SIEMENS

Training manual

Sinumerik 808D Programming and Operating Procedures for Milling

Version 2013-01

Notes

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

SIEMENS

Contents

SIEMENS

Index

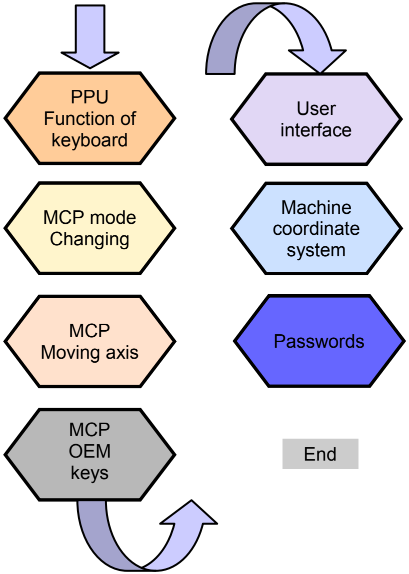
Preparation

Content

Unit Description

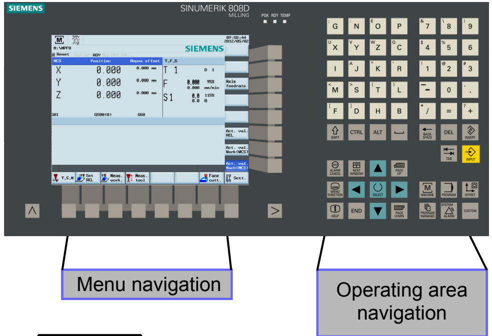
This unit describes the 808D PPU and MCP functionality, the coordinate system of a milling machine and how to enter passwords to access the system.

Unit Content



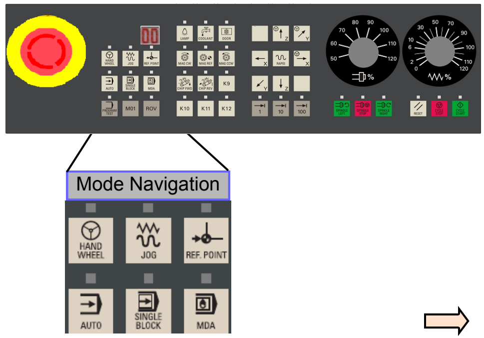
Basic Theory

PPU Function of keyboard



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

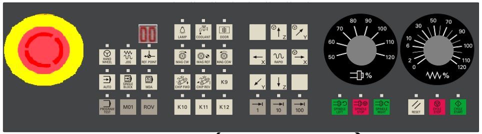
MCP mode Changing

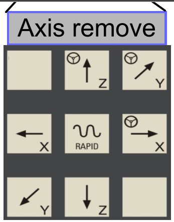


The 808D machine control panel (MCP) is used to select the machine operating mode :
JOG - MDA - AUTO

Preparation

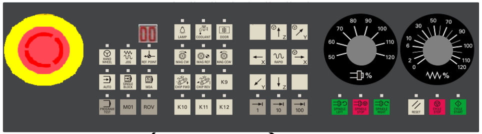
MCP Moving axis

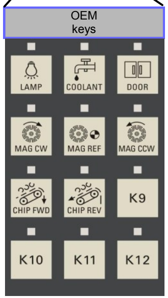




The 808D machine control panel (MCP) is used to control manual operation of the axis.
The machine can be moved with the appropriate keys.

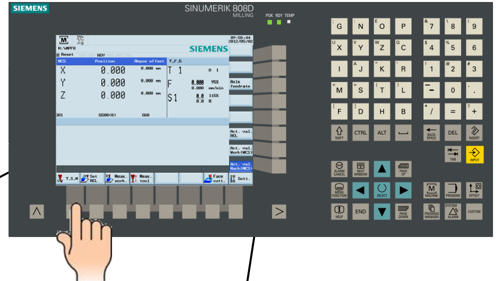
MCP OEM keys

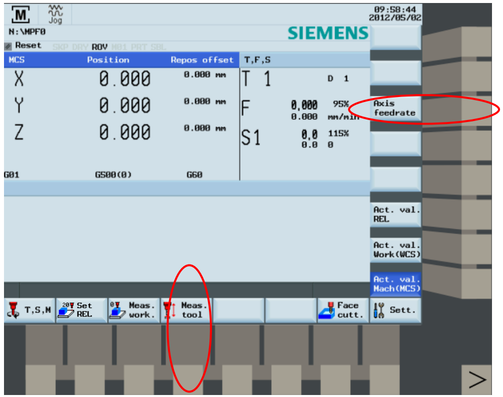




The 808D machine control panel (MCP) is used to control OEM machine functions.
The machine functions can be activated with the appropriate keys.

User interface





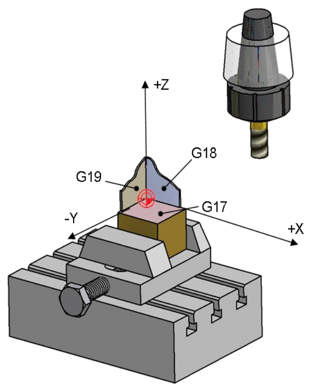
808D (PPU) has eight vertical softkeys (abbr. SKs) on the right of the screen. These SKs can be activated with the corresponding button (located on the right).

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

Preparation

SEQUENCE

Machine coordinate system



The Sinumerik 808D 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.

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

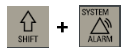
Customer's password = CUSTOMER
Manufacturer's password = SUNRISE

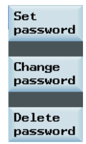
Changing password

Step 1

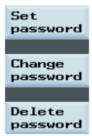
Usually the machine operator does not need to change the password.

The service mode is opened with the appropriate key combination.
In the service mode, the password can be activated and deactivated.





Step 2



Set password → Enter customer's or manufacturer's password

Change password → Change customer's or manufacturer's password

Delete password → Delete customer's or manufacturer's password

End

Notes

Content

Unit Description

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

Unit Content

Switch on the machine

Reference the machine

End

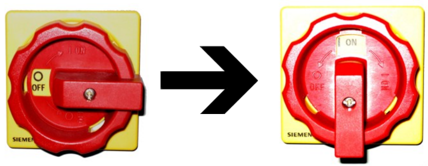
SEQUENCE

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

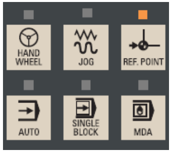
SIEMENS

SEQUENCE

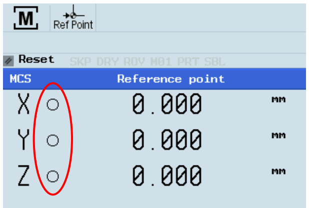
Reference the machine

After power on, the machine must first be referenced!

Step 1

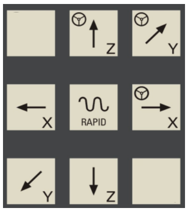


After power on, the machine will be in the reference point approach mode (default).



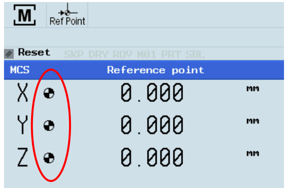
If the axis is not referenced, the non-referenced symbol (circle) is displayed between the axis identifier and the value.

Step 2



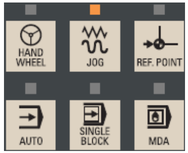
The axes are referenced with the corresponding axis traversing keys.

The traversing direction and keys are specified by the machine manufacturer.

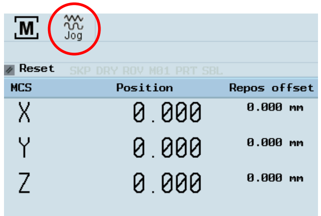


After completing the referencing procedure for all axes, the referenced symbol is displayed next to the axis identifier.

Step 3



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



The machine can now be operated in JOG mode.

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

End

Notes

Notes

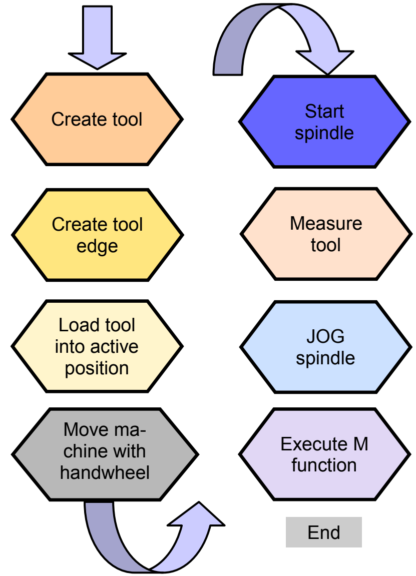
Tool Setup

Content

Unit Description

This unit describes how to create and set up tools.

Unit Content



SEQUENCE

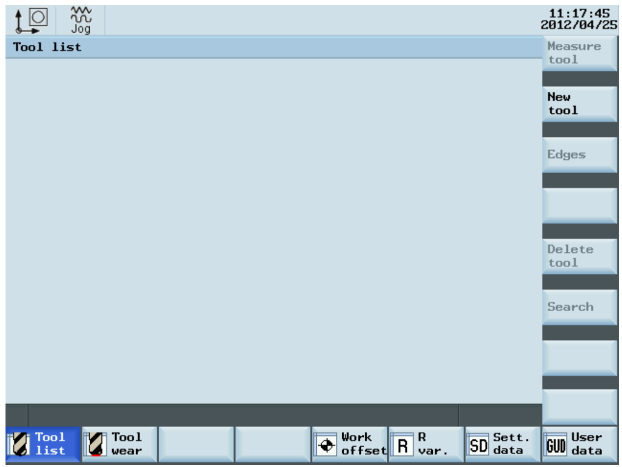
Create tool

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.

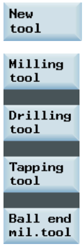


SEQUENCE

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.

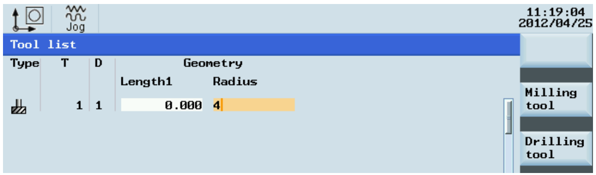


Select the type of tool required.

Enter “1” at “Tool No.”

Press the “OK” SK on the PPU.

Enter the ”Radius” of the milling tool.



Press the “Input” button on the PPU.

Create tool edge

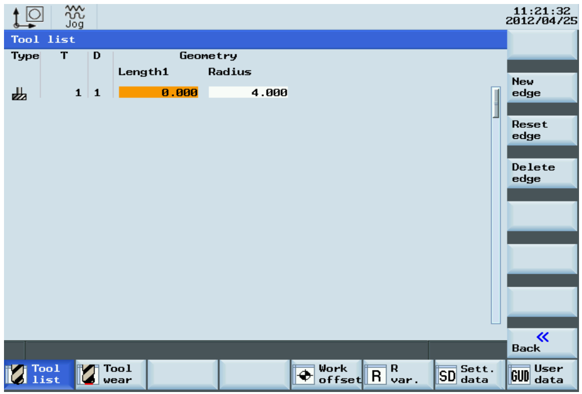
A tool must have been created and selected before creating a tool edge!

Step 1 Use “D” code to specify the tool edge. The system activates tool edge no. 1 per default at the start.

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 add a tool edge.



Press the “Edges” SK on the PPU.

Press the “New edge” SK on the PPU.

Tool Setup

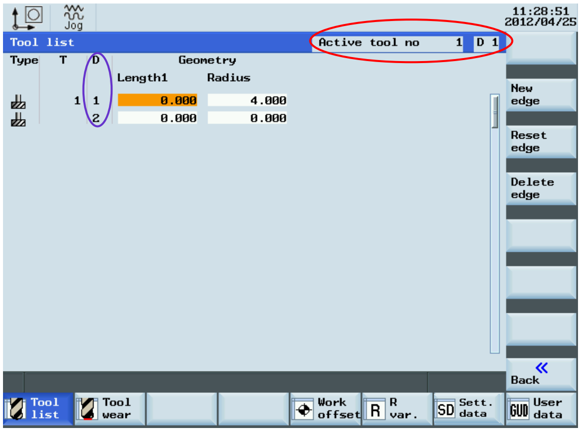
SIEMENS

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!

Load tool into Spindle

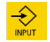
A tool must have been created in the system before it can be loaded into the active position.

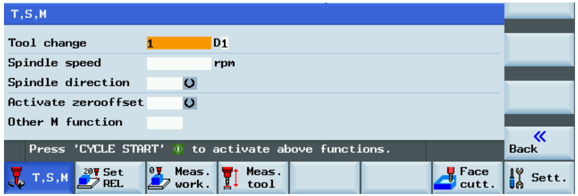
Press the “Machine” key on the PPU

Press the “JOG” key on the MCP

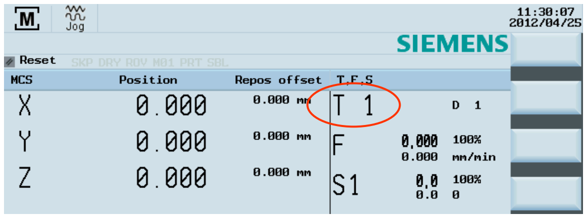
Press the “T.S.M” SK on the PPU

Enter tool number “1” in “T”





Press “CYCLE START” on the MCP



Press the “Back” SK on the PPU

SIEMENS

SEQUENCE

The tool are usually loaded manually into the spindle.

The tool will be automatically loaded into the spindle with an automatic tool changer.

Move machine with handwheel

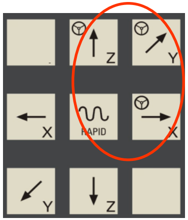
Make sure there is no obstruction when moving the tool to avoid a crash.

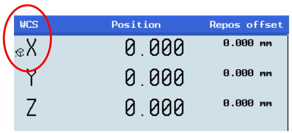
A handwheel can control the axis motion instead of the “JOG” button.

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





Under “WCS” or “MCS” state, a handwheel will be shown beside the axis symbols, showing the axis is chosen, and can be controlled with a handwheel.

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

The override increment is “0.001 mm”

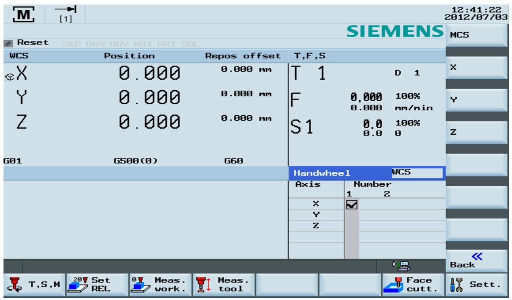
The override increment is “0.010 mm”

The override increment is “0.100 mm”

The selected axis can now be moved with the handwheel.

Press “JOG” on MCP to end the function of “Handwheel”.

Notes：If set the 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.



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

Tool Setup

SEQUENCE

Start spindle

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

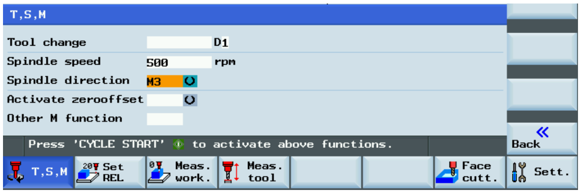
Start the spindle before adjusting tools as follows:

Press the “Machine” key on the PPU

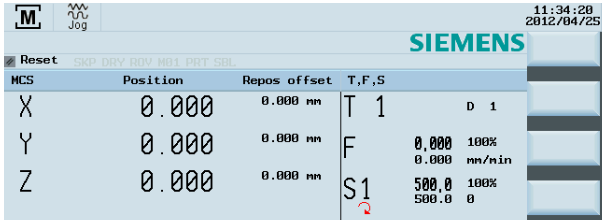
Press the “T.S.M” SK on the PPU

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

Measure tool

A tool must have been created and loaded before it can be measured!

Step 1 Measure length

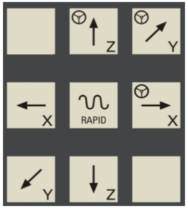
Press the “Machine” key on the PPU

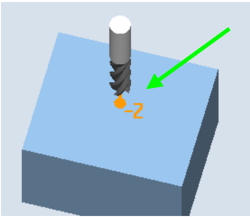
Press the “JOG” key on the MCP

Press the “Meas. tool” SK on the PPU

Press the “Measure manual” SK on the PPU

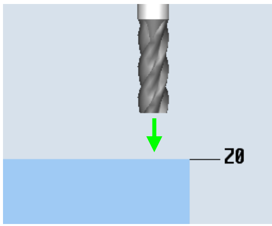
Press the axis keys on the MCP to move the tool to the set position above the workpiece.





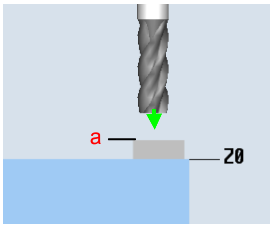
Note: The following text describes the required settings in the workpiece coordinate system
"X / Y / Z" zero points as: "X0" / "Y0" / "Z0"

Press the "Handwheel" key on the MPC and position the tool at location Z0 or a of the workpiece.



Move directly to zero point

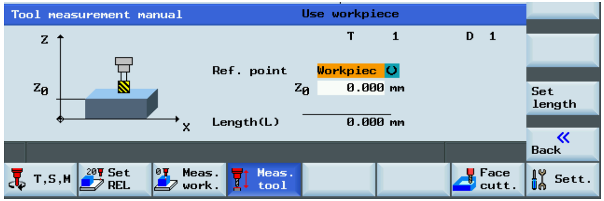
or



Use a setting block.

Use "SELECT" key to set the reference point as "workpiece" (In real measurement, the reference point can be set as either "workpiece" or " fixed point" if required.

Enter "0" for "Z0"
(If the setting block is used, then the value would be thickness a)



Press the "Set length" SK on the PPU

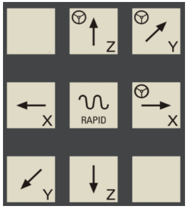
The measured tool length is now shown in "Length (L)". This value is also saved in the length value column of the corresponding tool list at the same time.

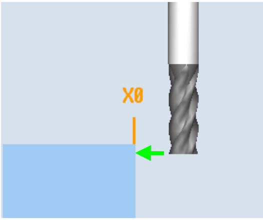
Step2 Measure diameter

Press the "Diameter" SK on the PPU

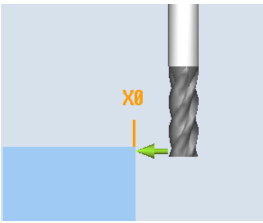
SEQUENCE

Press the axis keys on the MCP to move the tool to the set position.



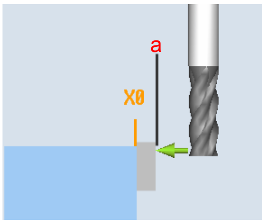


Press the "Handwheel" key on the MCP and position the tool at the location X0 or a of the workpiece.



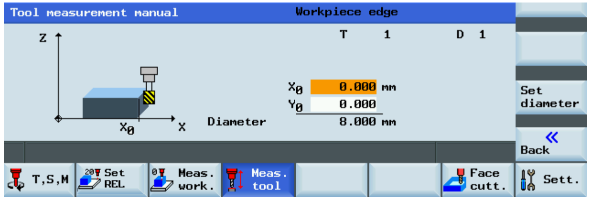
Move directly to zero point

or



Use a setting block.

Enter "0" at "X0"
Enter "0" at "Y0"
(This is the value of the width of a setting block if it is used. Select one of X0/Y0 according to requirement.)



Press the "Set diameter" SK on the PPU

Press the "Back" SK on the PPU

Tool Setup

SIEMENS

SEQUENCE

Jog spindle

A tool must be loaded to the spindle.

