

SIEMENS

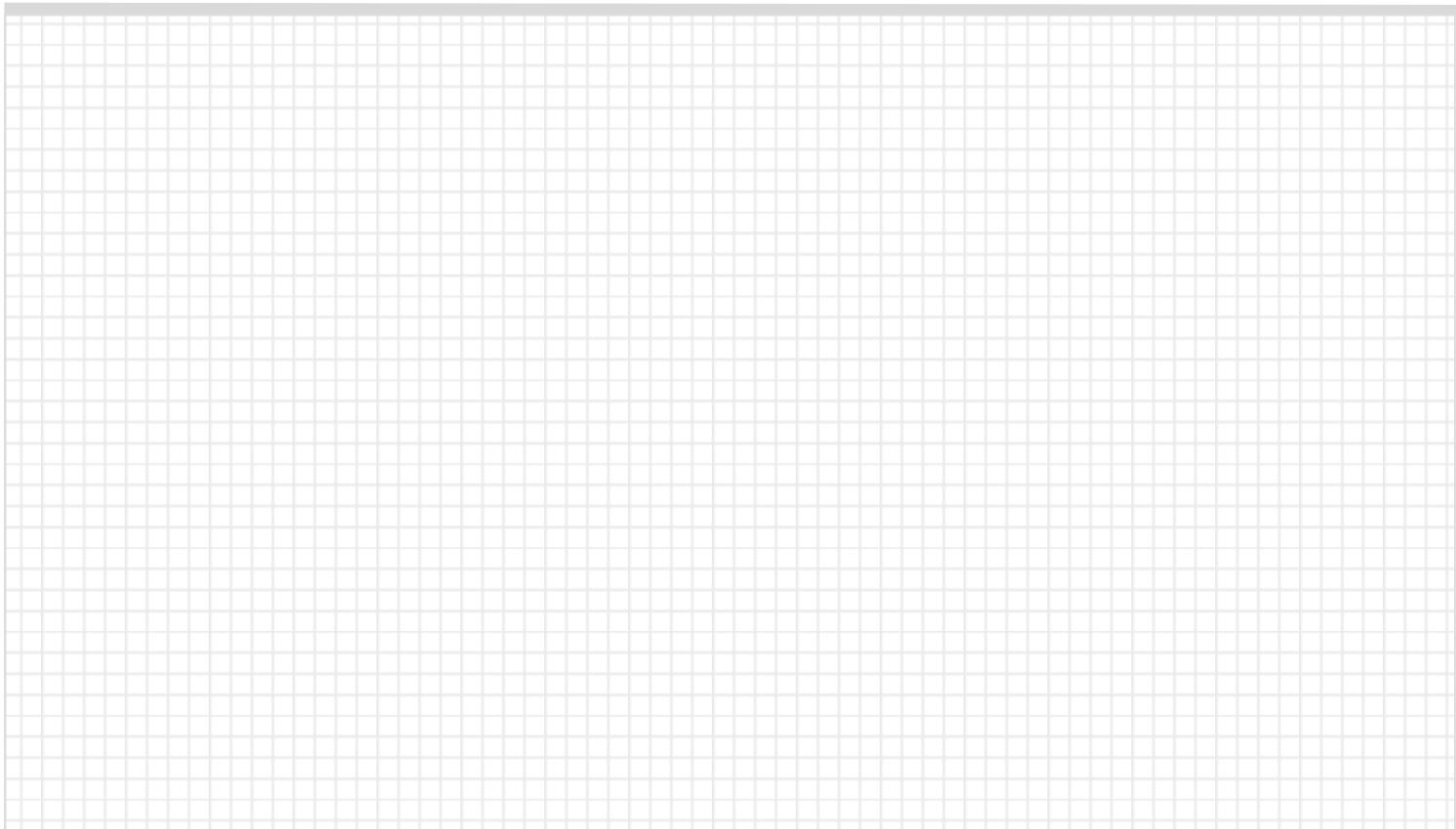


Training manual

Sinumerik 808D ADVANCED Programming and Operating Procedures for Turning

Version 2013-09

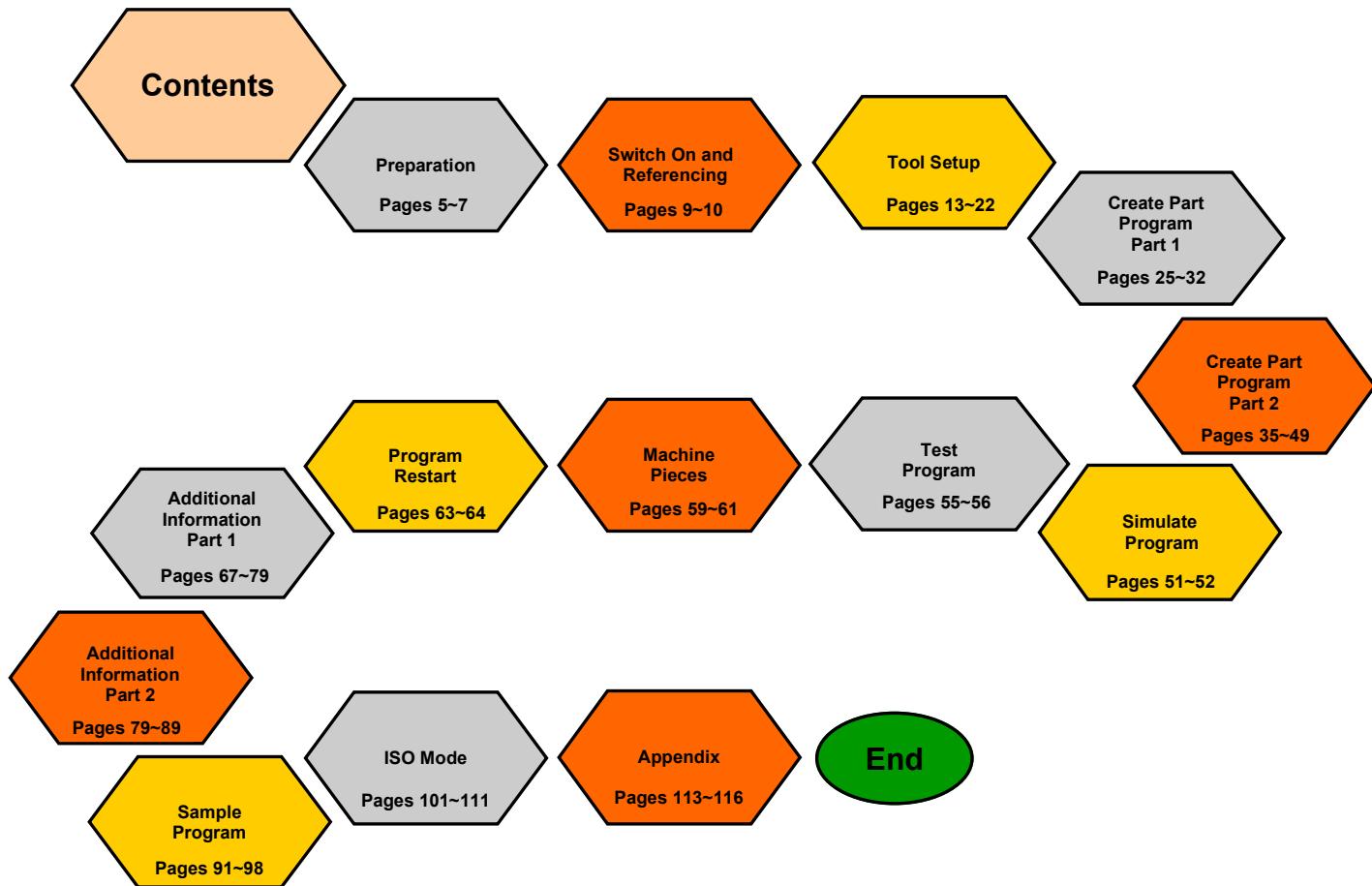
Notes





Basic knowledge of programming for turning is required,
before operating of a machine !

SIEMENS

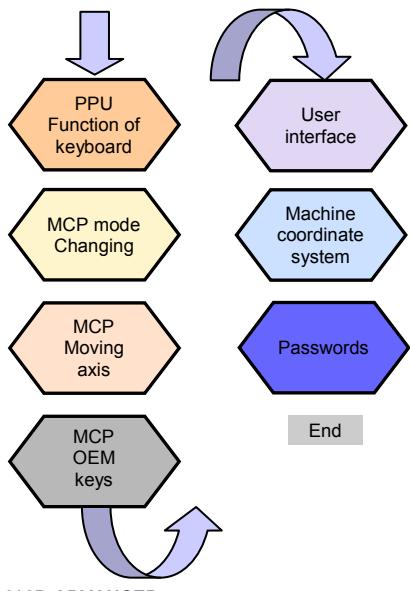


Notes

Absolute value and incremental value	28	Manual tool change	16
Editing part program	27	MDA	84
Executing function M	21	Moving axis with handwheel	17
Calculator	90	Part programming	25
Time change	78	Protection levels	7
Creating and measuring tools	13	Program execution	55
Creating zero offsets	82	Breakpoint search	63
Cycles	35	Reference point	10
Dry run	56	RS232c, USB, and network	67
Jogging spindle	21	Saving data	78
Help	76	Simulation	51
List of programming functions	115	Subprograms	85
Tool wear	61	Sample programs	93
Manual start spindle	81	Timers/counters	59
		ISO mode	103

Unit Description

This unit describes the 808D ADVANCED PPU and MCP functionality, the coordinate system of a turning machine and how to enter passwords to access the system.

Unit Content

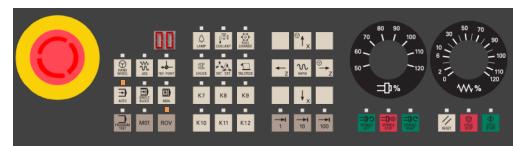
PPU Function of keyboard



Menu navigation

Operating area navigation

MCP mode Changing



The 808D machine control panel (MCP) is used to select the machine operating mode :
JOG - MDA - AUTO

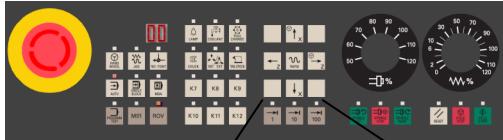
Mode Navigation



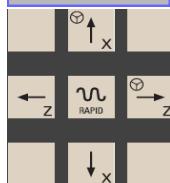
The 808D ADVANCED panel processing unit (PPU) is used to input data to the CNC and to navigate to operating areas of the system.

Preparation

MCP Moving axis



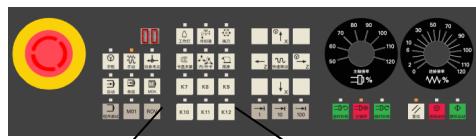
Axis movement



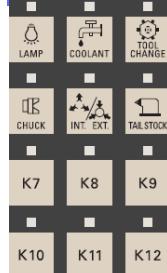
The 808D machine control panel (MCP) is used to control manual operation of the axis.

The machine can be moved with the appropriate keys.

MCP OEM keys



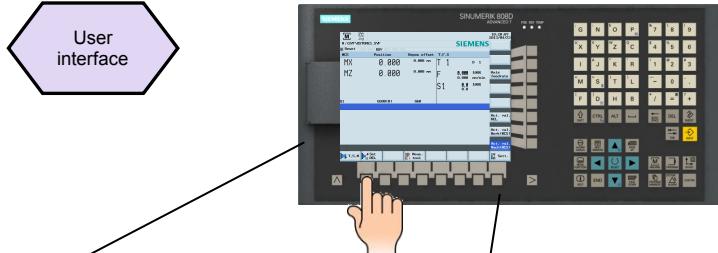
OEM key



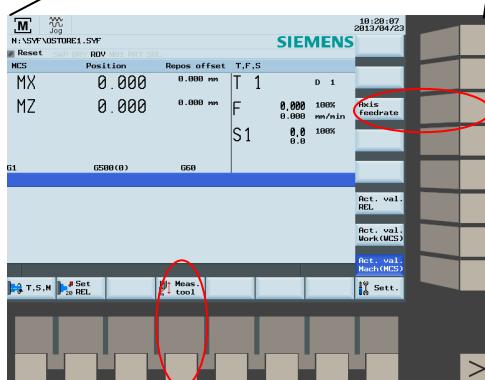
The 808D machine control panel (MCP) is used to control OEM machine functions.

The machine functions can be activated with the appropriate keys.

User interface



808D ADVANCED (PPU) has eight vertical softkeys (abbr. SK) on the right of the screen. These SK's can be activated with the corresponding button (located on the right).

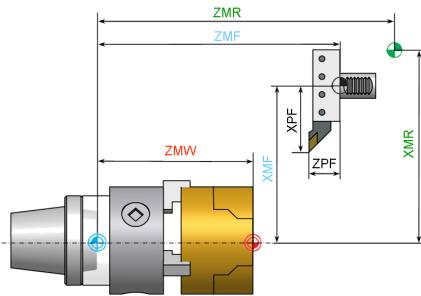


808D ADVANCED (PPU) has eight horizontal SK's on the bottom of the screen. These SK's can be activated with the corresponding button (located below).

Preparation

SEQUENCE

Machine coordinate system



The Sinumerik 808D ADVANCED uses a coordinate system which is derived from the DIN 66217 standard. The system is an international standard and ensures compatibility between machines and coordinate programming. The primary function of the coordinate system is to ensure that the tool length and tool radius are calculated correctly in the respective axis.



The machine zero point (M) is determined by the machine manufacturer and cannot be changed.



The workpiece zero point (W) is the origin of the workpiece coordinate system.



The reference point (R) is used for synchronizing the measuring system. Synchronizing is used for synchronizing the measuring system



The tool holder reference point (F) is used to determine the tool offset.

Passwords

Passwords at the control are used to set the user's right to access the system. Tasks such as "Basic Operating", "Advanced Operating" and commissioning functions all depend on the passwords.

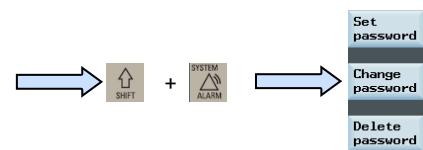
No password	Machine operator
Customer's password	Advanced operator
Manufacturer's password	OEM engineer
Customer's password	= CUSTOMER
Manufacturer's password	= SUNRISE

Changing password

Step 1



Usually the machine operator does not need to change the password.



The service mode is opened with the appropriate key combination.

In the service mode, the password can be activated and deactivated.

Step 2



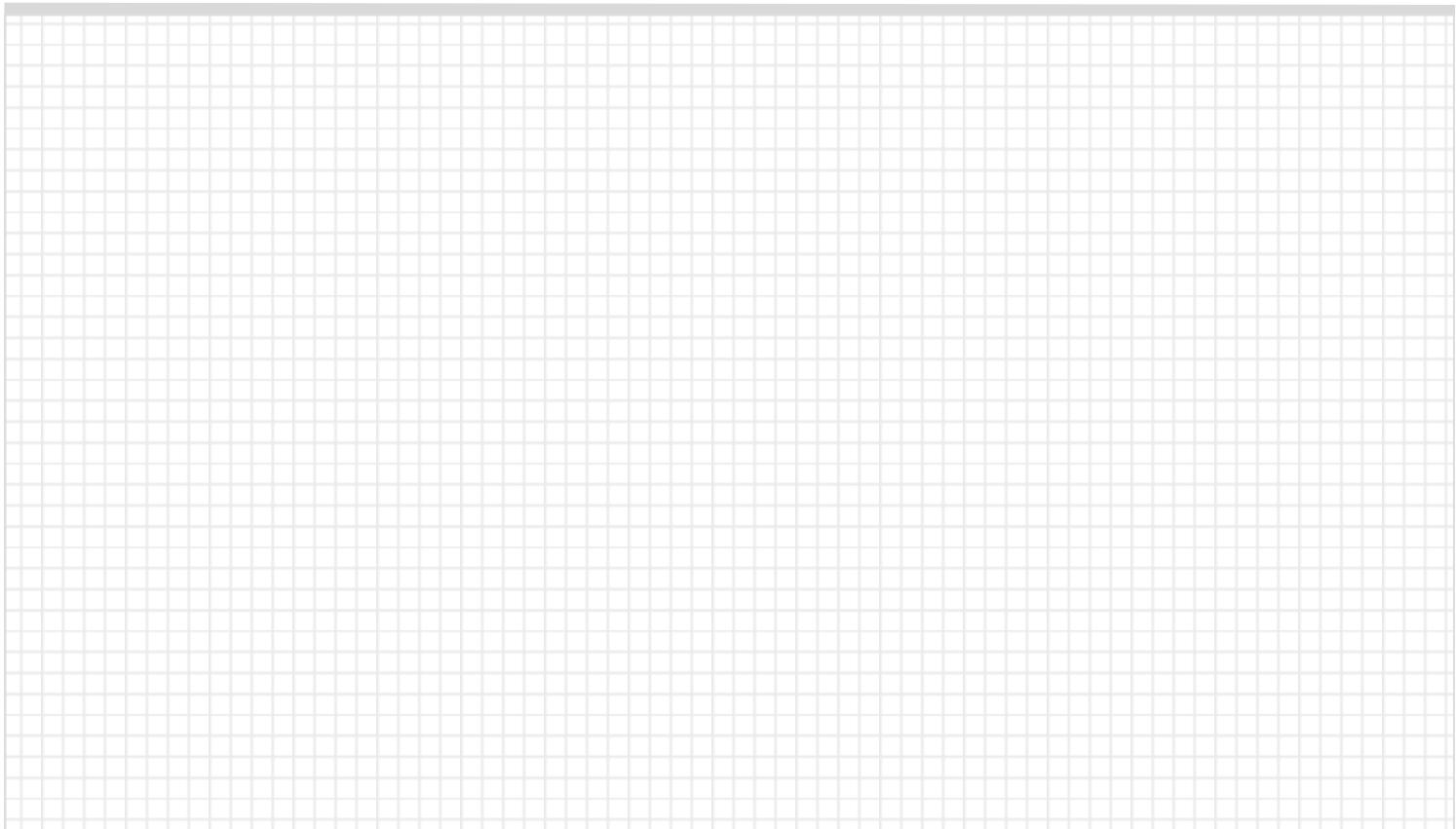
- Enter customer password
- Change customer password
- Delete customer password



End



Notes

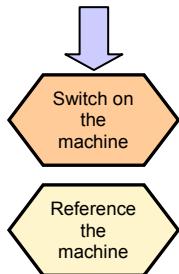


Content

Unit Description

This unit describes how to switch the machine on and reference it.

Unit Content



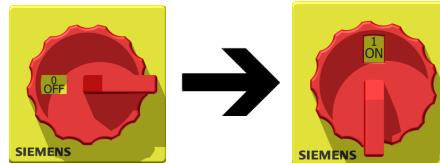
Switch on the machine



Please note the explicit switching on rules as specified by the machine manufacturer.

Step 1

Turn on the main switch of the machine.



The main switch is usually at the rear of the machine.

Step 2

Make sure you perform the following operation!



Release all the EMERGENCY STOP buttons on the machine!



End



Switch On And Referencing

SEQUENCE

Reference
the
machine

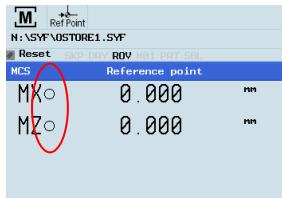


If your machine is configured with ABS encoder, you do not need to reference the axis of the machine.
If your machine is fitted with INC encoder, after power on the machine must first be referenced!

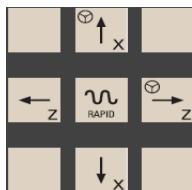
Step 1



After power on, the machine will be in the reference point approach mode (default).



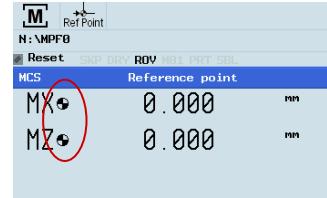
Step 2



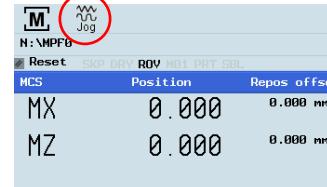
The axes are referenced with the corresponding axis traversing keys.

The traversing direction keys are specified by the machine manufacturer.

If the axis is not referenced, the non-referenced symbol (circle) is displayed between the axis identifier and the value.



Step 3



After completing the referencing procedure for all axes, the referenced symbol is displayed next to the axis identifier.

After returning to JOG mode, use the axis traversing keys to move the machine manually.

Now the machine can be operated in JOG mode.

During normal operation (JOG), the reference symbol is not shown on the screen.



End

Notes

A large rectangular area filled with a uniform grid of light gray lines, creating a pattern of small squares. This grid covers most of the page below the 'Notes' header, intended for handwritten notes.

Notes



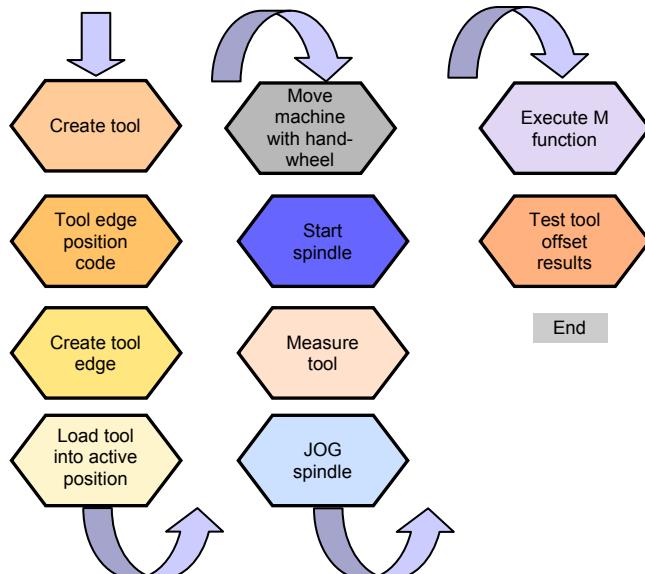


Content

Unit Description

This unit describes how to create and set up tools.

Unit Content



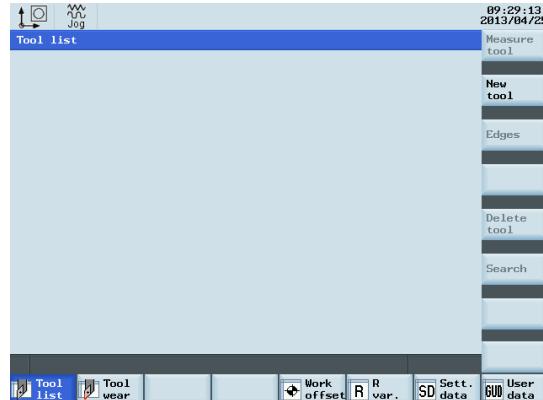
A tool must have been created and measured before executing the program.

Step 1 Please make sure the system is in JOG mode.

Press "Offset" on the PPU.



Press the "Tool list" SK on the PPU.





Tool Setup

SEQUENCE

Step 2



The range of tool numbers which can be created by this system is 1 ~32000. The machine can be loaded with a maximum of 64 tools / 128 tool edges.

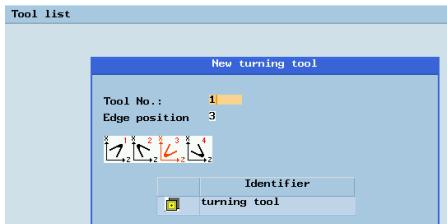
Press the "New tool" SK on the PPU.



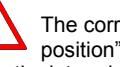
Select the type of required tool.



Enter "1" at "Tool No."



Enter "3" at "Edge position".



The correct "Edge position" selection directly determines the correct tool compensation which will be described in the next unit.

Press the "OK" SK on the PPU



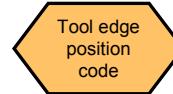
Enter the "Radius" and/or "Tip width" as required.



Press the "Input" button on the PPU



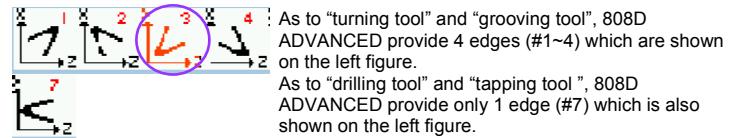
Programming and Operating — Turning



Principle of correct tool edge position code selection: Select the corresponding tool edge position code according to actual tool point direction!

Observe the relationship between the tool point direction and the positive direction of the X axis and the Z axis.

Find the corresponding position relationship in the figure below and enter the number in "Edge position", the red coordinate in the purple circle is the selected position code.

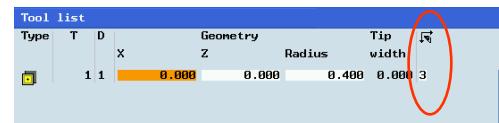


Note: Not every tool has eight position codes. All the options are shown above.



Note that the tool tip direction here is the direction after the correct tool offset, not only the direction in tool loading. And the correct of tool edge position code directly affects the tool tip radius compensation!

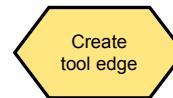
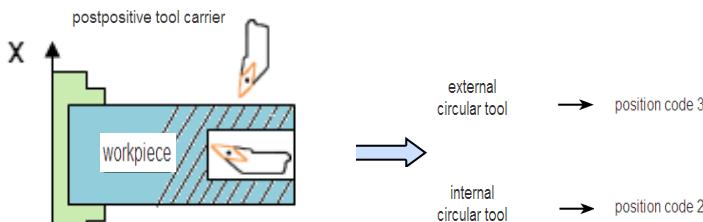
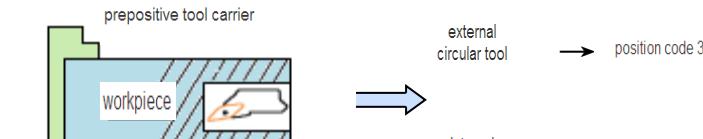
The tool edge position code can also be changed in the position showed in the figure.





SEQUENCE

Example Common tool edge position code choices are as follows:



A tool must have been created and selected with the cursor before creating a tool edge.

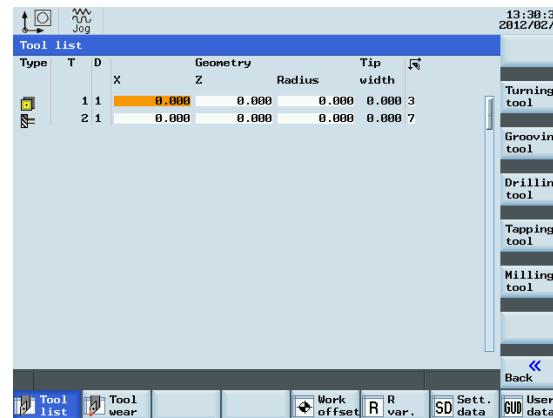
Step 1 Use "D" code to represent the tool edge. The system activates the No.1 tool edge as default at the beginning.

Press the "Offset" key on the PPU.



Press the "Tool list" SK on the PPU.

Use direction keys to select the tool which needs to have an additional a tool edge.



Press the "Edges" SK on the PPU.



Press the "New edge" SK on the PPU.

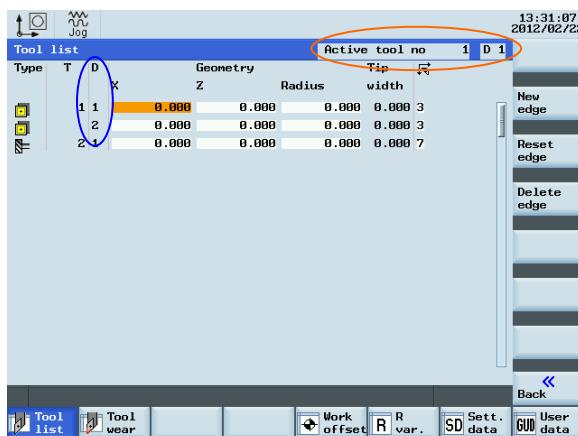


SEQUENCE

Step 2

A new tool edge can be added in this way and different lengths and radii can be entered as required.

The red circle shows the actual active tool and tool edge, the purple circle shows how many tool edges have been created and the related data for each tool edge.



!
A maximum of nine tool edges can be created for each tool!
Different tool lengths and radii can be saved in different tool edges as required.
Please select the right tool edge for machining according to requirement!



A tool must have been created in the system before it can be loaded into the active position.

Press the "Machine" key on the PPU.



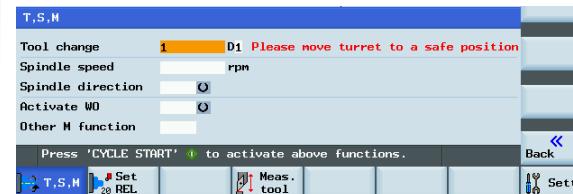
Press the "JOG" key on the MCP.



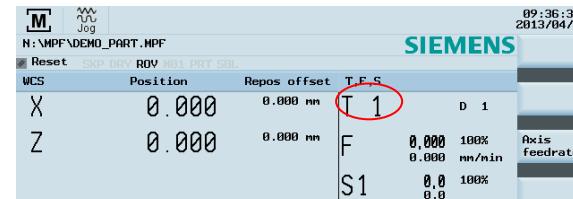
Press the "T.S.M" SK on the PPU.



Enter tool number "1" in "T".



Press "CYCLE START" on the MCP.

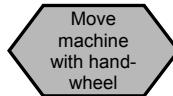


Press the "Back" SK on the PPU.





SEQUENCE



Make sure there is no obstruction when moving the tool to avoid a crash.

A handwheel can control the axis motion instead of the "JOG" button.

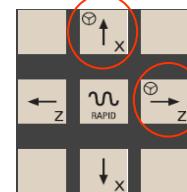
Press the "Machine" key on the PPU.



Press the "Handwheel" key on the MCP.



Select the axis you want to move with the appropriate keys on the MCP



MCS	Position	Repos offset
@X	0.000	0.000 mm
Z	0.000	0.000 mm

Under "WCS" or "MCS", a handwheel will be shown beside the axis symbols, representing that the axis can be moved using handwheel.

Select the required override increment according to the buttons on the right (this selection fits all axes)



The handwheel increment is "0.001 mm"



The handwheel increment is "0.010 mm"



The handwheel increment is "0.100 mm"



The selected axis can now be moved with the handwheel.

Press "JOG" on the MCP to end the "Handwheel" function.



Notes: if set MD14512[16]=80 , the system will deactivate the function of MCP for selecting the axis of handwheel, the user will have to activate "Handwheel" function with PPU softkey

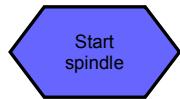
MCS		Position	Repos offset	T,F,S	
@X	0.000	0.000 mm		T 1	X
Z	0.000	0.000 mm		F	Z
				S1	8.0 100%
G1	G500(0)	G68			
		Handwheel MCS		SIEMENS	
		Axis Number		MCS	
		X 1			
		Z 2			
				Back	
		T,S,H Set REL		Meas. tool	
		Set REL		Sett.	

