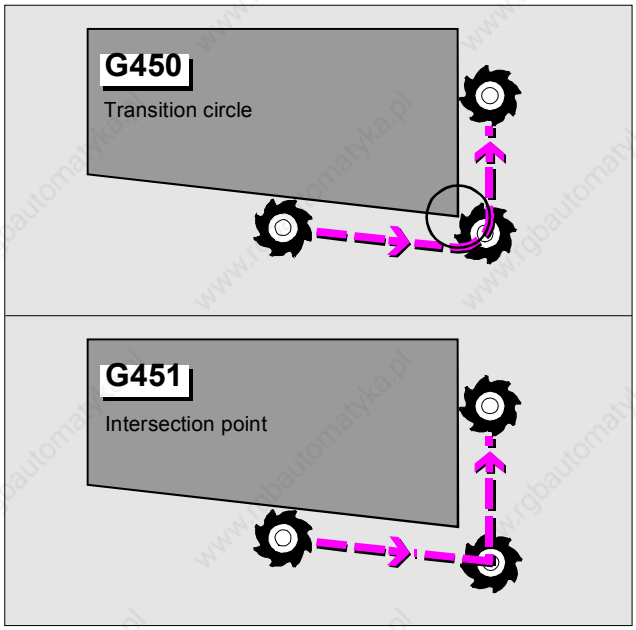
Programmable behavior for approach and exit

© Siemens AG 2000. All rights reserved.

Programmable contour travel mode, G450/G451

Programming N10 G41 G450 X... Y... Z... LF

G450	Transition circle, the tool travels around workpiece corners along a circular path with tool radius.
G451	Intersection point, the tool cuts free in the workpiece corner.



Tool travel behavior at workpiece corners

© Siemens AG 2000. All rights reserved.

© Siemens AG 2000. All rights reserved.

4. Programming Aids Assist the Programmer

Overview of frame concept 4-40

Translating/rotating the coordinate system, TRANS/ROT 4-42

Mirroring on the coordinate axes, MIRROR 4-44

Increasing/reducing size of contour, SCALE 4-45

© Siemens AG 1998. All rights reserved.

Overview of frame concept

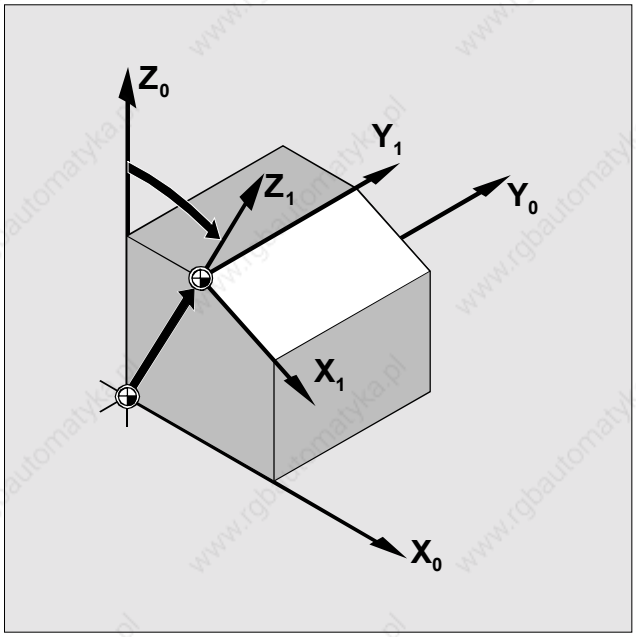
Programming

For the three-dimensional description of the workpiece coordinate system, the following functions are available.

TRANS/ATRANS	Translation of the zero point
ROT/AROT	Rotation
SCALE/ASCALE	Change of scale
MIRROR/AMIRROR	Mirroring

The actual coordinate system can be anywhere in space. This also allows skew contours to be produced.

Milling:

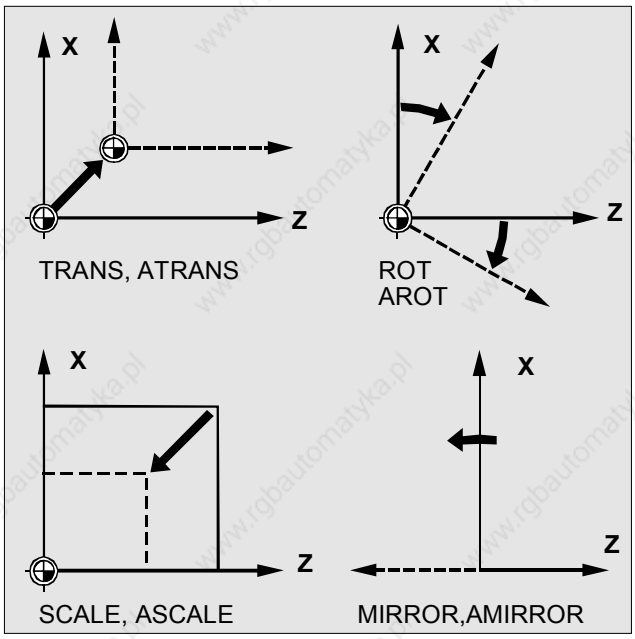


Programmable frames allow inclined contours to be machined

© Siemens AG 2000. All rights reserved.

Overview of frame concept (cont.)

Turning:



Programmable frames allow inclined contours to be machined

© Siemens AG 1998. All rights reserved.

Translating/rotating the coordinate system, TRANS/ROT

Programming

```
N30...G54 LF
N40  G90 TRANS X40 Y40 Z30  LF
N50  G90 AROT Z 30  LF
```

To switch off ZO: TRANS (without specifying axis)
To switch off rotation: ROT (without specifying angle)

In all cases, the complete frame is deleted here!

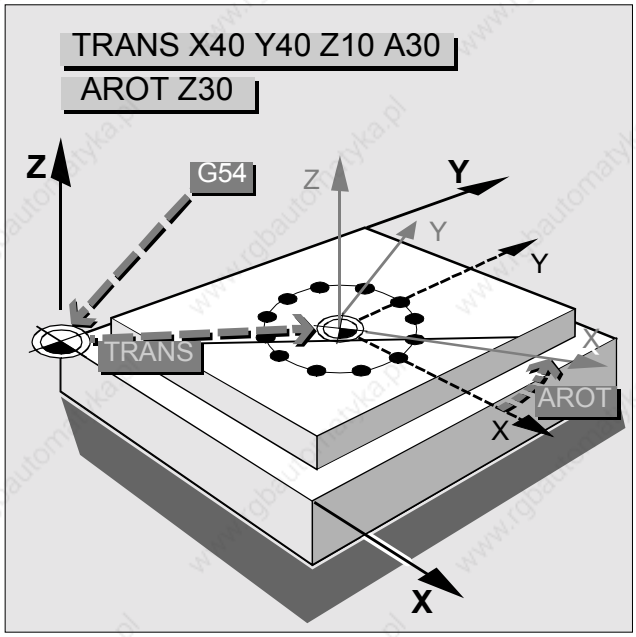
Parameters

TRANS	Absolute translation
ATRANS	Additive translation
X, Y, Z	Coordinates of zero offset in axial direction
ROT	Absolute rotation
AROT	Additive rotation

Parameters

X, Y, Z	Coordinate axis about which rotation occurs in angular degrees (positives sign = counterclockwise rotation)

Milling:



Changing the zero point for producing a drilling pattern

© Siemens AG 2000. All rights reserved.

Translating/rotating the coordinate system, TRANS/ROT

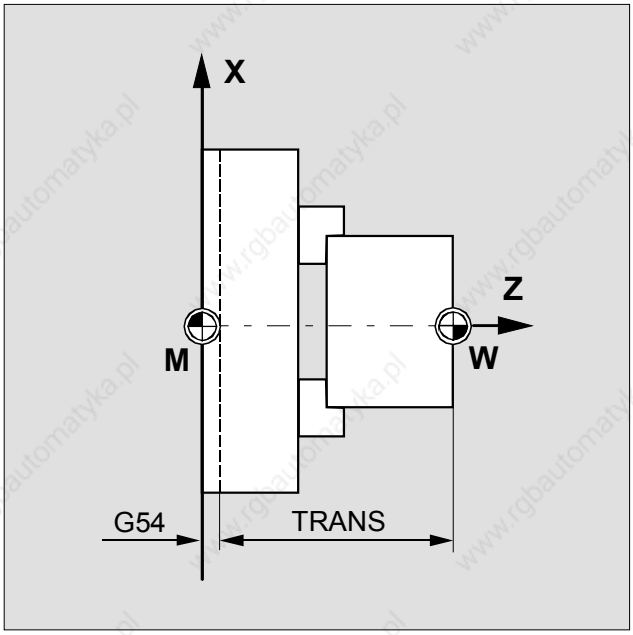
Programming

```
N30…G54
N40 TRANS Z150
```

Parameter	TRANS	Absolute translation
	ATRANS	Additive translation
	Z	Coordinates of zero offset in axial direction

On turning machines, a zero offset is usually sensible in the Z direction only.

Turning:



Changing the zero point in the Z direction

© Siemens AG 1998. All rights reserved.

Mirroring on the coordinate axes, MIRROR

Programming

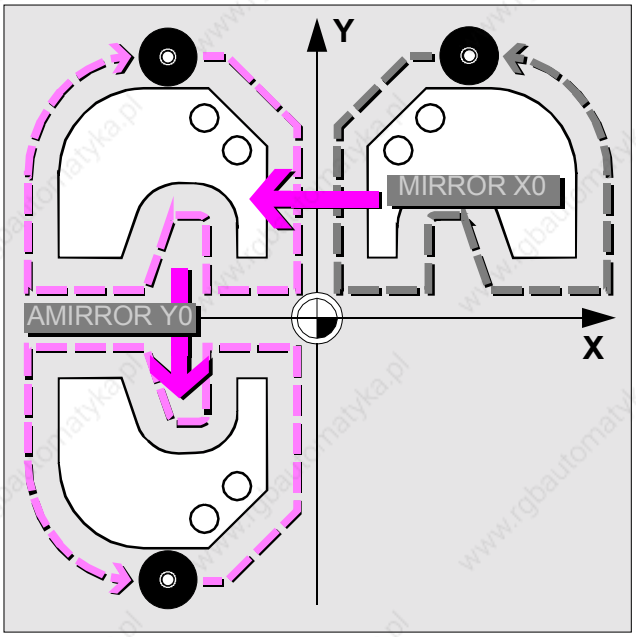
N10 MIRROR X0 LF
Switch off MIRROR (without defining axis)

In all cases, the complete frame is deleted here!

Parameters

MIRROR	Absolute mirroring
AMIRROR	Additive mirroring
X, Y, Z	Address with value 0 of the axis at which mirroring takes place.

When mirroring on a coordinate axis, the control changes
- the signs of the mirrored coordinates,
- the direction of rotation of circular interpolations and
- the machining directions (G41/G42).



No additional programming for symmetrical contours

© Siemens AG 2000. All rights reserved.

Increasing/reducing size of contour, SCALE

Programming

N10 SCALE X2 Y2 LF
Switch off SCALE (without defining axis)

In all cases, the complete frame is deleted here!

