



## SINUMERIK Operate programGUIDE reference

### Starting a new program

From the Program manager → select NC → arrow down to the directory you would like to create your program in (use right arrow to access sub folders) → select the New key → pick programGUIDE → type in the desired file name (no spaces) → press OK.



### Basic program structure

G-code programs can be freely programmed. The most important commands tend to be at the beginning of the program or an operation and will set you up to be able to machine any desired objects.

- Set a machine plane — G17, G18, G19
- Define metric or inch programming — G71 or G70
- Call a work offset — G54–G59 and G505–G599 (depending upon machine options)
- Call a Tool — T# and D# — M06 may be required depending upon the OEM
- Feed rate mode and speed — G94 fpm, G95 fpr, and F#
- Spindle commands — S#, M03 for CW, M04 for CCW, M05 for off
- Coolant — M08 on, M09 off (M-codes will vary depending upon the OEM)
- Desired machining operation

## Milling example

Figure 1

### Beginning of the program

N10 G17 G54 G70 G90 ; plane, work offset, inch, abs  
 N20 WORKPIECE(,,,"RECTANGLE",64,0,-1,-80,5.5,5.5) ;defines the stock for graphics  
 N30 T="endmill" ; calls tool based upon the name in the offset table  
 N40 D1 ; calls up cutting edge 1 for tool — can create up to nine offsets  
 N50 M6 ; tool change macro varies based upon the OEM  
 N60 M3 S1000  
 N70 G0 X-3 Y-3 Z.1 M8 ; rapid position move and coolant command  
 N80 G94 F20 ; feed per minute

### N230 body of the program

### End of the program

N300 M5 M9  
 N310 SUPA G0 D0 Z0 ; cancels all offsets and positions **based upon the machine coordinate system**  
 N320 M30

## Turning example

Figure 2

### Beginning of the program

N10 WORKPIECE(,,,"CYLINDER",192,0,-4,-3,2) ; workpiece defines the shape for graphics  
 N20 G18 G54 G70 G90  
 N30 DIAMON ; DIAMON= x diametric programming, DIAMOFF = x radial programming  
 N30 T="CNMG432\_OD" D1 ; calls up tool based upon name in offset table and cutting edge 1  
 N50 M3S1000  
 N60 G0X3Z.1  
 N70 G95 F.010 ; feed per revolution  
 N80 **Operation**

### Tool Change

N100 T3 D1 ;will call up turret position not tool name & cutting edge 1  
 N110 G96 S350 LIMS=3000 M3 ;css @ 350 with a max rpm of 3000  
 N120 G0X3Z.1  
 N130 G95 F.005 ; feed per revolution  
 N140 M08  
 N150 **Operation**

### End of the program

N180 M5 M9  
 N190 SUPA G0 D0 X0 Z0 ; cancels all offsets and positions **based upon the machine coordinate system**  
 N200 M30

## G-Code chart

Function	Command	Format
Rapid	G00	G00 X.. Y.. Z..
Linear Interp	G01	G01 X.. Y.. Z.. F..
Circle CW	G02	G02 X.. Y.. I.. J..
Circle CCW	G03	G03 X.. Y.. Z..
Dwell	G04	G04 F2.5
Exact Stop Non-Modal	G09	G09
XY Plane	G17	G17
ZX Plane	G18	G18
YZ Plane	G19	G19
Inch Mode	G70/G700	G70/G700
MM Mode	G71/G710	G71/G710
Thread Constant Pitch	G33	G33 Z.. K..
Thread Variable Pitch	G34	G34 Z.. K.. F..
CRC off	G40	G40
CRC Left	G41	G41 X.. Y..
CRC Right	G42	G42 X.. Y..
Tool Length Comp On	D1	T.. D1
Tool Length Comp Off	D0	D0
Exact Stop Modal	G60	G60
Corner Deceleration	G62	G62
Tapping Mode	G63	G63 X.. Y.. Z..
Continuous Path	G64	G64
Inverse Time Feed	G93	G93
Feed Per Minute	G94	G94
Feed Per Revolution	G95	G95
CSS	G96	G96
Max RPM for CSS	LIMS=	LIMS=
Constant speed	G97	G97

Function	Command	Format
Finish turn cycle	CYCLE95	See sample
Stock removal cycle/longitudanal	CYCLE95	See sample
Stock removal cycle/transverse	CYCLE95	See sample
Deep hole drill cycle	CYCLE83	See sample
Contour repetition	CYCLE95	See sample
Tap cycle left hand	CYCLE84	See sample
Deep hole and groove cycle	CYCLE83	See sample
Deep hole and groove cycle	CYCLE83	See sample
Multiple thread cycle	CYCLE93	See sample
Fine drilling cycle	CYCLE86	See sample
End cycle	MCALL	See sample
Counterboring cycle	CYCLE81	See sample
Countersinking cycle	CYCLE82	See sample
Deep hole drill cycle	CYCLE83	See sample
Tap cycle right hand	CYCLE84	See sample
Boring cycle	CYCLE85	See sample
Boring cycle	CYCLE86	See sample
Boring cycle	CYCLE87	See sample
Boring cycle	CYCLE88	See sample
Boring cycle	CYCLE89	See sample

Basic format for all cycles  
Sample #2 see note below  
M3 S2000 G90 G0 F120  
Mcall CYCLE81 (1,0,.1,-1.5)  
Y2  
Y3  
X2  
Y-1  
Mcall

**Published by  
Siemens Industry, Inc.**

390 Kent Avenue  
Elk Grove Village, IL 60007

1-800-879-8079

Order No. MBTD-PR007-0216

Printed in USA

© 2016 Siemens Industry, Inc.

[usa.siemens.com/cnc](http://usa.siemens.com/cnc)

This brochure contains only general descriptions or performance features, which do not always apply in the manner described in concrete application situations or may change as the products undergo further development. Performance features are valid only if they are formally agreed upon when the contract is closed.

Siemens is a registered trademark of Siemens AG. Product names mentioned may be trademarks or registered trademarks of their respective companies. Specifications are subject to change without notice.