Handwheel

Select the required axis on the right of the PPU; the selected axis is shown with a



SEQUENCE



A tool must have been loaded and the turret rotated to position.

Start the spindle before adjusting tools as follows:

Press the "Machine" key on the PPU.



Press the "JOG" key on the MCP.



Press the "T.S.M" SK on the PPU.



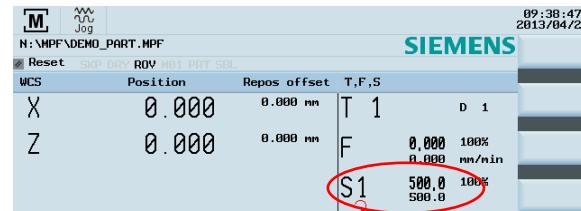
Enter "500" at "Spindle speed".



Select "M3" using the "Select" key on the PPU.



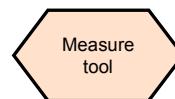
Press the "CYCLE START" key on the MCP.



Press "Reset" on the MCP to stop the spindle rotation.



Press the "Back" SK on the PPU.



A tool must have been created and the turret rotated before it can be measured!

Step 1 Measure length: X

Press the "Machine" key on the PPU.



Press the "JOG" key on the MCP.



Press the "Meas. tool" SK on the PPU.



Press the "Measure X" SK on the PPU.

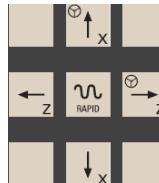
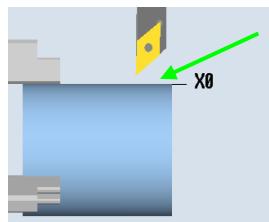




Tool Setup

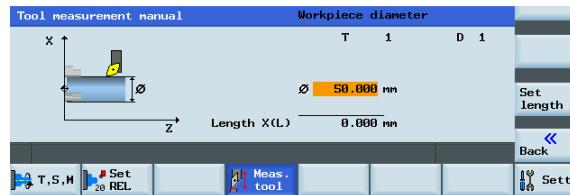
SEQUENCE

Use the traversing keys on the MCP to move the axis to the adjusted position.

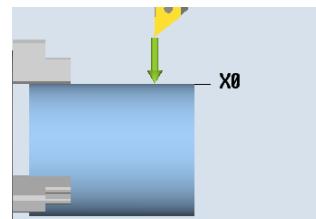


Note: "X=0" or "Z=0" in the workpiece coordinate system is shown as "X0" / "Z0" in the following text.

Enter 50 in " \emptyset "
(this is the diameter of the workpiece)



Use the
"Handwheel"
key on the
MCP and
select a suit-
able feedrate
override to
move the tool
to X0.



Move directly to zero point.

Press the "Set length X" SK on the PPU.





Tool Setup

SIEMENS

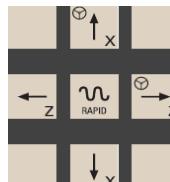
SEQUENCE

Step 2 Set length: Z

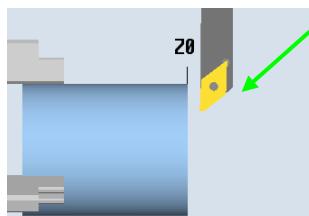
Press the "Set length Z" SK on the PPU.



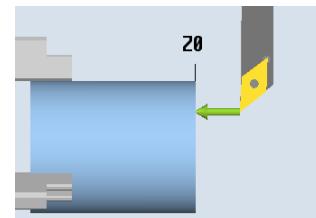
Set
length Z



Use the traversing keys on the MCP to move the axis to the adjusted position.

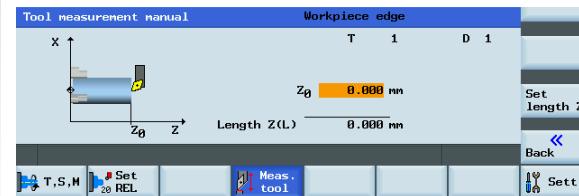


Use the "Handwheel" key on the MCP and select a suitable feedrate override to move the tool to Z0.



Move directly to zero point

Enter "0" in "Z0"
(this is the distance between the tool point and the zero point)



Press the "Set length Z" SK on the PPU.



Press the "Back" SK on the PPU.





Tool Setup

SEQUENCE



A tool must have been loaded and the turret rotated to the position!

Press the "Machine" key on the PPU.



Press the "JOG" key on the MCP.



Press the spindle direction key on the MCP to start/stop the spindle.



Press "Spindle left" on the MCP to start the spindle in the counter-clockwise direction.



Press "Spindle stop" on the MCP to stop the spindle.



Press "Spindle right" on the MCP to start the spindle in the clockwise direction.



M		Jog	SIEMENS	
N:\MPF\DEMO_PART.MPF			09:41:02 2013/04/25	
Reset		SPK DIRY REV HDT PRT SBL		
MCS	Position	Repos offset	T,F,S	
X	0 .000	0 .000 mm	T 1	D 1
Z	0 .000	0 .000 mm	F 0 .000 100% 0 .000 mm/min	Axis feedrate
			S1 100 .0 100% 100 .0	



Please make sure all the machine axes are in safe positions before executing the M function!

Press the "Machine" key on the PPU.



Press the "T.S.M" SK on the PPU.



Use the direction key to move the highlighted cursor to "Other M function" and enter "8". This will start the coolant.



Press "CYCLE START" on the MCP.



The coolant function key on MCP is active.



Press the "Reset" key on the MCP to stop the coolant function.



Press the "Back" SK on the PPU.





Tool Setup

SEQUENCE



The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

In order to ensure the machine safety and correctness, the results of the tool offset should be tested appropriately.

Press the "Machine" key on the PPU. M MACHINE

Press the "MDA" key on the MCP. MDA

Press the "Delete file" SK on the PPU. Delete file

Enter the test program recommended on the right (can also be customized). G500; select offset panel as required
T1 D1
G00 X0 Z5

Press the "ROV" key to ensure the "ROV" function is active (lit up). ROV

Note: The ROV function activates the feedrate override switch under the G00 function.

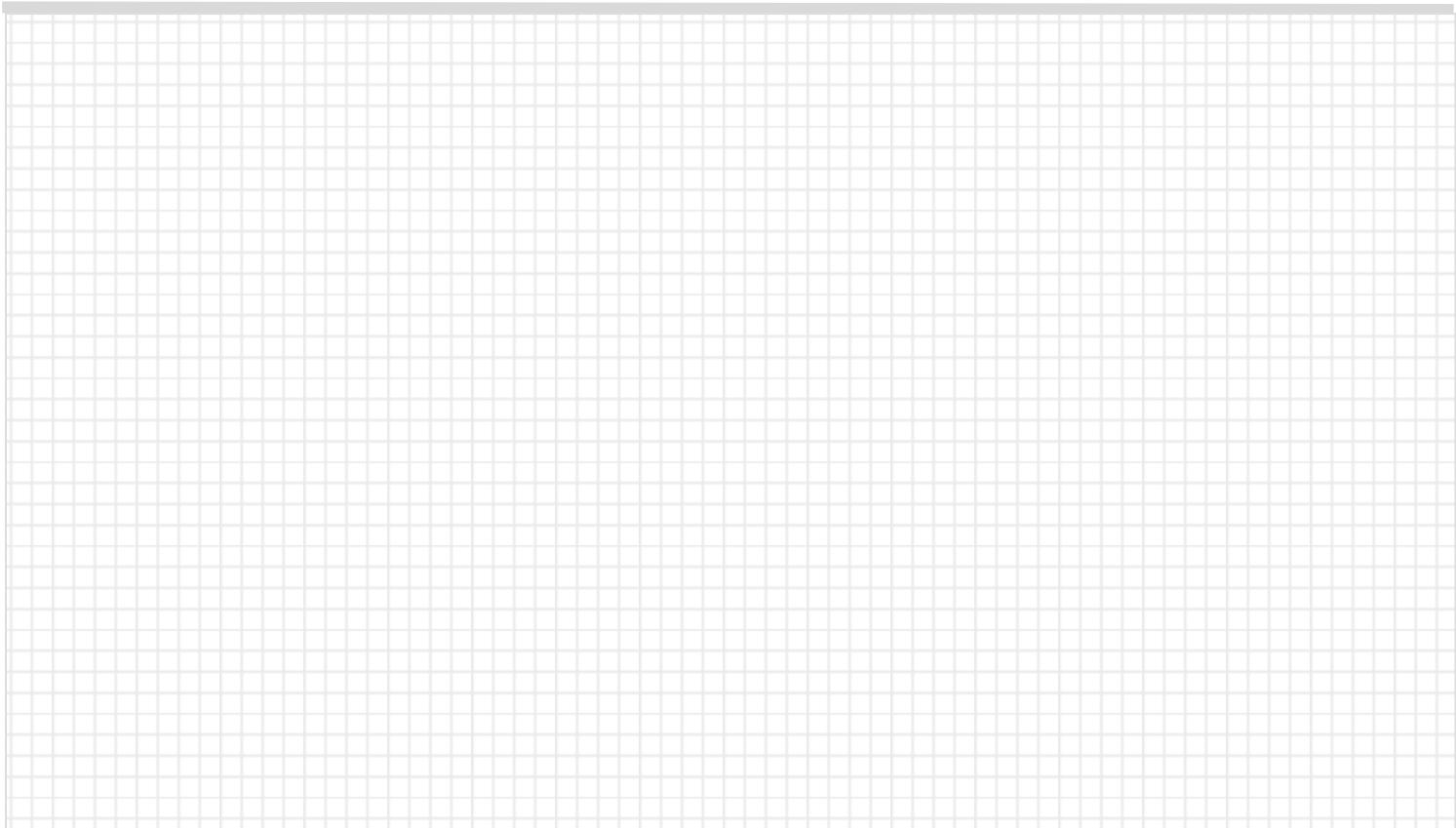


Make sure the feedrate override on the MCP is at 0%!

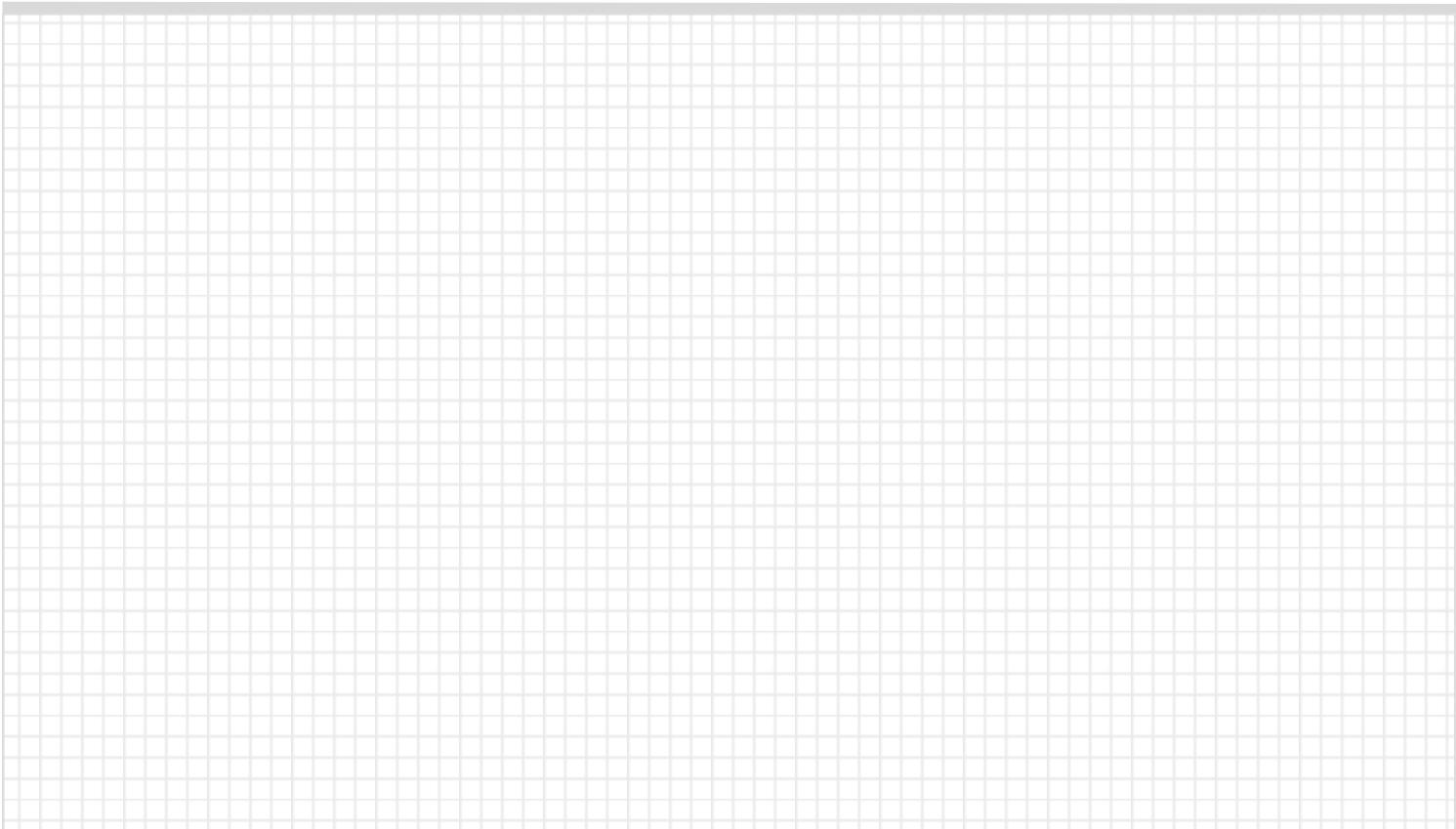
Press "CYCLE START" on the MCP. CYCLE START

Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast and observe whether the axis moves to the set position.

Notes



Notes

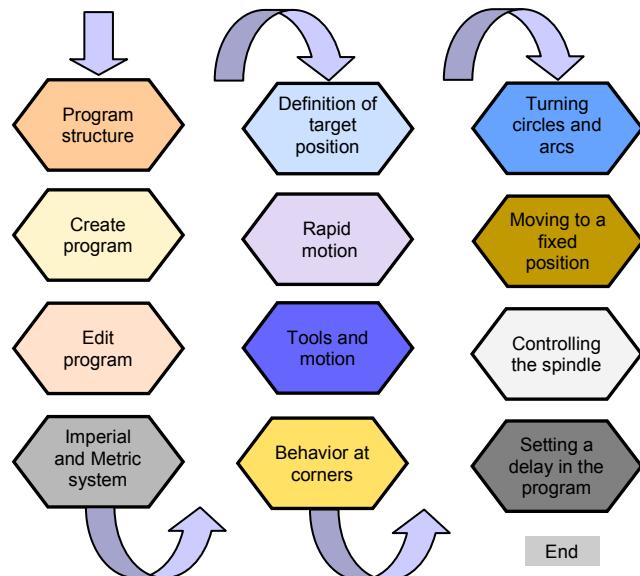


Content

Unit Description

This unit describes how to create and edit a part program, and get to know the most important CNC commands required to produce a workpiece.

Unit Content



BASIC THEORY

Program structure

A standard program structure is not needed but is recommended in order to provide clarity for the machine operator. Siemens recommends the following structure:

Return to change tool	N5 G17 G90 G54 G71
T, F, S function	N10 T1 D1 N15 S5000 M3 G95 F0.3
Geometry data / motion	N20 G00 X100 Z2 N25 G01 Z-5 N30 X105 N35 G00 SUPA X300 Z50 D0
Return to change tool	
T, F, S function	N40 T2 D1 N45 S3000 M3 G95 F0.2
Geometry data / motion	N50 G00 X99 Z2 N55 G01 Z-5 N60 X105 N65 G00 SUPA X300 Z50 D0
Return to change tool	
T, F, S function	N70 T3 D1 N75 S3000 M3 G95 F0.2
Geometry data / motion	N80 G00 X105 Z-25 N85 G01 X90 N90 X105 N95 G00 SUPA X300 Z50 D0
Return to change tool	
End/stop position	M30

Create Part Program Part 1

BASIC THEORY

Create program

The following sequence should be followed to create a part program:

Step 1

Programs can be created with the "program manager". You can select the "program manager" using the key located on the PPU.



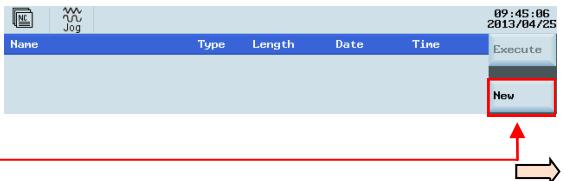
Step 2

Select NC as the storage location for the program. Programs can only be created in the NC.



Step 3

Create a new program with the "New" SK on the right of the PPU.



Step 4

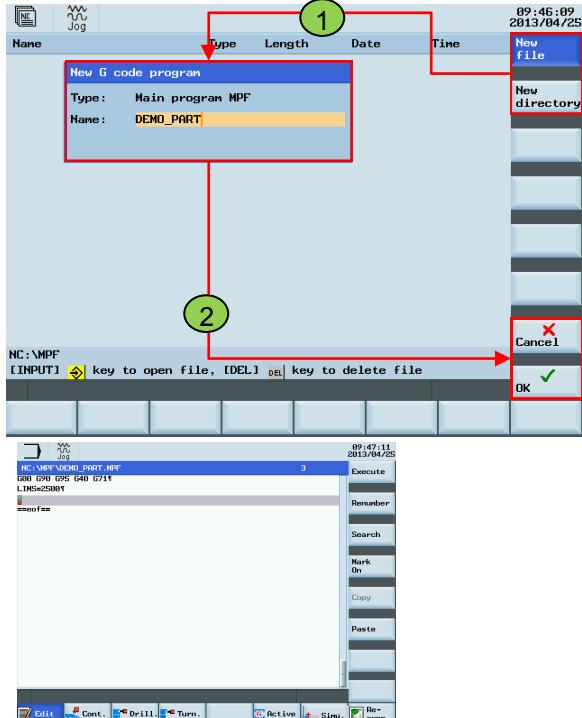
You can choose "New" or "New directory".

Choose "New" to create a program.

Choose "New directory" to create a folder.

Step 5

Now the program is open and can be edited.



After editing the system will save it automatically.

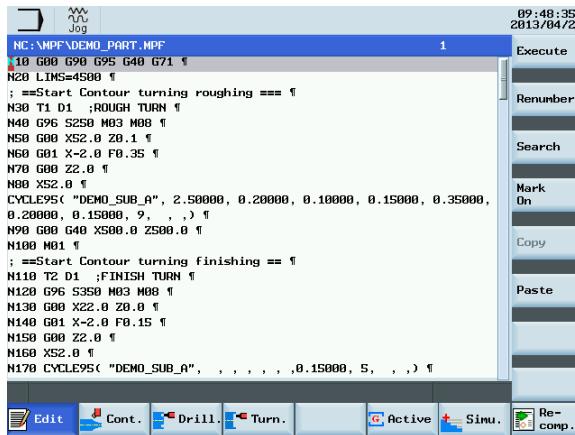
End

Create Part Program Part 1

BASIC THEORY

Edit program

The program shown in the editor can be created and edited with the correct keys.



808D ADVANCED

Inches and mm

G71

With G71 at the header, the geometry data will be in the metric unit system, feedrates in the default metric system.

[Return to change tool](#)

T, F, S function

Geometry data / motion

[Return to change tool](#)

N5 G17 G90 G54 G71

N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X100 Z1
N25 G01 X-0.5
N30 Z2
N35 G00 X200 Z50

G70

With G70 at the header, the geometry data will be in the imperial (inches) unit system, the feedrate in the default metric system.

[Return to change tool](#)

T, F, S function

Geometry data / motion

[Return to change tool](#)

N5 G17 G90 G54 G70

N10 T1 D1
N15 S5000 M3 G95 F0.2
N20 G00 X10 Z0.2
N25 G01 X-0.2
N30 Z0.2
N35 G00 X10 Z10



BASIC THEORY

Definition of target position

G50

All absolute path data will be relative to this position. The position is written in the G500 (basic) zero offset.

Or

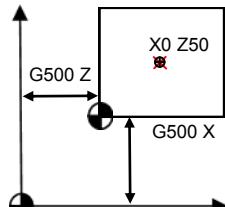
G54 G55 G56 G57 G58 G59

With G500 = 0, the offset for the work-piece can be stored in the G54 workpiece offset.

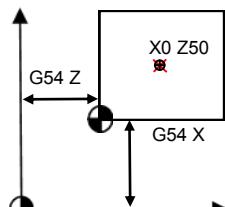
Or

G500 + G54

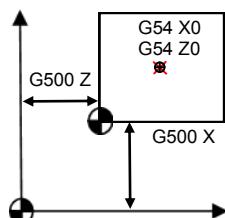
With G500 >< 0 and is activated, the value in G500 will be added to the value in G54.



N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X₅₀ Z₅
N25 G01 Z-5
N30 Z₅
N35 G00 Z₅₀ X₁₀₀



N5 G17 G90 G54 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X₀ Z₅
N25 G01 Z-5
N30 Z₅
N35 G00 Z₅₀ X₁₀₀



N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 G₅₄ X₂₀ Z₅
N25 G01 Z-5
N30 Z₅
N35 G00 G₅₃ Z₅₀ X₁₀₀

G90

Absolute positioning; with G90 at the beginning of the program, the geometry data which follows will be interpreted relative to the active zero point in the program, usually with G54 or G500 or G500 + G54.

N5 G17 G90 G54 G71

N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X₁₀₀ Z₅
N25 G01 Z-20
N30 Z₅
N35 G00 Z₅₀₀ X₁₀₀

G91

Relative positioning; with G91 you can add an incremental value (G91 defined data is the relative positioning using the present position as the start point).

Finally you should change the program to absolute positioning with G90.

N5 G17 G90 G54 G70

N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X_{3.93} Z_{0.196}
N25 G01 G₉₁ Z-0.787
N30 Z_{0.196}
N35 G00 G₉₀ Z_{19.68} X₁₀

Create Part Program Part 1

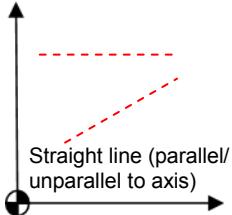
SIEMENS

BASIC THEORY

Rapid motion

G00

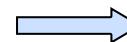
When G00 is active in the program, the axis will traverse at the maximum axis speed in a straight line.



N5 G17 G90 G54 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-20
N30 Z5
N35 G00 Z500 X200

- Feedrate
- Spindle speed
- Feed type
- Spindle direction

N5 G17 G90 G54 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 Z200



The feedrate is defined in the program with "F". Two types of feedrate are available:

1. Feed per minute → G94
2. Feed per revolution of the spindle → G95

N5 G17 G90 G54 G71

G94
Defines the feedrate in terms of time mm/min.



G95
Defines the feedrate in terms of spindle revolutions mm/rev.

N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 Z200

S

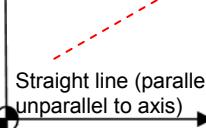
The spindle speed is defined with "S" **S5000**

M3/M4

The spindle direction is defined with M3 and M4, clockwise/counter-clockwise respectively.

G01

When G01 is active in the program, the axis will traverse at the programmed feedrate in a straight line, according to the feedrate type defined by G94 or G95.



Tools and motion

T1 D1

With the "T" command the new tool can be selected, the "D" command is used to activate the tool length offset.



N5 G17 G90 G54 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 Z200



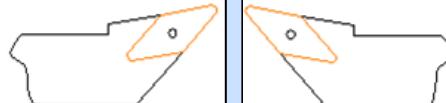
BASIC THEORY



Activation/
deactivation of the
tool radius compensa-
tion when working on
the part contour.

G41 / G42 and G40

With G41/G42,
the radius compensa-
tion of the tool will be
done in the direction
of travel.



G41: Compensation
to left.

G41 → direction
along the tool
motion, the tool is
always on the left of
the contour.

Arrow indicates the
direction of tool motion
along the contour

G42: Compensation
to right.

G40: Compensation
of the radius can be
deactivated.



BASIC THEORY

Turning circles and arcs

The circle radius shown in the example on the right can be produced with the specified part program code. When milling circles and arcs, you must define the circle center point and the distance between the start point / end point and the center point on the relative coordinate.

When working in the XZ coordinate system, the interpolation parameters I and K are available.

Two common types of defining circles and arcs:

①: G02/G03 X_Z_I_K_;

②: G02/G03 X_Z_CR=;

Arcs $\leq 180^\circ$, CR is a positive number

Arcs $> 180^\circ$, CR is a negative number

N5 G17 G90 G500 G71

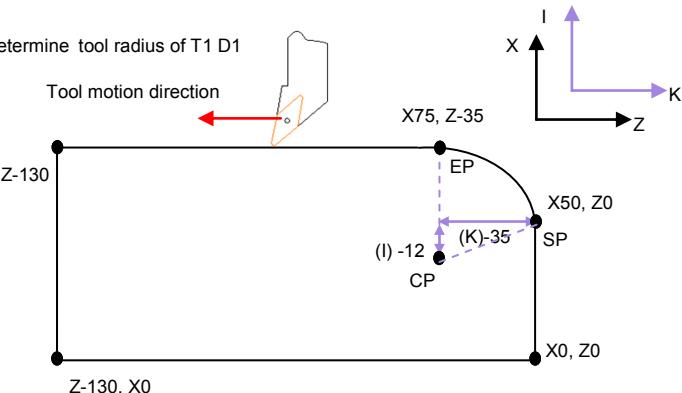
 N10 T1 D1
 N15 S5000 M3 G95 F0.3
 N20 G00 X0 Z2
 N25 G01 Z0
 N30 G42 X50
 N45 G03 X75 Z-35 I-12 K-35
 N50 G01 Z-130
 N60 G40 X120 Z-140
 N35 G00 X300 Z500

Note:

N45 can also be written as follows
 N45 G03 X75 Z-35 CR=37

Determine tool radius of T1 D1

Tool motion direction



SP = start point of circle

CP = center point of circle

EP = end point of circle

I = defined relative increment from start point to center point in X

K = defined relative increment from start point to center point in Z

G2 = define circle direction in traversing direction = G2 clockwise

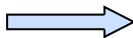
G3 = define circle direction in traversing direction = G3 counter-clockwise

BASIC THEORY

Moving to a
fixed
position

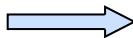
Using the code **G74**,
the machine can
move to the
reference point
automatically.

N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 X=0 ; reference point



Using the code **G75**,
the machine can
move to the fixed
position defined by
machine supplier
automatically.

N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 Z=0 ; reference point
N40 G75 X=0 ; fixed point



Controlling
the spindle

The following functions can be
used to influence the operation of
the spindle:

M3 accelerate to programmed
speed clockwise.

M4 accelerate to programmed
speed counter-clockwise.

M5 spindle decelerate to stop.

M19 orient the spindle to a
specific angular position.

Setting a
delay in the
program

G04 can be used to pause the
tools' movements during opera-
tion

G04 F5: Program dwells for 5 s

This makes the surface of the
workpiece much smoother.

N5 G17 G90 G500 G71

N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 M5
N35 Z5 M4
N40 M5
N45 M19
N50 G00 X200 Z50



N5 G17 G90 G500 G71

N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 G04 F5
N35 Z5 M4
N40 M5
N45 M19
N35 G00 X200 Z50



Notes



Notes



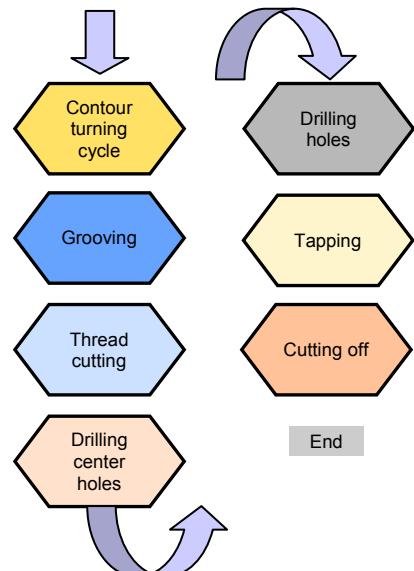
Create Part Program Part 2

Content

Unit Description

This unit describes how to create and edit a part program, and get to know the most important CNC commands required to produce a workpiece.
Part 2

Unit Content



BASIC THEORY

Contour turning cycle

Step 1

The easiest way to perform roughing/finishing along the contour is to use the "contour turning" cycle function. By selecting the "Turn." SK, you can enter the cycle and set parameters.



The "Contour turning" SK can be found on the right vertical menu.



The related parameters can be set on the screen.

14:12:22
2013/04/19

10 Stock removal

NC : MPF\DEMO_PART.MPF

N60 G01 X-2.0 F0.35 1

H70 G00 Z2.0 1

N80 XS2.0 1

CYCLE95C "DEMO_SUB_A", 2.50000, 0.20000, 0.10000, 0.15000, 0.35000, .20000, 0.15000, 9, , , 1

H70 G00 X0.0 Y2.000.0 Z2.000.0 1

N10 G01 M01 1

; ==Start Contour turning finishing == 1

H110 T2 D1 ;FINISH TURN 1

H120 G95 G33B M03 M08 1

H130 G00 X22.0 Z0.0 1

H140 G01 X-2.0 F0.15 1

H150 G00 Z2.0 1

H160 XS2.0 1

H170 CYCLE95C "DEMO_SUB_B", , , , , 0.15000, 5, , , 1

H180 G00 G40 XS0.0 Z500.0 1

N190 M01 1

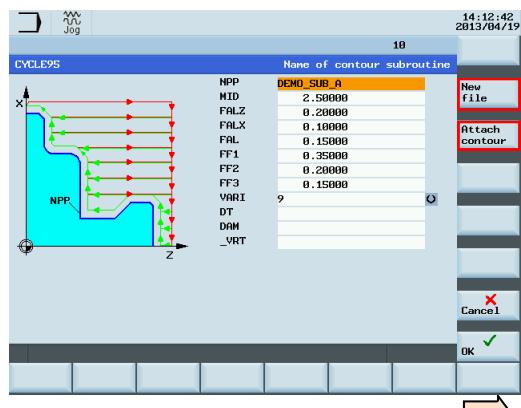
; ==Start Grooving ===== 1

H200 T3 D1 ;GROOVE 1

H210 G95 S200 M03 M08 1

H220 G00 XS5.0 Z0.0 1

Edit Cont. Drill Turn Active Simu Re-comp.