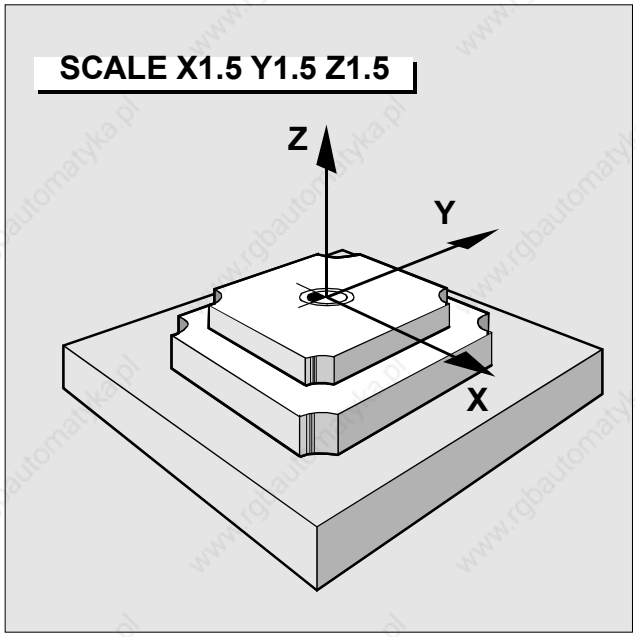
Parameters

SCALE	New scale factor
ASCALE	Additive scale factor
X, Y, Z	Axes with scale factor in the direction of which the contour is to be increased or reduced in size.

If transformation follows with ATRANS, the offset values are also scaled.

Any contours that you wish to increase or reduce in size are best defined in a subroutine.

You can define an individual scale factor for each axis.



No additional programming for similar contours

© Siemens AG 1998. All rights reserved.

© Siemens AG 2000. All rights reserved.

5. Programming Preparatory Functions

Feedrate programming, G93 to G97 5-48

Exact stop, G9/G60 5-49

Feedrate in continuous path mode, G64, G641 5-50

Programming the spindle motion 5-51

© Siemens AG 2000. All rights reserved.

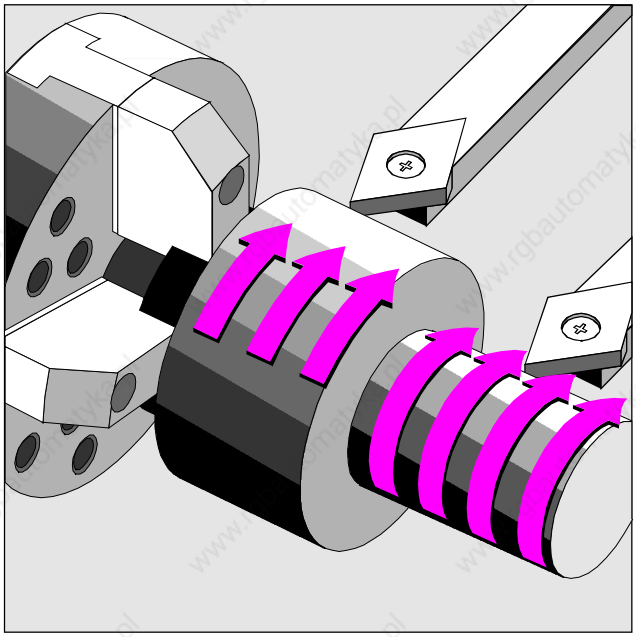
Feedrate programming, G93 to G97

Programming

```
N5 G90 G00 X... Y... Z... LF
N10   G94 F500   G01...M3 LF
```

G93 F	Feedrate in 1/min (inverse time feedrate)
G94 F	Constant speed and feedrate in mm/min
G95 F	Constant speed and feedrate in mm/revolution
G96 S	Constant cutting velocity in m/min
F	Feedrate in mm/revolution
G97	Switch off G96, store the last speed setpoint of G96 as constant speed.

The maximum feedrates and speeds are defined by the machine manufacturer.



Automatic speed control for constant cutting velocity

© Siemens AG 2000. All rights reserved.

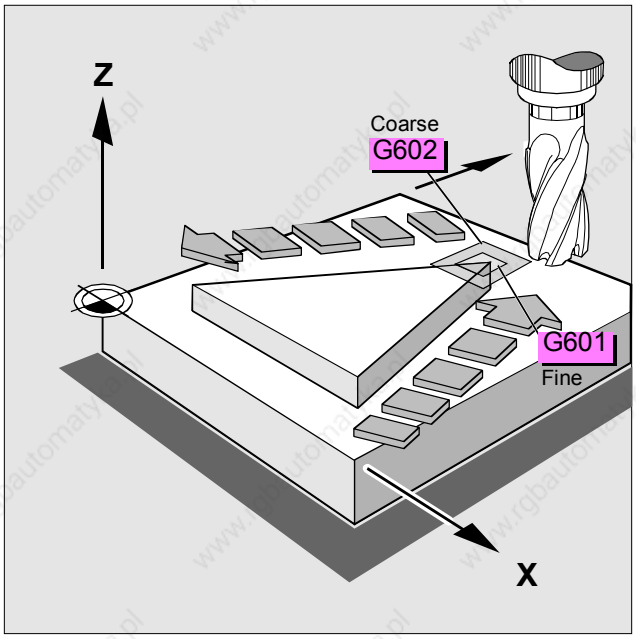
Exact stop, G9/G60

Programming

G601 Exact positioning fine
G602 Exact positioning coarse
G603 Exact positioning at end of interpolation
G9 Exact positioning, active in the block
G60 Exact positioning, modal, active until deselected by G64, G641.

The exact positioning functions are used in order to produce sharp outside corners or to finish inside corners to the required dimension.

The exact positioning limits are defined in the machine data.



Producing sharp outside corners

© Siemens AG 2000. All rights reserved.

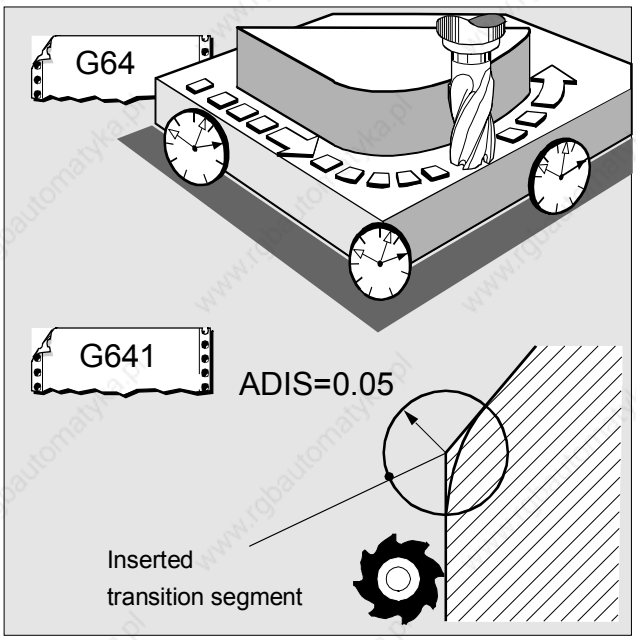
Feedrate in continuous path mode, G64, G641

Programming

G64 Continuous path mode

G641 Block transition with contour transition rounding.
The rounding distance (in mm) is programmed via
ADIS = for G1, G2, G3 or
ADISPOS = for G0.

Both functions work with "look ahead" velocity control.



Optimization of the production results

© Siemens AG 2000. All rights reserved.

Programming the spindle motion

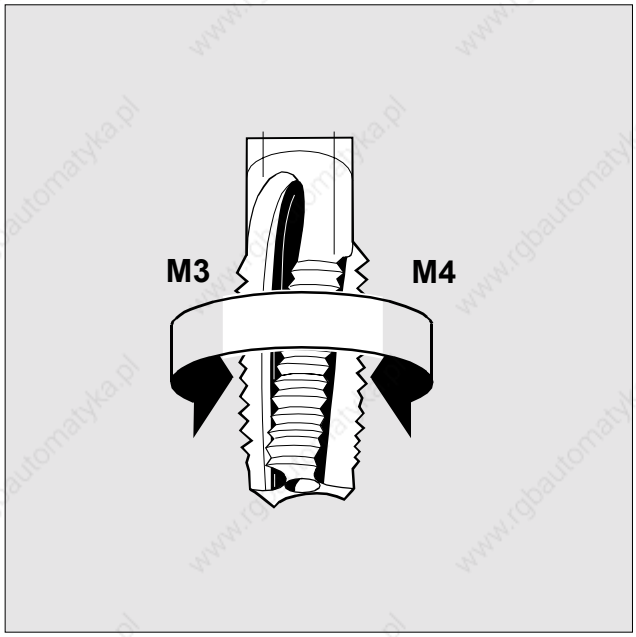
Programming

```
N05 SETMS(2) LF
N10 G1 F300 X70 Y20 S270 M3 LF
```

Parameters

S, Sn	Spindle speed in rpm
M3, Mn=3	Clockwise rotation
M4, Mn=4	Counterclockwise rotation
M5, Mn=5	Spindle stop
SETMS(n)	Spindle n becomes master spindle
n	Spindle number

If the M commands are programmed in a block with axial motion, the commands before the axial motion are effective.



Programming the direction of spindle rotation

© Siemens AG 2000. All rights reserved.

© Siemens AG 2000. All rights reserved.

6. Subroutine Technique and Cycles

Subroutine technique	6-55
The use of cycles	6-57
Explanation of parameters	6-58
Drilling, centering, CYCLE81	6-62
Drilling, counterboring, CYCLE82	6-63
Deep hole drilling, CYCLE83	6-64
Rigid tapping, CYCLE84	6-65
Tapping with floating tapholder, CYCLE840	6-66
Bore 1, CYCLE85	6-67
Bore 2, CYCLE86	6-68
Bore 3, CYCLE87	6-69
Bore 4, CYCLE88	6-70
Bore 5, CYCLE89	6-71
Row of holes, HOLES1	6-72

© Siemens AG 2000. All rights reserved.

Hole circle, HOLES2 6-73

Elongated holes on a circle, LONGHOLE 6-74

Slots on a circle, SLOT1 6-75

Circumferential slot, SLOT2 6-76

Rectangular pocket milling, POCKET1 6-77

Circular pocket milling, POCKET2 6-78

Thread cutting, CYCLE90 6-79

Recessing cycle, CYCLE93 6-80

Undercut cycle, CYCLE94 6-81

Stock removal cycle, CYCLE95 6-82

Thread undercut, CYCLE96 6-83

Thread cutting, CYCLE97 6-84

Chaining of threads, CYCLE98 6-85

© Siemens AG 2000. All rights reserved.

Subroutine technique

Programming

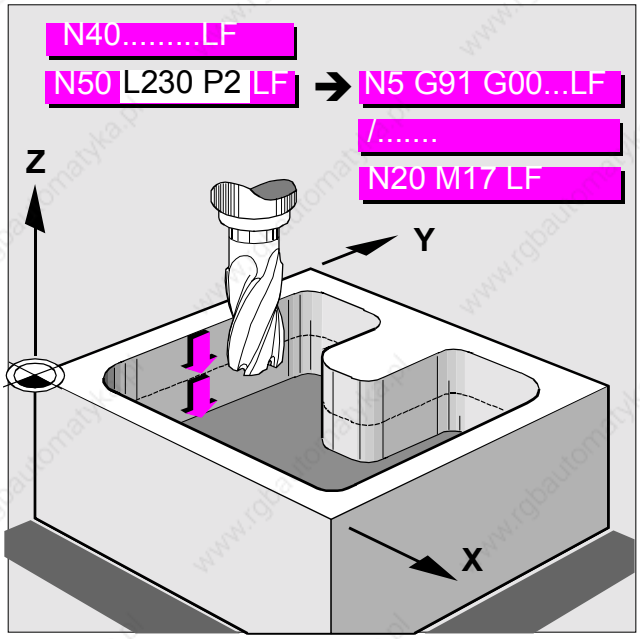
```
N40 G0 X500 Y500 Z500 LF
N50 L230 P2 LF
```

L... Subroutine call
P... Number of repeats (max. 9999)

Subroutine nesting: nesting to a depth of eleven

The end of the subroutine and the return jump to the main program is programmed with M17. The subroutine must be called in a separate NC block.