Press the “Machine” key on the PPU.

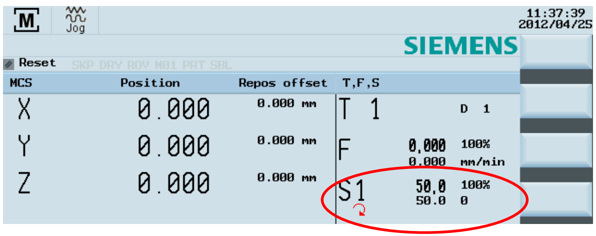
Press the “JOG” key on the MCP.

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.



Execute M function

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 button on MCP is active.

Press the “Reset” key on the MCP to stop the coolant function.

Press the “Back” SK on the PPU.

Notes

Notes

Workpiece Setup

Content

Unit Description

This unit describes how to set the workpiece offset and test the tool results.

Unit Content

Manual start spindle

Create workpiece offset

Test tool offset results

End

SEQUENCE

Manual start spindle

A tool must have been loaded into the spindle.

Before measuring, the spindle can be started as follows:

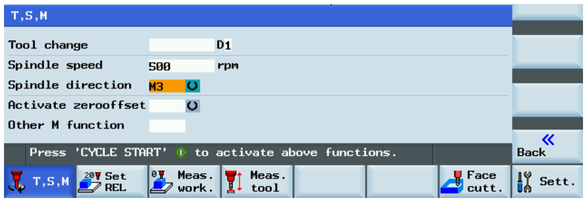
Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

Press the “T.S.M” SK on the PPU.

Enter “500” in “Spindle speed” on the PPU.

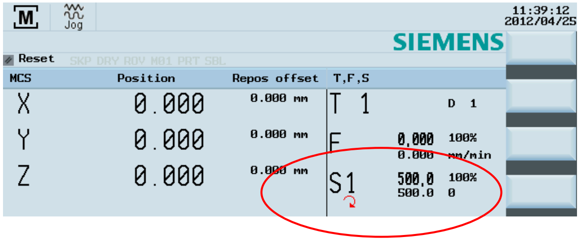
Select “M3” as the “Spindle direction” using the “Select” key on the PPU.



Press “CYCLE START” on the MCP.

SIEMENS

SEQUENCE



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

Press the “Back” SK on the PPU.

Create workpiece offset

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

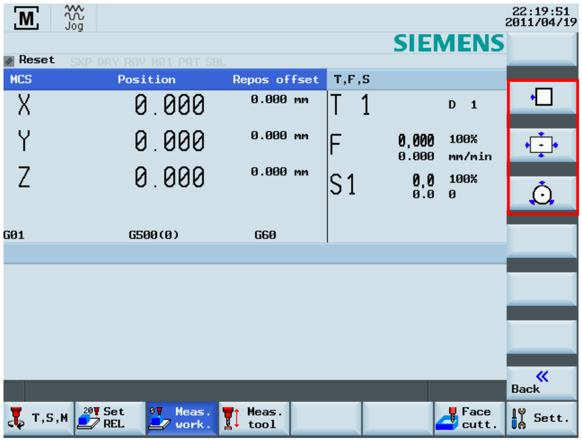
Make sure the active tool is the measured tool!

Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

Press the “Meas. work.” SK on the PPU.

As the following red frame shows, 808D provides the user with three methods of using tools to simplify the operating process.



Workpiece Setup

SEQUENCE

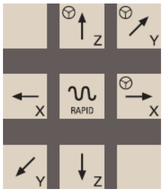
Method1 This method is normally for setting the zero point of the work-piece at the edge of the workpiece.

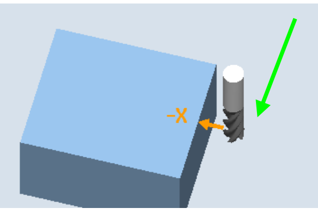
Using a tool that has a measured “Tool length & radius”, move the tool to a known position on the workpiece. Using either JOG or Handwheel, scratch an edge and then calculate the zero point of the workpiece. The process of setting the “X” zero point (“X0”) is described below.

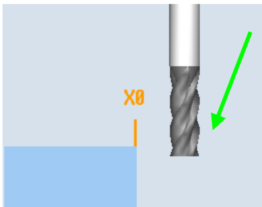
Press the corresponding SK of the first icon on the right-hand side of the PPU.

Press the appropriate SK to select the feed axis which needs to be set up.

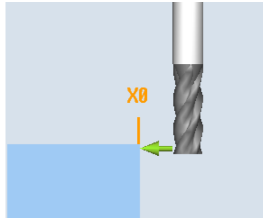
Press the axis traverse keys to move the tool to the required set-ting position in the X axis.



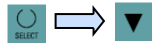




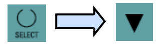
Press the “Handwheel” key on the MCP to position the tool at the X0 edge of the work-piece.



Select “Save in” Offset “G54” (or other offset).

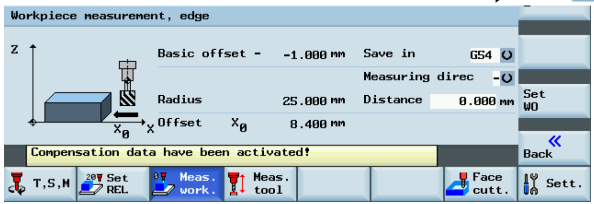


Select “Measuring direction” as “-”. (This value should be chosen according to realities)



Set “Distance” as “0”.

Press the “Set WO” SK on the PPU.



“Step 2” must be repeated for the setting of Y and Z zero points.
If you change the tool because of wear/damage during the machining process, you must remeasure the length of the tool.

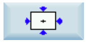
SIEMENS

SEQUENCE

Method 2 This method is normally used for setting the workpiece zero point at the center point of a rectangular workpiece.

Using tools with a measured “length and radius”, move them to the four edges of the rectangular workpiece. Using either JOG or Handwheel, scratch an edge and then calculate the zero point of the workpiece.

Press the corresponding SK of the second icon on the right-hand side of the PPU.



Observing the figure on the PPU, move the coordinate axis following the orange arrow to move the tool to the specified position and scratch the edge of the workpiece.

Press the ”Save P1” SK on the PPU to save the coordinate axis of the 1st position in the system.

Save P1

Repeat the process for positions 2, 3 and 4. (When the setting is complete, the buttons will be shown in blue.)

Save P1

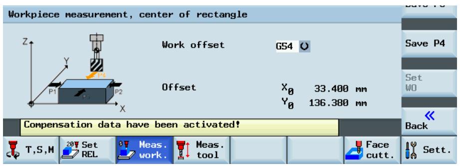
Save P2

Save P3

Save P4

Press the “Set WO” SK on the PPU.

Set WO



You have then finished setting the zero point of the workpiece as the center point of the rectangular workpiece.

Method 3 This method is normally used for setting the zero points at the center point of a circular workpiece.

Using tools with a measured “length and radius”, move them to the three edges of the circular workpiece. Using either JOG or Handwheel, scratch an edge and then calculate the zero point of the workpiece.

Press the corresponding SK of the third icon on the right-hand side of the PPU.

Observing the figure on the PPU, move the coordinate axis following the orange arrow to move the tool to the specified position and scratch the edge of the workpiece.

Press the ”Save P1” SK on the PPU to save the coordinate axis of the 1st position in the system.

Save P1

Repeat the process for positions 2 and 3. （When the setting is complete, the buttons will be shown in blue.）

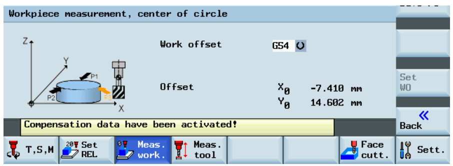
Save P1

Save P2

Save P3

Press the “Set WO” SK on the PPU.

Set WO



You have then finished setting the zero point of the workpiece as the center point of the circular workpiece.

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 "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) →

G54 (select offset panel as required)
T1 D1
G00 X0 Y0 Z5

Press the "ROV" key to ensure that the "ROV" function is activated (the function is activated when the light on the key is on). → 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. Observe whether the axis moves to the set position.

Notes

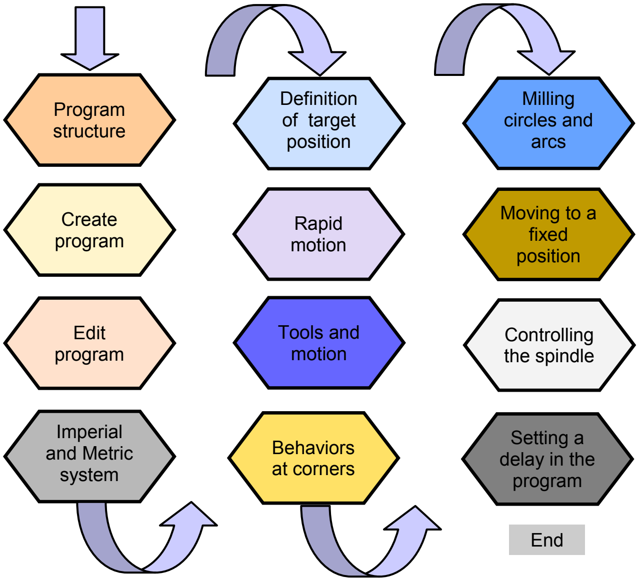
Create Part Program Part 1

Content

Unit Description

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

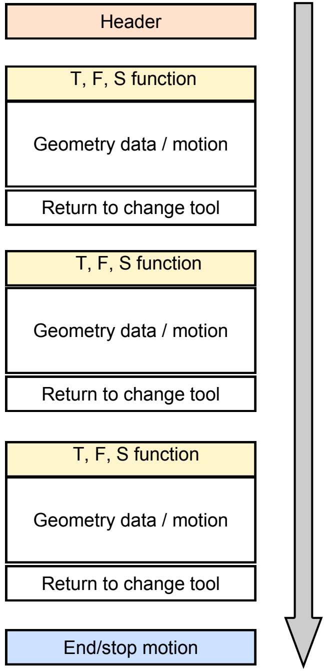
Unit Content



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:




```
N5 G17 G90 G54 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X100 Y100 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 D0

N40 T2 D1 M6
N45 S3000 M3 G94 F100
N50 G00 X50 Y50 Z5
N55 G01 Z-5
N60 Z5
N65 G00 Z500 D0

N70 T3 D1 M6
N75 S3000 M3 G94 F100
N80 G00 X50 Y50 Z5
N85 G01 Z-5
N90 Z5
N95 G00 Z500 D0

N100 G00 G40 G53 X0 Y0 Z500 D0
M30
```

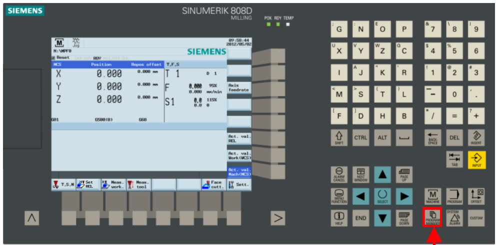
SEQUENCE

Create program

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

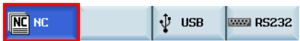
Step 1

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

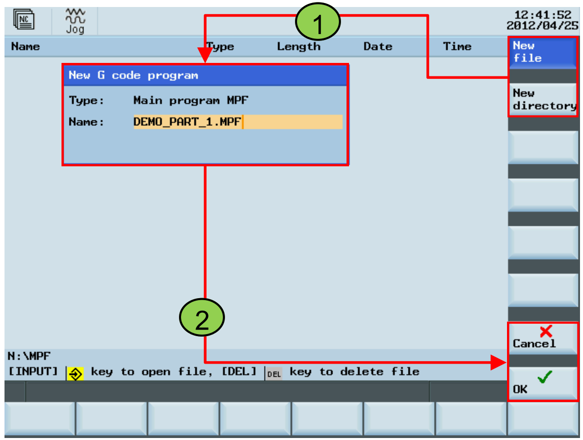
Create a new program with the “New” SK on the right of the PPU.

Step 4

You can choose “New” or “New directory”.

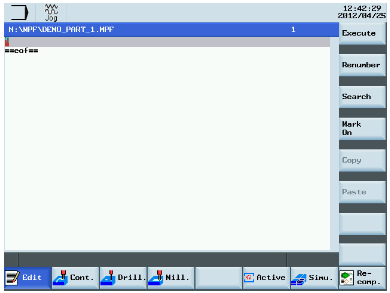
Choose “New” to create a program.

Choose “New directory” to create a file.



Step 5

Now the program is opened and can be edited.



The system will save it automatically after editing.

End

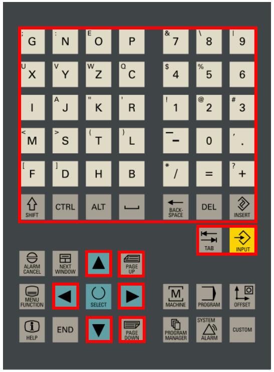
Create Part Program Part 1

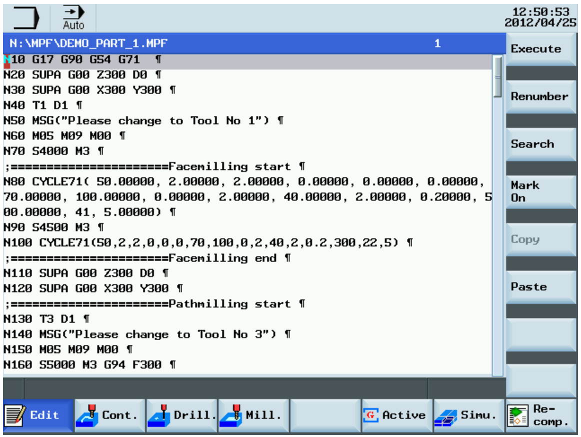
SIEMENS

Basic Theory

Edit program

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





Inches and mm

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

Header	N5 G17 G90 G54 G71
T, F, S function	N10 T1 D1 M6 N15 S5000 M3 G94 F300
Geometry data / motion	N20 G00 X100 Y100 Z5 N25 G01 Z-5 N30 Z5
Return to change tool	N35 G00 Z500 D0

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

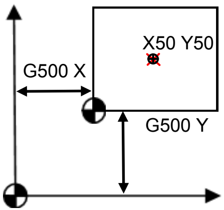
Header	N5 G17 G90 G54 G70
T, F, S function	N10 T1 D1 M6 N15 S5000 M3 G94 F300
Geometry data / motion	N20 G00 X3.93 Y3.93 Z5 N25 G01 Z-0.787 N30 Z0.196
Return to change tool	N35 G00 Z19.68 D0

Create Part Program Part 1

Basic Theory

Definition of target position

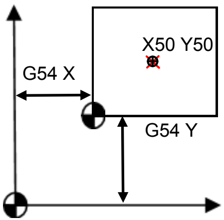
G500
All absolute path data will be relative to this position. The position is written in the G500 (basic) zero offset.




```
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-20
N30 Z5
N35 G00 Z500 D0
```

Or

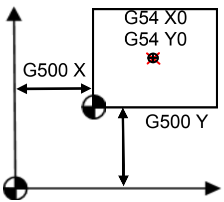
G54 G55 G56 G57 G58 G59
With G500 = 0, the offset for the workpiece can be stored in the G54 workpiece offset.




```
N5 G17 G90 G54 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X0 Y0 Z5
N25 G01 Z-20
N30 Z5
N35 G00 Z500 D0
```

Or

G500 + G54
With G500 unequal to 0 and be activated, the value in G500 will be added to the value in G54.




```
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 G54 X20 Y20 Z5
N25 G01 Z-20
N30 Z5
N35 G00 G53 Z500 D0
```

G90
Absolute positioning; with G90 at the header, 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 M6
N15 S5000 M3 G94 F300
N20 G00 X100 Y100 Z5
N25 G01 Z-20
N30 Z5
N35 G00 Z500 D0
```

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.

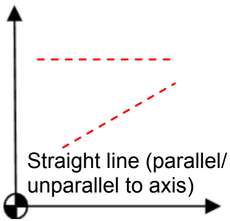
```
N5 G17 G90 G54 G70

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X3.93 Y3.93 Z0.196
N25 G01 G91 Z-0.787
N30 Z0.196
N35 G00 G90 Z19.68 D0
```

SIEMENS

Rapid motion

G00

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




```
N5 G17 G90 G54 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 D0
```

Tools and motion

T1 D1 M06

Using the "T" command, the new tool can be selected. The "D" command is used to activates the tool length offset.

M06 can be also used for machines with automatic tool changer.

```
N5 G17 G90 G54 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-20
N30 Z5
N35 G00 Z500 D0
```

- **Feedrate**
- **Spindle speed**
- **Feed type**
- **Spindle direction**

In the program, the feed rate is defined with "F". Two types of feed rate are available:

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

```
N5 G17 G90 G54 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 D0
```

G94

Defines the feed rate in terms of time (unit: mm/min).

G95

Defines the feed rate in terms of spindle revolutions (unit: mm/rev).

```
N5 G17 G90 G54 G71

N10 T1 D1 M6
N15 S5000 M3 G95 F0.3
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 D0
```

S

The spindle speed is defined with "S"

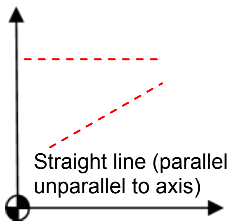
S5000

M3/M4

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

G01

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



Behavior at corners

Activation/ deactivation of the tool radius compensation when working on the part contour.

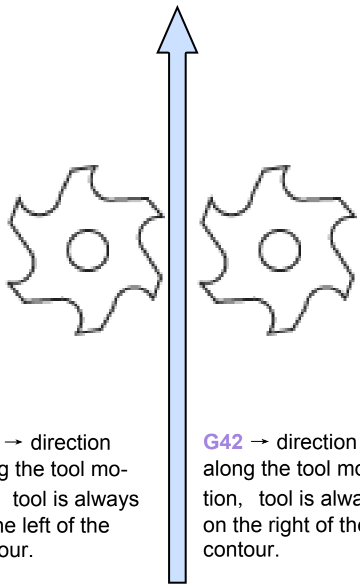
G41 / G42 and G40

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

G41： Compensation to left

G42： Compensation to right

G40： Compensation of the radius can be deactivated



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

G42 → direction along the tool motion, tool is always on the right of the contour.

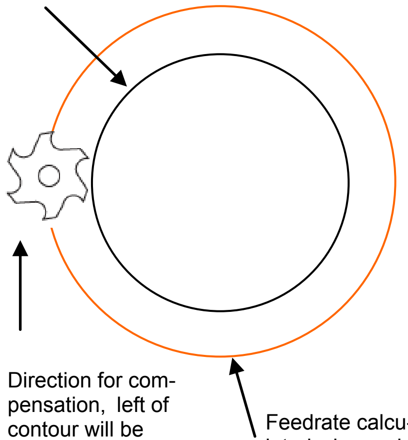
Arrow indicates the direction of tool motion along the contour.

When traversing circular contours with cutter radius compensation, it should be decided whether the feed rate should be calculated along the contour of the workpiece or along the path defined by the center point of the cutting tool.

When using a contour with a feed rate defined by the CFC code, the feed rate will be constant at the contour, but in some cases, it may cause increases in the feed rate of the tool.

This increase could damage the tool if excessive material is encountered at the contour; this function is normal for finish cutting of contours.

The CFTCP command ensures a constant feed rate, however a constant feed rate may not be ensured at the contour, which may cause deviations in surface finish.



The result of the two commands will be such that the cutter goes very fast around a corner or slow on the contour.

SIEMENS

Basic Theory

Milling 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 XY coordinate system, the interpolation parameters I and J are available.