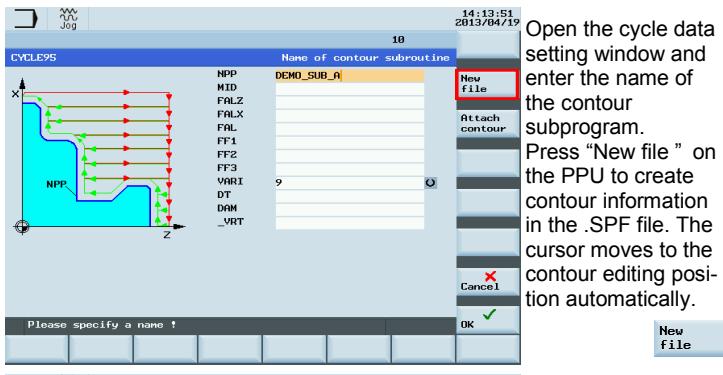


Create Part Program Part 2

BASIC THEORY

New file

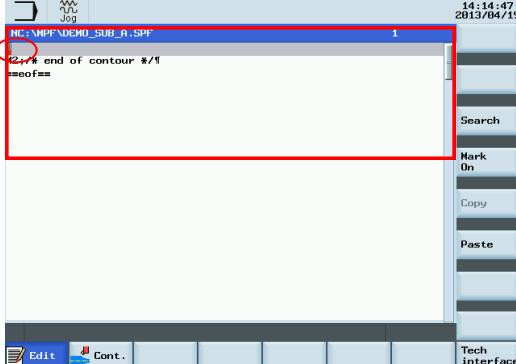
By selecting the “New file” SK, the contour turning data can be inserted into Sub Program File (.SPF). You can edit and change it when selected. The sequence is as follows:



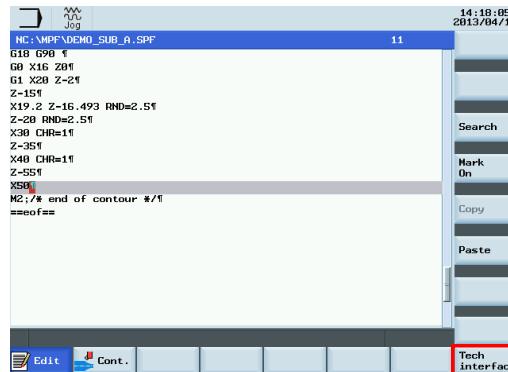
Open the cycle data setting window and enter the name of the contour subprogram. Press “New file ” on the PPU to create contour information in the .SPF file. The cursor moves to the contour editing position automatically.

New file

Make sure the cursor is in the editing position (shown in the figure on the left).



After opening the contour data setting window, please make the following settings:



Enter appropriate coordinates based on the data from the technical drawing.

Step 2 Radius and chamfers

The radii and the chamfer can be produced using the contour editor, in conjunction with the roughing or finishing cycles.

RND and CHR/CHF can be found in the additive description of the T contour.

RND = Radii

CHR = Chamfer
(specified side length of isosceles triangle with chamfer as base line)

CHF = Chamfer
(specified base line length of isosceles triangle with chamfer as base line)

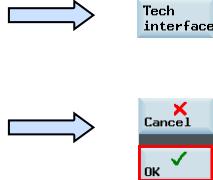
Create Part Program Part 2

SIEMENS

BASIC THEORY

After completing the steps, the system will return to the Edit interface, press "Tech interface" SK on the PPU to return to the interface for setting the cycle data.

After finishing the parameter settings of CYCLE95, press the "OK" SK on the PPU to insert the corresponding cycles in the main program.



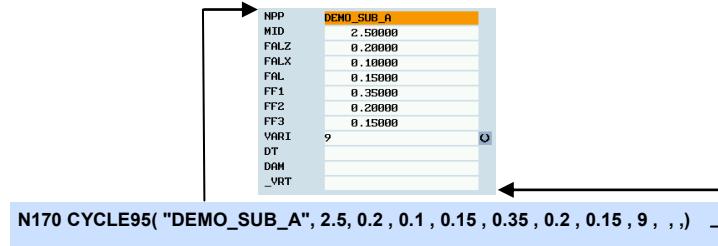
```

NC_MPFNDEMOPART.MPF          10   14:12:22
N60 G01 X<-2.0 F0.35
N70 G90 Z0.0
N90 XS2.0
CYCLE95C("DEMO_SUB_A", 2.50000, 0.20000, 0.10000, 0.15000, 0.35000,
, 0.20000, 0.15000, 9, , ,)
N90 G90 G40 XS98.0 Z598.0
N100 M01
; ==Start Contour turning finishing ==
N110 T2 D1 :FINISH TURN
N120 G96 S550 M03 M08
N130 G00 X22.0 Z0.0
N140 G01 X<-2.0 F0.15
N150 G00 Z22.0
N160 XS2.0
N170 CYCLE95("DEMO_SUB_B", , , , , 0.15000, 5, , ,)
N180 G90 G40 XS98.0 Z598.0
N190 M01
; ==Start Grooving =====
N200 T3 D1 :GROOVE
N210 G96 S200 M03 M08
N220 G00 XS5.0 Z0.

```

Toolbars at the bottom: Edit, Cont., Drill, Turn, Active, Simu., Re-comp.

After all the settings take effect, the selected cycle and set data will be transferred to corresponding part program automatically (for further information, see next page).



```

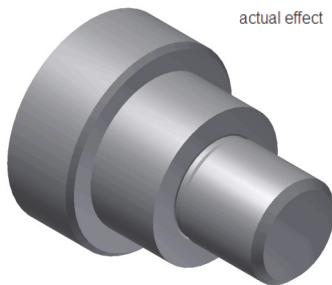
G18 G90
G0 X16 Z0
G1 X20 Z-2
Z-15
X19.2 Z-16.493 RND=2.5
Z-20 RND=2.5
X30 CHR=1
Z-35
X40 CHR=1
Z-55
X50
M2; /* end of contour */

```

BASIC THEORY

Parameters	Meanings	Remarks
NPP=DEMO:DEMO_E	Subprogram name: "DEMO" (:"DEMO_E" is created automatically)	The first two positions of the name must be letters.
MID=2.5	Maximal feed depth 2.5 mm	
FALZ=0.2	Finishing allowance at the vertical axis is 0.2 mm	
FALX=0.1	Finishing allowance at the horizontal axis is 0.1 mm	
FAL=0.15	Contour finishing allowance is 0.15 mm	
FF1=0.35	Roughing feedrate is 0.35 mm/rev	
FF2=0.2	Feedrate with back cut is 0.2 mm/rev	
FF3=0.15	Finishing feedrate is 0.15 mm/rev	
VARI=9	Do horizontal complete machining externally	For other parameters, please refer to the standard manual

actual effect



Create Part Program Part 2

SIEMENS

BASIC THEORY

Grooving

The easiest way to produce a groove is to use CYCLE93

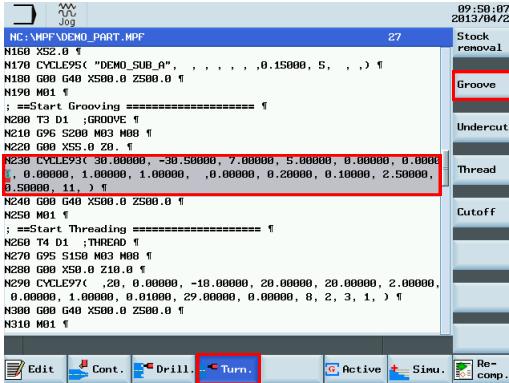
The cycle can be found and parameterized with the "Turn." SK.

 Turn.

The relevant cycle can now be found using the vertical SKs on the right.

 Groove

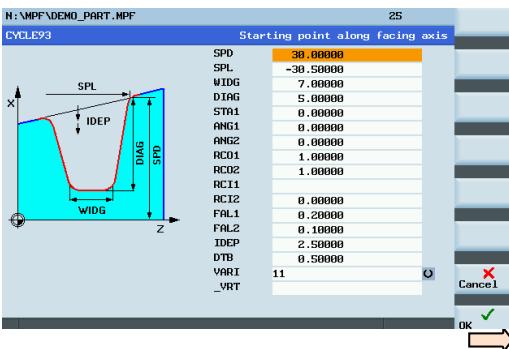
Select "Groove" using the vertical SKs and parameterize the cycle according to requirement.



```

NC:\MPF\DEMO_PART.MPF
27
N168 G52 Z0 0
N170 CYCLE93( "DEMO_SUB_A", , , , , 0.15000, 5, , , ) 1
N180 G00 G40 X500.0 Z500.0 1
N190 M01 1
; ==Start Grooving =====
N200 T3 D1 ;GROOVE 1
N210 G95 S200 M03 M08 1
N220 G00 X55.0 Z9.0 1
N230 CYCLE93( 30.00000, -30.50000, 7.00000, 5.00000, 0.00000, 0.00000
, 0.00000, 1.00000, 1.00000, 0.00000, 0.20000, 0.10000, 2.50000,
0.50000, 11, ) 1
N240 G00 G40 X500.0 Z500.0 1
N250 M01 1
; ==Start Threading =====
N260 T4 D1 ;THREAD 1
N270 G95 S150 M03 M08 1
N280 G00 X50.0 Z10.0 1
N290 CYCLE97( ,20, 0.00000, -18.00000, 20.00000, 20.00000, 2.00000,
0.00000, 1.00000, 0.01000, 29.00000, 0.00000, 8, 2, 3, 1, ) 1
N300 G00 G40 X500.0 Z500.0 1
N310 M01 1

```



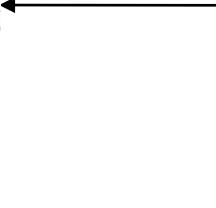
With the "OK" SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.

The machine will cut a groove at the position specified in the cycle.

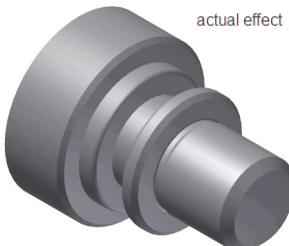


BASIC THEORY

SPD	30.00000
SPL	-30.50000
WIDG	7.00000
DIAG	5.00000
STA1	0.00000
ANG1	0.00000
ANG2	0.00000
RCO1	1.00000
RCO2	1.00000
RCI1	
RCI2	0.00000
FAL1	0.20000
FAL2	0.10000
IDEP	2.50000
DTB	0.50000
VARI	11
_VRT	



N230 CYCLE93(30.00000, -30.50000, 7.00000, 5.00000, 0.00000, 0.00000,
0.00000, 1.00000, 1.00000, ,0.00000, 0.20000, 0.10000, 2.50000, 0.50000, 11,)



Parameters	Meanings	Remarks
SPD=30	Starting coordinate at horizontal axis is 30	
SPL=-30.5	Starting coordinate at vertical axis is -30.5	
WIDG=7	Groove width is 7 mm	
DIAG=5	Groove depth is 5 mm	
STA1=0 (range 0°~180°)	Angle between contour and vertical axis is 0°	
ANG1=0 (range 0°~89.999°)	Angle between positive vertical axis and groove cliff near starting point is 0°	
ANG2=0 (range 0°~89.999°)	Angle between positive vertical axis and groove incline away from starting point is 0°	
RCO1=1	Forward angle length away from machining starting point is 1mm	
RCO2=1	Reverse angle length away from machining starting point is 1mm	
RCI1=0	Groove bottom with no reverse angle (near groove machining starting point)	
RCI2=0	Groove bottom with no reverse angle (away from groove machining starting point)	
FAL1=0.2	Finishing allowance at the bottom of groove is 0.2 mm	
FAL2=0.1	Finishing allowance at groove side is 0.1 mm	
IDEP=2.5	Feed depth is 2.5 mm	
DTB=0.5	Pause 0.5 s at the bottom of groove	
VARI=11	Use CHR to calculate the reverse angle	For other parameters please refer to the standard manual



Create Part Program Part 2

SIEMENS

BASIC THEORY

Thread cutting

The easiest way to cut a thread is to use CYCLE99

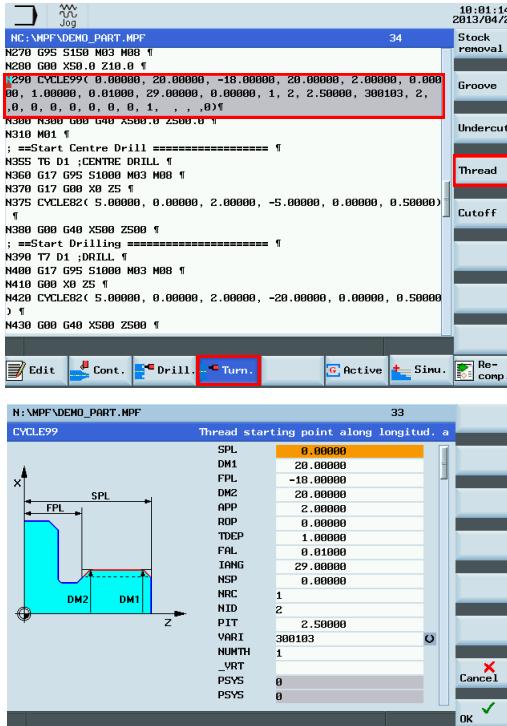
The cycle can be found and parameterized with the "Turn." SK.

 Turn.

The relevant cycle can now be found using the vertical SKs on the right.

 Thread → Thread long.

Select "Thread" and "Thread long," using the vertical SKs and parameterize the cycle according to requirements.



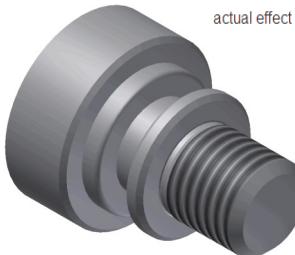
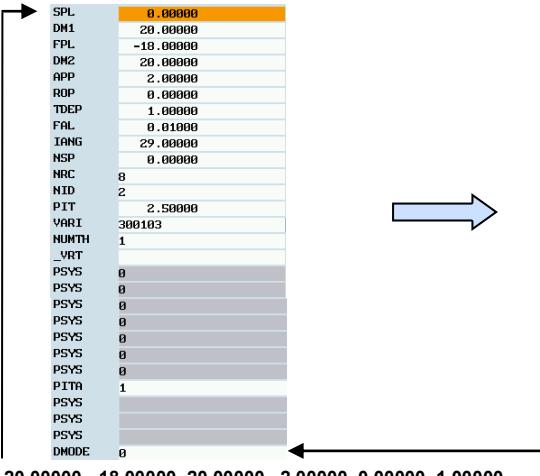
With the "OK" SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.
The machine will cut a thread at the position specified in the cycle.



Create Part Program Part 2

SIEMENS

BASIC THEORY



Parameters	Meanings	Remarks
SPL=0	Thread start point coordinate at vertical axis is 0	
FPL=-18	Thread end point coordinate at vertical axis is -18 mm	
DH1=20	Thread diameter at start point is 20 mm	
DH2=20	Thread diameter at end point is 20 mm	
APP=2	Reverse distance is 2 mm	
ROP=0	End distance is 0 mm	
TDEP=1	Thread depth is 1 mm	
FAL=0.01	Finishing allowance is 0.01 mm	
IANG=29	Feed along the same face, feed angle is 29°	IANG<0: feed along two faces in turn
NSP=0 (range 0°~359.9999°)	In comparison with the starting point, the angle offset of the first thread cutting point is 0°	
NRC=8	Roughing cutting 8 times	
NID=2	Empty tool cutting steps 2	
PIT=2.5	Thread distance is 2.5 mm	
VARI=300103	Machining externally, constant cross session	For other parameters, please refer to the standard manual
NUMTH=1	Thread number of multi-head thread is 1	
PITA=1	Select data in the PIT and in mm	
DMODE=0	Thread types	

Create Part Program Part 2

SIEMENS

BASIC THEORY

Drilling center holes

The easiest way to drill a center hole prior to drilling is to use either CYCLE81 or CYCLE82.

CYCLE81: Without dwell at current hole depth

CYCLE82: With dwell at current hole depth

The cycle can be found and parameterized with the "Drill." SK.



The relevant cycle can now be found using the vertical SKs on the right.

Center drilling → Center drilling

Select "Center drilling" using the vertical SKs , and then select "Center drilling" parameterize the cycle according to requirement.

```

NC:\WPF\DEMO_PART.MPF          41
N270 G95 S1500 M03 M08 1
N280 G00 X50.0 Z10.0 1
N290 CYCLE89( 0.00000, 20.00000, 20.00000, 2.00000, 0.00000
00, 1.00000, 0.01000, 29.00000, 0.00000, 1, 2, 2.50000, 300103, 2,
.0, 0, 0, 0, 0, 0, 0, 1, ., ., 0) 1
N300 N390 G00 G60 G40 X500.0 Z500.0 1
N310 M01 1
; ==Start Centre Drill ===== 1
N355 T6 D1 ;CENTRE DRILL 1
N360 G17 G95 S1000 M03 M08 1
N370 G17 G60 X0 Z5 1
N375 CYCLE82( 5.00000, 0.00000, 2.00000, -5.00000, 0.00000, 0.50000)
1
N380 G00 G60 X500 Z500 1
; ==Start Drilling ===== 1
N390 T7 D1 ;DRILL 1
N400 G17 G95 S1000 M03 M08 1
N410 G60 X0 Z5 1
N420 CYCLE82( 5.00000, 0.00000, 2.00000, -20.00000, 0.00000, 0.50000)
1
N430 G00 G60 X500 Z500 1

Edit Cont. Drill Turn. Active Simu. Re-comp.

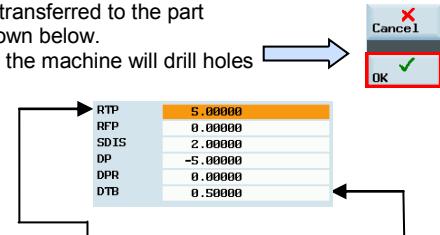
N:\WPF\DEMO_PART.MPF          48
CYCLE82
RTP: 5.00000
RFP: 0.00000
SDIS: 2.00000
DP: -5.00000
DPR: 0.00000
DTB: 0.50000

Modal call
Cancel
OK ✓

```

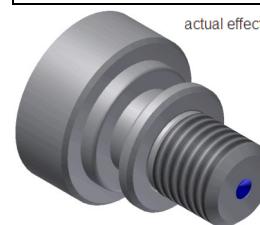
With the "OK" SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.

If there is no other operation, the machine will drill holes at the current position.



N375 CYCLE82(5.00000, 0.00000, 2.00000, -5.00000, 0.00000, 0.50000)

Parameters	Meanings
RTP=5	Coordinate value of turning position is 5 (absolute)
RFP=0	Coordinate value of hole edge starting position under workpiece zero point surface is 0 (absolute)
SDIS=2	Safety distance, feed path changes from quick feed to machine feed 2 mm away from RFP face (frequently used values 2~5)
DP=-5	Coordinate position of final drilling depth is -5 mm (absolute)
DTB=0.5	Dwell of 0.5 sec at final drilling depth



Create Part Program Part 2

SIEMENS

BASIC THEORY

Drilling holes

The easiest method to drill holes is with CYCLE81/82: Without/with dwell at current hole depth.

CYCLE83: Each drilling operation needs a withdrawal distance during deep hole drilling.

The cycle can be found and parameterized with the "Drill." SK.



The relevant cycle can now be found using the vertical SKs on the right.

Center drilling → Center drilling

Select "Center drilling" using the vertical SKs ,and then select "Center drilling" and parameterize the cycle according to requirements.

NC:\MPF\DEMO_PART.NPF 47

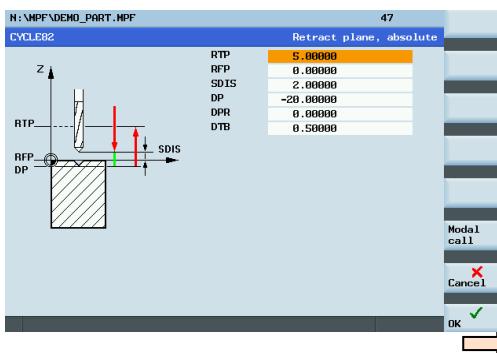
```

N360 G17 G95 S1000 M03 M08 1
N374 G17 G90 X0 Z5 1
N375 CYCLE82( 5.00000, 0.00000, 2.00000, -5.00000, 0.00000, 0.50000)
1
N380 G00 G40 XS00 Z500 1
; ==Start Drilling =====
N394 T1 D1 ;DRILL 1
N400 G17 G95 S1000 M03 M08 1
N410 G00 X0 Z5 1
N420 CYCLE82( 5.00000, 0.00000, 2.00000, -20.00000, 0.00000, 0.50000)
1
N430 G00 G40 XS00 Z500 1
; ==Start Tapping =====
N440 T1 D1 ;TRAP HOLE 1
N450 G17 G95 S500 M3 M08 1
N460 G00 X0 Z5 1
N470 CYCLE84( 5.00000, 0.00000, 2.00000, -18.00000, 0.00000, 0.50000
, -3.12.00000, 0.00000, 200.00000, 200.00000, 3, 0, 0, 0, 0.0000
0) 1
N480 G0 G40 XS00 Z500 1
; ==Start Cut-Off =====

```

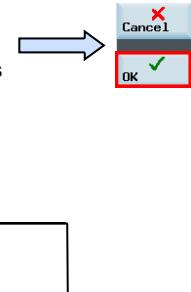
Drilling centering
Center drilling
Deep hole drilling
Boring
Thread
Deselect modal

Edit Cont. Drill Turn Active Simu Re-comp.



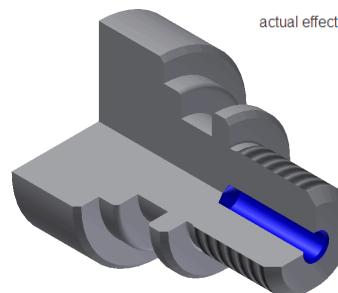
With the "OK" SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.

If there is no other operation, the machine will drill holes at the current position.



N420 CYCLE82(5.00000, 0.00000, 2.00000, -20.00000, 0.00000, 0.50000)

For RTP, RFP, SDIS, DP, DPR and DTB and related commands, see page 50



Create Part Program Part 2

SIEMENS

BASIC THEORY

Tapping

The easiest way to tap a hole is to use CYCLE84: Solid tap holder CYCLE840: With floating tap holder. The cycles can be found and parameterized using the "Drill." SK.



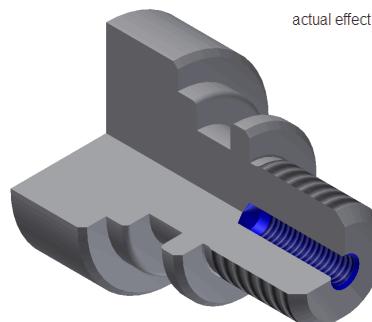
The relevant cycle can now be found using the vertical SKs on the right.

Thread → **Rigid tapping**

Select "Thread" using the vertical SKs and then select "Rigid tapping." and parameterize the cycle according to requirement.

The screenshot shows the SIMATIC Manager interface with the NC program N:\MPF\DEMO_PART.NPF open. The program contains several G-code commands, including cycles for drilling and tapping. On the right, a vertical stack of function keys (SKs) is displayed. The 'Thread' SK is highlighted with a red box. Below the NC program, the 'Drill' button is also highlighted with a red box. The 'CYCLE84' dialog box is open, showing parameters for rigid tapping. The 'OK' button at the bottom of the dialog box is also highlighted with a red box.

With the "OK" SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below. If there is no other operation, the machine will drill holes at the current position.

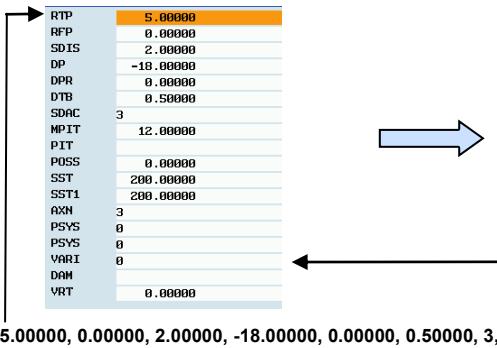


actual effect





BASIC THEORY



Parameters	Meanings	Remarks
DTB=0.5	Pause 0.5 s during final tapping to thread depth (discontinuous cutting)	
SDAC=3	Spindle state after cycle is M3	Enter values 4/5→M4/M5
MPIT=12 (value range:M3~M48)	Thread distance is same as values corresponding to the thread size M12	Negative value→rotate thread left
POSS=0	Spindle stops at 0° (unit: °)	
SST=200	Tapping thread spindle speed is 200 r/min	
SST1=200	Retraction spindle speed is 200 r/min	Direction is opposite to SST SST1=0 →speed is same as SST
AXN=3	AXN is tool axis, , use Z axis under G17	
VARI=0	Tapping is active	
VRT=0	Retraction value during discontinuous cutting is 1 mm	VRT>0→retraction value is fixed



Data in SST and SST1 control the spindle speed and the Z axis feed position synchronously.
During execution of CYCLE 84 the switches of the feedrate override and the cycle stop (maintaining feed) switch are not active.

For descriptions of RTP, RFP, SDIS, DP and DTB, please see page 50

Create Part Program Part 2

SIEMENS

BASIC THEORY

Cutting off

The easiest way to cut off a part is to use CYCLE92. The cycle can be found and parameterized using the "Turn." SK.



The relevant cycle can now be found using the vertical SKs on the right.



Select "Cutoff" using the vertical SKs and parameterize the cycle according to requirements.

```

N:\MPP\DEMO_PART.MPF      S9      10:07:22
N450 G17 G95 S500 H3 M08 1
N460 G00 X0 Z5 1
N470 CYCLE84( 5.00000, 0.00000, 2.00000, -18.00000, 0.00000, 0.50000
, 3, -0.00000, , 0.00000, 200.00000, 200.00000, 3, 0, 0, 0, , 0.00000
) 1
N480 G00 G40 XS00 Z500 1
; ==Start Cut-off =====
N320 T5 D1 ;CUT-OFF 1
N330 G16 G96 S200 H03 M08 1
N340 G00 XS5.0 Z10.0 1
N350 CYCLE92( 40.00000, -50.00000, 0.00000, -1.00000, 0.50000,
, 200.00000, 2500.00000, 3, 0.20000, 0.00000, 500.00000, 0, 0, 1, 0, 1100
) 1
N351 G00 G40 XS00 Z500 1
N360 G00 G40 XS00.0 Z500.0 1
N370 M30 1
==eof==

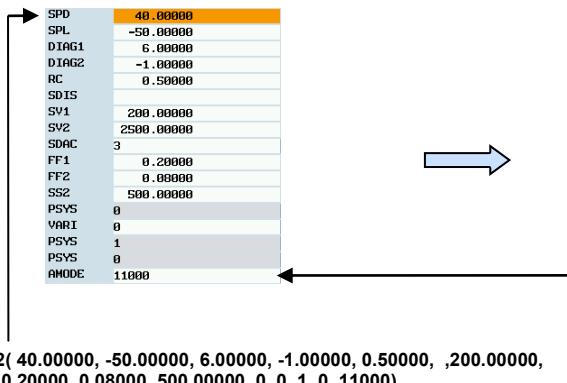
```

CYCLE92		Starting point along facing axis
SPL	40.00000	
SPL	-50.00000	
DIAG1	5.00000	
DIAG2	-1.00000	
RC	0.50000	
SDIS		
SV1	200.00000	
SV2	2500.00000	
SDMC	3	
FF1	0.20000	
FF2	0.08000	
SS2	500.00000	
PSYS	0	
VARI	0	
PSYS	1	
PSYS	0	
AMODE	11000	

With the "OK" SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.
The machine will cut off a part at the position specified in the cycle.

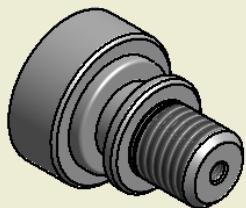
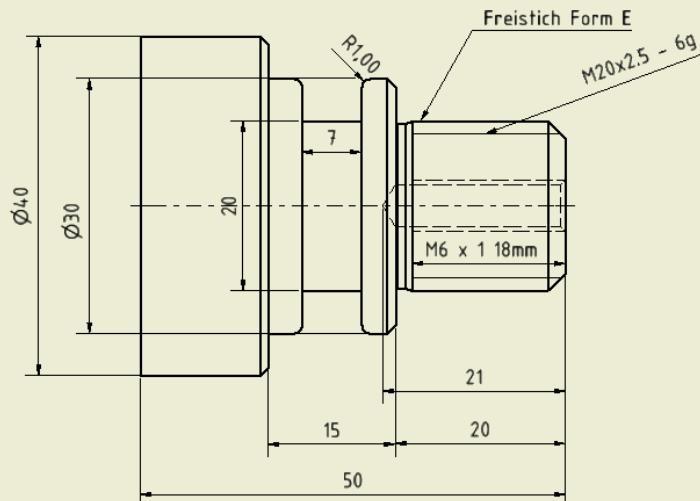


BASIC THEORY



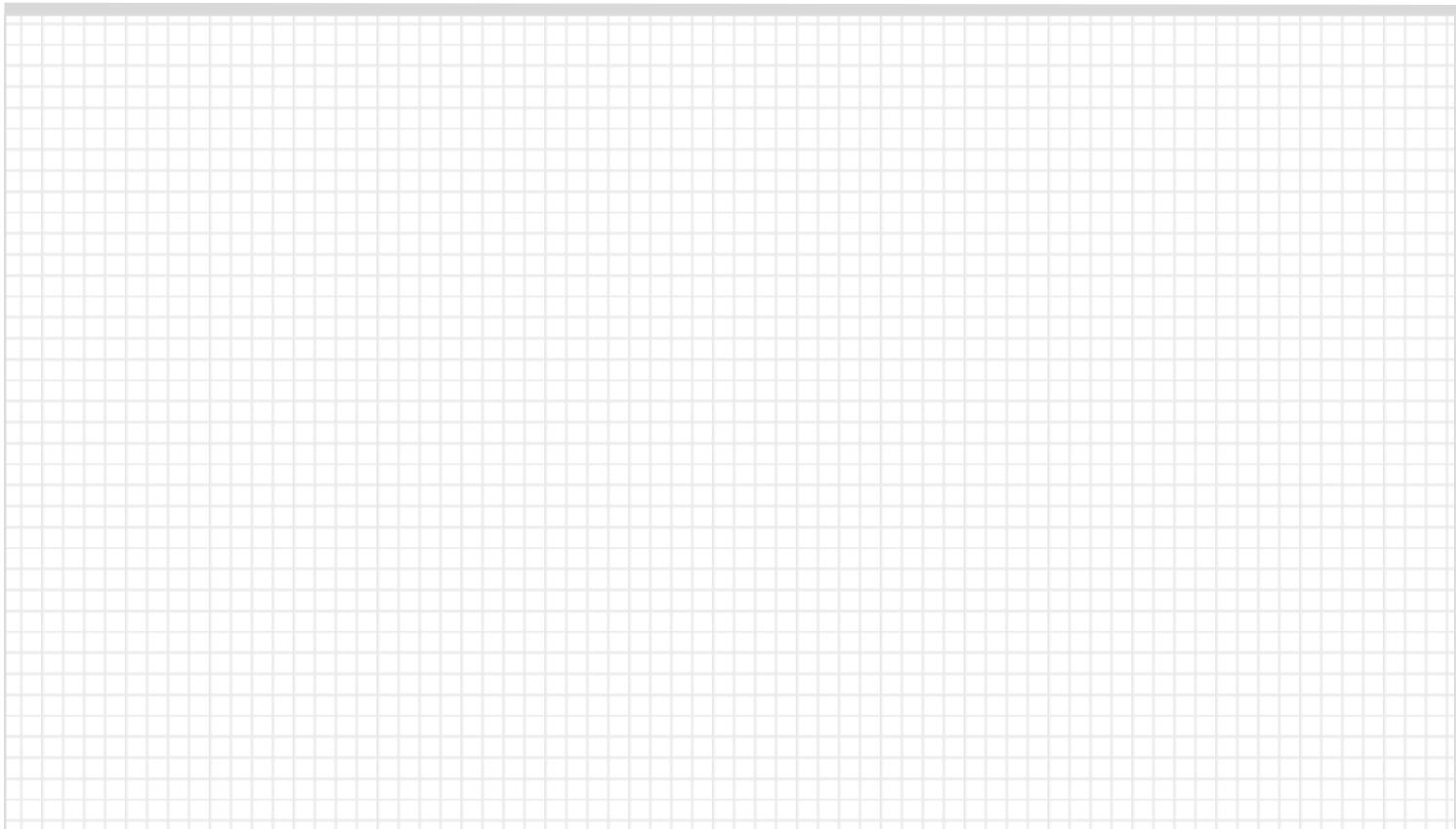
Parameters	Meanings	Remarks
DING1	The speed is reduced at depth of 6 mm	
DING2	When cutting off the final depth is -1 mm	
RC	Width of reverse angle is 0.5 mm	Or can be set as the radii of reverse circle
SV1	Fixed cutting speed is 200 mm/min	
SV2	Maximal spindle speed during fixed cutting is 2500 r/min	
SDAC=3	Spindle rotation direction is M3	SDAC=4 → spindle rotation direction M4
FF1=0.2	Depth feedrate when reaching the reduced speed (DING1)	
FF2=0.08	DING2 feedrate is 0.08 mm/min	
SS2=500	Reduced spindle speed (until final depth) is 500 r/min	
VARI=0	Retract to the position defined by SDP+SDIS	VARI=1 → no retraction
AMODE=11000	Machining shape is reverse angle	AMODE=10000 → reverse circle
For descriptions of SDIS, see page 50		
For descriptions of SPD and SPL, see page 47		

Notes



				Datum	Name
Erschafft		Kontrolliert		20.11.2011	adscre24
Horn					
		DEMO_PART_TURNING_1		1	
Status	Änderungen	Datum	Name		A4

Notes



Simulate Program

Content

Unit Description

This unit describes how to simulate a part program before executing it in AUTO mode.