Milling:



Machining in several steps

© Siemens AG 2000. All rights reserved.

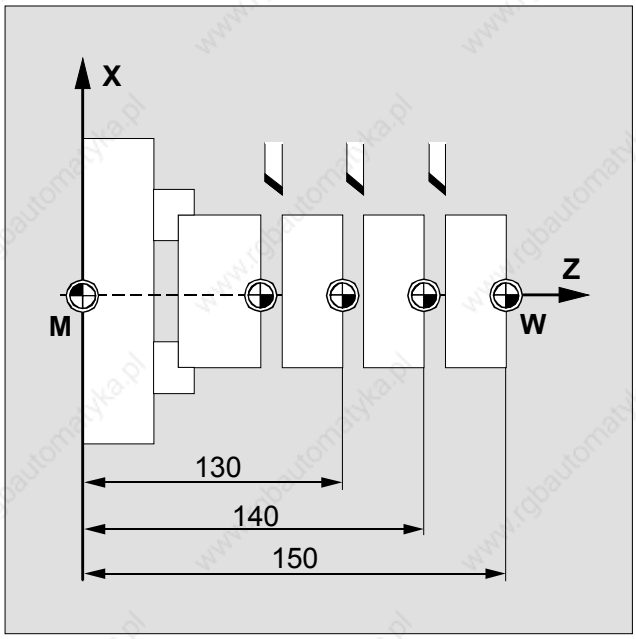
Subroutine technique (cont.)

Programming

```
N10 TRANS X0 Z150
N20   L20
N30 TRANS X0 Z140
N40   L20
N50 TRANS X0 Z130
N60   L20
```

L... Subroutine call

Turning:



Machining in several steps

© Siemens AG 2000. All rights reserved.

The use of cycles

Precondition

The machining plane (G17, G18, G19) feed and speed must be defined before calling the cycle.

Cycle call

Cycle calls must always be in a separate block!

For example:
CYCLE81 (RTP, RFP, SDID, DP, DPR) L_F

Programming support for cycles

The program editor in the control offers a programming support function for generating cycle calls.

You will find a detailed description of all cycles in the Programming Guides for the technologies turning and milling/drilling.

© Siemens AG 2000. All rights reserved.

Explanation of parameters

Parameter	Explanation
AFSL	Angle for slot length (enter without sign)
ANG1	Flank angle 1: at the side of the groove defined by the start point (enter without sign) Values: 0<=ANG1<89.999 degrees
ANG2	Flank angle 2: at other side (enter without sign) Values: 0<=ANG2<89.999
APP	Run-in path (enter without sign)
CDIR	Direction of machining Values: 2 (for G2) 3 (for G3)
CPA	Center point circle/pocket abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
CRAD	Corner radius (enter without sign)
DAM	CYCLE95: Path length after which each roughing cut is interrupted for chip breaking CYCLE83: Amount of degression (enter without sign)
DBH	Distance between holes (enter without sign)
DIAG	Groove depth (enter without sign)
DIATH	Nominal thread diameter, outer diameter of thread
DM1	Diameter of thread at start point
DM2	CYCLE97: Diameter of thread at end point CYCLE98: Diameter at first intermediate point
DM3	Diameter at second intermediate point
DM4	Diameter at end point
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DT	Dwell time for chip breaking during roughing
DTB	CYCLE82, -83, -86, -88, -89: Dwell time at final drilling depth (chip breaking) CYCLE93: Dwell time at recess base CYCLE84, -840: Dwell time at thread depth (chip breaking) CYCLE85: Dwell time at drilling depth (chip breaking)
DTS	Dwell time at start point and for stock removal
ENC	Tapping with/without encoder Values: 0 = with encoder 1 = without encoder
FAL	Correct finishing allowance for contour (enter without sign)
FAL1	Finishing allowance at recess base
FAL2	Finishing allowance at edges
FALX	Finishing allowance in face axis (enter without sign)
FALZ	Finishing allowance in longitudinal axis (enter without sign)
FDEP	First drilling depth (absolute)

© Siemens AG 2000. All rights reserved.

Explanation of parameters (cont.)

Parameter	Explanation
FDIS	Distance of first hole from reference point (enter without sign)
FDPR	First drilling depth relative to reference plane (enter without sign)
FF1	Feedrate for roughing without relief cut
FF2	Feedrate for insertion in relief cut elements
FF3	Feedrate for finishing
FFD	Feedrate for depth infeed
FFP1	Feedrate for surface machining
FFP2	Feedrate for final allowance
FFR	Feedrate
FORM	Definition of form CYCLE94: E (for Form E) F (for Form F) CYCLE96: A (for Form A) D (for Form D)
FPL	End point of thread in longitudinal axis
FRF	Feedrate factor for first drilling depth (enter without sign) Values: 0.001 ... 1
IANG	Insertion angle Values: "+" (for edge infeed) "–" (for alternating edge infeed)
IDEP	Insertion depth (enter without sign)
INDA	Indexing angle
KDIAM	Core diameter, inside diameter of thread
LENG	Length of elongated hole/slot length/pocket length (enter without sign)
MID	Maximum insertion depth for insertion (enter without sign)
MIDF	Maximum insertion depth for final allowance
MPIT	Thread pitch as thread size CYCLE97: 3 (for M3) ... 60 (for M60) CYCLE84, -840: 3 (for M3) ... 48 (for M48)
NID	Number of noncuts (enter without sign)
NPP	Name of the contour subprogram
NRC	Number of roughing cuts (enter without sign)
NSP	Start point offset for first thread start (enter without sign)
NUM	Number of holes/elongated holes/slots
NUMTH	Number of thread starts (enter without sign)
PIT	Thread pitch; values: 0.001 ... 2000.000 mm
PRAD	Pocket radius (enter without sign)
PO1	Start point of thread in the longitudinal axis
PO2	First intermediate point in the longitudinal axis

© Siemens AG 2000. All rights reserved.

Explanation of parameters (cont.)

Parameter	Explanation
PO3	Second intermediate point
PO4	End point of thread in the longitudinal axis
POSS	Spindle position for oriented spindle stop in cycle (in degrees)
PP1	Thread pitch 1 as value (enter without sign)
PP2	Thread pitch 2 as value (enter without sign)
PP3	Thread pitch 3 as value (enter without sign)
PRAD	Pocket radius (enter without sign)
RAD	Radius of circle (enter without sign)
RCI1	Radius/chamfer 1, inside: at start point side
RCI2	Radius/chamfer 2, inside
RCO1	Radius/chamfer 1, outside: at the side defined by the start point
RCO2	Radius/chamfer 2, outside
RFF	Return feedrate
RFP	Reference plane (absolute)
ROP	Exit path (enter without sign)
RPA	Return path in abscissa of the active plane (enter incrementally with sign)
RPAP	Return plane in the applicate (enter incrementally with sign)
RPO	Return path in the ordinate of the active plane (enter incrementally with sign)
RTP	Return plane (absolute)
SDAC	Direction of rotation after end of cycle Values: 3, 4 or 5
SDIR	Direction of rotation Values: 3 (for M3) 4 (for M4)
SDIS	Safety distance (enter without sign)
SDR	Direction of rotation for return Values: 0 (automatic reversal of direction of rotation) 3 or 4 (for M3 or M4)
SPCA	Abscissa of a reference point on a straight line (absolute)
SPCO	Ordinate of this reference point (absolute)
SPD	Start point in the facing axis (enter without sign)
SPL	Start point of contour/thread in the longitudinal axis
SSF	Speed during finishing
SST	Speed for tapping
SST1	Speed for return
STA1	Starting angle Values: -180 to 180 degrees
TDEP	Thread depth (enter without sign)
TYPTH	Thread type: values: 0=inside thread 1=outside thread

© Siemens AG 2000. All rights reserved.

Explanation of parameters (cont.)

Parameter	Explanation
VARI	Machining mode SLOT1, SLOT2, POCKET1, POCKET2: 0=complete machining 1=solid machining to finishing allowance 2=only machining of finishing allowance CYCLE83: 0=chip breaking 1=stock removal CYCLE93: 1...8 11...18 CYCLE95: 1...12 CYCLE97, CYCLE98: 1...4
WID	Slot width/pocket width (enter without sign)
WIDG	Slot width (enter without sign)

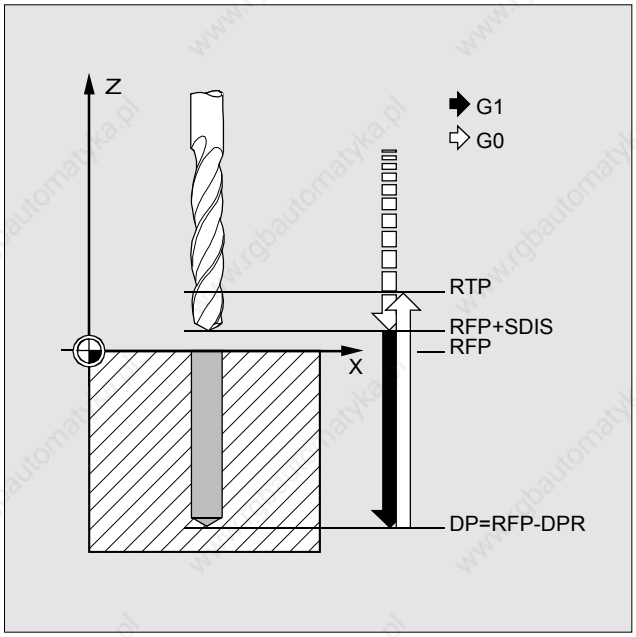
© Siemens AG 2000. All rights reserved.

Drilling, centering, CYCLE81

Sequence The "Drilling, centering" cycle produces a simple hole.

Programming CYCLE81 (RTP, RFP, SDIS, DP, DPR)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)



Sequence of motions and parameters in the "Drilling, centering" cycle

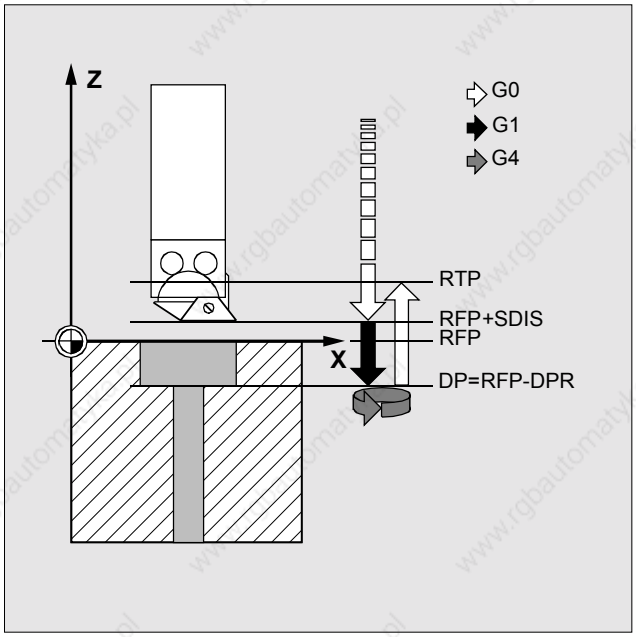
© Siemens AG 2000. All rights reserved.

Drilling, counterboring, CYCLE82

Sequence The "Drilling, counterboring" cycle produces a simple hole. When the final drilling depth has been reached, a dwell time can be activated.

Programming CYCLE82 (RTP, RFP, SDIS, DP, DPR, DTB)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at final drilling depth (chip breaking)



Sequence of motions and parameters in the "Drilling, counterboring" cycle

© Siemens AG 2000. All rights reserved.