```
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X-20 Y-20 Z5
N25 G01 Z-5
N30 G41 X0 Y0
N35 Y50
N40 X100
N45 G02 X125 Y15 I-12 J-35
N50 G01 Y0
N55 X0
N60 G40 X-20 Y-20
N35 G00 Z500 D0
```

Note:
N45 can also be written as follows
N45 G02 X125 Y15 CR=37

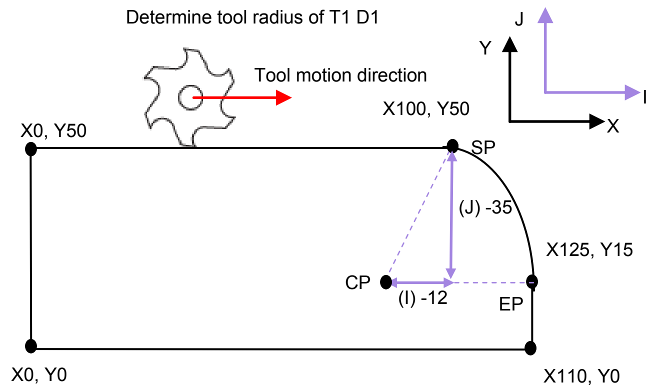
Two common types of defining circles and arcs:

①: G02/G03 X_Y_I_J_;
②: G02/G03 X_Y_CR=_;

Arcs ≤180°, CR is a positive number

Arcs >180°, CR is negative number

When milling circles, you can only use ① to define the program!



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

J = defined relative increment from start point to center point in Y

G2 = define circle direction in traversing direction = G2 clockwise

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

SIEMENS

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 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 Z=0 ;reference point
```

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

```
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 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.

```
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 M5
N35 Z5 M4
N40 M5
N45 M19
N50 G00 Z500 D0
```

Setting a delay in the program

G04 can be used to pause the tools' movements during operation

G04 F5： Program pause of 5 s
This makes the surface of the workpiece much smoother

```
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 G04 F5
N35 Z5 M4
N40 M5
N45 M19
N35 G00 Z500 D0
```

Notes

Notes

Create Part Program Part 2

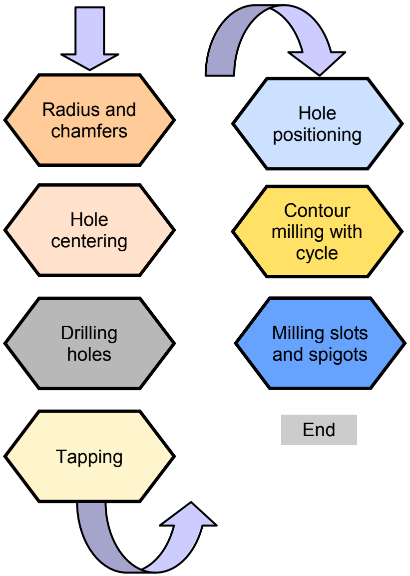
Content

Unit Description

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

Part 2

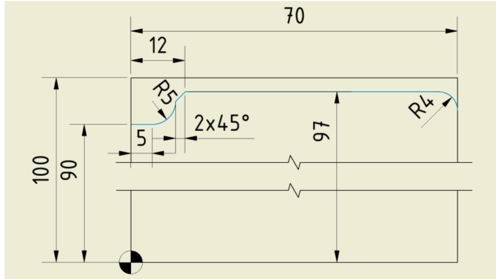
Unit Content



Basic Theory

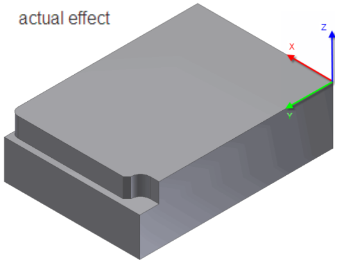
Radius and chamfers

The two radii and the chamfer shown in the diagram can be produced with the code marked in the program below.



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)

actual effect



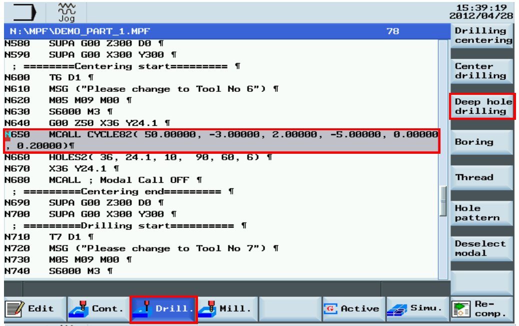
```
N55 SUPA G00 Z300 D0
N60 SUPA G00 X300 Y300
N65 T3 D1
N70 MSG("Please change to Tool No 3")
N75 M05 M09 M00