Unit Content



Simulate program
(axis do not move)

End

Simulate program
(axis do not move)



A part program must have been created before it can be tested using "Simulation".

Step 1

The part program must be opened using the "Program Manager".

```
NC : \MPF\DEMO_PART.MPF
10 G00 G90 G95 G48 G71
N20 LIMS=4500
; ==Start Contour turning roughing ===
N30 T1 D1 ;ROUGH TURN
N40 G96 S250 M03 M08
N50 G00 X52.0 Z0.1
N60 G01 X-2.0 F0.35
N70 G00 Z22.0
N80 X52.0
CYCLE95( "DEMO_SUB_A", 2.50000, 0.20000, 0.10000, 0.15000, 0.35000,
0.20000, 0.15000, 9, , , ) ;
N90 G00 G40 X500.0 Z500.0
N100 M01
; ==Start Contour turning finishing ==
N110 T2 D1 ;FINISH TURN
N120 G96 S350 M03 M08
N130 G00 X22.0 Z0.0
N140 G01 X-2.0 F0.15
N150 G00 Z22.0
N160 X52.0
N170 CYCLE95( "DEMO_SUB_A", , , , , 0.15000, 9, , )
```

10:08:54
2013/04/25

Execute
Renumber
Search
Mark On
Copy
Paste

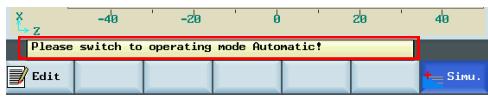
Edit Cont. Drill. Turn. Active Simu. Re-comp.



SEQUENCE

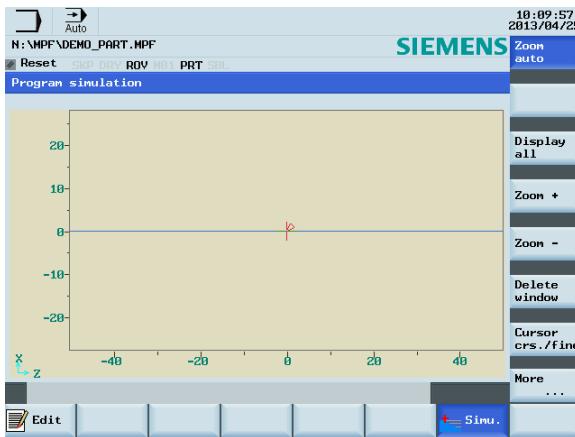
Step 2

Press the "Simu." SK on the PPU.



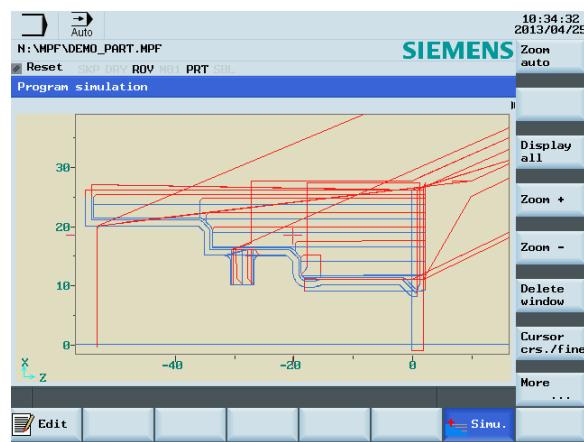
If the control is not in the correct mode, a message will be displayed at the bottom of the screen.

If this message is displayed at the bottom of the screen, press the "AUTO" mode key on the MCP.



Step 3

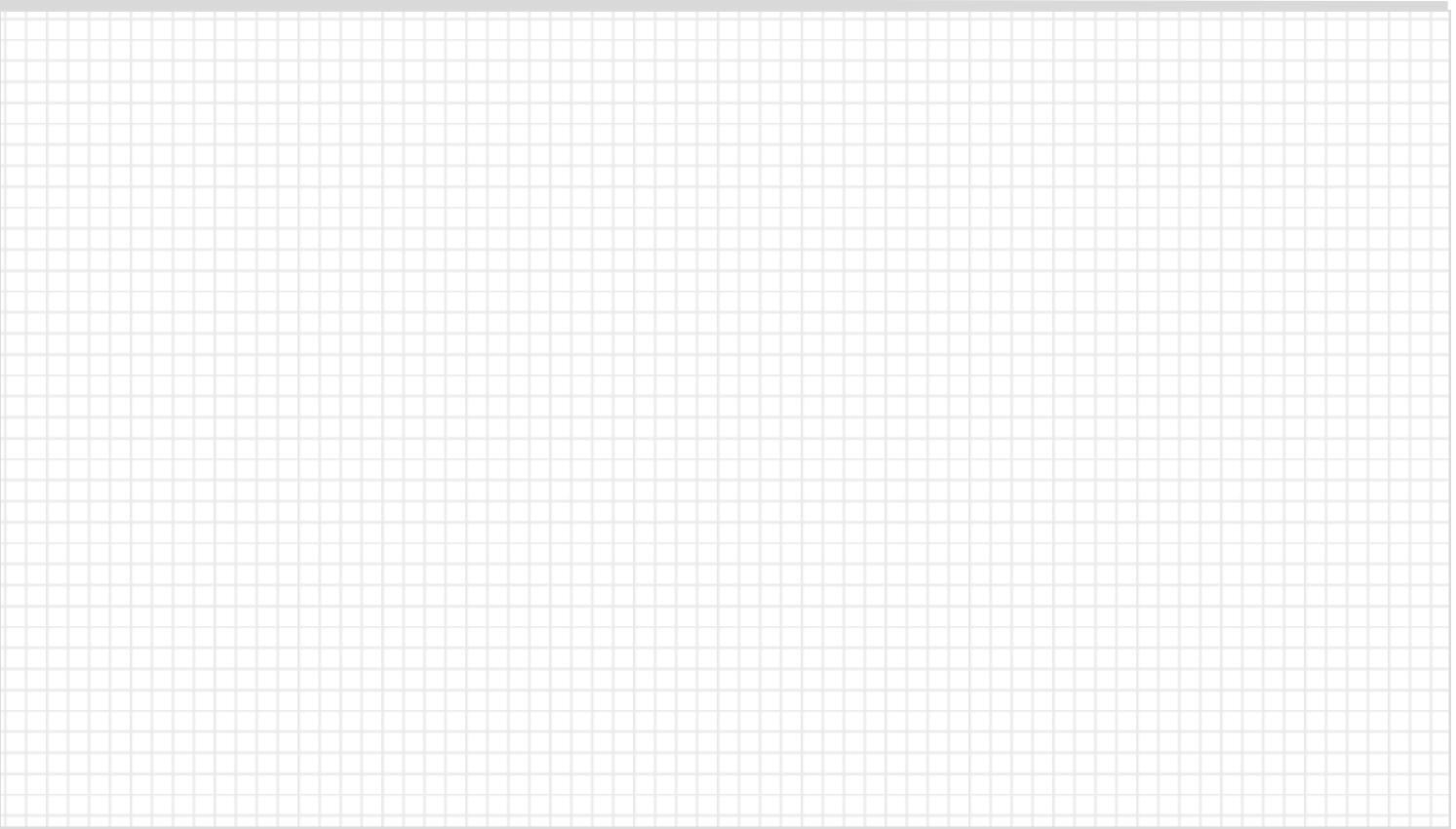
Press the "CYCLE START" key on the MCP.



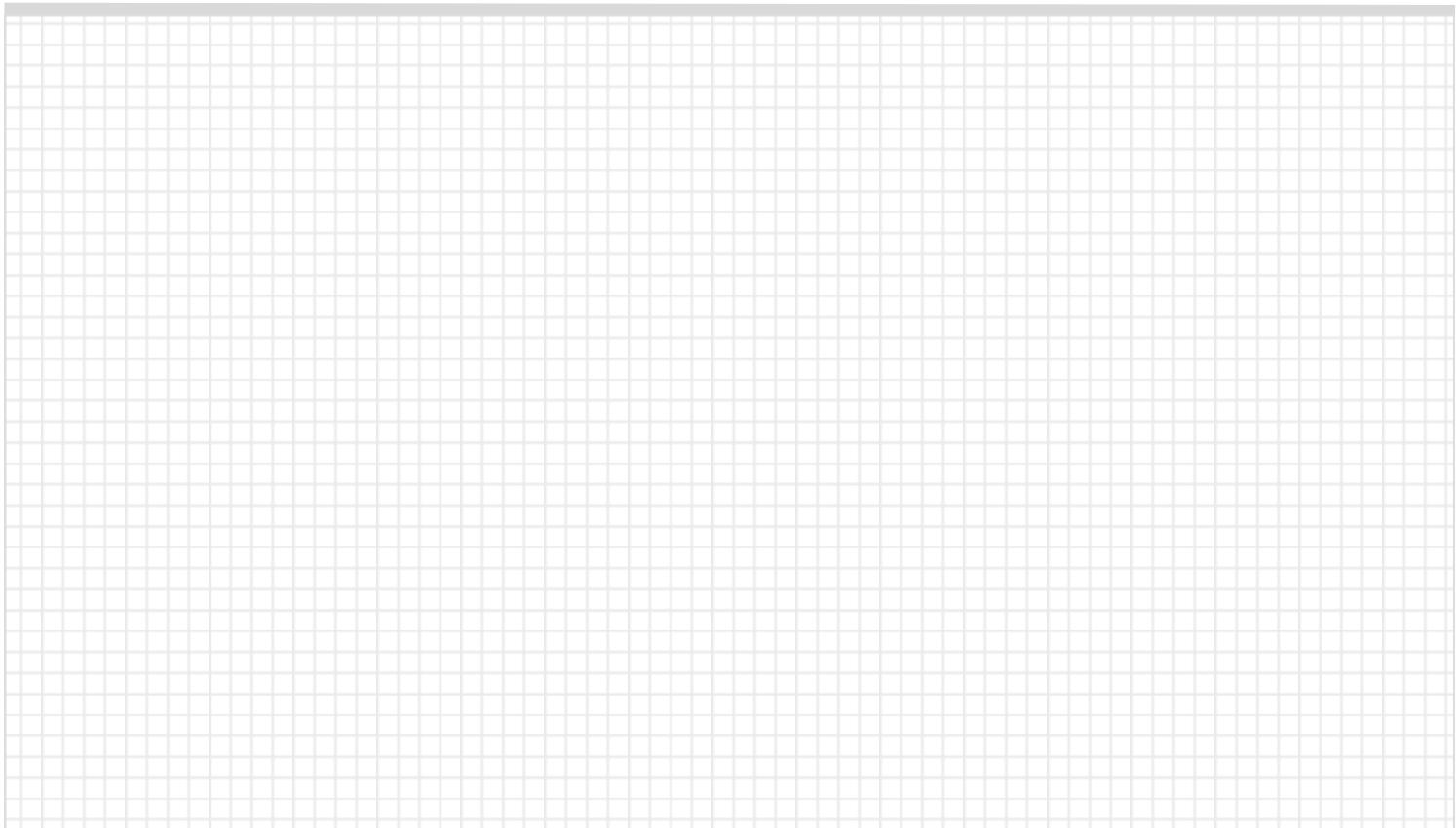
Press the "Edit" SK on the PPU to return to the program.



Notes



Notes



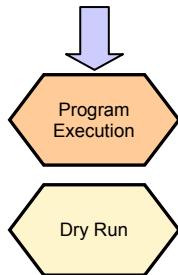


Content

Unit Description

This unit describes how to load the program in “AUTO” mode and test the part program at fixed speed.

Unit Content



End

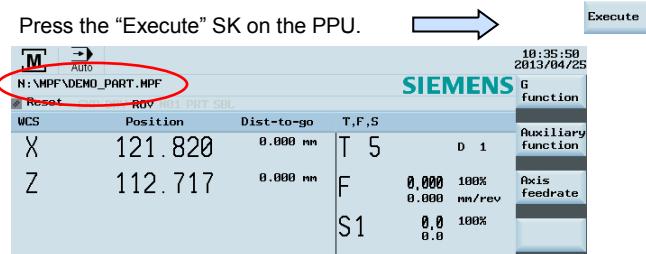
SEQUENCE



Before the part program can be loaded and executed in AUTO mode, it must be tested using the simulation function in “Edit”.



Press the “Execute” SK on the PPU.



The control is now in AUTO mode with the current opened program storage path being displayed and the AUTO lamp on the MCP is on.



Now the program is ready to start and the actual operation will be described in the next section!



SEQUENCE



Before executing the "Dry Run", please change the offset value appropriately for the real workpiece size in order to avoid cutting the real workpiece during the dry run and avoid unnecessary danger!

Note: The following operation is based on the finished "program execution".

Step 1



The data in the "Dry run feedrate" must first be set and checked!

Press the "Offset" key on the PPU.



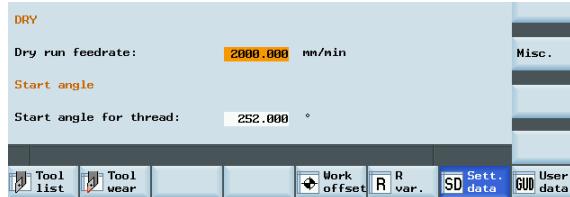
Press the "Sett. data" SK on the PPU.



Use the traversing key to move to the required position.
The position is now highlighted.



Enter the required feedrate in mm/min, enter "2000" in the example.



Press the "Input" key of the PPU.



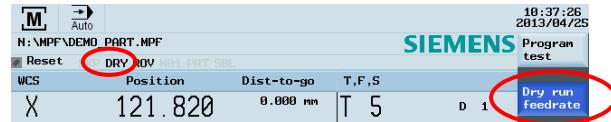
Press the "Machine" key on the PPU.



Press the "Prog. cont." SK on the PPU.



Press the "Dry run feedrate" SK on the PPU.



Note: The "DRY" symbol is shown and the "Dry run feedrate" SK is highlighted in blue.

Press the "Back" SK on the PPU.



Step 2



Make sure the feedrate override on the MCP is 0%.

Press "Door" on the MCP to close the door of the machine. (If you don't use this function, just close the door in the machine manually.)



Press "CYCLE START" on the MCP to execute the program.



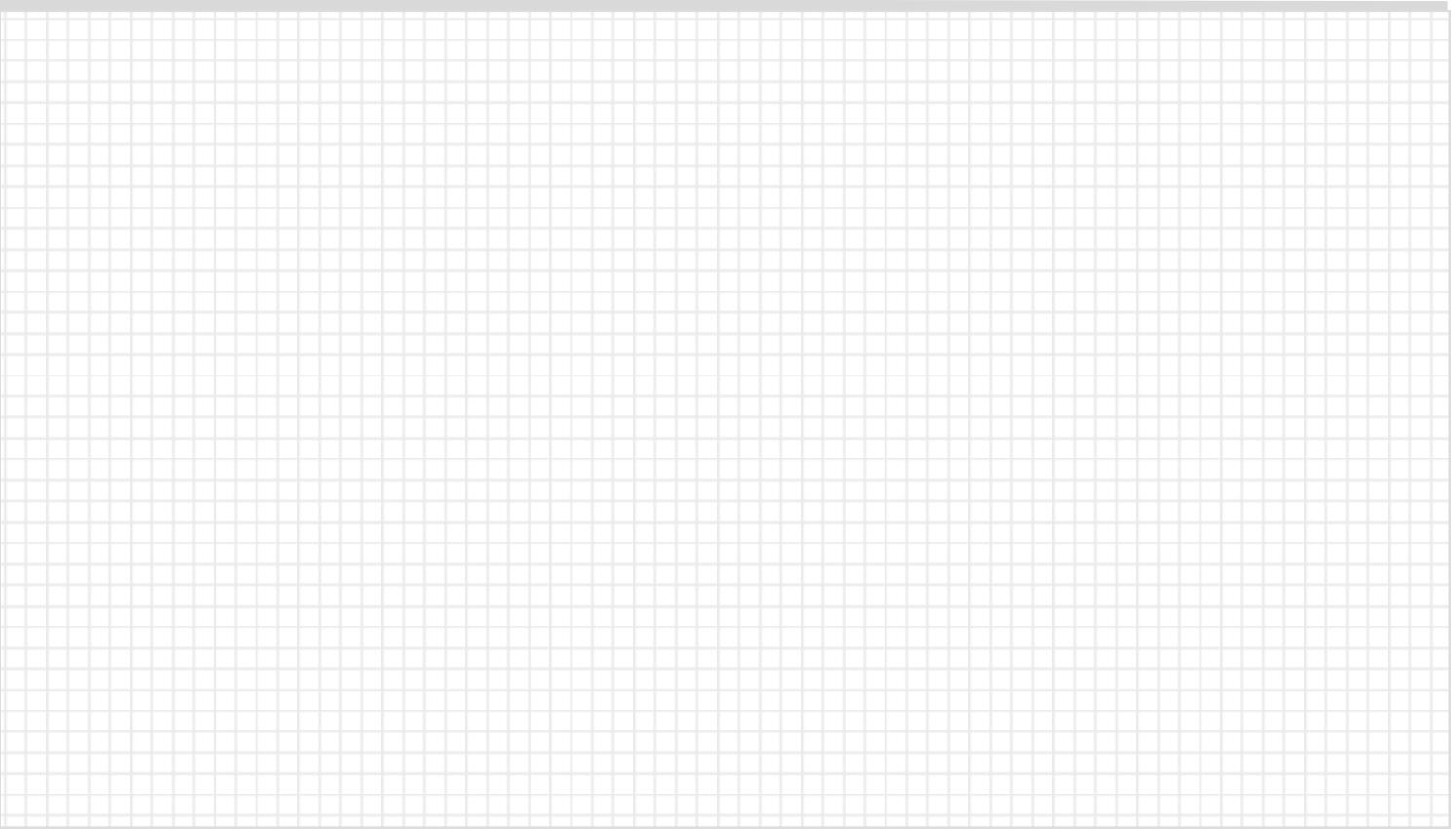
Turn the feedrate override gradually to the required value.



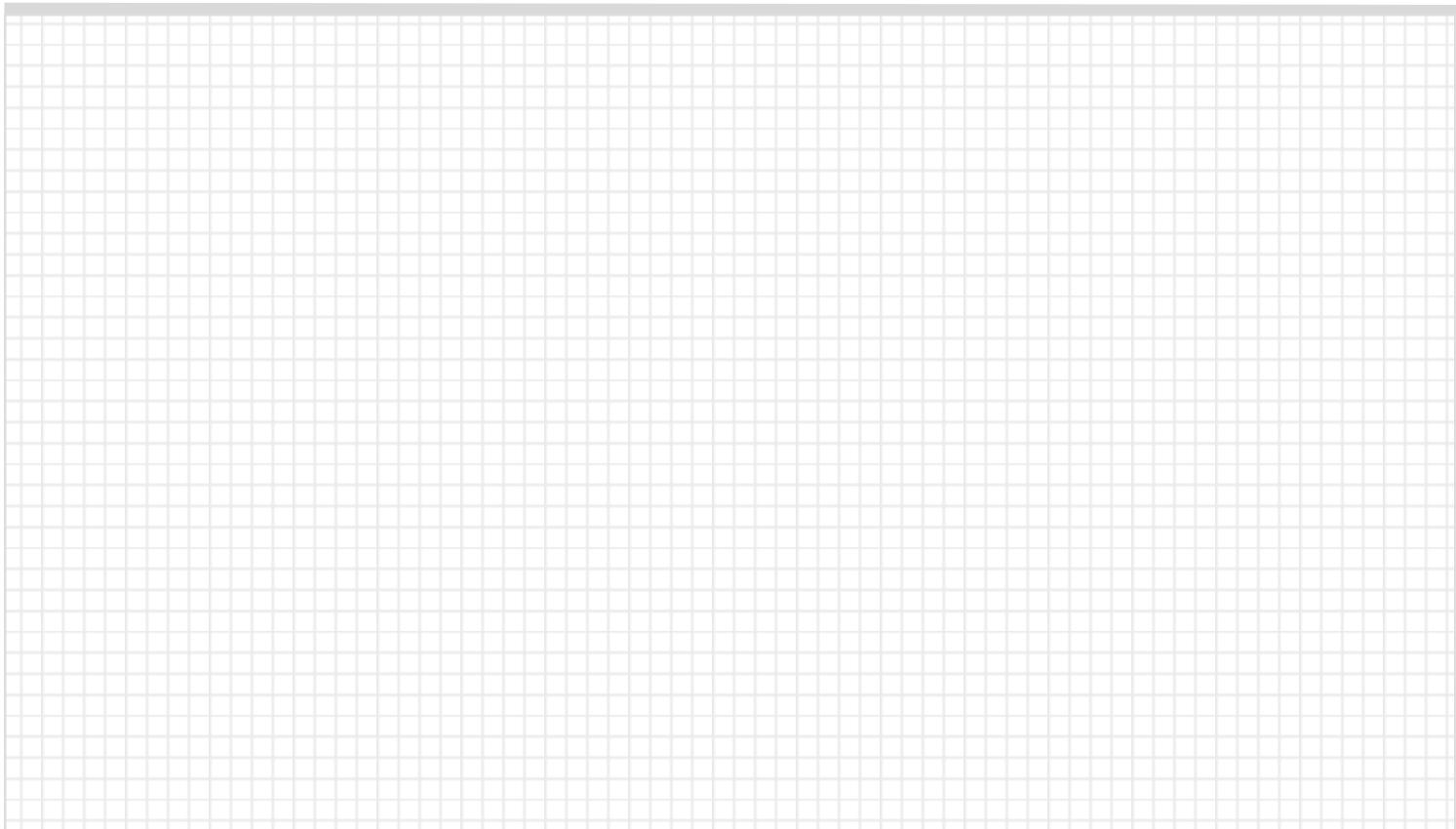
After finishing the dry run, please turn the changed offset back to the original value in order to avoid affecting the actual machining!



Notes



Notes



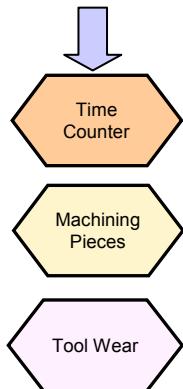


Content

Unit Description

This unit describes how to use the “Time, counter” function and how to machine pieces and the compensation setting for the tool wear.

Unit Content



End

SIEMENS

SEQUENCE



Make sure the machine has been referenced before machining work-pieces!

Step 1

Press the “Machine” key on the PPU.



Press the “Auto” key on the MCP.



Press the “Time counter” SK on the PPU.



Block display	DEMO_PART.MPF	Time, counter	counter
	M10 G00 G90 G95 G40 G71 1 N20 LIMs=4500 1 ; ==Start Contour turning roughing === 1	Cycle time 0000:00:00h Time left 0000:06:59h	Act. val. REL
	N30 T1 D1 ;ROUGH TURN 1 M40 G96 S250 M03 M08 1 N50 G00 X52.0 Z0.1 1 N60 G01 X-2.0 F0.35 1	Counter No 0	Act. val. Work(MCS)
			Act. val. Mach(MCS)



Machine Pieces

SEQUENCE

"Cycle time" shows how long the program has been running.

Cycle time 0000:00:00h

"Remaining time" shows how much time remains before the program ends.

Time left 0000:00:00h



The "Remaining time" can only be counted after a successful cycle run of a part program!

Select "Yes" or "No" to decide whether to activate the counter (press the "Select" key to activate the choice).

Enter the number of workpieces you require to be machined in "Required".

Required 45

"Actual" shows the number of work-pieces that have been machined.

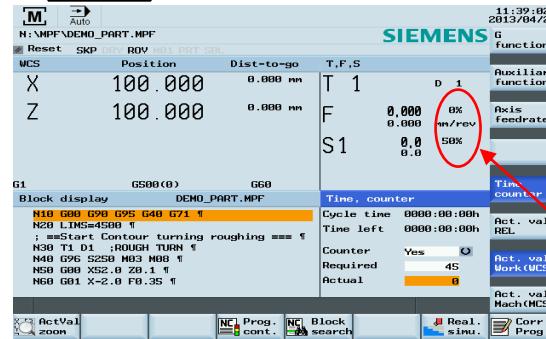
Actual 8

Block display	DEMO_PART.MPF	Time, counter	counter
N10 G00 G90 G95 G40 G71 1	Cycle time 0000:00:00h		
N20 LIMs=4500 1	Time left 0000:00:00h		Act. val. REL
; ==Start Contour turning roughing == 1			
N30 T1 D1 ;ROUGH TURN 1			
N40 G96 S250 M03 M08 1			
N50 G00 X52.0 Z0.1 1			
N60 G01 X-2.0 F0.35 1			

Machine Pieces



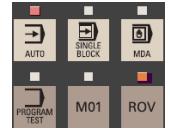
Make sure the program is correct before machining pieces!



Set the program in the ready-to-start status as shown on the left in accordance with the "Program execution" sequences.

Perform the relevant safety precautions!

Make sure that only "AUTO" mode and "ROV" mode are active.

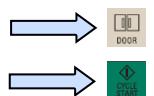


Note: M01 function → program will stop at the position where there is M01 code.

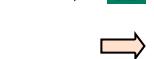


Make sure that the feedrate override on the MCP is 0%!

Press "Door" on the MCP to close the door of the machine. (If you don't use this function, just close the door on the machine manually.)



Press "CYCLE START" on the MCP to execute the program.



Turn the feedrate override gradually to the required value.

Machine Pieces

SEQUENCE

Tool
Wear



The tool wear compensation must distinguish the direction of compensation clearly!

Step 1

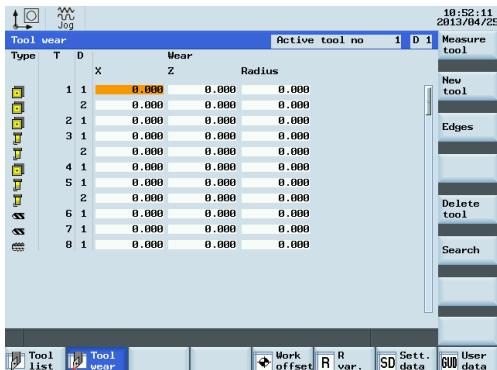
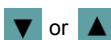
Press the "Offset" key on the PPU.



Press the "Tool wear" SK on the PPU.



Use the direction keys to select the required tools and their edges.



Step 2

Set the tool length wear parameter of axis X in "Length X", the sign determines the direction of wear compensation.

Set the tool length wear parameter of axis Z in "Length Z", the sign determines the direction of wear compensation.

Positive value: The tool moves away from the workpiece
Negative value: The tool moves closer to the workpiece

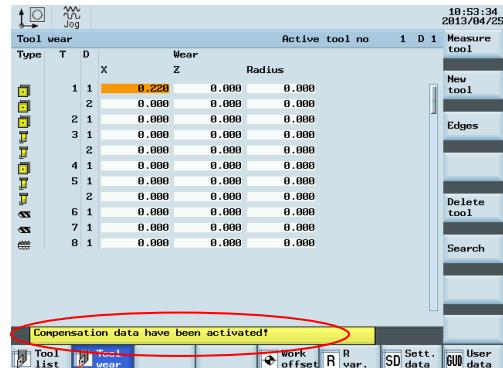
Press "Input" on the PPU to activate the compensation →

Set the tool radius wear parameter in "Radius", the sign determines the direction of wear compensation.