Deep hole drilling, CYCLE83

Sequence

The "Deep-hole drilling" cycle produces holes up to the final drilling depth by depth infeed in several steps the maximum amount of which can be programmed.

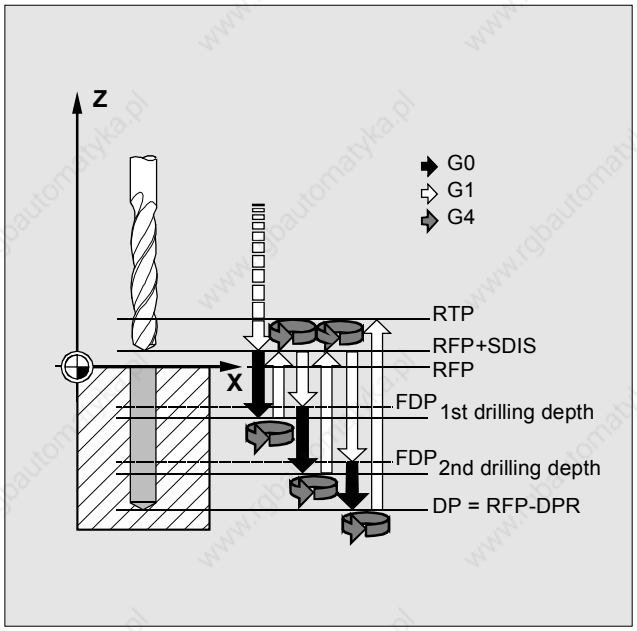
The drill can either be withdrawn from the reference plane after each infeed depth for stock removal or by 1 mm each time for chip breaking.

Programming

CYCLE83 (RTP, RFP, SDIS, DP, DPR, FDEP, FDPR, DAM, DTB, DTS, FRF, VARI)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at final drilling depth (chip breaking)
FDEP	First depth (absolute)

Other parameters: see Chapter "Explanation of parameters"



Sequence of motions and parameters in the "Deep-hole drilling" cycle with stock removal

© Siemens AG 2000. All rights reserved.

Rigid tapping, CYCLE84

Sequence

CYCLE84 produces tapped holes without using a floating tapholder.

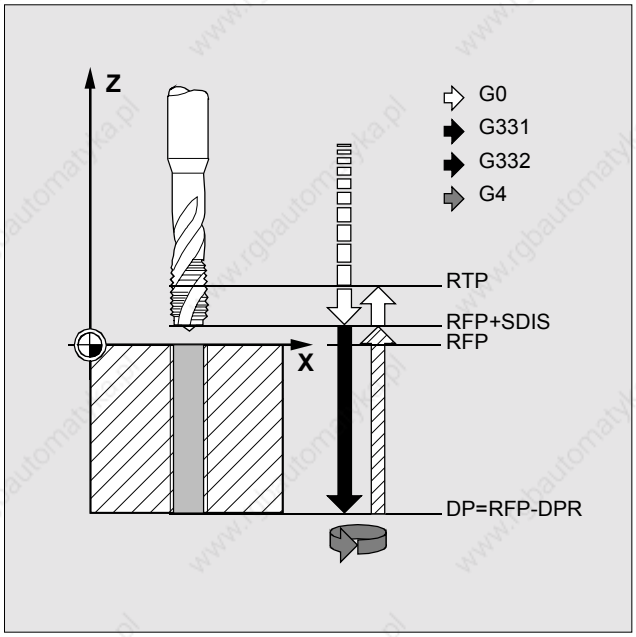
This cycle can be applied only if the spindle designated for drilling is capable of working in the position-controlled spindle mode.

Programming

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

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at final drilling depth (chip breaking)

Other parameters: see Chapter "Explanation of parameters"



Sequence of motions and parameters in the "Rigid tapping" cycle

© Siemens AG 2000. All rights reserved.

Tapping with floating tapholder, CYCLE840

Sequence

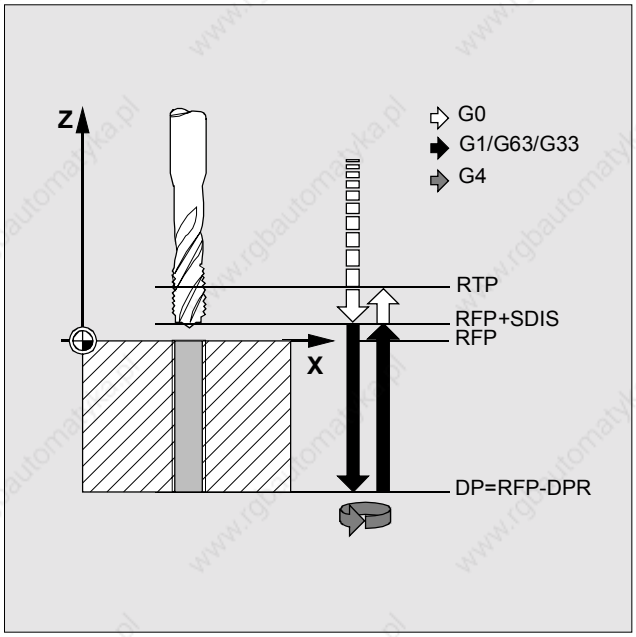
With this cycle, tapped holes can be produced with floating tapholder
- without encoder and
- with encoder.

For tapping without floating tapholder (rigid taping) another cycle is used (CYCLE84, see previous page).

Programming

CYCLE840 (RTP, RFP, SDIS, DP, DPR, DTB, SDR, SDAC, ENC, MPIT, PIT)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at final drilling depth (chip breaking)
	Other parameters: see Chapter "Explanation of parameters"



Sequence of motions and parameters in the "Tapping with floating tapholder/encoder" cycle

© Siemens AG 2000. All rights reserved.

Bore 1, CYCLE85

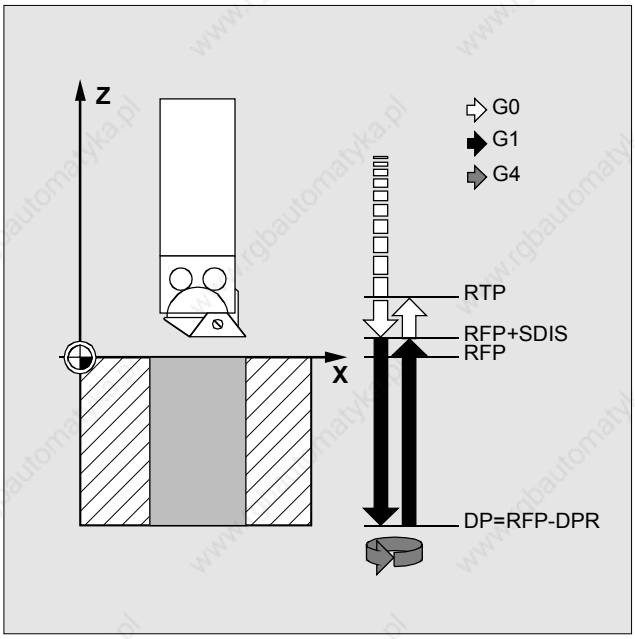
Sequence

In the "Bore 1" cycle, the inward and outward motions are performed at a feedrate that must be specified under the relevant parameters.

Programming

CYCLE85 (RTP, RFP, SDIS, DP, DPR, DTB, FFR, RFF)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at final drilling depth (chip breaking)
FFR	Feedrate
RFF	Return feedrate



Sequence of motions and parameters in the "Bore 1" cycle

© Siemens AG 2000. All rights reserved.

Bore 2, CYCLE86

Sequence

In the "Bore 2" cycle, an oriented spindle stop is performed on reaching the drilling depth. The tool then travels at rapid traverse to the programmed return positions and from there to the return plane.

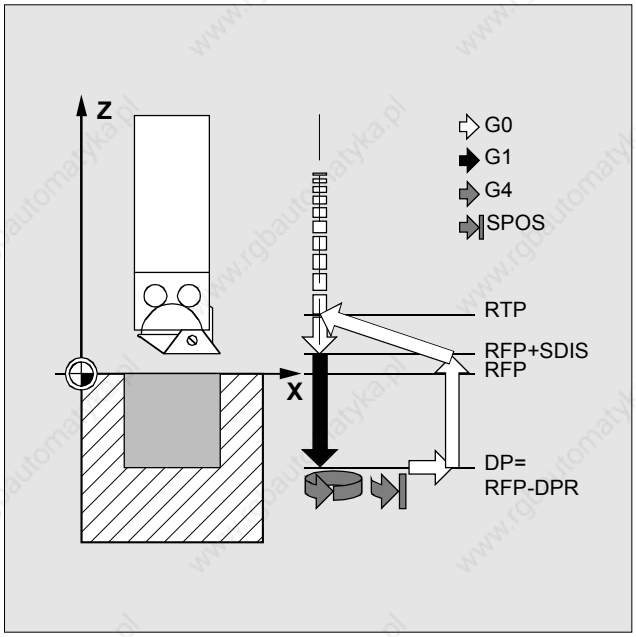
This cycle can only be used if the spindle designated for drilling is technically capable of operating in the position-controlled spindle mode.

Programming

CYCLE86 (RTP, RFP, SDIS, DP, DPR, DTB, SDIR, RPA, RPO, RPAP, POSS)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at Final drilling depth (chip breaking)

Other parameters: see Chapter "Explanation of parameters"



Sequence of motions and parameters in the "Bore 2" cycle

© Siemens AG 2000. All rights reserved.

Bore 3, CYCLE87

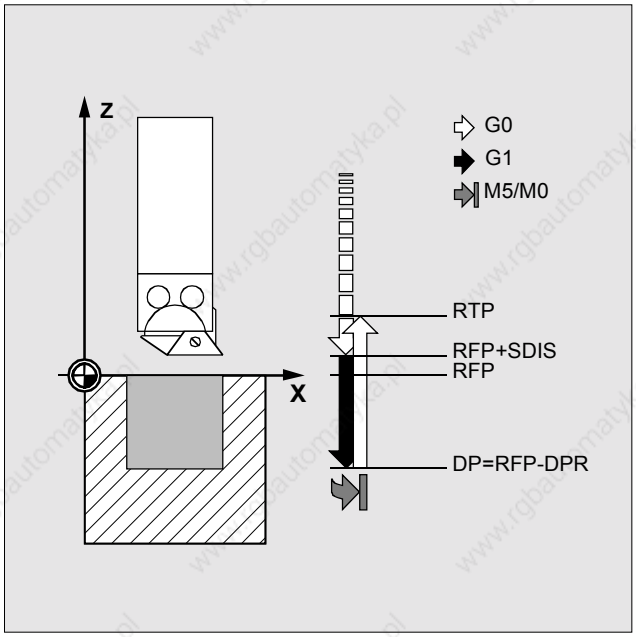
Sequence

In the "Bore 3" cycle, a spindle stop without orientation is effected on reaching the final drilling depth, followed by a programmed stop. By pressing the NC Start key, the upwards movement is continued at rapid traverse until the retraction plane is reached.

Programming

CYCLE87 (RTP, RFP, SDIS, DP, DPR, SDIR)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
SDIR	Direction of rotation 3 (for M3) 4 (for M4)



Sequence of motions and parameters in the "Bore 3" cycle

© Siemens AG 2000. All rights reserved.