N80 S5000 M3 G94 F300
N85 G00 X-6 Y92
N90 G00 Z2
N95 G01 F300 Z-10
N100 G41 Y 90
N102 G01 X 5
N105 G01 X12 RND=5
N110 G01 Y97 CHR=2
N115 G01 X70 RND=4
N120 G01 Y90
N125 G01 G40 X80
N130 G00 Z50
```

Hole centering

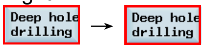
The easiest way to center drill a hole prior to drilling is to use either CYCLE81 or CYCLE82

CYCLE81: Without delay at current hole depth

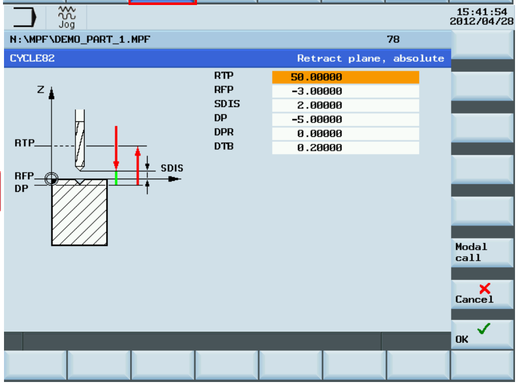
CYCLE82: With delay at current hole depth



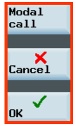
The relevant cycle can now be found using the vertical softkey on the right.



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



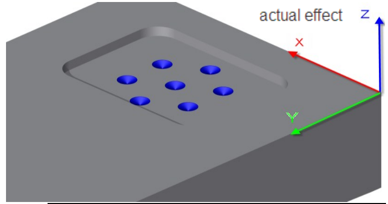
With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

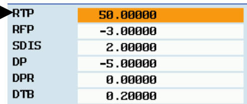


This will drill a hole at the current position.

With the Modal call SK, holes will be centered at subsequent programmed positions until cancelled with the MCALL command in the part program.

The information is transferred as shown below.





N325 MCALL CYCLE82(50.000, -3.000, 2.000, -5.000, 0.000, 0.200)
N330 X20 Y20 ; Hole will be centered
N335 X40 Y40 ; Hole will be centered
N340 MCALL
N345 X60 Y60 ; Hole will not be centered

Parameters	Meanings
RTP=50	Coordinate value of turning position is 50 （absolute）
RFP=-3	Coordinate value of hole edge starting position under workpiece zero point surface is 3 （absolute）
SDID=2 （frequently used values 2~5）	Safety distance, feed path changes from quick feed to machine feed 2 mm away from RFP face
DP=-5	Coordinate position of final drilling depth is -5 （absolute）
DTB=0.2	Delay of 0.2 s at final drilling depth

SIEMENS

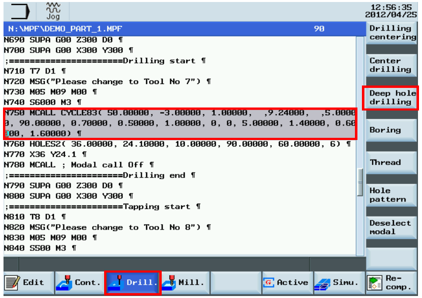
Basic Theory

Drilling holes

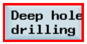
The easiest method to drill holes is with CYCLE81/82：Without/with delay at current hole depth

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

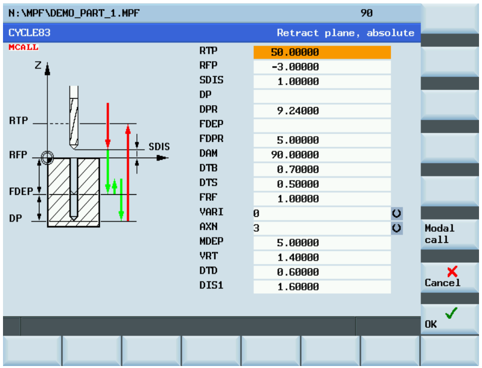
The cycle can be found and parameterized with the "Drill." SK.



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



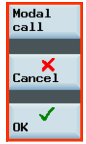
Select "Deep hole drilling" using the vertical SKs and parameterize the cycle according to requirements.

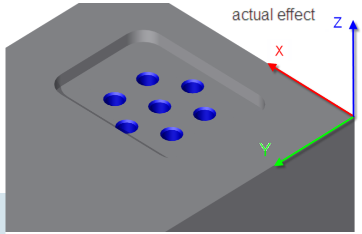


With the "OK" SK, the values and cycle call will be transferred to the part program as shown below. This will drill a hole at the current position.

With the "Modal call" SK, holes will be drilled at subsequently programmed positions until cancelled with the MCALL command in the part program.

The information is transferred as shown below.





RTP	50.00000
RFP	-3.00000
SDIS	1.00000
DP	
DPR	9.24000
FDEP	
FDPR	5.00000
DAM	90.00000
DTB	0.70000
DTS	0.50000
FRF	1.00000
VARI	0
AXN	3
MDEP	5.00000
VRT	1.40000
DTD	0.60000
DIS1	1.60000

For specific parameter commands, see the next page

N325 MCALL CYCLE83(50.00000, -3.00000, 1.00000, ,9.24000, ,5.00000, 90.00000, 0.70000, 0.50000, 1.00000, 0, 0, 5.00000, 1.40000, 0.60000, 1.60000)
N330 X20 Y20 ; Hole will be drilled
N335 X40 Y40 ; Hole will be drilled
N340 MCALL
N345 X60 Y60 ; Hole will not be drilled

Basic Theory

For descriptions of RTP, RFP, SDIS and DP, please see page 40		
FDEP=5	Reach first drilling hole depth. Z axis coordinate is -5 (absolute coordinate value)	
FDPR=5	From the reference plane, drill downwards 5mm	
DAM=90	Decrement is 90	
DTB=0.7	Pause 0.7 s during final tapping of thread depth (discontinuous cutting)	DTB <0: Unit is r
DTS=0.5	Stops at the start position for 0.5 s (for VARI=1, removal active)	DTS <0: Unit is r
FRF=1 (range:0.001~1)	Original effective feed rate remains unchanged	Feed rate modulus
VARI=0	Interruption in drilling is active	VARI=1 retraction of active quill back to reference plane
AXN=3	AXN is tool axis, under appointed G17 use Z axis	The value of AXN decides which axis to use
MDEP=5	Minimal drilling depth 5 mm	This parameter activates only when DAM <0
VRT=1.4	Interruption in drilling, the retraction value of the quill is 1.4 mm	VRT=0 → retraction value is 1mm VRT>0 → retraction value is appointed value
DTD=0.6	Pauses at the position of final drilling depth for 0.6 s	DTD <0:unit is r, DTD =0:same as DTB
DIS1=1.6	When reinserting a quill, you can program a distance limit of 1.6 mm	For specific explanations please refer to the standard handbook

DAM parameter

①DAM≠0, the first drilling operation（FDPR）cannot exceed the drilling depth. As of the second drilling operation, the drilling is acquired from the last depth operation（drilling depth=last drilling depth-DAM）. The calculated drilling must be >DAM. If the calculated drilling is ≤DAM, as of the next feed, the DAM value will be the feed depth until the end of the feed. If the last remaining depth is <DAM, then drilling is performed automatically until the required depth is reached.

②DAM=0, drilling depth each time is same as the 1st drilling depth（FDPR）,
In case the residual depth <2xFDPR, the last 2 cutting depth are half of the residual depth.

Example：40 mm deep hole as an example, with DAM=2 mm and DAM=0 mm feed

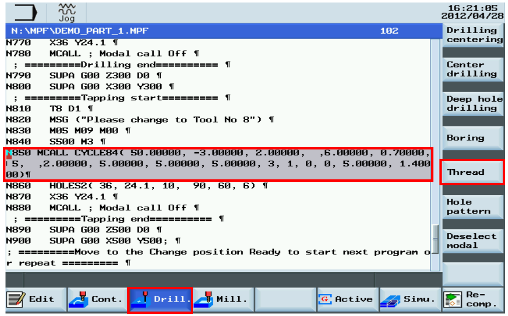
Feed times	Every feed depth/ mm DAM=2	Actual depth/ mm	Feed times	Every feed depth/ mm DAM=0	Actual depth/ mm
1.	FDPR=10	-10	1.	FDPR=10	-10
2.	FDPR-DAM=10-2=8	-18	2.	FDPR=10	-20
3.	(FDPR-DAM)-DAM =8-2=6	-24	3.	FDPR=10	-30
4.	(FDPR-2DAM)-DAM =6-2=4	-28	Remaining depth =10＜2xFDPR, the remaining depth distribute by the last two drilling		
5.	(FDPR-3DAM)-DAM =4-2=2	-30	4.	5	-35
6.	DAM=2	-32	5.	5	-40
7.	DAM=2	-34	6.		
8.	DAM=2	-36	7.		
9.	DAM=2	-38	8.		
10.	DAM=2	-40	9.		

Create Part Program Part 2

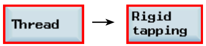
Basic Theory

Tapping

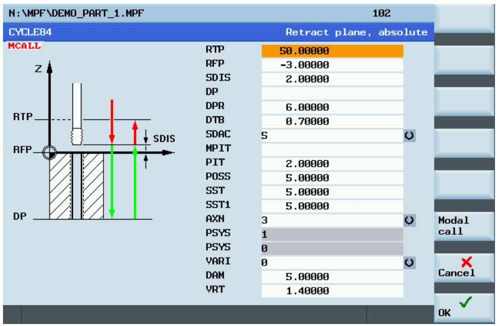
The easiest way to tap a hole is to use CYCLE84: Solid tap holder CYCLE840: With floating tap holder. The cycles can be found and parameterized using the "Drill." SK.



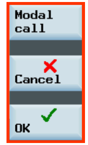
The relevant cycle can now be found using the vertical SKs on the right.



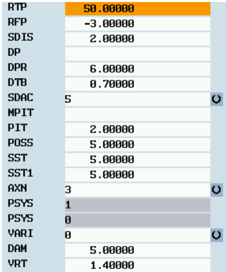
Select "Thread" using the vertical SKs ,and then select "Rigid tapping", and parameterize the cycle according to requirement.



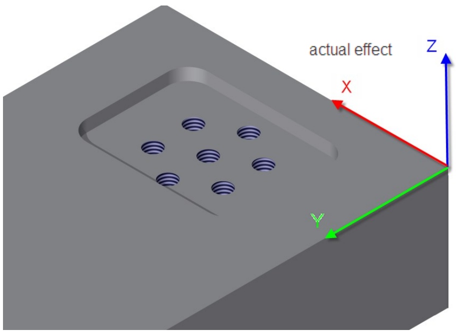
With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.
This will drill a hole at the current position.
If there is no other operation, the machine will drill holes in the current position.
With the "Modal call" SK, holes will be tapped at subsequently programmed positions until cancelled with the MCALL command in the part program.
Examples are shown on the next page .



Basic Theory



N325 MCALL CYCLE84(50.00000, -3.00000, 2.00000, ,6.00000, 0.70000, 5, ,2.00000, 5.00000, 5.00000, 5.00000, 3, 0, 0, 0, 5.00000, 1.40000)
N330 X20 Y20 ; Hole will be tapped
N335 X40 Y40 ; Hole will be tapped
N340 MCALL
N345 X60 Y60 ; Hole will not be tapped



For descriptions of RTP, RFP, SDIS, DP and DTB, please see page 40

For descriptions of AXH, VARI, DAM and VRT, please see page 42

Parameters	Meanings	Remarks
DTB=0.7	Pause 0.7 s during final tapping to thread depth (discontinuous cutting)	
SDAC=5	Spindle state after cycle is M5	Enter values 3/4→M3/M4
PIT=2（Range of values:0.001~2000 mm）	Right hand thread with 2mm pitch	Evaluate value→left hand thread
POSS=5	Spindle stops at 5º （unit: º）	
SST=5	Tapping thread spindle speed is 5 r/min	
SST1=5	Retraction spindle speed is 5 r/ min	Direction is opposite to SST SST1=0 →speed is same as SST

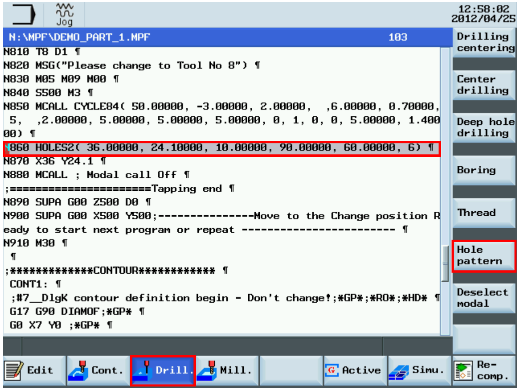
SST and SST1 control the spindle speed and the Z axis feed position synchronously. During execution of CYCLE 84, the switches of the feed rate override and the cycle stop (feed hold) are deactivated.

Create Part Program Part 2

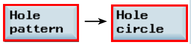
SIEMENS

Hole positioning

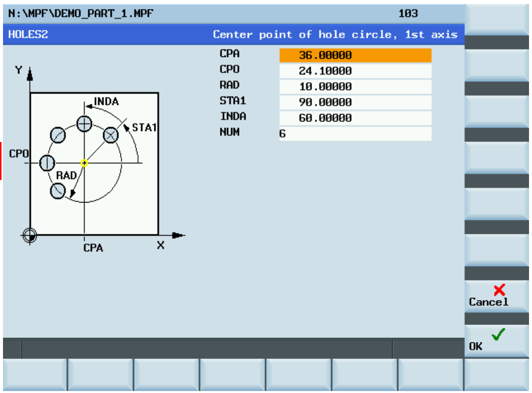
The easiest way to drill a series of holes is to use the pre-defined “Hole pattern” cycles. The cycles can be found and parameterized via the “Drill.” SK.



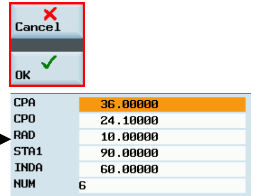
The relevant cycle can now be found using the vertical SKs on the right.



Select “Hole pattern” using the vertical SKs ,and then select “Hole circle”, and parameterize the cycle according to requirement.



With the “OK” SK, the values and cycle call will be transferred to the part program as shown below. This will drill holes at the positions defined from within the cycle.



N325 MCALL CYCLE82(50.00000, -3.00000, 2.00000, -5.00000, 0.00000, 0.20000)
N330 HOLES2(36.00000, 24.10000, 10.00000, 90.00000, 60.00000, 6)
N335 X36 Y24.1
N340 MCALL ; Modal Call OFF

Parameters	Meanings
CPA=36	Center of hole circle horizontal coordinate is 36 (absolute value)
CPO=24.1	Center of hole circle horizontal coordinate is 24.1 (absolute value)
RAD=10	Circle radius is 10 mm
STA1=90	Angle between the circle and horizontal coordinate is 90°
INDA=60	Angle between the circles is 60°
NUM=6	Drill 6 holes on circle
⚠	The cycle is used together with the drilling fixed cycle to decrease the hole clearance

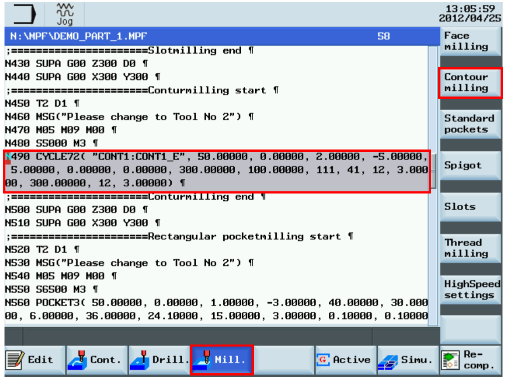
Create Part Program Part 2

SIEMENS

Basic Theory

Contour milling with cycle

The easiest way to rough and finish around a contour is to use the contour milling function.
The cycle can be found and parameterized via the “Mill.” SK.

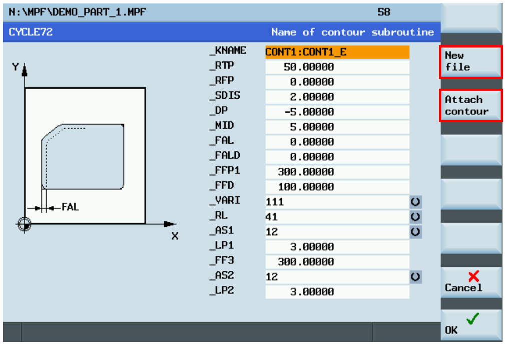


Mill.

The “Contour milling” SK can be found in the vertical SKs on the right.

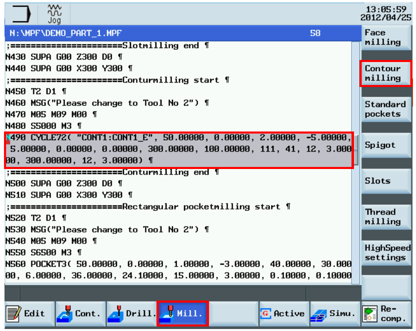
Contour milling

The parameterization is performed as in this figure.



Attach contour

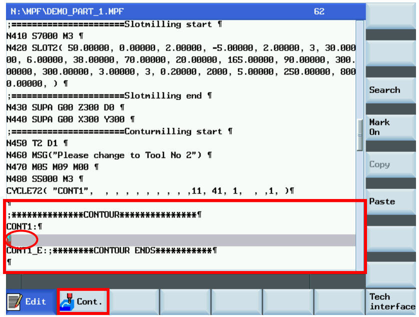
The contour can be edited and stored in the main program file after the M30 command when using the “Attach contour” SK.



Enter the cycle data setting according to the former operations in the screen and enter the name of the contour subprogram.

Press the “Attach contour” SK on the PPU. This will create a “contour information segment” at the end of the program, and the cursor will move to the writing position automatically.

Attach contour



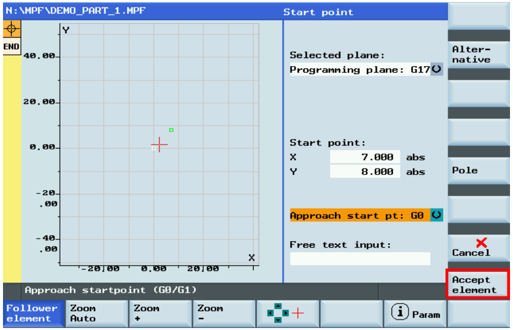
Make sure that the cursor has moved to the contour writing position (as shown in the figure).

Press the “Cont.” SK on the PPU to open the screen for setting the contour data.

Cont.

Basic Theory

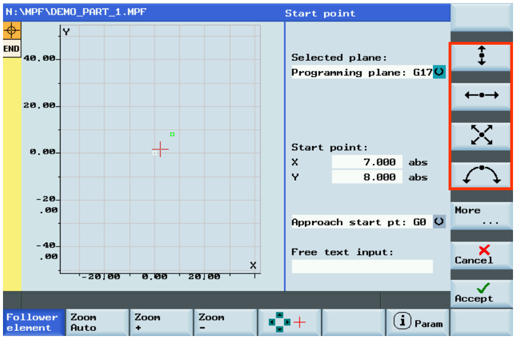
When you have opened the screen for setting the contour data, you can make the following settings:



Enter appropriate start point coordinates as in the machining figure and select the correct approach.

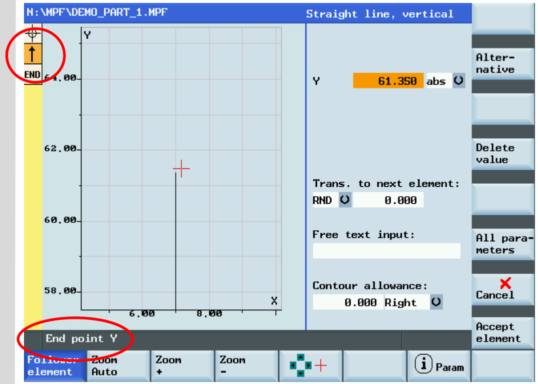
Press the “Accept element” SK on the PPU.

Accept element



Use the arrows on the PPU to select the direction and the shape of the contour milling.

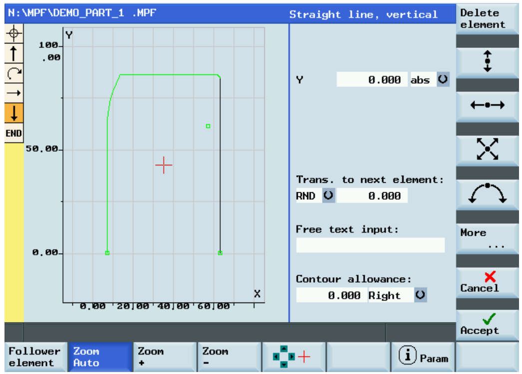
Enter the corresponding coordinate parameters.



The selected direction is shown at the top left side of PPU.

The meanings of the highlighted positions is shown at the bottom of the PPU screen.

Press the “Accept element” SK on the PPU.



Select different items (SKs) to set the contour until finishing editing the whole shape of the contour.

Press ”accept” SK on PPU to input the contour information in the main program

SIEMENS

Basic Theory

After completing the steps, the system will return to the edit interface. Press "Technical interface" on the PPU to return to the interface for setting the cycle data.

Tech interface

After finishing the parameter settings of CYCLE72, press the "OK" SK on the PPU to insert the corresponding cycles in the main program.

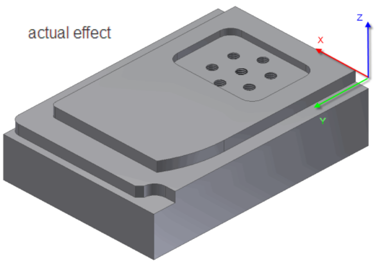
Cancel

OK

New

With the "New" SK and "Contour milling", the operation can be edited and saved in a subprogram.
The editing in the subprogram is the same as above.

With the "OK" SK, the values and the cycle call are transferred to the part program as shown below.



_KNAME	CONT1:CONT1_E
_RTP	50.00000
_RFP	0.00000
_SDIS	2.00000
_DP	-5.00000
_MID	5.00000
_FAL	0.00000
_FALD	0.00000
_FFP1	300.00000
_FFD	100.00000
_VARI	111
_RL	41
_AS1	12
_LP1	3.00000
_FF3	300.00000
_AS2	12
_LP2	3.00000

N245 CYCLE72("CONT1:CONT1_E", 50.00000, 0.00000, 2.00000, -5.00000, 5.00000, 0.00000, 0.00000, 300.00000, 100.00000, 111, 41, 12, 3.00000, 300.00000, 12, 3.00000)

For descriptions of RTP, RFP, SDIS and DP, please see page 40

Parameters	Meanings	Remarks
KNAME= CONT1:CONT1_E	Set the name of the contour subprogram as "CONT1" (":CONT1_E" is automatically created)	The first two positions of the program name must be letters
MID=5	The maximal feed depth is 5 mm	
FAL=0	Finishing allowance at the contour side is 0 mm	
FALD=0	Finishing allowance at the bottom plane is 0 mm	
FFP1=300	Tool feed rate on plane is 300 mm/min	
FFD=100	Feed rate after inserting the tool in the material is 100 mm/min	
VARI=111	Use G1 to perform rough machining, and back to the depth defined by the RTP+SDIS at the completion of the contour	For other parameters, please refer to the standard manual
RL=41 (absolute value)	PL=41→use G41 to make tool compensation on the left side of the contour	PL=40→G40, PL=42→G42
AS1=12	Approach the contour along the 1/4 circle on the path in space	For other parameters, please refer to the standard manual
LP1=3	The radius of the approaching circle is 20 mm	The length of the approaching path is along the line to approach
FF3=300	The feed rate during retraction of the path is 300 mm/min	
AS2=12	Return along the 1/4 circle on the path in space	Parameter explanations are the same as for AS1
LP2=3	The radius of the return circle is 20 mm	The length of the returning path is along the line to approach

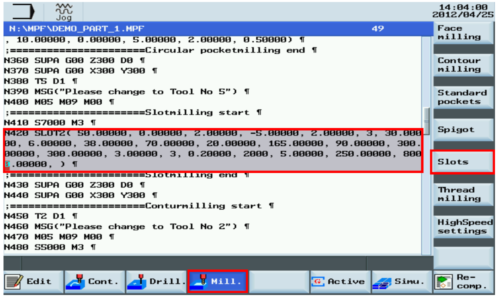
Create Part Program Part 2

SIEMENS

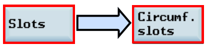
Basic Theory

Milling slots and spigots

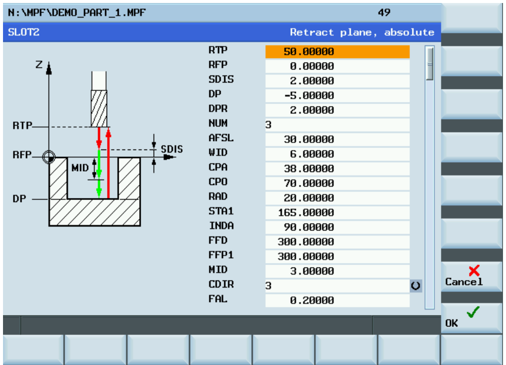
The easiest way to mill a slot is to use the SLOT2 cycle. The cycle can be found and parameterized via the “Mill.” SK.



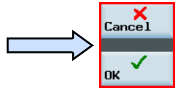
The relevant cycle can be found using the vertical SKs on the right.

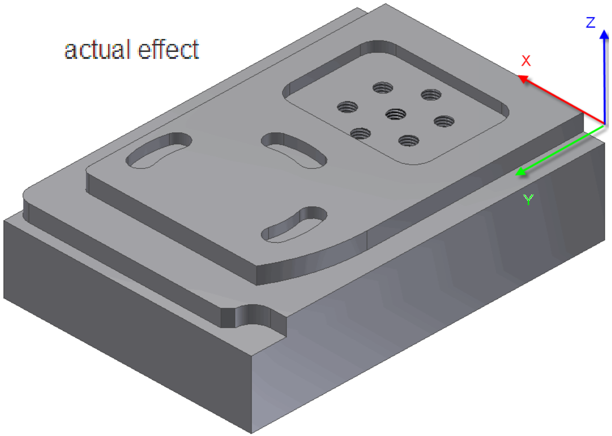


Select “slot” using the vertical SKs and parameterize the cycle according to requirement.



With the “OK” SK, the values and cycle call will be transferred to the part program as shown below.
This will perform milling at the position defined in the cycle.





Basic Theory

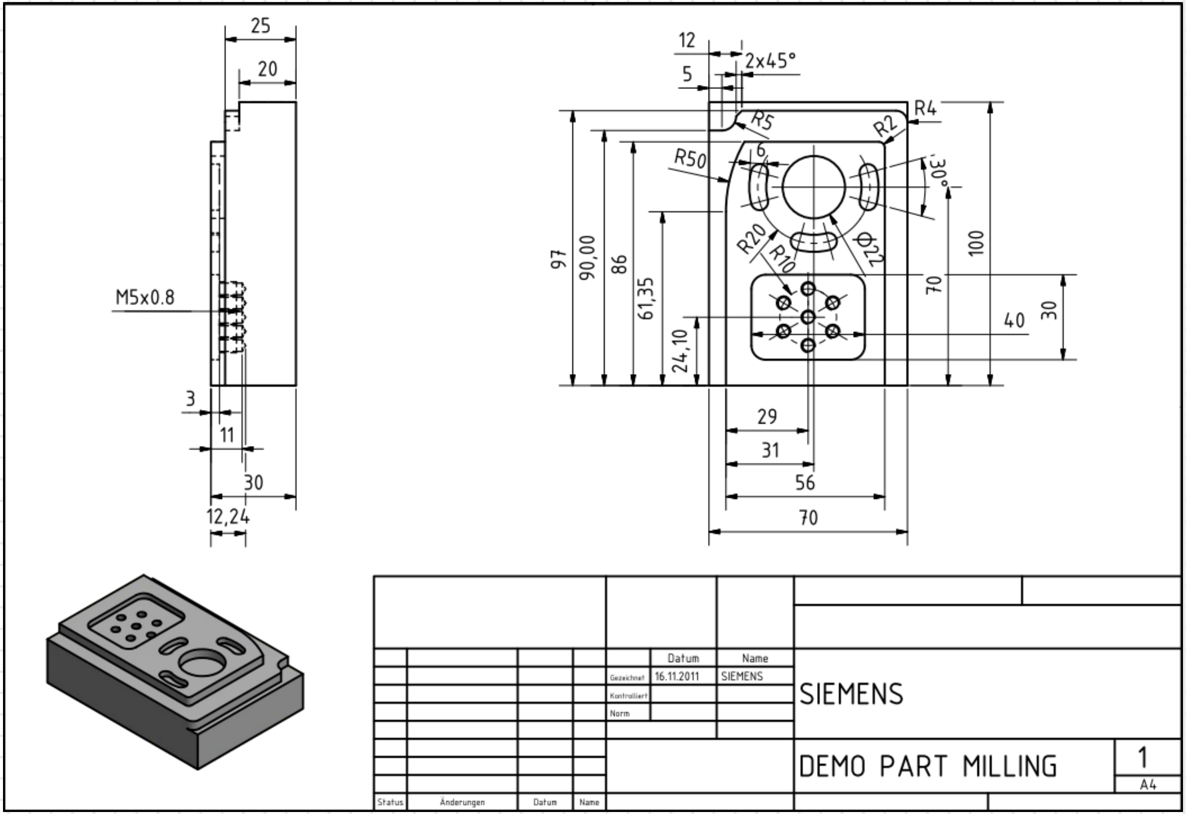
RTP	50.00000
RFP	0.00000
SDIS	2.00000
DP	
DPR	3.00000
NUM	3
AFSL	30.00000
WID	6.00000
CPA	38.00000
CPO	70.00000
RAD	20.00000
STA1	165.00000
INDA	90.00000
FFD	300.00000
FFP1	300.00000
MID	3.00000
CDIR	3
FAL	0.20000
VARI	0
MIDF	5.00000
FFP2	250.00000
SSF	3000.00000
FFCP	

N210 SLOT2(50.00000, 0.00000, 2.00000, , 3.00000, 3, 30.00000, 6.00000, 38.00000, 70.00000, 20.00000, 165.00000, 90.00000, 300.00000, 300.00000, 3.00000, 3, 0.20000, 2000, 5.00000, 250.00000, 3000.00000,)

For descriptions of RTP, RFP, SDIS, DP and DPR, please see page 40
For descriptions of CPA, CPO and RAD, please see page 45
For descriptions of FFD and FFP1, please see page 48

Parameters	Meanings	Remarks
NUM=3	Three slots on the circle	
AFSL=30	Angle slot length is 30°	⚠ AFSL and WID jointly decide the shape of the slot in the plane
WID=6	Slot width is 6 mm	
STA1=165	Start angle, angle between the effective work piece horizontal coordinate in positive direction and the first circle slot is 165°	
INDA=90	Incremental angle, angle between the slots is 90°	INDA=0, cycle will calculate the incremental angle automatically
MID=3	Maximal depth of one feed is 3 mm	MID=0 → complete the cutting of the slot depth
CDIR=3	Milling direction G3 (in negative direction)	Evaluate value 2→use G2 (in positive direction)
FAL=0.2	Slot side, finishing allowance is 0.2 mm	
VARI=0	The type of machining is complete machining	VARI=1→roughing VARI=2→finishing
MIDF=5	Maximal feed depth of the finishing is 5 mm	
FFP2=250	Feed rate of finishing is 250 mm/min	
SSF=3000	Spindle speed for finishing is 3000 mm/min	
⚠ If FFP2/SSF are not specified, then use the feed rate/spindle speed of rotation as default		
FFCP=	Feed rate at the center position on the circle path , unit is mm/min	
⚠ Before recalling the cycle, you must set the tool radius compensation value.		

Notes


25
20
M5x0.8
3
11
30
12,24
12
5
2x45°
R4
R5
R2
R50
6
30°
97
90,00
86
61,35
24,10
R20
R10
Φ22
100
70
40
30
29
31
56
70
Datum
Name
Gezeichnet
16.11.2011
SIEMENS
Kontrolliert
Norm
SIEMENS
DEMO PART MILLING
1
A4
Status
Änderungen
Datum
Name

Notes

SIEMENS

Content

SEQUENCE

Module Description

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

Module Content

Simulate program (Axis do not move)

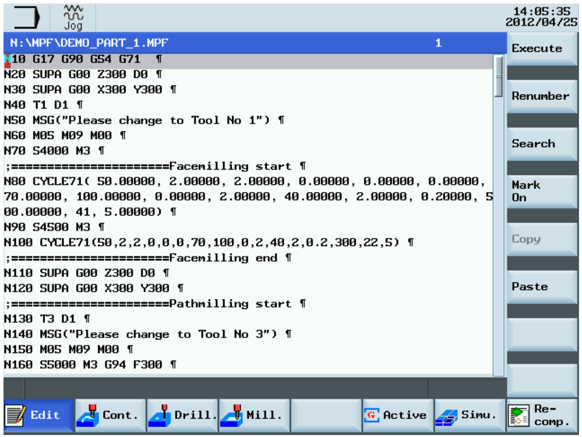
End

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" on PPU.

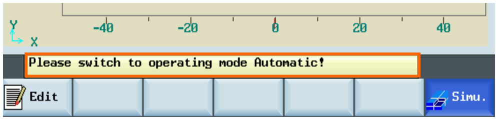


Simulate Program

SEQUENCE

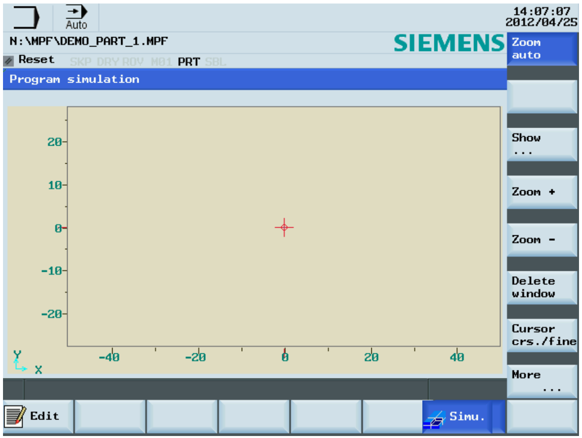
Step 2

Press the “Simu.” SK on the PPU.



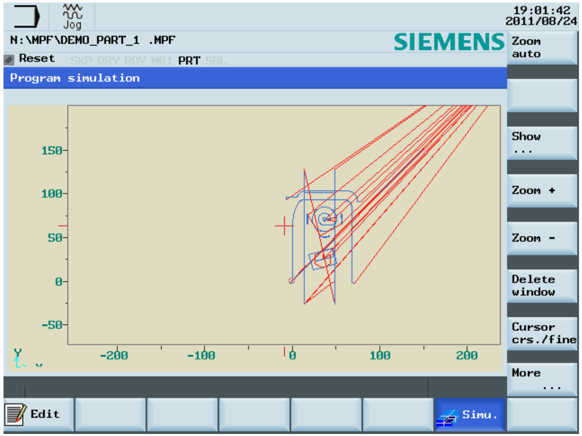
If the control is not in the correct mode, a message will be displayed at the bottom of the screen.

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.

End

Notes

Notes

Test Program

Content

Unit Description

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

Unit Content

Program Execution

Dry Run

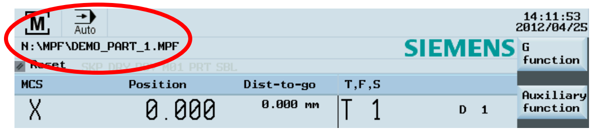
End

SEQUENCE

Program Execution

Before the part program can be loaded and executed in AUTO mode, it must be tested using the simulation function mentioned previously!

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!

Test Program

SEQUENCE

Dry Run

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

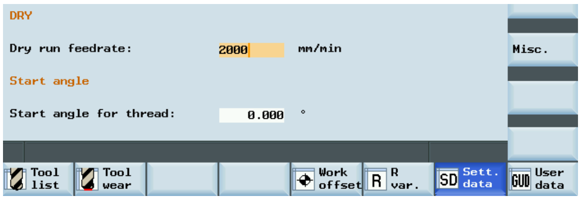
The data in the "Dry run feedrate" must first be set and checked!

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.

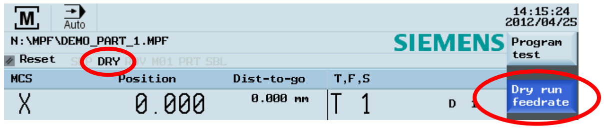


Press the "Input" key of the PPU.

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

Machine Pieces

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

Time Counter

PieceMachining

Tool Wear

End

Time Counter

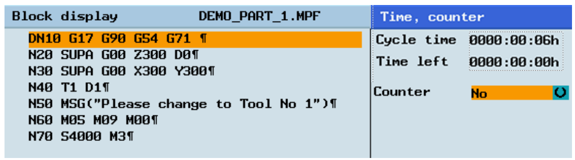
Make sure the machine has been referenced before machining workpieces !

Step 1

Press the “Machine” key on the PPU.

Press the “Auto” key on the MCP.

Press the “Time counter” SK on the PPU.



“Cycle time” shows how long the program has been running. → Cycle time 0000:00:06h

“Time left” shows how much time remains before the program ends. → Time left 0000:00:00h

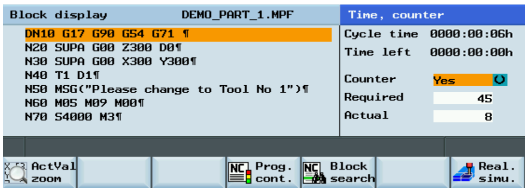
Step 2

The “Time left” 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). → SELECT

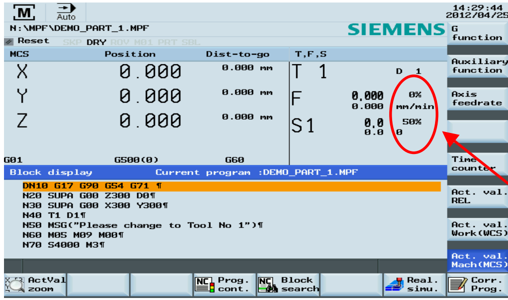
Enter the number of workpieces you require to be machined in “Required”. → Required 45

“Actual” shows the number of workpieces that have been machined. → Actual 8



Machine Pieces

Make sure the program is correct before machining pieces!

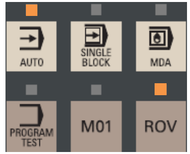


Set the program in the ready-to-start status as shown on the left in accordance with the “Program execution” sequences.

Perform the relevant safety precautions!

Make sure that only “AUTO” mode and “ROV” mode are activated

(or select the M01 function if required).



Notes: 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.) → DOOR

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

Turn the feedrate override gradually to the required value.

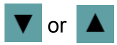
Tool Wear

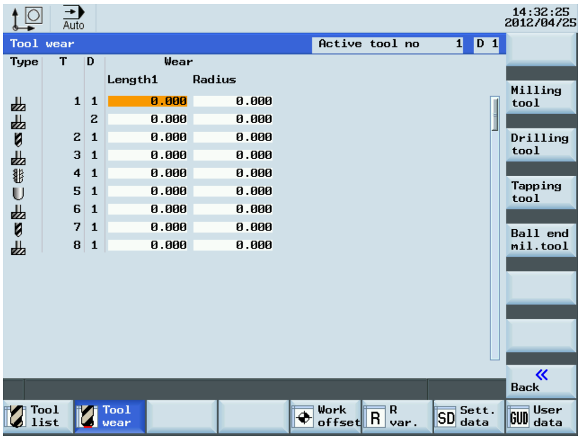
The tool wear compensation must distinguish the direction of compensation clearly!

Step 1

Press the “Offset” key on the PPU. 

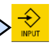
Press the “Tool wear” SK on the PPU. 

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

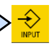


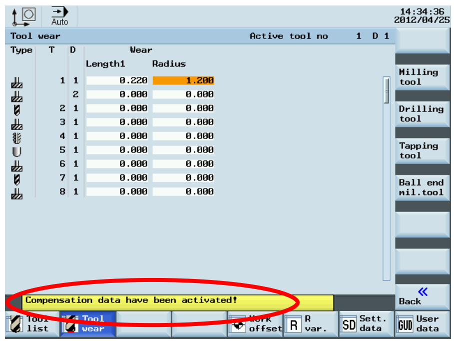
Step 2

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: tool is away from workpiece (set radius bigger than real one)
Negative value: tool is close to workpiece (set radius smaller than real one)

Press “Input” on the PPU to activate the compensation. 



Notes

Program Restart

SIEMENS

Content

Unit Description

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

Unit Content

Block Search

End

SEQUENCE

Block Search

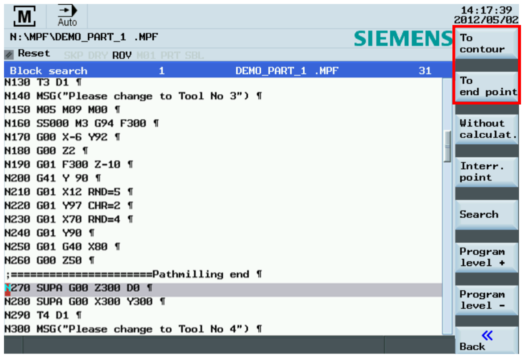
Press the “Machine” key on the PPU.

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.



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

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

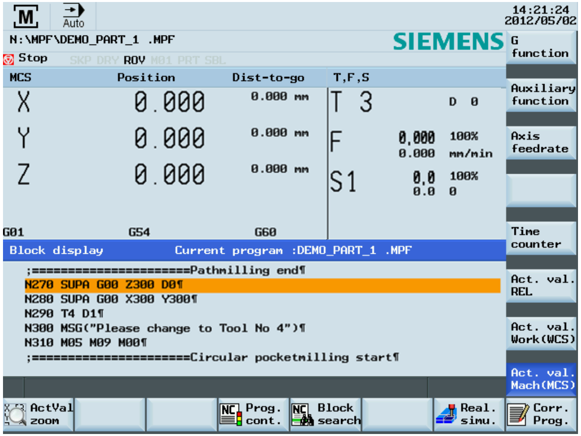
“To end point”: The program will continue from the line with the breakpoint.

Press the “To end point” SK on the PPU.
(can also press “To contour” if required)

Program Restart

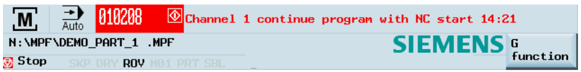
SEQUENCE

Press the “Back” SK on the PPU.



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

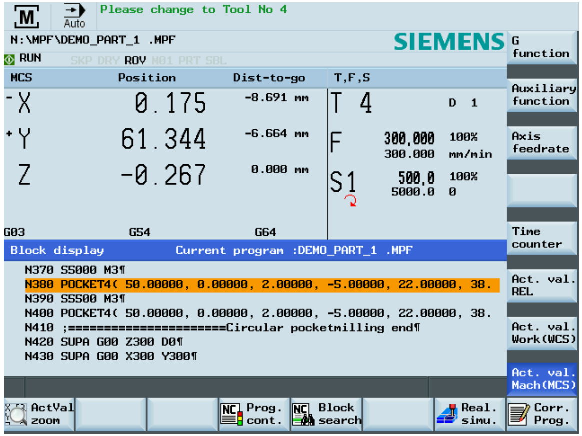
Press the “CYCLE START” key on the MCP to execute the program.



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

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

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



Notes

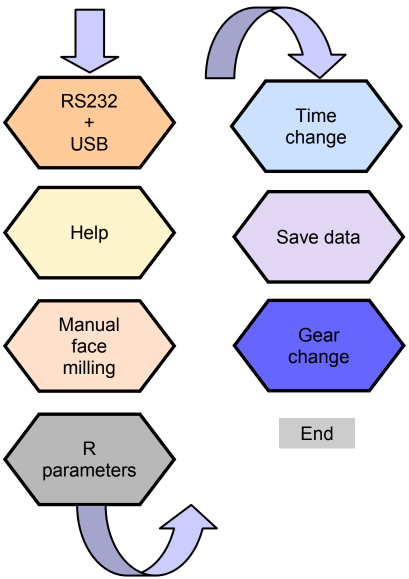
Notes

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

RS232 + USB

RS232 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 the PC.

Press “Program Manager” on the PPU. → PROGRAM MANAGER

Press the “RS232” SK on the PPU. → RS232

Press the “Settings” SK on the PPU. → Settings

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 transmis.	1a
Confirm overwrite	N

Press the “Save” SK on the PPU. → Save

Press the “Back” SK on the PPU. → Back

SEQUENCE

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

Press the "NC" SK on the PPU. → NC

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

Press the "Copy" SK on the PPU. → Copy

Press the "RS232" SK on the PPU. → RS232

Check the interface setting and start the communication software to receive the program on PC.
(Press "Receive Data" on SINUCOM PCIN to start the receive function.)

Press the "Send" SK on the PPU. → Send

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

```
Sending of data                19200,8,1,NONE
File:   _N_DEMO_PART_1 _MPF
From    N:
To      RS232
Bytes:  4095
```

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

```
RS232 communications are interrupted!
Please check the cable and the receive
program.
Continue transmission?
```

You can continue sending the part program.

Press the "OK" SK on the PPU. → OK

Or you can abort the sending of the part program

Press the "Cancel" SK on the PPU. → Cancel

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

Press "Program Manager" on the PPU. → PROGRAM MANAGER

Press the "RS232" SK on the PPU. → RS232

Press the "Accept" SK on the PPU. → Accept

```
Receiving of data              19200,8,1,NONE
File:
From    RS232
To
Bytes:
```

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.

```
Receiving of data              19200,8,1,NONE
File:   _N_DEMO_PART_1_MPF
From    RS232
To      /_N_MPF_DIR
Bytes:  903
```

SIEMENS

SEQUENCE

"USB" is used to transfer the programs to and from the NC.

Step 4 Use the "Copy" and "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. → NC

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

Press the "Copy" SK on the PPU. → Copy

Press the "USB" SK on the PPU. → USB

Press the "Paste" SK on the → Paste

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. → USB

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

Press the "Copy" SK on the PPU. → Copy

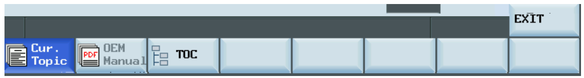
Press the "NC" SK on the PPU. → NC

Press the "Paste" SK on the → Paste

Help

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

Press the "Help" key on the PPU. → HELP



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

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

Press the "OEM Manual" SK on the PPU → OEM Manual

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

Press the "TOC" SK on the PPU. → TOC

The online help from the Siemens manual will be shown.

SIEMENS

SEQUENCE

Manual face cutting

“Face cutting” is used to cut the oversized materials on the rough face before starting to machine.

Step 1

Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

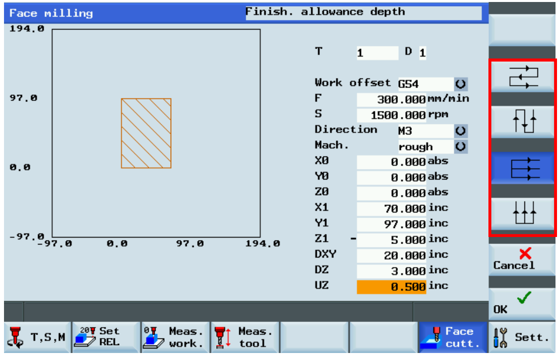
Press the “Sett.” SK on the PPU.

Enter appropriate values in “Retraction plane” and “Safety distance”.

Press the “Input” key on the PPU to activate the settings.

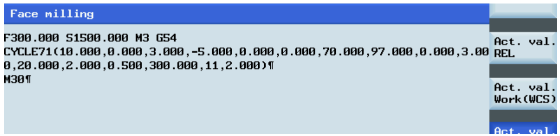
Step 2

Press “Face cutt.” SK on PPU.



Enter appropriate data in the ”Face Milling” window according to the machining requirement. Use the button on the right side of the PPU to select the cutting path of the tool during machining.

Press the “OK” SK on the PPU



The system now automatically creates the programs.

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

Press the “Cycle Start” key on the MCP.

Adjust the override on the MCP gradually to the required values.

SIEMENS

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:

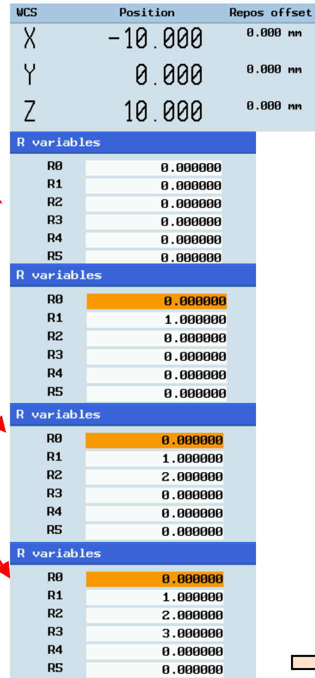
Reprocessing stop

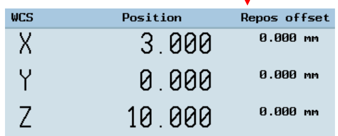
Programming the STOPRE command in a block will stop block preprocessing and buffering. The following block is not executed until all preprocessed and saved blocks have been executed in full. The preceding block is stopped in exact stop (as with G9).

Press the "Offset" key on the PPU.

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


```
N10 G17 G90 G54
N20 T1 D1
N30 S2500 M03 M08
N40 G00 X-10.0 Y0 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
```





SEQUENCE

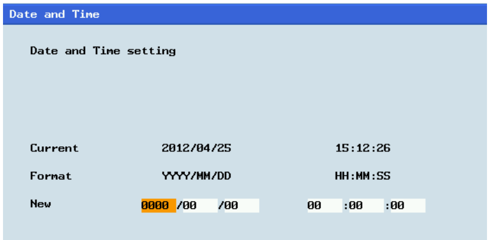
Time change

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

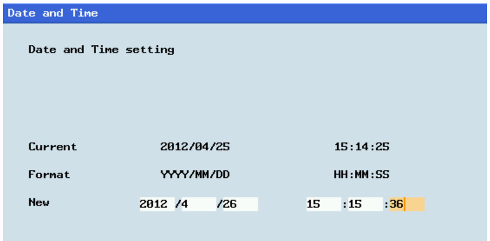
Press “Shift” and “Alarm” on the PPU simultaneously.

Make sure the password is set to the “CUSTOMER” access level.

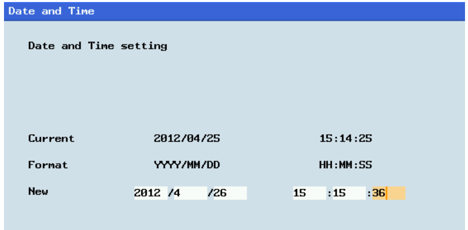
Press the “Date time” SK on the PPU.



Enter a new “Date” and “Time”.



Press the “OK” SK on the PPU.



Press the “Cancel” SK on the PPU to abort the operation.

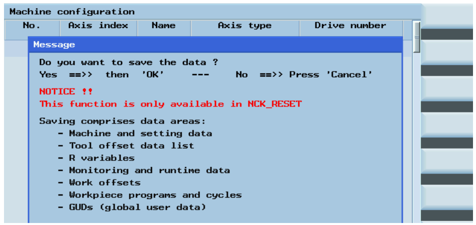
Save data

“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.

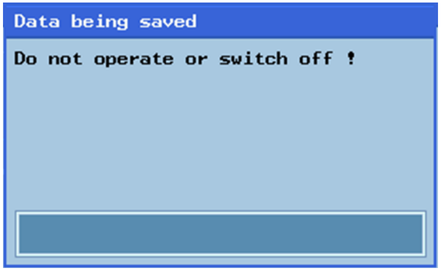
Press the “Save data” SK on the PPU.



SIEMENS

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!

Gear change

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

Additional Information Part 2

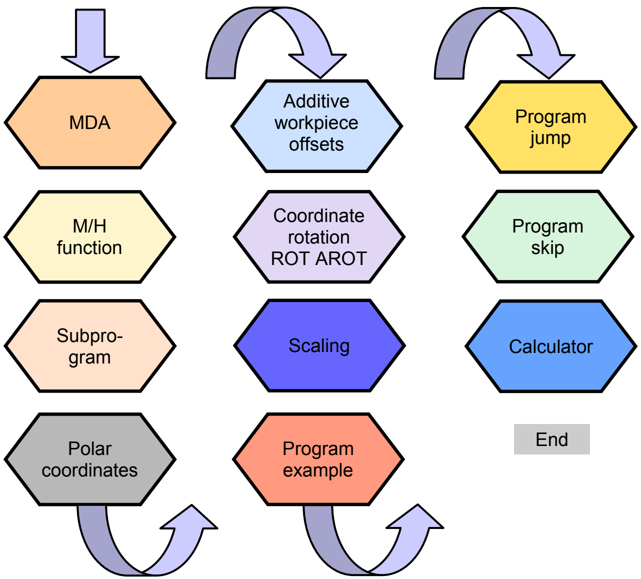
Content

SEQUENCE

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



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

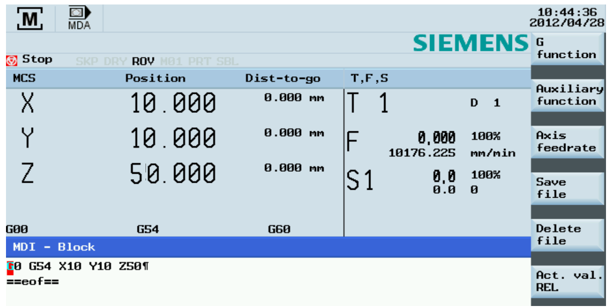
Press the “Delete file” SK on the PPU.

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.



M/H function

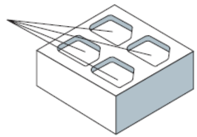
The M function initiates switching operations, such as "Coolant ON/ OFF". Various M functions have already been assigned a fixed functionality by the CNC manufacturer. The M functions not yet assigned are reserved for free use of the machine tool manufacturer.
With H functions, the meaning of the values 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.

Specified M function	Explanation	Specified M function	Explanation
M0	Stop program	M6	Tool change
M1	Stop program with conditions	M7 / M8	Coolant on
M2	End program	M9	Coolant off
M30	End program and back to the beginning	M40	Select gear stage automatically
M17	End subprogram	M41~M45	Change spindle gear
M3 / M4 / M5	Spindle CW/CCW/ Stop		

Subprogram

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.

Subprogram for positions of the four pockets.



Example

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.

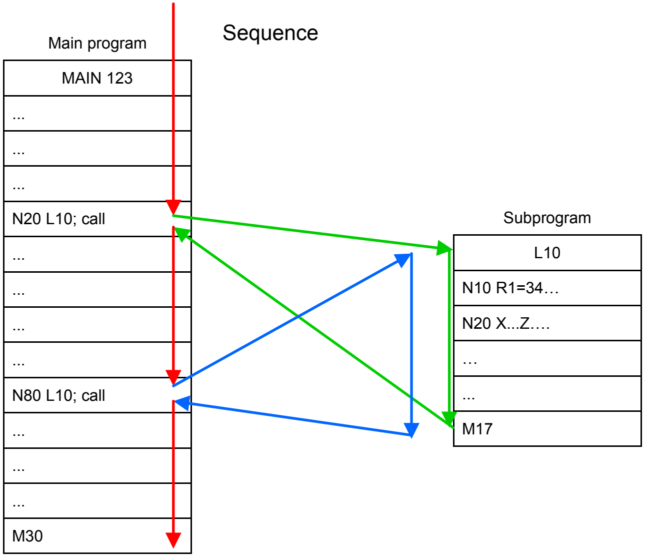
The subprogram should be given a unique name enabling it to be selected from several subprograms. When you create the program, the program name may be freely selected.
However, the following rule should be observed:

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

Example: LRAHMEN7

SIEMENS

SEQUENCE



Subprograms can be called from a main program, and also from another subprogram. In total, up to eight 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, Y, 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 G17: 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.

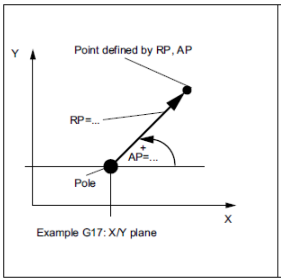
G110 Pole specification relative to the setpoint position last programmed (in the plane, e.g. with G17: X/Y)
(when using G110, please always take the current position of the tool as the reference point to specify the new pole)

G111 Pole specification relative to the origin of the current workpiece coordinate system (in the plane, e.g. with G17: X/Y)

G112 Pole specification, relative to the last valid pole; retain plane

Programming example

```
N10 G17                           ; X/Y plane
N20 G111 X17 Y36                  ; pole coordinates in the current workpiece
AP=45 RP=50                         coordinate system
...
N80 G112 X35.35 Y35.35            ; new pole, relative to the last pole as a
AP=45 RP=27.8                       polar coordinate
N90 ... AP=12.5 RP=47.679         ; polar coordinate
N100 ... AP=26.3 RP=7.344 Z4      ; polar coordinate and Z axis (= cylinder
                                    coordinate)
```



Additive workpiece offsets

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

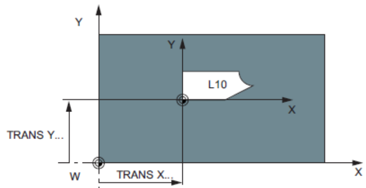
- For recurring shapes/arrangements in various positions on the workpiece
- When selecting a new reference point for dimensioning

This results in the current workpiece coordinate system.

TRANS X...Y... Z... ; programmable offset (absolute)
ATRANS X...Y... Z... ; programmable offset, additive to existing offset (incremental)
TRANS ; without values, clears old commands for offset

Programming example

```
N20 TRANS X20.0 Y15.0        programmable offset
L10                          subprogram call
```



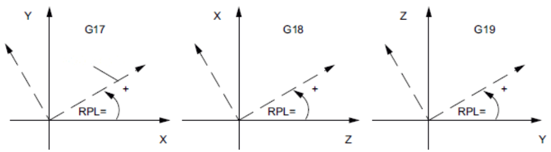
Coordinate rotation ROT AROT

The programmable rotation ROT, AROT can be used:

The rotation is performed in the current plane G17, G18 or G19 using the value of RPL=...specified in degrees.

ROT RPL=... ; programmable rotation offset (absolute).
AROT RPL=… ; programmable offset, additive to existing offset (incremental)
ROT ; without values, clears old commands for offset

N10 G17
N20 AROT RPL=45 additive 45 degree rotation
L10 subprogram call

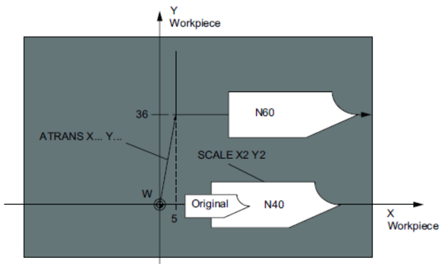


Scaling

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

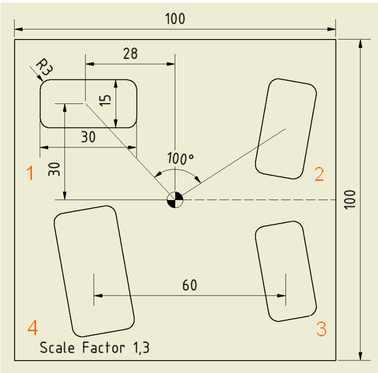
If a program contains SCALE or ASCALE, this must be programmed in a separate block.
Programming example
N10 G17
N20 SCALE X2.0 Y2.0 ; contour is enlarged two times in X and Y
L10 subprogram call



Program example

This describes and analyzes the additive offset, coordinate rotation, scaling functions mentioned above.

Machining target dimension drawing and the final effect are as follows:

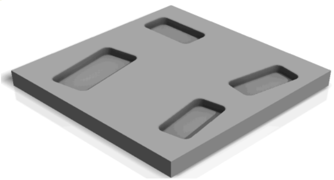


Drawing 1—original workpiece machining

Drawing 2—coordinate rotates 100º

Drawing 3—①Drawing 2 along X axis mirror image
②Coordinate rotates 20º

Drawing 4—①Drawing 3 along Y axis moves 60 in negative direction
②enlarge 1.3 times in X and Y direction



In this example, the positive direction of the XY coordinate axis is different when machining each groove!

```
N10  SUPA G00 Z300 D0
N15  SUPA G00 X0 Y0
N20  G17 T1 D1
N25  MSG ("change to 1 tool")
N30  M5 M9 M00
N35  S5000 M3 G94 F300
N40  G00 X-28 Y 30
N45  G00 Z2
N50  LAB1:
N65  POCKET3( 50, 0, 2, -5, 30, 15, 3, -28, 30, 0, 5, 0, 0, 300, 100, 0, 11, 5, , , 5, 3, )
N70  LAB2:
N75  M01
N80  ROT RPL=-100
N85  REPEAT LAB1 LAB2 P1
N90  M01
N95  AMIRROR X=1
N100 AROT RPL=-20
N105 M01
N110 REPEAT LAB1 LAB2 P1
N115 AROT RPL=10
N120 ATRANS Y-60
N125 AROT RPL=-10
N130 ASCALE X1.3 Y1.3
N135 REPEAT LAB1 LAB2 P1
N140 M30
```

N10 SUPA→cancel all settable offsets
N15
N20 coordinate plane G17，use tool 1
N25
N30
N35
N40
N45
N50 LAB1:milling start sign
N65 milling rectangular groove（depth 5 mm, length 30 mm, width 15 mm, corner radius 3 mm, groove datum coordinate （X-28,Y30), groove longitudinal axis and plane X axis clamping angle 0º）
N70 LAB2:milling groove end sign
N75
N80 coordinate axis rotates 100º in positive direction
N85 machining the same groove at the new position
N90
N95 along the new X axis to change the mirror image
N100 coordinate axis rotates -20º in positive direction
N105
N110 machining the same groove at the new position
N115 coordinate axis rotates －10º in negative direction
N120 Y axis coordinate moves 60 in negative direction
N125
N130 groove enlarged 1.3 times in the X, Y direction.
N135 machining the same groove at the new position
N140 end

Program jump

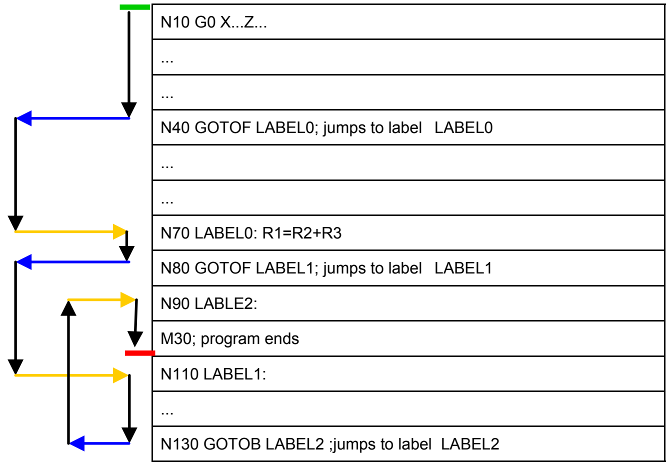
NC programs process their blocks in the sequence in which they were arranged when they were written. The processing sequence can be changed by introducing program jumps. The jump destination can be a block with a label or with a block number. This block must be located within the program. The unconditional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

Program execution



Unconditional jump example

Additional Information Part 2

Basic Theory

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 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-20
N30 Z5
...
N85 T2 D1 M6 ; Tool change
N90 S5000 M3 G94 F300
; N95 G00 X60 Y55 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.

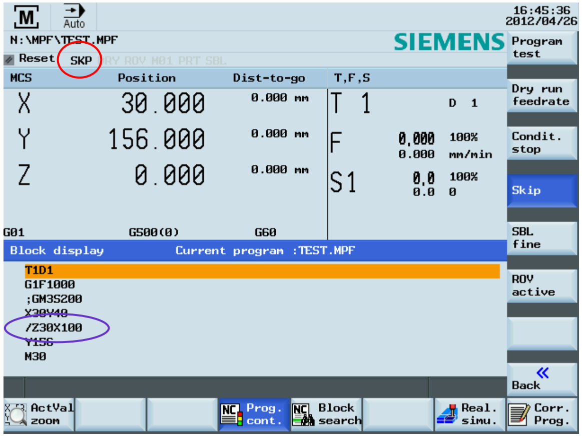
Method 2

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.



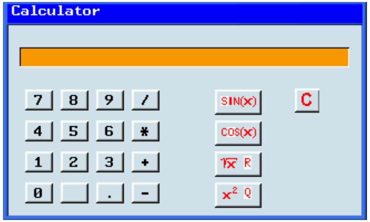
When ”SKP” is displayed (red circle), the skip function has been activated. After activating “SKP”, using “/” at the beginning of the program string (shown in purple circle), the string will be skipped without influencing the execution.

Basic Theory

Calculator

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 “=“ SK on the PPU. =



SEQUENCE

C Delete — Press this SK to delete the contents in the calculator.

Back — Press this SK to exit the calculator screen.

Accept — 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

Sample Program

SIEMENS

Content

Unit Description

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

Unit Content

Milling program 1

Milling program 2

Milling program 3

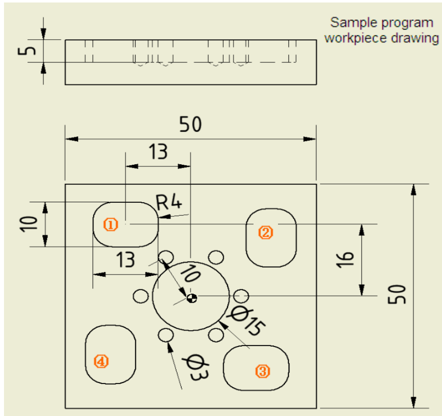
End

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

Milling program 1

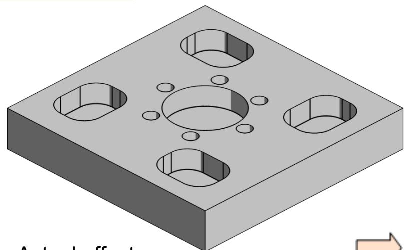
Make sure all the preparations and safety measures have been performed before machining !



The zero point of the workpiece is located at the center point of the workpiece

Tool information：

T1 Milling tool D50
T2 Milling tool D8



Actual effect

Machining Process

```
N10  G17 G90 G54 G60 ROT
N20  T1 D1; FACEMILL
N30  M6
N40  S4000 M3 M8
N50  G0 X-40 Y0
N60  G0 Z2
; ========Start face milling=========
N70  CYCLE71( 50, 1, 2, 0, -25, -25, 50, 50, 0, 1, , , 0, 400, 11, )
N80  S4500
N90  CYCLE71( 50, 1, 2, 0, -25, -25, 50, 50, 0, 1, , , 0, 400, 32, )
; ========End face milling==========
N100 G0 Z100
N110 T2 D1 ; ENDMILL D8
N120 M6
N130 S4000 M3
N140 M8 G0 X-13 Y16
N150 G0 Z2
; ========Start rectangular pocket roughing==========
N160 _ANF:
N170 POCKET3( 50, 0, 2, -5, 13, 10, 4, -13, 16, 0, 5, 0.1, 0.1, 300, 200, 2, 11, 2.5, , , , 2, 2)
; ==Adaptive rotation around Z axis==
N180 AROT Z90
N190 _END:
```

N10

N20 tool 1 is plane milling tool

N30

N40

N50

N60

; =========Start face milling========

N70 start point（X-25，Y-25）, the length and the width are 50 mm, feedrate 400 mm/min, along the direction parallel to the X axis to perform roughing.

N80

N90 repeat the process in N80 , the difference between the two：along the alternate direction parallel to the X axis to perform finishing

; =========End face milling=========

N100

N110 tool 2 is face milling tool, diameter 8 mm

N120

N130

N140

N150

; ===Start ① rectangular pocket roughing===

N160 _ANF: Milling start sign

N170 milling rectangular groove（depth 5 mm, length13 mm, width 10 mm, corner radius 4 mm, groove base point coordinate（X-13,Y16), angle between groove vertical axis and plane X axis is 0°), feedrate 300 mm/min, milling direction G2, rough machining, use G1 vertical groove center to insert.

; ==Adaptive rotation around Z axis==

N180 rotation in positive direction 90º

N190 _END: Milling end sign

```
; =======Repeat rectangular pocket milling 3 times==========
N200 REPEAT _ANF _END P=3
; ======Cancel rotation=======
N210 ROT
N220 S4500 M3
; ========Start rectangular pocket finishing=========
N230 _ANF1:
N240 POCKET3( 50, 0, 2, - 5, 13, 10, 4, -13, 16, 0, 2.5, 0.1, 0.1, 300, 200, 2, 2, 2.5, , , , 2, 2)
; ==Adaptive rotation around Z axis==
N250 AROT Z90
N260 _END1:
; ========Repeat rectangular pocket milling 3 times=========
N270 REPEAT _ANF1 _END1 P=3
N280 ROT
; ========Cancel rotation=========
```

; ====Repeat ② ③ ④ rectangular pocket milling 3 times=====

N200 Repeat N160 ~ N190 operation three times

; =======Cancel rotation=========

N210 cancel all the coordinate rotation commands

N220

; ===Start ① rectangular pocket finishing===

N230 _ANF1: Milling start sign

N240 milling rectangular groove（depth、length、width、corner radius、base point、corner angles are the same as the above parameters), plane feedrate300 mm/min, depth direction feedrate200 mm/min, milling direction G2, finish machining.

; ==Adaptive rotation around Z axis===

N250 rotation in positive direction 90º

N260 _END1: Milling end sign

; ====Finishing ② ③ ④ rectangular pocket milling =====

N270 repeat N230~N260 operation three times

N280 cancel all the coordinate rotation commands

; =====Cancel rotation===========

Sample Program

Machining Process

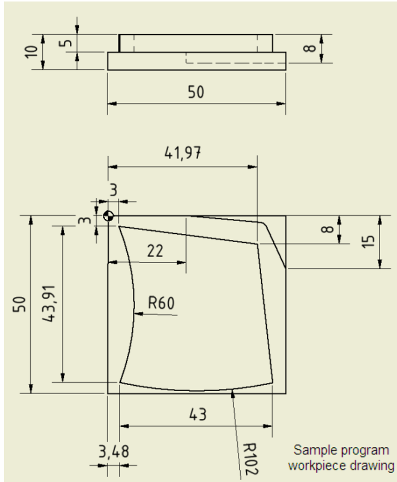
N290 **G0 X0 Y0**
; =============Start circular pocket roughing=============
N300 **POCKET4(50, 0, 2, -5, 7.5, 0, 0, 2.5, 0.1, 0.1, 300, 200, 0, 21, 2, , , 4, 1)**
N310 **S4500 M3**
; =============Start circular pocket finishing=============
N320 **POCKET4(50, 0, 2, -5, 7.5 , 0, 0, 5, 0.1, 0.1, 300, 200, 0, 12, 2, , , 4, 1)**
N330 **G0 Z100**
; =========Start drilling=========
N340 **T3 D1 ;DRILL D3**
N350 **M6**
N360 **S5000 M3**
N370 **G0 X0 Y0**
N380 **MCALL CYCLE81(50, 0, 2, -5, 0)**
N390 **HOLES2(0, 0, 10, 45, 60, 6)**
N400 **MCALL**
N410 **M30**

N290 back to workpiece zero point
; =====Start circular pocket roughing=====
N300 milling circular groove（depth 5 mm, radius 7.5 mm, groove base point coordinate（X0,Y0), angle between groove vertical axis and plane X axis is 0°), milling direction is positive, rough machining.
N310
; =====Start circular pocket finishing=====
N320 milling circular groove（depth 5 mm, radius 7.5 mm, groove basic point coordinate（X0,Y0), the clamping angle between the groove vertical axis and plane X axis is 0), finish machining allowance 0.1 mm, milling direction is positive, finish machining, use G1 vertical groove center to insert.
N330 G0 Z100
; =========Start drilling=========
N340 3 tool is drilling tool diameter 3 mm
N350
N360
N370 back to workpiece zero point
N380 drilling depth 5 mm, use "MCALL" mode to use command, means drilling position decided by the parameters in N490
N390 circular line hole forms cycle command（circular center point coordinate（X0,Y0), radius 10 mm, angle between the line with first hole and circular center point and the X axis in positive direction is 45º, angle between the holes is 60º, circular hole number 6 个）
N400 cancel mode use
N410 M30

Drawing

Milling program 2

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

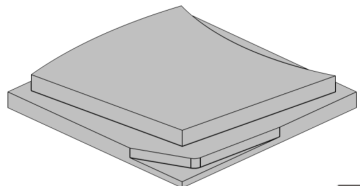


Sample program workpiece drawing

Workpiece zero point is located in the top left corner.

Tool information:

T1 Milling tool D50
T2 Milling tool D12
T4 Milling tool D10


Actual effect

Sample Program

Machining Process

```
N10   G17 G90 G60 G54
N20   T1 D1 ;FACEMILL D50
N30   M6
N40   S3500 M3
N50   G0 X0 Y0
N60   G0 Z2
 ; =======Start face milling========
N70   CYCLE71( 50, 1, 2, 0, 0, 0, 50,  -50 , ,
1, 40, , 0.1, 300, 11, )
N80   S4000 M3
N90   CYCLE71( 50, 0.1, 2, 0, 0, 0, 50, -50 ,
, 1, 40, , 0, 250, 32, )
 ; ======Start contour milling======
N100  T2 D2 ;END MILL
N110  M6
N120  S3500 M6
N130  CYCLE72( "CON1:CON1_E", 50, 0, 2,
-5, 2, 0.1, 0.1, 300, 300, 11, 42, 1, 4, 300, 1, 4)
 ; =======Start path milling with radius
compensation ========
N140  T4 D1 ;ENDMILL D10
N150  M6
N160  S4000 M3
N170  G0 X55 Y-15
N180  G0 Z2
N190  G1 F300 Z-8
N200  G42 G1 Y-15 X50
N210  G1 X44 Y-2 RND=2
N220  G1 Y0 X 22
N230  G40 Y30
N240  M30
```

N10
N20 tool 1 is milling tool, diameter 50 mm
N30
N40
N50 back to workpiece zero point
N60
; =========Start face milling======
N70 start point (X0，Y0)，the length and the width are 50 mm, feedrate 300 mm/min, finishing allowance 0.1 mm, along the direction parallel to the X axis to perform the rough machining
N80
N90 start point（X0，Y0)，the length and the width are 50 mm, feedrate 250 mm/min, finishing allowance 0, along the direction parallel to the X axis to perform the finish machining
; =====Start contour milling=======
N100 tool 2 is milling tool
N110
N120
N130 contour cutting depth 5 mm, all finishing allowances 0.1 mm, the feedrate of surface machining and cutting direction 300 mm/min, use G42 to activate the compensation, use G1 to do rough machining, approaching path is along a straight line, length 4 mm, the parameters of feedrate/path/length in retraction and approach are equal.
; ====Start path milling with radius compensation ===
N140 tool 4 is face milling tool, diameter 10 mm
N150
N160
N170
N180
N190
N200 G42 activate tool radius compensation
N210 starts from（X44,Y-2）insert a reverse circle, radius is 2 mm
N220 （X22,Y0）is the reverse circle point
N230 G40 cancel tool radius compensation
N240

```
;*************CONTOUR************
CON1:

;#7__DlgK contour definition begin - Don't
change!;*GP*;*RO*;*HD*
G17 G90 DIAMOF;*GP*
G0 X3 Y3 ;*GP*
G2 X3.27 Y-40.91 I=AC(-52.703) J=AC(-
19.298) ;*GP*
G3 X46.27 Y-47 I=AC(38.745) J=AC
(54.722) ;*GP*
G1 X42 Y-8 ;*GP*
X3 Y3 ;*GP*
;CON,0,0.0000,4,4,MST:0,0,AX:X,Y,I,J
;*GP*;*RO*;*HD*
;S,EX:3,EY:3;*GP*;*RO*;*HD*
;ACW,DIA:0/35,EX:3.27,DEY:-
43.91,RAD:60;*GP*;*RO*;*HD*
;ACCW,DIA:0/35,DEX:43,EY:-
47,RAD:102;*GP*;*RO*;*HD*
;LA,EX:42,EY:-8;*GP*;*RO*;*HD*
;LA,EX:3,EY:3;*GP*;*RO*;*HD*
;#End contour definition end - Don't
change!;*GP*;*RO*;*HD*

CON1_E:;****** CONTOUR ENDS*******
```

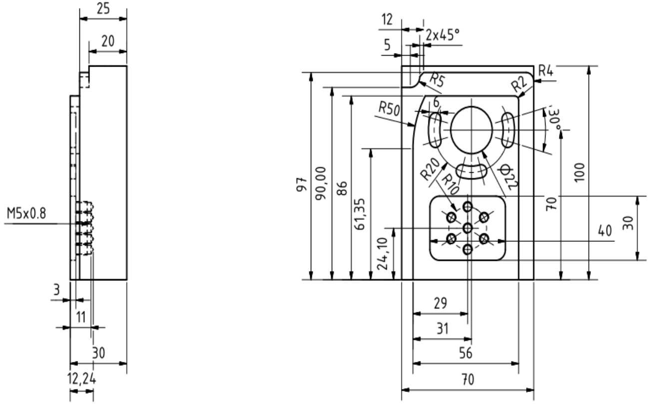
This program is additional description information created by the system automatically after finishing the programming of the rough cutting CYCLE72 and does not affect the system execution.

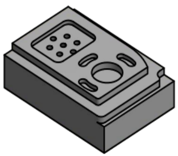
Sample Program

Drawing

Milling program 3

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





Tool information

T1 Milling tool D50
T2 Milling tool D12
T3 Milling tool D10
T4 Milling tool D16
T5 Milling tool D5
T6 Drilling tool D10
T7 Drilling tool D5
T8 Tap D6

Machining Process

```
N10  G17 G90 G54 G71
N20  SUPA G00 Z300 D0
N30  SUPA G00 X300 Y300
N40  T1 D1
N50  MSG ("Please change to Tool No 1")
N60  M05 M09 M00
N70  S4000 M3
; =======Face milling start========
N80  CYCLE71( 50, 2, 2, 0, 0, 0, 70, 100,
0, 2, 40, 2, 0.2, 500, 41, 5)
N90  S4500 M3
N100 CYCLE71( 50, 2, 2, 0, 0, 0, 70, 100,
0, 2, 40, 2, 0.2, 300, 22, 5)
; ========Face milling end========
N110 SUPA G00 Z300 D0
N120 SUPA G00 X300 Y300
; =======Path milling start========
N130 T3 D1
N140 MSG( "Please change to Tool No 3")
N150 M05 M09 M00
N160 S5000 M3 G94 F300
N170 G00 X-6 Y92
N180 G00 Z2
N190 G01 F300 Z-10
N200 G41 Y 90
N210 G01 X12 RND=5
N220 G01 Y97 CHR=2
N230 G01 X70 RND=4
N240 G01 Y90
N250 G01 G40 X80
N260 G00 Z50
; ========Path milling end========
```

N10
N20
N30
N40
N50 hint: change to tool 1
N60
N70
; =======Face milling start=======
N80 start point (X0,Y0), machining length: X→70 mm, Y→100 mm, angle between vertical axis and X axis is 0°, finishing allowance 0.2 mm, feedrate 500 mm/min, along the alternate direction parallel to the Y axis to perform the finishing

N90
N100 repeat N80 contour process, the difference in the feedrate is 300 mm/min along the single direction parallel to the Y axis to perform the finishing

; =========Face milling end=======
N110
N120
; ======Path milling start=========
N130
N140 hint: change to tool 3
N150
N160 feedrate 300 mm/min
N170
N180
N190
N200 left side radius compensation
N210 circle, milling radius is 5 mm
N220 incline, milling side length is 2 mm
N230
N240
N250 cancel tool radius compensation
N260
; =========Path milling end======

SIEMENS

Machining Process

N270 SUPA G00 Z300 D0
N280 SUPA G00 X300 Y300
N290 T4 D1
N300 MSG ("Please change to Tool No 4")
N310 M05 M09 M00
; ===Circular pocket milling start====
N320 S5000 M3
N330 POCKET4(50, 0, 2, -5, 22, 38, 70, 2.5, 0.2, 0.2, 300, 250, 0, 21, 10, 0, 5, 2, 0.5)
N340 S5500 M3
N350 POCKET4(50, 0, 2, -5, 22, 38, 70, 2.5, 0.2, 0.2, 250, 250, 0, 22, 10, 0, 5, 2, 0.5)
; ===Circular pocket milling end====
N360 SUPA G00 Z300 D0
N370 SUPA G00 X300 Y300
N380 T5 D1
N390 MSG ("Please change to Tool No 5")
N400 M05 M09 M00
; ========Slot milling start========
N410 M3 S7000
N420 SLOT2(50, 0, 2, , 3, 3, 30, 6, 38, 70, 20, 165, 90, 300, 300, 3, 3, 0.2, 0, 5, 250, 3000,)
; ========Slot milling end========

N270
N280
N290
N300 hint: change to tool 4
N310
; ====Circular pocket milling start===
N320
N330 milling circular groove（depth 5 mm, radius 22 mm, groove center coordinate （X38,Y70), finishing allowance 0.2 mm, plane machining feedrate 300 mm/min, depth machining feedrate 250 mm/min, milling in positive direction, along helical path insert to do rough machining, helical path radius 2 mm, insert depth 0.5 mm）
N340
N350 repeat N370 milling process, the difference is the machining allowance.
; ====Circular pocket milling end===
N360
N370
N380
N390 hint: change to tool 5
N400
; ========Slot milling start=======
N410
N420 milling slot（depth 3 mm, machining 3 slots, slot angle 30°, slot width 6 mm, basic circle center point coordinate（X38,Y70）, basic circle radius 20 mm, start angle 165°, slot incremental angle 90°, depth machining feedrate 300 mm/min, plane machining feedrate 300 mm/min, milling direction G3, slot edge finishing allowance 0.2 mm, complete machining ways, finishing machining feedrate 250 mm/min, spindle speed rate 3000 r/min
; ========Slot milling end=======

N430 SUPA G00 Z300 D0
N440 SUPA G00 X300 Y300
; ======Contour milling start======
N450 T2 D1
N460 MSG ("Please change to Tool No 2")
N470 M05 M09 M00
N480 S5000 M3
N490 CYCLE72("CONT1:CONT1_E", 50, 0, 2, -5, 5, 0, 0, 300, 100, 111, 41, 12, 3, 300, 12, 3)
; ======Contour milling end=======
N500 SUPA G00 Z300 D0
N510 SUPA G00 X300 Y300
; =Rectangular pocket milling start==
N520 T2 D1
N530 MSG ("Please change to Tool No 2")
N540 M05 M09 M00
N550 S6500 M3
N560 POCKET3(50, 0, 1, -3, 40, 30, 6, 36, 24.1, 15, 3, 0.1, 0.1, 300, 300, 0, 11, 12, 8, 3, 15, 0, 2)
N570 POCKET3(50, 0, 1, -3, 40, 30, 6, 36, 24.1, 15, 3, 0.1, 0.1, 300, 300, 0, 12, 12, 8, 3, 15, 0, 2)
; ==Rectangular pocket milling end==

N430
N440
; ======Contour milling start======
N450
N460 hint: change to tool 2
N470
N480
N490 contour cutting depth 5 mm, surface machining feedrate 300 mm/min, cutting direction feedrate 100 mm/min, use G41 to activate compensation, use G1 to do rough machining, back to the machining plane at the end of the contour, approach path is along 1/4 circle in space, length 3 mm, the parameters of feedrate//path/length for retraction and approach are equal.
; ======Contour milling end=======
N500
N510
; =Rectangular pocket milling start==
N520
N530 hint: change to tool 2
N540
N550
N560 milling rectangle groove（depth 3 mm, length 40 mm, width 30 mm, corner radius 6 mm, groove base point coordinate（X36,Y24.1), angle between groove vertical axis and plane X axis is 15°), finishing allowance 0.1 mm, feedrate surface machining and cutting direction machining is 300 mm/min, milling in positive direction, rough machining, use G1 vertical groove center to insert.
N570 repeat N600 milling process, the difference is the machining allowance.
; ==Rectangular pocket milling end==

Sample Program

SIEMENS

Machining Process

```
N580  SUPA G00 Z300 D0
N590  SUPA G00 X300 Y300
; ========Centering start=========
N600  T6 D1
N610  MSG ("Please change to Tool No 6")
N620  M05 M09 M00
N630  S6000 M3
N640  G00 Z50 X36 Y24.1
N650  MCALL CYCLE82( 50, -3,  2,  -5, 0, 0.2)
N660  HOLES2( 36, 24.1, 10,  90, 60, 6)
N670  X36 Y24.1
N680  MCALL ; Modal Call OFF
; =========Centering end=========
N690  SUPA G00 Z300 D0
N700  SUPA G00 X300 Y300
; =========Drilling start==========
N710  T7 D1
N720  MSG ("Please change to Tool No 7")
N730  M05 M09 M00
N740  S6000 M3
N750  MCALL CYCLE83( 50, -3, 1,  , 9.24,  ,5, 90,  0.7,  0.5,  1,  0,  3,  5,  1.4, 0.6,  1.6)
N760  HOLES2( 36, 24.1, 10,  90, 60, 6)
N770  X36 Y24.1
N780  MCALL ; Modal call Off
; =========Drilling end==========
```

N580
N590
; =========Centering start=======
N600
N610 hint:change to tool 6
N620
N630
N640
N650 CYCLE82 mode recall command active →drilling depth 5 mm, last drilling depth (delayed milling) stops for 0.2 s
N660 hole arrangement circular center coordinate (X36,Y24.1), circular radius 10 mm, start angle 90°, angle between the holes is 60°, circular hole number 6
N670 continue drilling with (X36,Y24.1) as for the center point
N680 cancel mode recall command
; =========Centering end========
N690
N700
; ===========Drilling start========
N710
N720 hint:change to tool 7
N730
N740
N750 CYCLE83 mode recall command active →drilling depth 9.24 mm, first drilling depth 5 mm, degression 90, last drilling depth (delayed milling) stops for 0.7 s, stops at the start point for 0.5 s, first drilling feed modules is 1, select Z axis as the tool axis, machining type is delayed milling, tool axis is Z axis, minimal depth 5 mm, every retraction is 1.4 mm, drilling depth stops for 0.6 s , reinsert lead distance 1.6 mm
N760 hole arrangement circular center coordinate (X36,Y24.1), circular radius 10 mm, start angle 90°, angle between the holes is 60°, circular hole number 6
N770 continue drilling with (X36,Y24.1) as the center point
N780 cancel mode recall instruction
; =========Drilling end===========

```
N790  SUPA G00 Z300 D0
N800  SUPA G00 X300 Y300
; =========Tapping start=========
N810  T8 D1
N820  MSG ("Please change to Tool No 8")
N830  M05 M09 M00
N840  S500 M3
N850  MCALL CYCLE84( 50, -3,  2,   ,  6,  0.7, 5,  ,  2,  5,  5,  5,  3,  0,  0,  0,  5, 1.4 )
N860  HOLES2( 36, 24.1, 10,  90, 60, 6)
N870  X36 Y24.1
N880  MCALL ; Modal call Off
; =========Tapping end==========
N890  SUPA G00 Z500 D0
N900  SUPA G00 X500 Y500;
; =========Move to the change position Ready to start next program or repeat =========
N910   M30
```

N790
N800
; ==========Tapping start========
N810
N820 hint:change to tool 8
N830
N840
N850 CYCLE84 mode recall active→drilling depth 6 mm, last tapping depth (delayed milling) stops for 0.7 s, after the cycle, the spindle M5 stops, machining dextrorotation thread, size 2 mm , spindle stop position is 5° , the tapping speed and the retraction speed of the spindle are 5 r/min, select Z axis as the tool axis, incremental drilling depth 5 mm, retraction value is 1.4 mm
N860 hole arrangement circular center coordinate (X36,Y24.1), circular radius 10 mm, start angle 90°, angle between the holes is 60°, circular hole number 6
N870 continue drilling with X36,Y24.1) as the center tapping
N880 cancel mode recall instruction
; ==========Tapping end=========
N890
N900
; ======Move to the change position Ready to start next program or repeat =======
N910

SIEMENS

Machining Process

```
N1100 ;*************CONTOUR************
N1110 CONT1:
N1120 ;#7__DlgK contour definition begin -
Don't change!;*GP*;*RO*;*HD*
N1130 G17 G90 DIAMOF;*GP*
N1140 G0 X7 Y0 ;*GP*
N1150 G1 Y61.35 ;*GP*
N1160 G2 X13.499 Y86 I=AC(57) J=AC
(61.35) ;*GP*
N1170 G1 X63 RND=2 ;*GP*
N1180 Y0 ;*GP*
N1190 ;CON,0,0.0000,4,4,MST:0,0,AX:X,Y,I,J,
TRANS:1;*GP*;*RO*;*HD*
N1200 ;S,EX:7,EY:0;*GP*;*RO*;*HD*
N1210 ;F,LFASE:0;*GP*;*RO*;*HD*
N1220 ;LU,EY:61.35;*GP*;*RO*;*HD*
N1230 ;ACW,DIA:210/0,EY:86,AT:0,RAD:50;*
GP*;*RO*;*HD*
N1240 ;LR,EX:63;*GP*;*RO*;*HD*
N1250 ;R,RROUND:2;*GP*;*RO*;*HD*
N1260 ;LD,EY:0;*GP*;*RO*;*HD*
N1270 ;#End contour definition end - Don't
change!;*GP*;*RO*;*HD*
N1280 CONT1_E:;************* CONTOUR
ENDS ************
```

This section is additional description information created by system automatically after finishing the programming of the rough cutting CYCLE95 and does not affect the system execution.

Notes

Notes

ISO
Mode

Content

Unit Description

This unit describes the ISO operating functions in 808D, 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.

Unit Content

ISO function switch

ISO code explanation

ISO program transfer and operation

End

Basic Theory

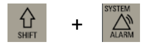
ISO function switch

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

ISO function switch

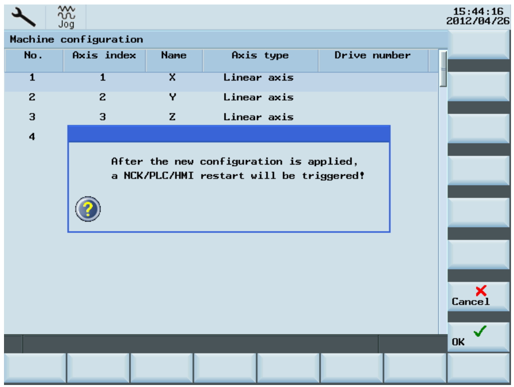
Method 1

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



Press the “ISO mode” SK on the right.

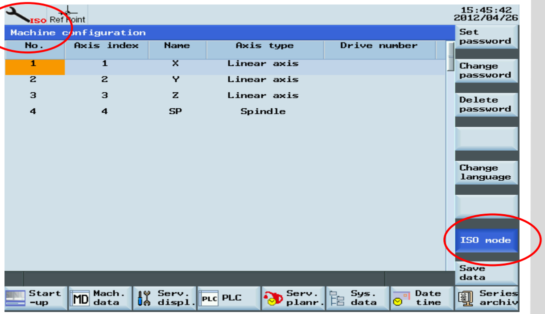
A dialog box appears prompting whether to activate the new setting. Select the “OK” SK to activate it.



ISO
Mode

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.

Method 2

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
N5 G17 G90 G54 G71 ¶
N20 T1 H1¶
N25 MSG("Tool No 1 in use")¶
N35□S4000 M3¶
N40 CYCLE71( 50.00000, 2.00000, 2.00000, 0.00000, 0.00000, 0.0000
N45 S4500 M3¶
```

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

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

ISO
code
explanation

All the ISO codes described in this unit can be implemented in the ISO mode of the 808D 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
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	
G90/G91	Absolute/incremental programming	
G94/G95	Feedrate F in mm/min / mm/r	As DIN
S	Spindle speed	As DIN
，R	Reverse circle（note the form there must be "，" before R parameter）	RND
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	End of Subroutine	M17

Basic Theory

In DIN mode, the tool length is activated automatically, but in ISO mode, you must activate the tool length via G code.

G43/G44 and G49

Use G43/G44, the tool length compensation value will be activated.

G43: Tool length compensation in positive direction

G44: Tool length compensation in negative direction

G49: Cancel tool length compensation

H01→Offset value 20.0
H02→Offset value -30.0
H03→Offset value 30.0
H04→Offset value -20.0

G90 G43 Z100.0 H01; Z will reach 120.0

G90 G43 Z100.0 H02; Z will reach 70.0

G90 G44 Z100.0 H03; Z will reach 70.0

G90 G44 Z100.0 H04; Z will reach 120.0

Note: In DIN mode, you must open the H code list in the tool list. For information on the opening method, please refer to the instructions for H code on **page 104**

G98 : Fixed cycle back to the original point

G99 : Fixed cycle back to R point

G80 : Cancel the fixed cycle

Pausing function G04

G04 X5.0→delay 5 s

G04 P5→delay 5 ms

N5 G90 T1 M06

N10 M3 S2000; spindle rotation

N20 G99 G81 X300 Y-250 Z-150 R-10 F120; after orientation drilling, back to R point

N30 X1000. ; after orientation drilling, back to R point

N40 G04 X2.0 ; delay 2 s

N50 G98 Y-550 ; after orientation drilling, back to start point

N60 G80 ; cancel the fixed cycle

N70 M5 ; spindle rotation stop

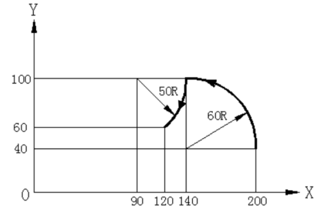
N80 M30

Code G02 and G03

G02 circular interpolation in positive direction

G03 circular interpolation in negative direction

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



Method 1 (use incremental to describe circular radius)

G92 X200.0 Y40.0 Z0

G90 G03 X140.0 Y100.0 I-60.0 F300.0

G02 X120.0 Y60.0 I-50.0

Method 2 (use parameter R to describe circular radius)

G92 X200.0 Y40.0 Z0

G90 G03 X140.0 Y100.0 R60.0 F300

G02 X120.0 Y60.0 R50.0

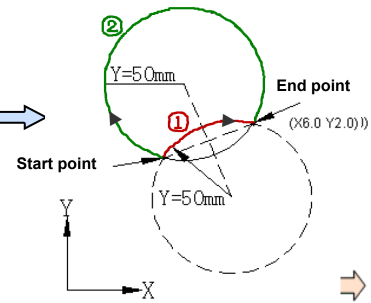
When specifying circle radii with parameter R

Circles less than 180° is 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



ISO Mode

SIEMENS

Basic Theory

Frequently used letter meanings of typical fixed cycle codes in ISO

P.	Descriptions	Unit	Applied range and note
X/Y	Cutting end point X/Z absolute coordinate values		G73 / G74 / G76 G81 ~ G87 / G89
Z	The distance incremental value between R point and the bottom of the hole, or the absolute coordinate value of the bottom of the hole		G73 / G74 / G76 G81 ~ G87 / G89
R	The distance incremental value between the start point plane and R point or the absolute coordinate value of R point		G73 / G74 / G76 G81 ~ G87 / G89
Q	The depth of every cut （incremental value）		G73 / G83
	Offset value （incremental value）		G76 / G87
P	The delay time at the bottom of the hole	ms	G74 / G76 / G89 G81 ~ G87
F	The feedrate of the cutting	mm/min	G73 / G74 / G76 G81 ~ G87 / G89
K	The repeat times of the fixed cycle		G73 / G74 / G76 G81 ~ G87 / G89

In 808D, the default ISO program feed distance unit is mm！（X100→100mm）

Note：change the parameter 10884=0，to make X100 → 100 um / X100. → 100 mm

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！

G73 fast-speed deep hole drilling

Common programming structures：

G73 X—Y—Z—R—Q—F—K

Motion process：

① Drilling motion (-Z) → intermediate feed

② Motion at the bottom of the hole → none

③ Retraction motion（+Z）→ fast feed

G73 application example program：

```
M3 S1500 ;spindle rotation
G90 G99 G73 X0 Y0 Z-15 R-10 Q5 F120
          ;after orientation drill 1st hole, back to R point
Y-50      ;after orientation drill 2nd hole, back to R point
Y-80      ;after orientation drill 3rd hole, back to R point
X10       ;after orientation drill 4th hole, back to R point
Y10       ;after orientation drill 5th hole, back to R point
G98 Y75   ;after orientation drill 6th hole, back to R point
G80       ;cancel fixed cycle
G28 G91 X0 Y0 Z0     ;back to reference point
M5        ;spindle rotation stop
M30
```

G74 reverse tapping cycle

Common programming structures：

G74 X—Y—Z—R—P—F—K

Motion process：

① Drilling motion（-Z）→ cutting feed

② Motion at the bottom of the hole → spindle rotation in positive direction

③ Retraction motion（+Z）→ cutting feed

G74 application example program：

```
M4 S100 ;spindle rotation
G90 G99
G74 X300 Y-250 Z-150 R-120 P300 F120
;after orientation drill 1st hole, back to R point
Y-550     ;after orientation drill 2nd hole, back to R point
Y-750     ;after orientation drill 3rd hole, back to R
pointX1000     ;after orientation drill 4th hole, back to R pointY-
550     ;after orientation drill 5th hole, back to R pointG98
Y750 ;after orientation drill 6th hole, back to R
pointG80        ;cancel fixed cycle
G28 G91 X0 Y0 Z0     ;back to reference point
M5          ;spindle rotation stop
M30
```

G76 Boring cycle
Common programming structures:
G76 X—Y—Z—R—Q—P—F—K
Motion process：
① Drilling motion (-Z) → cutting feed
② Motion at the bottom of the hole → spindle stop directional
③ Retraction motion (+Z) → fast feed

G76 application example program：

M3 S500 ;spindle rotation
G90 G99
G76 X300 Y-250 Z-150 R-100 Q5 P1000 F120
;after orientation bore 1st hole, then move 5 mm, stop for 1 s at the bottom of the hole, back to the R point.
Y-50 ;bore 2nd hole (the same as 1st hole)
Y-80 ;bore 3rd hole (the same as 1st hole)
X10 ;bore 4th hole (the same as 1st hole)
Y10 ;bore 5th hole (the same as1st hole)
G98 Y-750 ;bore 6th hole, then move 5 mm, stop for 1s at the bottom of the hole, back to the start point position plane
G80 ;cancel fixed cycle
G28 G91 X0 Y0 Z0 ;back to reference point
M5 ;spindle rotation stop
M30

G81 Drilling cycle (fixed point drilling)
Common programming structures：
G81 X—Y—Z—R—F—K
Motion process：
① Drilling motion (-Z) → cutting feed
② Motion at the bottom of the hole → none
③ Retraction motion (+Z) → fast feed

G81 application example program：

M3 S2000 ;spindle rotation
G90 G99 G81 X300 Y-250 Z-150 R-10 F120
;after orientation drill 1st hole, back to R point
Y-550 ;after orientation drill 2nd hole, back to R point
Y-750 ;after orientation drill 3rd hole, back to R point
X1000 ;after orientation drill 4th hole, back to R point
Y-550 ;after orientation drill 5th hole, back to R point
G98 Y-750;after orientation drill 6th hole, back to start plane
G80 ;cancel fixed cycle

G82 Drilling cycle (countersink drilling)
Common programming structures:
G82 X—Y—Z—R—P—F—K
Motion process：
① Drilling motion(-Z) → cutting feed
② Motion at the bottom of the hole → pause
③ Retraction motion (+Z) → fast feed

G82 application example program：

M3 S2000 ;spindle rotation
G90 G99 G82 X300 Y-250 Z-150 R-100 P1000 F120
;after orientation drill 1st hole, stop for 1 s at the bottom of the hole, back to the R point.
Y-550 ;drill 2nd hole (the same as 1st hole)
Y-750 ;drill 3rd hole (the same as 1st hole)
X1000 ;drill 4th hole (the same as 1st hole)
Y-550 ;drill 5th hole (the same as 1st hole)
G98 Y-750 ;drill 6th hole, stop for 1 s at the bottom of the hole, back to the start point position plane
G80 ;cancel fixed cycle
G28 G91 X0 Y0 Z0 ;back to reference point
M5 ;spindle rotation stop
M30

G83 Drilling cycle (deep hole drilling)
Common programming structures
G83 X—Y—Z—R—Q—F—K
Motion process：
① Drilling motion (-Z) → intermission feed
② Motion at the bottom of the hole → None
③ Retraction motion (+Z) → fast feed

G83 application example program：

M3 S2000 ;spindle rotation
G90 G99 G83 X300 Y-250 Z-150 R-100 Q15 F120
;after orientation drill 1st hole, back to R point
Y-550. ;after orientation drill 2nd hole, back to R point
Y-750. ;after orientation drill 3rd hole, back to R point
X1000. ;after orientation drill 4th hole, back to R point
Y-550. ;after orientation drill 5th hole, back to R point
G98 Y-750. ;after orientation drill 6th hole, back to start plane
G80 ;cancel fixed cycle
G28 G91 X0 Y0 Z0 ;back to reference point
M5 ;spindle rotation stop
M30

G84 Tapping cycle

Common programming structures:

G84 X—Y—Z—R—P—F—K

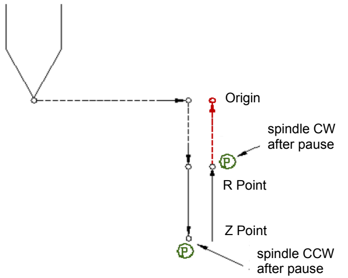
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → spindle rotation in negative direction

③ Retraction motion (+Z) → cutting feed

G84 execution operation graphic:
With command G99 without operation in red line
With command G98 with operation in red line



G85 boring cycle

Common programming structures:

G85 X—Y—Z—R—F—K

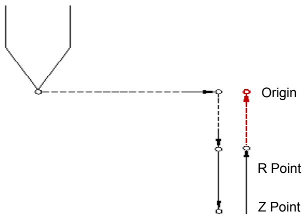
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → none

③ Retraction motion (+Z) → cutting feed

G85 execution operation graphic:
With command G99 without operation in red line
With command G98 with operation in red line
Except that the spindle is not rotating at the bottom of the hole, G85 is same as G84



G86 boring cycle

Common programming structures:

G86 X—Y—Z—R—F—K

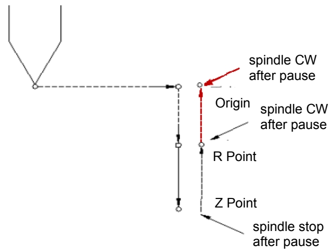
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → spindle stop

③ Retraction motion (+Z) → fast feed

G86 execution operation graphic:
With command G99 without operation in red line
With command G98 with operation in red line
Except for the stop at the bottom of the hole, G86 is same as G81



G89 boring cycle

Common programming structures:

G89 X—Y—Z—R—P—F—L

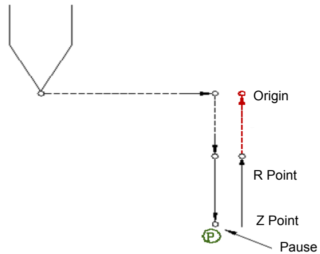
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → pause

③ Retraction motion (+Z) → cutting feed

G89 execution operation graphic:
With command G99 without operation in red line
With command G98 with operation in red line
Except that the spindle stops at the bottom of the hole, G89 is same as G85



SIEMENS

Basic Theory

G87 Boring cycle Ⅰ / reverse boring cycle Ⅱ

Common programming structures：

G87 X—Y—Z—R—Q—P—F—L

Motion process：

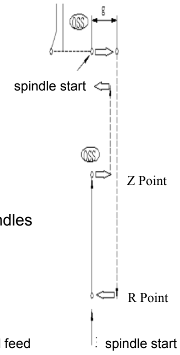
① Drilling motion（-Z） →cutting feed

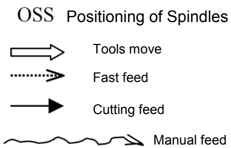
② Motion at the bottom of the hole → spindle stops

③ Retraction motion（+Z） → manual operation or fast feed

G87 execution operation graphic：

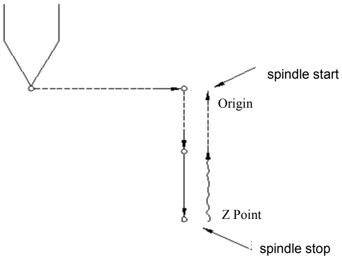
Fixed cycle Ⅱ→

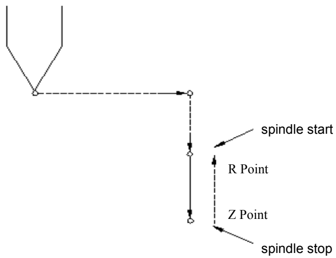




G87 execution operation graphic：

Fixed cycle Ⅰ





ISO program transfer and operation

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

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

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

Press the “USB” SK on the PPU. → USB

Use the “Cursor + Select” keys to select the required program which is then highlighted. ▼ + SELECT

Press the “Copy” SK on the PPU. → Copy

Press the “NC” SK on the PPU. → NC

Press the “Paste” SK on the PPU. → Paste

A specified ISO program is then stored in the 808D 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 have their own rules. Suitable changes must be made at the appropriate positions so that you can run the ISO programs!

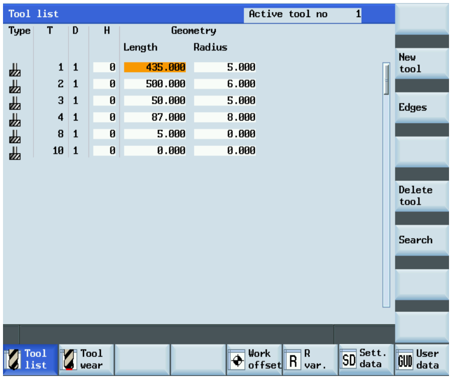
SIEMENS

Basic Theory

Beginning of the program

Common ISO program:
Beginning is "O"
ISO mode of 808D:
Not compatible with the programs beginning with "O"

Common ISO program	808D ISO program
O0001; G0 X50 Y50 Z50 M5 G04 X5 M3 S1000 . . .	O0001; Delete this line G0 X50 Y50 Z50 M5 G04 X5 M3 S1000 . . .



H code

In 808D standard DIN mode, you must open the H list in the tool list first and fill in the data accordingly

2 common methods

① Direct use of the ISO switch button on the PPU to enter ISO mode.
(We recommend the 1st method!)

② Enter code G291 in MDA mode and execute. When the "Reset" is not used, the H list in the tool list is open.

Note: Every tool only can use the H value corresponding to the edge.
In the graphic above, T2 H1 cannot be executed.

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.

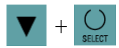
Tool and workpiece setup →simulation→test→machining.

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 PPU.

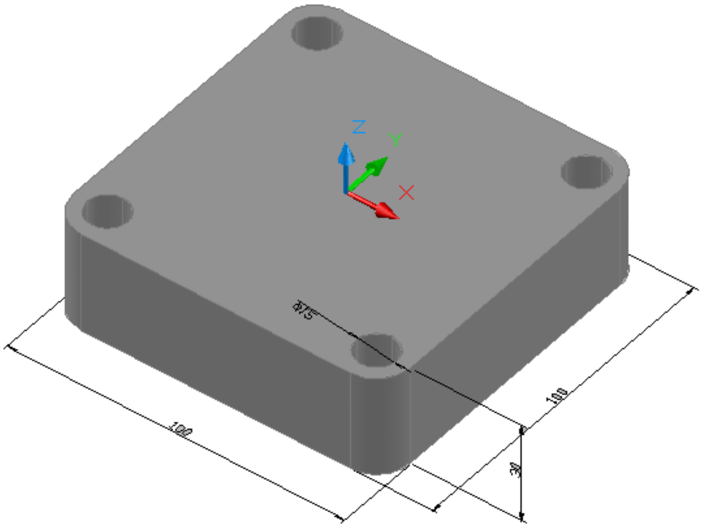
A specified ISO program is then stored in the USB and can be executed as required.

ISO
Mode

Basic Theory

Step 5 Sample program

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 as follows:

```
N10  G291
N20  T1M6
N30  G0G54G90G40
N40  M3S1200F200
N50  G43H1Z50
N60  G0X0Y-70
N70  Z5M8
N80  G1Z-5
N90  G01G41X20D1
N100 G03X0Y-50R20
N120 G1X-50,R10
N130 Y50,R10
N140 X50,R10
N150 Y-50,R10
N160 X40
N170 X0
N180 G03X-20Y-70R20
N190 G1G40X0
N200 G0Z50
N210 T2M6
N220 M3S3000F100
N230 G43H2Z50
N240 G0X40Y-40
N250 Z20
N260 G81Z-2R10
N270 Y40
N290 X-40
N300 Y-40
N310 G80
N320 G0Z50

N330 T3M6
N340 M3S3000F100
N350 G43H3Z50
N360 G73Z-20R10Q5
N370 Y40
N380 Y-40
N390 X40
N400 Y40
N410 G80
N420 G0G40G90G49Z100
N430 M09
N440 G290
N450 M30
```

Note：This program opens/exits ISO mode with theG291/G290 command. It is recommended to use the first method to open ISO mode — using the ISO mode active button on the PPU (described above)

Standard Siemens programming. Machining the same workpiece as described above (can be compared with the ISO code).

```
N10   T1D1M6 ; contour milling tool
N20   G54G90G40G17
N30   M3S2000M8
N40   G0Z25
N50   X0Y-70
N55   CYCLE72( "CONT1:CONT1_E", 50,
0, 2, -5, 2.5, 0.1, 0.1, 200, 200, 111, 41, 2,
20, 200, 2, 20)
N60   T2D1M6 ; quill,  drill center hole
N70   M3S2500M8
N80   MCALL CYCLE82( 50, 0, 2, 0, 2, 0)
N90   CYCLE802( 111111111, 111111111,
40, -40, 40, 40, -40, 40,
-40, -40,  ,)
N100   MCALL
N110   T3D1M6 ; quill; deep hole drilling
N120   M3S2500M8
N130   MCALL CYCLE83( 50, 0, 2,
-20,  ,-5,  ,3, 0.5, 1, 1, 1, 3, 3, 0,  ,0)
N140   CYCLE802( 111111111, 111111111,
40, -40, 40, 40, -40, 40,
-40, -40,  ,)
N150   MCALL
N160   G0G40G90Z60
N170   M09M05
N180   M30
```

The next paragraph describes the codes of the contour. The system will generate them automatically and does not affect the program execution.

```
*************CONTOUR************
CONT1:
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
N190 G17 G90 DIAMOF;*GP*
N200 G0 X0 Y-50 ;*GP*
N210 G1 X-50 RND=10 ;*GP*
N220 Y50 RND=10 ;*GP*
N230 X50 RND=10 ;*GP*
N240 Y-50 RND=10 ;*GP*
N250 X0 ;*GP*
;CON,0,0.0000,5,5,MST:0,0,AX:X,Y,I,J;*GP*;*RO*;*HD*
;S,EX:0,EY:-50;*GP*;*RO*;*HD*
;LL,EX:-50;*GP*;*RO*;*HD*
;R,RROUND:10;*GP*;*RO*;*HD*
;LU,EY:50;*GP*;*RO*;*HD*
;R,RROUND:10;*GP*;*RO*;*HD*
;LR,EX:50;*GP*;*RO*;*HD*
;R,RROUND:10;*GP*;*RO*;*HD*
;LD,EY:-50;*GP*;*RO*;*HD*
;R,RROUND:10;*GP*;*RO*;*HD*
;LL,EX:0;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*

CONT1_E:;****CONTOUR ENDS *****
```

Notes

Notes

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 clockwise
G03	Circular interpolation counter-clockwise
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

SIEMENS

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 work 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 work offset
G153	Non-modal skipping of the settable work offset including base frame

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	Exact stop window, course, with G60, G9

SIEMENS

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 for feedrate F

Group 14: Absolute/incremental dimension modally effective	
Name	**Meaning**
G90 *	Absolute dimensions data input
G91	Incremental dimension data input

Group 15: Feedrate / Spindle modally effective	
Name	**Meaning**
G94	Feedrate mm/min
G95	Feedrate F in mm/spindle revolutions

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 the plane (SAR)

Group 47: External NC languages modally effective	
Name	**Meaning**
G290 *	Siemens mode
G291	External mode

SIEMENS

Technical Support Contact

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

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

Useful Siemens Websites

SINUMERIK Internet address

Further product information can be found at the following web site:

http://www.siemens.com/sinumerik

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