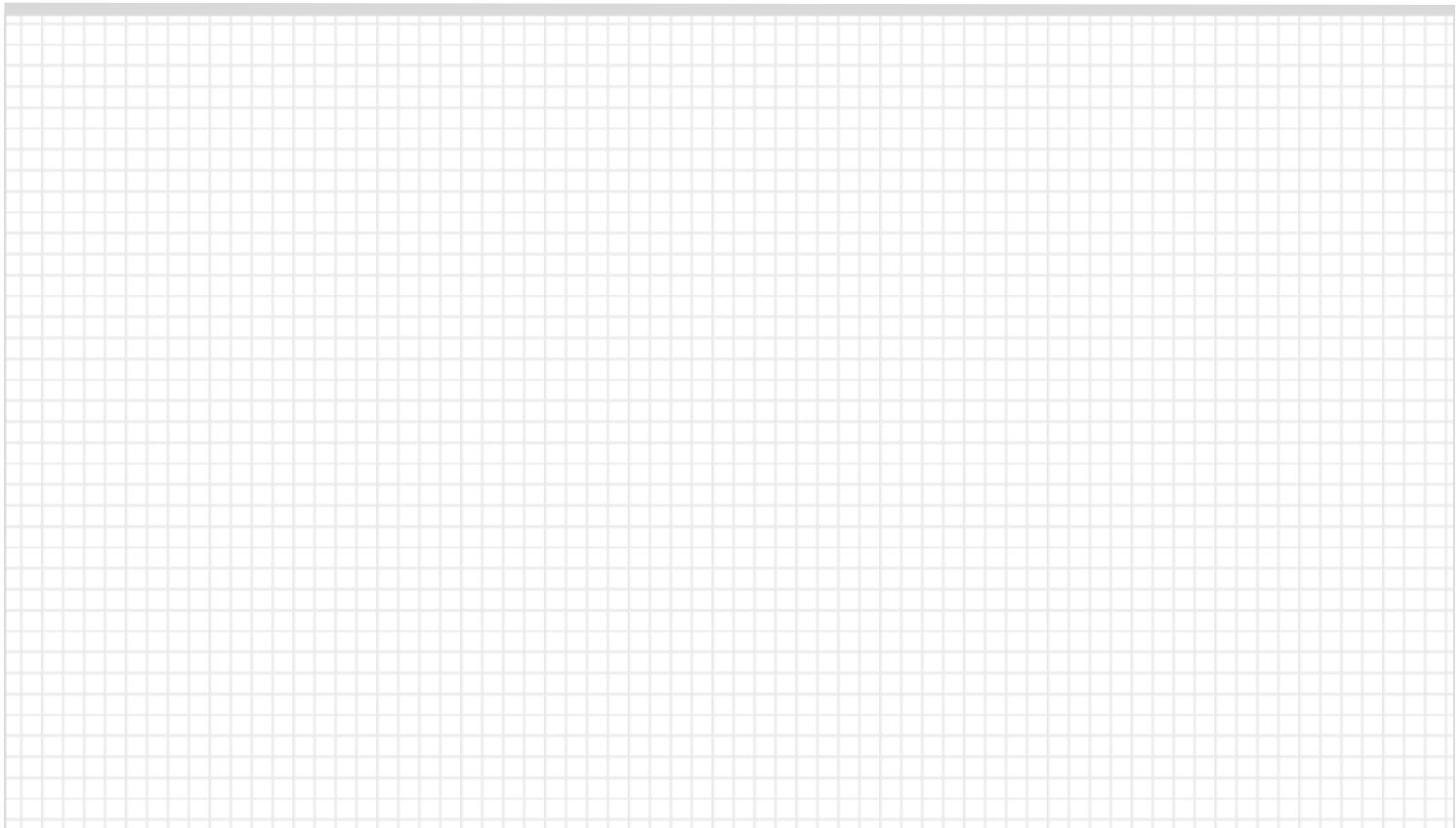
Positive value: The tool moves away from the workpiece (set radius is larger than the actual radius)

Negative value: The tool moves closer to workpiece (set radius is smaller than the actual radius)

Press "Input" on the PPU to activate the compensation →



Notes



Program Restart

Content

Unit Description

This unit describes how to restart the part program after a tool has been changed due to damage, or re-machining has to be performed.

Unit Content



Block search

End

SEQUENCE

Block search

Press the “Machine” key on the PPU.



Press the “Auto” key on the MCP.



Press the “Block search” SK on the PPU



Press the “Interr. point” SK on the PPU
and the cursor will move to the last
interrupted program line.



Note: The cursor can be moved to the required program block
with the traversing keys.

```

MACHINE
Auto
SIEMENS
N:\AMPF\DEMO_PART.MPF
N10 G00 G90 G95 G40 G71
N20 L1M5G45000
; ==Start Contour turning roughing ===
N30 T1 D1 _ROUGH TURN
N40 G96 S250 M03 M08
N50 G00 X52.0 Z0.1
N60 G01 X-2.0 F0.35
N70 G00 Z2.0
N80 G02 Z2.0
CYCLEPS("DEMO_SUB_A", 2.50000, 0.20000, 0.10000, 0.15000, 0.35000,
0.20000, 0.15000, 0.10000, 0.05000)
N90 G00 G40 X50.0 Z50.0
N100 M01
; ==Start Contour turning finishing ==
N110 T2 D1 _FINISH TURN
N120 G96 S350 M03 M08
N130 G00 X22.0 Z0.0
N140 G01 X-2.0 F0.15
N150 G00 Z2.0

```

Note: The “To contour” and “To end point” functions.

“To contour”: The program will continue from the line before the breakpoint.

“To end point”: The program will continue from the line with the breakpoint.

Press the “To end point” SK on the PPU
(or select “To contour” as required).





SIEMENS

SEQUENCE

MCS Position Dist-to-go T,F,S
X 499.780 0.000 mm T 1 D 1
Z 500.000 0.000 mm F 0.000 0%
S1 8.0 50%

G1 GS00(0) G60
Block display Current program :DEMO_PART.MPF
: :=Start Contour turning finishing = 1
N110 T2 D1 ;FINISH TURN 1
N120 G96 S350 M03 M08 1
N130 G00 X22.0 Z0.0 1
N140 G01 X-2.0 F0.15 1
N150 G00 Z22.0 1
N160 X52.0 1

ActVal zoom NC Prog. cont. NC Block search Real. simu. Corr. Prog.



The feedrate override must always be set to 0%!
Make sure the correct tool is selected before continuing!

Press the “CYCLE START” key on the → MCP to execute the program.



Turn the feed rate override on the MCP gradually to the required value.

MCS Position Dist-to-go T,F,S
- X 409.015 -387.016 mm T 2 D 1
- Z 404.828 -404.828 mm F 3325.110 300%
S1 500.0 100%

G0 GS00(0) G60
Block display Current program :DEMO_PART.MPF
N110 G96 S350 M03 M08 1
N130 G00 X22.0 Z0.0 1
N140 G01 X-2.0 F0.15 1
N150 G00 Z22.0 1
N160 X52.0 1
N170 CYCLE95("DEMO_SUB_A", , , , , 0.15000, S, , ,) 1
N180 G00 G40 XS00.0 ZS00.0 1

ActVal zoom NC Prog. cont. NC Block search Real. simu. Corr. Prog.

Press the “CYCLE START” key on the MCP to execute the program.

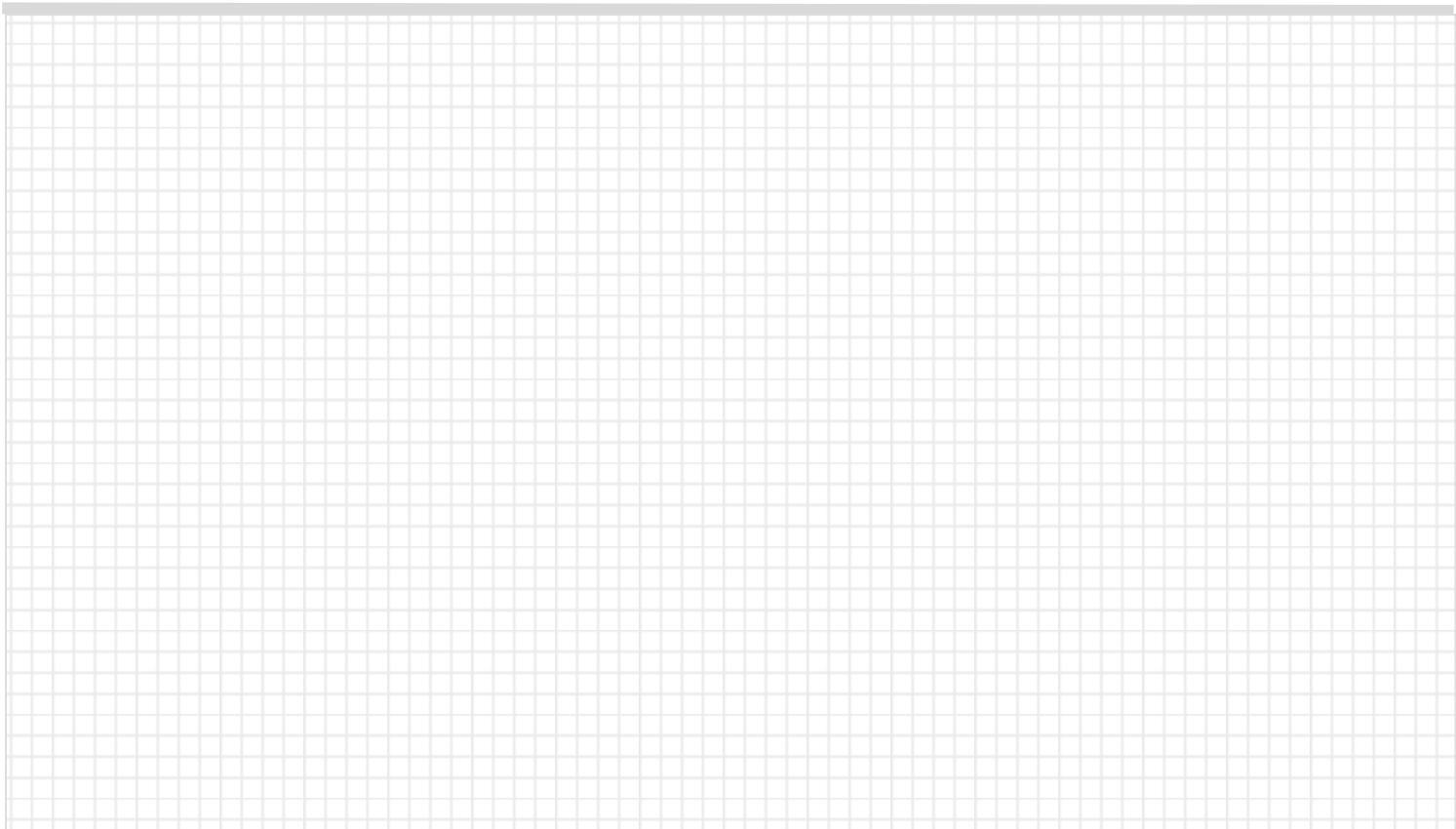


M 010208 Channel 1: Continue program with MC start 10:57
N:\MPP\DEMO_PART.MPF
Stop SKP DRY RDY H01 PRT SEL
SIEMENS G function

Alarm 010208 is shown at the top prompting to press the “CYCLE START” key to continue the program.



Notes

A large rectangular area filled with a uniform grid of light gray lines, creating a pattern of small squares. This grid covers most of the page below the 'Notes' header, intended for handwritten notes.

Notes

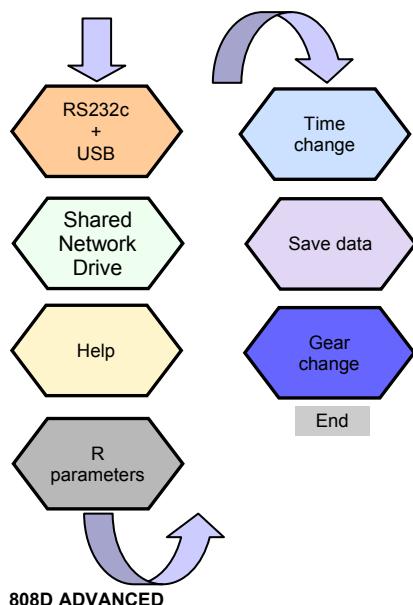


Content

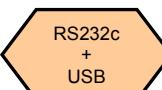
Unit Description

This unit describes how to perform simple tasks on the machine and provides some additional information which may be required to operate the machine correctly.

Unit Content



SEQUENCE



RS232c is used to transfer the programs to and from the NC.

Step 1 It is recommended to use the "Sinucom PCIN" communication SW provided by Siemens to transfer the standard program.

Adjust the parameter settings on the PPU to match the settings of the communication SW on PC.

Press "Program Manager" on the PPU.

Press the "RS232" SK on the PPU.

Press the "Settings" SK on the PPU.

Adjust the parameters in "Communication settings" to match the settings of communication SW on PC.

Communications settings	
Device	RTS CTS
Baud rate	19200
Stop bits	1
Parity	None
Data bits	8
End of transm.	1a
Confirm overwrite	No

Press the "Save" SK on the PPU.

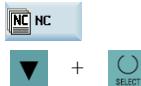
Press the "Back" SK on the PPU.



SEQUENCE

Step 2 Transfer a part program to a PC from the PPU.

Press the "NC" SK on the PPU.



Use "Cursor + Select" to select the required part program. The selected program will be highlighted.



Press the "Copy" SK on the PPU.



Press the "RS232" SK on the PPU.



Check the interface setting and start the communication software to receive the program from PC.

(Press "Receive Data" on SINUCOM PCIN to start the receive function.)

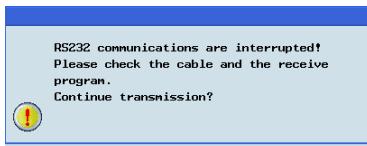
Press the "Send" SK on the PPU.



The PPU will display a window showing the progress of the transfer.



If there is a problem during transfer of the part program, a window will be displayed.



You can continue sending the part program.

Press the "OK" SK on the PPU.



Or you can abort the sending of the part program.

Press the "Cancel" SK on the PPU.



Step 3 Transfer a part program to the PPU from a PC.

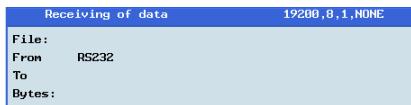
Press "Program Manager" on the PPU.



Press the "RS232" SK on the PPU.



Press the "Receive" SK on the PPU.



Check the interface setting and start the communication software to send the program from PC.

(Press "Send Data" on SINUCOM PCIN to send data.)

The PPU will display a window showing the progress of the transfer.



SEQUENCE



“USB” is used to transfer the programs to and from the NC.

- Step 4 Use the “Copy” “Paste” SKs to transfer the part program from NC to USB.

Connect a USB device with sufficient memory to the USB interface on the PPU.

Press the “NC” SK on the PPU.



Use “Cursor + Select” to select the required part program. The selected program will be highlighted



Press the “Copy” SK on the PPU.



Press the “USB” SK on the PPU.



Press the “Paste” SK on the PPU.



- Step 5 Use the “Copy” and “Paste” SKs to transfer the part program from USB to NC.

Connect the USB device with the stored target programs to the USB interface on the PPU.

Press the “USB” SK on the PPU.



Use “Cursor + Select” to select the required part program. The selected program will be highlighted



Press the “Copy” SK on the PPU.



Press the “NC” SK on the PPU.



Press the “Paste” SK on the PPU.



SEQUENCE



A shared network drive can be made using an ethernet connection between the PC and the PPU so the transferring and backup of NC programs can be performed easier.

Step 1 Set PPU IP address.

Connect PC using a network cable to the rear X130 ethernet port on the PPU

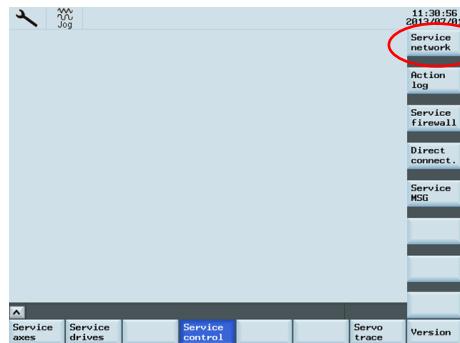
Press key: +

Press key:

Press "Serv. Dispil." SK



Press "Service control" SK

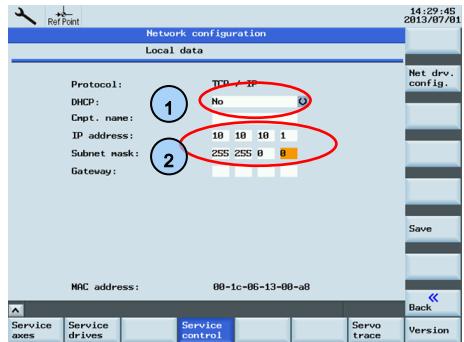


Press "Network Info" button to enter the "Local Configuration Data"



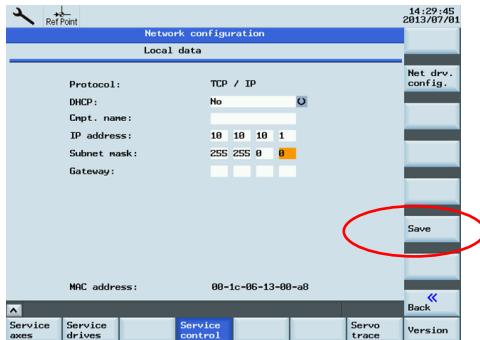
In the "local configuration data" enter the relevant parameters.

- ① DHCP is set to "No"
- ② IP address and subnet mask can be set according to requirements.
(screenshot right given only as an example)



"Local Configuration Data" setting finished, press the "Save" button to activate the data set.

When the "data storage end" is displayed, the input data activation effect.



SEQUENCE

Step 2 Set the PC's static IP address.

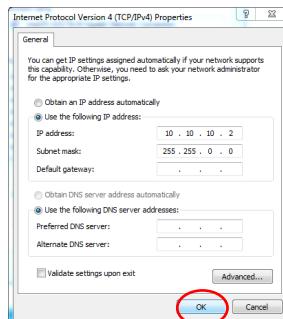
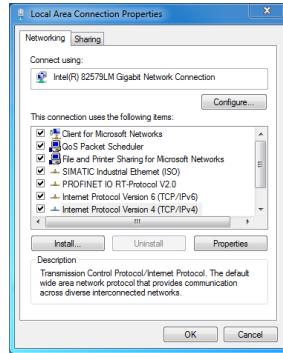
Ensure PC/PG is connected using a network cable to rear X130 PPU Ethernet port.

Open the PC's network connection settings, in the "local area connection properties" select "Internet Protocol (TCP / IP)" And double click "Properties".

In the dialog box, select "Use the following IP address" and fill in the required IP address.
(Shown right only given as an example)
Select "OK" to complete the setup.

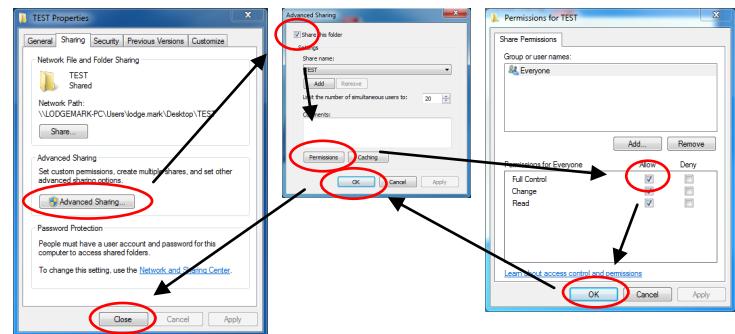
Note: The address "10.10.10.2" setting is based on the first step in the IP address of the PPU.

PPU and PC IP address should be kept in the same network segment.



Step 3 On PC create a shared folder.

Anywhere on your PC create a new folder with a simple name (do not use special characters). This example creates a folder named "Test". Once created, right-click the folder and select "Properties." then select the pull down menu "Sharing".



In the dialog window, select "Advanced Sharing"
Then check "Share this folder"
Then select "Permissions" and check "Full control".
Select "OK" - "OK" - "Close" to activate the settings.
In this folder you can put some machining program.

Step 4 Add the network drive on the PPU side to activate the shared folder, and online processing



In the "Network drive configuration" screen select "Net drv. Config."



Additional Information Part 1

SIEMENS

SEQUENCE

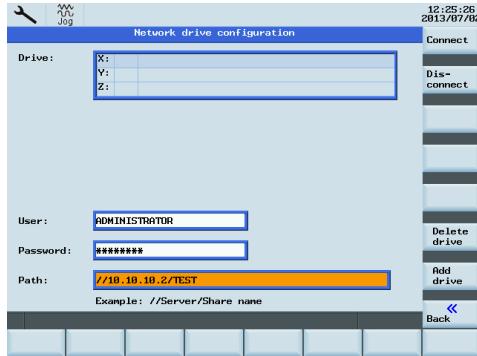
In the “Network Drive Configuration” enter PC login user name, password, and path of where shared folder is. In accordance to the format required.

Server: IP address
Share Name: the name of the shared folder

Note: Use “TAB” key to switching between different tasks boxes.

Add drive Press “Add Drive” SK to add it to the specified drive letter

After set successful, the screen will displayed “Network drive added successfully” while the set path is automatically written to the “drive” Window.

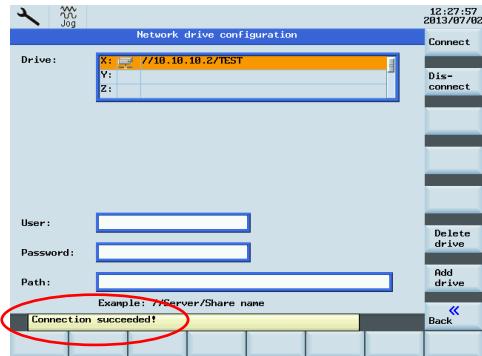


If the connection is lost select the drive path and press “Connect.” SK

Connect

This will re-establish the connection with PC/PG.

This will be shown with the text “Connection succeeded”

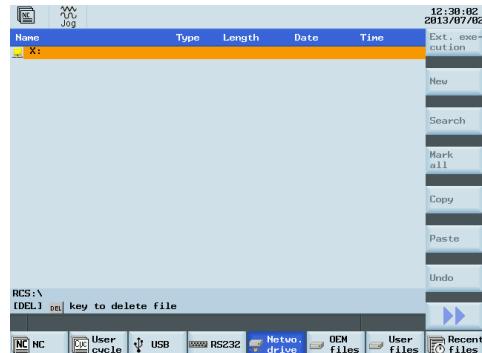


Press “Program Manager” Button

Press “netwo. drive” SK to enter the network drive interface.



Press “INPUT” Button to open network drive to PC/PG.



Additional Information Part 1

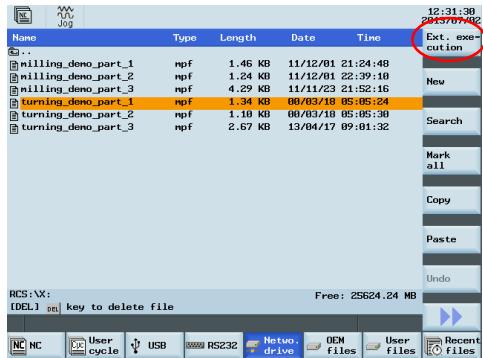
SIEMENS

SEQUENCE

You can now see the content of the shared folder with all the NC programs.

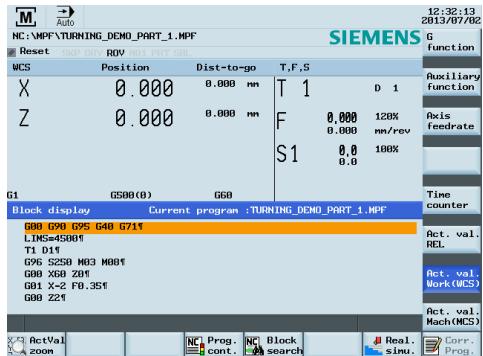
You can select the file you require to execute in AUTO mode, click "Exe. Execution".

Ext. exe-
cution



The system will automatically jump to AUTO mode, select the appropriate NC program.
Press the "Cycle Start" button for machining operation.

CYCLE
START



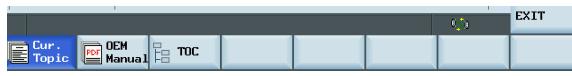
Note: You can also use the "Copy", "Paste" key to achieve "NC", "USB" and "Network Drive" moving files.



Help

The PPU has an online help which shows the contents of standard documents.

Press the "Help" key on the PPU.



Press the "Cur. Topic" SK on the PPU.



The help information related to the current topic will be shown on screen.

Press the "OEM Manual" SK on the PPU.



The online help manual of the OEM will be shown on the screen.

Press the "TOC" SK on the PPU.



The online help from the Siemens manual will be shown.

SEQUENCE

R parameters

The arithmetic parameters are used in a part program for value assignment, and also for some necessary value calculations. The required values can be set or calculated by the control system during program execution. Some of the common arithmetic functions are shown below:

Arithmetic parameters	Meaning
+	Addition
-	Subtraction
*	Multiplication
/	Division
=	Equals
Sin()	Sine
COS()	Cosine
TAN()	Tangent
ASIN()	Arcsine
ACOS()	Arccosine
ATAN2(,)	Arctangent2
SQRT()	Square root
ABS()	Absolute value

Note:

Preprocessing stop

Programming the STOPRE command in a block will stop block preprocessing and buffering. The following block is not executed until all preprocessed and saved blocks have been executed in full. The preceding block is stopped in exact stop (as with G9).

The following program shows the interaction of the part program and the R variables screen.

Press the "Offset" key on the PPU.



Press the "R var." SK on the PPU.



N10 G18 G90 G54

N20 T1 D1

N30 S2500 M03 M08

N40 G00 X-10.0 Z10

N50 R1=0 R2=0 R3=0

N60 STOPRE

N70 M00

N80 R1=1

N90 STOPRE

N100 M00

N110 R2=2

N120 STOPRE

N130 M00

N140 R3=R1+R2

N150 STOPRE

N160 G00 X=R3

N170 M30

WCS	Position	Dist-to-go
X	3.000	0.000 mm
Z	10.000	0.000 mm



WCS	Position	Dist-to-go
X	-10.000	0.000 mm
Z	10.000	0.000 mm

R variables
R0
R1
R2
R3
R4
R5

R variables
R0
R1
R2
R3
R4
R5

R variables
R0
R1
R2
R3
R4
R5

R variables
R0
R1
R2
R3
R4
R5



Additional Information Part 1

SIEMENS

SEQUENCE

Time change

You can change the time on the control if required when the clocks changes from summer time to winter time.

Press "Shift" and "Alarm" on the PPU simultaneously.



Make sure the password is set to the "CUSTOMER" access level.

Press the "HMI" SK on the PPU.



Press the "Date time" SK on the PPU.



Date and Time		
Date and Time setting		
Current	2011/10/08	05:08:42
Format	YYYY/MM/DD	HH:MM:SS
New	0000/00/00	00 :00 :00

Enter a new "Date" and "Time"

Date and Time		
Date and Time setting		
Current	2011/10/08	05:09:49
Format	YYYY/MM/DD	HH:MM:SS
New	2012 /02 /22	12 :37 :42

Press the "OK" SK on the PPU to up date



Date and Time		
Date and Time setting		
Current	2012/02/22	12:37:49
Format	YYYY/MM/DD	HH:MM:SS
New	2012 /02 /22	12 :37 :42

Press the "Cancel" SK on the PPU to abort the operation.



Save data

"Save data" enables the complete system to be backed up on the system CF card so that there is a system backup available to the operator.

Press "Shift" and "Alarm" on the PPU simultaneously.

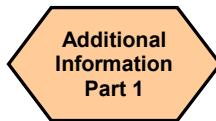


Make sure the password is set to the "CUSTOMER" access level.

Press the "Save data" SK on the PPU.

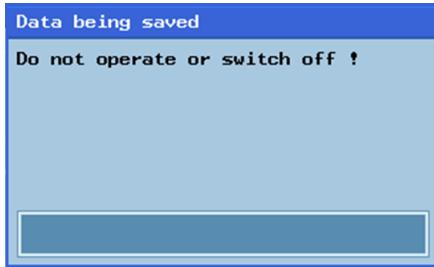


Machine configuration				
No.	Axis index	Name	Axis type	Drive number
Message				
Do you want to save the data ? Yes ==> then 'OK' --- No ==> Press 'Cancel' NOTICE !! This function is only available in NCK_RESET Saving comprises data areas: - Machine and setting data - Tool offset data list - R variables - Monitoring and runtime data - Work offsets - Workpiece programs and cycles - GUDs (global user data)				



SEQUENCE

Press the "OK" SK on the PPU.



While the control is saving data to the system, do not operate or switch off the control!



When a machine has a manual gearbox on the spindle, it is the responsibility of the operator to change gear at the correct place in the part program.

If the machine tool manufacturer has fitted an automatic gearbox, the following M-codes can be used to change gear in the part program.

Gear stages M40, M41, M42, M43, M44 and M45 are available.