Bore 4, CYCLE88

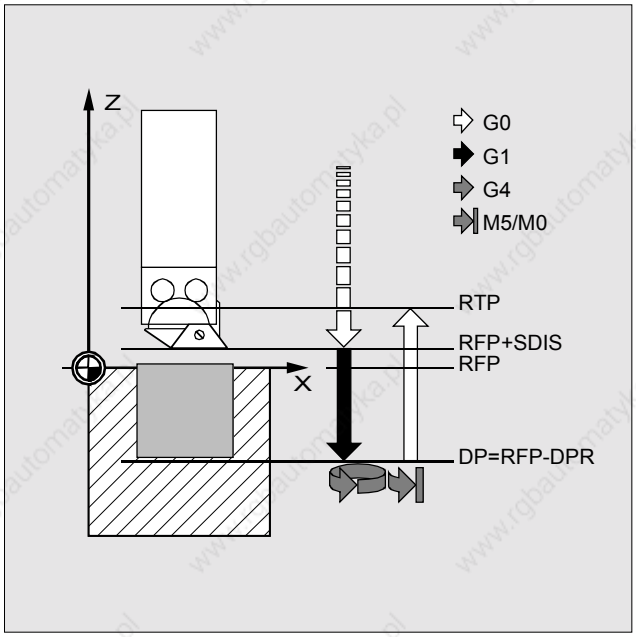
Sequence

In the "Bore 4" cycle, a dwell time is effected on reaching the final drilling depth together with a spindle stop without orientation and a programmed stop. By pressing the NC Start key, the upwards movement is performed at rapid traverse until the retraction plane is reached.

Programming

CYCLE88 (RTP, RFP, SDIS, DP, DPR, DTB, SDIR)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	Dwell time at final drilling depth (chip breaking)
SDIR	Direction of rotation 3 (for M3) 4 (for M4)



Sequence of motions and parameters in the "Bore 4" cycle

© Siemens AG 2000. All rights reserved.

Bore 5, CYCLE89

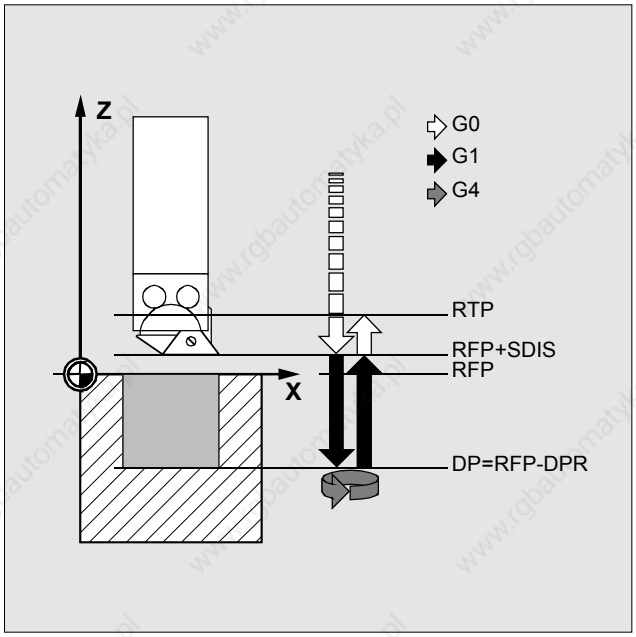
Sequence

In the "Bore 5" cycle, the inwards and the upwards movements are performed at the F value programmed before the cycle call. On reaching the final drilling depth, a dwell time can be effected.

Programming

CYCLE89 (RTP, RFP, SDIS, DP, DPR, DTB)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DTB	CYCLE82: Dwell time at final drilling depth (chip breaking)



Sequence of motions and parameters in the "Bore 5" cycle

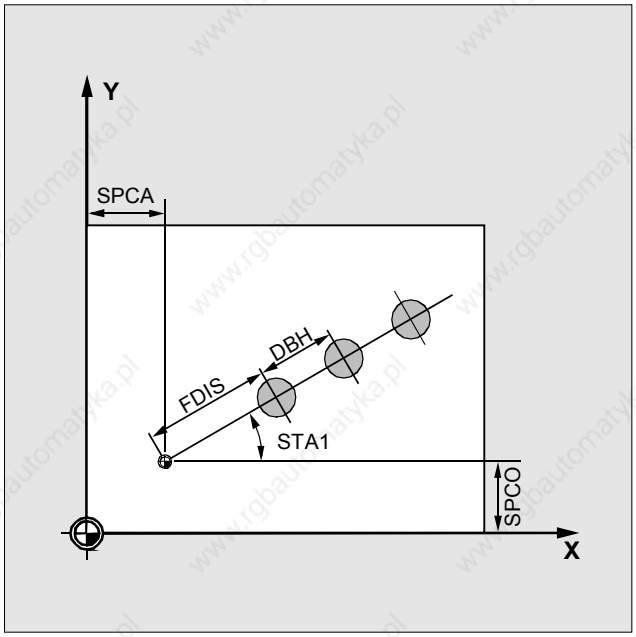
© Siemens AG 2000. All rights reserved.

Row of holes, HOLES1

Sequence With this cycle, a row of holes can be produced on a straight line.

Programming HOLES1 (SPCA, SPCO, STA1, FDIS, DBH, NUM)

SPCA	Abscissa of a reference point on the straight line (absolute)
SPCO	Ordinate of this reference point (absolute)
STA1	Starting angle Values: -180 to 180 degrees
FDIS	Distance to first hole from reference point (enter without sign)
DBH	Distance between holes (enter without sign)
NUM	Number of holes/elongated holes/slots



Parameters in the "Row of holes" drilling pattern

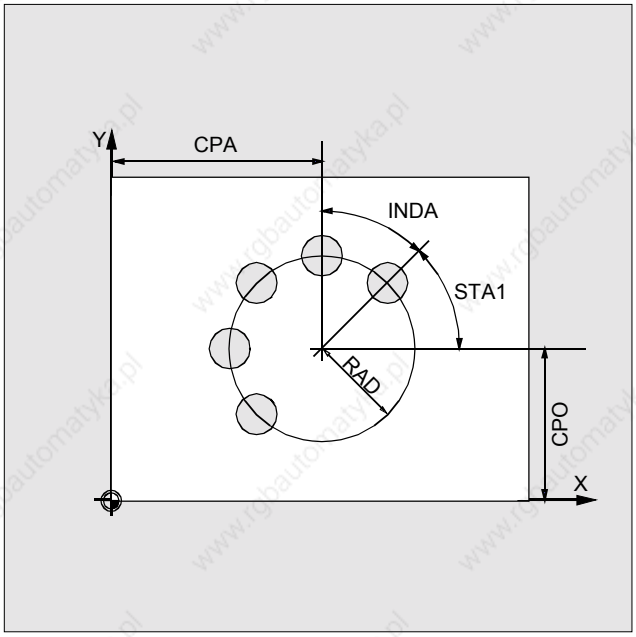
© Siemens AG 2000. All rights reserved.

Hole circle, HOLES2

Sequence With this cycle, a hole circle can be machined.

Programming HOLES2 (CPA, CPO, RAD, STA1, INDA, NUM)

CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
RAD	Radius of circle (enter without sign)
STA1	Starting angle Values: -180 to 180 degrees
INDA	Indexing angle
NUM	Number of holes/elongated holes/slots



Parameters in the "Hole circle" drilling pattern

© Siemens AG 2000. All rights reserved.

Elongated holes on a circle, LONGHOLE

Sequence

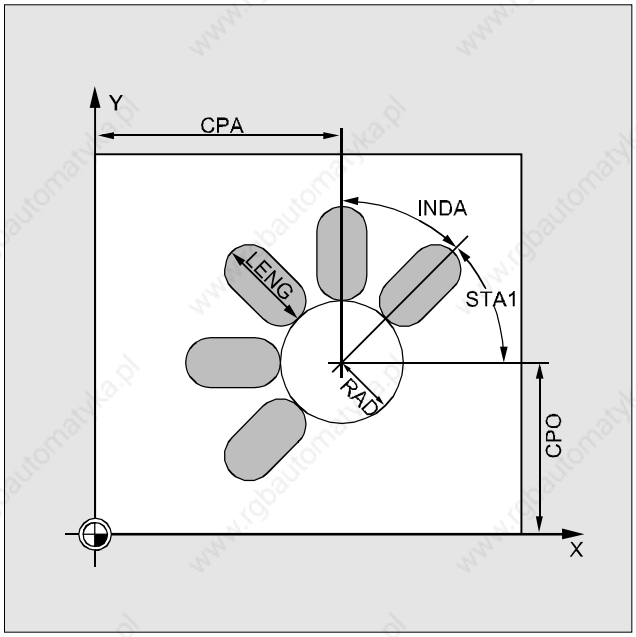
With this cycle, elongated holes can be produced arranged on a circle.
The width of the elongated hole is determined by the tool diameter.

Programming

LONGHOLE (RTP, RFP, SDIS, DP, DPR, NUM, LENG, CPA, CPO, RAD, STA1, INDA, FFD, FFP1, MID)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
NUM	Number of holes/elongated holes/slots
LENG	Length of elongated hole/slot length/pocket length (without sign)
CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
RAD	Radius of circle (enter without sign)
STA1	Starting angle Values: -180 to 180 degrees
INDA	Indexing angle

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Elongated holes on a circle" milling pattern

© Siemens AG 2000. All rights reserved.

Slots on a circle, SLOT1

Sequence

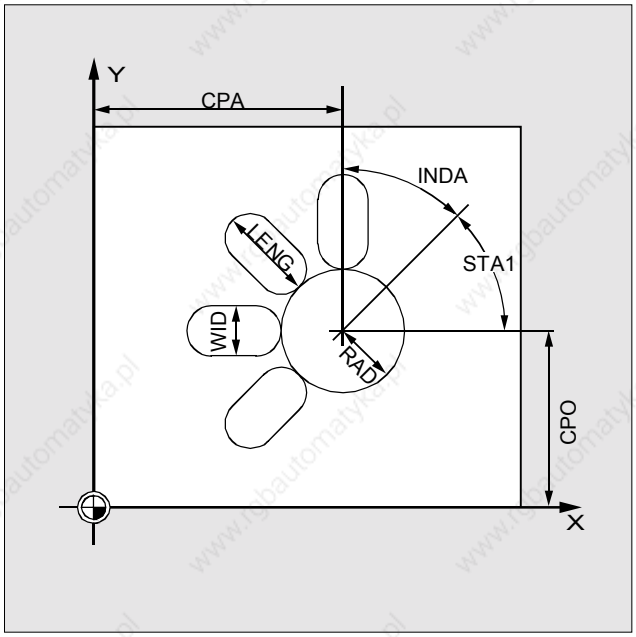
With this cycle, slots can be produced arranged on a circle. The longitudinal axis of the slots has a radial orientation In contrast to the elongated hole, a value is specified for the slot width.
The SLOT1 cycle is a combined roughing/finishing cycle.

Programming

SLOT1 (RTP, RFP, SDIS, DP, DPR, NUM, LENG, WID, CPA, CPO, RAD, STA1, INDA, FFD, FFP1, MID, CDIR, FAL, VARI, MIDF, FFP2, SSF)

LENG	Length of elongated hole/slot length/pocket length (without sign)
WID	Slot width/pocket width (enter without sign)
CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
RAD	Radius of circle (enter without sign)
STA1	Starting angle, values: -180 to 180 degrees
INDA	Indexing angle

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Slots on a circle" milling pattern

© Siemens AG 2000. All rights reserved.