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

Example:

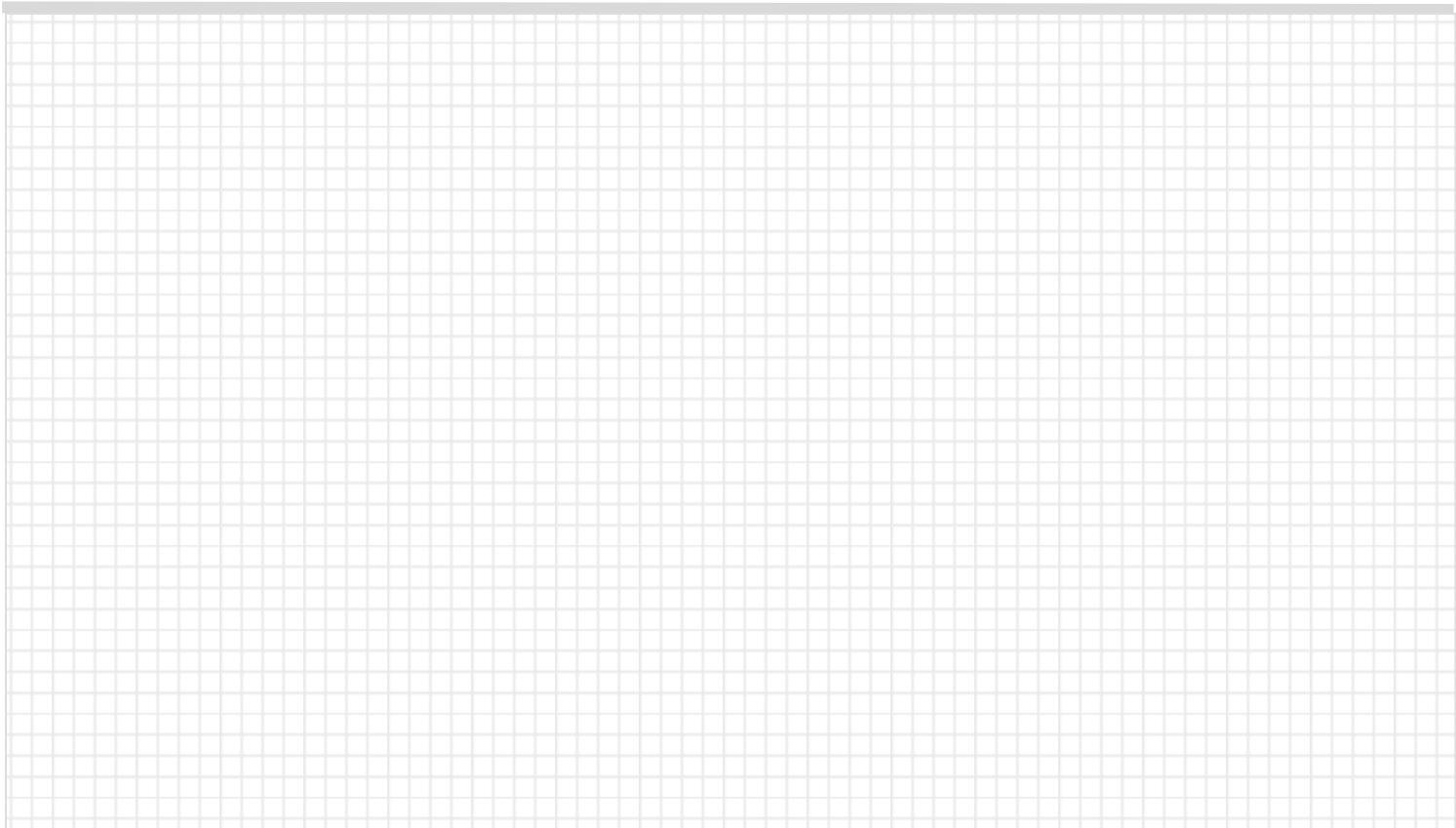
The machine tool manufacturer specifies a speed range for each gear stage:

S0...500	Gear stage 1 → M41
S400..1200	Gear stage 2 → M42
S1000..2000	Gear stage 3 → M43

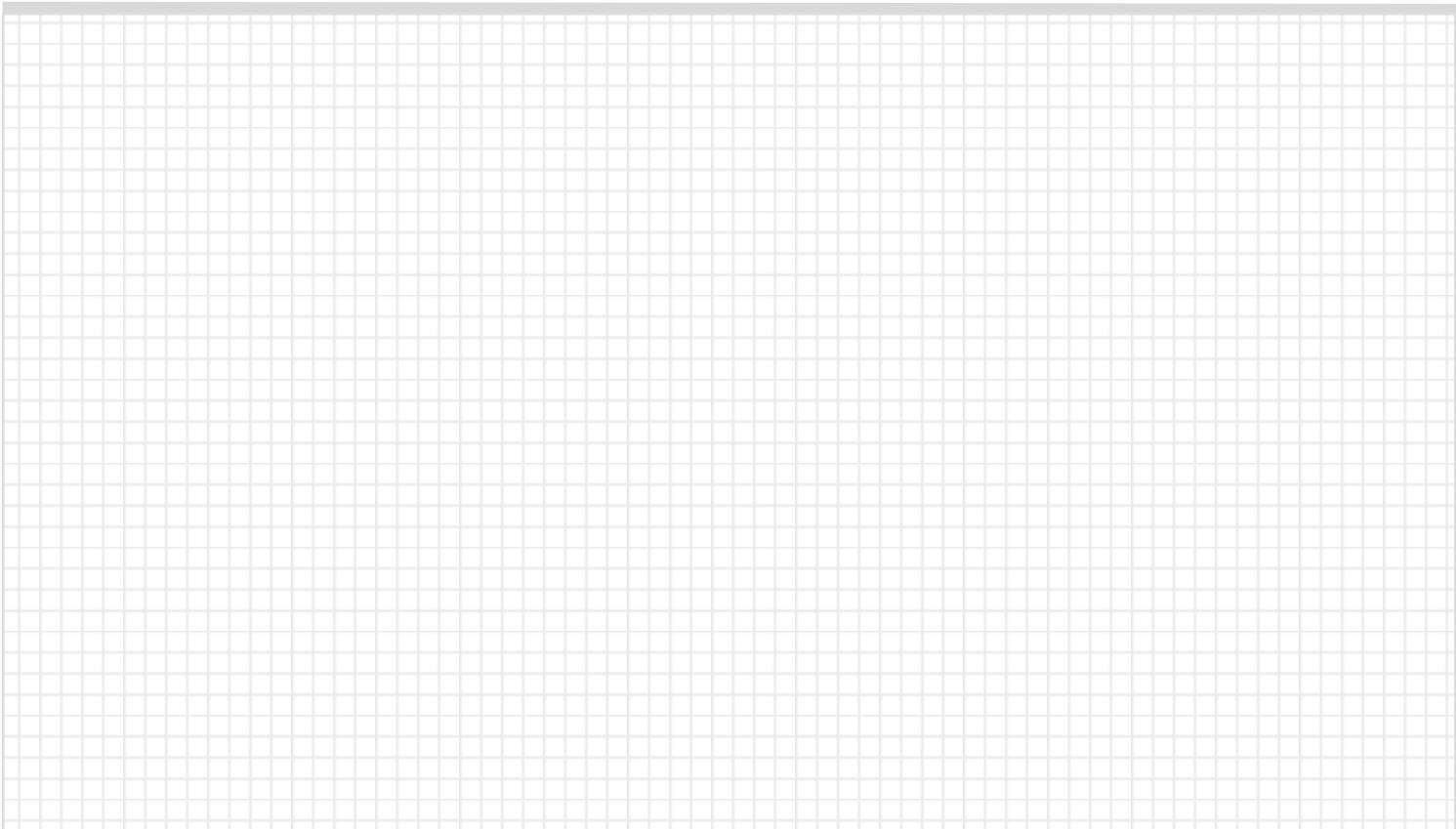
If the operator is manually selecting the gear stage in the part program, it is the operator's responsibility to select the correct gear stage according to the required speed.



Notes



Notes



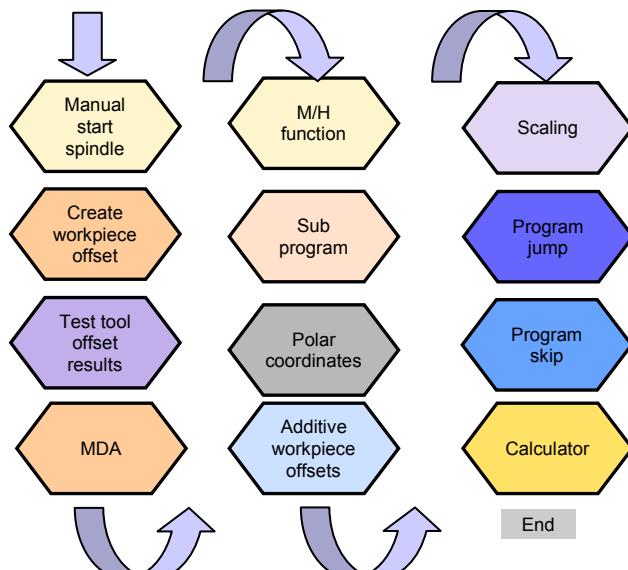
Content

Unit Description

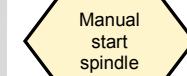
This unit describes how to perform simple tasks on the machine and provides some additional information which may be required to operate the machine correctly.

Part 2

Unit Content



SEQUENCE



A tool must be loaded and rotated to the position.

Before measuring, the spindle can be started as follows:

Press the "Machine" key on the PPU.



Press the "JOG" key on the MCP.



Press the "T.S.M" SK on the PPU.



Enter "500" in "Spindle speed" on the PPU.



Select "M3" as the "Spindle direction" using the "Select" key on the PPU.

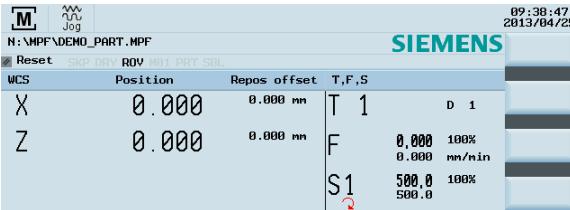


Press "CYCLE START" on the MCP.





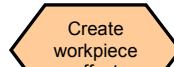
SEQUENCE



Press the “Reset” key on the MCP to stop the spindle rotation.



Press the “Back” SK on the PPU.



A tool must have been created and measured before it can be used to set the workpiece offset.



Make sure the active tool is the measured tool!

Press the “Machine” key on the PPU.



Press the “JOG” key on the MCP.



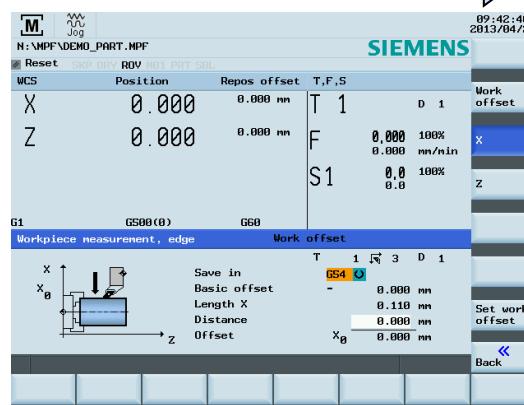
Press the “Offset” key on the PPU.



Press the “Work offset” SK on the PPU.



Press the “Meas.work.” SK on the PPU.



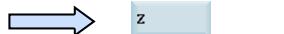


SEQUENCE

Step 2

Using a tool that has a measured “Tool length”, move the tool to a known position on the workpiece. Using either JOG or Handwheel, scratch an edge and then calculate the zero point of the workpiece.
The process of setting the zero point (“Z0”) is described below.

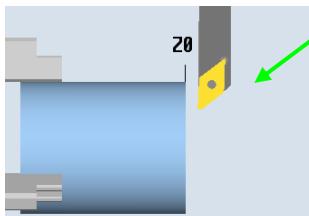
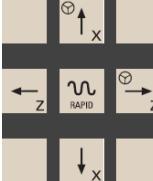
Press the SK on the PPU to select the required setting axis.



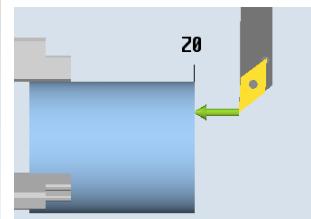
z



Press the axis traverse keys to move the tool to the required setting position in the Z axis.



Press the “Handwheel” key on the MCP to move the tool to the Z0 position on the workpiece.



Enter tool number “1” in “T”.



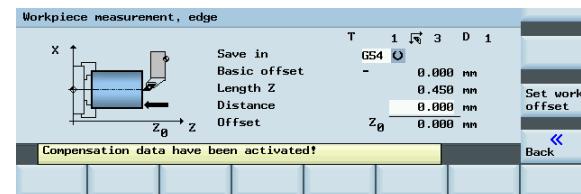
Set “Save in” as “G54” (or other offset).



Set “Distance” as “0”



Press the “Set work offset” SK on the PPU.



Repeat the operations to set the “X” zero point.



Press the “Back” SK on the PPU after measuring.



Workpiece Setup

SEQUENCE

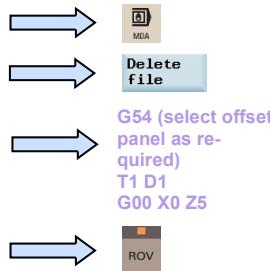
Test tool offset results



The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

In order to ensure the machine safety and correctness, the results of the tool offset should be tested appropriately.

Press the "MDA" key on the MCP.



Press the "Delete file" SK on the PPU.

Enter the test program recommended on the right.

Press the "ROV" key to ensure the "ROV" function is active (the function is activated when the light on the key is on).

Note: The ROV function activates the feedrate override switch under the G00 function.



Make sure the feedrate override on the MCP is at 0%!

Press "CYCLE START" on the MCP.



Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast and observe whether the axis moves to the set position.

MDA

In MDA mode, you can enter and execute single and multiple lines of NC codes.

Use MDA to move the axis to a fixed position.

Press the "Machine" key on the PPU.



Press the "MDA" key on the PPU.



Press the "Delete file" SK on the PPU.



Enter correct NC code to move the axis to the required position...



Make sure the feedrate override on the MCP is at 0%!

Press "CYCLE START" on the MCP to execute the MDA program.



Turn the feedrate override on the MCP gradually to the required value.

SIEMENS					
Stop	SKP	DRY	ROV	MDI	PRT
WCS	Position	Dist-to-go	T,F,S		G function
X	40.000	0.000 mm	T 1	D 1	Auxiliary function
Z	50.000	0.000 mm	F 0.000 10000.000 mm/min	100%	Axis feedrate
			S1	0.0 0.0 0	Save file
G00	G54	G60			Delete file
MDI - Block					
#00 G54 X40 Z50! =eof=					
Act. val. REL					

SEQUENCE

M/H Function

The M function initiates switching operations, such as "Coolant ON/OFF". Various M functions have already been assigned a fixed functionality by the CNC manufacturer. The M functions not yet assigned are reserved for free use of the machine tool manufacturer. With H functions, the meaning of the values of a specific H function is defined by the machine tool manufacturer. M codes and H functions created by the OEM should be backed up by the machine tool manufacturer.

Sub program

Frequently used machining sequences, e.g. certain contour shapes, are stored in subprograms. These subprograms are called at the appropriate locations in the main program and then executed.

The structure of a subprogram is identical to that of the main program, but a subprogram contains M17 - end of program in the last block of the program sequence. This means a return to the program level where the subprogram was called.

The subprogram should be given a unique name enabling it to be selected from several subprograms. When you create the program, the program name may be freely selected.

However, the following rule should be observed:

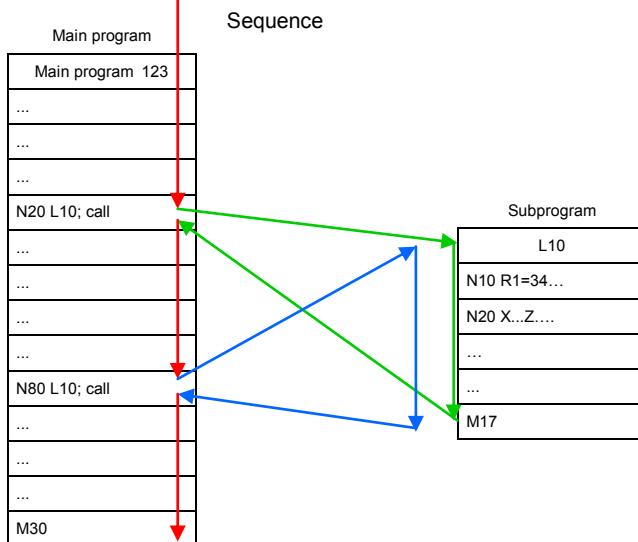
The name can contain letters, numbers and underscores and should be between 2 and 8 characters long.

Example: LRAHMEN7

Specified M Function	Explanation	Specified M function	Explanation
M0	Stop program	M7 / M8	Coolant on
M1	Stop program with conditions	M9	Coolant off
M2	End program	M10 / M11	Chuck close / release
M30	End program and back to the beginning	M20 / M21	Tailstock release / close
M17	End subprogram	M40	Select gear stage automatically
M3 / M4 / M5	Spindle CW/CCW/ Stop	M41~M45	Change spindle gear



SEQUENCE



Subprograms can be called from a main program, and also from another subprogram. In total, up to 8 program levels, including the main program, are available for this type of nested call.

Polar coordinates

In addition to the common specification in Cartesian coordinates (X, Z), the points of a workpiece can also be specified using polar coordinates.

Polar coordinates are also helpful if a workpiece or a part of it is dimensioned from a central point (pole) with specification of the radius and the angle.

The polar coordinates refer to the plane activated with G17 to G19. In addition, the third axis perpendicular to this plane can be specified. When doing so, spatial specifications can be programmed as cylindrical coordinates.

The polar radius RP= specifies the distance of the point to the pole. It is saved and must only be written in blocks in which it changes, after the pole or the plane has been changed.

The polar angle AP= is always referred to the horizontal axis (abscissa) of the plane (for example, with G18: X axis). Positive or negative angle specifications are possible. The positive angle is defined as follows: Starting from the plus direction of X axis and rotates CCW.

It is saved and must only be written in blocks in which it changes, after the pole or the plane has been changed.

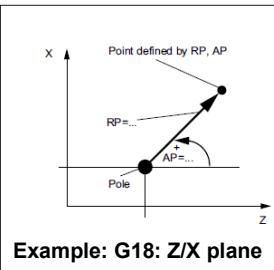
SEQUENCE

- G110 Pole specification relative to the setpoint position last programmed (in the plane, e.g. with G18: Z/X)
(when using G110, please always take the current position of the tool as the reference point to specify the new pole)
- G111 Pole specification relative to the origin of the current workpiece coordinate system (in the plane, e.g. with G18: Z/X)
- G112 Pole specification, relative to the last valid pole; retain plane

Programming example

N10 G18 ; Z/X plane
N20 G111 X17 Z36 ; pole coordinates in the current workpiece coordinate system...

N80 G112 AP=45 RP=27.8 ; new pole, relative to the last pole as a polar coordinate
; polar coordinate
; polar coordinate and Z axis (= cylinder coordinate)



Additive
workpiece
offsets

The programmable workpiece offsets TRANS and ATRANS can be used in the following cases:

- For recurring shapes/arrangements in various positions on the workpiece
- When selecting a new reference point for dimensioning

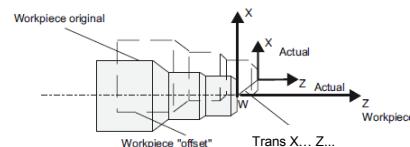
This results in the current workpiece coordinate system.

TRANS X... Z... ; programmable offset (absolute)
ATRANS X... Z... ; programmable offset, additive to existing offset (incremental)

TRANS ; without values, clears old commands for offset

Programming example
N20 TRANS X20.0 Z15.0
L10

programmable offset
subprogram call



SEQUENCE

Scaling

A scale factor can be programmed for all axes with SCALE, ASCALE. The path is enlarged or reduced by this factor in the specified axis. The currently set coordinate system is used as the reference for the scale change.

SCALE X... Z... ; programmable rotation offset (absolute)
ASCALE X... Z... ; programmable offset, additive to existing offset
(incremental)

If a program contains SCALE or ASCALE, this must be programmed in a separate block.

Programming example
N10 G17

N20 SCALE X2.0 Z2.0 ; contour is enlarged two times in X and Z
L10 subprogram call

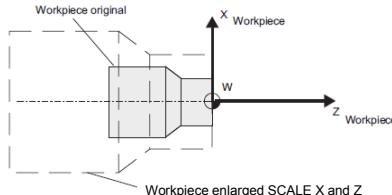
Program
jump

NC programs process their blocks in the sequence in which they were arranged when they were written. The processing sequence can be changed by introducing program jumps. The jump destination can be a block with a label or with a block number. This block must be located within the program. The unconditional jump command requires a separate block.

GOTOF+ label: Jump forward (in the direction of the end block of the program)

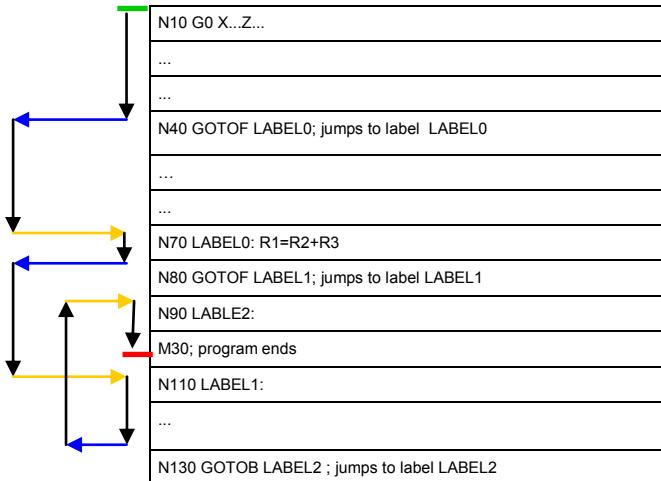
GOTOB+ label: Jump backward (in the direction of the start block of the program)

Label: Name of the selected string (standing for the required jump program block) or block number



SEQUENCE

Program execution



Unconditional jump example

Program skip

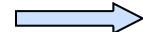
Method 1

“,” code

Using “,” code at the beginning of the block can skip this string.

“,” can also be used to add remarks to the block.

See the figure on the right for an example of use.



N5 G90 G500 G71

N10 T1 D1 M6

N15 S3000 M3 G94 F300

N20 G00 X50 Z5

N25 G01 Z-20

N30 Z5

...
N85 T2 D1 M6 ; change tools

N90 S3000 M3 G94 F300

; N95 G00 X60 Z10

...

Using “,” code at the beginning of the program block N95, this string will be skipped without execution.

Using “,” code to add a remark to the N85 function, without any influence on the execution.

Additional Information Part 2

SIEMENS

SEQUENCE

Method 2

Press the "Machine" key on the PPU.



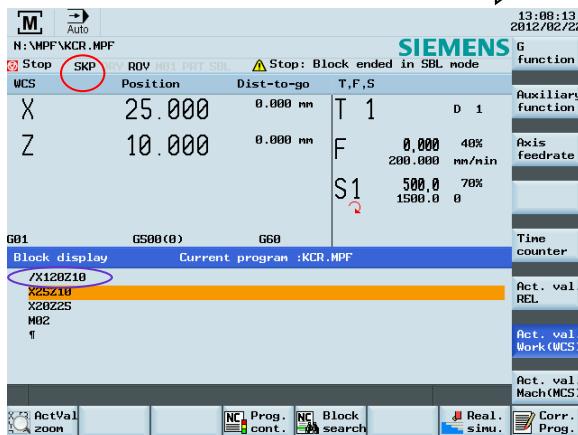
Press the "Auto" key on the MCP.



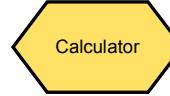
Press the "Prog cont." SK on the PPU.



Press the "Skip" SK on the PPU.

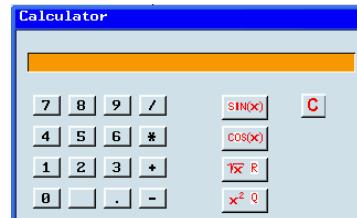


When "SKP" is displayed (red circle), the skip function has been activated. After activating "SKP", using "/" at the beginning of the program string (shown in purple circle), the string will be skipped without influencing the execution.



You can use the calculator to calculate contour elements, values in the program editor, tool offsets and workpiece offsets and enter the results on the screen.

Press the "=" on the PPU.



SEQUENCE**C**
Delete**«**
Back**✓**
Accept

Press this SK to delete the contents in the calculator.

Press this SK to exit the calculator screen.

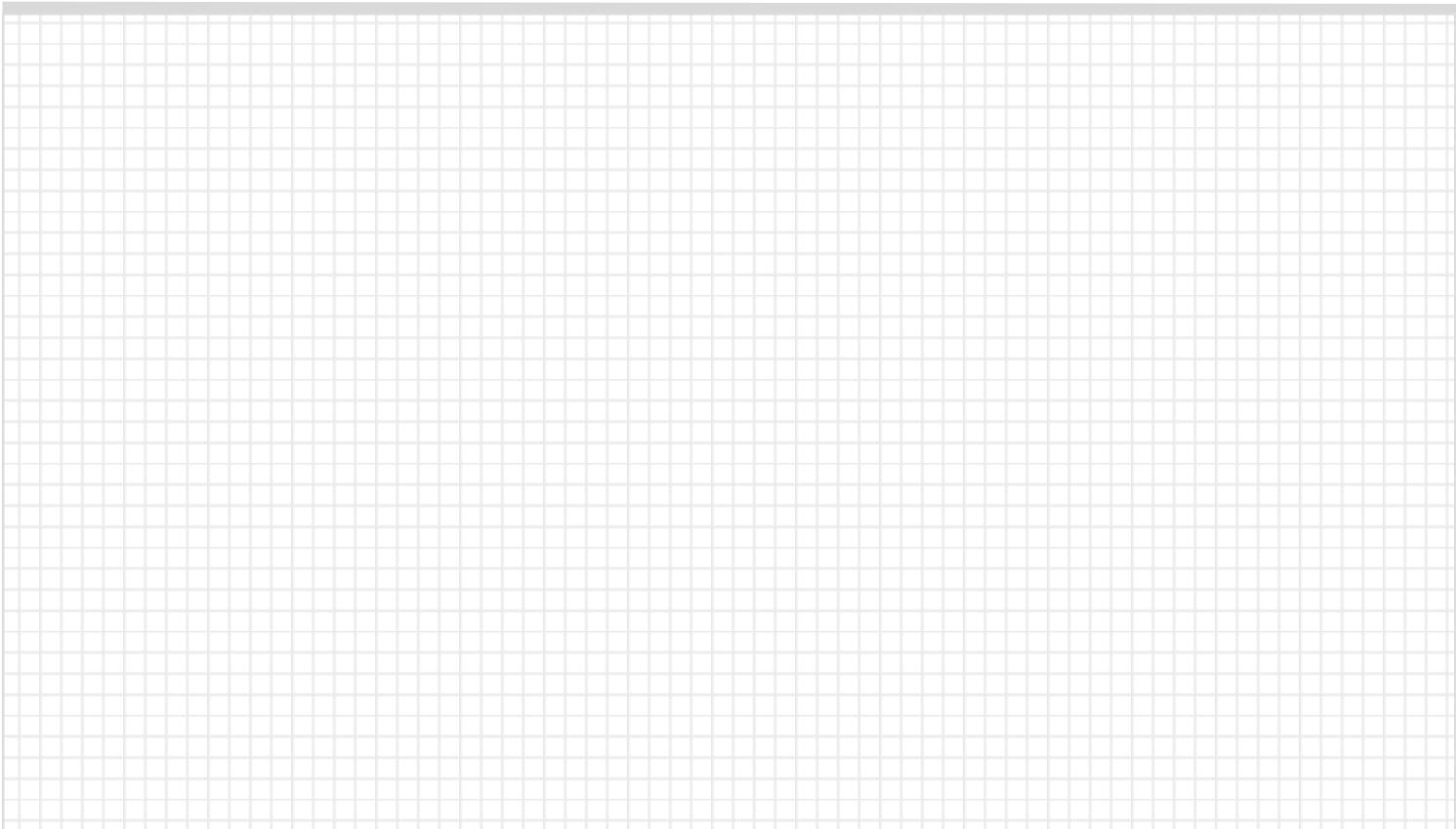
Use this SK to accept the input and write the values to the required position.

If the input field is already occupied by a value, the calculator will take this value into the input line.

Use the "Accept" SK to enter the result in the input field at the current cursor position of the part program editor. The calculator will then close automatically.



Notes

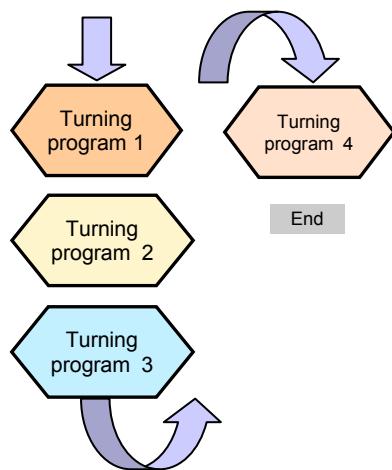


Content

Unit Description

This unit shows three typical program examples of frequently used turning cycles and the corresponding machining diagrams with detailed explanations.

Unit Content



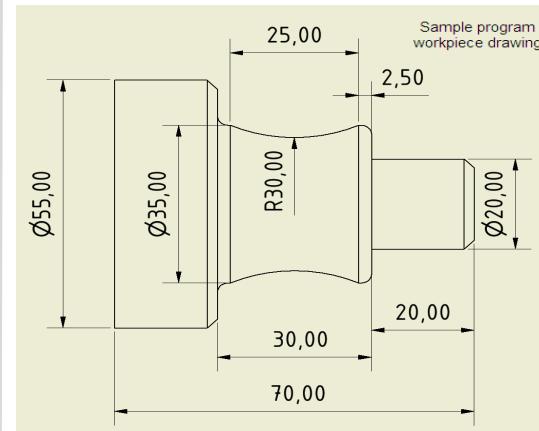
Note: All the program examples in this book are only for reference. If you want to perform actual operations, please adjust the tool offset, coordinate moving range, workpiece plane settings, etc. according to the actual machine conditions!

DRAWING

Turning
program 1

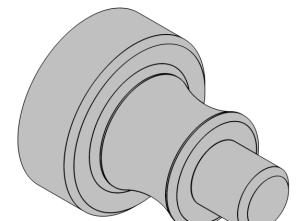


Make sure all the preparations and safety measures have been performed before machining!



Tool information:

T1 Turning tool D0.8
T2 Turning tool D0.8



Actual effect



Machining Process

```

N10 G00 G90 G95 G40 G71
N20 LIMS=4500
N30 T1 D1
; ======Start face turning=====
N40 G96 S250 M03 M08
N50 G00 X60 Z0
N60 G01 X-2 F0.35
N70 G00 Z2
N80 G00 X60
; ======End face turning=====
; ======Start contour turning
roughing without back cut=====
N90 CYCLE95("SUB_PART_1", 1.5,
0.2, 0.1, , 0.5, 0.3, 0.2, 9, , , )
N100 T2 D1
N110 G96 S250 M03 M08
;=====Start contour turning
finishing with back cut=====
N120 CYCLE95("SUB_PART_1A",
0.5, , , 0.2, 0.4, 0.3, 0.2, 9, , , )
N130 M30

```

```

N10 spindle feedrate in mm/r
N20 set spindle upper limit 4500 r/min
N30
; ======Start face turning=====
N40 constant cutting speed 250 m/min
N50
N60 feedrate is 0.35 mm/r
N70
N80
; ======End face turning=====
; ==Start contour turning roughing
without back cut===
N90 maximal feed depth 1.5 mm,
vertical axis finishing allowance 0.2 mm,
horizontal axis finishing allowance 0.1
mm, roughing feedrate 0.5 mm/r, feed
along the negative direction of the Z axis
to do complete machining.
N100
N110 constant cutting speed 250 m/min
; ==Start contour turning finishing
with back cut===
N120 maximal feed depth 0.5 mm,
contour finishing allowance 0.2 mm,
feedrate 0.3 mm/r with back cut, finishing
feedrate 0.2 mm/r, feed along the nega-
tive direction of the Z axis to do complete
machining.
N130

```

SUB_PART_1.SPF

```

G18 G90
G0 X16 Z0
G1 X20 Z-2
Z-20
X35 RND=2
Z-50 RND=2
X55 CHR=2
Z-70
M2; /* end of contour */

```

SUB_PART_1A.SPF

```

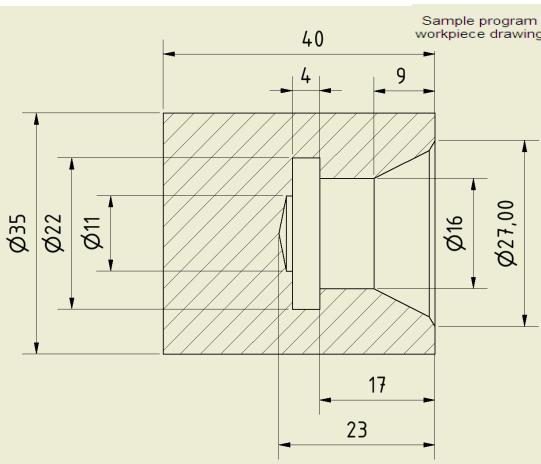
G18 G90
G0 X35 Z-22.5
G2 I=AC(89.544) Z-47.5 K=AC(-35)
G1 Z-49.5
M2; /* end of contour */

```

DRAWING

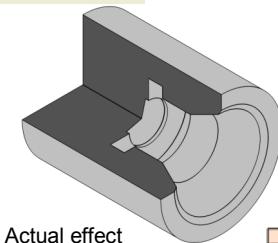
Turning
program 2

Make sure all the preparations and safety measures have been performed before machining!