Circumferential slot, SLOT2

Sequence

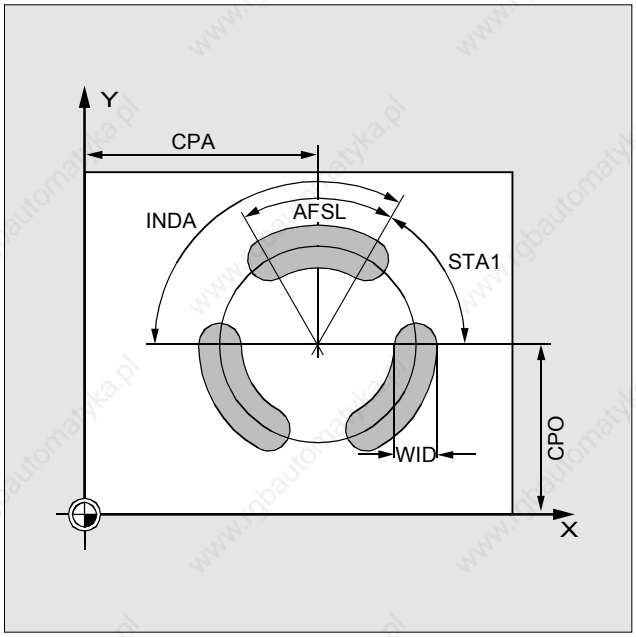
With this cycle, circular slots can be produced arranged on a circle.
The SLOT2 cycle is a combined roughing/finishing cycle.

Programming

SLOT2 (RTP, RFP, SDIS, DP, DPR, NUM, AFSL, WID, CPA, CPO, RAD, STA1, INDA, FFD, FFP1, MID, CDIR, FAL, VARI, MIDF, FFP2, SSF)

LENG	Length of elongated hole/slot length/pocket length (without sign)
WID	Slot width/pocket width (enter without sign)
CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
RAD	Radius of circle (enter without sign)
STA1	Starting angle, Values: -180 to 180 degrees
INDA	Indexing angle
AFSL	Angle for slot length (enter without sign)

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Circumferential slot" milling pattern

© Siemens AG 2000. All rights reserved.

Rectangular pocket milling, POCKET1

Sequence

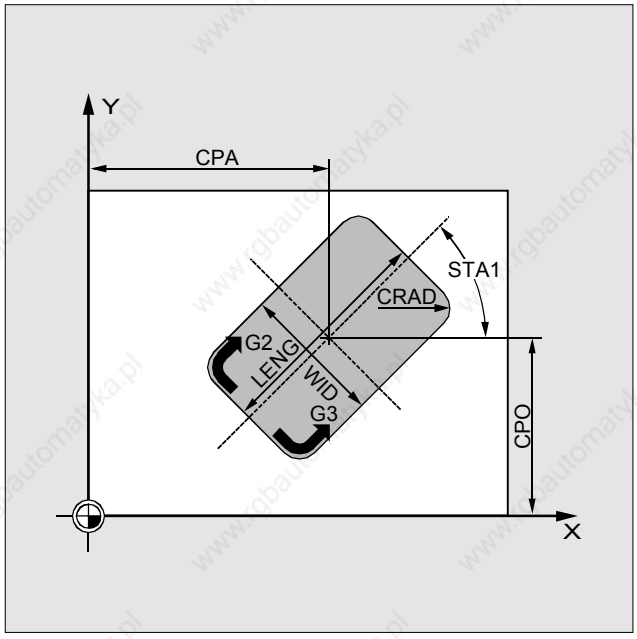
With this cycle, rectangular pockets can be produced in any location in the machining plane.

The cycle is a combined roughing/finishing cycle.

Programming

POCKET1 (RTP, RFP, SDIS, DP, DPR, LENG, WID, CRAD, CPA, CPO, STA1, FFD, FFP1, MID, CDIR, FAL, VARI, MIDF, FFP2, SSF)

LENG	Length of elongated hole/slot length/pocket length (enter without sign)
WID	Slot width/pocket width (enter without sign)
CRAD	Corner radius (enter without sign)
CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
STA1	Starting angle Values: -180 to 180 degrees Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Rectangular pocket" cycle

© Siemens AG 2000. All rights reserved.

Circular pocket milling, POCKET2

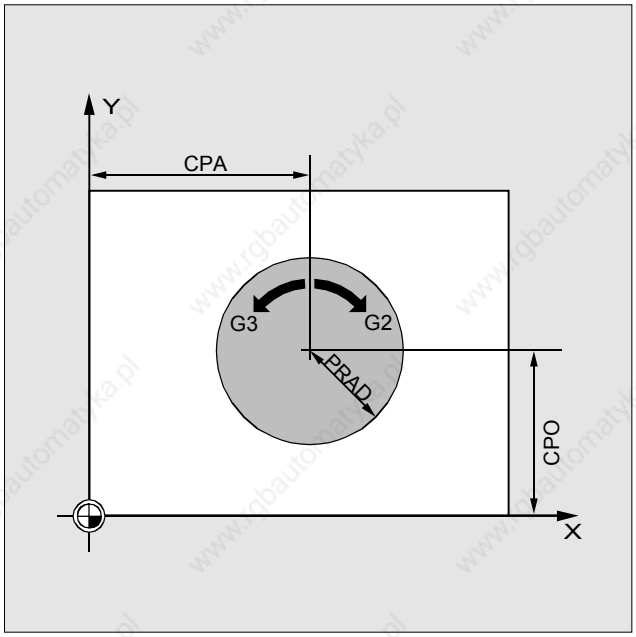
Sequence

With this cycle, circular pockets can be produced.
The cycle is a combined roughing/finishing cycle.

Programming

POCKET2 (RTP, RFP, SDIS, DP, DPR, PRAD, CPA, CPO, FFD, FFP1, MID, CDIR, FAL, VARI, MIDF, FFP2, SSF)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
PRAD	Pocket radius (enter without sign)
CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)
FFD	Feedrate for depth infeed
FFP1	Feedrate for surface machining
MID	Maximum insertion depth (without sign)
CDIR	Direction of machining, 2 (for G2), 3 (for G3)
	Other parameters: see Chapter "Explanation of parameters"



Parameters in the "circular pocket" cycle

© Siemens AG 2000. All rights reserved.

Thread cutting, CYCLE90

Sequence

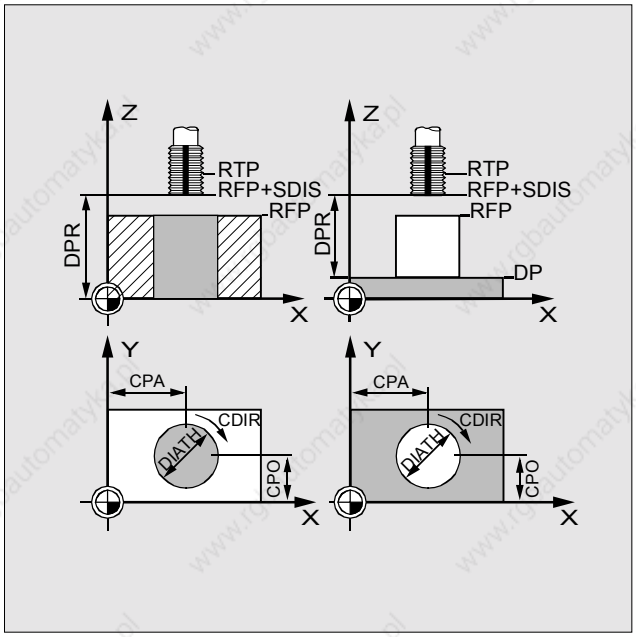
With this cycle, internal and external threads can be produced. The path in thread milling is based on helical interpolation.
All three geometry axes of the present plane are involved in this motion.

Programming

CYCLE90 (RTP, RFP, SDIS, DP, DPR, DIATH, KDIAM, PIT, FFR, CDIR, TYPTH, CPA, CPO)

RTP	Return plane (absolute)
RFP	Reference plane (absolute)
SDIS	Safety distance (enter without sign)
DP	Final drilling depth/elongated hole depth/slot depth/pocket depth (absolute)
DPR	Final drilling depth/elongated hole depth/slot depth/pocket depth relative to reference plane (enter without sign)
DIATH	Nominal thread diameter, outside diameter of thread
CDIR	Direction of machining: 2 (for G2), 3 (for G3)
CPA	Center point circle/pocket, abscissa (absolute)
CPO	Center point circle/pocket, ordinate (absolute)

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Thread milling" cycle

© Siemens AG 2000. All rights reserved.

Recessing cycle, CYCLE93

Sequence

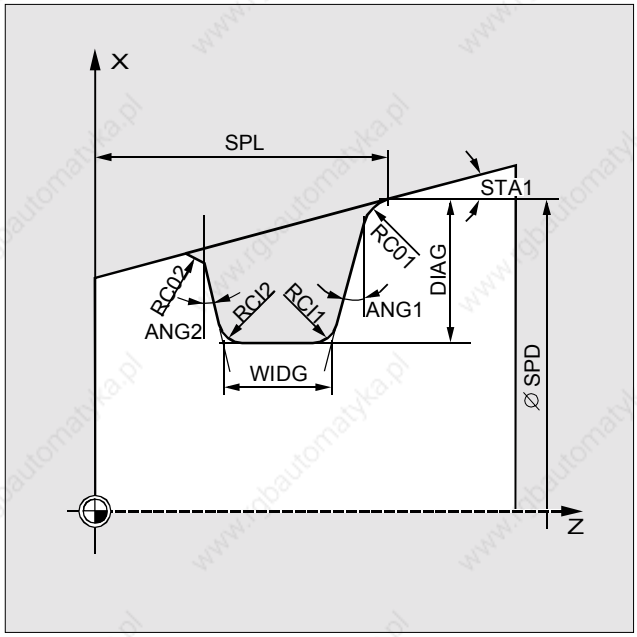
The recessing cycle allows symmetrical and asymmetrical recesses to be produced for longitudinal and facing operations on any straight contour elements. Outside and inside recesses can be made.

Programming

CYCLE93 (SPD, SPL, WIDG, DIAG, STA1, ANG1, ANG2, RCO1, RCO2, RCI1, RCI2, FAL1, FAL2, IDEP, DTB, VARI)

SPD	Start point in the facing axis (enter without sign)
SPL	Start point of contour/of thread in the longitudinal axis
WIDG	Slot width (enter without sign)
DIAG	Slot depth (enter without sign)
ANG1	Flank angle 1: at side of insertion defined by the start point (enter without sign)
ANG2	Flank angle 2: at other side (without sign)
RCO1	Radius/chamfer 1, outside: at side defined by start point
RCO2	Radius/chamfer 2, outside
RCI1	Radius/chamfer 1, inside: at start point side
RCI2	Radius/chamfer 2, inside

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Recessing" cycle for longitudinal machining

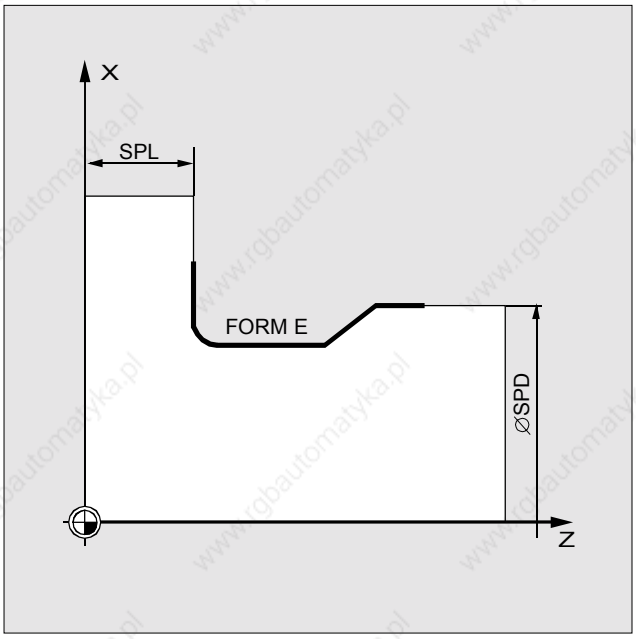
© Siemens AG 2000. All rights reserved.

Undercut cycle, CYCLE94

Sequence With this cycle, undercuts to DIN 509 of forms E and F can be produced with usual loading for a finished part diameter > 3 mm.