Tool information:

- T1 Turning tool D0.8
- T10 Turning tool D0.8
- T13 Drilling tool D10
- T110 Grooving tool D0.2
Tool tip width 3



Machining Process

```

N10  G54 G00 G90 G95 G40 G71
N20  LIMS=4500
N30  T1 D1
N40  G96 S250 M03 M08
; =====Start face turning=====
N50  G00 X35 Z0
N60  G01 X-2 F0.35
N70  G00 Z2
N80  G00 X35
; =====End face turning=====
N90  T13 D1
; =====Start drilling=====
N100 G95 S1000 M4 G17
N110 G00 Z1 X0
N120 CYCLE83( 10, 0, 2, -23, 0,
-10, , 5, , 1, 0, 1, 5, 0, , 0 )
N130 G18
; =====End drilling=====
N140 T10 D1
; ==Start contour turning roughing==
N150 CYCLE95("PART_SUB_2", 1.5,
0.2, 0.1, , 0.5, 0.3, 0.2, 11, , , )
; ==End contour turning roughing==
N160 T110 D1
N170 G96 S250 M03 M08
N180 G00 Z1 X0
N190 G1 F0.3 Z-17
; =====Start grooving=====
N200 CYCLE93( 16, -17, 4, 3, , ,
, , , , 1, , 13, )
; =====End grooving=====
N210 M30

```

Machining Process

PART_SUB_2.SPF

CONTOUR

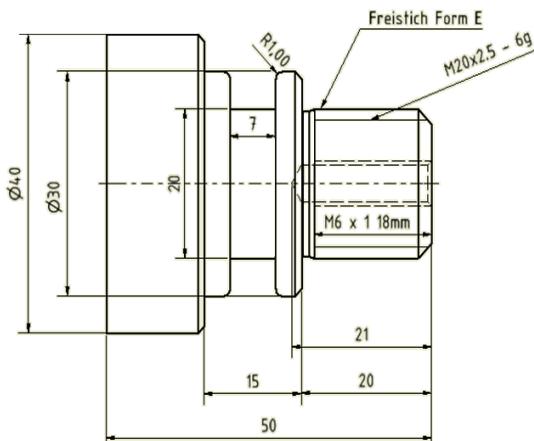
```
G18 G90
G0 X27 Z0
G1 X24.11 Z-.89
X16 Z-9
Z-21
X10
M2; /* end of contour */
```



DRAWING

Turning
program 3

Part of the cycles in the program are taken as examples in Section 5, "Create Part Program Part 2"!



Tool information:

T1 Turning tool D0.8

T2 Turning tool D0.8

T3 Grooving tool D0.2

Tool tip width 2

T4 Turning tool D0.8

T5 Grooving tool D0.2

Tool tip width 3

T6 Drilling tool D10

T7 Drilling tool D10

T8 Tap D12



Machining Process

N10 G00 G90 G95 G40 G71
N20 LIMS=4500

; ==Start contour turning roughing==
N30 T1 D1 ;ROUGH TURN

N40 G96 S250 M03 M08

N50 G0 X52.0 Z0.1

N60 G01 X-2.0 F0.35

N70 G00 X52.0 Z2.0

N80 CYCLE95("SUB_PART_3", 2.5,
0.2, 0.1, 0.15, 0.35, 0.2, 0.15, 9, , ,)
N90 G00 G40 X500.0 Z500.0

N100 M01

; ==Start contour turning finishing ==
N110 T2 D1 ;FINISH TURN

N120 G96 S350 M03 M08

N130 G00 X22.0 Z0.0

N140 G01 X-2.0 F0.15

N150 G00 Z2.0

N160 X52.0

N170 CYCLE95("SUB_PART_3", , ,
, , , 0.15, 5, , ,)

N180 G00 G40 X500.0 Z500.0

N190 M01

; =====Start grooving=====

N200 T3 D1 ;GROOVE

N210 G96 S200 M03 M08

N220 G00 X55.0 Z0.

N230 CYCLE93(30, -30.5, 7, 5, 0, 0,
0, 1, 1, , 0, 0.2, 0.1, 2.5, 0.5, 11,)

N240 G00 G40 X500.0 Z500.0

N250 M01

; =====End grooving=====

N10 spindle feedrate in mm/r
N20 set spindle upper limit 4500 r/min
; ==Start contour turning roughing==
N30

N40 constant cutting speed 250 m/min
N50 feedrate is 0.35 mm/r
N60

N70 maximal feed depth 2.5 mm,
vertical axis finishing allowance 0.2 mm,
horizontal axis finishing allowance 0.1 mm,
contour finishing allowance 0.15 mm,
roughing feedrate 0.35 mm/r,
feedrate 0.2 mm/r with back cut, feed
along negative direction of Z axis to do
complete machining.
N90 G40—cancel tool radius compensa-
tion

N100 delay changing tool
; ==Start contour turning finishing ==
N110
N120
N130
N140
N150
N160
N170 finishing feedrate 0.15 mm/min,
feed along negative direction of Z axis to
do complete machining.
N180
N190

; =====Start grooving=====

N200
N210
N220
N230 grooving start point (X30,Y-30.5),
groove width 7mm, depth 5 mm, angle
between contour and Z axis is 0°,
groove bottom finishing allowance 0.2 mm,
angle between groove middle both
sides and X axis is 0°, tooth face finish-
ing allowance 0.1 mm, feed depth 2.5 mm,
delay 0.5 s at base of groove, de-
fine reverse angle by entering side
length(CHR method).
N240
N250

; =====End grooving=====

Machining Process

; ======THREAD=====

```
N260 T4 D1 ; THREAD
N270 G95 S150 M03 M08
N280 G00 X50.0 Z10.0
N290 CYCLE99( 0, 20, -18, 20, 2, 0,
1, 0.01, 29, 0, 8, 2, 2.5, 300103, 1, ,
0, 0, 0, 0, 0, 0, 1, , , , 0 )
N300 G00 G40 X500.0 Z500.0
N310 M01
```

; ======CENTER DRILL=====

```
N355 T6 D1 ;CENTER DRILL
N360 G95 S1000 M03 M08
N370 G17 G00 X0 Z5
N375 CYCLE82( 5, 0, 2, -5, 0, 0.5 )
N380 G00 G40 X500 Z500
```

; ======DRILL=====

```
N390 T7 D1 ;DRILL
N400 G95 S1000 M03 M08
N410 G00 X0 Z5
N420 CYCLE82( 5, 0, 2, -20, 0, 0.5 )
N430 G00 G40 X500 Z500
```

; ======TAP HOLE=====

```
N440 T8 D1 ;TAP HOLE
N450 G95 S500 M3 M08
N460 G00 X0 Z5
N470 CYCLE84( 5, 0, 2, -18, 0, 0.5,
3, 12, , 0, 200, 200, 3, 0, 0, 0, , 0 )
N480 G0 G40 X500 Z500
```

; ======Start thread cutting=====

```
N260
N270 G95→spindle feedrate in mm/r
N280
N290 size of thread 2.5 mm, on Z
axis start point →end point: 0→20,
diameter at start point / end point are
both 20 mm, reverse distance 2 mm,
ending distance 0 mm, thread depth 1
mm, finishing allowance 0.01 mm, feed
angle 29°, first thread start point offset
0 mm, rough cutting 8 times, idle tool
cutting 2 mm, thread machining path is
thread string number 1
```

N300 G40→cancel tool radius com-
pensation
N310 delay changing tool

; ======Start center drilling=====

```
N355
N360
N370
N375 drilling depth 5 mm, delay time
at final drilling depth is 0.5 s
(discontinuous drilling)
```

N380 ; ======Start drilling=====

```
N390
N400
N410
N420 drilling depth 20 mm, delay time
at final drilling depth is 0.5 s
(discontinuous drilling)
```

N430 ; ======Start tapping hole=====

```
N440
N450
N460
N470 tapping depth 18 mm, deep
drilling delays 0.5 s (discontinuous drill-
ing), spindle rotating direction is M3
when withdrawn, thread size is M12,
spindle stop position is 0°, tapping speed
and turning speed are both 200 mm/min,
tool axis is Z axis, machining way is
tapping, withdraw path is 1 mm
(discontinuous drilling)
N480
```

; ======CUT OFF=====

```
N320 T5 D1 ;CUT-OFF
N330 G18 G96 S200 M03 M08
N340 G00 X55.0 Z10.0
N350 CYCLE92( 40, -50, 6, -1, 0.5, ,
200, 2500, 3, 0.2, 0.08, 500, 0, 0, 1,
0, 11000 )
N351 G00 G40 X500 Z500
N360 G00 G40 X500.0 Z500.0
N370 M30
```

; ======Cutting off=====

```
N320
N330
N340
N350 cutting off start point (X40, Y-50),
depth for speed reduction (diameter) 6
mm, final depth -1, 0.50000, constant
cutting speed is 200 mm/min, maximum
speed at fixed speed is 2500 r/min, spin-
dle rotating direction is M3, feedrate depth
is 0.2 mm/min when rotational speed is
reached, reduced feedrate (until the final
depth) is 0.08 mm/min, reduced speed
(until the final depth) is 500 r/min, ma-
chining path returns to basic plane, alter-
native mode is reverse angle.
N351
N360
N370
```

PART_SUB_3.SPF

```
G18 G90
G0 X16 Z0
G1 X20 Z-2
Z-15
X19.2 Z-16.493 RND=2.5
Z-20 RND=2.5
X30 CHR=1
Z-35
X40 CHR=1
Z-55
X50
M2; /* end of contour */
```

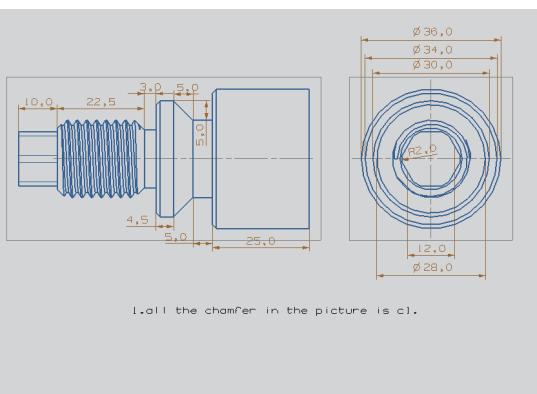
CONTOUR

Machining Process

Turning
program 4



Make sure all the preparations and safety measures have been performed before machining!



Sample program
workpiece drawing

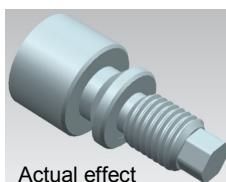
Tool information:

T1: rough/finish turn D0.2

T3: Thread

T9: Grooving 3 mm wide tip

T10: Milling cutter Φ=12



```

N10  G0X100Z200
N20  T1 D1
N30  M8
N40  M3S1000
N50  CYCLE95( "PART_SUB_4",
1.00000, , , 0.10000, 0.12000, 0.12000,
,1, , )
N60  M3S1800
N70  CYCLE95( "PART_SUB_4",
1.00000, , , 0.10000, 0.12000, 0.12000,
0.05000, 5, , )
N80  M3S1000
N90  CYCLE94( 20, -35.5, "E", 0)
N100 G0X100Z200
N110 M5
N120 T9D1
N130 M3S500
N140 CYCLE99( -10.00000, 20.00000,
-32.50000, 20.00000, 3.00000, 2.00000,
1.56300, 0.10000, ,0.00000, 7, 1,
2.50000, 300101, 1, ,0, 0, 0, 0, 0, 0, 1,
, ,0)
N150 G0X100Z200
N160 M5
N170 T3D1
N180 M3S600
N190 CYCLE93( 40.00000, -35.50000,
5.00000, 10.00000, ,45.00000, , ,
1.00000, ,0.100000, 0.100000,
2.00000, ,5, , )

```

```

N10
N20
N30
N40
N50  Rough cycle, cutting depth
1.5mm, finish allowance 0.1mm, rough
cutting feed rate 0.12mm/rev undercut
federate 0.12 cutting in the longitudinal
direction.
N60
N70  Finish cycle ,cutting depth
1.5mm, finish allowance 0.1mm, rough
cutting feed rate 0.12mm/rev undercut
feedrate 0.12 , finish feedrate 0.05
cutting in the longitudinal direction.
N80
N90  undercut "form E" with a starting
point X 20, Z-35.5)
N100
N110
N120
N130
N140  Thread cycle, pitch 2.5mm, start
point in Z-10 end point Z-35.5, parallel
thread with dia 20mm, Run in 3mm, run
out 2mm, thread depth 1.563mm, finish
allowance 0.1mm, number of rough cuts
7, non cutting passes 1, constant in-
feed .
N150
N160
N170
N180
N190  Groove, start point X40,Z-35.5,
width of groove 5mm, depth 10mm,
infeed depth 1mm, angle 1 45deg ,
finish allowance on sides 0.1mm, infeed
depth of cut 2mm, CHF chamfer type.

```

Machining Process

N200 G0X100Z200
N210 T1 F400 G94
N220 G0 X50 Z60 SPOS=0
N230 SETMS(2)
N240 M3 S2000
N250 TRANSMIT
N260 G90 G17
N270 DIAMOF
N280 G1 Z-10 F100
N290 G1 X8 Y0 G42 RND=2
N300 X4Y6.928
N310 RND=2
N320 X-4Y6.928 RND=2
N330 X-8 Y0
N340 RND=2
N350 X-4Y-6.928
N360 RND=2
N370 X4Y-6.928
N380 RND=2
N390 X8 Y0
N400 RND=2
N410 G40 G0 Z40 RND=2
N420 M5
N430 TRAFOOF
N440 SETMS
N450 G54 G18 G0 X50 Z60
N460 Z30
N470 M30

N200 Feedrate in mm/min 400
N210 Rotational positioning 0 deg
N220 Set the second spindle as main
spindle
N240
N250 Activate TRANSMIT function
N260 Activate G17 plane
N270 Deactivate dim programming
N280
N290 Activate TRC G42, radius 2 mm
N300
N310
N320
N330
N340
N350
N360
N370
N380
N390
N400
N410 Cancel TRC, radius 2mm
N420
N430 Deactivate TRANSMIT function
N440 Set the first spindle as main spindle
N450
N460
N470

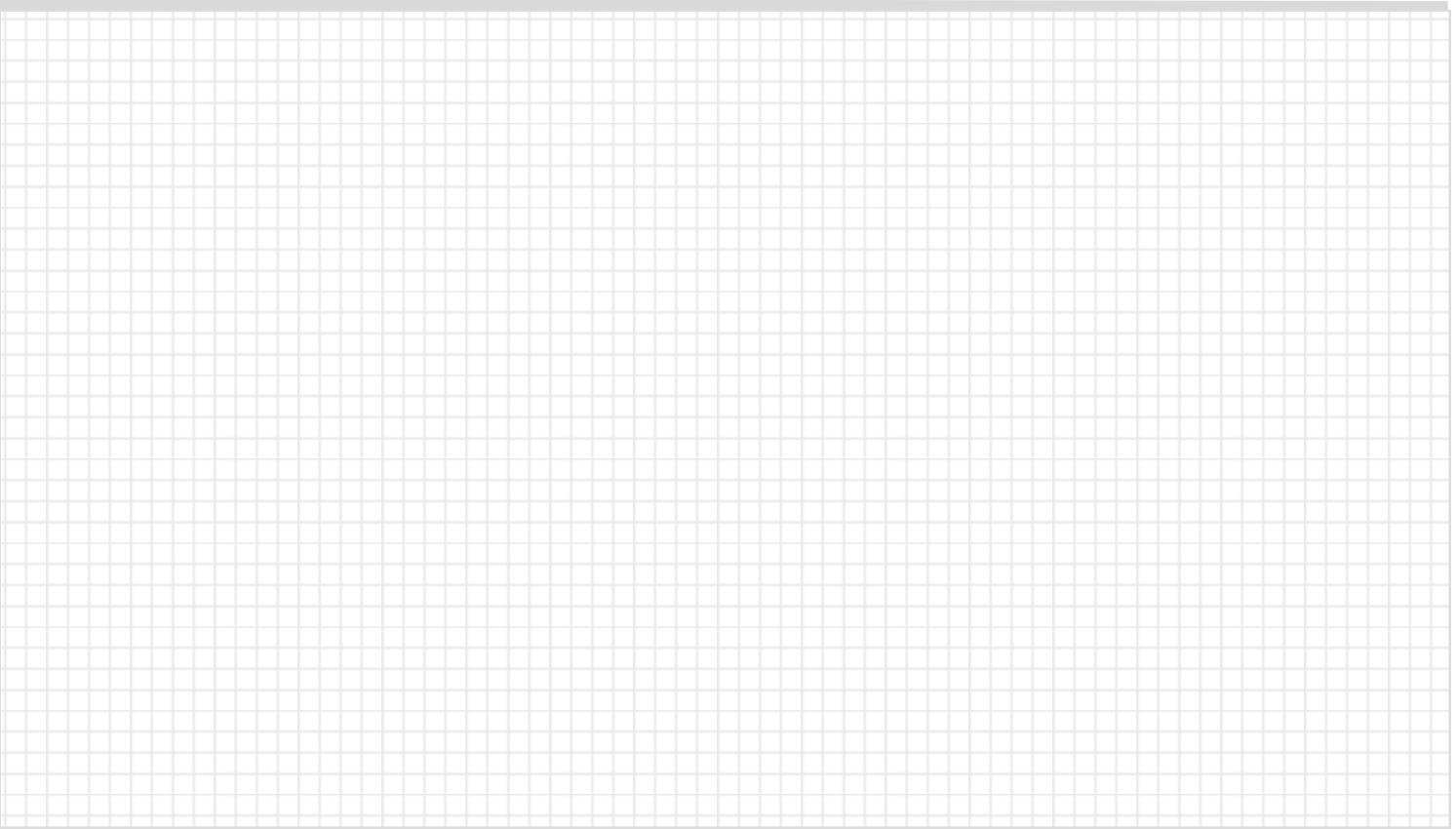
PART_SUB_4.SPF

G18 G90 DIAMON
G0 Z0 X-.5
G1 X20
Z-35.5
X30 CHR=1
Z-50
X36 CHR=1
Z-75
X46
M2 ; /* end of contour */

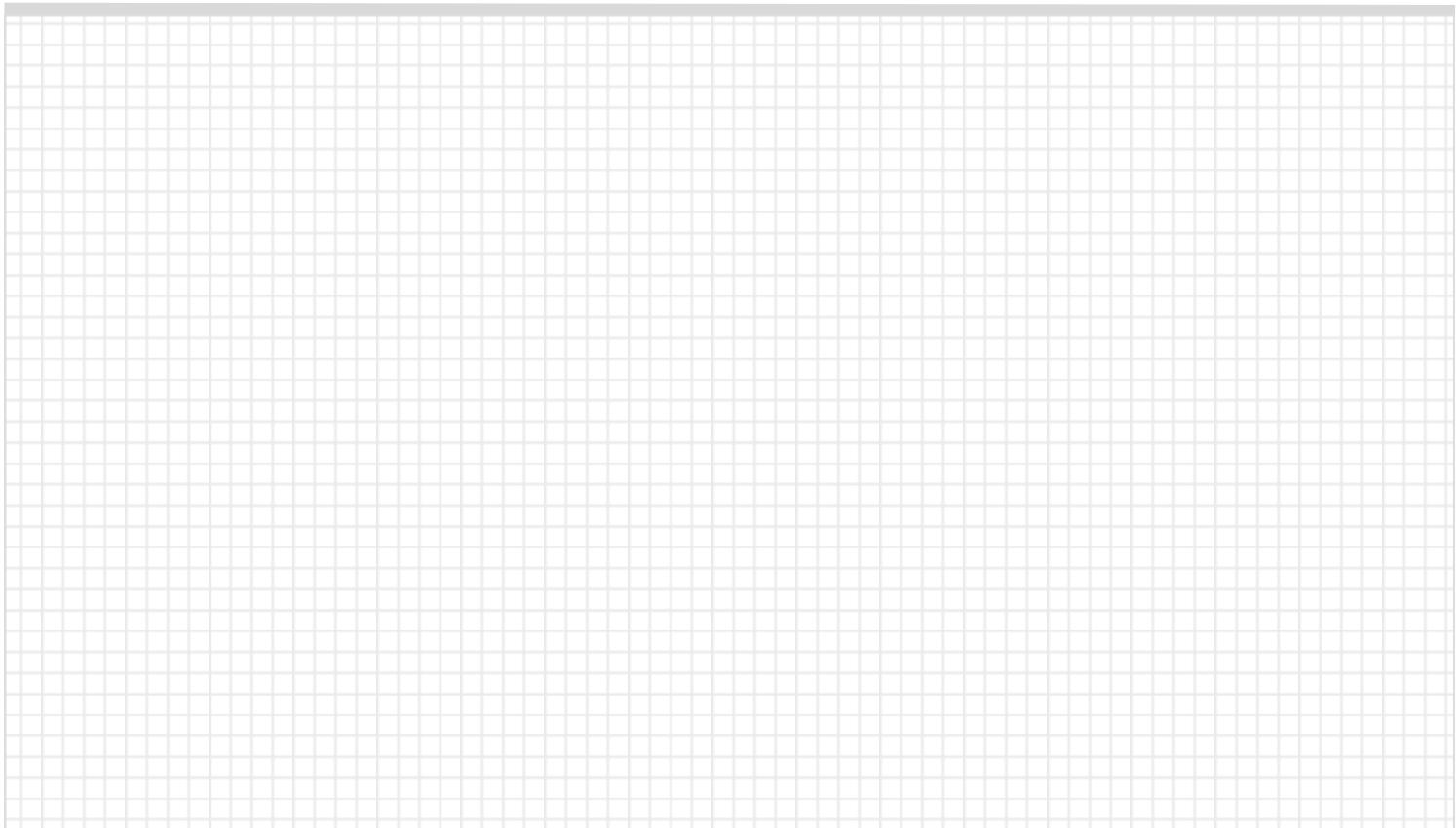
Sub program



Notes



Notes





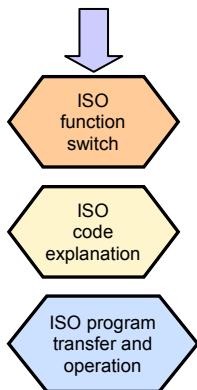
Content

Module Description

This unit describes the ISO operating functions in 808D ADVANCED, compares the similarities and differences of the machining code in DIN mode and ISO mode and shows how to transfer and implement the ISO machining program.

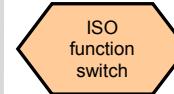
All the ISO codes described in this unit can be implemented in the ISO mode of the 808D ADVANCED system.

Module Contents



End

BASIC THEORY



Siemens standard machining codes are implemented in DIN mode. The 808D ADVANCED also provides appropriate functions for implementing the ISO commands, but the ISO mode must be activated during operation.

ISO function switch

Method 1

Press the “Shift” + “System - Alarm” keys →
on the PPU. Input the manufacturer’s password (“SUNRISE”)



Press the “ISO mode” SK on the right. →

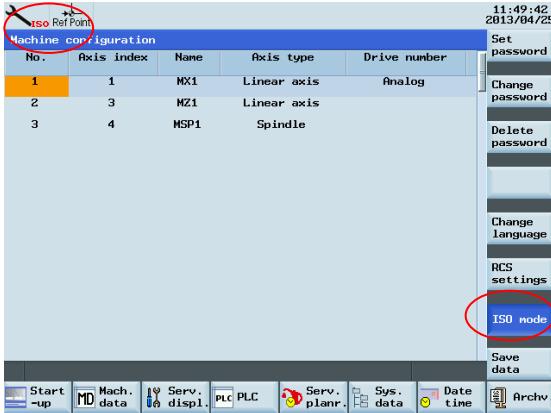


A dialog box appears prompting whether to activate the new setting. Select the “OK” SK to activate it.



BASIC THEORY

After pressing "OK", the system restarts automatically. After restarting, press "Shift" + "System - Alarm" again and if the symbol in the red circle appears, ISO mode is already activated.



A red ISO appears at the top of the screen and the ISO mode button on the right is highlighted in blue.



When using method 2 to activate the ISO mode, it will exit ISO mode and return to the default DIN mode via "Reset" button or after finishing the machining program.

Insert G291 in the first line of the ISO part program to be executed and insert G290 in front of M30.

```
N0 G291
M5 G17 G90 G54 G71 1#
H20 T1 H1#
M25 MSG("Tool No. 1 in use")#
N3G54000 M31
N40 CYCLE1(G S0.0000, 2.0000, 2.0000, 0.0000, 0.0000, 0.0000
N45 S4500 M31
```

! G291/G290 commands must be set separately in a line!

If ISO is displayed at the top of the screen, it is activated.



All the ISO codes described in this unit can be implemented in the ISO mode of the 808D ADVANCED system!

Brief description of typical, frequently used ISO codes

ISO code	Description	Compare with DIN
G00	Orientation (rapid traverse)	As DIN
G1	Linear difference	As DIN
G17/G18/G19	XY plane / ZX plane / YZ plane	As DIN
G20/G21	Input in inch/mm	G70/G71
G32	Equal lead thread cutting	G33
G41/G42/G40	Left tool tip radius compensation / right tool tip radius compensation / cancel tool radius compensation	As DIN
G54 ~ G59	Select workpiece coordinate system	As DIN
G80	Cancel fixed cycle	
G98/G99	Feedrate F in mm/min / mm/r	G94/G95
S	Spindle speed	As DIN
R	Reverse circle	RND
, C	Reverse bevel angle (note the form there must be ", " before C parameter)	CHF/CHR
M3/M4/M5	Spindle right / spindle left / spindle stop	As DIN
M98 P_L_	Subprogram call (P+ subprogram name/L+ times)	Program name +L_
M99	Subprogram end	M17

BASIC THEORY

G98: Spindle in mm/min

G99: Spindle in mm/r

G80: Cancel fixed cycle

Pausing function **G04**

G04 X5.0→dwell 5 s

G04 P5→dwell 5 ms

M3 S2000; spindle rotation

G98 F500 G01 X100; feedrate is 500 mm/min

G92 X50 W-20 F2 ;F is the thread lead

G04 X2.0 ;delay 2 s

G99 G01 U10 F0.01 ;feedrate is 0.01 mm/r

G00 G80 Z50 M30 ;cancel this cycle

M5 ; spindle rotation stop

M30

Tool function **T** code

Tool offset number	X	Z
00	0.000	0.000
01	0.000	0.000
02	12.000	-23.000
03	24.560	13.542

T code has two functions:
 ①→change automatically
 ②→execute tool offset

Code form **T ΔΔ OO**

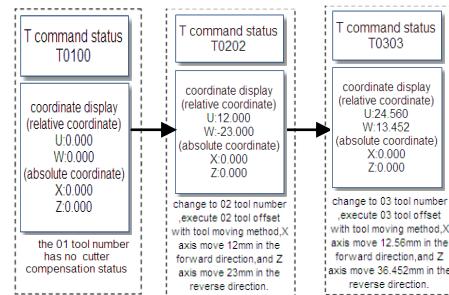
ΔΔ:

Enter target tool number

OO:

Input tool offset number

Note: When using G291 to activate ISO mode, you must set machine data MD10890=0, or the tool path can not be implemented.



Code **G02** and **G03**

G02 circular interpolation CW

Motion path: Start point→end point

CW (rear tool coordinate system) / CCW (front tool coordinate system)

G03 circular interpolation CCW

Motion path: Start point→end point

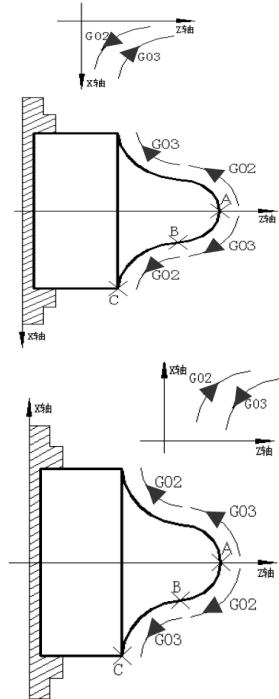
CCW (rear tool coordinate system) / CW (front tool coordinate system)

You can specify the circle end point in the following X/Z address for both. You can also describe circle radii with I, K incremental or use parameter R to specify radii directly.

When specifying circle radii with parameter **R**

Circles less than 180° are assigned positive values
G02 X6.0 Y2.0 R50.0

Circles greater than 180° are assigned negative values
G02 X6.0 Y2.0 R-50.0



BASIC THEORY

Frequently used letter meanings of typical fixed cycle codes in ISO mode

P.	Descriptions	Unit	Applied range and note
X/Z	Cutting end point X/Z absolute coordinate values	mm	G90 / G94 / G74 G75 / G92 / G76
U/W	Absolute coordinate difference between start point and end point at X/Z	mm	G90 / G94 / G74 G75 / G92 / G76
	X/Z tool retraction / finishing allowance	mm	G73
R	Radii—difference between start point and end point		G90 / G94 / G92
	Each radial/shaft (X/Z axis) tool retraction e	mm	G71 / G72 / G74 / G75
	Cutting times d		G73
	Thread finishing d / thread cone i	mm	G76
P	Single radial cutting cycle at X axis Δi	0.001mm	G74
	Feed at X axis Δi	0.001mm	G75
	Thread finishing turning time m /thread retraction length r Angle between two nearby thread teeth a / thread tooth height k	time / 0.1 times thread lead / 0.001mm	G76
Q	Feed at Z axis Δk	0.001 mm	G74
	Single radial cutting cycle at Z axis Δk	0.001 mm	G75
	Minimum roughing thread Δd_{min} Thread first cut depth Δd	0.001 mm	G76
F	Cutting feed speed	mm	G90 / G71 / G72 G73 / G94 / G74 / G75
	Thread lead in metric system F(l)	mm	G92 / G76
I	Thread teeth/inch in inch system		G92 / G76

Brief introduction of typical fixed cycle codes in ISO mode



For the meaning of letters when programming typical fixed cycles, please refer the figure on the left!