Programming CYCLE94 (SPD, SPL, FORM)

SPD	Start point in the facing axis (enter without sign)	
SPL	Start point of contour/of thread in the longitudinal axis	
FORM	Definition of form	E (for Form E)
		F (for Form F)



Parameters in the "Undercut" cycle

© Siemens AG 2000. All rights reserved.

Stock removal cycle, CYCLE95

Sequence

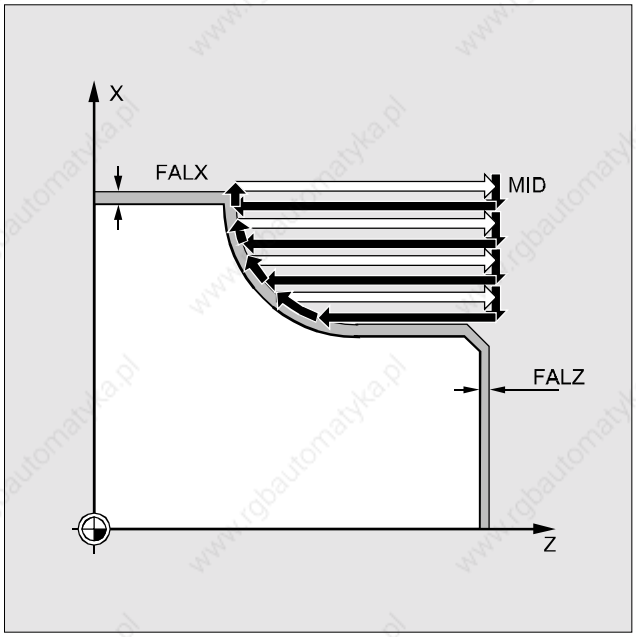
With the "Stock removal" cycle, a contour programmed in a subroutine can be produced from a blank by means of paraxial stock removal.
The contour can include relief-cut elements.

The technology (roughing/finishing/complete machining) is selectable. The cycle can be called from any collision-free position.

Programming

CYCLE95 (NPP, MID, FALZ, FALX, FAL, FF1, FF2, FF3, VARI, DT, DAM)

NPP	Name of the contour program
MID	Maximum insertion depth (enter without sign)
FALZ	Finishing allowance in the longitudinal axis (without sign)
FALX	Finishing allowance in the facing axis (without sign)
FAL	Correct finishing allowance for contour (without sign)
	Other parameters: see Chapter "Explanation of parameters"



Sequence of motions and parameters in the "Stock removal" cycle

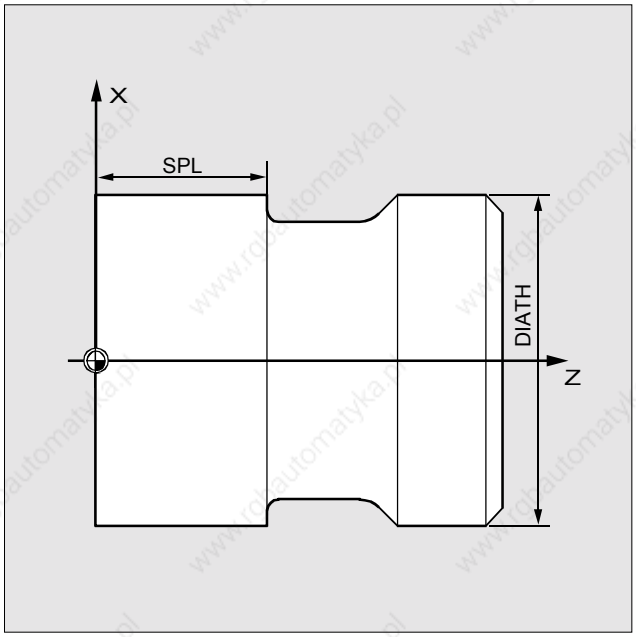
© Siemens AG 2000. All rights reserved.

Thread undercut, CYCLE96

Sequence With this cycle, form A, B, C, D thread undercuts can be produced in accordance with DIN 13 for parts with metric ISO thread.

Programming CYCLE96 (DIATH, SPL, FORM)

DIATH	Rated diameter, outside diameter of thread	
SPL	Start point of contour/of thread in the longitudinal axis	
FORM	Definition of form	E (for Form E) F (for Form F)



Parameters in the "Thread undercut" cycle

© Siemens AG 2000. All rights reserved.

Thread cutting, CYCLE97

Sequence

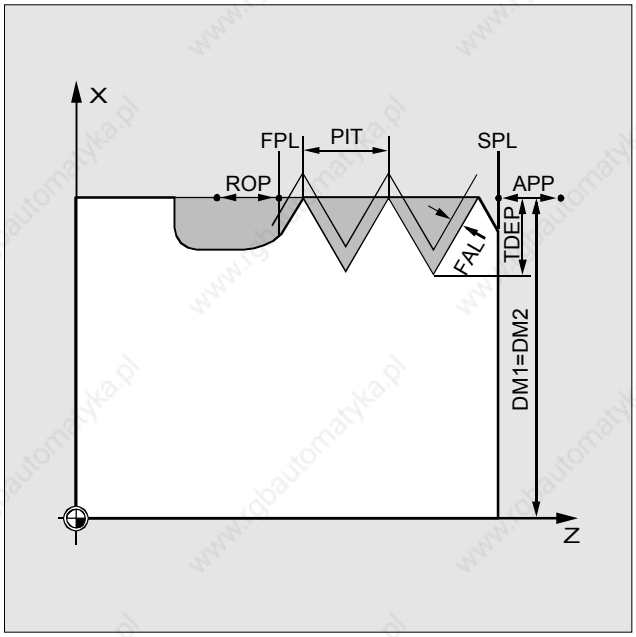
With the "Thread cutting" cycle, cylindrical and tapered outside and inside threads can be produced in longitudinal and facing operations, both of the single thread and multiple thread type.
A prerequisite for using this cycle is a speed-controlled spindle with position measuring system. In the case of multiple threads, the individual threads are machined one after the other.

Programming

CYCLE97 (PIT, MPIT, SPL, FPL, DM1, DM2, APP, ROP, TDEP, FAL, IANG, NSP, NRC, NID, VARI, NUMTH)

SPL	Start point of contour/thread in the longitudinal axis
FPL	End point of thread in the longitudinal axis
PIT	Pitch; Values: 0.001 ... 2000.000 mm
ROP	Exit path (enter without sign)
FAL	Correct finishing allowance for contour (enter without sign)
TDEP	Thread depth (enter without sign)
APP	Run-in path (enter without sign)
DM1	Diameter of thread at start point
DM2	Diameter of thread at end point

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Thread cutting" cycle

© Siemens AG 2000. All rights reserved.

Chaining of threads, CYCLE98

Sequence

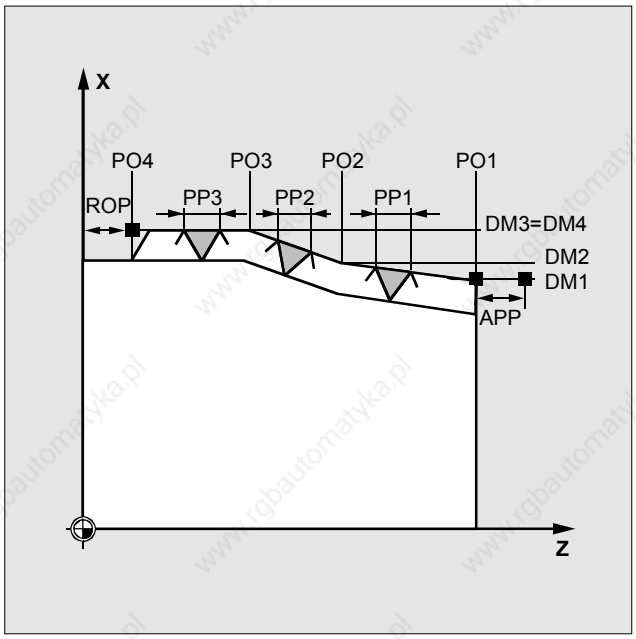
This cycle allows production in longitudinal and facing operations of several cylindrical or taper threads located one behind the other and possibly having different leads.

Programming

CYCLE98 (PO1, DM1, PO2, DM2, PO3, DM3, PO4, DM4, APP, ROP, TDEP, FAL, IANG, NSP, NRC, NID, PP1, PP2, PP3, VARI, NUMTH)

PO1	Start point of thread in the longitudinal axis
DM1	Diameter of thread at start point
PO2	First intermediate point in the longitudinal axis
DM2	Diameter at first intermediate point
PO3	Second intermediate point
DM3	Diameter at second intermediate point
PO4	End point of thread in the longitudinal axis
DM4	Diameter at end point
APP	Run-in path (enter without sign)
ROP	Exit path (enter without sign)
PP1 to PP3	Pitch 1 to 3 as value (enter without sign)

Other parameters: see Chapter "Explanation of parameters"



Parameters in the "Thread chaining" cycle

© Siemens AG 2000. All rights reserved.

© Siemens AG 2000. All rights reserved.

7. Predefined Switching and Auxiliary Functions

List of M commands 7-88

Forms 7-89

© Siemens AG 2000. All rights reserved.

List of M commands

M0*	Programmed stop
M1*	Optional stop
M2*	End of program (main program)
M30*	End of program as M2
M17*	End of subroutine

M3	Spindle clockwise
M4	Spindle counterclockwise
M5	Spindle stop
M6	Tool change
M70	Change to axis mode

M40	Automatic gear change
M41	Gear stage 1
M42	Gear stage 2
M43	Gear stage 3
M44	Gear stage 4
M45	Gear stage 5

Extended address notation is not permissible for functions marked with an asterisk "".*

Machine manufacturer (MH7.1)

All free M function numbers can be assigned by the machine manufacturer. For example, with switching functions for controlling clamping devices or for activating/deactivating further machine functions.

© Siemens AG 2000. All rights reserved.

Forms

You can enter your own user-specific functions here.

© Siemens AG 2000. All rights reserved.

Forms

© Siemens AG 2000. All rights reserved.

8. Program Key

List of G functions 8-92

© Siemens AG 2000. All rights reserved.