G90 shaft cutting cycle

Programming structure:

Cylinder cutting

G90 X U Z W F;

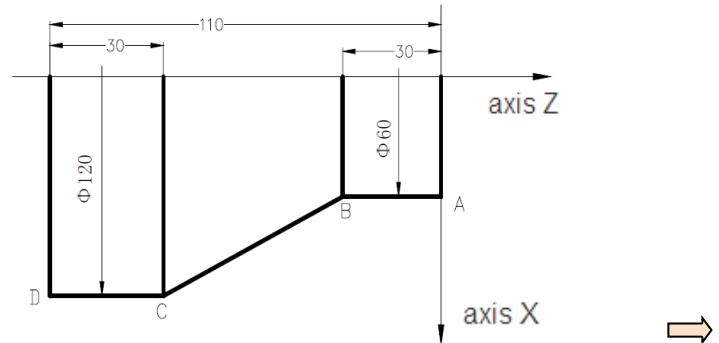
Cone cutting

G90 X U Z W R F;

Note: Please follow the specified structures when programming!

G90 sample example:

O0002;
M3 S300 G0 X130 Z3
G90 X120 Z_110 F200 A→D, Φ120 cutting
X110 Z_30 ——————
X100
X90
X80
X70
X60
G0 X120 Z_30 A→B, Φ60 cutting,
G90 X120 Z_44 R_7.5 F150 divided 6 times,
Z_56 R_15 each time feed in 10 mm
Z_68 R_22.5
Z_80 R_30 ——————
M30 B→C, cone cutting
divided four times



BASIC THEORY

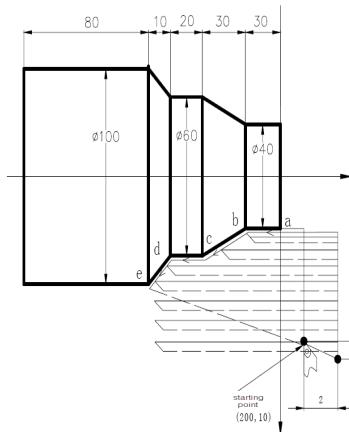
G71 shaft roughing cycle

Programming structures:

G71 U(Δd)—R(e);**G71 P(ns)—Q(nf)—U(Δu)—W(Δw)—F—S—T
N(ns)...**

...

...

N(nf);**P(ns) / Q(nf):** Indicating start/end point of finishing program block path**Note: Please follow the specified structures when programming!****G71** sample program:**O0004;****G00 X200 Z10 M3 S800****G71 U2 R1**

; each feed in 4 mm, retraction 2 mm

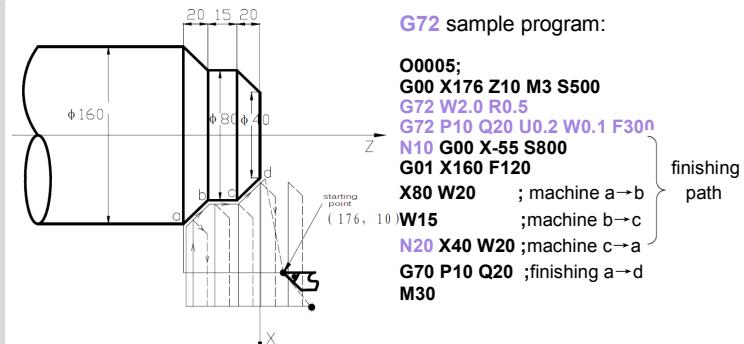
G71 P80 Q120 U0.5 W0.2 F200; for a ~ e roughing, X axis allowance 1 mm
Z axis allowance 2 mm**N80 G00 X40 S1200****G01 Z-30 F100** ;machine a→b
X60 W-30 ;machine b→c
W-20 ;machine c→d**N120 X100 W-10** ;machine d→e
G70 P80 Q120 ;finishing a→e
M30**G72** radical roughing cycle

Programming structures:

G72 W(Δd)—R(e);**G72 P(ns)—Q(nf)—U(Δu)—W(Δw)—F—S—T;
N(ns)...**

...

...

N(nf);**P(ns) / Q(nf):** Indicating start/end point of finishing program block path**Note: Please follow the specified structures when programming!****G72** sample program:**O0005;****G00 X176 Z10 M3 S500****G72 W2.0 R0.5****G72 P10 Q20 U0.2 W0.1 F300****N10 G00 X-55 S800****G01 X160 F120****X80 W20** ; machine a→b
W15 ; machine b→c**N20 X40 W20** ; machine c→a**G70 P10 Q20** ; finishing a→d**M30**

BASIC THEORY

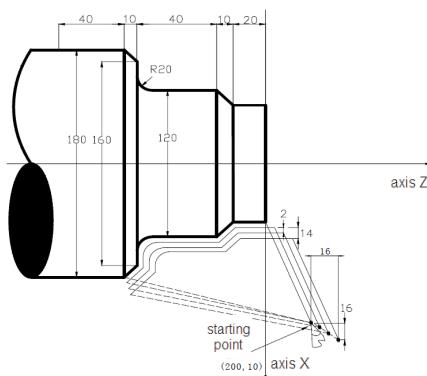
G73 closed cutting cycle

Programming structures:

G73 U(Δ i)—W(Δ k)—R(d);**G73 P(ns)—Q(nf)—U(Δ u)—W(Δ w)—F—S—T;****N(ns)...**

...

...

N(nf);**P(ns) / Q(nf):** Indicating start/end point of finishing program block path**Note:** Please follow the specified structures when programming!**G73 sample program:****O0006;****G99 G00 X200 Z10 M3 S500****G73 U1.0 W1.0 R3**

; tool retraction at X axis 0.2 mm, at Z axis 1 mm

G73 P14 Q19 U0.5 W0.3 F0.3

; roughing, keep 0.5 mm finishing allowance at X axis and 0.3 mm at Z axis

N14 G00 X80 W-40**G01 W-20 F0.15 S600****X120 W-10****W-20****G02 X160 W-20 R20****N19 G01 X180 W-10****G70 P14 Q19 ; finishing M30****G70 finishing cycle**

Programming structures:

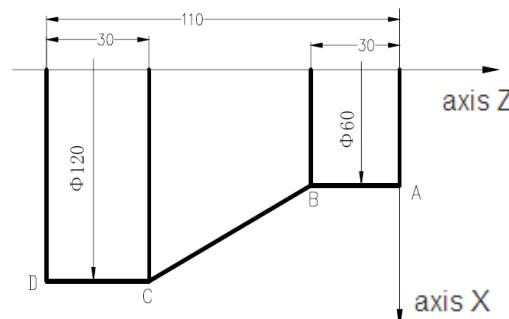
G70 P(ns)—Q(nf);**P(ns) / Q(nf):** Indicating start/end point of finishing program block path**Note:** T / S / F used in G70 must be specified in G71/G72/G73 fixed cycles before G70.**G94 radical cutting cycle**

Programming structures:

Face cutting

G94 X / U—Z / W—F;

Cone face cutting

G94 X / U—Z / W—R—F;**Note:** Please follow the specified structures when programming!

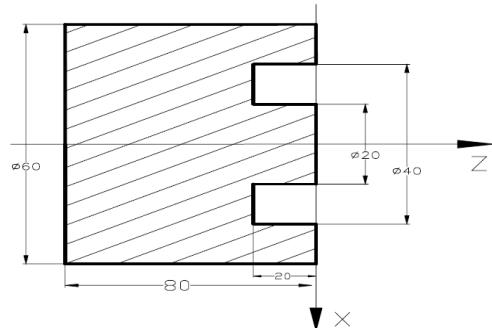
BASIC THEORY

G74 shaft grooving multi-cycles

Programming structures:

G74 R(e);
G74 X U—Z / W—P(Δi)—Q(Δk)—R(Δd)—F;

Note: Please follow the specified structures when programming!

**G74** sample program:

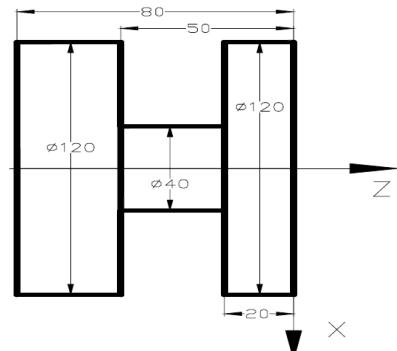
```
O007;
M3 S1500
G0 X40 Z5
G74 R0.5
; set each radical tool retraction
0.5 mm
G74 X20 Z60 P3000 Q5000 F50
; Z axis feed in 5 mm each time,
tool retraction 0.5 mm, back to
start point (Z5) after feeding to
end point (Z60), then X axis feed
in 3 mm, repeat the process till
the end
M30
```

G75 radical grooving multi-cycles

Programming structures:

G75 R(e);
G75 X U—Z / W—P(Δi)—Q(Δk)—R(Δd)—F;

Note: Please follow the specified structures when programming !

**G75** sample program:

```
O008;
M3 S500
G0 X125 Z-20
G75 R0.5
; set each radical tool retraction
0.5mm
G74 X40 Z-50 P6000 Q3000 F150
; X axis feed in 6 mm each time,
tool retraction 0.5 mm, back to
start point (X125) after feeding to
end point (X40), then Z axis feed
in 3 mm, repeat the process till the
end
G0 X150 Z50
M30
```

BASIC THEORY

G92 thread cutting cycle

Programming structures:

Straight thread cutting cycle in mm

G92 X / U—Z / W—F;

Straight thread cutting cycle in inches

G92 X / U—Z / W—I;

Cone thread cutting cycle in mm

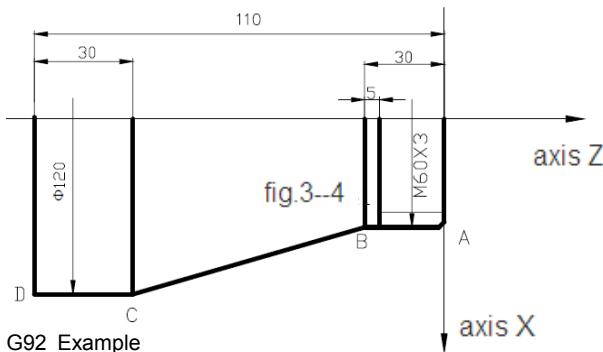
G92 X / U—Z / W—R—F;

Cone thread cutting cycle in inches

G92 X / U—

Z / W—R—I;

Note: Please follow the specified structures when programming!



G92 sample program:

O0012;

M3 S1500

G0 X150 Z50 T0101; thread tool

G0 X65 Z5

G92 X58.7 Z-28 F3

; machining thread, divided into 4 cutting times, 1st cut : 1.3 mm
X57.7 ; 2nd cut: 1 mm

X57 ; 3rd cut: 0.7 mm

X56.9 ; 4th cut: 0.1 mm

M30

G76 thread cutting multi-cycles

Programming structures:

G76 P(m)(r)(a)—Q(Δd_{min})—R(d);

G76 X / U—Z / W—R(i)—P(k)—Q(Δd)—F(l);

Note: Please follow the specified structures when programming!

G76 sample program:

O0013;

M3 S3000

G0 X80 Z5

G76 P020560 Q150 R0.1

; finishing repeat times 2, reverse width 0.5 mm, tool angle 60°, minimum cutting depth 0.15 mm, finishing allowance 0.1 mm

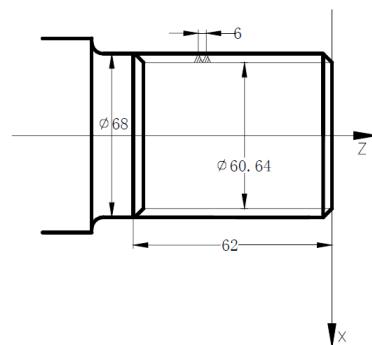
G76 X60.64 Z-62 P3680 Q1800 F6

; thread teeth height 3.68 mm, first thread cutting depth 1.8 mm, thread lead 6 mm

G00 X100 Z50

M30

cutting point enlarged picture



ISO Mode

BASIC THEORY

ISO
program
transfer and
operation



The ISO mode function provided by the 808D ADVANCED can easily operate the existing ISO program!

Step 1 Transfer ISO files in USB device to 808D ADVANCED.

Connect the USB device with the stored target programs to the USB interface on the PPU.

Press the "USB" SK on the PPU.



Use the "Cursor + Select" keys to select the required program which is then highlighted.

Press the "Copy" SK on the PPU.



Press the "NC" SK on the PPU.



Press the "Paste" SK on the PPU.



A specified ISO program is then stored in the 808D ADVANCED system and can be edited and executed as described above.

Step 2 Make the necessary changes to the ISO programs.



Programs in ISO mode in the 808D ADVANCED have their own rules. Suitable changes must be made at the appropriate positions so that you can run the ISO programs!



Common ISO prog.	808D ISO prog.
O0001; G0 X100 Z100 M5 G04 X5 M3 S1000 ...	O0001; delete G0 X100 Z100 M5 G04 X5 M3 S1000 ...

Beginning of the program

Common ISO programs:
Beginning is "O"

ISO mode in 808D ADVANCED:
Not compatible with programs beginning with "O"

Common ISO prog.	808D ISO prog.
T0707; G0 X45. Z3. G94 X-1. Z0 F0.2 ...	T0701 ; can also create 07 tool edge in tool list if required G0 X45 Z3 G94 X-1 Z0 F0.2 ...

T code

Common ISO programs:
The default active tool offset number is same as the tool number

ISO mode in 808D ADVANCED:
Tool active method T ΔΔ OO
No matter what the tool number is, the default active tool offset is 01

Note:

- If you use the SKs on the PPU to activate ISO mode, you can use T0701 directly
- If you use G291 to activate ISO mode, you must set machine parameter MD10890=0 first and then you can use T0701
No matter which way to activate ISO mode, the default active tool offset number is 01. If you want to use T0707 further, you must create tool edge number 7 in the 7th tool (each tool has a maximum of nine tool edges)



BASIC THEORY

G90/G94 与 G71

ISO mode in 808D

ADVANCED:

Must add relevant codes of G00/G01 between two cycles, or alarms will be displayed

Common ISO programs:
Two cycles can be executed continuously

Note: Alarm numbers 10255/15100/14082/10932 are available

F / T / S in G71-G75

ISO mode in 808D

ADVANCED:

F must be edited in the 2nd line

Common ISO programs:
F position is optional

Note: Alarm number 61812 is available

808D ISO prog.

N70 G90 X43 Z-130

N80 X41;

N85 G0 X45 Z3

; must add program here

N89 G71 U1.5 R1 F0.3;

...

Common ISO prog.

N70 G90 X43 Z-130

N80 X41;

N89 G71 U1.5 R1 F0.3;

...

F / T / S in G70

ISO mode in 808D

ADVANCED:

Must be edited between G71 cycle blocks (N100~N200)

Common ISO programs:

- ① can be edited in G71 cycle blocks (N100 ~ N200)
- ② or can be edited in line G70

808D ISO prog.

N89 G71 U1.5 R1 F0.3

N90 G71 P100 Q170 U0.5 W-0.2;

N100 G01 X16 Z0 F0.15

; F0.15 speed during G70

...

N200 G0 X45 Z3;

N210 G70 P100 Q170 ;

...

Common ISO prog.

N89 G71 U1.5 R1 F0.3

N90 G71 P100 Q170 U0.5 W-0.2;

N100 G01 X16 Z0;

...

N200 G0 X45 Z3;

N210 G70 P100 Q170 F0.15

; F0.15 is the speed during G70 and can be written anywhere between N100 ~ N200

808D ISO prog.

N89 G71 U1.5 R1 ;

N90 G71 P100 Q170 U0.5 W-0.2 F0.3

; the speed of F must be edited in the 2nd line of G71

N100 G01 X16 Z0;

...

Common ISO prog.

N89 G71 U1.5 R1 **F0.3**

; the speed of F can be written in this line or in N90

N90 G71 P100 Q170 U0.5 W-0.2;

N100 G01 X16 Z0;

...

Reversing angle and reversing circle

Common ISO programs:

Linear reversing angle code: L

Circle reversing angle code: D

ISO mode in 808D ADVANCED:

Linear reversing angle code:

CHR (specified side length of isosceles triangle with chamfer as base line)

CHF (specified base line length of isosceles triangle with chamfer as base line)

Circle reversing angle code: RND



Note: If the L/D command is used in ISO mode in the 808D ADVANCED, the system will automatically skip the program line in which it lies without any operation.



BASIC THEORY

Step 3 Program execution.



Make sure the current system is in ISO mode!
Make sure all preparations and safety measures have been performed!

Operate as described above.

Tool and workpiece setup → simulation → test → machining

Step 4 Transfer the ISO files in the 808D to the USB device.

Connect the USB device with sufficient memory to the USB interface on the PPU.

Press the "NC" SK on the PPU.



Use the "Cursor + Select" keys to select the required program which is then highlighted.



Press the "Copy" SK on the PPU.



Press the "USB" SK on the PPU.



Press the "Paste" SK on the



A specified ISO program is then stored in the USB and can be executed as required.



Step 5

Sample program (target workpiece is the same as in Section 5 "Create Part Program Part 2").



Make sure the current system is in ISO mode!
Make sure all preparations and safety measures have been performed!

ISO programs can be executed in the 808D ADVANCED as follows:

**G291
G99 M3 S800 F0.3**

**T1
G0 X42 Z2
G71 U1 R0.5
G71 P101 Q102 U0.5 W0 F0.3
N101 G01 X0 Z0 S1200 F0.1
X20,C2
Z-20
X30,C2
W-15
U10 R3
Z-50
N102 X42
G70 P101 Q102
G0 X50
Z50**

**T3 M3 S500
G0 X22 Z4
G92 X20 Z-18 F2.5**

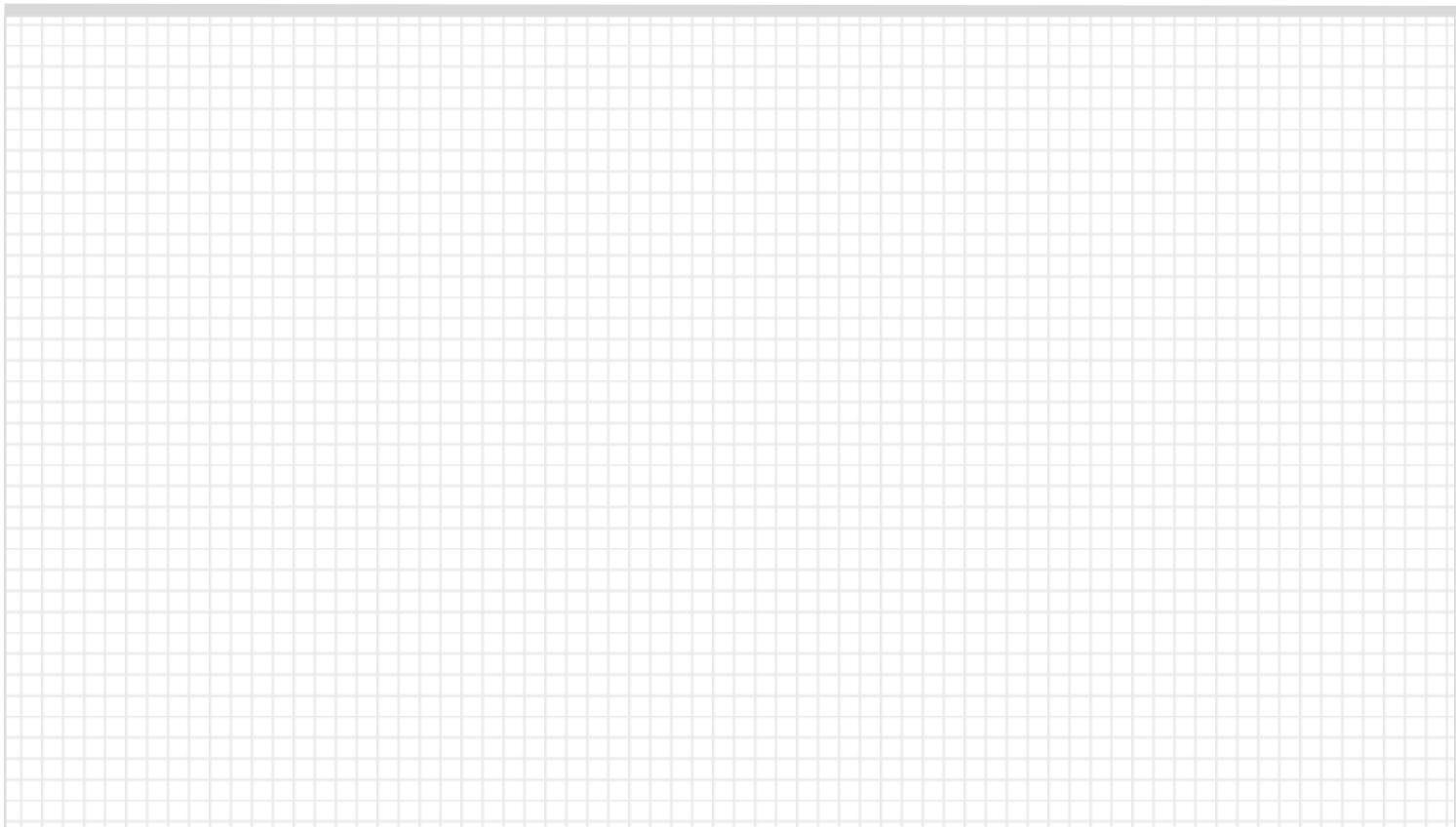
**X19
X18.5
X18
X17.5
X17
X16.8
X16.75
X16.75
G0 X50
Z50**

**T2 M3 S400 F0.2
G0 X32 Z-24
G75 R2
G75 X20 Z-31 P3000 Q3000
G0 X50
Z50
G0 X0 Z5**

**T5
M3 S500 F0.2
G74 R1
G74 X0 Z-21 P1000 Q5000 F0.2
G0 Z50
X50
G290
M30**



Notes



Appendix

Content

Unit Content



G
Functions

Technical
Support
Contact

Useful
Siemens
Websites

End

G Functions

Group 1: Modally valid motion commands

Name	Meaning
G00	Rapid traverse
G01 *	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
CIP	Circular interpolation through intermediate point
CT	Circular interpolation: tangential transition
G33	Thread cutting with constant lead
G331	Thread interpolation
G332	Thread interpolation — Retraction

Group 2: Non-modally valid motion. dwell

Name	Meaning
G04	Dwell time preset
G63	Tapping without synchronization
G74	Reference point approach with synchronization
G75	Fixed point approach
G147	SAR - approach with a straight line
G148	SAR - retract with a straight line
G247	SAR - approach with a quadrant
G248	SAR - retract with a quadrant
G347	SAR - approach with a semicircle
G348	SAR - retract with a semicircle

Group 3: Programmable frame

Name	Meaning
TRANS	Translation
ROT	Rotation
SCALE	Programmable scaling factor
MIRROR	Programmable mirroring
ATRANS	Additive translation
AROT	Additive programmable rotation
ASCALE	Additive programmable scaling factor
AMIRROR	Additive programmable mirroring
G110	Pole specification relative to the last programmed setpoint position
G111	Pole specification relative to origin of current workpiece coordinate system
G112	Pole specification relative to the last valid POLE

Group 6: Plane selection

Name	Meaning
G17	X/Y plane
G18 *	Z/X plane
G19	Y/Z plane

Group 7: Tool radius compensation

Name	Meaning
G40 *	Tool radius compensation OFF
G41	Tool radius compensation, left of contour
G42	Tool radius compensation, right of contour

Group 8: Settable zero offset

Name	Meaning
G500 *	Settable zero offset OFF
G54	1st settable zero offset
G55	2nd settable zero offset
G56	3rd settable zero offset
G57	4th settable zero offset
G58	5th settable zero offset
G59	6th settable zero offset

Group 9: Frame suppression

Name	Meaning
G53	Non-modal skipping of the settable zero offset
G153	Non-modal skipping of the settable zero offset including base frame suppression

Group 10: Exact stop-continuous-path mode

Name	Meaning
G60 *	Exact positioning
G64	Continuous-path mode

Group 11: Exact stop, non-modal

Name	Meaning
G09	Non-modal exact stop

Group 12: Exact stop window modally effective

Name	Meaning
G601 *	Exact stop window
G602	G60, G9 course stop window

Group 13: Workpiece measuring inch/metric

Name	Meaning
G70	Inch dimension data input
G71 *	Metric dimension data input
G700	Inch dimension data input, also for feedrate F
G710	Metric dimension data input, also feedrate F

Group 14: Absolute/Incremental dimension modally effective

Name	Meaning
G90 *	Absolute dimension data input
G91	Incremental dimension data input

Group 15: Feedrate spindle feedrate modally effective

Name	Meaning
G94	Feedrate F in mm/min
G95 *	Spindle feedrate in mm/r
G96	Constant cutting rate ON (in mm/r m/min)
G97	Constant cutting OFF

Group 16: Feedrate override modally effective

Name	Meaning
CFC *	Feedrate override with circle ON
CFTCP	Feedrate override OFF

Group 18: Behavior at corner when working with tool radius compensation

Name	Meaning
G450 *	Transition circle
G451	Point intersection

Group 44: Path segmentation with SAR modally effective

Name	Meaning
G340 *	Approach and retraction in space (SAR)
G341	Approach and retraction in plane (SAR)

Group 47: External NC languages modally effective

Name	Meaning
G290 *	Siemens mode
G291	External mode

Transformations

Name	Meaning
TRACYL	Cylinder. Peripheral surface transformation
TRANSMIT	Transmit: Polar transformation
TRAFOOF	Deactivate transformation

Technical Support

If you have any questions about this product or this manual, please contact the hotline:

Phone	+86 1064 719990
Fax	+86 1064 719991
E-mail	4008104288.cn@siemens.com

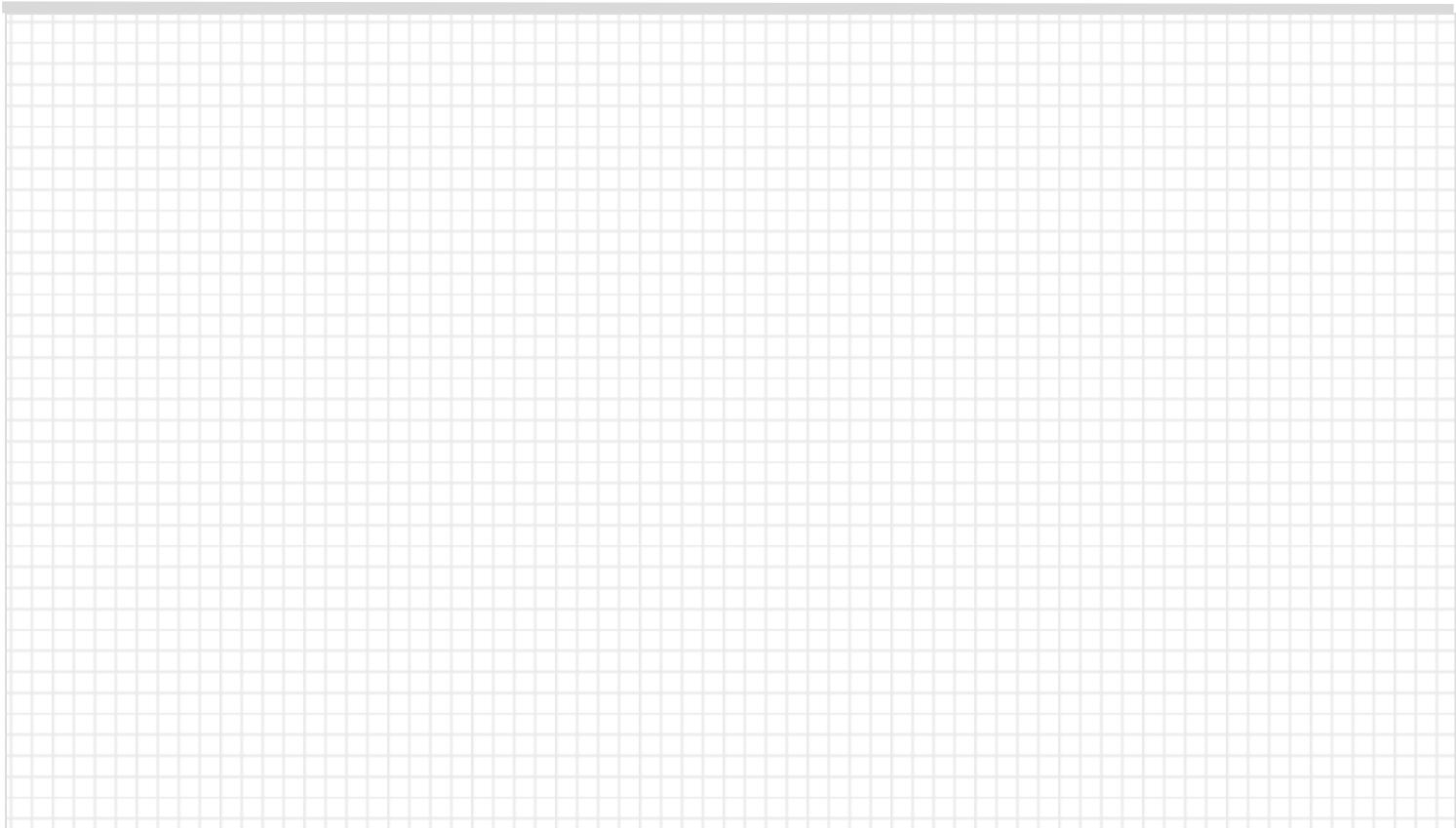
SINUMERIK internet address

Further product information can be found at the following website:

<http://www.siemens.com/sinumerik>



Notes

A large rectangular area filled with a uniform grid of light gray lines, creating a pattern of small squares. This grid covers most of the page below the 'Notes' header, intended for handwritten notes.

Everything ever wanted to know about SINUMERIK 808D:

www.automation.siemens.com/mcms/m2/en/automation-systems/cnc-sinumerik/sinumerik-controls/sinumerik-808/Pages/sinumerik-808.aspx

Everything about shopfloor manufacturing:

www.siemens.com/cnc4you

Everything about the SINUMERIK Manufacturing Excellence portfolio of services:

www.siemens.com/sinumerik/manufacturing-excellence

Information about CNC training:

www.siemens.com/sinumerik/training

Siemens AG
Industry Sector
Motion Control Systems
P.O.Box 3180
91050 ERLANGEN
GERMANY

Subject to change without prior notice
Order No.:
Dispostelle 06311
WÜ/35557 WERK.52.2.01 WS
11113.0
Printed in Germany
© Siemens AG 2012

The information provided in this brochure contains merely general descriptions or characteristics of performance which in actual case of use do not always apply as described or which may change as a result of further development of the products. An obligation to provide the respective characteristics shall only exist if expressly agreed in the terms of contract.

All product designations may be trademarks or product names of Siemens AG or supplier companies whose use by third parties for their own purposes could violate the rights of the owners.