List of G functions

Group 1: Modal motion commands				
Name	**No.**	**Meaning**	**m/n**	**Def.**
G0	1.	Rapid traverse motion	m	
G1	2.	Linear interpolation	m	Def.
G2	3.	Circular interpolation clockwise	m	
G3	4.	Circular interpolation counterclockwise	m	
CIP	5.	Circular interpolation through point	m	
ASPLINE #	6.	Akima spline	m	
BSPLINE #	7.	B spline	m	
CSPLINE #	8.	Cubic spline	m	
POLY ##	9.	Polynomial interpolation	m	
G33	10.	Thread cutting with constant lead	m	
G331	11.	Rigid tapping	m	
G332	12.	Return (rigid tapping)	m	
G58		Axial-programmable absolute zero offset	m	
G59		Axial-programmable additive zero offset	m	
OEMIPO1 ###	13.	OEM interpolation 1 *)	m	
OEMIPO2 ###	14.	OEM interpolation 2 *)	m	

*) The OEM user can include two additional types of interpolation. The OEM user can alter the names.
If no function has been programmed out of this group for modal G functions, the default setting (which can be altered by machine data) applies: $MC_GCODE_RESET_VALUES
Vocabulary word does **not** apply for SINUMERIK 810D; ## does **not** apply for SINUMERIK FM-NC/810D;
does **not apply for SINUMERIK 810D/FM-NC/NCU571**

Group 2: Non-modal motion commands, dwell time				
G4	1.	Dwell time preset	n	
G63	2.	Tapping without synchronization	n	
G74	3.	Reference point approach with synchronization	n	
G75	4.	Fixed point approach	n	
REPOSL	5.	Repositioning on contour, linear	n	
REPOSQ	6.	Repositioning on contour in quarter circle	n	
REPOSH	7.	Repositioning on contour in semi-circle	n	
REPOSA	8.	Repositioning on contour with all axes	n	
REPOSQA	9.	Repositioning on contour with all axes, geometry axis in quarter circle	n	
REPOSHA	10.	Repositioning on contour with all axes, geometry axes in semi-circle	n	

m: modal, n: non-modal, Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 3: Write memory				
Name	**No.**	**Meaning**	**m/n**	**Def.**
TRANS	1.	TRANSLATION: translation, programmable	n	
ROT	2.	ROTATION: rotation, programmable	n	
SCALE	3.	SCALE: scaling, programming	n	
MIRROR	4.	MIRROR: mirroring, programmable	n	
ATRANS	5.	Additive translation, programmable	n	
AROT	6.	Additive rotation, programmable	n	
ASCALE	7.	Additive scaling, programmable	n	
AMIRROR	8.	Additive mirroring, programmable	n	
TOFRAME	9.	Place currently programmed frame on tool coordinate system	n	
G25	10.	Minimum working area limitation/spindle speed limitation	n	
G26	11.	Maximum working area limitation/spindle speed limitation	n	
G110	12.	Pole programming relative to the last programmed setpoint position	n	
G111	13.	Pole programming relative to the zero of the present WCS	n	
G112	14.	Pole programming relative to the last valid pole	n	

Group 4: FIFO				
STARTFIFO	1.	Execute and fill preprocessing buffer in parallel	m	Def.
STOPFIFO	2.	Stop processing: fill preprocessing memory	m	

Group 6: Plane selection				
G17	1.	Plane selection 1st - 2nd geometry axis	m	Def.
G18	2.	Plane selection 3rd - 1st geometry axis	m	
G19	3.	Plane selection 2nd - 3rd geometry axis	m	

m: modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 7: Tool radius compensation				
Name	**No.**	**Meaning**	**m/n**	**Def.**
G40	1.	No tool radius compensation	m	
G41	2.	Tool radius compensation left of contour	m	
G42	3.	Tool radius compensation right of contour	m	

Group 8: Settable zero offset				
G500	1.	Deselection G54 - G5xx, reset settable frame	m	Def.
G54	2.	1st settable zero offset	m	
G55	3.	2nd settable zero offset	m	
G56	4.	3rd settable zero offset	m	
G57	5.	4th settable zero offset	m	
G5xx	nth	nth settable zero offset	m	
G599	100.	100th settable zero offset	m	

Group 9: Frame suppression				
G53	1.	Suppression of current frame	n	
SUPA	2.	Suppression of current zero offset, including programmed and handwheel offsets (DRF), external zero offset and PRESET offset		

Group 10: Exact stop, continuous path mode				
G60	1.	Velocity reduction, exact positioning	m	Def.
G64	2.	Continuous path mode	m	
G641	3.	Continuous path mode with programmable rounding distance	m	

Group 11: Exact stop blockwise				
G9	1.	Velocity reduction, exact stop	n	

m: modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 12: Block change criteria at exact stop (G60/G09)				
Name	**No.**	**Meaning**	**m/n**	**Def.**
G601	1.	Block change at exact stop fine	m	**Def.**
G602	2.	Block change at exact stop coarse	m	
G603	3.	Block change at end of interpolation block	m	

Group 13: Workpiece dimensioning inch/metric				
G70	1.	Input system inch	m	
G71	2.	Input system metric	m	Def.

Group 14: Workpiece dimensioning absolute/incremental				
G90	1.	Absolute dimension input	m	Def.
G91	2.	Incremental dimension input	m	

Group 15: Feedrate type				
G93 #	1.	Inverse time feedrate coding	m	
G94	2.	Linear feedrate in mm/min, inch/min	m	Def.
G95	3.	Revolutional feedrate in mm/rev, inch/rev	m	
G96	4.	Constant cutting velocity ON		
G97	5.	Constant cutting velocity OFF		

The vocabulary word does **not** apply for SINUMERIK FM-NC/810D.

Group 16: Feed correction at inside and outside curvature				
CFC	1.	Constant feed at contour	m	Def.
CFTCP	2.	Constant feed in tool center point	m	
CFIN	3.	Constant feed at inside curvature	m	

m: modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 17: Approach/retraction behavior, tool compensation				
Name	**No.**	**Meaning**	**m/n**	**Def.**
NORM	1.	Normal position at start/end point	m	Def.
KONT	2.	Travel around contour at start/end point	m	

Group 18: Corner behavior, tool compensation				
G450	1.	Transition circle	m	Def.
G451	2.	Intersection of equidistances	m	

Group 19: Curve transition at beginning of spline				
BNAT #	1.	Natural curve transition at the first spline block	m	Def.
BTAN #	2.	Tangential curve transition to the first spline block	m	
BAUTO #	3.	Definition of the 1st spline section through the following 3 points	m	

Group 20: Curve transition at end of spline				
ENAT #	1.	Natural curve transition to the next traversing block	m	Def.
ETAN #	2.	Tangential curve transition at the beginning of spline	m	
EAUTO #	3.	Definition of the last spline section through the last 3 points	m	

Group 21: Acceleration profile				
BRISK	1.	Brisk non-smoothed path acceleration	m	Def.
SOFT	2.	Soft smoothed path acceleration	m	
DRIVE ##	3.	Velocity-related path acceleration	m	

m: modal
n: Non-modal
Def.: Default
The vocabulary word does **not** apply for SINUMERIK 810D.
The vocabulary word applies **only** for SINUMERIK FM-NC.

© Siemens AG 2000. All rights reserved.

List of G functions

Group 22: Tool compensation type				
Name	**No.**	**Meaning**	**m/n**	**Def.**
CUT2D	1.	2 1/2D tool compensation	m	Def.
CUT2DF	2.	2 1/2D tool compensation relative to current frame (inclined plane)	m	
CUT3DC #	3.	3D tool compensation peripheral milling	m	
CUT3DF #	4.	3D tool compensation peripheral milling	m	
CUT3DFS #	5.	3D tool compensation face milling with constant tool orientation, independent of active frame	m	
CUT3DFF #	6.	3D tool compensation face milling with constant tool orientation, independent of active frame	m	

The vocabulary word does **not** apply for SINUMERIK FM-NC/810D.

Group 23: Collision detection at inside contours				
CDOF	1.	Collision detection off	m	Def.
CDON	2.	Collision detection on	m	

Group 24: Feedrate control				
FFWOF	1.	Feedforward control off	m	Def.
FFWON	2.	Feedforward control on	m	

Group 25: Tool orientation				
ORIWKS #	1.	Tool orientation in workpiece coordinate system	m	Def.
ORIMKS #	2.	Tool orientation in machine coordinate system	m	
ORIPATH #	3.	Tool orientation path	m	

The vocabulary word does **not** apply for SINUMERIK FM-NC/810D/NCU571.

Group 26: Repositioning point				
RMB	1.	Return to start of block	m	
RMI	2.	Return to interruption point	m	Def.
RME	3.	Return to end of block	m	

m: modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 27: Tool compensation on change of orientation at outside corners				
Name	**No.**	**Meaning**	**m/n**	**Def.**
ORIC #	1.	Orientation changes are superimposed on the circle block to be inserted	m	Def.
ORID #	2.	Orientation changes are performed before the circle block	m	

Group 28: Working area limitation on/off				
WALIMON	1.	Working area limitation on	m	Def.
WALIMOF	2.	Working area limitation off	m	

Group 29: Radius – diameter				
DIAMOF	1.	Diameter programming off	m	Def.
DIAMON	2.	Diameter programming on	m	

Group 30: Compressor on/off				
COMPOF #	1.	Compressor off	m	Def.
COMPON #	2.	Compressor on	m	

Group 31: OEM - G group				
G810 #	1.	OEM G function		Def.
G811 #	2.	OEM G function		
G812 #	3.	OEM G function		
G813 #	4.	OEM G function		
G814 #	5.	OEM G function		
G815 #	6.	OEM G function		
G816 #	7.	OEM G function		
G817 #	8.	OEM G function		
G818 #	9.	OEM G function		
G819 #	10.	OEM G function		

Two groups are reserved for the OEM user who can use them to provide the end user with OEM functions in the final program. No.: Internal number for (e.g.) PLC interface. Def.: Default

The vocabulary word does **not** apply for SINUMERIK FM-NC/810D/NCU571.

m: Modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 32: OEM - G group				
G820 #	1.	OEM G function	**m/s**	**Def.**
G821 #	2.	OEM G function		Def.
G822 #	3.	OEM G function		
G823 #	4.	OEM G function		
G824 #	5.	OEM G function		
G825 #	6.	OEM G function		
G826 #	7.	OEM G function		
G827 #	8.	OEM G function		
G828 #	9.	OEM G function		
G829 #	10.	OEM G function		

Two groups are reserved for the OEM user who can use them to provide the end user with OEM functions in the final program.

No.: Internal number for (e.g.) PLC interface

m: Modal
n: Non-modal
Def.: Default

Group 33: Settable tool fine coding				
FTOCOF #	1.	Online active tool fine coding off	m	Def.
FTOCON #	2.	Online active tool fine coding on	m	

Group 34: Smoothing of tool orientation				
OSOF #	1.	Smoothing of tool orientation off	m	Def.
OSC #	2.	Constant smoothing of tool orientation	m	
OSS #	3.	Smoothing of tool orientation at end of block	m	
OSSE#	4.	Smoothing of tool orientation at beginning and end of block	m	

The vocabulary word does **not** apply for SINUMERIK FM-NC/810D/NCU571.

m: Modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

List of G functions

Group 35: Punching and nibbling				
Name	**No.**	**Meaning**	**m**	**Def.**
SPOF #	1.	Punching, nibbling off	m	Def.
SON #	2.	Nibbling on	m	
PON #	3.	Punching on	m	
SONS #	4.	Nibbling on in IPO cycle	m	
PONS #	5.	Punching on in IPO cycle	m	

Group 36: Punching with delay				
PDELAYON #	1.	Punching with delay on	m	Def.
PDELAYOF #	2.	Punching with delay off	m	

Group 37: Feedrate profile				
FNORM #	1.	Feed normal in accordance with DIN66025	m	Def.
FLIN #	2.	Feed linear variable	m	
FCUB #	3.	Feed variable according to cubic spline		

Group 38: Assignment of high-speed NCK inputs/outputs for punching/nibbling				
SPIF1 #	1.	High-speed NCK inputs/outputs for punching/nibbling byte 1	m	Def.
SPIF2 #	2.	High-speed NCK inputs/outputs for punching/nibbling byte 2	m	

Group 39: Programmable contour accuracy				
CPRECOF	1.	Programmable contour accuracy off	m	Def.
CPRECON	2.	Programmable contour accuracy on	m	

The vocabulary word does **not** apply for SINUMERIK FM-NC/810D/NCU571.

No.: Internal number for (e.g.) PLC interface

m: Modal
n: Non-modal
Def.: Default

© Siemens AG 2000. All rights reserved.

To: **SIEMENS AG** **A&D MC BMS** **P. O. Box 3180** **D-91050 Erlangen** **Federal Republic of Germany** Fax ++ 49 09131/98-1145)	**Suggestions** **Corrections** for Publication/Manual: SINUMERIK 840D/840Di SINUMERIK 810D/FM-NC User Documentation
From Name	Short Guide Programming Order No.: 6FC5298-6AB30-0BP0 Edition: 10.00
Company/department Address _____ _____ Telephone / _____ Telefax: /	Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections