



# OPTIMUM

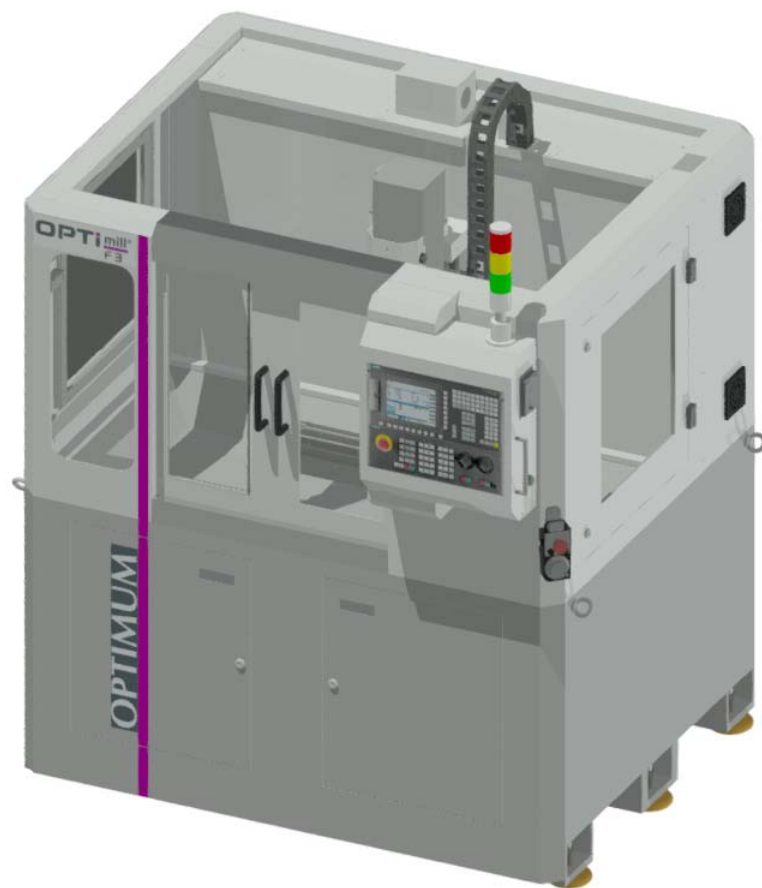
MASCHINEN - GERMANY

## Operating manual

Version 1.0

### CNC milling machine

# OPTi mill<sup>®</sup> F 3





## Table of contents

<b>1</b>	<b>Safety</b>	
1.1	Type plate .....	6
1.2	Glossary of symbols .....	6
1.3	Safety instructions (warning notes) .....	7
1.3.1	Classification of hazards .....	7
1.3.2	Other pictograms .....	7
1.4	Intended use .....	8
1.5	Reasonably foreseeable misuse .....	9
1.5.1	Avoiding misuse .....	9
1.6	Possible dangers caused by the CNC machine .....	9
1.7	Qualification of personnel .....	10
1.7.1	Target group .....	10
1.7.2	Authorized personnel .....	11
1.8	Operator positions .....	12
1.9	Safety devices .....	12
1.9.1	Lockable main switch .....	12
1.9.2	Emergency-stop button .....	13
1.9.3	Control technical protection .....	14
1.9.4	Polycarbonate windows .....	14
1.9.5	Prohibition, warning and mandatory labels .....	14
1.10	Safety check .....	14
1.11	Personal protective equipment .....	15
1.12	Safety during operation .....	15
1.13	Safety during maintenance .....	16
1.14	Disconnecting and securing the CNC machine .....	17
1.14.1	Using lifting equipment .....	17
1.14.2	Mechanical maintenance work .....	17
1.15	Unattended Operation .....	17
1.16	Accident report .....	18
1.17	Electrical system .....	18
1.18	Inspection deadlines .....	18
1.19	Clamping devices for workpieces and tools .....	18
1.20	Environmental protection and water conservation .....	18
<b>2</b>	<b>Technical data</b>	
2.1	Electrical connection .....	20
2.2	Milling spindle .....	20
2.3	Lubricant/coolant system .....	20
2.4	Compressed air .....	20
2.5	Dimensions .....	21
2.6	Working area, traverse path .....	21
2.7	Accuracy .....	21
2.8	Milling table .....	21
2.9	Environmental conditions - operation .....	22
2.10	Environmental conditions - storage .....	22
2.11	Emissions .....	22
2.12	Tools and tool holding fixtures .....	23
<b>3</b>	<b>Assembly and commissioning</b>	
3.1	Scope of delivery .....	25
3.2	Transport .....	25
3.3	Installation and assembly .....	26
3.3.1	Requirements regarding the installation site .....	26



3.3.2	Installation plan F3 .....	27
3.3.3	Machine mounting .....	28
3.3.4	Corrosion protection .....	28
3.3.5	Electrical connection .....	29
3.3.6	Current in the Protective Earth Ground Wire.....	30
3.3.7	Connection compressed air supply .....	31
3.4	First commissioning.....	31
3.4.1	Refilling the cooling lubricant.....	31
3.4.2	Cooling lubricants.....	31
3.5	Refill central lubrication system.....	33
3.6	Functional test and controls .....	35
3.6.1	Warming up the machine .....	35
<b>4</b>	<b>General information about CNC</b>	
4.1	Compensation of geometry .....	36
4.2	Coordinate systems on CNC-machine tools .....	36
4.2.1	Cartesian coordinate system.....	36
4.2.2	Polar coordinate system.....	37
4.2.3	Machine coordinate system.....	37
4.2.4	Workpiece coordinate system .....	37
4.2.5	Rotary axes and secondary axes .....	38
4.3	NC mathematics.....	38
4.3.1	Basics of the coordinate calculation .....	38
4.3.2	Parameters of a triangle .....	38
4.3.3	Angle on a triangle .....	38
4.4	Trigonometric functions .....	39
<b>5</b>	<b>User interface, machine control panel</b>	
5.1	Screen arrangement.....	40
5.2	Elements on the PPU(Panel Processing Unit) front .....	41
5.2.1	Key combination.....	45
5.3	Controls on the machine control panel .....	46
5.4	Protection levels .....	51
5.4.1	Passwords .....	52
5.4.2	Change passwords.....	52
<b>6</b>	<b>Operation</b>	
6.1	Safety .....	53
6.2	Control and indicating elements .....	53
6.3	Operational modes .....	54
6.4	Programming.....	54
6.5	Operation of the machine .....	55
6.5.1	Turning on the CNC- machine.....	55
6.5.2	Reference point approach after turning on.....	56
6.5.3	Using the electronic handwheel .....	58
6.5.4	Insert tool.....	58
6.5.5	Manual tool change .....	59
6.5.6	Clamping the workpiece .....	59
6.5.7	Turning off the CNC- machine.....	60
6.6	Operational modes .....	60
6.7	Programming.....	60
6.8	Starting the program.....	60
6.9	Central lubrication system .....	61
6.10	Data interfaces and current collection .....	61
6.11	Selecting the speed .....	61
6.11.1	Standard values for cutting speeds .....	61
6.11.2	Standard values for speeds with HSS – Eco – twist drilling .....	62



<b>7</b>	<b>M - Code list, M functions</b>	
7.1	M-function for milling machines according to PAL	153
7.2	G functions to PAL	155
<b>8</b>	<b>Notes, messages and error messages</b>	
<b>9</b>	<b>SINUMERIK 808 D</b>	
<b>10</b>	<b>Maintenance</b>	
10.1	Operating material	160
10.1.1	Machine lubricants	160
10.1.2	Cooling lubricants	160
10.2	Safety	161
10.2.1	Preparation	162
10.2.2	Restarting	162
10.3	Inspection and maintenance	163
10.3.1	Cleaning and replacing of the polycarbonate windows	165
10.3.2	Cleaning the electrical cabinet	166
10.4	Interlock switch sliding door	166
10.4.1	Drain the compressed air service unit	166
10.5	Cooling lubricants and tanks	167
10.5.1	Inspection plan for water-mixed cooling lubricants	168
<b>11</b>	<b>Ersatzteile - Spare parts - F3</b>	
11.1	Ersatzteilbestellung - Ordering spare parts	169
11.2	Elektrische Ersatzteile - Electrical spare parts	169
11.3	Schaltplan - Wiring diagram	169
11.4	Gehäuse - Housing	170
11.5	Unterbau - Substructure	171
11.6	Fräskopf - Milling head	172
11.7	Frästisch - Milling table	173
11.8	Frästisch - Milling table	174
11.9	Säule - Column	175
11.10	Zentralschmierung	178
11.11	Schaltplan	179
<b>12</b>	<b>Appendix</b>	
12.1	Copyright	186
12.2	Changes	186
12.3	Storage	186
12.4	Terminology/Glossary	187
12.5	Product follow-up	187
12.6	Re-export	188
12.7	Liability Claims for Defects / Warranty	189
12.8	Note regarding disposal / options to reuse:	189
12.8.1	Decommissioning	190
12.8.2	Disposal of new device packaging	190
12.8.3	Disposing of the old device	190
12.8.4	Disposal of electrical and electronic components	190
12.8.5	Disposal of lubricants and coolants	191
12.9	Disposal via municipal collection	191
12.10	RoHS, 2002/95/EC	191



## Preface

Dear customer,

Thank you very much for purchasing a product made by OPTIMUM.

OPTIMUM metal working machines offer a maximum of quality, technically optimum solutions and convince by an outstanding price performance ratio. Continuous enhancements and product innovations guarantee state-of-the-art products and safety at any time.

Before commissioning the machine please thoroughly read these operating instructions and get familiar with the machine. Please also make sure that all persons operating the machine have read and understood the operating instructions beforehand.

Keep these operating instructions in a safe place nearby the machine.

### Information

The operating instructions include indications for safety-relevant and proper installation, operation and maintenance of the machine. The continuous observance of all notes included in this manual guarantee the safety of persons and of the machine.

The manual determines the intended use of the machine and includes all necessary information for its economic operation as well as its long service life.

In the paragraph "Maintenance" all maintenance works and functional tests are described which the operator must perform in regular intervals.

The illustration and information included in the present manual can possibly deviate from the current state of construction of your machine. Being the manufacturer we are continuously seeking for improvements and renewal of the products. Therefore, changes might be performed without prior notice. The illustrations of the machine may be different from the illustrations in these instructions with regard to a few details. However, this does not have any influence on the operability of the machine.

Therefore, no claims may be derived from the indications and descriptions. Changes and errors are reserved!

Your suggestion with regard to these operating instructions are an important contribution to optimising our work which we offer to our customers. For any questions or suggestions for improvement, please do not hesitate to contact our service department.

**If you have any further questions after reading these operating instructions and you are not able to solve your problem with a help of these operating instructions, please contact your specialised dealer or directly the company OPTIMUM.**

Optimum Maschinen Germany GmbH

Dr.- Robert - Pfleger - Str. 26

D-96103 Hallstadt

Mail: [info@optimum-maschinen.de](mailto:info@optimum-maschinen.de)

Internet: [www.optimum-maschinen.com](http://www.optimum-maschinen.com)



## 1 Safety

This part of the operating instructions

- explains the meaning and use of the warning notes included in these operating instructions,
- defines the intended use of the CNC machine ,
- points out the dangers that might arise for you or others if these instructions are not observed,
- informs you about how to avoid dangers.

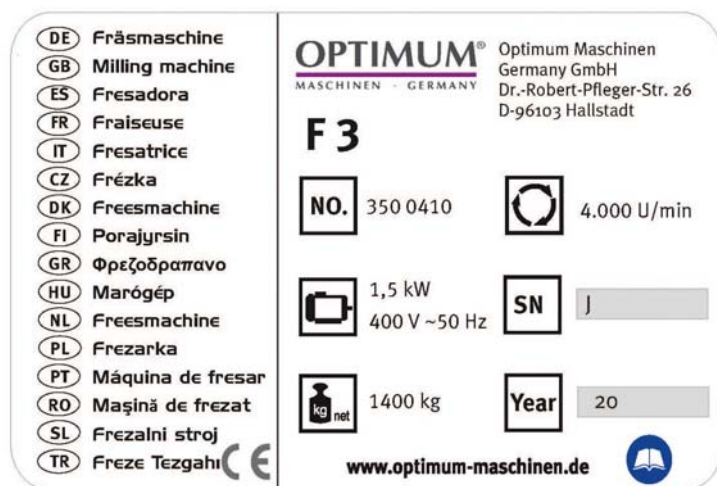
In addition to these operation instructions, please observe

- the applicable laws and regulations,
- the statutory provisions for accident prevention,
- the prohibition, warning and mandatory signs as well as the warning labels on the CNC machine.

Always keep this documentation close to the CNC machine.

If you would like to order another documentation for your CNC machine, please indicate the serial number of your CNC machine. The serial number is printed on the type plate.

### 1.1 Type plate



### 1.2 Glossary of symbols

- provides further instructions
- calls on you to act
- enumerations



## 1.3 Safety instructions (warning notes)

### 1.3.1 Classification of hazards

We classify the safety warnings into different categories. The table below gives an overview of the classification of symbols (ideogram) and the warning signs for each specific danger and its (possible) consequences.

Symbol	Warning alert	Definition / consequence
	<b>DANGER!</b>	Impending danger that will cause serious injury or death to people.
	<b>WARNING!</b>	A danger that can cause serious injury or death.
	<b>CAUTION!</b>	A danger or unsafe procedure that can cause personal injury or damage to property.
	<b>ATTENTION!</b>	Situation that could cause damage to the CNC-machine and products and other types of damage. No risk of injury to people.
	<b>INFORMATION</b>	Practical tips and other important or useful information and notes. No dangerous or harmful consequences for people or objects.

In case of specific dangers, we replace the pictogram with



### 1.3.2 Other pictograms





Warning: suspended loads!



Warning of oxidizing substances!



Caution, danger of explosive substances!



Warning: danger of slipping!



Protect the environment!



Contact address

## 1.4 Intended use

### WARNING!

#### In the event of improper use of the CNC machine

- will endanger personnel,
- the CNC machine and other material property of the operating company will be endangered,
- the correct function of the CNC machine may be affected.

The CNC machine is designed and manufactured to be used for milling and drilling cold metals or other non-flammable materials or materials that do not constitute a health hazard by using commercial milling and drilling tools.

Using this machine it is possible to perform dry processing as well as processing by using cooling lubricants. ➡ "Cooling lubricants" on page 160

The limit values of the balances of the tools need to be observed. ➡ "Tools and tool holding fixtures" on page 23

The CNC machine must only be installed and operated in a dry and well-ventilated place.

The CNC machine is designed and manufactured to be used in a non-explosive environment.

If the CNC machine is used in any way other than described above, modified without the Intended use approval of the company Optimum Maschinen Germany GmbH then the CNC machine is being used improperly.

We will not be held liable for any damages resulting from any operation which is not in accordance with the intended use.

We expressly point out that the guarantee or CE conformity will expire, if any constructive, technical or procedural changes are not performed by the company Optimum Maschinen Germany GmbH.

It is also part of intended use that you

- the limits of performance of the CNC machine are observed,
- the operating manual is observed,
- the inspection and maintenance instructions are observed.

### WARNING!

#### Severe injuries due to non-intended use.

It is forbidden to make any modifications or alternations to the operation values of the CNC machine. They could endanger the personnel and cause damage to the CNC machine.







## 1.5 Reasonably foreseeable misuse

Any other use other than that specified under "Intended use" or any use beyond the described use shall be deemed as non-intended use and is not permissible.

Any other use has to be discussed with the manufacturer.

It is only allowed to process metal, cold and non-inflammable materials with the milling machine.

In order to avoid misuse, it is necessary to read and understand the operating instructions before first commissioning.

Operators must be qualified.

### 1.5.1 Avoiding misuse

- Use of suitable cutting tools.
- Adapting the speed adjustment and feed to the material and workpiece.
- Clamp workpieces firmly and vibration-free.

#### ATTENTION!

**The workpiece is always to be fixed by a machine vice, jaw chuck or by another appropriate clamping tool such as for the clamping claws.**



#### WARNING!

Risk of injury caused by workpieces flying off.

Clamp the workpiece in the machine vice. Make sure that the workpiece is firmly clamped in the machine vice resp. that the machine vice is firmly clamped on the machine table.



- Use cooling and lubricating agents to increase the durability of the tool and to improve the surface quality.
- Clamp the cutting tools and workpieces on clean clamping surfaces.
- Sufficiently lubricate the machine.
- Correctly adjust the bearing clearance and the guidings.

#### ATTENTION!

**Do not use the drill chuck for milling tools. Never clamp a milling cutter into a drill chuck. Use a collet chuck with collets for the end mill.**



## 1.6 Possible dangers caused by the CNC machine

The CNC machine was tested for operational safety. The construction and type are state of the art.

Nevertheless, there is a residual risk as the CNC machine operates with

- rotating parts,
- electrical voltage and currents,
- compressed air,
- rapid moves.

We have used construction resources and safety techniques to minimize the health risk to personnel resulting from these hazards.

If the CNC machine is used and maintained by personnel who are not duly qualified, there may be a risk resulting from incorrect or unsuitable maintenance of the lathe.

#### INFORMATION

Everyone involved in the assembly, commissioning, operation and maintenance must

- be duly qualified,





- strictly follow these operating instructions.

In the event of improper use

- there may be a risk to personnel,
- there may be a risk of damage to the CNC machine and other material values,
- the correct function of the CNC machine may be affected.

Always switch off the CNC machine and disconnect it from the mains if you perform cleaning or maintenance works.

## WARNING!

**The CNC machine may only be used with functional safety devices. Disconnect the CNC machine immediately, whenever you detect a failure in the safety devices or when they are not fitted!**



**All additional parts of the machine which had been added by the customer need to be equipped with the prescribed safety devices.**

**This is your responsibility being the operating company!**

## 1.7 Qualification of personnel

### 1.7.1 Target group

This manual is addressed to

- the operating companies,
- operators having sufficient specialist knowledge,
- the maintenance personnel.

Therefore, the warning notes refer to both, operation and maintenance personnel of the CNC machine.

Determine clearly and explicitly who will be responsible for the different activities on the CNC machine (operation, setting up, maintenance and repair). Please note the name of the responsible person into an operators's log.

## INFORMATION

Unclear responsibilities constitute a safety risk!

Always lock the main switch after switching off the CNC machine. This will prevent it from being used by unauthorized persons.

The qualifications of the personnel for the different tasks are mentioned below:

### Operator

The operator has been instructed by the operating company regarding the assigned tasks and possible risks in case of improper behaviour. Any tasks which need to be performed beyond the operation in standard mode must only be performed by the operator, if so indicated in these instructions and if the operator has been expressly commissioned by the operating company.

### Qualified electrician

With professional training, knowledge and experience as well as knowledge of respective standards and regulations, qualified electricians are able to perform work on the electrical system and recognise and avoid any possible dangers.

Qualified electricians have been specially trained for the working environment, in which they are working and know the relevant standards and regulations.





### Qualified personnel

Thanks to professional training, knowledge and experience as well as knowledge of relevant regulations the qualified personnel is able to perform the assigned tasks and to independently recognise and avoid any possible dangers themselves.

### Instructed person

Instructed persons were instructed by the operating company regarding the assigned tasks and any possible risks of improper behaviour.

### 1.7.2 Authorized personnel

#### INFORMATION

**Sufficient expertise is required for working on the CNC machine. No one must work on the machine without having the necessary training, not even for a short while.**

**We recommend the use of the CNC software SinuTrain as an aid for training and operation.**

SinuTrain made by Siemens is the perfect software supplement for the CNC machine F3 of OPTIMUM.

This training software supports the rapid training for the operation of the control Sinumerik Siemens SINUMERIK 808D. Employees having little CNC-experience can learn the basics of the DIN-programming by using SinuTrain and are finally able to write and test programs using SINUMERIK 808D.

Please find SinuTrain and further information on the website of Siemens.  
<http://www.cnc4you.siemens.com>



#### WARNING!

**Inappropriate operation and maintenance of the CNC machine constitutes a danger for the personnel, objects and the environment.**

**Only authorized personnel may operate the CNC machine !**

Persons authorized to operate and maintain should be trained technical personnel and instructed by the ones who are working for the operating company and for the manufacturer.



#### The operating company must

- train the personnel,
- instruct the personnel in regular intervals (at least once a year) on
  - all safety standards that apply to the CNC machine,
  - operation of the CNC machine,
  - generally accepted engineering standards.
  - possible emergency situations,
- check the personnel's knowledge level,
- document training/instruction in a operation book,
- require personnel to confirm participation in training/instructions by means of a signature,
- check whether the personnel is working safety and risk-conscious and observes the operating instructions.
- define and document the inspection deadlines for the machine in accordance with the Factory Safety Act and perform an operational risk analysis in accordance with the Work Safety Act.

Obligations of  
the operating  
company

#### The operator must

- be specially trained in handling and programming the CNC machine,
- know and understand the program sequence and which effects the individual process parameters will have,
- keep an operator's log,

Obligations of  
the operator



- before taking the machine in operation
  - have read and understood the operating manual,
  - be familiar with all safety devices and instructions.

For work on the following CNC- machine parts there are additional requirements:

- Electric components or operating materials: Must only be worked on by a qualified electrician or person working under the instructions and supervision of a qualified electrician.

Additional requirements regarding the qualification

## 1.8 Operator positions

The operator position is in front of the CNC machine at the sight window or on the machine control panel.

## 1.9 Safety devices

Use the CNC machine only with properly functioning safety devices.

Stop the CNC machine immediately if safety device fails or is not functioning for any reason.

It is your responsibility!

If a safety device has been activated or has failed, the drilling machine must only be used if you

- the cause of the fault has been eliminated,
- you have verified that there is no danger to personnel or objects.

### WARNING!

**If you bypass, remove or deactivate a safety device in any other way, you are endangering yourself and other personnel working with the CNC machine. The possible consequences are:**



- **injuries due to tools, workpieces or fragments hereof which are flying off at high speed,**
- **contact with rotating or moving parts,**
- **fatal electrocution,**
- **pulling-in of clothes.**

The CNC machine includes the following safety devices:

- a lockable main switch,
- One EMERGENCY STOP push-button on the machine control panel and on the electronic handwheel,
- A locked, separating protective equipment around the CNC milling machine with sight windows made of break-proof Makrolon.
- Locking switch on the separating safety devices.

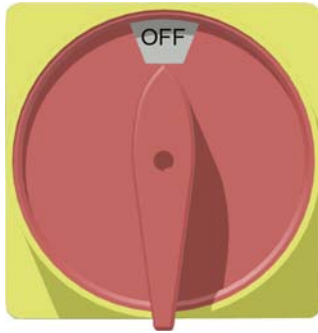
### 1.9.1 Lockable main switch

In the "0" position, the lockable main switch can be secured against accidental or non-authorised switching on by means of a padlock.

The power supply is interrupted by switching off the main plug.

Except for the areas marked by the pictogram in the margin. In these areas there might be voltage, even if the main switch is switched-off.





Img.1-1: Main switch

## WARNING!

**Dangerous voltage even if the main switch is switched off.**

The areas marked by the pictogram might contain live parts, even if the main switch is switched off.



## 1.9.2 Emergency-stop button

### ATTENTION!

The **EMERGENCY STOP** push button immediately stops the operation of the CNC machine.

Press the **EMERGENCY STOP** button only if there is a risk! If this push button is actuated in order to switch off the CNC machine in the standard operation the tool or workpiece might get damaged.

After having actuated the EMERGENCY-STOP mushroom switch, turn the knob of the particular push button to the right in order to restart the machine.



Img.1-2: Emergency-stop push button



## 1.9.3 Control technical protection

### WARNING!

If you bypass a controller you endanger yourself and other persons working on the CNC machine.

- injuries due to tools, workpieces or fragments hereof which are flying off at high speed,
- contact with rotating parts,
- fatal electrocution,
- pulling-in of clothes.

If you bypass a controller in exceptional cases (e.g. during electrical repairs) short term you must continuously monitor the CNC machine during this time.




## 1.9.4 Polycarbonate windows

Polycarbonate viewing window in chip protection, must be visual inspected by the customer responsible personnel at regular intervals to guarantee the operational safety of the CNC machine.

Polycarbonate viewing panes are subject to an ageing process and are classified as wear parts.

The aging of polycarbonate windows can not be detected by visual inspection. It is therefore necessary to replace the polycarbonate windows after a certain time.

Prolonged exposure from polycarbonate windows to cutting fluids can lead to accelerated ageing, i.e. deterioration of the mechanical properties (brittleness). Coolant vapours, detergents, greases and oils or other corrosive substances from the operator side can also lead to a deterioration of the polycarbonate windows. The result in reduced impact resistance of the polycarbonate windows.

 "Cleaning and replacing of the polycarbonate windows" on page 165

## 1.9.5 Prohibition, warning and mandatory labels

### INFORMATION

All warning and mandatory signs must be legible. They must be checked regularly.



## 1.10 Safety check

Check the CNC machine at least once per shift. Inform the person responsible immediately of any damage, defects or changes in the operating function.

Check all safety devices

- at the beginning of each shift (when the machine is operated continuously),
- once per day (during one-shift operation),
- once per week (when operated occasionally),
- after all maintenance and repair work.

Check that prohibition, warning and information signs and the labels on the CNC machine


- are legible (clean them, if necessary),
- and complete (replace them, if necessary).

### INFORMATION

Organise the checks according to the following table;





General check		
Equipment	Check	OK
Protective housing	Switching function, firmly bolted and not damaged	
Signs, Markers	Installed and legible	
Sight window	Check for mechanical damage (scratches, cracks). cracks,,  "Polycarbonate windows" on page 14	
Date:	checked by (signature):	

Functional check		
Equipment	Check	OK
EMERGENCY STOP push button	After actuating an EMERGENCY STOP push button the CNC machine must be switched off.	
Switch cabinet cooling	The cabinet cooling must be running.	
Separating protective equipment around the CNC machine	If the protective equipment is open it must not be possible to start program.	
Date:	checked by (signature):	

## 1.11 Personal protective equipment

For certain work personal protective equipment is required.

Protect your face and your eyes: Wear a safety helmet with facial protection when performing work where your face and eyes are exposed to hazards.

Wear protective gloves when handling pieces or tools with sharp edges.

Wear safety shoes when you assemble, disassemble or transport heavy components.

Use ear protection if the noise level (emission) in the workplace exceeds 80 dB (A).

Before starting work make sure that the required personnel protective equipment is available at the work place.

### CAUTION!

**Dirty or contaminated personnel protective equipment can cause illness. It must be cleaned after each use and at least once a week.**



## 1.12 Safety during operation

### WARNING!

**Before activating the CNC machine, ensure that this will not endanger other persons or cause damage to equipment.**

Avoid any unsafe work methods:

- The instructions mentioned in these operating instructions have to be strictly observed during assembly, operation, maintenance and repair.







- Do not work on the CNC machine, if your concentration is reduced, for example, because you are taking medication.
- Stay on the CNC machine until the program is terminated.

The running program can be identified by means of the signal lamp.

- Green light: Program run active
- Yellow light: Malfunction
- Red light: Actuated emergency stop push button



Img. 1-3: Signal lamp

- Safely and firmly clamp the workpiece before switching on the CNC machine.
- Never change the dosing of the coolant supply during operation.
- Never open the sliding door of the separating protective unit when the CNC-program is running.

## WARNING!

**When chipping magnesia materials (aluminium-/magnesium alloys), spontaneously inflammable or explosive particles (powder, dust, chips) might be generated, which might cause a fire and/or explosion (deflagration).**

**Magnesium is designated a dangerous material in the list of dangerous materials and preparations according to para. 4a of the Ordinance of Hazardous Substances.**

**In case of a fire with magnesium, only use appropriate and admitted extinguishing agents. Never extinguish using water. If burning magnesium is extinguished with water, this might lead to dangerous reactions (hydrogen gas). Water would be decomposed in its components hydrogen (H) and oxygen (O).**

**Only the following extinguishing agents are permissible:**

- solid extinguishing agent of fire class D (fires involving metals)
- dry covering salts for magnesium
- a mixture of sand and cast chips
- argon (Ar) or nitrogen (N<sub>2</sub>)

**If fine mist and smoke is generated at the workplace, suction units must be provided in order to avoid the accumulation of ignitable mixtures and emissions.**

We provide information about the specific dangers when working with and on the CNC machine in the descriptions for these types of work.



## 1.13 Safety during maintenance

Inform the operators in good time of any maintenance and repair works.

Report all safety-relevant changes and performance characteristics of the CNC machine. Any changes must be documented, the operating instructions updated and machine operators instructed accordingly.





### 1.14 Disconnecting and securing the CNC machine

Turn off the main switch of the CNC machine before starting any maintenance or repair work.

Use a padlock to prevent the switch from being turned on without authorization and keep the key in a safe place.

All machine parts as well as all dangerous voltages are switched off.

Excepted are only the positions which are marked with the adjoining pictogram. These positions may be live, even if the main switch is switched off.

Place a warning sign on the CNC machine.



#### **WARNING!**

**Live parts and moves of machine parts can injure you or others dangerously!**

**Proceed with extreme care if you cannot switch off switch due to required works (e.g. functional control).**



#### 1.14.1 Using lifting equipment

#### **WARNING!**

The use of unstable lifting and load suspension equipment that might break under load can cause severe injuries or even death. Observe the accident prevention regulations issued by your Employers Liability Insurance Association or other supervisory authorities responsible for your company.

Check that the lifting and load-suspension equipment are of sufficient load-bearing capability and are in perfect condition.

Fasten the loads carefully.

Never walk under suspended loads!



#### 1.14.2 Mechanical maintenance work

Remove or install protection safety devices before starting or after completing any maintenance work; this include:

- covers,
- safety instructions and warning signs,
- grounding cables.

If you remove protective or safety devices, re-fit them immediately after the completing the work.

Check if they are working properly!

### 1.15 Unattended Operation

CNC machines are designed for unattended operation. However, it may not be safe to let your machining process run unmonitored. As it is the shop owner's responsibility to set up the machine safely and use best practice machining techniques, it is also their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage in case of a hazardous situation.

For example, if there is a risk of fire due to the machined material, an appropriate fire suppression system must be installed to reduce the risk of harm to personnel, equipment and the building. Have a specialist supplier install monitoring tools, before allowing machines to run unattended.

It is especially important to select monitoring equipment that can immediately perform an appropriate action without human intervention to prevent an accident, should a problem be detected.



## 1.16 Accident report

Inform your supervisors and Optimum Maschinen Germany GmbH immediately in the event of accidents, possible sources of danger and any actions which almost led to an accident (near misses).

There are many possible causes for "near misses".

The sooner they are notified, the quicker the causes can be eliminated.

### INFORMATION

We specifically point out the dangers when describing the work with and on the lathe.



## 1.17 Electrical system

Have the machine and/or the electric equipment checked regularly. Immediately eliminate all defects such as loose connections, defective wires, etc.

A second person must be present during work on live components to disconnect the power in the event of an emergency. Disconnect the CNC machine immediately if there is a malfunction in the power supply!

Comply with the required inspection intervals in accordance with the factory safety directive, operating equipment inspection DGUV, formerly BVG.

The operator of the machine must ensure that the electrical systems and operating equipment are inspected with regards to their proper condition, namely,

- by a qualified electrician or under the supervision and direction of a qualified electrician, prior to initial commissioning and after modifications or repairs, prior to recommissioning
- and at certain intervals.

The deadlines must be set so that arising, foreseeable defects can be detected in time.

The relevant electro-technical rules must be followed during the inspection.

The inspection prior to initial commissioning is not required if the operator receives confirmation from the manufacturer or installer that the electrical systems and operating equipment comply with the accident prevention regulations, see conformity declaration.

Permanently installed electrical systems and operating equipment are considered constantly monitored if they are continually serviced by qualified electricians and inspected by means of measurements in the scope of operation (e.g. monitoring the insulation resistance).

## 1.18 Inspection deadlines

Define and document the inspection deadlines for the machine in accordance with § 3 of the Factory Safety Act and perform an operational risk analysis in accordance with § 6 of the Work Safety Act. Also use the inspection intervals in the maintenance section as reference values.

## 1.19 Clamping devices for workpieces and tools

### ATTENTION!

**Attention when taking over existing clamping devices. Please thoroughly check that the clamping device is appropriate for your CNC machine.**

- **Only use clamping devices with a complete inherent rigidity.**
- **Contact the manufacturer of the clamping device regarding the reuse of clamping devices after damage to the clamping device due to collisions.**
- **Correctly insert the workpiece and make sure that the machine is proper working condition.**



## 1.20 Environmental protection and water conservation

The CNC machine is a device to produce, handle and use materials which are hazardous to water according to para. 19g of the Water Resources Law.





Please follow the requirements of the Water Resources Law when operating, decommissioning or disassembling the CNC machine or parts hereof. Detailed information regarding this can be found in the Ordinance on Installations for the Handling of Substances Hazardous to Water (VAwS).



## 2 Technical data

The following information represents the dimensions and indications of weight and the manufacturer's approved machine data.

### 2.1 Electrical connection

Total connection	3 x 400V ~ 50Hz 2.5 KW
------------------	------------------------

### 2.2 Milling spindle

performance spindle drive	1.5 KW
Torque	10 Nm
Speed	200 - 4000 min <sup>-1</sup>
Spindle seat	ISO 30 / BT 30 x 45°

### 2.3 Lubricant/coolant system

Power of the coolant/lubricant pumps [W]	95
Tank capacity [l]	50

### 2.4 Compressed air

Connection [MPa]	0.8 (8 bar)	
Air consumption	0.01 - 0.015 m <sup>3</sup> / h at 0.8 MPa	
Recommended compressed air quality	ISO 8573.1 class 2	
Compressed air consumer	• Tool clamp / unclamp	
Compressed air service unit	Drainage filters	40 µm
	max. pressure	1.5 MPa
	Housing	Polycarbonate
	Recommended oil lubricator for compressed air	ISO VG32
	Quantity of oil Lubricator for compressed air	90ml
	Drainage	manually
	Compressed air connection thread	3/8"



## 2.5 Dimensions

Height [mm]	☞ "Installation plan F3" on page 27
Depth [mm]	
Width [mm]	
Total weight [kg]	1400

## 2.6 Working area, traverse path

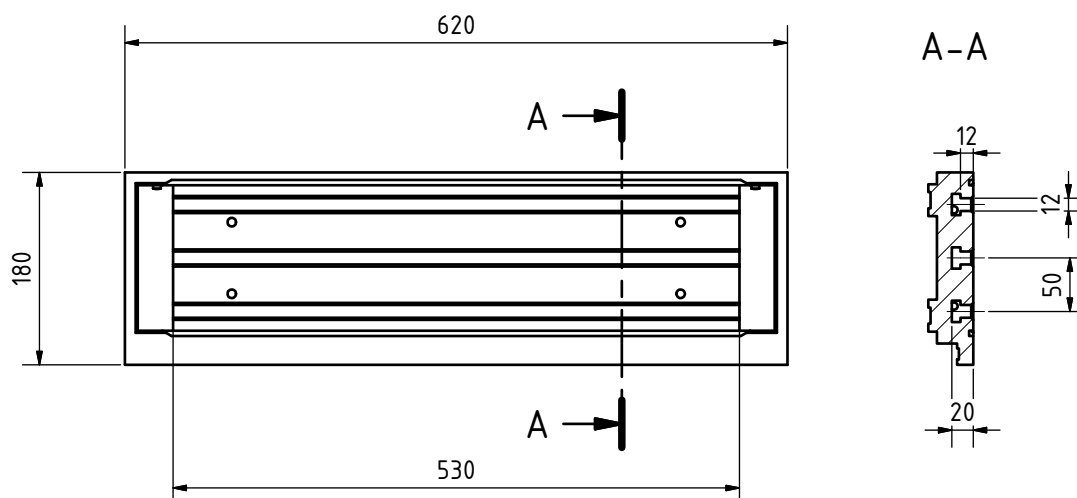
X-axis [mm]	355
Y-axis [mm]	190
Travel of Z axis [mm]	245
Z-axis Distance spindle - milling table [mm]	20 until 305

## 2.7 Accuracy

Repeat accuracy	$\pm 0,015\text{mm}$
Positioning accuracy	$\pm 0,015\text{mm}$

## 2.8 Milling table

Table length [mm]	620
Table width [mm]	180
T - slot size / number / distance [mm]	12 / 3 / 50
Max. load [kg]	30





## 2.9 Environmental conditions - operation

Temperature	19 - 21 °C (for an optimum milling result) 19 - 21 °C (for operation without malfunctions)
Admissible relative humidity	5...80 % no condensation
Compressed air	700...1060 hPa

## 2.10 Environmental conditions - storage

Temperature	5 - 45 °C
-------------	-----------

## 2.11 Emissions

The noise emission of the CNC machine is 80 dB(A).

If the CNC machine is installed in an area where various machines are in operation, the noise exposure (immission) on the operator of the drilling machine at the working place may exceed 80 dB(A).

### INFORMATION

This numerical value was measured on a new machine under the operating conditions specified by the manufacturer. The noise behaviour of the machine might change depending on the age and wear of the machine.

Furthermore, the noise emission also depends on production engineering factors, e.g. speed, material and clamping conditions.

### INFORMATION

The specified numerical value represents the emission level and does not necessarily a safe working level.

Though there is a dependency between the degree of the noise emission and the degree of the noise disturbance it is not possible to use it reliably to determine if further precaution measures are required or not.

The following factors influence the actual degree of the noise exposure of the operator:

- Characteristics of the working area, e.g. size or damping behaviour,
- other noise sources, e.g. the number of machines,
- other processes taking place in proximity and the period of time, during which the operator is exposed to the noise.

Furthermore, it is possible that the admissible exposure level might be different from country to country due to national regulations.

This information about the noise emission should, however, allow the operator of the machine to more easily evaluate the hazards and risks.

### CAUTION!

Depending on the overall noise exposure and the basic threshold values, machine operators must wear appropriate hearing protection.

We generally recommend the use of noise protection and hearing protection.





## 2.12 Tools and tool holding fixtures

### CAUTION !

**When using tools with larger diameters or at higher speeds!**

The balancing of the tools has to amount to

- 0 - 6000 min<sup>-1</sup> - G 6,3
- from a speed of 6000 min<sup>-1</sup> - G 2,5

according to DIN / ISO 1940.









### 3 Assembly and commissioning

#### INFORMATION

The CNC-machine is delivered pre-assembled. It is delivered in a transport box.



#### 3.1 Scope of delivery

Compare the delivery volume with the attached packing list.

Check the status of the CNC machine immediately upon receipt and claim possible damages at the last carrier also if the packing is not being damaged. In order to ensure claims towards the freight carrier we recommend you to leave the machines, devices and packing material for the time being in the status at which you have determined the damage or to take photos of this status. Please inform us about any other claims within six days after receipt of delivery.

Check if all parts are firmly seated.

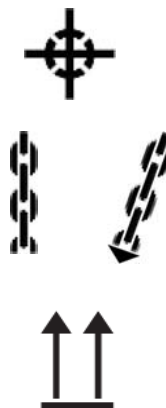
#### 3.2 Transport

##### WARNING!

Severe or fatal injuries may occur if the machine or parts of the machine tumble or fall down from the forklift truck or from the transport vehicle. Follow the instructions and information on the transport box:



- Centres of gravity
- Load suspension points  
(Marking of positions for the load suspension point)
- Prescribed transportation position  
(Marking of the top surface)
- Means of transport to be used
- Weights



##### WARNING!

The use of damaged lifting and load suspension equipment without sufficient load capacity that might break under load can cause severe injuries or even death.

Check that the lifting and load suspension equipment has sufficient load capacity and that it is in perfect condition.

Observe the accident prevention regulations.

Fasten the loads carefully.

**Never walk under suspended loads!**

- Check the substructure. The substructure must bear the load.
- Disassemble the lateral parts of the wooden box.
- The CNC machine is lifted and transported with an appropriate handling device to the installation place by means of a fork-lift truck.
- Disassemble the clamping bolts which are used to fix the machine on the pallet.





- ➔ Lift the CNC machine carefully from the pallet of the transportation box by means of a crane or a fork-lift truck. "Total weight [kg]" on page 21.
- ➔ Bring the CNC machine with an appropriate handling device, e.g. Electric pallet truck or fork-lift truck at their firm position.
- ➔ Make sure that no add-on-pieces are damaged or cause paintwork is damaged during transport.

## WARNING!

**The use of unstable lifting and load suspension equipment that might break under load can cause severe injuries or even death.**



"Machine mounting" on page 28

## 3.3 Installation and assembly

### 3.3.1 Requirements regarding the installation site

Organize the working area around the CNC machine according to the local safety regulations.

The working area for operating, maintenance and repair must not be restricted. Follow the prescribed safety areas and escape routes according to VDE 0100 part 729 as well as the environmental conditions for the operation of the CNC machine.

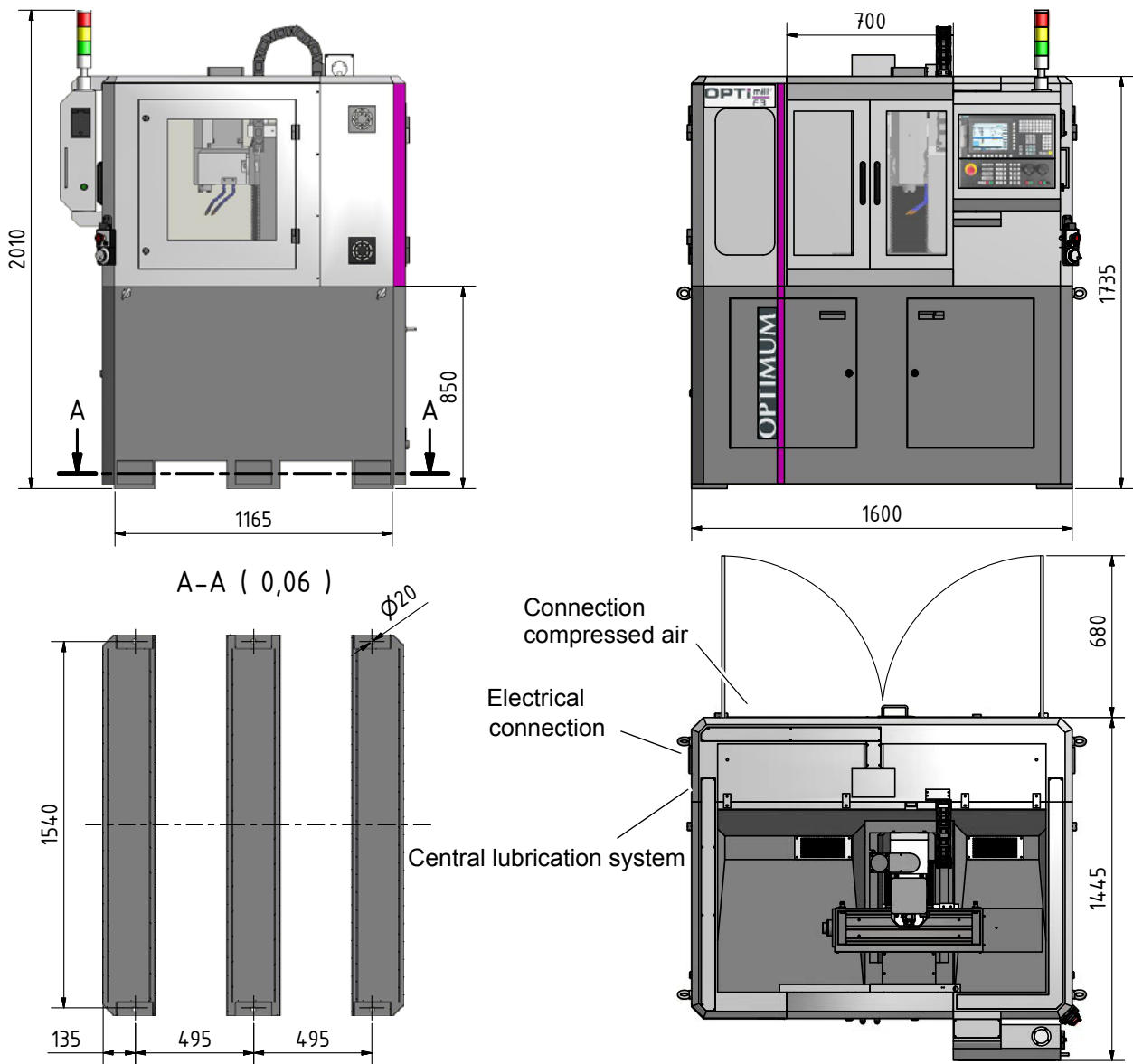
## INFORMATION

The main switch of the CNC machine must be freely accessible.





### 3.3.2 Installation plan F3

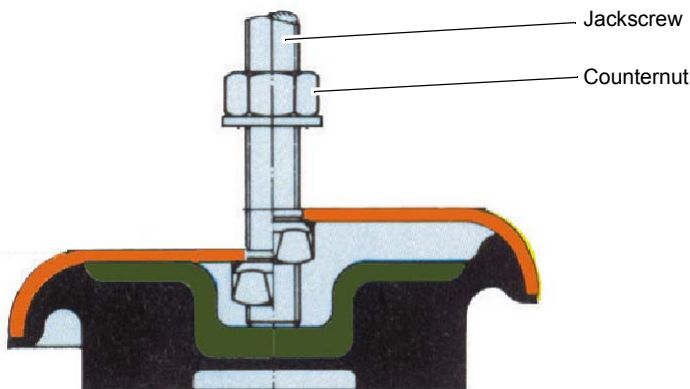




### 3.3.3 Machine mounting

#### Anchor-free assembly

- Align the CNC machine with a machine spirit level. The slope deviation of all levels must not exceed 0.04/1000mm.
- Adjust the height by screwing in or screwing out of the levelling screw. Screwing in the levelling screw causes that the rubber plate lifts itself as on the drawing from the same element.
- Fix the height adjustment by the the jackscrew with the help of the counternut.
- Check the alignment of the machine after a few days of usage.



Img.3-1: Oscillating element

#### Anchored assembly

Use the anchored assembly in order to attain a firm connection to the ground. An anchored assembly is always reasonable if parts are manufactured to the maximum capacity of the CNC machine.

#### ATTENTION!

**An insufficient rigidity of the substructure leads to superposition of vibrations between the CNC machine and the substructure (natural frequency of the components). Critical speeds and moves in the axis with displeasing vibrations are rapidly achieved in case of insufficient rigidity of the whole system and will lead to bad milling results.**



- Check the correct alignment of the machine after a few days of usage.

### 3.3.4 Corrosion protection

- A corrosion protection is applied on the machine table and on the guiding surfaces for transport and storage. Remove the anti-corrosive agent from the CNC machine before first commissioning. Therefore, we recommend you to use paraffin.



### 3.3.5 Electrical connection

- Check the fusing (fuse) of your electrical supply according to the technical instructions regarding the total connected power of the milling machine.
- Firmly connect the machine.

#### CAUTION!

**Install the connection cable of the machine in such a way that people will not stumble over it.**



Please verify if the type of current, voltage and protection fuse correspond to the values specified. A protective earth ground wire connection must be available.

Main Fuse 16A.

With an internal EMC filter the leakage current of the frequency converter of milling spindle is greater than 3.5 mA. We ask for due attention while executing machine tests within the framework of industrial safety guidelines.

#### ATTENTION!

**When delivered the machine is equipped with a plug for electrical connection. It only serves for acceptance and test purposes. In order to operate the machine it is necessary to remove this plug and to connect the machine directly with a power supply.**



Firmly connect the CNC machine to the terminal box. It is not allowed to connect the machine using a standard 16A CEE plug, since the stray current of the frequency converter is exceeding the admissible value of 3.5mA (refer to EN 50178 / VDE 5.2.11.1).

#### ATTENTION!

Depending on the quality of the network, there is a risk of machine malfunctions under extreme conditions. If necessary and in order to exclude retroactive effects on the internal power supply system, the operator should install a line filter on the machine. Therefore, at workplaces with lots of powerful consumers, it might also be necessary to use a system for network compensation. Please consult your electricity supplier regarding this.



#### ATTENTION!

Frequency converters (drive regulators) might trigger the FI circuit breaker of your electrical supply. In order to avoid malfunction, an FI circuit breaker switch sensitive to pulse current or to universal current may be required.



#### ATTENTION!

**Ensure that all 3 phases (L1, L2, L3) and the ground wire are connected correctly.  
The neutral conductor (N) of its power supply is not connected.**



### 3.3.6 Current in the Protective Earth Ground Wire

The degree of the leakage current in the protective earthing conductor depends on whether the internal EMC filter in the Emerson M200 frequency converter is used for spindle rotation speed control or not. The standard frequency converter is fitted with an internal EMC filter. Instructions on how to remove the internal filter is in the converter manual.

- With the EMC filter, the leakage current is 15.4 mA AC 230V 50Hz (1 phase supply, phase-neutral power supply, neutral point earthed).
- Without an EMC filter, the leakage current is <1.9 mA (phase-neutral power supply, neutral point earthed).

Therefore, a fixed earth connection is required and the minimum cross section of the protective earthing conductor must conform to local safety regulations for devices with high leakage current. This is achieved by providing a permanent fixed earthing connection with two independent conductors, each having a cross section the same as the power supply cord or greater. To simplify this, the converter is provided with two earth terminals. Both earthing connections are required to comply with the standard EN 61800-5-1.

Since a direct current may be caused by the frequency converter in the protective earthing conductor, if an upstream residual current device (ELCB / RCD) is required in the network, the following guidelines must be followed:

There are three common types of FI (ELCB / RCD):

- AC - to detect AC fault currents
- A - to detect AC fault currents and pulsating DC fault currents (provided the DC current reaches zero at least once every half cycle).
- B - to detect AC fault currents, pulsating DC fault currents and smooth DC residual currents.

Type AC should never be used in converters.

Type A can only be used for single-phase converters.

Type B must be used for 3-phase converters.

When using an external EMC filter, to avoid false error shutdowns, a time delay of at least 50 ms is required. The leakage current can exceed the threshold trigger value for an error shutdown if the phases are not switched on at the same time.

#### Line systems

The CNC milling machine is designed for TN and TT line systems with a grounded neutral point.

#### Prohibited operation

Operation on TN line systems with grounded external conductors is prohibited.

Operation on TT line systems without grounded neutral points is prohibited.

Operation on IT line systems is not permitted. In an IT line system, all of the conductors are insulated with respect to the PE protective conductor – or connected to the PE protective conductor through an impedance. Operation on an IT line system is not permitted.

#### Permissible line supplies

##### Operation on TN and TT line systems

##### TN line system

The TN line system in accordance with IEC 60364-1 (2005) transmits the PE conductor to the installation via a conductor. Generally, in a TN line system the neutral point is grounded. There are versions of a TN line supply with a grounded line the conductor, e.g. with grounded L1.

A TN line system can transfer the neutral conductor N and the PE protective conductor either separately or combined.



## TT system

In a TT line system, the transformer grounding and the installation grounding are independent of one another. There are TT line supplies where the neutral conductor N is either transferred – or not.

### 3.3.7 Connection compressed air supply

- Connect the compressed air supply with at least 6.5 bars to the quick-action coupling of the compressed air maintenance unit.
- Adjust a pressure of 6.3 bars using the set screw of the maintenance unit.

#### ATTENTION!

In order to ensure a failure-free operation of the machine it is necessary that the required air pressure is continuously applied on the machine at constant quality. In case of insufficient air supply, for instance interruptions occur during tool change.



## 3.4 First commissioning

### 3.4.1 Refilling the cooling lubricant

#### INFORMATION

The CNC machine is delivered without cooling lubricant.

- Fill the coolant / lubricant tank with an appropriate cooling lubricant via the machining room CNC machine. ☞ "Cooling lubricants" on page 160



#### ATTENTION!

Failure of the pumps in case of dry running. The pumps are lubricated by the cooling lubricant. Do not start up the pumps without cooling lubricant.



#### INFORMATION

Use a water-soluble environmentally friendly emulsion as cooling lubricant procured from a specialist retailer.

Make sure that the cooling lubricant is properly absorbed.

Respect the environment when disposing of lubricants and coolants. Follow the manufacturer's disposal instructions.



### 3.4.2 Cooling lubricants

#### INFORMATION

The lathe is lacquered with a **one-component paint**. Consider this fact when selecting your cooling lubricant.

The company Optimum Maschinen Germany GmbH does not assume any guarantee for subsequent damages due to unsuitable cooling lubricants.

The flashpoint of the emulsion must be higher than 140°C.

When using non-water-miscible cooling lubricants (oil content > 15%) with a flashpoint, ignitable aerosol air mixtures might develop. There is a potential danger of explosion.

The selection of cooling lubricants and slideway oils, lubricating oils or greases as well as their care are being determined by the machine operator or operating company.

Therefore, Optimum Maschinen Germany GmbH cannot be held liable for machine damages caused by unsuitable coolants and lubricants as well as by inadequate maintenance and servicing of the coolant. In case of problems with the cooling lubricant and the slideway oil or grease, please contact your mineral oil supplier.





## CAUTION!

**The cooling lubricant needs to be checked at least weekly, including during downtimes, with regard to its concentration, ph-value, bacteria and fungal decay.**



### "Cooling lubricants and tanks" on page 167

We would like to ask you to have the following machine-related properties of the cooling lubricant confirmed in writing by the manufacturer of the cooling lubricant.

- The products must comply with the provisions of the current statutory regulations and the employers' liability insurance association.
- Request documentation for the products such as the product description VKIS and EC safety data sheet from the cooling lubricants manufacturer. The EC safety data sheet gives you information about the water hazard class.

They need to be environmentally friendly and workplace-friendly. Thus, they need to be free of nitrite, PCB, chlorine and nitrosatable diethanolamine (DEA), according to TRGS 611.

- The manufacturer should be able to provide a certificate concerning skin tolerance.
- The mineral oil content according to DIN 51417 should be at least 40% in the concentrate.
- If possible, it should be universally applicable for all chippings and materials.
- Long service life of the emulsion e.g. long-term stable and resistant to bacteria.
- Safe corrosion protection according to DIN 51360/2.
- Re-emulsifiable and non-adhesive according to VKIS sheet 9: Sticking and residue behaviour.
- It should not attack the varnish of the machine according to VDI 3035.
- It should not attack any machine elements (metals, elastomers).
- Low foaming behaviour of the emulsion.
- It should be as finely dispersed as possible in order to avoid clogging in the needle slot screen.





## 3.5 Refill central lubrication system

The CNC machine is equipped with a central lubrication system.

### INFORMATION

The CNC milling machine is delivered without lubricating oil. The central lubricating system is located on the left-hand side of the machine housing.

The central lubrication system and the choke manifold system is preset and must not be adjusted or re-adjusted during the commissioning of the machine. The discharge volume is 1.0 cm<sup>3</sup> per cycle.

➔ Refill the reservoir with oil through the fill cap.

☞ "Cooling lubricants" on page 160

- The central lubrication system supplies the lubricating points automatically.
- The cycle time is fixed and can not be changed.
- The discharge volume per cycle can be reduced, however, should remain at the factory setting.

### Function:

Lubricator is a motor-driven, spring discharge piston pump. The motor incorporates a gear reduction which determines the operating cycle time of the pump piston. The SLR-chokes are cyclically supplied with the set flow rate of oil. The delivery rate is distributed over the various chokes to the lubrication points.

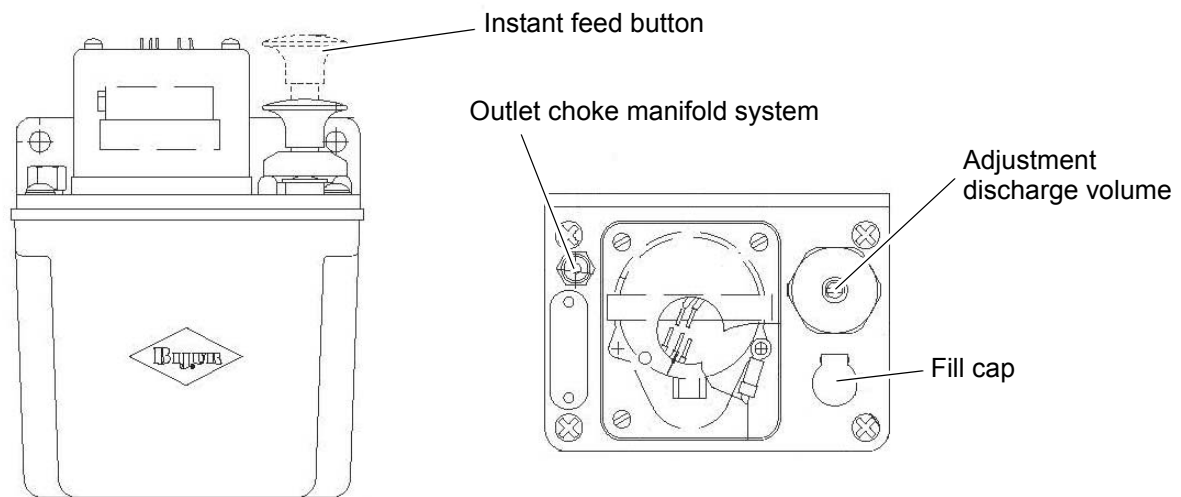
### Technical data:

Cycle time at 50Hz - 72 min, Cycle time at 60Hz - 60 min,

Reservoir capacity 0.474 liter, Lubricator inlet filter 40 µm, Discharge pressure 1.4 to 3.4 bar (20-50 psi), Factory discharge setting 1.0 cm<sup>3</sup>

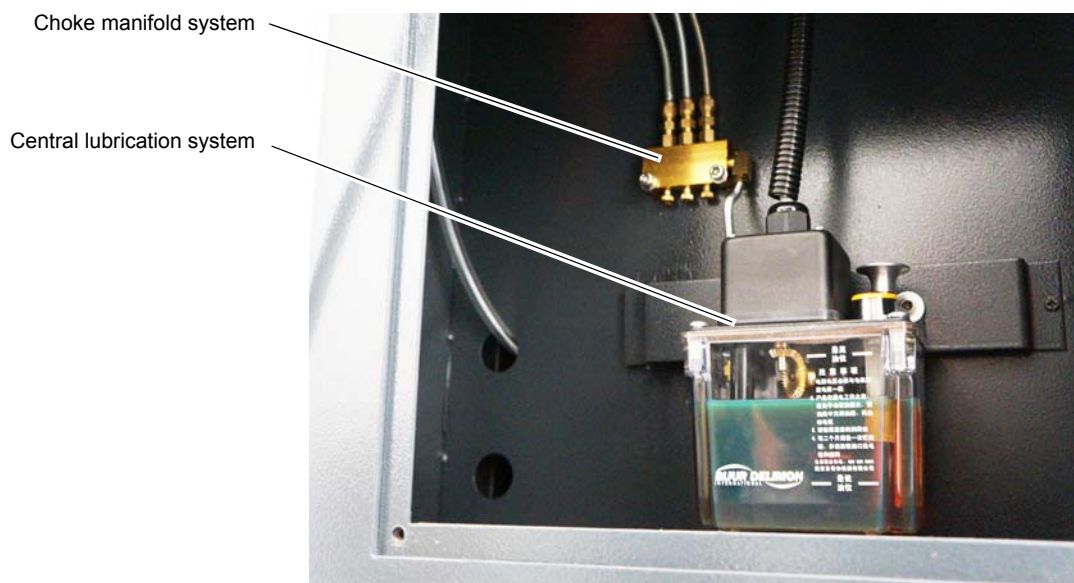
Discharge volume per cycle 0.2 cm<sup>3</sup> to 1.0 cm<sup>3</sup> adjustable

Viscosity range 35-1750 cSt (150 to 8,000 SSU) at operating temperature



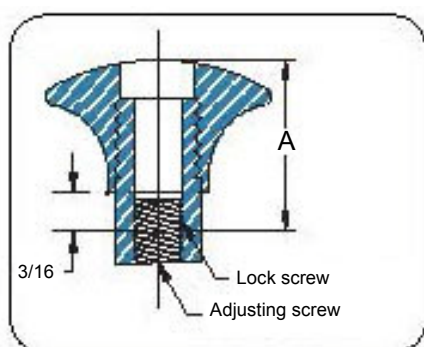
Img.3-2: central lubrication





## Reducing the discharge volume per cycle:

To reduce oil delivery, remove lock screw, measure "A", turn adjusting screw clockwise, increasing "A" by "B" dimension, as shown on table.



B	Discharge
11.2 mm	0.2 cc
8.4 mm	0.4 cc
5.6 mm	0.6 cc
2.8 mm	0.8 cc
0 mm	1.0 cc



### 3.6 Functional test and controls


#### Rotation coolant pump

- Check the direction of rotation of the CNC machine. There must be a right-handed rotating field. If the direction of rotation is wrong, exchange two of the three phase conductors. At this, the correct direction of rotation only refers to the drive of the cooling lubricant pumps.

#### INFORMATION

The rotational direction of the coolant pump can not be checked in the built in state. If no coolant flows the rotational direction of the coolant pump may be wrong.

Proceeding to set the direction of rotation:

-  "Turning on the CNC- machine" on page 55
- Close the sliding door and turn on the coolant supply.
- Check if cooling agent flows.



#### WARNING!

**Never change the dosing of the cooling lubricant supply when a program is running.**

- If no coolant is flowing and thus the rotational direction of the coolant pump may be wrong,
  - first check the coolant level in the coolant tank,
  - and then replace two of three phases (e.g. L1 and L2) in the control cabinet on the circuit breaker of the engine of coolant pump against each other.



#### WARNING!

**Swapping the phases must be performed by a qualified electrician!**

- Check all oil levels and filling levels of lubricants in the reservoirs.
- Perform a safety test.



#### 3.6.1 Warming up the machine

#### ATTENTION!

If the CNC machine and in particular the milling spindle is immediately operated at maximum load when it is cold it may result in damages.

If the machine is cold, e.g. directly after having transported the machine, it should be warmed up at a spindle speed of only 500 1/min for the first 30 minutes.





## 4 General information about CNC

### 4.1 Compensation of geometry

It is necessary to be able to measure any currently actual position of the CNC-controlled axis in order to perform tool moves on workpieces. The measured value is related to a machine fixed zero position and is compared to the target position which is predefined by the programs.

#### Required knowledge:

- coordinate systems of the machine and workpiece
- reference points of the machine, tool and workpiece
- type of distance measurement
- options of dimensioning and dimension compensation

#### Basics:

For chip removal relative moves between the tool and the workpiece are required. For programming all moves are related to the resting workpiece.

### 4.2 Coordinate systems on CNC-machine tools

#### Types of coordinate systems

Coordinate systems allow the exact description of all points on a working plane, respectively in an area.

Generally they are divided into

- Cartesian coordinate system and
- the polar coordinate system

#### 4.2.1 Cartesian coordinate system

A Cartesian coordinate system also called a rectangular coordinate system possesses two coordinate axes (two-dimensional Cartesian coordinate system) or also three coordinate axes (three-dimensional Cartesian coordinate system) which are perpendicular to one another in order to exactly describe the points.

On a two-dimensional Cartesian coordinate system, e.g. on the X, Y-coordinate system, each point is clearly defined on the plane by indicating the coordinates (X,Y).

The distance from the Y-axis is called the X-coordinate and the distance from the X-axis is called Y-coordinate. Those coordinates may possess positive or negative algebraic signs.

The three-dimensional Cartesian coordinate system is required to display and determine the position of special workpieces, e.g. milling parts.

In order to clearly describe a point in the space, three coordinates are required which are named according to the corresponding axes X-, Y- or Z-coordinates.

Such three-dimensional coordinate systems with positive and negative areas on the coordinate axis allow the exact description of any locations, e.g. in the working area of a milling machine, independent from where the zero point of the workpiece is set.



#### 4.2.2 Polar coordinate system

In the Cartesian coordinate system a point is described by e.g. its X- and Y-coordinate. For rotation-symmetric outlines, e.g. circular drilling images the required coordinates can only be calculated with considerable effort.

In the polar coordinate system a point is described by means of its distance (radius  $r$ ) to the coordinate origin and its angle ( $\alpha$ ) to the defined axis. The angle ( $\alpha$ ) is related to the X-axis of the X, Y coordinate system. In opposite direction it is negative.

#### 4.2.3 Machine coordinate system

The machine coordinate system of the CNC machine tool is determined by the manufacturer. It cannot be changed. The position of the origin point for the machine coordinate system, also called machine zero point cannot be changed.

Any tool moves are generally defined in a standardized, right-handed coordinate system.

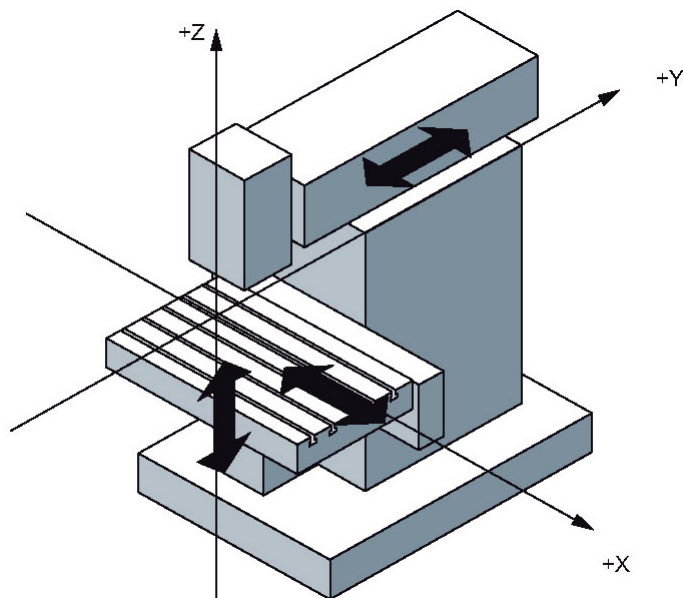
Turns from +X to +Y are created in +Z direction, which result in a right-handed screw.

##### Z-axis:

According to the standard it is equal to the working spindle or continues in positive direction starting from the workpiece. For milling it continues directed to the spindle perpendicular on the clamping surface (only for perpendicular milling machines) for several spindles one main spindle is determined.

##### X-axis:

Continues horizontally and parallel to the clamping surface  
for vertical Z-axis: +X to the right  
for horizontal Z-axis: +X to the left  
.



Img.4-1: Vertical Z-axis

##### Y-axis:

At a right angle to the Z- and X- axis in a way that a right-handed coordinate system is resulting

#### 4.2.4 Workpiece coordinate system

The workpiece coordinate system is determined by the programmer. It can be changed. The location of the origin point for this workpiece coordinate system, also called workpiece origin point is generally user-defined.



## 4.2.5 Rotary axes and secondary axes

NC machines with rotary table or swivel head

Rotary axis: A B C

Positive turn around X, Y, Z (right- hand-rule)

NC machines with several feed axes

Secondary axis: U V W

Parallel to the X-,Y-,Z-axis

## 4.3 NC mathematics

### 4.3.1 Basics of the coordinate calculation

For the CNC programming the corresponding points of the outline which is to be machined need to be entered. In most cases if the drawing is suitable for NC purposes, it is possible to directly transfer these coordinate points from the drawing. In some cases it may be necessary to calculate the coordinates.

In the frame of the automation those coordinates are calculated by an NC programming system at external working places and the data are directly transferred to the machine. Therefore, in most cases the NC programming is directly performed on the product (3D pattern) in the construction or in the process engineering department.

For the computer-aided programming the switch and path information are entered over the keyboard in the dialogue using the menu technique.

### 4.3.2 Parameters of a triangle

In order to calculate the missing coordinates the relations valid for a triangle are very useful. There are several options to describe a triangle. Some of the following parameters e.g. corners, angles or sides are being used.

### 4.3.3 Angle on a triangle

The angles on a triangle determine the type of triangle. Depending on the size of the individual angles we distinguish between acute-angle, obtuse angle or rectangular triangles.

On triangles the following relation is applied:

the sum of the angles a, b and g in a triangle always amounts to 180°.

$$a + b + g = 180^\circ$$

If two angles are known it is possible to determine the third unknown angle by means of this formula.

### Rectangular triangle

The rectangular triangle has a special meaning in the analytic geometry as the sides of such a triangle are having a definite mathematic relation to one another.

On a rectangular triangle the single sides are specially named.

- The longest side is located opposite to the right angle and is named hypotenuse.
- The two sides of the triangle which are forming the right angle are named cathetus.
- The side opposite the angle a is named opposite leg.
- The side adjacent to the angle a is named adjacent leg.

On a rectangular triangle the right angle is described by an quarter circle and a point in the angle.

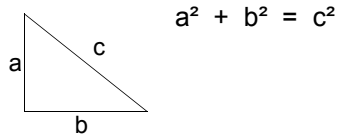
In a rectangular triangle it applies:

In a rectangular triangle you can calculate the missing leg if the other leg lengths are known. To do so, use the Pythagorean theorem.



The Greek Pythagoras (from about 580 to 496 before Jesus Christ) had been the first person to prove the following mathematic relation which had later on been named the Pythagorean theorem.

The sum of the cathetus square is equal to the hypotenuse square and expressed as a equation:



#### 4.4 Trigonometric functions

The trigonometric functions describe the relations between the angles and the sides of a rectangular triangle. With the help of these trigonometric functions it is possible to calculate unknown leg lengths with an unknown angle and a known leg. It is depending on which side and which angle are known in order to choose the appropriate trigonometric function e.g. the sinus function, the cosine function or the tangent function.

For the calculation of unknown legs the corresponding equation needs to be transformed as described in the following example:

Known are: the angle and the length of the adjacent leg

Looking for: the length of the opposite leg

It applies:  $\tan \alpha = \text{opposite leg} / \text{adjacent leg}$

The results is:

$\text{opposite leg} = \text{adjacent leg} \times \tan \alpha$

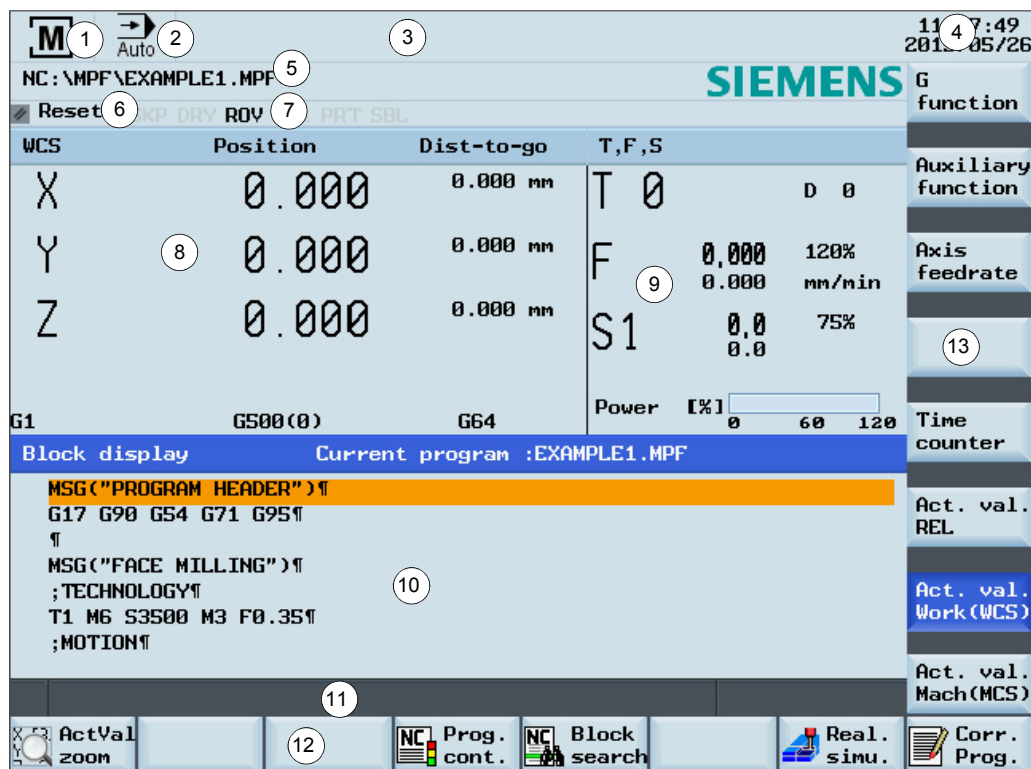




## 5 User interface, machine control panel

### 5.1 Screen arrangement

Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"



Img.5-1: Screen arrangement

#### Status area

- ① Active operating area
- ② Active operating mode
- ③ Alarm and message prompt area
- ④ Current time and date
- ⑤ Program file name
- ⑥ Program status indication
- ⑦ Active program control modes

#### Application area

- ⑧ Actual value window
- ⑨ T, F, S window
- ⑩ Operating window with program block Mode Display / Data Display

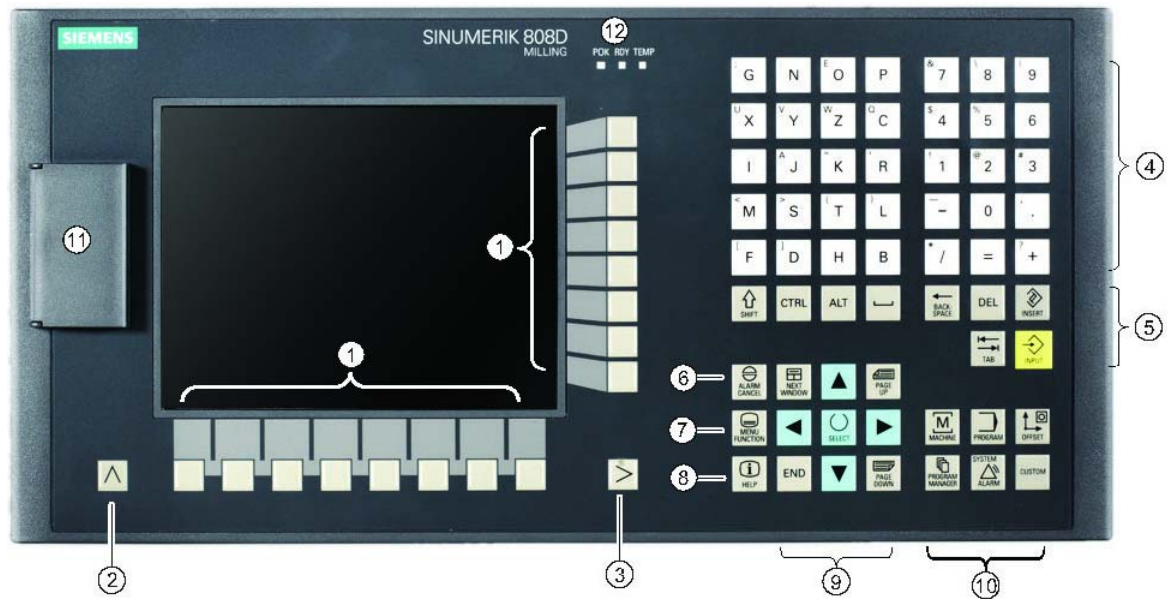
#### Tip and softkey area

- ⑪ Information line
- ⑫ Horizontal softkey bar
- ⑬ Vertical softkey bar












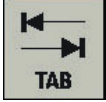


5.2 Elements on the PPU(Panel Processing Unit) front



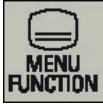

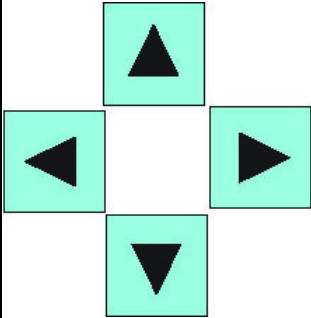





Img.5-2: Panel Processing Unit

Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
		Description
①		<b>Vertical and horizontal softkeys</b> Calls specific menu functions
②		<b>Return key</b> Returns to the next higher-level menu.
③		<b>Menu extension key</b> No function is assigned to this key. Reserved for future use.
④		<b>Alphabetic and numeric keys</b> You use these keys to enter characters or NC commands. Holding down <SHIFT> while pressing an alphabetic or numeric key allows you to enter the upper character shown on the key.










Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
		Description
⑤	Control keys	
		Shift key
		Control key
		Alternate key
		Space key
		<b>Backspace key</b> Deletes a character selected to the left of the cursor.
		<b>Delete key</b> Deletes the selected file or character.
		Insert key
		<b>Tab key</b> <ul style="list-style-type: none"> <li>• Indents the cursor by several characters.</li> <li>• Toggles between the input field and the selected program name.</li> </ul>
		Input key <ul style="list-style-type: none"> <li>• Confirms your entry of a value.</li> <li>• Opens a directory or program.</li> </ul>
⑥		<b>Alarm cancel key</b> Cancels alarms and messages that are marked with this symbol



Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
		Description
⑦		<b>Menu function key</b> Opens the wizard main screen.
⑧		<b>Help key</b> Calls the context-sensitive help for the selected window, alarm, message, machine data, setting data, or end-user wizard.
⑨	Cursor keys	
		<b>Cursor keys</b> up/down/left/right keys
		<b>Next window key</b> <b>No function is assigned to this key. Reserved for future use.</b>
		<b>End key</b> Moves the cursor to the end of a line.
		<b>Page up key</b> Scrolls upwards on a menu screen
		<b>Page down key</b> <b>Scrolls downwards on a menu screen</b>
		<b>Selection key</b> <ul style="list-style-type: none"> <li>• Toggles in selection lists and selection fields between several options.</li> <li>• Enters the "Set-up menu" dialog at NC start-up.</li> </ul>




Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
		Description
⑩	Operating area keys	
		Opens the "Machine" operating area
		Opens the "Program" operating area
		Opens the "Offset" operating area
		Opens the "Program" operating area
		Pressing this key opens the "Alarm" operating area. Holding down <SHIFT> while you press this key opens the "System" operating area.
		Enables user's extension application, for example, to generate user dialogs with the EasyXLanguage function. For more information about this function, refer to the SINUMERIK 808D Function Manual.
⑪	USB interface	Connects to a USB device Examples: <ul style="list-style-type: none"> <li>• Connects to an external USB memory sticker to transfer data between the USB sticker and the CNC.</li> <li>• Connects to an external USB keyboard for use as an external NC keyboard.</li> </ul>
⑫	<b>Status LEDs</b> 	<b>LED "POK"</b> Lights up green: The power supply for the CNC is switched on.
		<b>LED "RDY"</b> Lights up green: The CNC is ready for operation.
		<b>LED "TEMP"</b> Unlit: The CNC temperature is within the specified range. Lights up orange: The CNC temperature is out of range.



### 5.2.1 Key combination

Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"


Element	Description
<ALT> + <X>	Opens the "Machine" operating area
<ALT> + <V>	Opens the "Program" operating area
<ALT> + <C>	Opens the "Offset" operating area
<ALT> + <B>	Opens the "Program" operating area
<ALT> + <M>	<b>Opens the "Alarm" operating area</b>
<ALT> + <N> <SHIFT> + 	<b>Opens the "System" operating area</b>
<ALT> + <H>	<b>Calls the online help system.</b>
<ALT> + <L>	<b>Enables input of lowercase letters.</b>
<ALT> + <S>	Applicable only when the user interface language is Chinese. Calls the input method editor for entering Chinese characters.
<=>	Calls the pocket calculator. Note that this function is not applicable in MDA mode.
<CTRL> + <B>	Selects text in program blocks.
<CTRL> + <C>	Copies the selected text.
<CTRL> + <D>	Shows pre-defined slides on the screen.
<CTRL> + <P>	Captures screens
<CTRL> + <R>	Restarts the HMI
<CTRL> + <S>	Saves start-up archives




## 5.3 Controls on the machine control panel





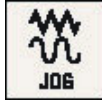
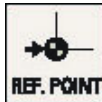





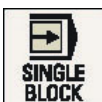
Img.5-3: Machine control panel

Controls on the machine control panel	
Element	Function
	<p><b>Drive voltage</b></p> <p>Push button with indicator light</p> <ul style="list-style-type: none"><li>Indicator light On, drive voltage activated</li><li>Indicator light Off, drive voltage deactivated</li></ul>

Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
①		<p>&lt;Emergency stop button&gt;</p> <p>Activate the button in situations where</p> <ul style="list-style-type: none"><li>life is at risk.</li><li>there is the danger of a machine or workpiece being damaged.</li></ul> <p>All drives will be stopped with the greatest possible braking torque.</p>









Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
②		<b>Handwheel key (with an LED status indicator)</b> Controls the axis movement with external handwheels.
③		<b>Tool number display</b> Displays the current tool number
④	Operating mode keys (all with LED status indicators)	
		Operating mode "JOG"
		Operating mode "REF. POINT" (reference point approach)
		Operating mode "AUTO" (automatic mode)
		Operating mode "MDA" Manual program input, automatic execution
⑤	Program control keys (all with LED status indicators)	
		<b>Program test key</b> Disables the output of setpoints to axes and the spindle. The control system only "simulates" the traverse movements in order to verify the correctness of the program.
		<b>Conditional stop key</b> Stops the program at every block in which miscellaneous function M01 is programmed.
		<b>Rapid override key</b> Adjusts axis feedrate override
		<b>Single block key</b> Activates single block execution mode
⑥	User-defined keys (all with LED status indicators)	









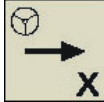
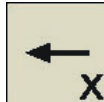
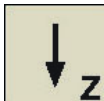

Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
		<b>Machine luminaire control key</b> Pressing this in any operating mode switches on/off the machine luminaire. LED on: The machine luminaire is switched on. LED off: The machine luminaire is switched off.
		<b>Coolant control key</b> Pressing this key in any operating mode switches on/off the coolant supply. LED on: The coolant supply is switched on. LED off: The coolant supply is switched off.
		<b>Safety door control key</b> When all axes and the spindle stop operation, pressing this key unlocks the safety door. LED on: The safety door is unlocked. LED on: The safety door is locked.
		<b>Clock wise magazine rotation</b> (active only in JOG mode) Pressing this key rotates the magazine clockwise. LED on: The magazine rotates clockwise. LED off: The magazine stops clockwise rotation.  <b>INFORMATION</b> The F3 does not have a tool changer. Therefore, the key has no function.
		<b>Reference point approach of the magazine</b> (active only in JOG mode) Pressing this key approaches the magazine to the reference point. LED on: The magazine is reference point approached. LED off: The magazine is not yet referenced.  <b>INFORMATION</b> The F3 does not have a tool changer. Therefore, the key has no function.
		<b>Counterclockwise magazine rotation</b> (active only in JOG mode) Pressing this key rotates the magazine counterclockwise. LED on: The magazine rotates counterclockwise. LED off: The magazine stops counterclockwise rotation.  <b>INFORMATION</b> The F3 does not have a tool changer. Therefore, the key has no function.





Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
		<b>Forward rotation of the chip conveyor</b> (active only in JOG mode) Pressing this key in any operating mode starts the forward rotation of the chip conveyor. LED on: The chip conveyor starts forward rotation. LED off: The chip conveyor stops rotation.  <b>INFORMATION</b> The F3 does not have a chip conveyor. Therefore, the key has no function. 
		<b>Reverse rotation of the chip conveyor</b> (active only in JOG mode) Keeping pressing this key in any operating mode rotates the chip conveyor in reverse order. Releasing the key changes the chip conveyor to the previous forward rotation or stop state. LED on: The chip conveyor starts reverse rotation. LED off: The chip conveyor stops reverse rotation.  <b>INFORMATION</b> The F3 does not have a chip conveyor. Therefore, the key has no function. 
		<b>User-defined keys</b>  <b>INFORMATION</b> The custom keys are not used. Therefore, the keys have no function. 
⑦	Axis traversing keys	
		<b>X axis key</b> Traverses the X axis in the positive direction.
		<b>X axis key</b> Traverses the X axis in the negative direction.
		<b>Z axis key</b> Traverses the Z axis in the negative direction.
		<b>Z axis key</b> Traverses the Z axis in the positive direction.







Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
		<b>Y axis key</b> Traverses the Y axis in the positive direction.
		<b>Y axis key</b> Traverses the Y axis in the negative direction.
		<b>Rapid traverse overlay key</b> Traverses the selected axis at rapid traverse speed while pressing the relevant axis key.
		Inactive key. No function is assigned to this key.
		Incremental feed keys (with LED status indicators) Sets increments desired for the axis to traverse.
⑧	Spindle control keys	
		Starts the spindle counterclockwise
		Stops the spindle
		Starts the spindle clockwise
		<b>Spindle speed override switch</b> Makes the spindle rotate at the specified speed override.
⑨	Program state keys	



Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
		<b>Cycle stop key</b> Stops the execution of NC programs
		<b>Cycle start key</b> Starts the execution of NC programs
		<b>Reset key</b> Resets NC programs Cancels alarms that meet the cancel criterion.
⑩		<b>Feedrate override switch</b> Traverses the selected axis at the specified feedrate override.

## 5.4 Protection levels

SINUMERIK 808D provides a concept of protection levels for enabling data areas. Different protection levels control different access rights.

The control system delivered from SIEMENS is set by default to the lowest protection level 7 (without password).

If the password is no longer known, the control system must be reinitialized with default machine data. All passwords are then reset to default passwords for this software release.

### ATTENTION!

**Before you boot the control system with default machine data, make sure that you have backed up your data; otherwise, all data is lost after rebooting with the default machinedata.**



Protection level	Locked by	Area
0	Siemens password	Siemens, reserved
1	Manufacturer password	Machine manufacturers
2	Reserved	
3 - 6	End-user password (Default password: "CUSTOMER")	End users
7	No password	End users



## Protection level 1

Protection level 1 requires a manufacturer password. With this password entry, you can perform the following operations:

- Entering or changing all machine data
- Conducting NC commissioning

## Protection level 3-6

Protection level 3-6 requires an end-user password. With this password entry, you can perform the following operations:

- Entering or changing part of the machine data
- Editing programs
- Setting offset values
- Measuring tools

## Protection level 7

Protection level 7 is set automatically if no password is set and no protection level interface signal is set. The protection level 7 can be set from the PLC user program after you set the bits in the user interface.

In the menus listed below the input and modification of data depends on the set protection level:

- Tool offsets
- Work offsets
- Setting data
- RS232 settings
- Program creation / program correction

## 5.4.1 Passwords

### INFORMATION



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



## 5.4.2 Change passwords

### Step 1

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

→ Press  Shift+  System Alarm



### Step 2



→ Enter customer's or manufacturer's password.

→ Change customer's or manufacturer's password.

→ Delete customer's or manufacturer's password.



## 6 Operation

### 6.1 Safety

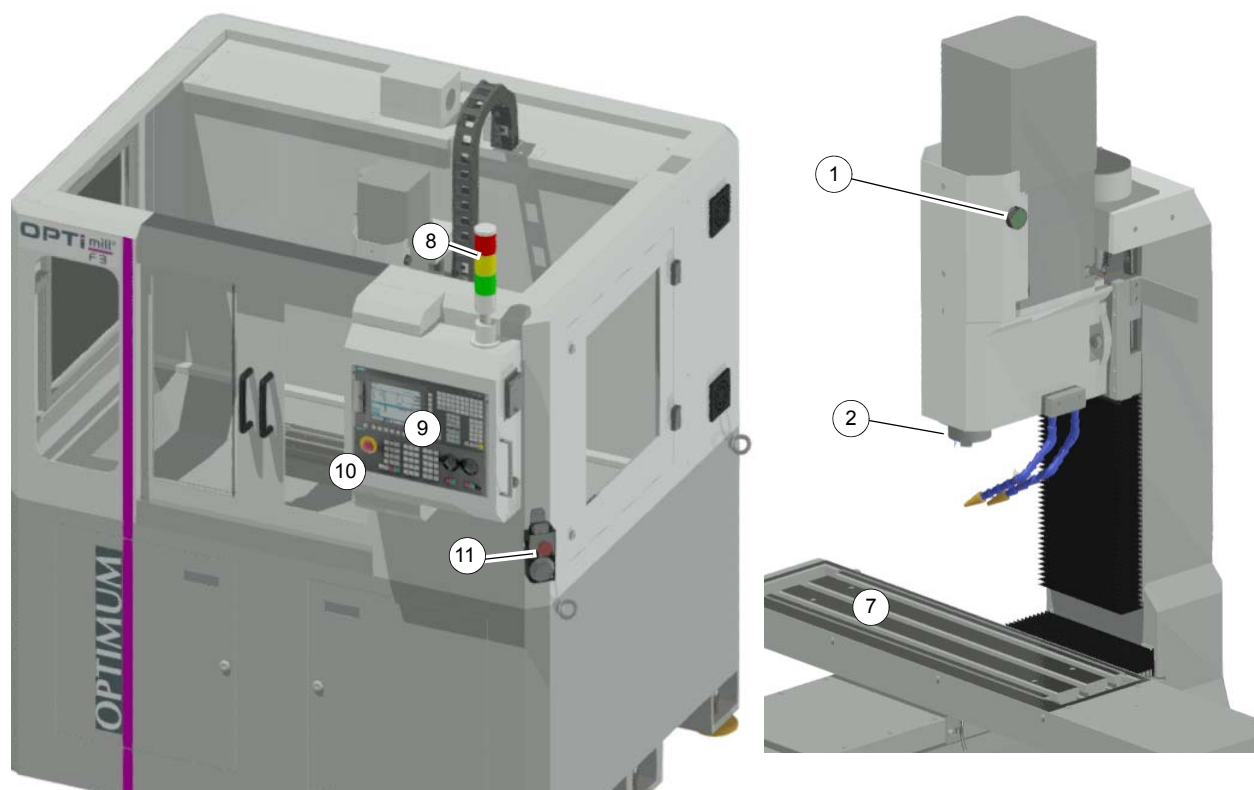
Commission the CNC machine only under the following conditions:

- The CNC machine is in proper working order.
- The CNC machine is used as intended.
- The operating manual is followed.
- All safety devices are installed and activated.

All failures should be eliminated immediately. Stop the CNC machine immediately in the event of any failure in operation and make sure that it cannot be started-up accidentally or without authorization. Notify the person responsible immediately of any modification.

☞ "Safety during operation" on page 15

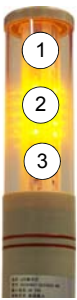
### 6.2 Control and indicating elements



Img.6-1: F3

No.	Description
1	Touch-switch to release and clamp the tool.
2	Milling spindle
7	Milling table
8	Signal lamp (if the indicator lamp lights up green --> CNC- program is running)
9	Machine control panel
10	Emergency-stop button
11	Handwheel for manual travelling with Emergency Stop push button and acknowledgement button

## Signal lamp

	Pos. No.	Colour	Meaning
	①	Red	Lights up when activity the EMERGENCY STOP push button
	②	Yellow	Lights up when a disturbance or in set-up operation such as opened safety housing
	③	Green	Lights up in the operating mode "automatic mode" resp. "program run"

## 6.3 Operational modes

### Manually controlled operation

The manually controlled operation is possible in the "JOG" and "MDA" operating modes. Refer to chapter "Manual mode" of the operating instructions of "SINUMERIK 808D".

In JOG mode, you can perform the following machining operations:

- Measuring tools
- Measuring the workpiece
- Setting parameters for face machining of a workpiece blank
- Setting the spindle speed and direction, activating other M function and changing the tool,
- Setting the axis positions in the relative coordinate system.

In MDA mode, you can create programs, load existing programs from directories in the "Program Manager" into the MDA buffer, or execute the current program.

### Automatic mode

Refer to chapter "Setup machine" of the operating instructions for "SINUMERIK 808D".

## 6.4 Programming

For further working steps please proceed as described in the operating instructions "Part programming, system, programming and cycles" for SINUMERIK 808D.

### Manual resp. part programming:

For this kind of programming, the programs must be created manually and entered in the control unit. The direct programming in the DIN-Code is a complex method which requires lots of skills. Nowadays this task is mostly taken over by CAD/CAM systems which directly create an operating program using a graphical user interface.


### Automatic programming:

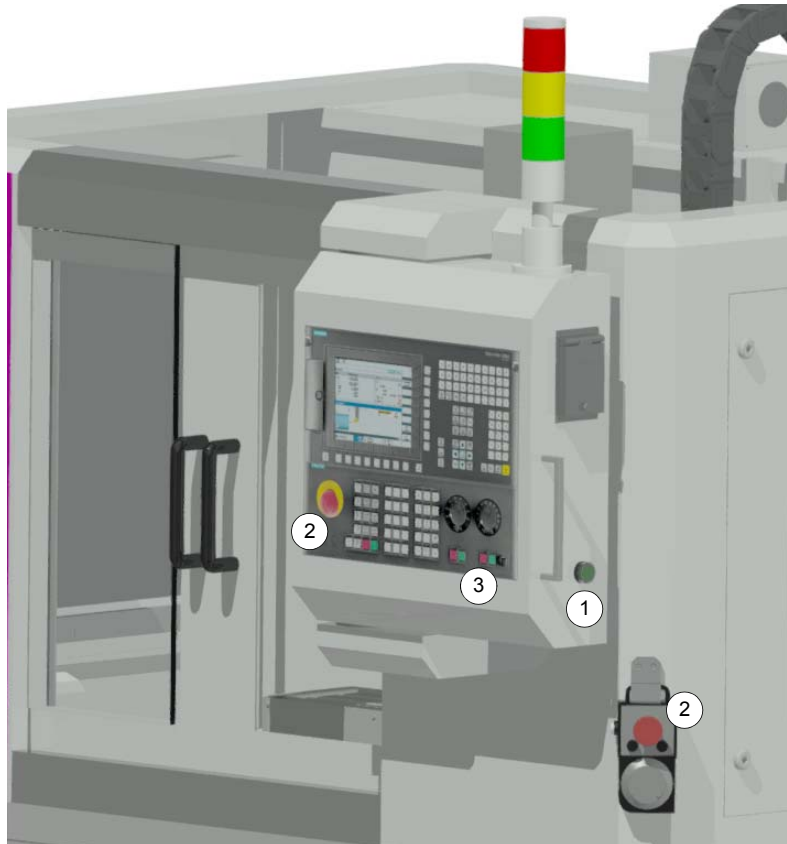
Construction data are transmitted (semi-) automatically to an executable program by means of the CAD/CAM program (for instance a 3D-CAD program including downstream co-processor). For this kind of programming, a 3D model is designed using a PC. The motion-sequence of the machine is calculated by means of an operating sequence which is predefined by the user. These programs access the tool data base which includes all tool parameters (speed, feed, diameter, etc.). Due to this systematic program structure, the user is able to create a complete program in a short period of time without having any knowledge of the individual program commands and their syntax.



## 6.5 Operation of the machine

### 6.5.1 Turning on the CNC- machine

- Switch on the main switch.  "Lockable main switch" on page 12
- Wait until the control is completely started.
- Press the push button "Drive control ON" (1).
- Unlock the "Emergency Stop push buttons" (2).
  - on the MCP,
  - on the electronic handwheel.
- Close - if not yet closed - the sliding door.
- Press the pushbutton "Reset" (3).



Img.6-2: Operating area

## 6.5.2 Reference point approach after turning on

### INFORMATION

If your machine is configured with ABS encoder (808D ADVANCED), you do not need to reference the axis of the machine.

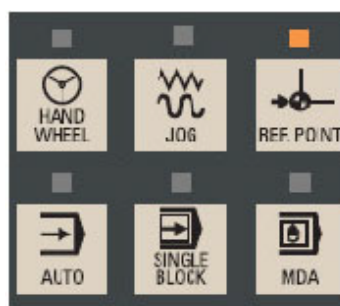
If your machine is fitted with INC encoder (808D), after power on, the machine must first be referenced!

Without existing reference points (machine zero points) you cannot start and run programs in the control.

With the beginning of the reference point approach the axes should be located in a central position as possible.

The following information indicated serve as preliminary information. Further information can be found in the Siemens manual.

After switching on the F3 is in the mode reference point approach area, the LED on the <REF POINT> button lights.

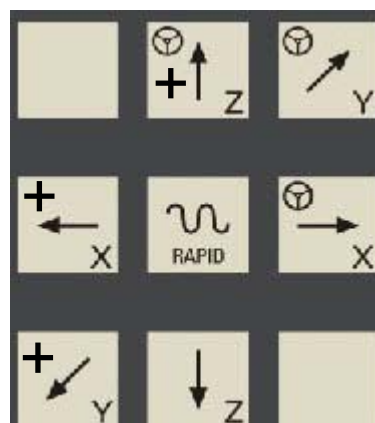


As long as the axes are not referenced, the symbol (circle) between the axis and the corresponding value is displayed.

<div> <div>M</div> <div>Ref Point</div> </div>		
NC: \MPF\EXAMPLE1.MPF		
<div> <div>Reset</div> <div>SKP DRY ROV M01 PRT SBL</div> </div>		
MCS	Reference point	
MX1	0.000	mm
MY1	0.000	mm
MZ1	0.000	mm

→ The axes are referenced with the "axis traversing keys."

Make sure that the "feed override switch" is not set to "zero".





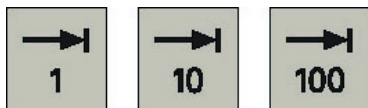


- Traverse each axis to the machine zero point until the referenced symbol is shown on the respective axis.

Once the axis approaches the reference point, the referenced symbol must be shown next to the axis.

M Ref Point		
NC: \MPP\EXAMPLE1.MPF		
Reset SKP DRY ROV M01 PRT SBL		
MCS	Reference point	
MX1	0.000	mm
MY1	0.000	mm
MZ1	0.000	mm

After returning to the "JOG" mode, the axes can be moved manually. Use the Increment button to specify the increment, or press the <JOG> button again to stop the increment again.



- If possible, read the upcoming messages on the display, remove the error messages, such as e.g. insufficient compressed air, ...
- Default values for example should apply for the feed rate in the "JOG" mode, be entered on the side of the standard values. Press the button <OFFSET> to get to the settings.

M JOG		
NC: \MPP\EXAMPLE1.MPF		
Reset SKP DRY ROV M01 PRT SBL		
MCS	Position	Repos offset
MX1	0.000	0.000 mm
MY1	0.000	0.000 mm
MZ1	0.000	0.000 mm

- For the following working steps, please proceed as described in „Operation and programming“ of the Siemens SINUMERIK 808D Operation instructions.

Setting data		11:28:45 2015/05/26
JOG data		Time counter
JOG feedrate:	0.000 mm/min	
Spindle speed:	0.000 rpm	
Spindle data		
Minimum:	0.000 rpm	
Maximum:	10000.000 rpm	
Limitation with G96:	100.000 rpm	
DRY		Misc.
Dry run feedrate:	5000.000 mm/min	
Start angle		
Start angle for thread:	0.000 °	
Tool list	Tool wear	Work offset R var. SD Sett. data GUD User data





## 6.5.3 Using the electronic handwheel

The electronic handwheel can always be used when

- the CNC- machine is referenced,
- the sliding door is opened,
- the LED on the button <HANDWHEEL> lights up.

Press the button <HANDWHEEL> on the machine control panel to use the electronic handwheel.



	Pos. No.	Meaning	Description
	①	Rotary switch	Selector switch for individual axes control.
	②	Rotary switch	Selector switch for the feed speed (3 stages).
	③	Hand wheel	Hand wheel for individual axes travel.
	④	<EMERGENCY STOP button>	The EMERGENCY STOP push button switches off the CNC- machine.
	⑤	Push button/ acknowledgement button	In the setting mode resp. if the sliding door is open it is necessary to actuate the acknowledgement button in order to expressly allow the movement of the individual axes.

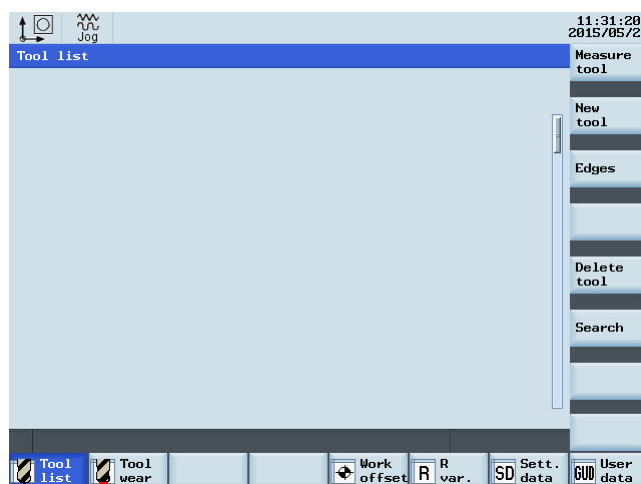
## 6.5.4 Insert tool

### INFORMATION

Before you can run a CNC program, at least one tool must have been created and measured in the tool memory.

→ Change over to the mode <JOG>.

- Press the "Offset" key on the machine control panel.
- Press the "Tool list" softkey
- For the following working steps, please proceed as described in „Operation and programming“ of the Siemens SINUMERIK 808D Operation instructions.



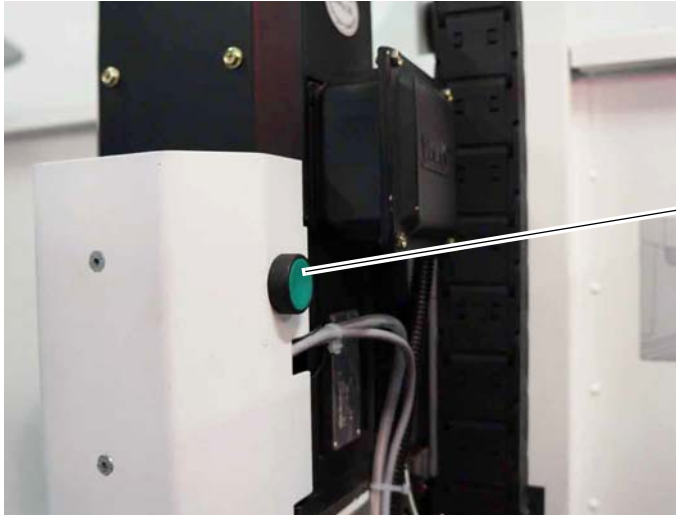


## 6.5.5 Manual tool change

### ATTENTION!

**Hold your tool if there is a tool inserted in the spindle.**

- Press the push button "Open/Close door" and open the sliding door.
- Press the pushbutton "manual tool change" to loosen or to clamp the tool.



Img.6-3: Spindle head

Pushbutton manual  
changing the tool



## 6.5.6 Clamping the workpiece

### ATTENTION!

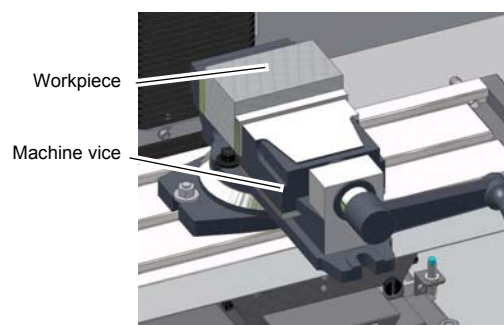
**The workpiece is always to be fixed by a machine vice, jaw chuck or by another appropriate clamping tool such as for the clamping claws.**

**When setting and using already created programs observe the safety clearance in order to avoid collisions with the selected clamping means.**

### WARNING!

**Risk of injury caused by workpieces flying off.**

- Clamp the workpiece in the machine vice.
- Make sure that the workpiece is firmly clamped in the vise.



Img.6-4: Clamping the workpiece





## 6.5.7 Turning off the CNC- machine

→ Push the EMERGENCY stop push button.

For a long-term standstill of the CNC machine switch it off at the main switch.

🔧 "Disconnecting and securing the CNC machine" on page 17

## 6.6 Operational modes

### Manually controlled operation

The manually controlled operation is possible in the "JOG" and "MDA" operating modes. Refer to chapter 5 "Manual mode" of the operating instructions of "SINUMERIK 808D".

### Automatic mode

Refer to chapter 6 "Setup machine" of the operating instructions for "SINUMERIK 808D".

## 6.7 Programming

For further working steps please proceed as described in the operating instructions "Part programming, system, programming and cycles" for SINUMERIK 808D.

### Manual resp. part programming:

For this kind of programming, the programs must be created manually and entered in the control unit. The direct programming in the DIN-Code is a complex method which requires lots of skills. Nowadays this task is mostly taken over by CAD/CAM systems which directly create an operating program using a graphical user interface.

### Automatic programming:

Construction data are transmitted (semi-) automatically to an executable program by means of the CAD/CAM program (for instance a 3D-CAD program including downstream co-processor). For this kind of programming, a 3D model is designed using a PC. The motion-sequence of the machine is calculated by means of an operating sequence which is predefined by the user. These programs access the tool data base which includes all tool parameters (speed, feed, diameter, etc.). Due to this systematic program structure, the user is able to create a complete program in a short period of time without having any knowledge of the individual program commands and their syntax.

### DIN Code and ISO Code:

Use the procedure for switching or activating the programming language in the manual operation and programming of the "SINUMERIK 808D".

## 6.8 Starting the program

Adjusting the dosing for the coolant supply on the spindle head before starting the program. Any change in the dosing must only be performed during the setup operation. The requirement of coolant supply is switched on over your CNC- programs.

### WARNING!

**Never change the dosing of the cooling lubricant supply and never seize into the machine when a program is running.**



### CAUTION!

**Before starting the program, the sliding door of the separating protective equipment must be closed.**

- Completely close the separating protective equipment.
- Change to "AUTO/MDA" mode.





For the following working steps, please proceed as described in „Operation and programming“ of the Siemens "SINUMERIK 808D" Operation instructions.

## 6.9 Central lubrication system

The CNC machine is equipped with a central lubrication system.

The lubricating system is used to maintain an oil film on the slideways, the bearings, the ledges and the ball screws and to reduce their wear.

In case of a failure or a fault in the central lubricating system, a stick-slip effect may occur. This effect describes the jerky sliding of solid bodies moving opposite one another. For instance: creaking doors and rattling windscreen wipers.

## 6.10 Data interfaces and current collection

When connecting data interfaces make sure that the data cable runs to the control unit interface via the shortest possible distance. The cable routing can be conducted along the measuring system lines. However the cable in the switch cabinet must never be routed near the drive units of the NC axis or the frequency converter itself. Electromagnetic radiation causes errors during data transfer (EMC problems).

The control is equipped with the following data interfaces.

- X130 Ethernet port (RJ45 connector, only 808D ADVANCED)
- USB connection
- Connection for power supply

## 6.11 Selecting the speed

The correct speed is an important factor for milling. The speed determines the cutting speed by which the cutting edges cut the material. By selecting the correct cutting speed, the service life of the tool is increased and the working result is optimized.

The optimum cutting speed mainly depends on the material and on the material of the tool. With tools (milling cutters) made of hard metal or ceramic insert it is possible to work with higher speeds than with tools made of high-alloy high speed steel (HSS). You will achieve the correct cutting speed by selecting the correct speed.

In order to determine the correct cutting speed for your tool and for the material to be cut you may refer to the following standard values or a table reference book (e.g. Tabellenbuch Metall, Europa Lehrmittel, ISBN 3808517220).

The required speed is calculated as follows:

$$n = \frac{V}{\pi \times d}$$

n = speed in min<sup>-1</sup> (revolutions per minute)

V = cutting speed in m/min (meter per minute)

d = tool diameter in m (Meter)

### 6.11.1 Standard values for cutting speeds

[ m/min ] with high-speed steel and hard metal in upcut milling.

Tool	Steel	Grey cast iron	Al alloy age-hardened
Plain mill and side milling cutters [ m/min ]	10 - 25	10 - 22	150 - 350
Relieved form cutters [ m/min ]	15 - 24	10 - 20	150 - 250



Inserted -tooth cutter with SS [ m/min ]	15 - 30	12 - 25	200 - 300
Inserted-tooth cutter with HM [ m/min ]	100 - 200	30 - 100	300 - 400

The results are the following standard values for speeds in dependence of the milling cutter diameter, cutter type and material.

Tool diameter [ mm ] shell end mill and plain milling cutter	Steel 10 - 25 m/min	Grey cast iron 10 - 22 m/min	Al alloy age-hardened 150 - 350 m/min
	Speed [ min <sup>-1</sup> ]		
35	91 - 227	91 - 200	1365 - 3185
40	80 - 199	80 - 175	1195 - 2790
45	71 - 177	71 - 156	1062 - 2470
50	64 - 159	64 - 140	955 - 2230

Tool diameter [ mm ] Form cutters	Steel 15 - 24 m/min	Grey cast iron 10 - 20 m/min	Al alloy cured 150 - 250 m/min
	Speed [ min <sup>-1</sup> ]		
4	1194 - 1911	796 - 1592	11900 - 19000
5	955 - 1529	637 - 1274	9550 - 15900
6	796 - 1274	531 - 1062	7900 - 13200
8	597 - 955	398 - 796	5900 - 9900
10	478 - 764	318 - 637	4700 - 7900
12	398 - 637	265 - 531	3900 - 6600
14	341 - 546	227 - 455	3400 - 5600
16	299 - 478	199 - 398	2900 - 4900

## 6.11.2 Standard values for speeds with HSS – Eco – twist drilling

Material	Drill diameter										Cooling <sup>3)</sup>
		2	3	4	5	6	7	8	9	10	
Steel, unalloyed, up to 600 N/mm <sup>2</sup>	n <sup>1)</sup>	5600	3550	2800	2240	2000	1600	1400	1250	1120	E
	f <sup>2)</sup>	0.04	0.063	0.08	0.10	0.125	0.125	0.16	0.16	0.20	
Structural steel, alloyed, quenched and subsequently drawn, up to 900N/mm <sup>2</sup>	n <sup>1)</sup>	3150	2000	1600	1250	1000	900	800	710	630	E/oil
	f <sup>2)</sup>	0.032	0.05	0.063	0.08	0.10	0.10	0.125	0.125	0.16	
Structural steel, alloyed, quenched and subsequently drawn, up to 1200 N/mm <sup>2</sup>	n <sup>1)</sup>	2500	1600	1250	1000	800	710	630	560	500	Oil
	f <sup>2)</sup>	0.032	0.04	0.05	0.063	0.08	0.10	0.10	0.125	0.125	
Stainless steels up to 900 N/ mm <sup>2</sup> e.g. X5CrNi1810	n <sup>1)</sup>	2000	1250	1000	800	630	500	500	400	400	Oil
	f <sup>2)</sup>	0.032	0.05	0.063	0.08	0.10	0.10	0.125	0.125	0.16	
1): Speed [ n ] in r/min											
2): Feed [ f ] in mm/r											
3): Cooling: E = Emulsion; oil = cutting oil											



- The above mentioned indications are standard values. In some cases it may be advantageous to increase or decrease these values.
- When drilling a cooling or lubricating agent should be used.
- For stainless materials (e.g. VA – or NIRO steel sheets) do not center as the material would compact and the drill bit will become rapidly blunt.
- The workpieces need to be tensed in flexibly and stably (vice, screw clamp).

## INFORMATION

Friction during the cutting process causes high temperatures at the cutting edge of the tool. The tool should be cooled during the milling process. By cooling with an appropriate coolant lubricant you will achieve a better working result and longer durability of the tool.



## INFORMATION

Use a water-soluble and non-pollutant emulsion as a cooling agent. This can be acquired from authorised distributors.

Make sure that the cooling agent is properly retrieved. Respect the environment when disposing of any lubricants and coolants. Follow the manufacturer's disposal instructions.



## INFORMATION

The CNC milling machine is lacquered with a one-component paint. Observe this fact when selecting your cooling lubricant.





## Brief instruction 808D Milling



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

The information provided in this brief instruction contains merely general descriptions or characteristics of performance which in actual case of use do not always apply as described or which may change as a result of further development of the products.

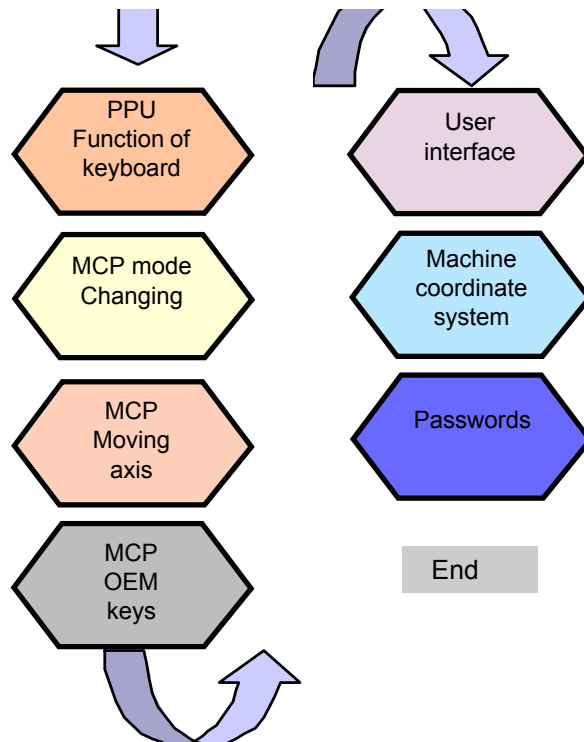
This brief instruction is not the replacement for Siemens 808D manuals and only serves as a reference for quickly find already-known operations and functions.

## Preparation

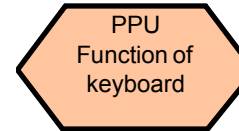
### Description

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.

### Content

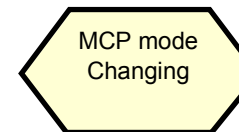


## Basic Theory



Menu navigation

Operating area  
navigation



Mode navigation

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



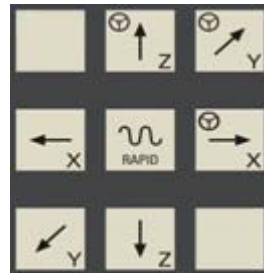
The 808D panel processing unit (PPU) is used to input data to the CNC and to navigate to operating areas of the system.



MCP  
Moving  
axis



Axis movement



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

The machine can be moved with the appropriate keys.

MCP  
OEM  
keys



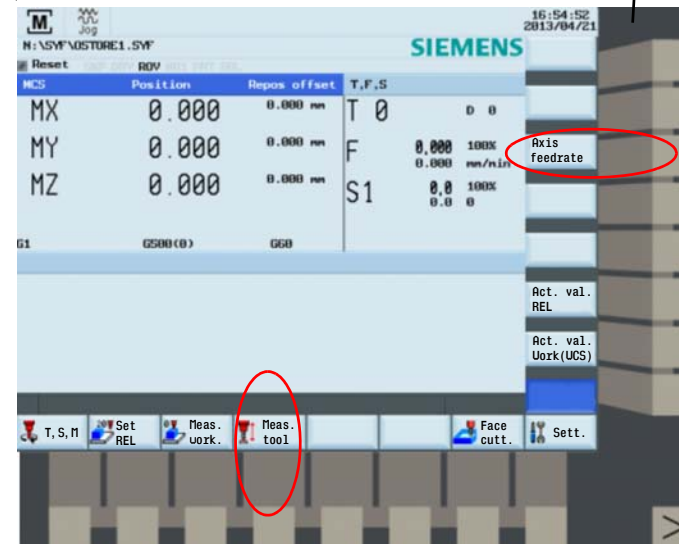
OEM keys



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

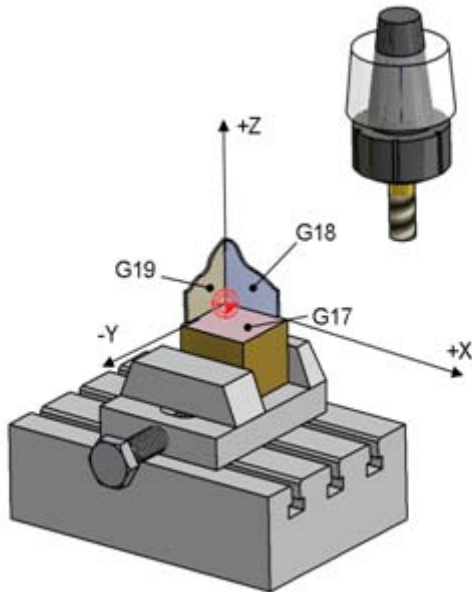
The machine functions can be activated with the appropriate keys.

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).

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.

## SEQUENCE

## 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

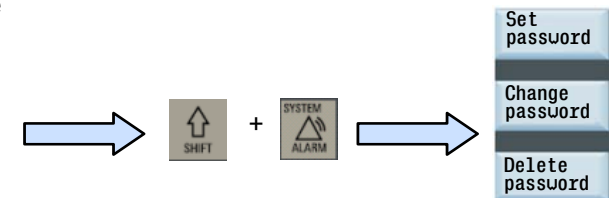
Changing  
password

## 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.

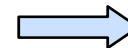


## Step 2

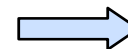
Set  
password

Change  
password

Delete  
password



Enter customer's or manufacturer's password



Change customer's or manufacturer's password



Delete customer's or manufacturer's password

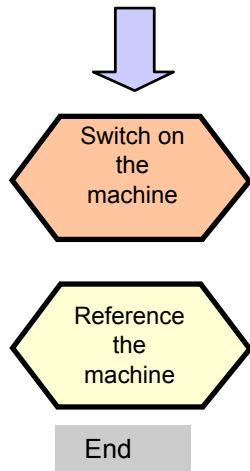
## End

## Switch On and Referencing

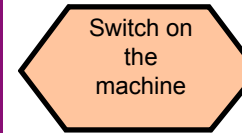
### Description

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

### Content



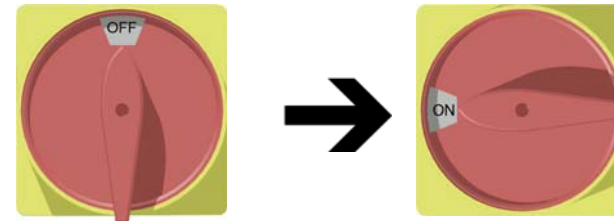
### SEQUENCE



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

#### Step 1

Turn on the main switch of the machine.



#### Step 2

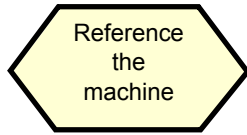
Make sure you perform the following operation!



Release all the EMERGENCY STOP buttons on the machine!

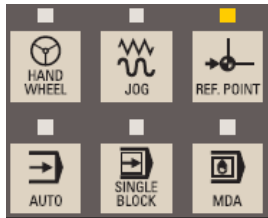
End

## SEQUENCE



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

## Step 1

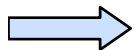
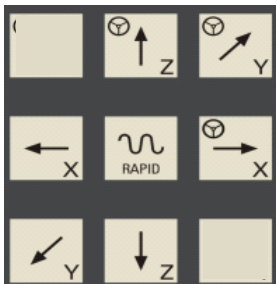


After power on, the machine will be in the reference point approach mode (default).

Ref Point		
N:\SYF\STORE1.SVF		
Reset	SKP DRY ROV M01 PRT SBL	
MCS	Reference point	
MX	0.000	mm
MY	0.000	mm
MZ	0.000	mm

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.

Ref Point		
N:\SYF\STORE1.SVF		
Reset	SKP DRY ROV M01 PRT SBL	
MCS	Reference point	
MX	0.000	mm
MY	0.000	mm
MZ	0.000	mm

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.

Jog		
N:\SYF\STORE1.SVF		
Reset	SKP DRY ROV M01 PRT SBL	
MCS	Position	Repos offset
MX	0.000	0.000 mm
MY	0.000	0.000 mm
MZ	0.000	0.000 mm

The machine can now be operated in JOG mode.

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

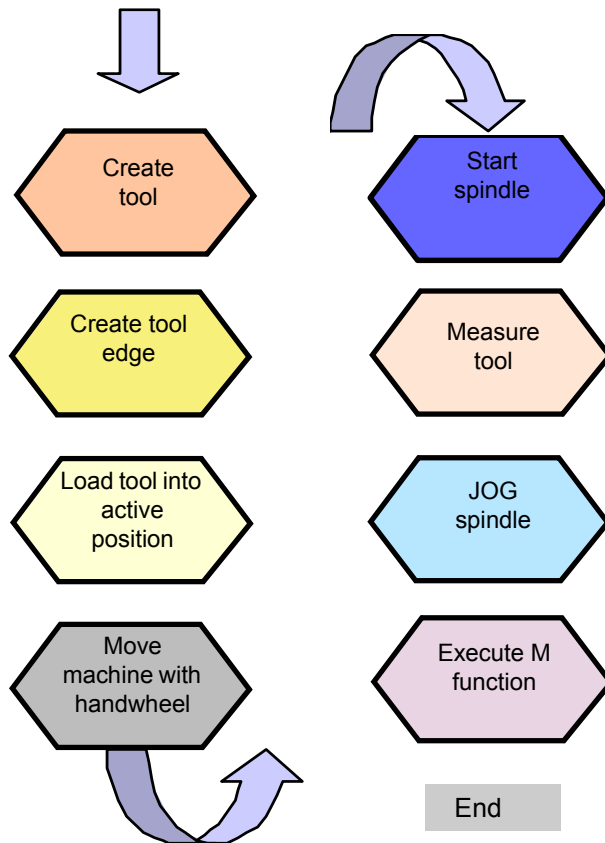
## End

## Tool Setup

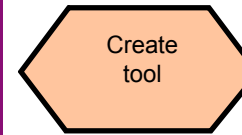
### Description

This unit describes how to create and set up tools.

### Content



## SEQUENCE



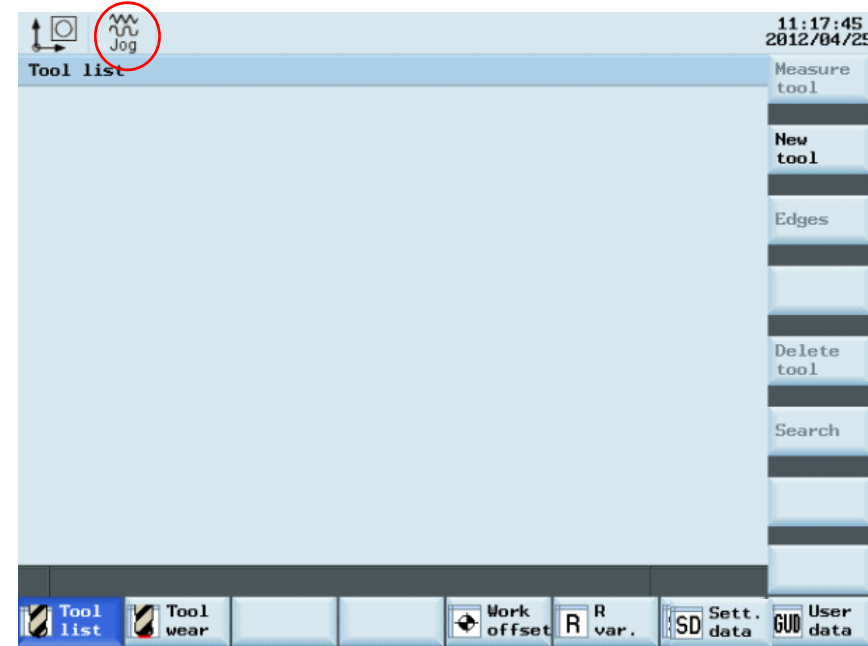
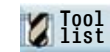
A tool must have been created and measured before executing the program.

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

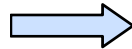
## Step 2



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.



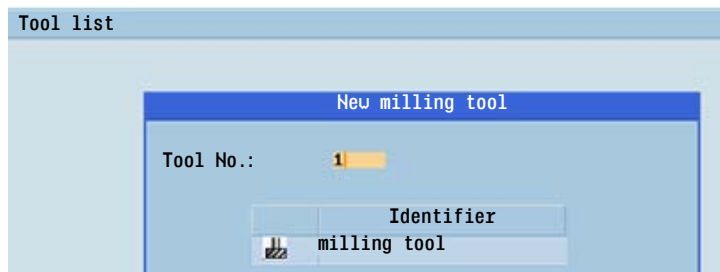
New tool

Select the type of tool required.

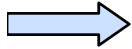


Milling tool

Enter "1" at "Tool No."

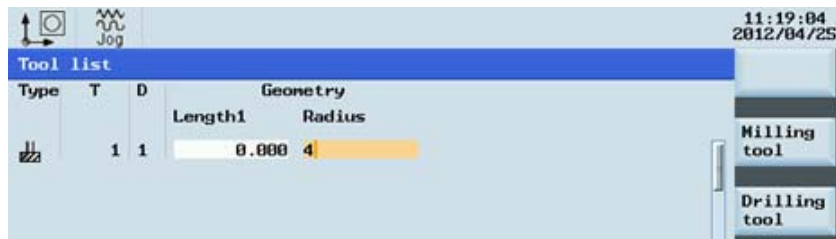


Press the "OK" SK on the PPU.



OK

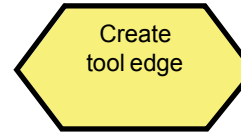
Enter the "Radius" of the milling tool.



Press the "Input" button on the PPU.



INPUT



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.



OFFSET

Press the "Tool list" SK on the PPU.

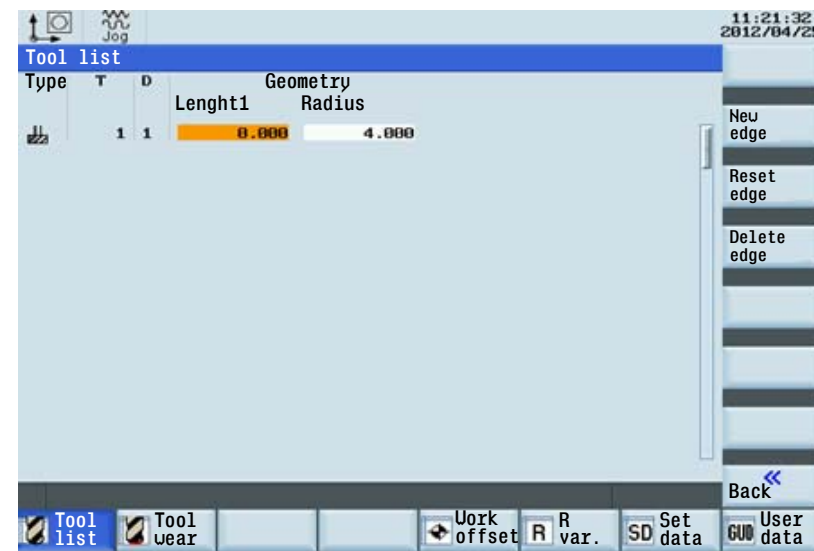


Tool list

Use direction keys to select the tool which needs to add a tool edge.



▼ Or ▲

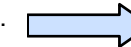


Press the "Edges" SK on the PPU.



Edges

Press the "New edge" SK on the PPU.



New edge

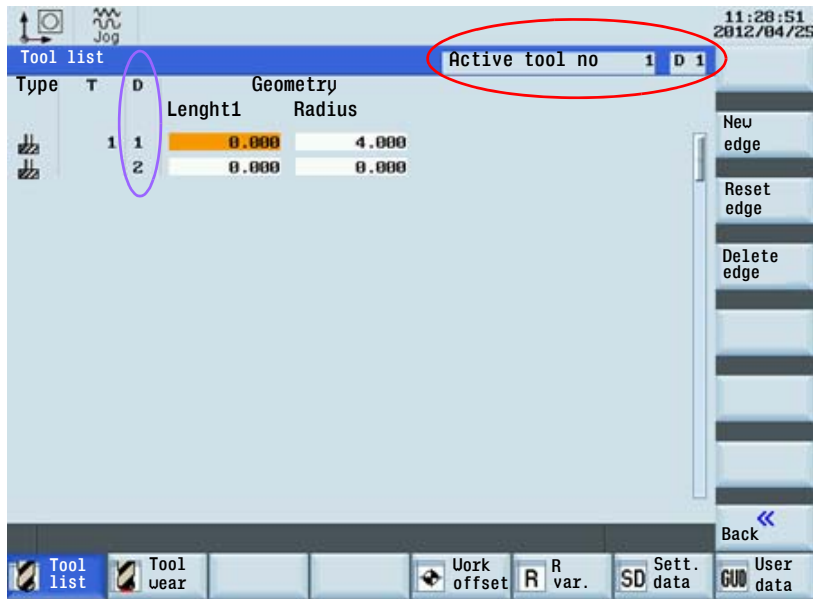


## SEQUENCE

## Step 2

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

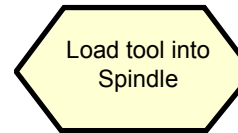
The red circle shows the actual active tool and tool edge, the purple circle shows how many tool edges have been created and the related data for each tool edge.



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



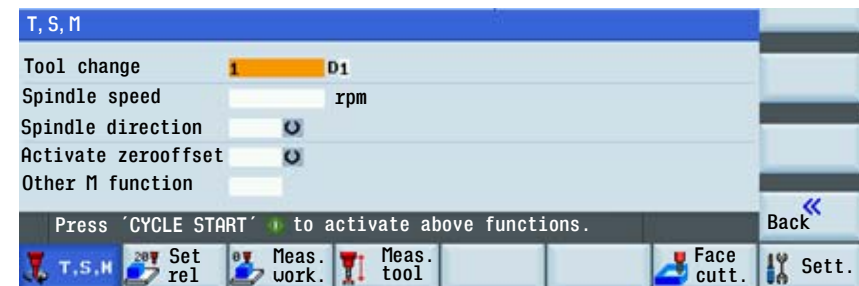
Press the "JOG" key on the MCP



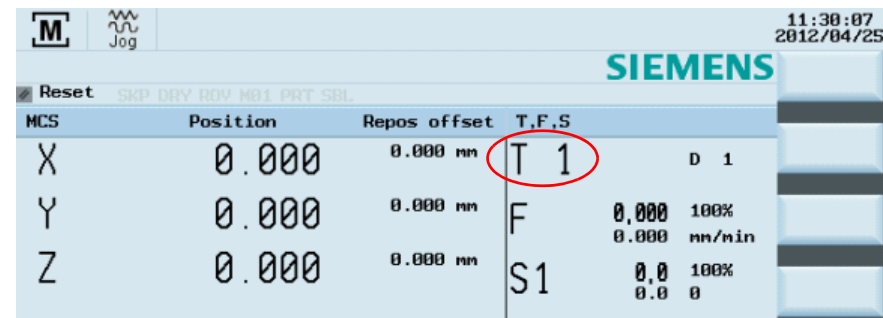
Press the "T.S.M" SK on the PPU



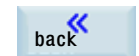
Enter tool number "1" in "T"



Press "CYCLE START" on the MCP



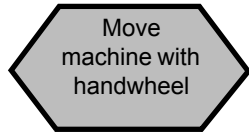
Press the "Back" SK on the PPU



## SEQUENCE

The tool are usually loaded manually into the spindle.

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



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.

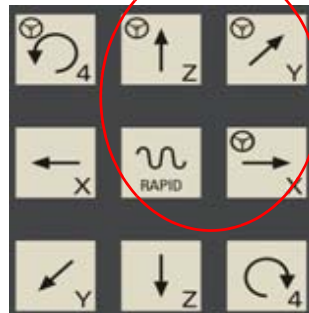
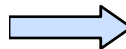
Press the “Machine” key on the PPU



Press the “Handwheel” key on the MCP



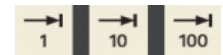
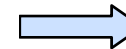
Select the axis you want to move with the appropriate keys. on the MCP



WCS	Position	Repos. offset
X	0.000	0.000 mm
Y	0.000	0.000 mm
Z	0.000	0.000 mm

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.

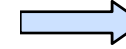
Select the required override increment according to the buttons on the right (this selection fits all axes)



The handwheel increment is “0.001 mm”



The handwheel increment is “0.010 mm”



The handwheel increment is “0.100 mm”

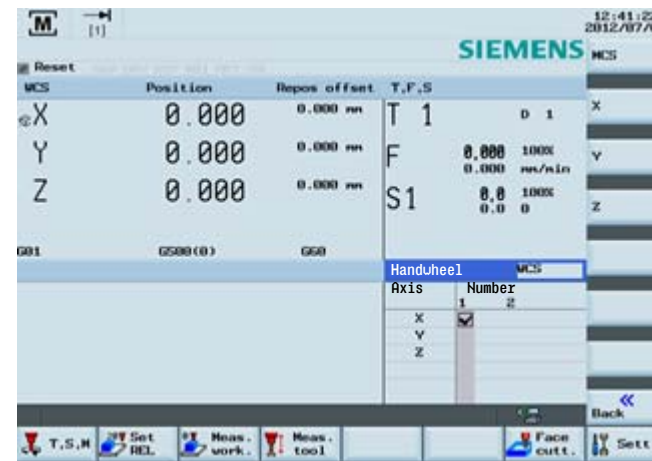


The selected axis can now be moved with the handwheel.

Press “JOG” on 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.

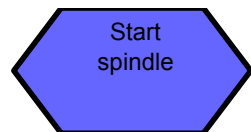


Handwheel

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



## SEQUENCE



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

Start the spindle before adjusting tools as follows:

Press the "Machine" key on the PPU



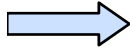
Press the "JOG" key on the MCP



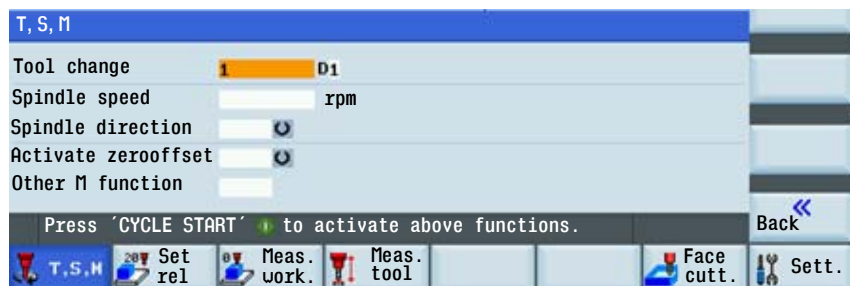
Press the "T.S.M" SK on the PPU



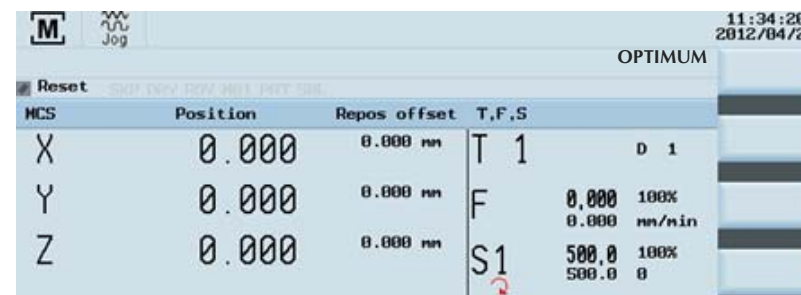
Enter "500" at "Spindle speed"



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



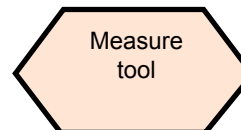
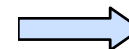
Press the "CYCLE START" key on the MCP



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



Press the "Back" SK on the PPU



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

### Step 1 Measure length

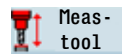
Press the "Machine" key on the PPU



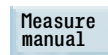
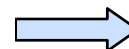
Press the "JOG" key on the MCP



Press the "Meas. tool" SK on the PPU

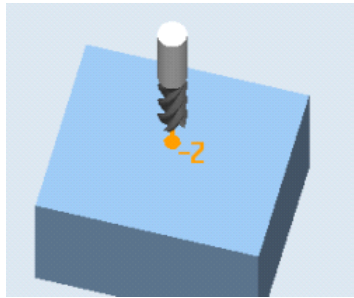


Press the "Measure manual" SK on the PPU



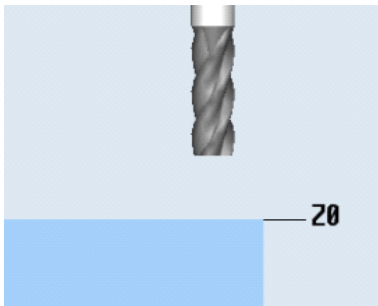
## SEQUENCE

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

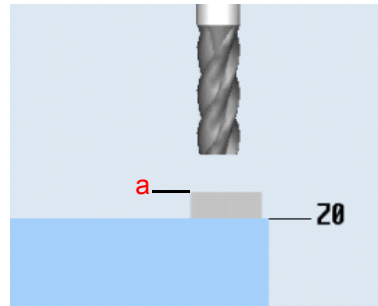


**Note:** The following text describes the required settings in the workpiece coordinate system  
“X / Y / Z” zero points as: “X0” / “Y0” / “Z0”

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



or



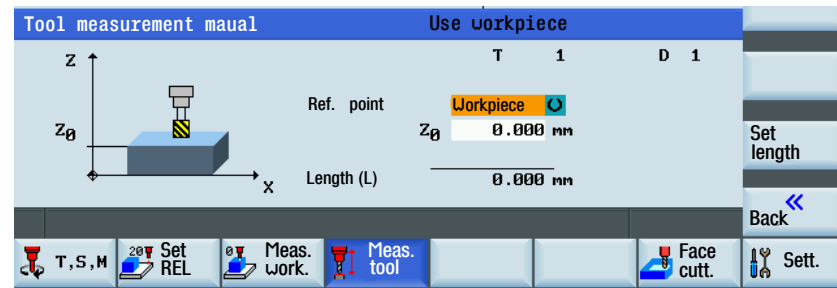
Move directly to zero point

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.

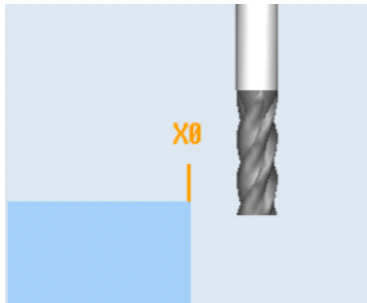
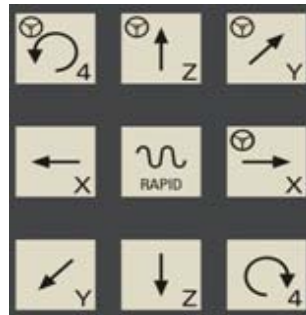
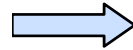
## Step 2 Measure diameter

Press the “Diameter” SK on the PPU

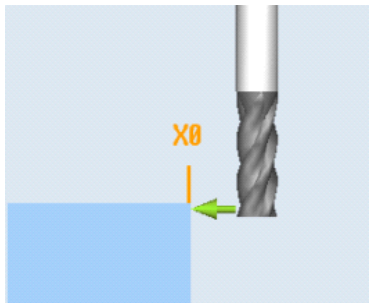
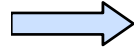


## SEQUENCE

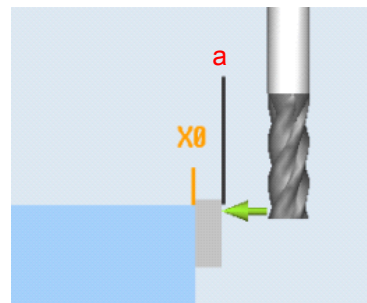
Press the axis keys on the MCP to move the tool to the set position.



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



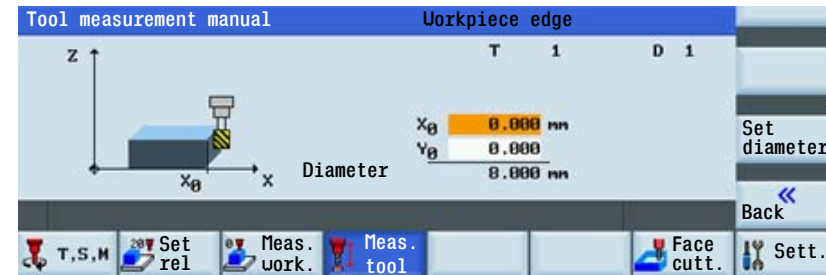
or



Move directly to zero point

Use a setting block.

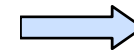
Enter "0" at "X0"  
Enter "0" at "Y0"  
(This is the value of the width of a setting block if it is used. Select one of X0/Y0 according to requirement.)



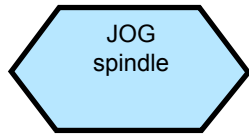
Press the "Set diameter" SK on the PPU



Press the "Back" SK on the PPU



## SEQUENCE



A tool must be loaded to the spindle.

Press the "Machine" key on the PPU



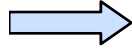
Press the "JOG" key on the MCP



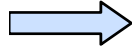
Press the spindle direction key on the MCP to start/stop the spindle.



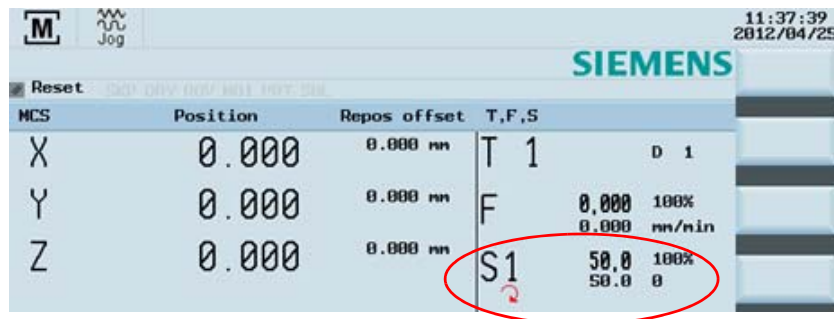
Press "Spindle left" on the MCP to start the spindle in the counter-clockwise direction.



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

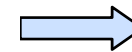


Press "Spindle right" on the MCP to start the spindle in the clockwise direction.

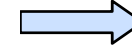


Please make sure all the machine axes are in safe positions before executing the M function!

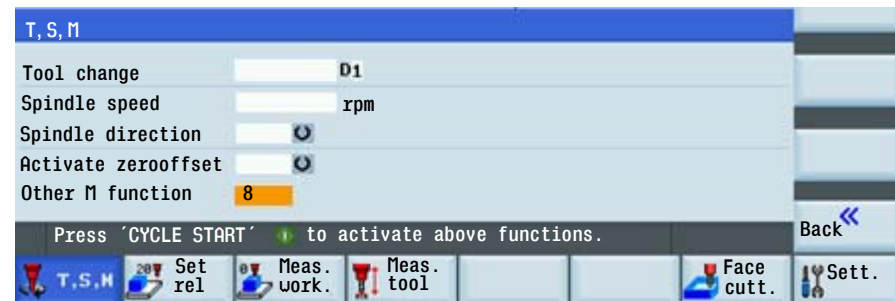
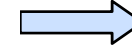
Press the "Machine" key on the PPU



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



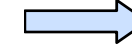
Use the direction key to move the highlighted cursor to "Other M function" and enter "8". This will start the coolant.



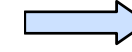
Press "CYCLE START" on the MCP.



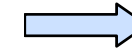
The coolant function button on MCP is active.



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



Press the "Back" SK on the PPU.

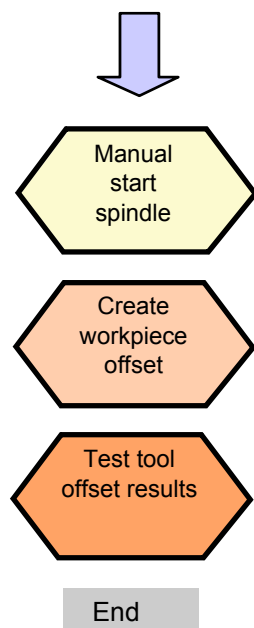


## Workpiece Setup

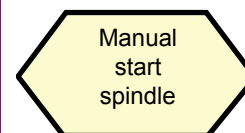
### Description

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

### Content



## SEQUENCE



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



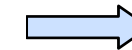
Press the “JOG” key on the MCP



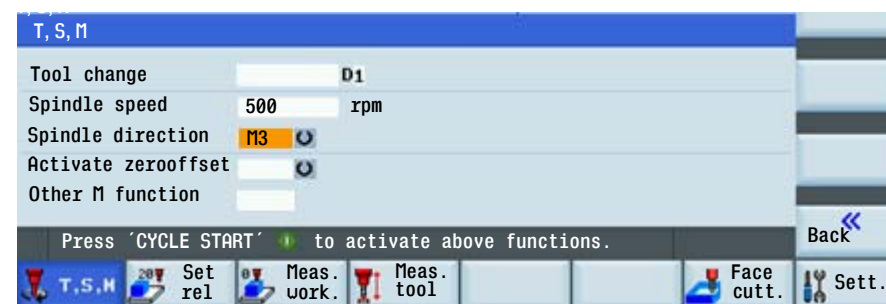
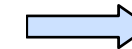
Press the “T.S.M” SK on the PPU.



Enter “500” at “Spindle speed”



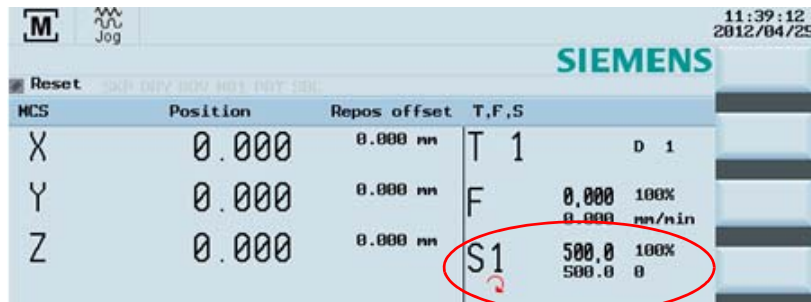
Select “M3” using the “Select” key on the PPU



Press the “CYCLE START” key on the MCP



## SEQUENCE



MCS	Position	Repos offset	T,F,S
X	0.000	0.000 mm	T 1 D 1
Y	0.000	0.000 mm	F 0.000 100% 0.000 mm/min
Z	0.000	0.000 mm	S1 500.0 100% 500.0 0

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



Press the "Back" SK on the PPU



Create workpiece offset

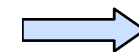


A tool must have been created and measured before it can be used to set the workpiece offset.

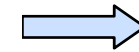


Make sure the active tool is the measured tool!

Press the "Machine" key on the PPU.



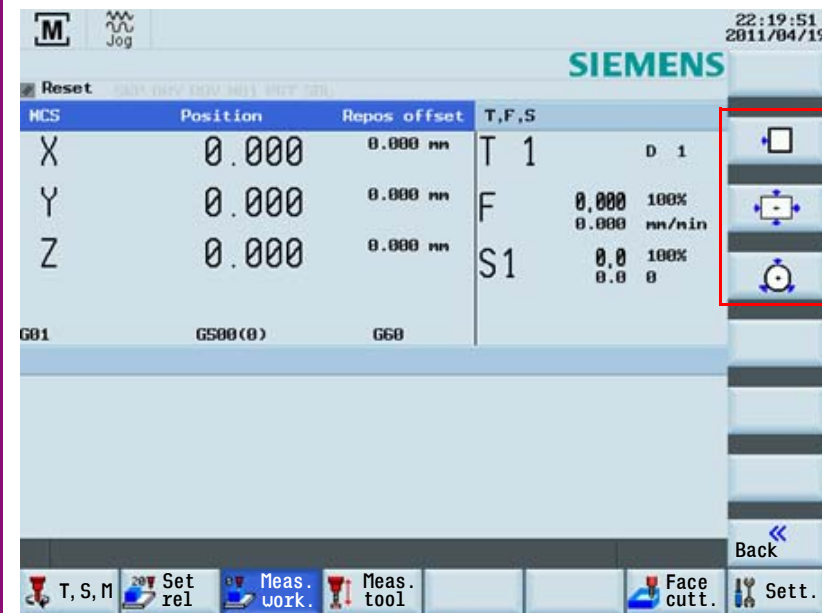
Press the "JOG" key on the MCP.



Press the "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.





## SEQUENCE

**Method 1** This method is normally for setting the zero point of the workpiece 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.

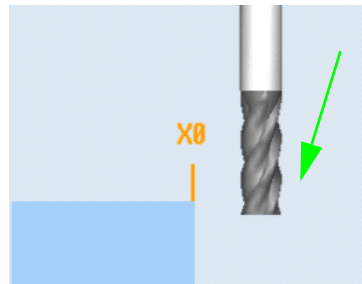
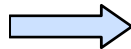
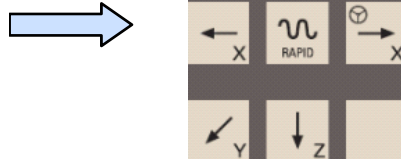
Press the corresponding SK of the first icon on the right-hand side of the PPU.



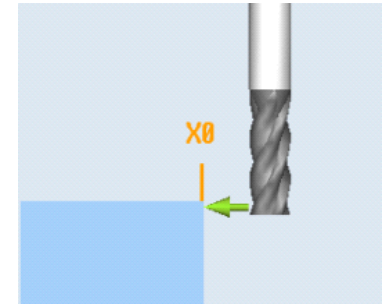
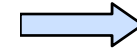
Press the appropriate SK to select the feed axis which needs to be set up.



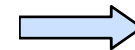
Press the axis traverse keys to move the tool to the required setting position in the X axis.



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



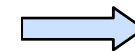
Select “Save in” Offset “G54” (or other offset).



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.

## 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.



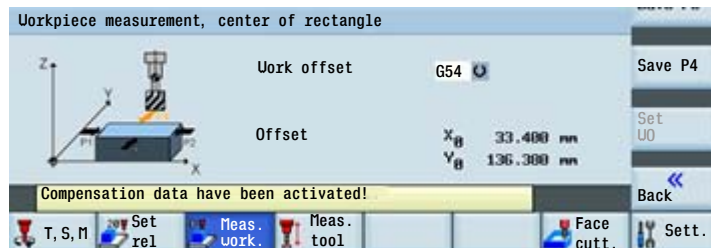
Repeat the process for positions 2, 3 and 4.  
(When the setting is complete, the buttons will be shown in blue.)



Press the "Set WO" SK on the PPU.



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.

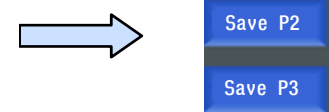


Observing the figure on the PPU, move the coordinate axis following the orange arrow to move the tool to the specified position and scratch the edge of the workpiece.

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



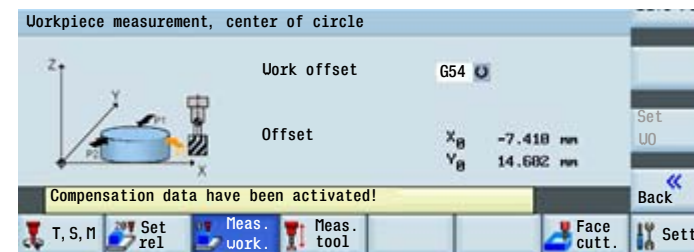
Repeat the process for positions 2 and 3.  
(When the setting is complete, the buttons will be shown in blue.)



Press the "Set WO" SK on the PPU.

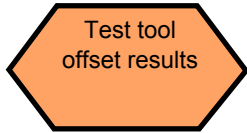


You have then finished setting the zero point of the workpiece as the center point of the circular workpiece.





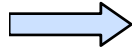
## SEQUENCE



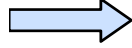
The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

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

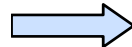
Press the “Machine” key on the PPU



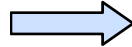
Press the “MDA” key on the MCP.



Press the “Delete file” SK on the PPU.

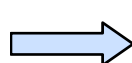


Enter the test program recommended on the right.  
(can also be customized)



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

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

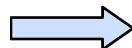


Note: The ROV function activates the feedrate override switch under the G00 function.



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

Press “CYCLE START” on the MCP.



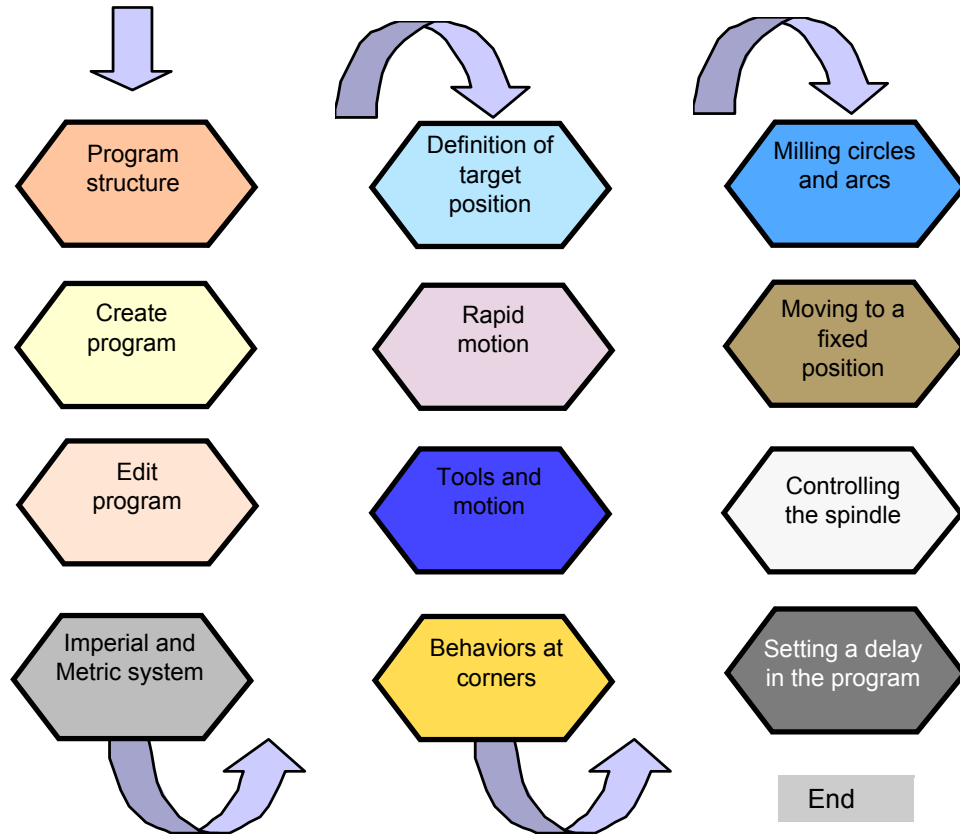
Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast. Observe whether the axis moves to the set position.

## Create Part Program Part 1

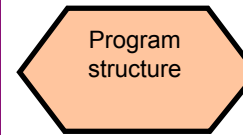
## Description

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.

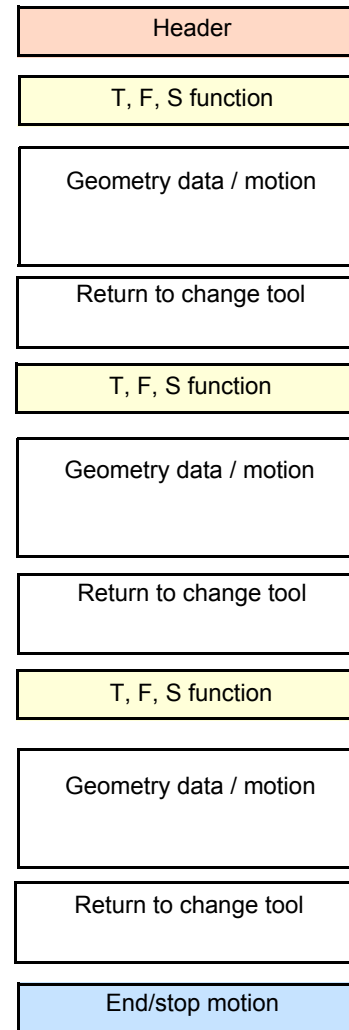
## Content



## Basic Theory



A standard program structure is not needed but is recommended in order to provide clarity for the machine operator. We recommend 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.



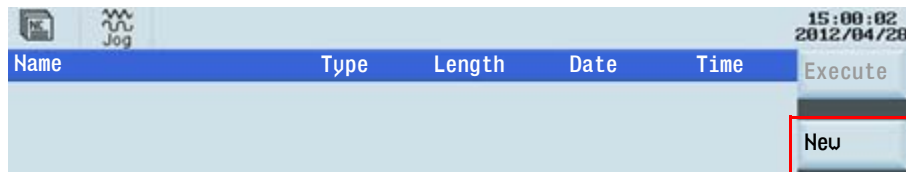
## Step 2

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.

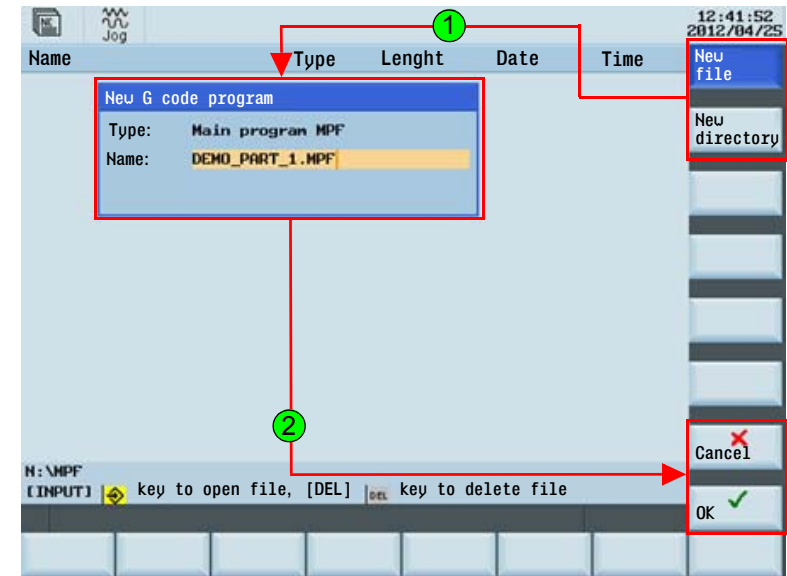


## Step 4

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

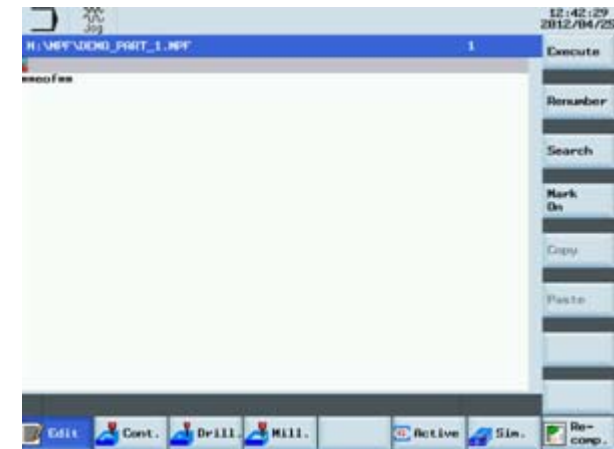
Choose “New” to create a program.

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

## 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

N20 G00 X100 Y100 Z5

N25 G01 Z-5

N30 Z5

N35 G00 Z500 D0

Geometry data / motion

Return to change tool

## 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

N20 G00 X3.93 Y3.93 Z5

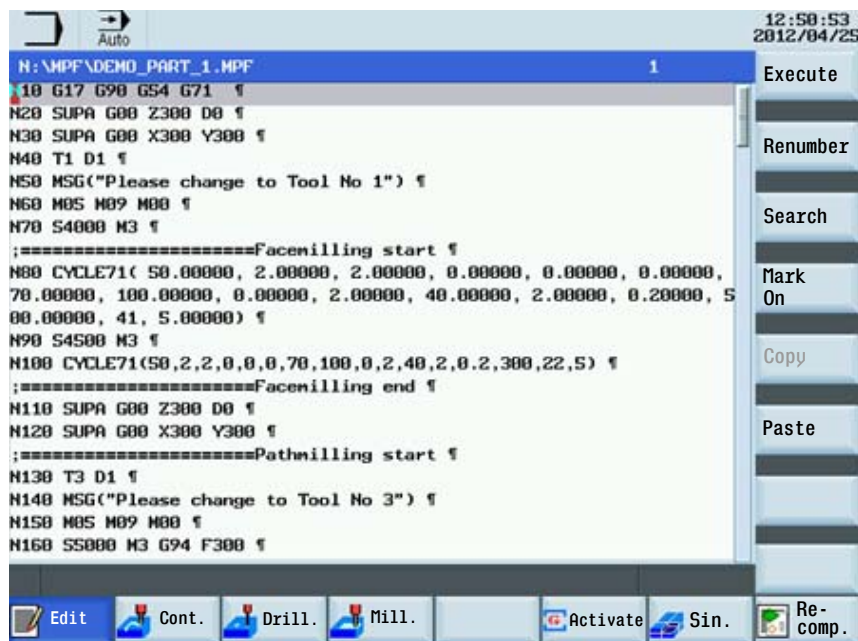
N25 G01 Z-0.787

N30 Z0.196

N35 G00 Z19.68 D0

Geometry data / motion

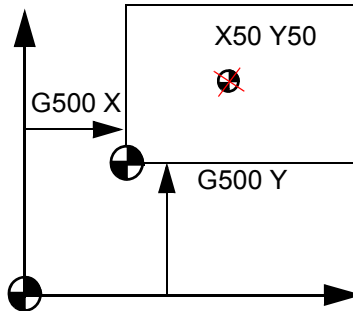
Return to change tool



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

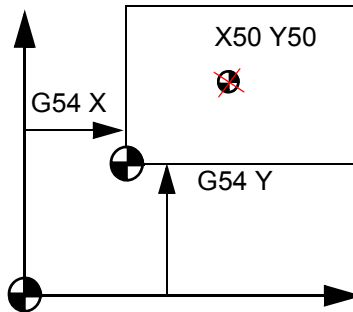
N30 **Z5**

N35 G00 **Z500** D0

Or

**G54 G55 G56 G57  
G58 G59**

With G500 = 0, the offset for the work-piece 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

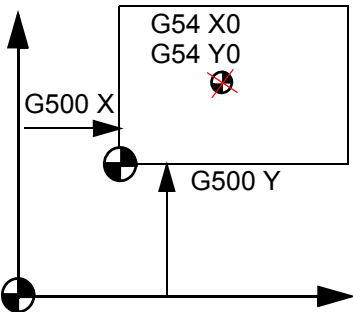
N30 **Z5**

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.

**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 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

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**

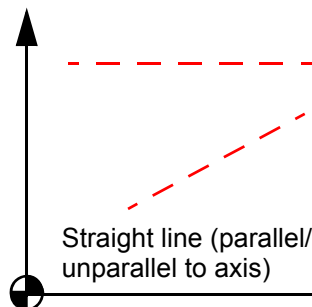
N30 **Z0.196**

N35 G00 **G90 Z19.68** D0

## 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

- 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

**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).

**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.

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

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

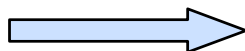
N35 G00 Z500 D0

## Tools and motion

**T1 D1 M06**

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

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



N5 G17 G90 G54 G71

N10 T1 D1 M6

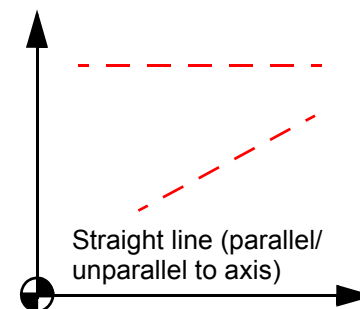
N15 S5000 M3 G94 F300

N20 G00 X50 Y50 Z5

N25 G01 Z-20

N30 Z5

N35 G00 Z500 D0



## Basic Theory

## Behaviors at corners

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

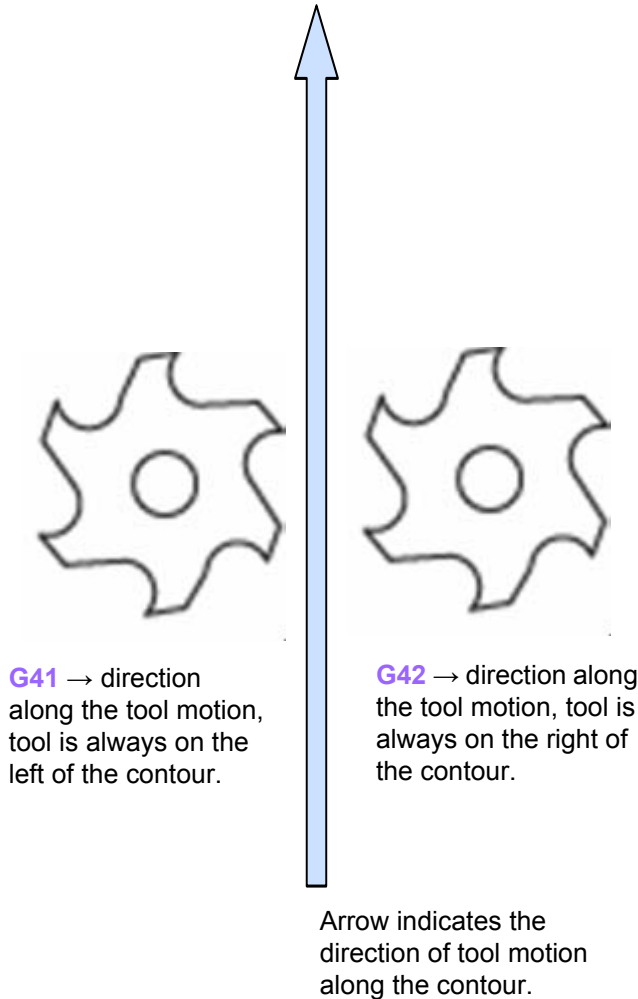
## G41 / G42 and G40

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



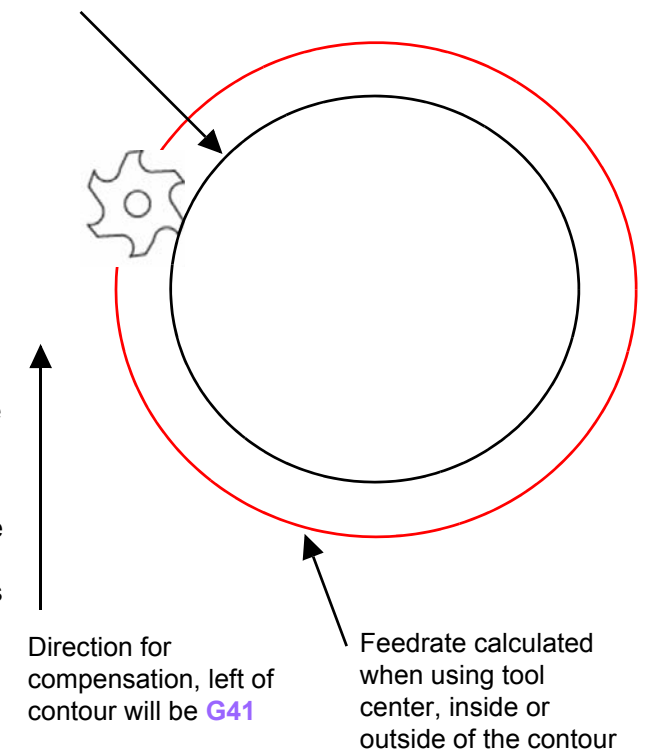
When traversing circular contours with cutter radius compensation, it should be decided whether the feed rate should be calculated along the contour of the workpiece or along the path defined by the center point of the cutting tool.

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.

Contour feedrate with **CFC**



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



## 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.

Two common types of defining circles and arcs:

- ① :G02/G03 X\_Y\_I\_J\_;
- ② :G02/G03 X\_Y\_CR=\_;

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

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



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

```

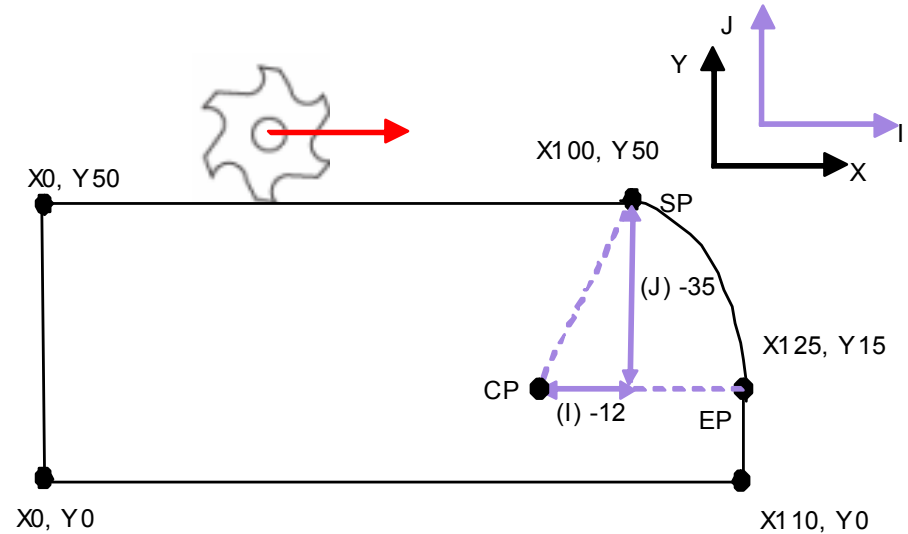
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X-20 Y-20 Z5
N25 G01 Z-5
N30 G41 X0 Y0
N35 Y50
N40 X100
N45 G02 X125 Y15 I-12 J-35
N50 G01 Y0
N55 X0
N60 G40 X-20 Y-20
N35 G00 Z500 D0
  
```

## Note:

N45 can also be written as follows

```
N45 G02 X125 Y15 CR=37
```

Determine tool radius of T1 D1



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

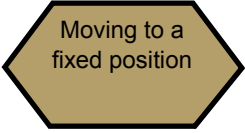
J = defined relative increment from start point to center point in Y

G2 = define circle direction in traversing direction = G2 clockwise

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



## Basic Theory



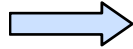
Moving to a fixed position

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

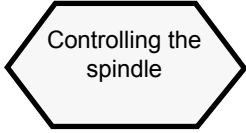


```
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 Z=0 ;reference point
```

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

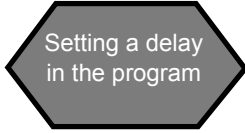
**M4** accelerate to programmed speed counter-clockwise

**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



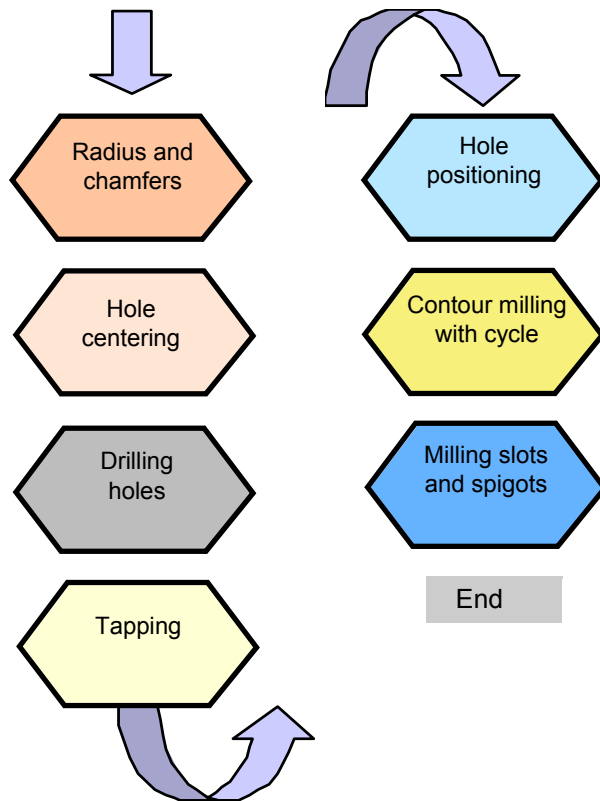
```
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 G04 F5
N35 Z5 M4
N40 M5
N45 M19
N35 G00 Z500 D0
```

## Create Part Program Part 2

## 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.

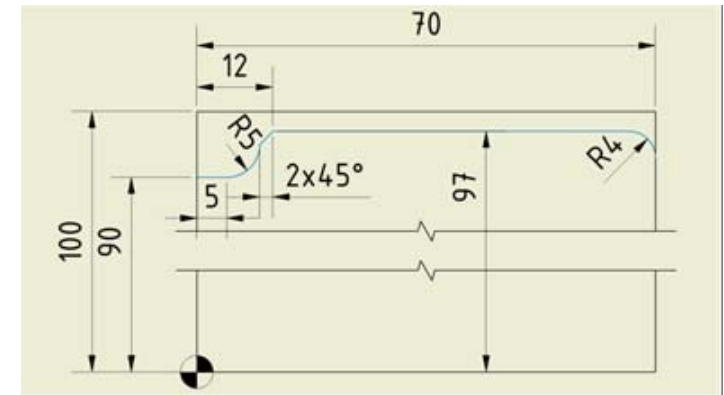
## Content



## 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)

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

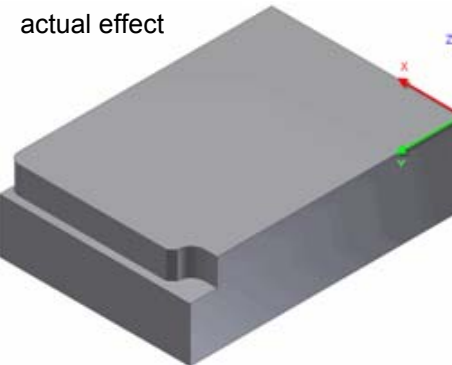
N110 G01 Y97 CHR=2

N115 G01 X70 RND=4

N120 G01 Y90

N125 G01 G40 X80

N130 G00 Z50



## Basic Theory



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

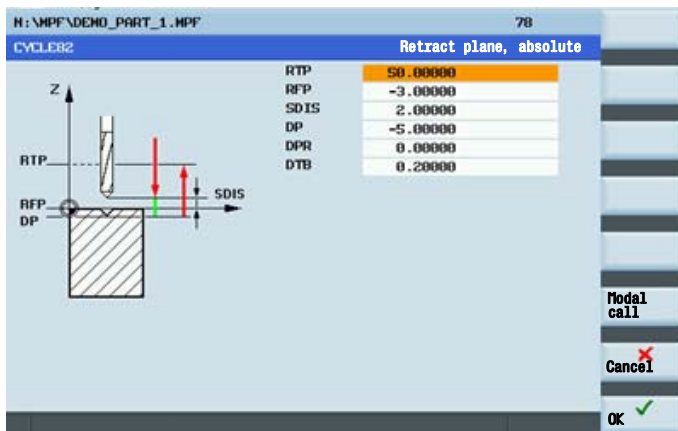
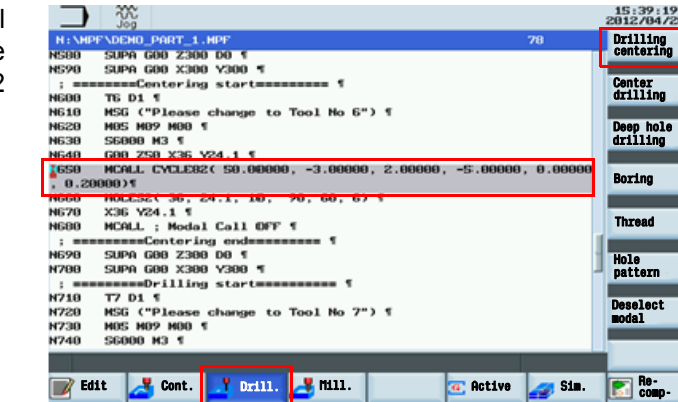
CYCLE82: With delay at current hole depth



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

Drilling centering → Center drilling

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

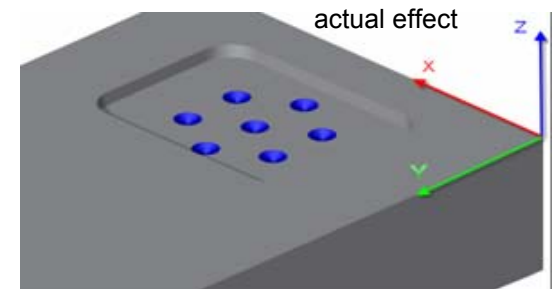
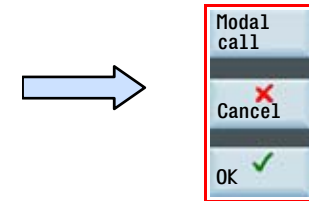


With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

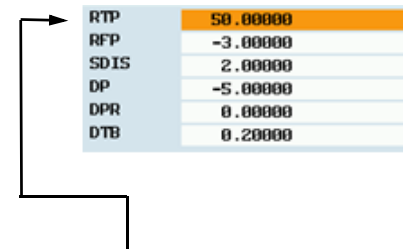
This will drill a hole at the current position.

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.



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



RTP	50.00000
RFP	-3.00000
SDIS	2.00000
DP	-5.00000
DPR	0.00000
DTB	0.20000

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

## 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.

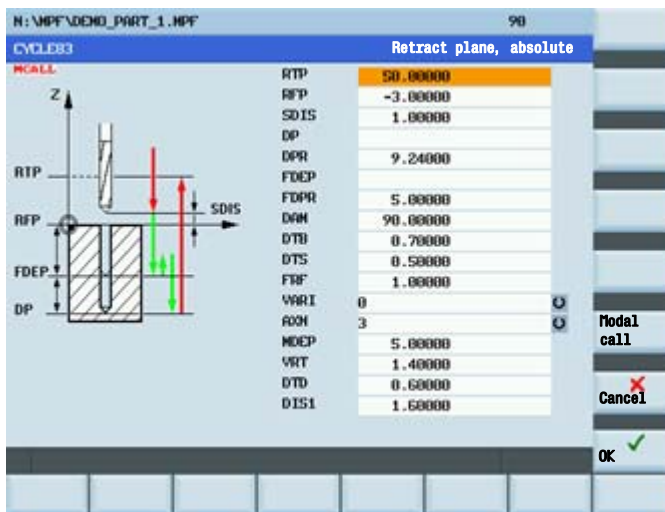
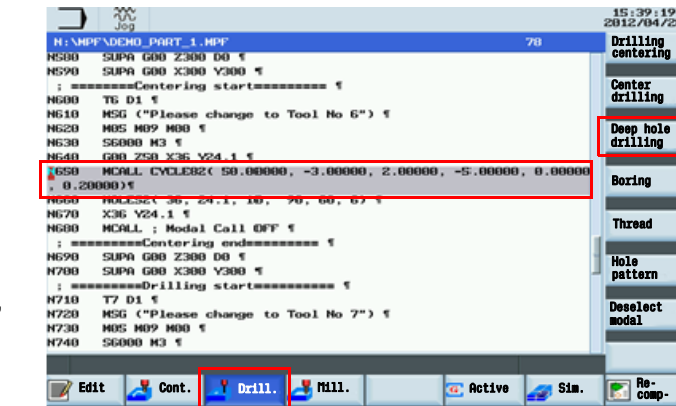


The relevant cycle can now be found using the vertical SKs on the right.



Deep hole drilling

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

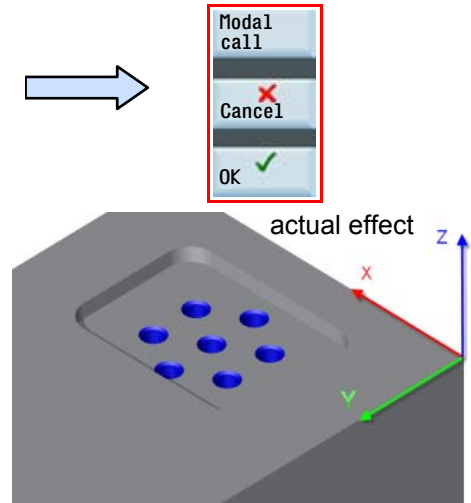


With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

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 91

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

① DAM≠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.

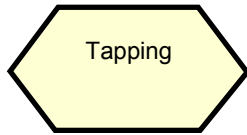
② DAM=0, drilling depth each time is same as the 1st drilling depth (FDPR),

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

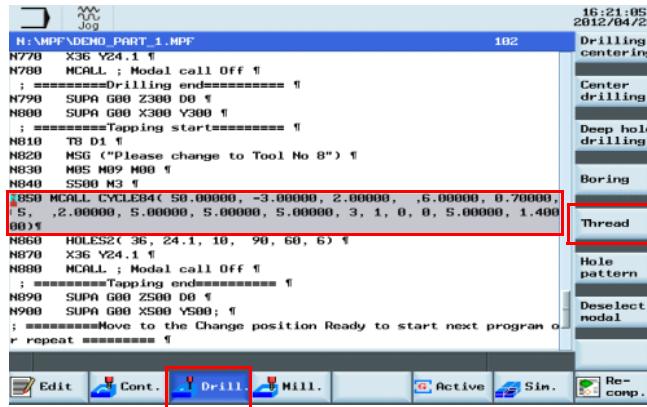
Feed times	Every feed depth/mm DAM=2	Actual depth/mm	Feed times	Every feed depth/mm DAM=0	Actual depth/mm
1.	FDPR=10	-10	1.	FDPR=10	-10
2.	FDPR-DAM=10-2=8	-18	2.	FDPR=10	-20
3.	(FDPR-DAM)-DAM=8-2=6	-24	3.	FDPR=10	-30
4.	(FDPR-2DAM)-DAM=6-2=4	-28	Remaining depth =10 < 2xFDPR, the remaining depth distribute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

## Basic Theory



The easiest way to tap a hole is to use  
CYCLE84: Solid tap holder  
CYCLE840: With floating tap holder.

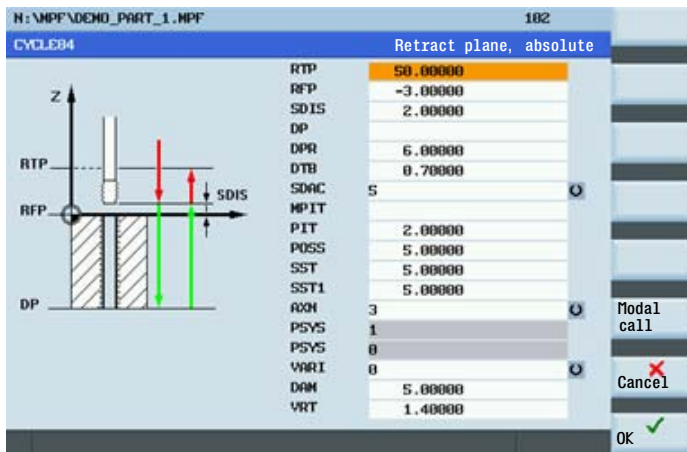
The cycles can be found and parameterized using the "Drill." SK.



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.



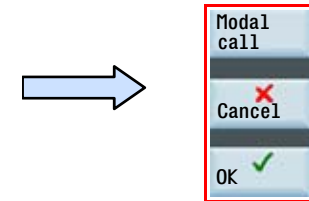
With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

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

RTP	50.00000	
RFP	-3.00000	
SDIS	2.00000	
DP		
DPR	6.00000	
DTB	0.70000	
SDAC	5	
MPIT		
PIT	2.00000	
POSS	5.00000	
SST	5.00000	
SST1	5.00000	
AXN	3	
PSYS	1	
PSYS	0	
VARI	0	
DAM	5.00000	
VRT	1.40000	



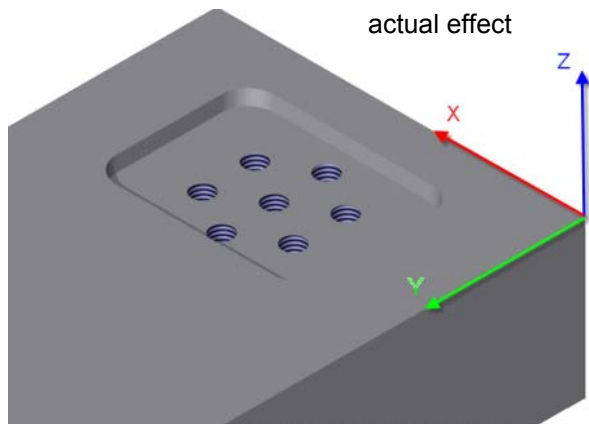
N325 MCALL CYCLE84( 50.00000, -3.00000, 2.00000, ,6.00000, 0.70000, 5, ,2.00000, 5.00000, 5.00000, 5.00000, 3, 0, 0, 0, 5.00000, 1.40000)

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 and DP, please see Page 91

For descriptions of AXH, VARI, DAM and VRT, please see Page 93

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:0.001~2 000 mm)	Right hand thread with 2mm pitch	Evaluate value→left hand thread
POSS=5	Spindle stops at 5° (unit: °)	
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.	

## Basic Theory

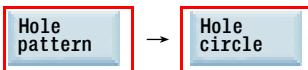


The easiest way to drill a series of holes is to use the pre-defined “Hole pattern” cycles.

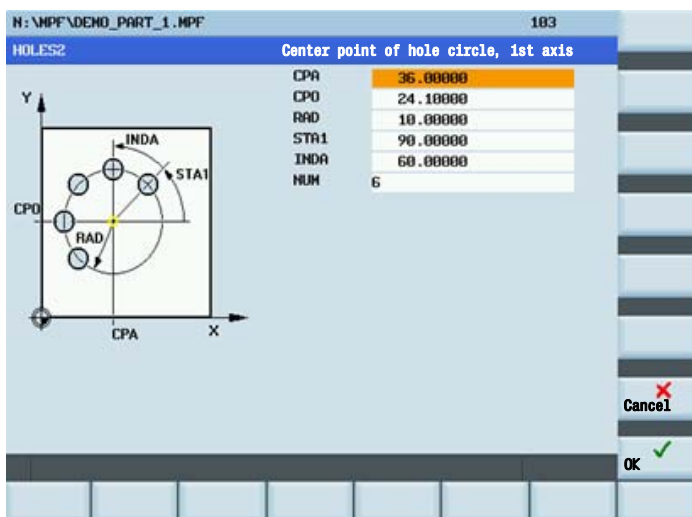
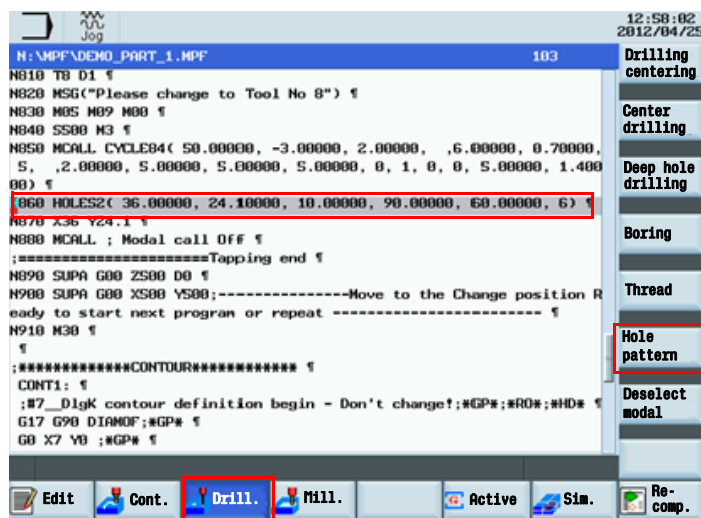
The cycles can be found and parameterized via the “Drill.” SK.



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

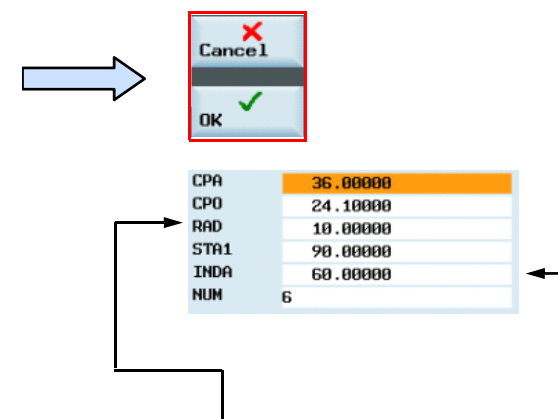


Select “Hole pattern” using the vertical SKs, and then select “Hole circle”, and parameterize the cycle according to requirement.



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.

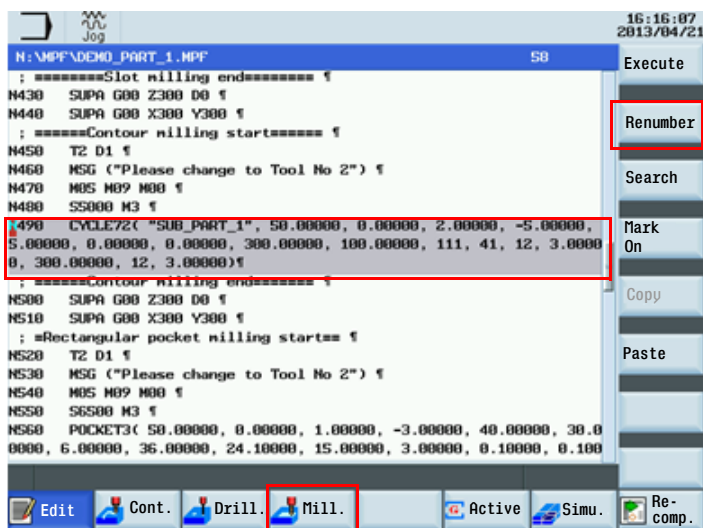


## Basic Theory

Contour milling  
with cycle

The easiest way to rough and finish around a contour is to use the contour milling function.

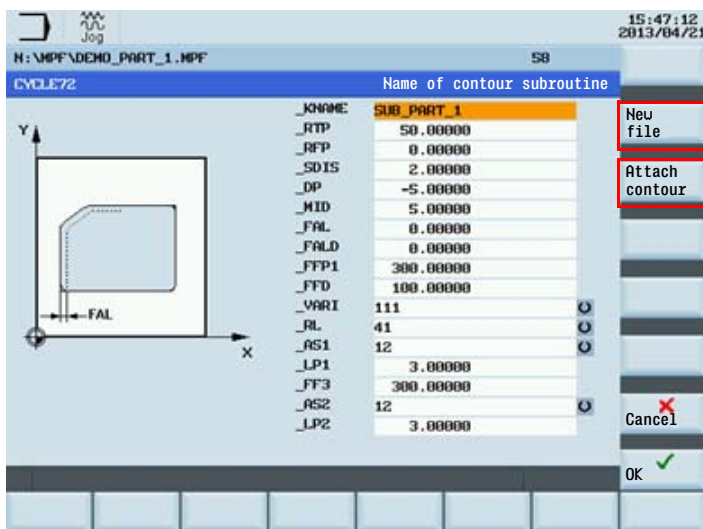
The cycle can be found and parameterized via the "Mill." SK.



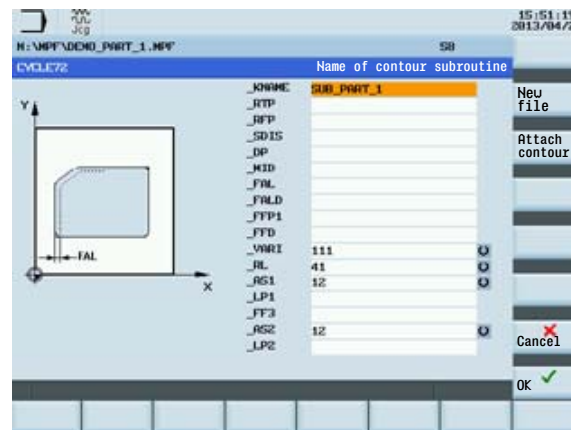
The "Contour milling" SK can be found in the vertical SKs on the right.

Contour  
milling

The parameterization is performed as in this figure.

New  
file

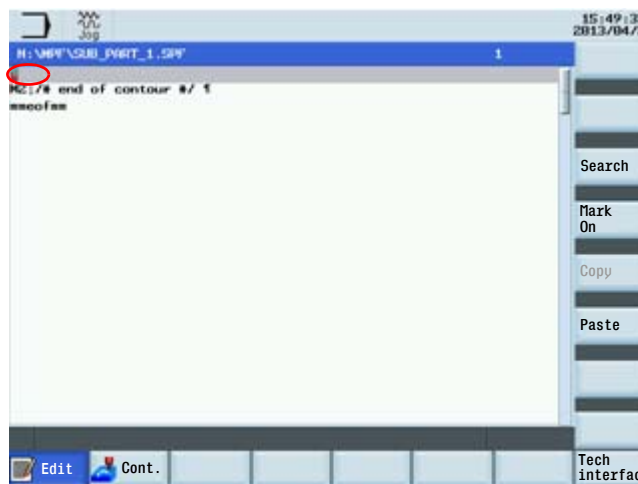
By selecting the "New file" SK, the contour turning data can be inserted into Sub Program File (.SPF). You can edit and change it when selected. The sequence is as follows:



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

Contour  
milling

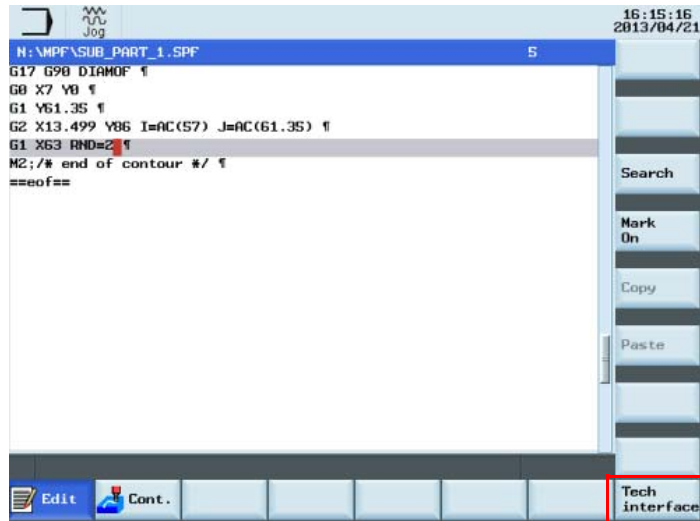
Press "New file" on the PPU to create contour information in the .SPF file. The cursor moves to the contour editing position automatically.

New  
file

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

## Basic Theory

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



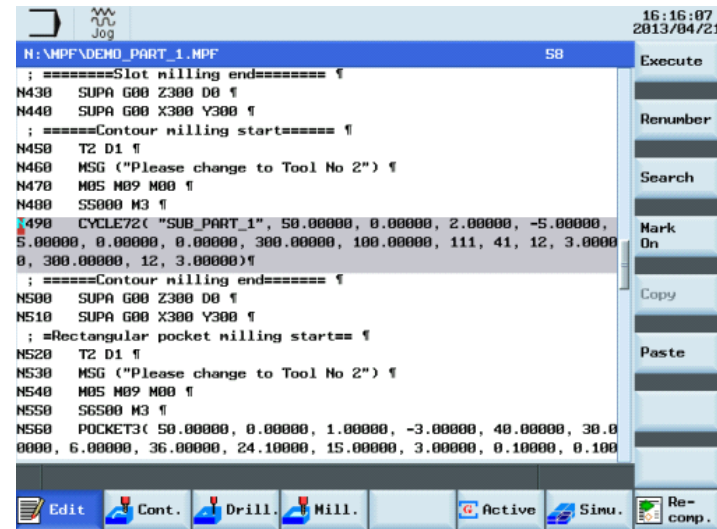
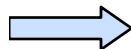
Enter appropriate coordinates based on the data from the technical drawing.

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.

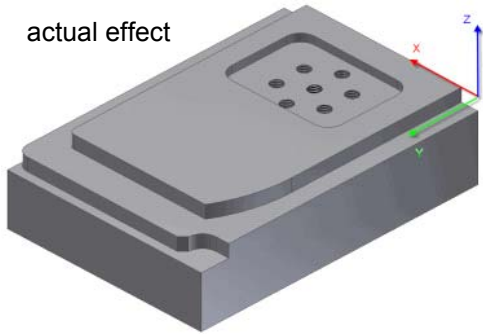


After all the settings take effect, the selected cycle and set data will be transferred to corresponding part program automatically (for further information, see next page).

**Basic Theory**

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

actual effect



KNAME	SUB_PART_1
_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( "SUB\_PART\_1", 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 91

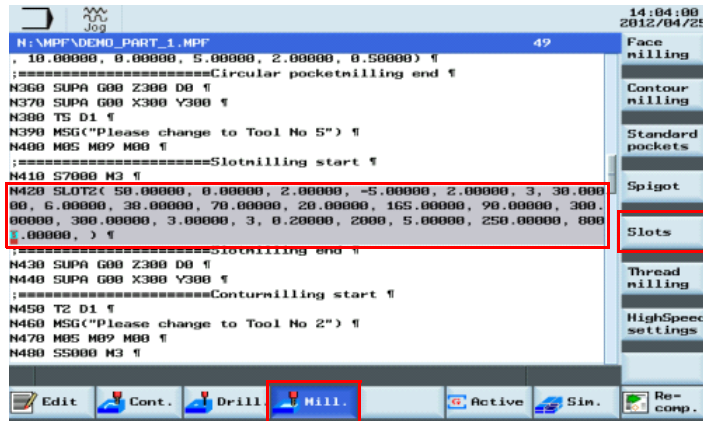
Parameters	Meanings	Remarks
KNAME= CONT1:CONT1_E	Set the name of the contour subprogram as "CONT1" ("CONT1_E" is automatically created)	The first two positions of the program name must be letters
MID=5	The maximal feed depth is 5 mm	
FAL=0	Finishing allowance at the contour side is 0 mm	
FALD=0	Finishing 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

## 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.



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

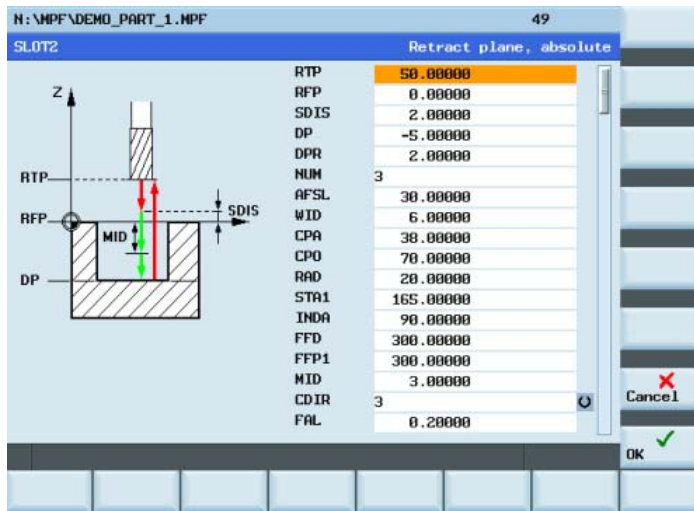
This will perform milling at the position defined in the cycle.



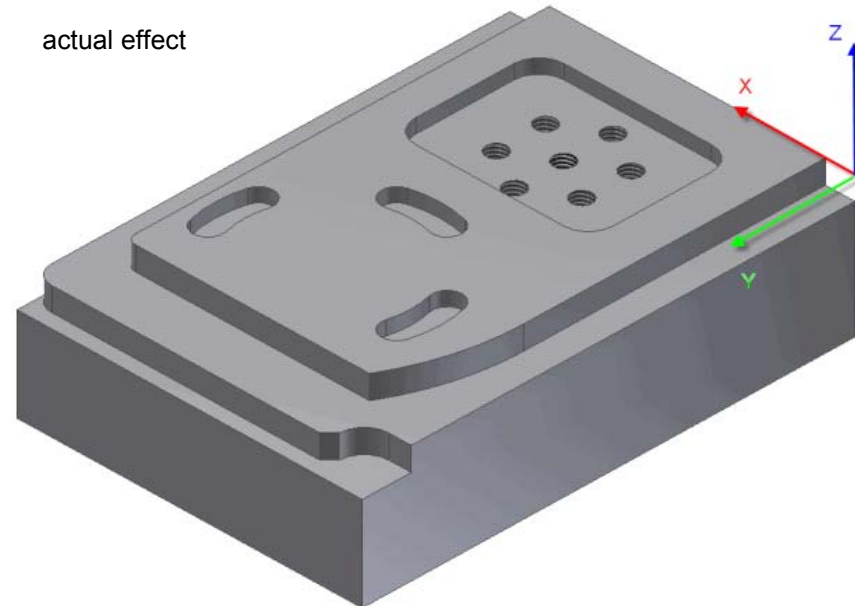
The relevant cycle can be found using the vertical SKs on the right.



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



actual effect



## 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	8000.00000
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, )

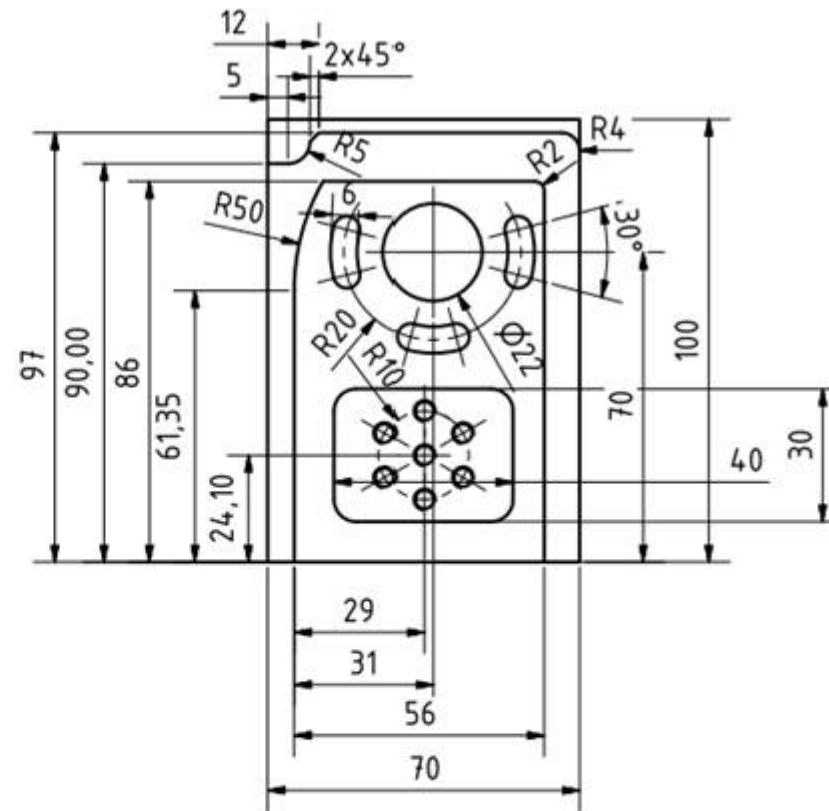
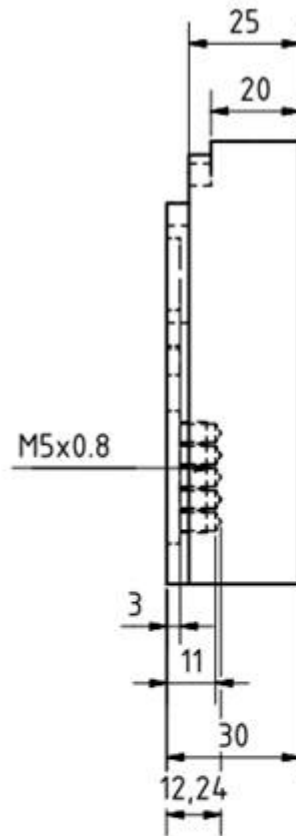
For descriptions of RTP, RFP, SDIS, DP and DPR, please see Page 91

For descriptions of CPA, CPO and RAD, please see Page 96

For descriptions of FFD and FFP1, please see Page 99

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 rpm	
	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.	





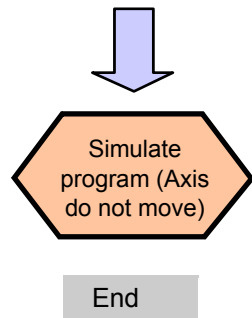
					Datum	Name			
				Gesichtet					
				Kontrolliert					
				Norm					
							DEMO PART MILLING		1
									A4
Status	Änderungen	Datum	Name						

## Simulate Program

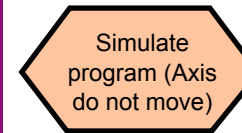
### Description

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

### Content



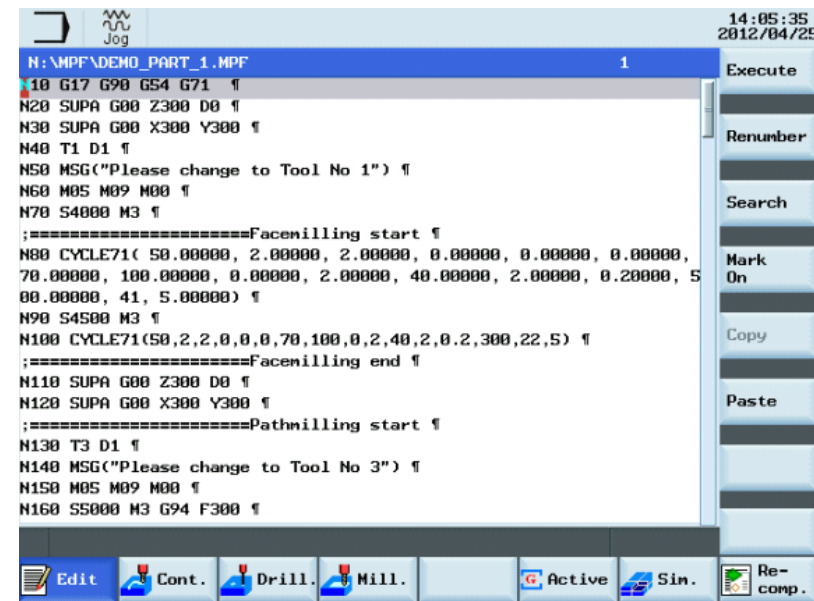
### Basic Theory



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

#### Step 1

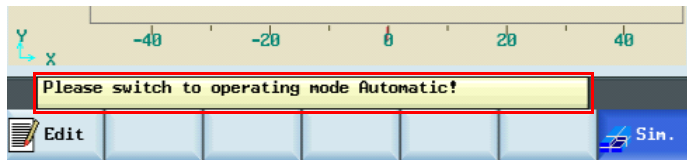
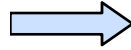
The part program must be opened using the "Program Manager" on PPU.



## SEQUENCE

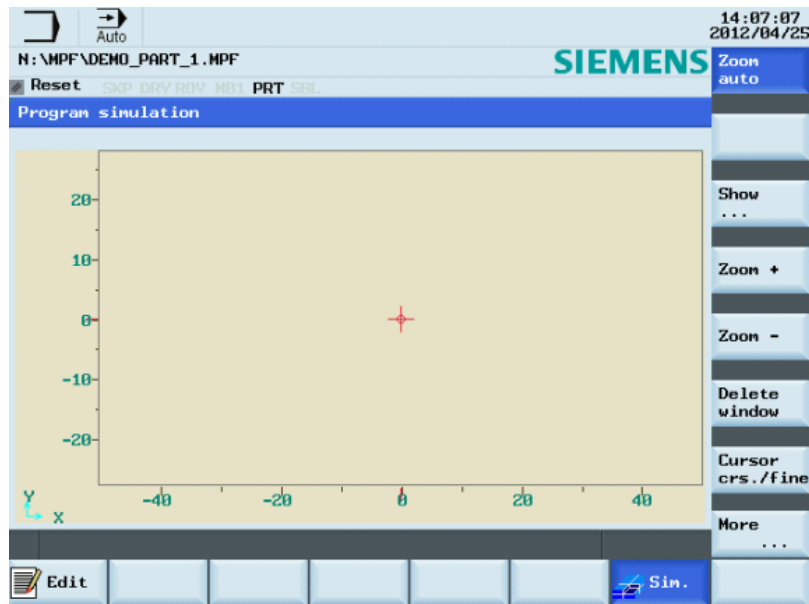
## Step 2

Press the "Simu." SK on the PPU.



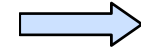
If the control is not in the correct mode, a message will be displayed at the bottom of the screen.

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

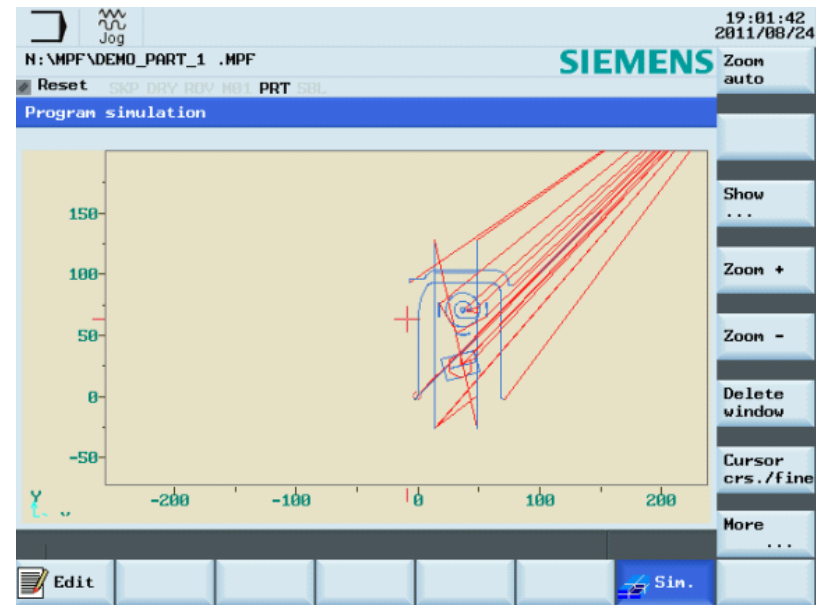


## Step 3

Press the "CYCLE START" key on the MCP.



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



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



End

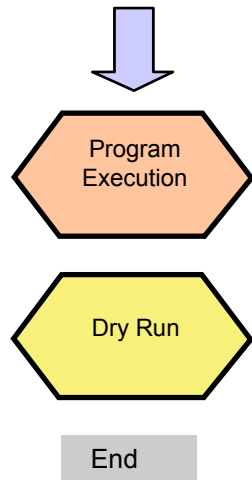


## Test Program

### Description

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

### Content



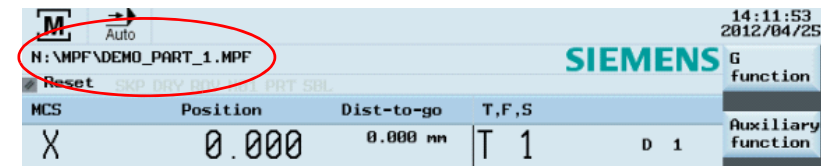
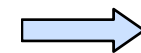
### Basic Theory



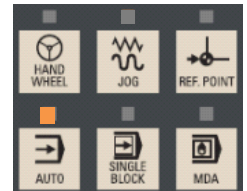
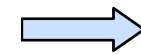
Before the part program can be loaded and executed in AUTO mode, it must be tested using the simulation function mentioned previously!



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!

## SEQUENCE



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

**Note:** The following operation is based on the finished “program execution” screen

## Step 1



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

Press the “Offset” key on the PPU.



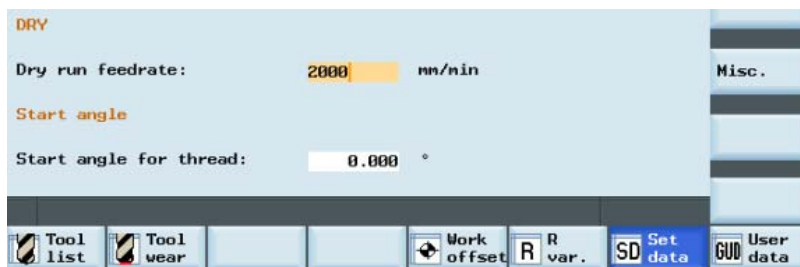
Press the “Sett. data” SK on the PPU.



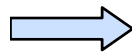
Use the traversing key to move to the required position. The position is now highlighted.



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



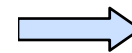
Press the “Input” key of the PPU.



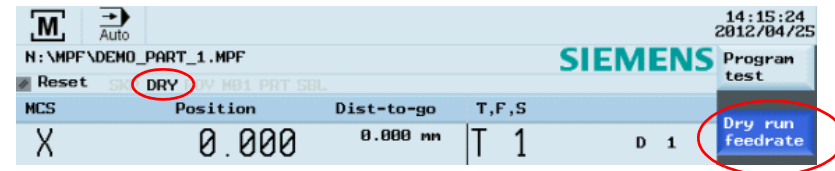
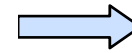
Press the “Machine” key on the PPU.



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.

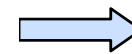


## Step 2

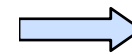


Make sure the feedrate override on the MCP is 0%.

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.



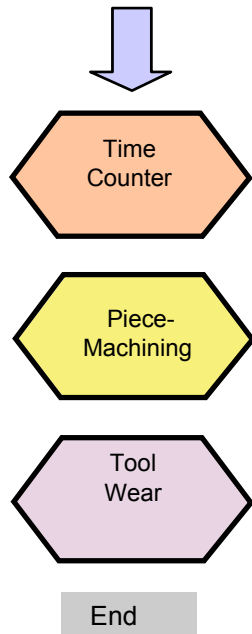
After finishing the dry run, please turn the changed offset back to the original value in order to avoid affecting the actual machining!

## Machine Pieces

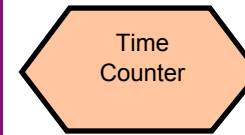
### Description

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

### Content



### Basic Theory



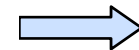
Make sure the machine has been referenced before machining workpieces!

#### Step 1

Press the "Machine" key on the PPU.



Press the "Auto" key on the MCP.



Press the "Time counter" SK on the PPU.



Block display	DEMO_PART_1.MPF	Time, counter
DN10 G17 G90 G54 G71		Cycle time 0000:00:06h
N20 SUPA G00 Z300 D0		Time left 0000:00:00h
N30 SUPA G00 X300 Y300		
N40 T1 D1		
N50 MSG("Please change to Tool No 1")		Counter No
N60 M05 M09 M00		
N70 S4000 M3		

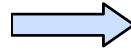
## SEQUENCE

"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.



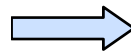
Time left 0000:00:00h

Step 2

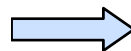


The "Time left" can only be counted after a successful cycle run of a part program!

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



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



Required 45

"Actual" shows the number of workpieces that have been machined.

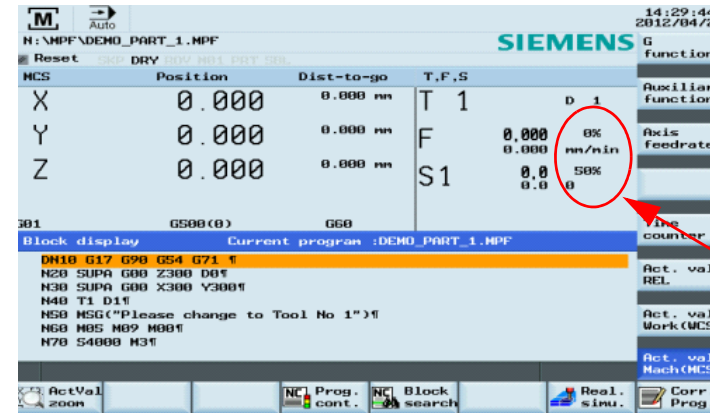


Actual 8

Block display	DEMO_PART_1.MPF	Part counter	Part timer
DN10 G17 G90 G54 G71 #		Cycle time	0000:00:06h
N20 SUPA G00 Z300 D0#		Remaining ti	0000:00:00h
N30 SUPA G00 X300 Y300#		Counter	Yes
N40 T1 D1#		Required	45
N50 MSG("Please change to Tool No 1")#		Actual	8
N60 M05 M09 M00#			
N70 S4000 M3#			



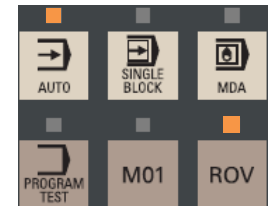
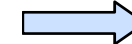
Make sure the program is correct before machining pieces!



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.

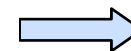


Make sure that the feedrate override on the MCP is 0%!

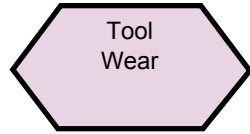
Press "Door" on the MCP to close the door of the machine. (If you don't use this function, just close the door on the machine manually.)



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



## SEQUENCE



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

## Step 1

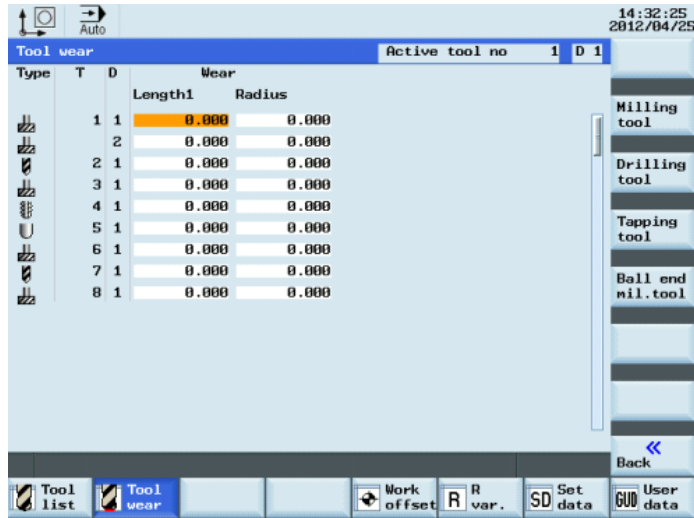
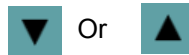
Press the "Machine" key on the PPU.



Press the "Auto" key on the MCP.



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



Sett. data

## 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.

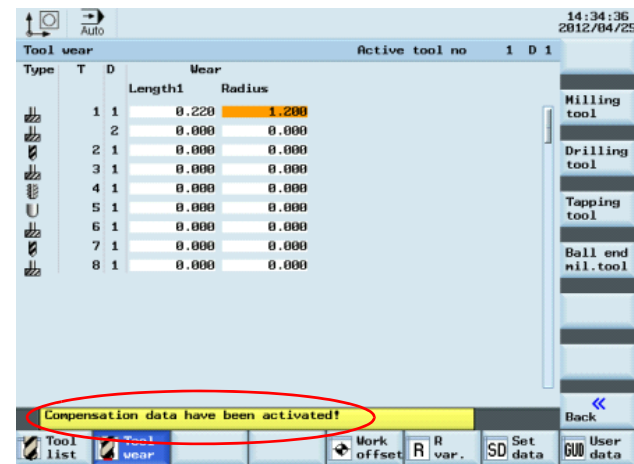
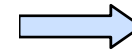


Set the tool radius wear parameter in "Radius", the sign determines the direction of wear compensation.

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.

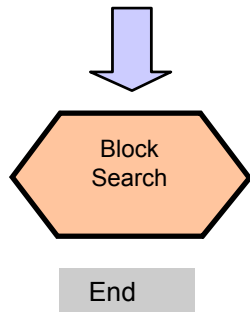


## Program Restart

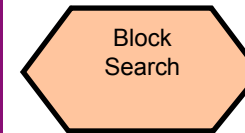
### Description

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

### Content



### Basic Theory



Press the "Machine" key on the PPU.



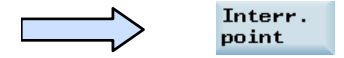
Press the "Auto" key on the MCP.



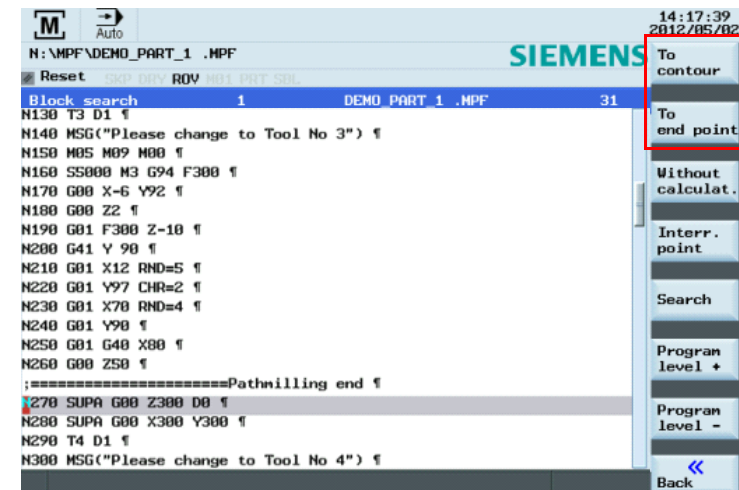
Press the "Block search" SK on the PPU.



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.

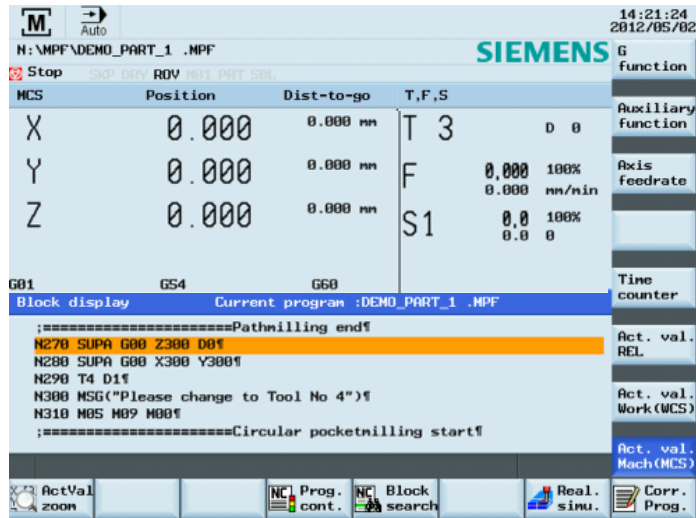
"To contour": The program will continue from the line before the breakpoint.

"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)



## SEQUENCE



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.

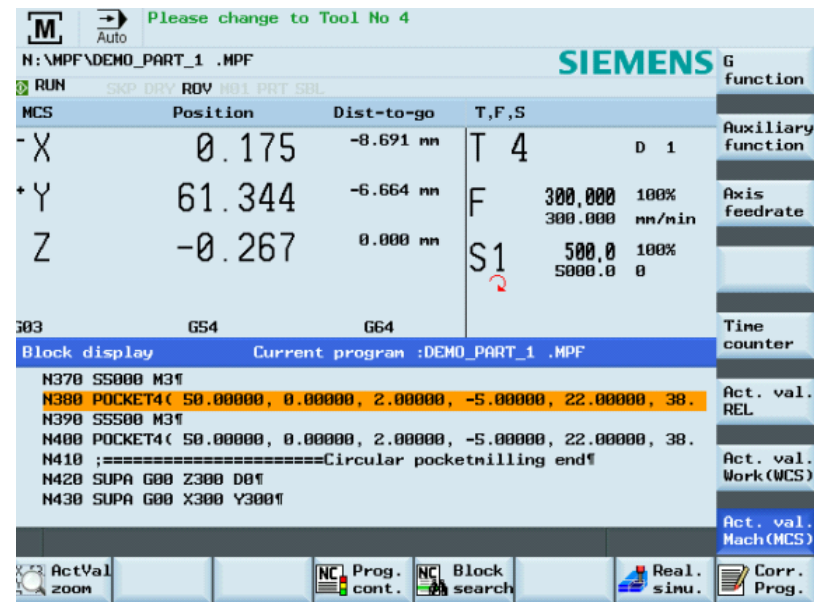


Alarm 010208 is shown at the top prompting to press the “CYCLE START” key to continue the program.

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



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



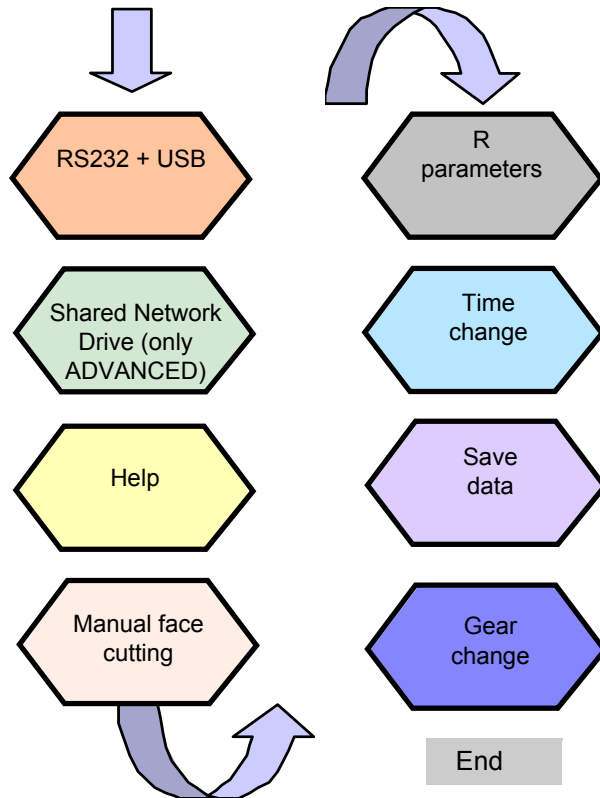


## Additional Information Part 1

## 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.

## Content



## SEQUENCE



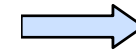
RS232 is used to transfer the programs to and from the NC.

## 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.

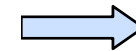
Press “Program Manager” on the PPU.



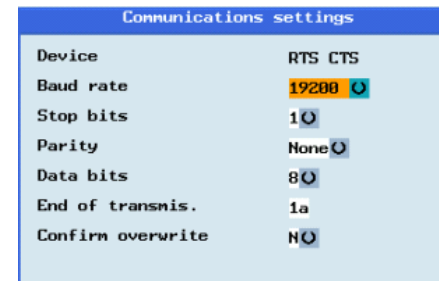
Press the “RS232” SK on the PPU.



Press the “Settings” SK on the PPU.



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



Press the “Save” SK on the PPU.



Press the “Back” SK on the PPU.



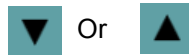
## SEQUENCE

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

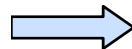
Press the "NC" SK on the PPU.



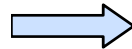
Use "Cursor + Select" to select the required part program. The selected program will be highlighted.



Press the "Copy" SK on the PPU.



Press the "RS232" SK on the PPU.



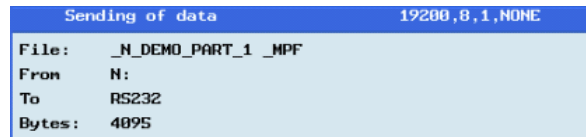
Check the interface setting and start the communication software to receive the program on PC.

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

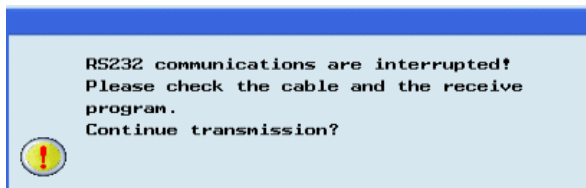
Press the "Send" SK on the PPU.



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

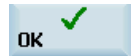


If there is a problem during transfer of the part program, a window will be displayed.



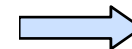
You can continue sending the part program.

Press the "OK" SK on the PPU.



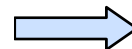
Or you can abort the sending of the part program.

Press the "Cancel" SK on the PPU.

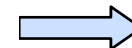


**Step 3** Transfer a part program to the PPU from a PC.

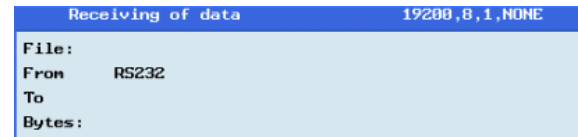
Press "Program Manager" on the PPU.



Press the "RS232" SK on the PPU.



Press the "Accept" SK on the PPU.



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

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

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



## SEQUENCE



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

**Step 4** Use the "Copy" 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. →



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



Press the "Copy" SK on the PPU. →



Press the "USB" SK on the PPU. →



Press the "Paste" SK on the PPU. →



**Step 5** Use the "Copy" and "Paste" SKs to transfer the part program from NC to USB.

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

Press the "USB" SK on the PPU. →



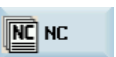
Use "Cursor + Select" to select the required part program.  
The selected program will be highlighted.



Press the "Copy" SK on the PPU. →



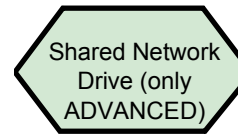
Press the "NC" SK on the PPU. →



Press the "Paste" SK on the PPU. →



## SEQUENCE



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

**Step 1** Set PPU IP address.

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

Press key: +

Press key:

Press "Serv. Displ." SK →



Press "Serv. Displ." SK →

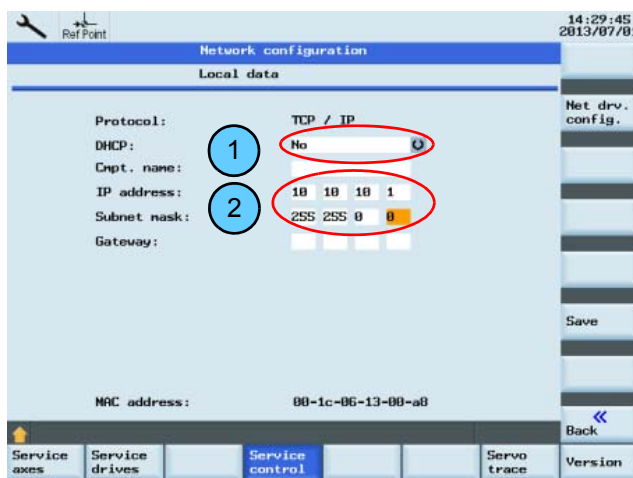


Press "Network Info" button to enter the "Local Configuration Data"



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

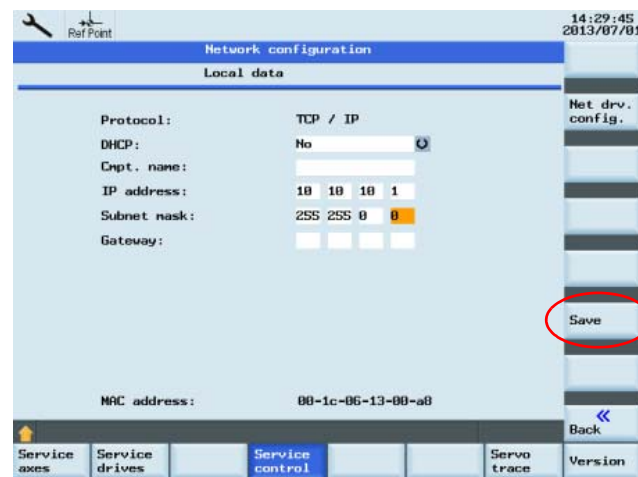
- ① DHCP is set to "No"
- ② IP address and subnet mask can be arbitrarily set according to need. (Right given only as an example)



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

Save

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



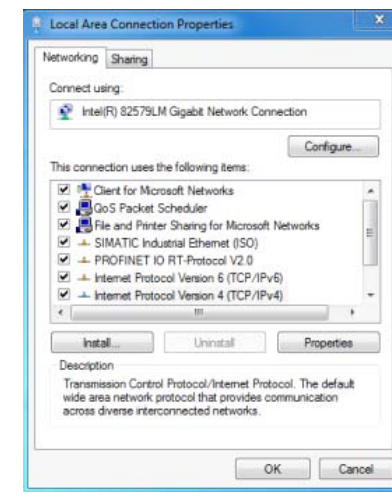
## SEQUENCE

Step 2 Set the PC's static IP address.

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

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

And double click "Properties".



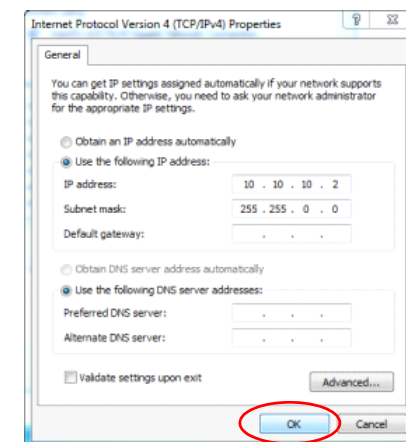
In the dialog box, select "Use the following IP address" and fill in the required IP address.

(Shown right only given as an example)

Select "OK" to complete the setup.

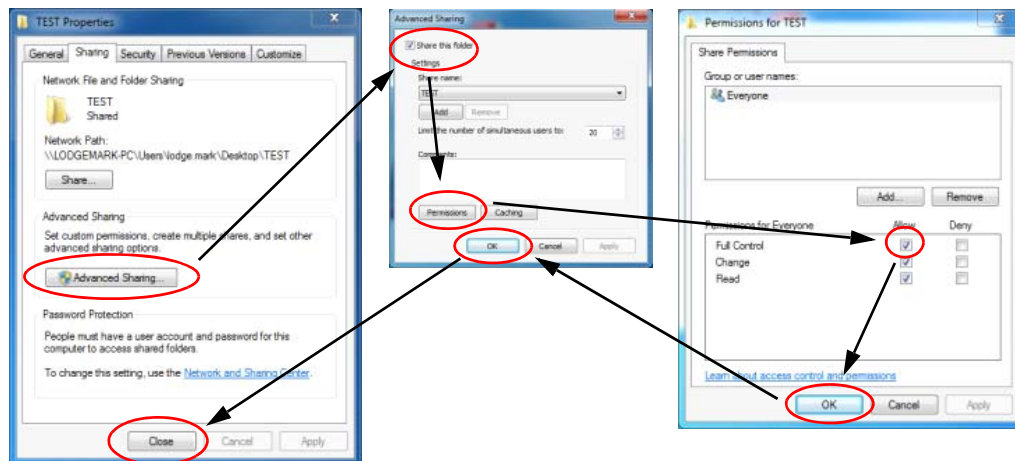
Note: The address "10.10.10.2" setting is based on the first step in the IP address of the PPU.

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



**Step 3** On PC create a shared folder.

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



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

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

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

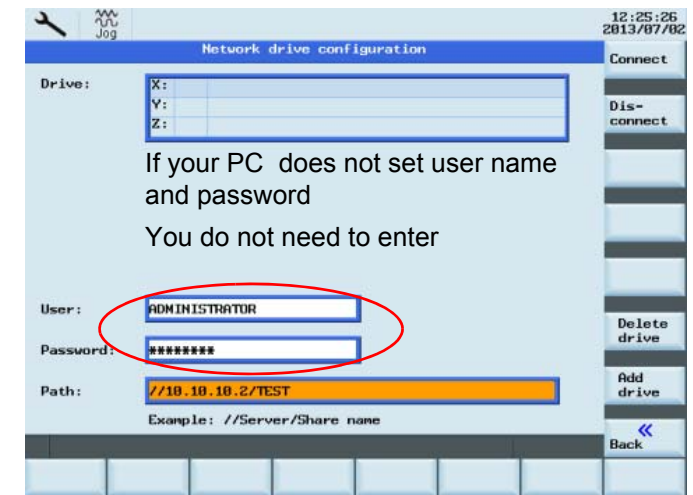
**SEQUENCE**

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

Server: IP address

Share Name: the name of the shared folder

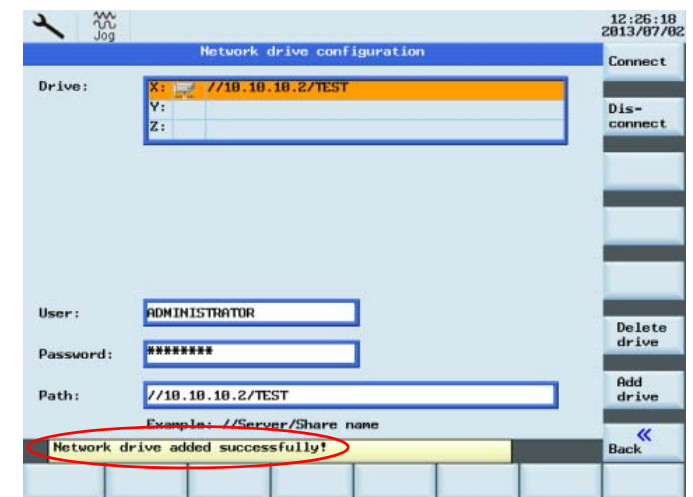
Note: Use "TAB" key to switching between different tasks boxes.



**Add drive**

Press "Add Drive" SK to add it to the specified drive letter

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



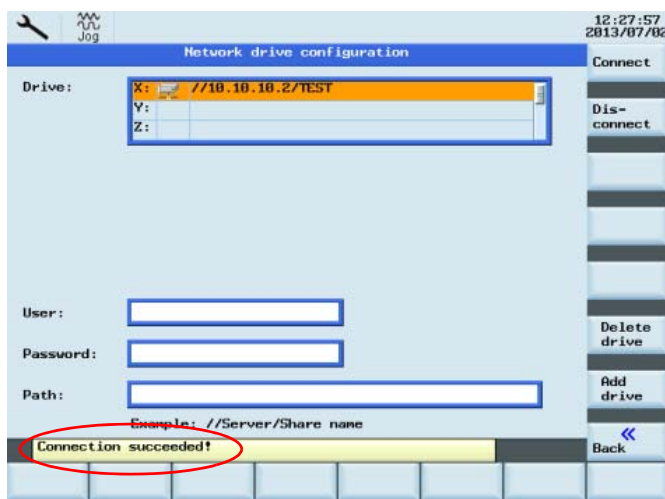


If the connection is lost select the drive path and press "Connect." SK

Connect

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

This will be shown with the text "Connection succeeded"

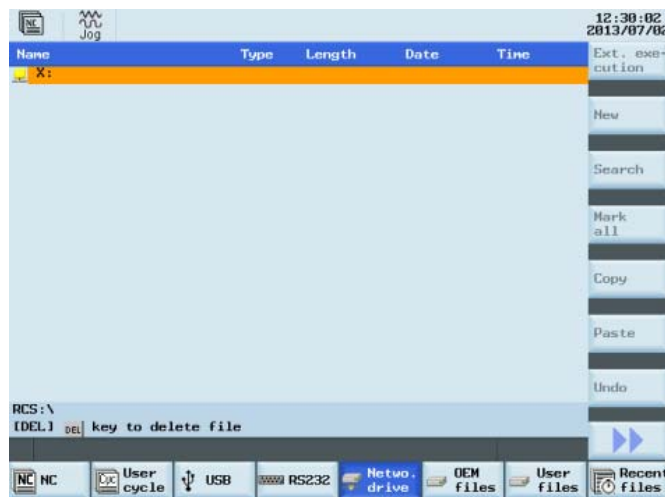


Press "Program Manager" Button

Press "netwo. Drive" SK to enter the network drive interface.



Press "Enter" Button to open network drive to PC/PG.

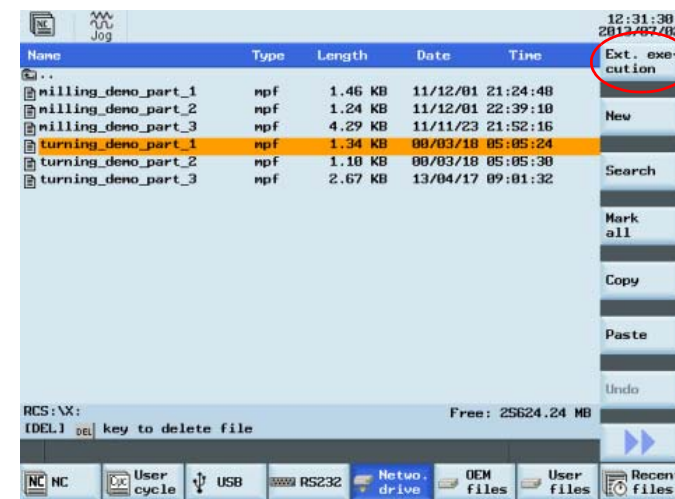


## SEQUENCE

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

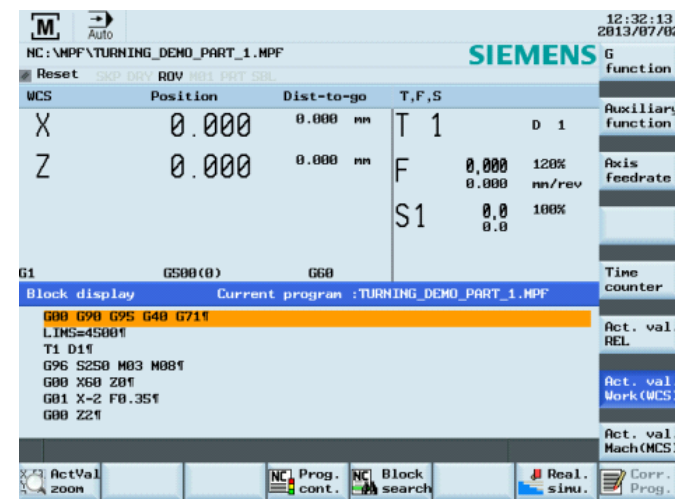
You can select the file you require to execute in AUTO mode, click "Exe. Execution".

Ext. execution



The system will automatically jump to AUTO mode, select the appropriate NC program.

Press the "Cycle Start" button for machining operation.



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

## Help

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

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

Press the "Help" key on the PPU.



Press the "Help" key on the PPU.



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

Press the "OEM Manual" SK on the PPU.



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

Press the "TOC" SK on the PPU.



The online help from the Siemens manual will be shown.

## Manual face cutting

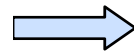
"Face cutting" is used to cut the oversized materials on the rough face before starting to machine.

## Step 1

Press the "Machine" key on the PPU.



Press the "JOG" key on the MCP.

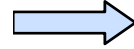


Press the "Sett." SK on the PPU.



Enter appropriate values in "Retraction plane" and "Safety distance".

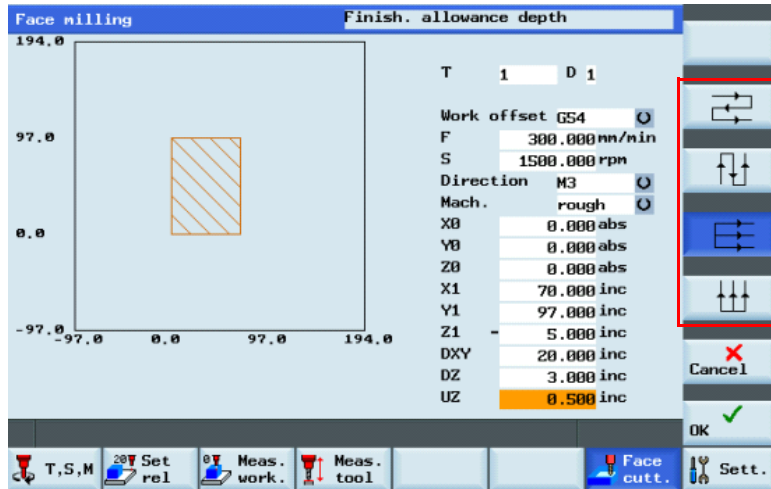
Press the "Input" key on the PPU to activate the settings.





## Step 2

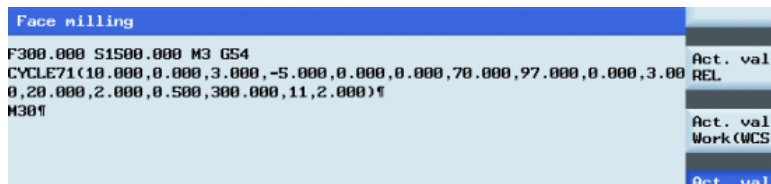
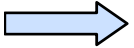
Press "Face cutt." SK on PPU.



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.



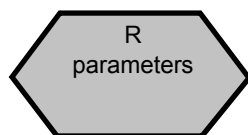
Make sure that the override value on the MCP is 0%!

Press the "Cycle Start" key on the MCP.



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

## SEQUENCE



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

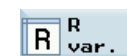
Programming the STOPRE command in a block will stop block preprocessing and buffering. The following block is not executed until all preprocessed and saved blocks have been executed in full. The preceding block is stopped in exact stop (as with G9).

Below shows the connection between program and “R variables” status windows

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

WCS	Position	Repos offset
X	-10.000	0.000 mm
Y	0.000	0.000 mm
Z	10.000	0.000 mm

R variables	
R0	0.000000
R1	0.000000
R2	0.000000
R3	0.000000
R4	0.000000
R5	0.000000

R variables	
R0	0.000000
R1	1.000000
R2	0.000000
R3	0.000000
R4	0.000000
R5	0.000000

R variables	
R0	0.000000
R1	1.000000
R2	2.000000
R3	0.000000
R4	0.000000
R5	0.000000

R variables	
R0	0.000000
R1	1.000000
R2	2.000000
R3	3.000000
R4	0.000000
R5	0.000000

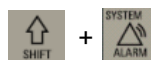
WCS	Position	Repos offset
X	3.000	0.000 mm
Y	0.000	0.000 mm
Z	10.000	0.000 mm

## SEQUENCE



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

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



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

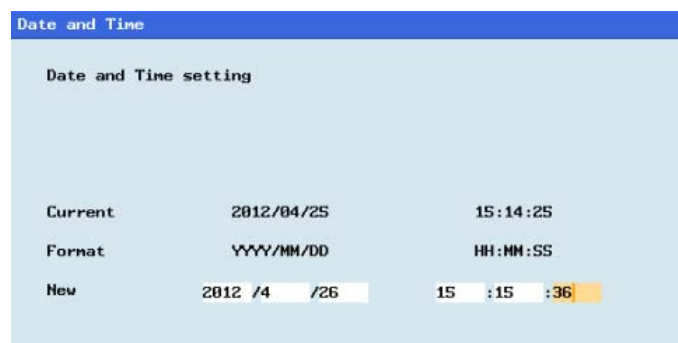
Press the “HMI” SK on the PPU.



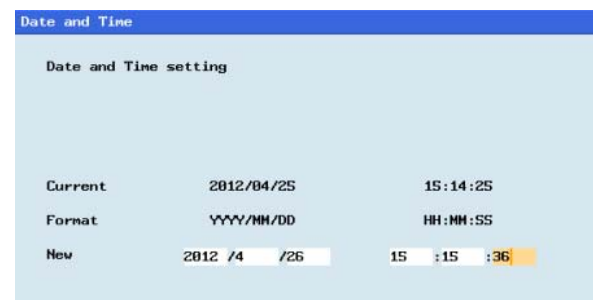
Press the “Date time” SK on the PPU.



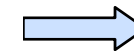
Enter a new “Date” and “Time”.



Press the “OK” SK on the PPU.



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



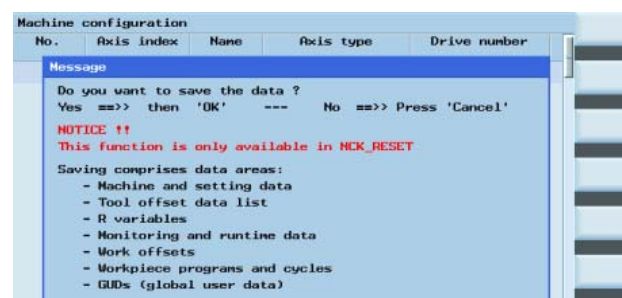
“Save data” enables the complete system to be backed up on the system CF card so that there is a system backup available to the operator.

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

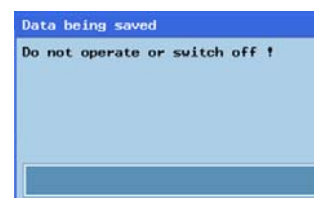
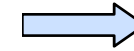


Make sure the password is set to the “CUSTOMER” access level.

Press the “Save data” SK on the PPU.



Press the “OK” SK on the PPU.



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

**SEQUENCE**

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

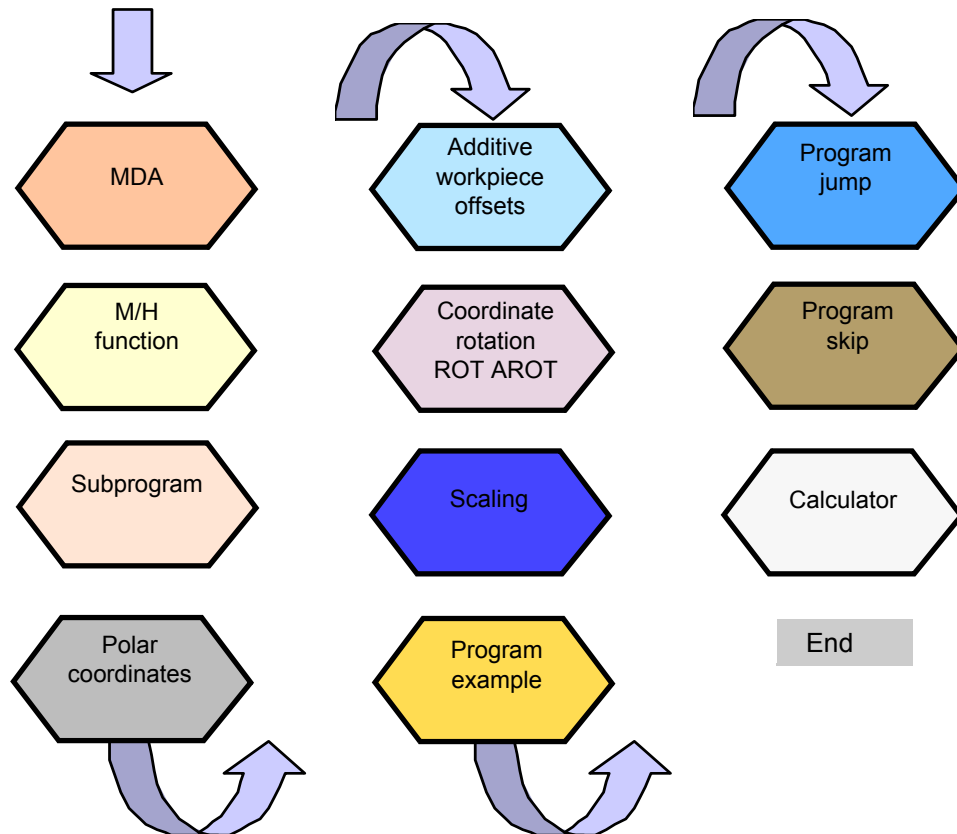
If the operator is manually selecting the gear stage in the part program, it is the operator's responsibility to select the correct gear stage according to the required speed.

## Additional Information Part 2

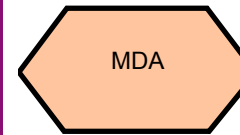
## Description

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.

## Content



## SEQUENCE



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

Use MDA to move the axis to a fixed position. →

Press the “Machine” key on the PPU. →



Press the “MDA” key on the PPU. →



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.

10:44:36  
2012/04/28

SIEMENS

Stop SKP DRV ROV MD1 PRT SBL

MCS	Position	Dist-to-go	T,F,S
X	10.000	0.000 mm	T 1 D 1
Y	10.000	0.000 mm	F 0.000 100% 10176.225 mm/min
Z	50.000	0.000 mm	S1 0.0 100% 0.0 0
G00	G54	G60	

MDI - Block

G0 G54 X10 Y10 Z50  
==eof==

G  
function

Auxiliary  
function

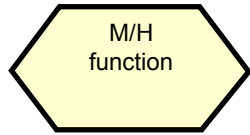
Axis  
feedrate

Save  
file

Delete  
file

Act. val.  
REL

## SEQUENCE

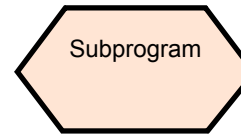


The M function initiates switching operations, such as "Coolant ON/OFF". Various M functions have already been assigned a fixed functionality by the CNC manufacturer. The M functions not yet assigned are reserved for free use of the machine tool manufacturer.

With H functions, the meaning of the values of a specific H function is defined by the machine tool manufacturer.

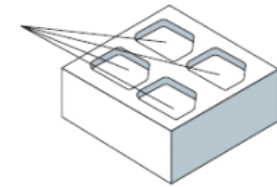
M codes and H functions created by the OEM should be backed up by the machine tool manufacturer.

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		



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.

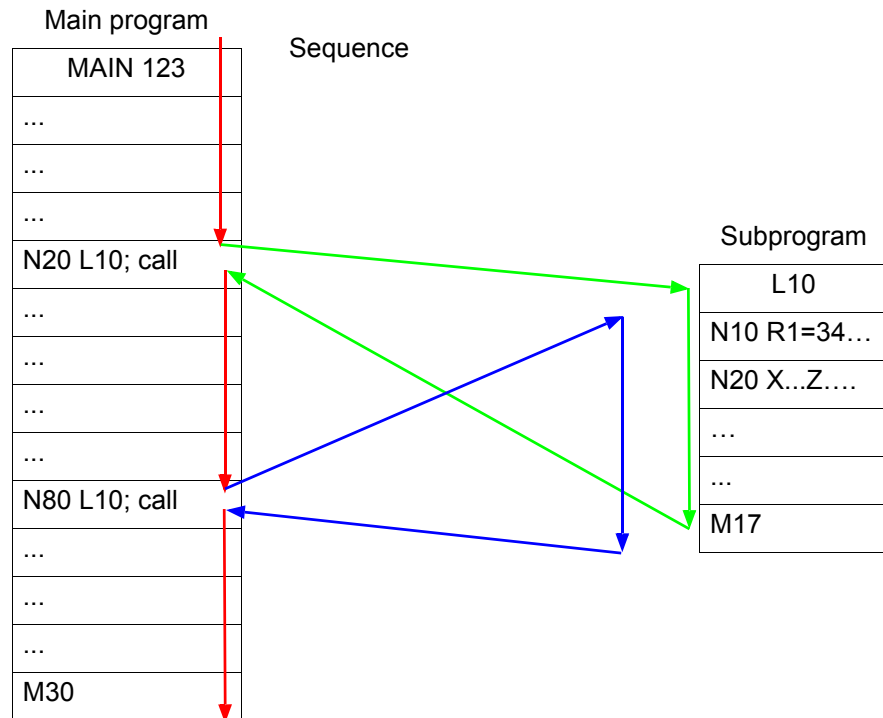
The subprogram should be given a unique name enabling it to be selected from several subprograms. When you create the program, the program name may be freely selected.

However, the following rule should be observed:

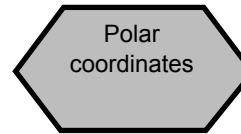
The name can contain letters, numbers and underscores and should be between 2 and 8 characters long.

Example: LRAHMEN7

## 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.



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.



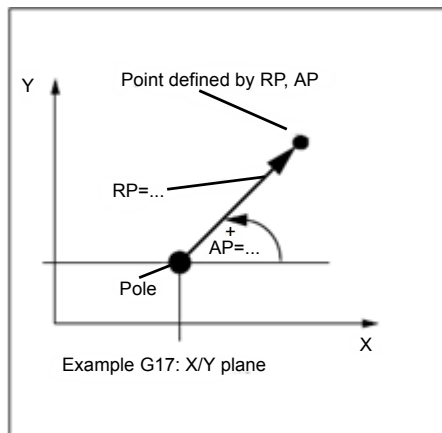
**Basic Theory**

- 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.

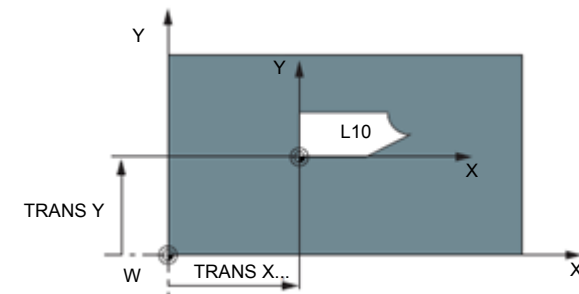
```

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
  
```



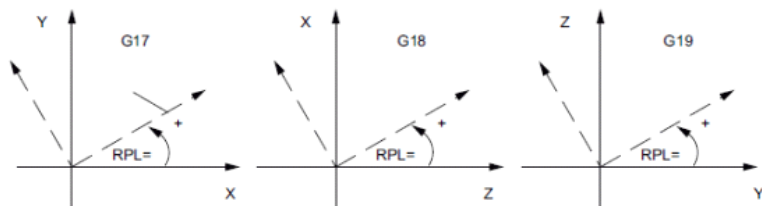
## SEQUENCE

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

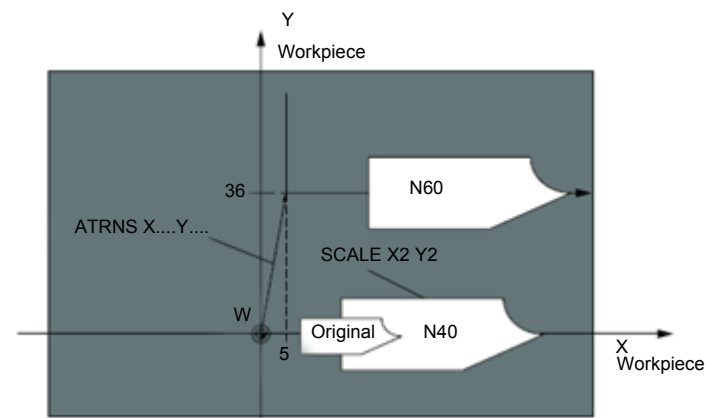
A scale factor can be programmed for all axes with SCALE, ASCALE. The path is enlarged or reduced by this factor in the specified axis. The currently set coordinate system is used as the reference for the scale change.

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

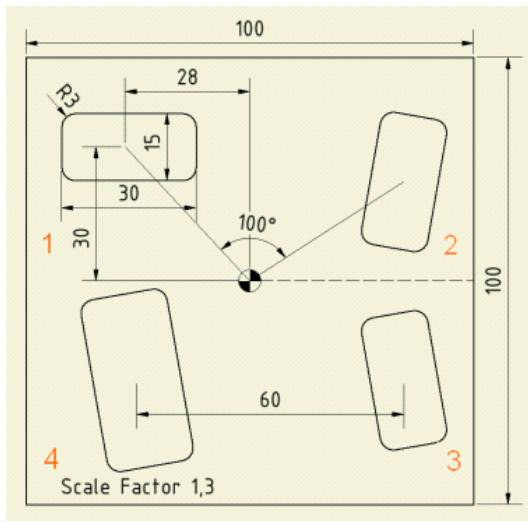


## SEQUENCE

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 100°

Drawing 3 — ① Drawing 2 along X axis mirror image

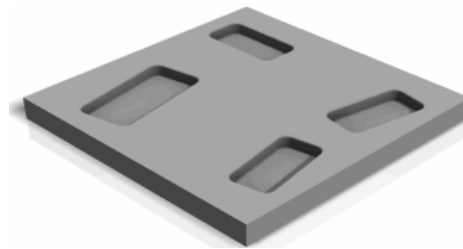
② Coordinate rotates 20°

Drawing 4 — ① Drawing 3 along Y axis moves 60 in negative direction

② enlarge 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!



<b>N10</b>	SUPA G00 Z300 D0	<b>N10</b>	SUPA→cancel all settable offsets
<b>N15</b>	SUPA G00 X0 Y0	<b>N15</b>	
<b>N20</b>	G17 T1 D1	<b>N20</b>	coordinate plane G17, use tool 1
<b>N25</b>	MSG ("change to 1 tool")	<b>N25</b>	
<b>N30</b>	M5 M9 M00	<b>N30</b>	
<b>N35</b>	S5000 M3 G94 F300	<b>N35</b>	
<b>N40</b>	G00 X-28 Y 30	<b>N40</b>	
<b>N45</b>	G00 Z2	<b>N45</b>	
<b>N50</b>	LAB1:	<b>N50</b>	LAB1:milling start sign
<b>N65</b>	POCKET3( 50, 0, 2, -5, 30, 15, 3, -28, 30, 0, 5, 0, 0, 300, 100, 0, 11, 5, , , 5, 3, )	<b>N65</b>	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 0°)
<b>N70</b>	LAB2:	<b>N70</b>	LAB2:milling groove end sign
<b>N75</b>	M01	<b>N75</b>	
<b>N80</b>	ROT RPL=-100	<b>N80</b>	coordinate axis rotates 100° in positive direction
<b>N85</b>	REPEAT LAB1 LAB2 P1	<b>N85</b>	machining the same groove at the new position
<b>N90</b>	M01	<b>N90</b>	
<b>N95</b>	AMIRROR X=1	<b>N95</b>	along the new X axis to change the mirror image
<b>N100</b>	AROT RPL=-20	<b>N100</b>	coordinate axis rotates -20° in positive direction
<b>N105</b>	M01	<b>N105</b>	
<b>N110</b>	REPEAT LAB1 LAB2 P1	<b>N110</b>	machining the same groove at the new position
<b>N115</b>	AROT RPL=10	<b>N115</b>	coordinate axis rotates -10° in negative direction
<b>N120</b>	ATrans Y-60	<b>N120</b>	Y axis coordinate moves 60 in negative direction
<b>N125</b>	AROT RPL=-10	<b>N125</b>	
<b>N130</b>	ASCALE X1.3 Y1.3	<b>N130</b>	groove enlarged 1.3 times in the X,Y direction.
<b>N135</b>	REPEAT LAB1 LAB2 P1	<b>N135</b>	machining the same groove at the new position
<b>N140</b>	M30	<b>N140</b>	end

## SEQUENCE



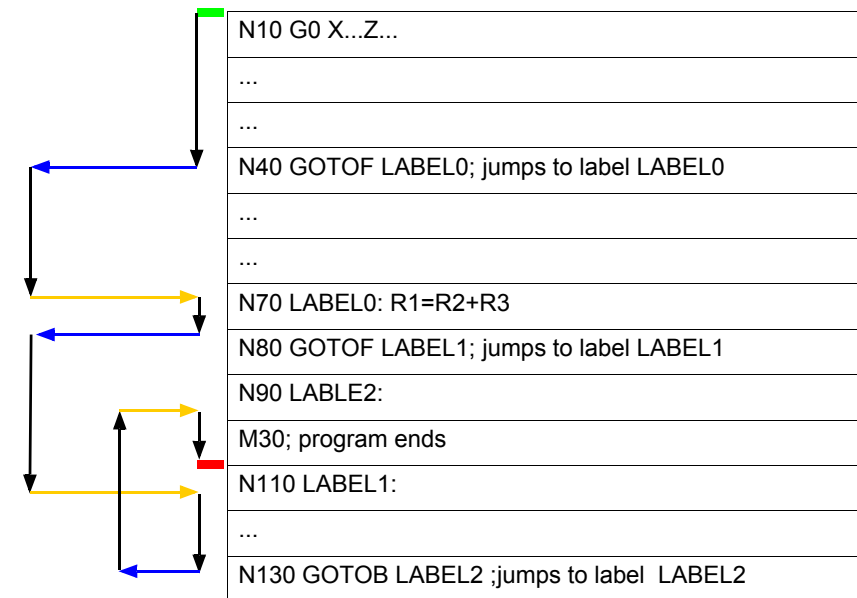
NC programs process their blocks in the sequence in which they were arranged when they were written. The processing sequence can be changed by introducing program jumps. The jump destination can be a block with a label or with a block number. This block must be located within the program. The unconditional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

## SEQUENCE



## 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.

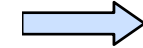
Using “;” code to add a remark to the N85 function, without any influence on the execution.

## Method 2

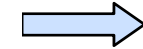
Press the “Machine” key on the PPU.



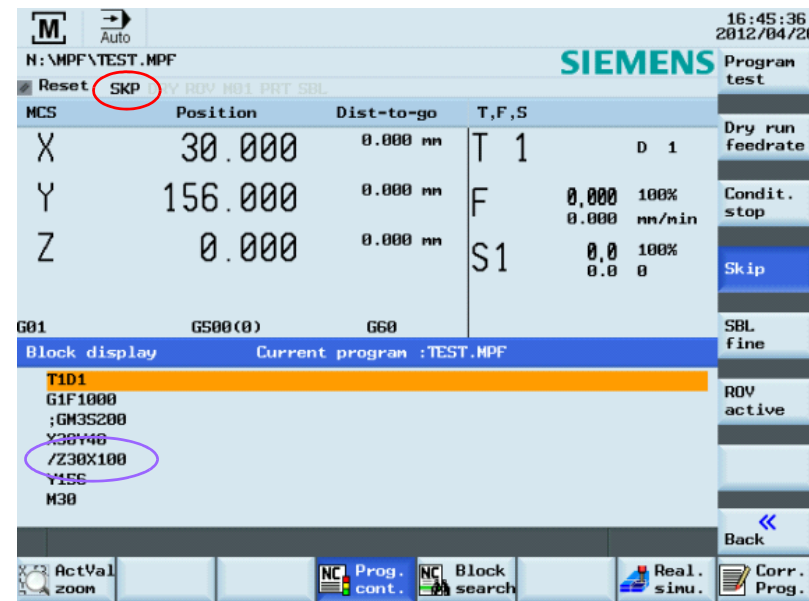
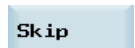
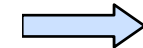
Press the “Auto” key on the MCP.



Press the “Prog cont.” SK on the PPU.



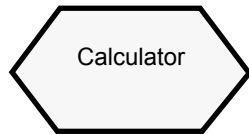
Press the “Prog cont.” SK on the PPU.



When “SKP” is displayed (red circle), the skip function has been activated.

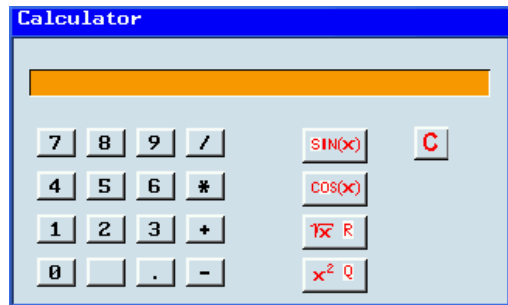
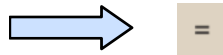
After activating “SKP”, using “/” at the beginning of the program string (shown in purple circle), the string will be skipped without influencing the execution.

## SEQUENCE



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.



Press this SK to delete the contents in the calculator.



Press this SK to exit the calculator screen.



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

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

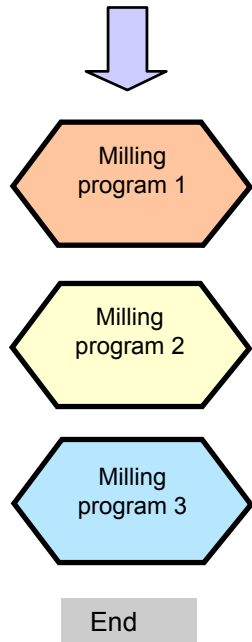
Use the “Accept” SK to enter the result in the input field at the current cursor position of the part program editor. The calculator will then close automatically.

## Sample Program

### Description

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

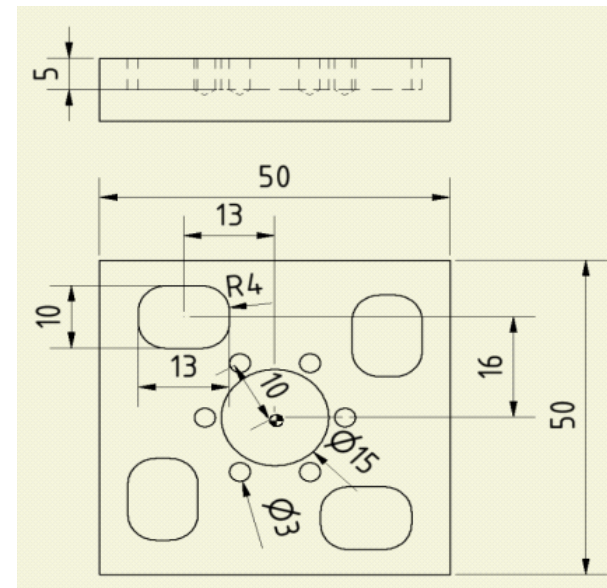
### Content



### Drawing



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

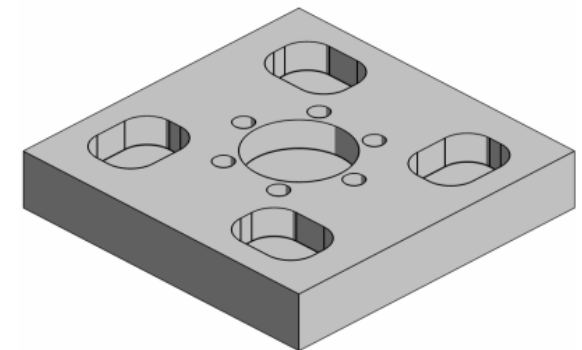


The zero point of the workpiece is located at the center point of the workpiece

### Tool information:

T1 Milling tool D50

T2 Milling tool D8





## 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 90°
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 90°
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=====

```

## 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** 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

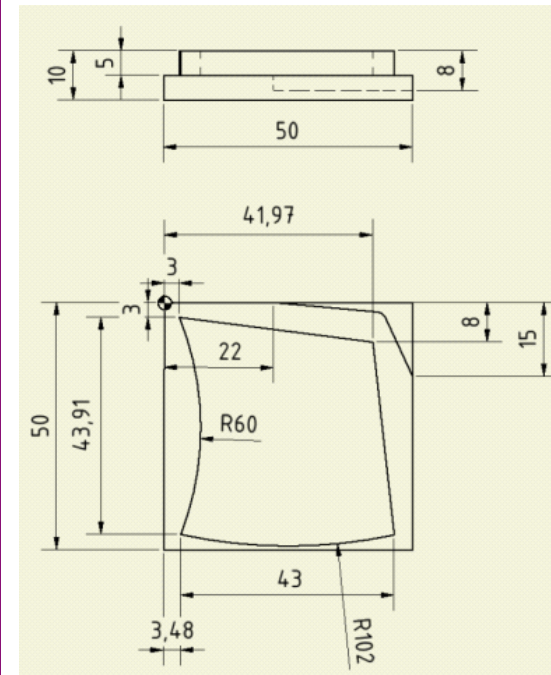
**N390** circular line hole forms cycle  
 command(circular center point coordinate(X0,Y0),  
 radius 10 mm, angle between the line with first hole  
 and circular center point and the X axis in positive  
 direction is 45°, angle between the holes is 60°,  
 circular hole number 6 →)

**N400** cancel mode use  
**N410** M30

## Drawing



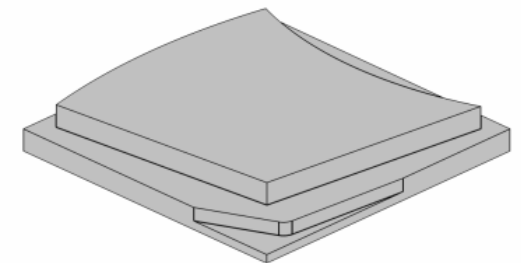
Make sure all the preparations and safety measures  
have been performed before machining!



Workpiece zero point  
is located in the top  
left corner.

## Tool information:

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



## 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( "SUB_PART_2", 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

```

SUB\_PART\_2.SPF

\*\*\*CONTOUR\*\*\*\*

```

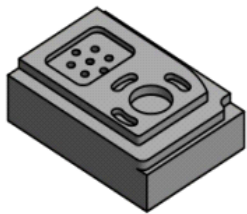
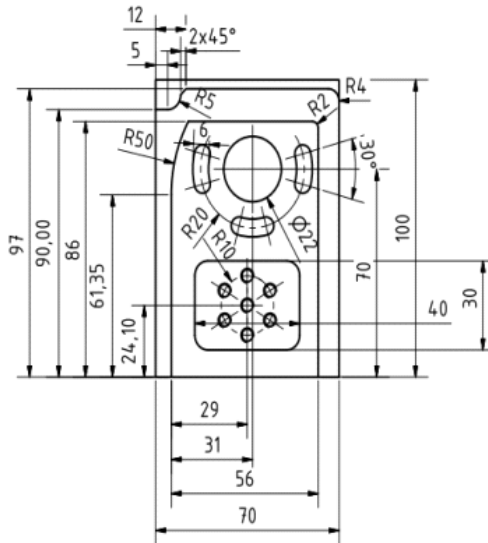
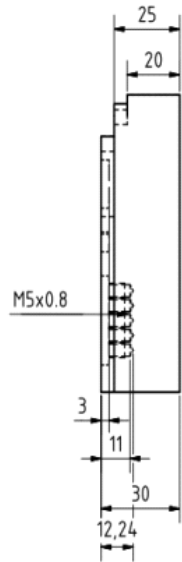
G17 G90
G0 X3 Y3
G2 X3.27 Y-40.91 I=AC(-52.703) J=AC(-19.298)
G3 X46.27 Y-47 I=AC(38.745) J=AC(54.722)
G1 X42 Y-8
X3 Y3
M2;/* end of contour */

```

## Drawing



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	T5 Milling tool D5
T2 Milling tool D12	T6 Drilling tool D10
T3 Milling tool D10	T7 Drilling tool D5
T4 Milling tool D16	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=====

```

## 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( "SUB_PART_3", 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==

```

## 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, 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→ rilling 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

```

**Machining Process**

SUB\_PART\_3.SPF

\*\*\*CONTOUR\*\*\*\*

G17 G90 DIAMOF

G0 X7 Y0

G1 Y61.35

G2 X13.499 Y86 I=AC(57) J=AC(61.35)

G1 X63 RND=2

Y0

M2;/\* end of contour \*/

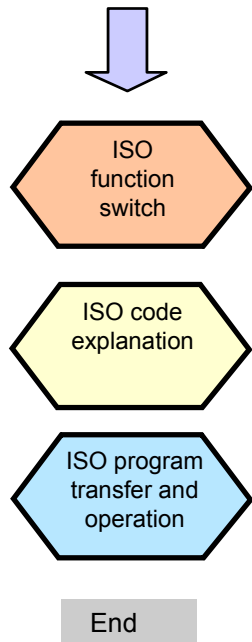


## ISO Mode

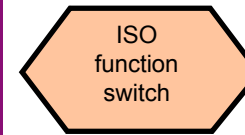
### Description

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. The examples in ISO mode chapter can be run in 808D ISO mode.

### Content



### Basic Theory



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

#### ISO function switch

##### Method 1

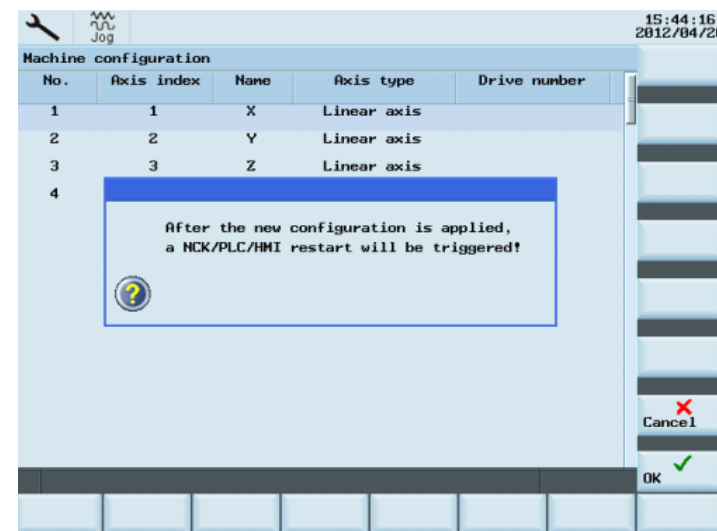
Press the “Shift” + “System - Alarm” keys on the PPU.  
Input the manufacturer’s password (“SUNRISE”)



Press the “ISO mode” SK on the right.



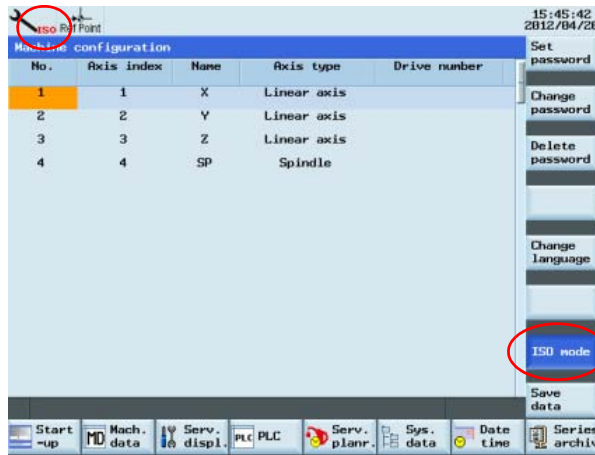
A dialog box appears prompting whether to activate the new setting. Select the “OK” SK to activate it.



## Basic Theory

After pressing “OK”, the system restarts automatically.

After restarting, press “Shift” + “System - Alarm” again and if the symbol in the red circle appears, ISO mode is already activated.



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

### 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 F1
N20 T1 H1
N25 MSG("Tool No 1 in use")
N30 S4000 M3
N40 CYCLE71( 50.00000, 2.00000, 2.00000, 0.00000, 0.00000, 0.00000
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**

**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

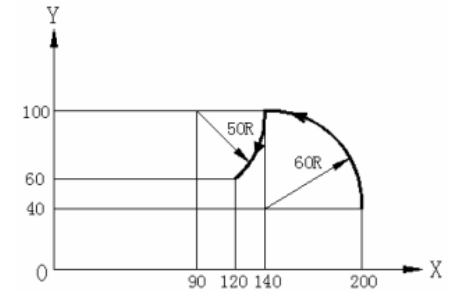
N5 **G90 T1 M06**  
N10 **M3 S2000**; spindle rotation  
N20 **G99 G81 X300 Y-250 Z-150**  
R-10 F120; **after orientation drilling, back to R point**  
N30 **X1000.** ; **after orientation drilling, back to R point**  
N40 **G04 X2.0** ; **delay 2 s**  
N50 **G98 Y-550** ; **after orientation drilling, 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**

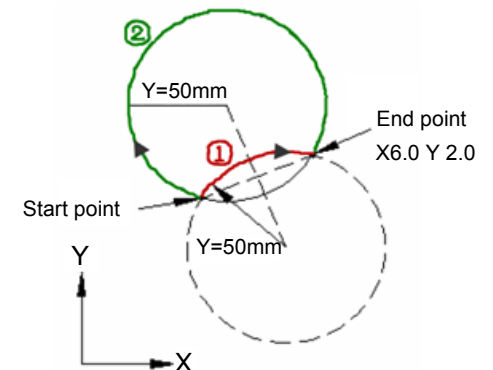
When specifying circle radii with parameter R

Circles less than 180° are assigned positive values

① → **G02 X6.0 Y2.0 R50.0**

Circles greater than 180° are assigned negative values

② → **G02 X6.0 Y2.0 R-50.0**



## Basic Theory

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

P.	Descriptions	Unit	Applied range and note
X/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

**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

**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** 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 point

**X1000** ;after orientation drill 4th hole, back to R point

**Y-550** ;after orientation drill 5th hole, back to R point

**G98 Y750** ;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**

## Basic Theory

### 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 as 1st 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

### 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

**G28 G91 X0 Y0 Z0** ;back to reference point

**M5** ;spindle rotation stop

**M30**

### 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

**M3**

### 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**

**Basic Theory****G84** Tapping cycle

Common programming structures:

**G84** X—Y—Z—R—P—F—K

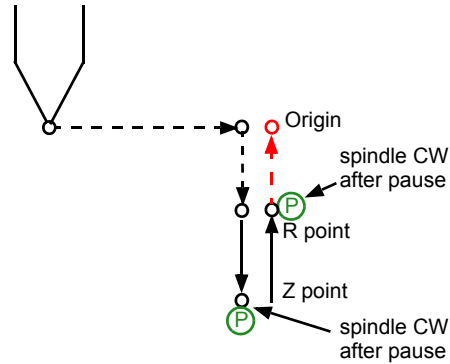
Motion process:

- 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

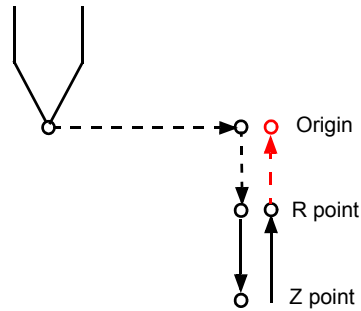
Motion process:

- 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

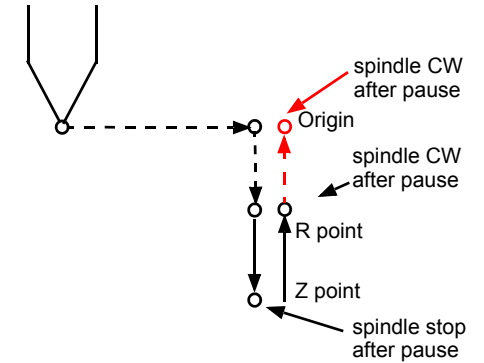
Motion process:

- 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

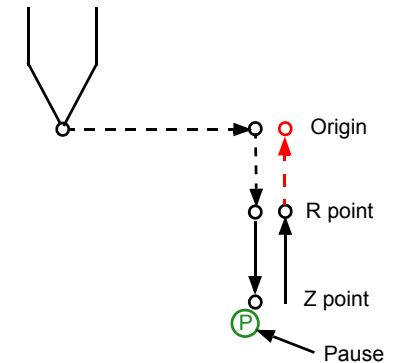
Motion process:

- 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**

## Basic Theory

**G87** Boring cycle I / reverse boring cycle II

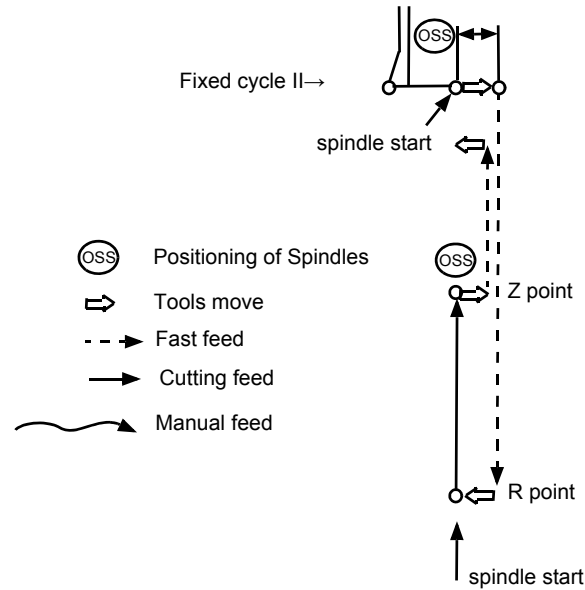
Common programming structures:

**G87** X—Y—Z—R—Q—P—F—L

Motion process:

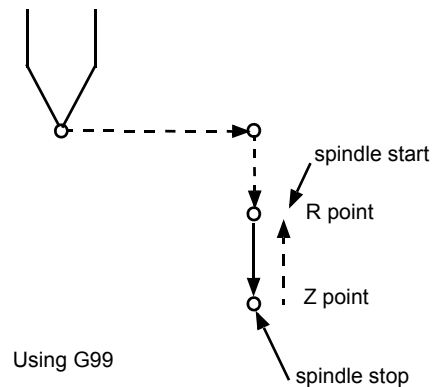
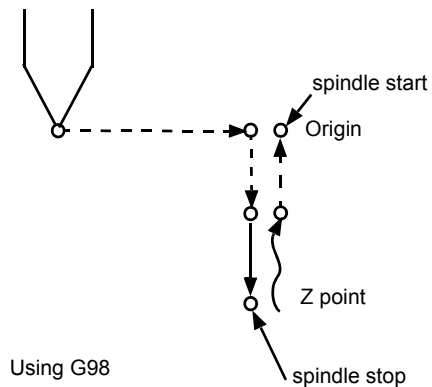
- ① Drilling motion (-Z) → cutting feed
- ② Motion at the bottom of the hole → spindle stops
- ③ Retraction motion (+Z) → manual operation or fast feed

**G87** execution operation graphic:



**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.



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



Press the "Copy" SK on the PPU.



Press the "NC" SK on the PPU.



Press the "Paste" SK on the PPU.



A specified ISO program is then stored in the 808D 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!

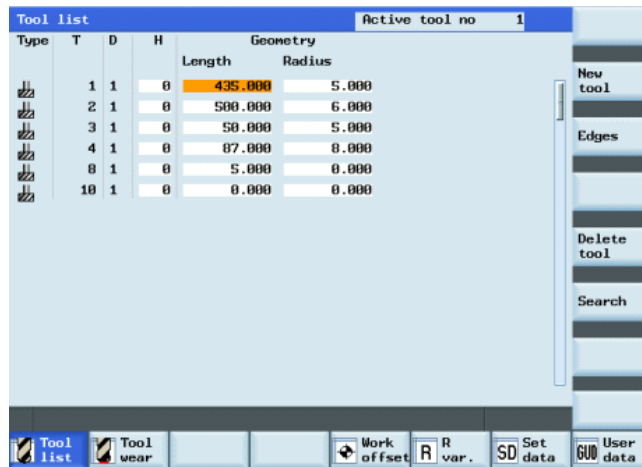


## Basic Theory

## 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;	<b>O0001;Delete this line</b>
G0 X50 Y50 Z50 M5	G0 X50 Y50 Z50 M5
G04 X5	G04 X5
M3 S1000	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.

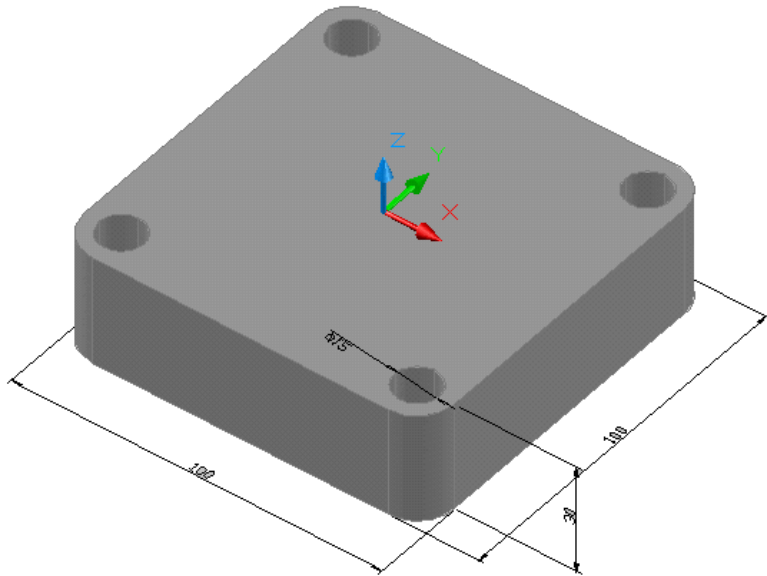
## Basic Theory

## 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	N210 T2M6
N20 T1M6	N220 M3S3000F100
N30 G0G54G90G40	N230 G43H2Z50
N40 M3S1200F200	N240 G0X40Y-40
N50 G43H1Z50	N250 Z20
N60 G0X0Y-70	N260 G81Z-2R10
N70 Z5M8	N270 Y40
N80 G1Z-5	N290 X-40
N90 G01G41X20D1	N300 Y-40
N100 G03X0Y-50R20	N310 G80
N120 G1X-50,R10	N320 G0Z50
N130 Y50,R10	
N140 X50,R10	N330 T3M6
N150 Y-50,R10	N340 M3S3000F100
N160 X40	N350 G43H3Z50
N170 X0	N360 G73Z-20R10Q5
N180 G03X-20Y-70R20	N370 Y40
N190 G1G40X0	N380 Y-40
N200 G0Z50	N390 X40
	N400 Y40
	N410 G80
	N420 G0G40G90G49Z100
	N430 M09
	N440 G290
	N450 M30

**Note:** This program opens/exits ISO mode with the G291/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)

**Basic Theory**

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( "SUB_PART_4", 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

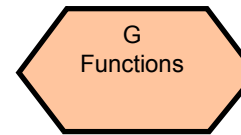
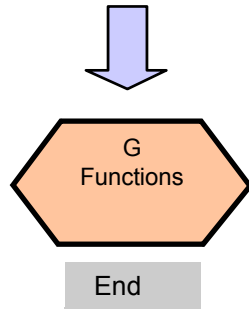
N160 G0G40G90Z60
N170 M09M05
N180 M30

; SUB_PART_4.SPF

G17 G90 DIAMOF
G0 X0 Y-50
G1 X-50 RND=10
Y50 RND=10
X50 RND=10
Y-50 RND=10
X0
M2;/* end of contour */
```

## Appendix

## Content

**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

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

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

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



## 7 M - Code list, M functions

### 7.1 M-function for milling machines according to PAL

No.	Function
M00	Programmed stop
M01	Selected stop
M02	Program end
M03	Starting the milling spindle in positive direction (clockwise)
M04	Starting the milling spindle in negative direction (anti-clockwise)
M05	Stopping the milling spindle
M06	Automatic tool change
M07	
M08	Cooling lubricant pump ON
M09	Cooling lubricant pump OFF
M10	
M11	
M12	
M13	
M14	
M15	
M16	
M17	Return from subroutine
M18	
M19	Spindle stop in defined limit position
M20	Program end with resetting (Reset)
M21	
M22	
M23	
M24	
M25	
M26	Clamp workpiece
M27	Loosen workpiece
M28	
M29	
M30	

No.	Function
M31	Opening a lock
M32	
M33	
M34	
M35	
M36	
M37	
M38	
M39	
M40	
M41	
M42	
M43	
M44	
M45	
M46	
M47	
M48	
M49	
M50	
M51	
M52	
M53	
M54	
M55	
M56	
M57	
M58	
M59	
M60	Workpiece change
M61	
M62	
M63	





No.	Function
M64	
M65	
M66	
M67	
M68	
M69	
M70	
M71	
M72	
M73	
M74	
M75	
M76	
M77	
M78	
M79	
M80	
M81	
M82	
M83	
M84	
M85	
M86	
M87	
M88	
M89	
M90	
M91	
M92	
M93	
M94	
M95	
M96	
M97	
M98	

No.	Function
M99	



## 7.2 G functions to PAL

G0	Travelling in rapid traverse
G1	Linear interpolation during process
G2	Circular interpolation clockwise
G3	Circular interpolation anti-clockwise
G4	Retention period
G9	Accurate stop
G10	Travelling in rapid feed in polar coordinates
G11	Linear interpolation with polar coordinates
G12	Circular interpolation clockwise with polar coordinates
G13	Circular interpolation anti-clockwise with polar coordinates
G45	Linear tangential travelling on a contour
G46	Linear tangential travelling off the contour
G64	Accurate stop off



## 8 Notes, messages and error messages

All messages and alarms are displayed in plain text on the control panel. The alarm text contains the date, time and a suitable symbol for the cancel criterion.

Alarms and messages are displayed separately according to the following criteria:

- Alarms and messages in the part program.
- Alarms and messages from the PLC and alarms and messages that concern the machine.

The description of the alarms and messages in the part program and other alarms and messages from the PLC are in the Siemens manual.

Number	Message
700000	User alarm 01
700001	The manual mode of MGZ is active
700002	Air pressure is not enough (bigger 0.6 Mpa)
700003	The door is open
700004	The power sequence is wrong(1L\2L\3L, change
700005	User alarm 06
700006	User alarm 07
700007	User alarm 08
700008	User alarm 09
700009	User alarm 10
700010	HHU is active
700011	Not able to lock tool in expected time
700012	Spindle in braking progress
700013	Operation while chuck is not closed
700014	Gear-change time out
700015	Gear level position error
700016	DRIVES NOT READY
700017	Operation chuck when sp. or part prog. is running
700018	COOLING MOTOR OVERLOAD
700019	COOLANT LIQUID POSITION IN LOW LEVEL
700020	LUBRICATING MOTOR OVERLOAD
700021	LUBRICANT LIQUID POSITION IN LOW LEVEL
700022	TURRET MOTOR OVERLOAD
700023	PROGRAMMED TOOL NUM. > MAX. TURRET NUMBER
700024	Max. tool number setting error
700025	NO POSITION SIGNALS FROM TURRET
700026	Not able to find expected tool in monitor time
700027	APPROACH REF.POINT AGAIN AFTER ROT. MONITORING



Number	Message
700028	Tool is not locked
700029	Reminding information for 1st service plan
700030	Alarm for 1st service plan
700031	Magazin not in spindle pos. or original pos.
700032	Magazin in spindle pos. and original pos.
700033	Magazine turn key when magazine or sp. not ready
700034	Block search, tool in spindle <> programmed tool
700035	Spindle not reach tool-release pos. in time
700036	Spindle not reach tool-lock pos. in time
700037	Do not move MGZ when Z axis under the tool change pos.
700038	User alarm 39
700039	Turn magazine when alarm or tool not retracted
700040	Start ATC when Z axis not in tool change pos.
700041	Move Z axis when ATC not in original position
700042	ATC not finish action in monitor time
700043	Change tool when magazine not in run mode
700044	Magazine motor overload
700045	ATC motor overload
700046	User alarm 47
700047	User alarm 48
700048	User alarm 49
700049	Reference point x-axis not reached
700050	Reference point z-axis not reached
700051	Wrong spindle direction started
700052	Watchdog timer JOG-program
700053	Spindle override not 100%
700054	Spindle is not started
700055	Feed override =0%
700056	Change of spindle direction not possible in thread
700057	User alarm 58
700058	User alarm 59
700059	safety door not closed, NC start not possible
700060	Channel not in reset, change PRT not possible
700061	User alarm 62
700062	User alarm 63



Number	Message
700063	User alarm 64
700064	User alarm 65
700065	User alarm 66
700066	User alarm 67
700067	User alarm 68
700068	User alarm 69



## 9 SINUMERIK 808 D

The Sinumerik 808D complete documentation consists of the manuals listed below, which are accompanying the machine separately.

**Manuals are strictly necessary for the operator, CNC programmer and also for the maintenance and repair personnel of the F3.**

- Programming and operating manual (milling)
  - Part 1 Milling operation
  - Part 2 Programming milling
  - Part 3 Programming milling in ISO code

**Manuals that are required for the maintenance and repair personnel.**

- Service manual
- Diagnostic manual
- Function manual
- Commissioning manual

**Additional useful manuals and information**

- Training Manual Operation and Programming Milling
- PLC Manual subroutines
  - Describes e.g. making a connection to the RS232 interface.
- Online help for programming and operation (milling)

**Manuals that are required to install the control and the components on a machine:**

- Mechanical installation manual
- Electrical installation
- Parameter Manual

All manuals are in PDF format - even in other languages - can be downloaded from the Siemens website. Date of issue of this manual, the Programming milling manual are currently only available in German, English, Russian, Portuguese and Chinese language.

<http://support.automation.siemens.com>

**For any questions regarding the CNC control, please contact:**

**Siemens AG, A&D techsupport**

**Phone (+49) 0180 50 50 222**

**mailto: techsupport@ad.siemens.de**

**Siemens AG Hotline, Helpline**

**Phone (+49) 0180 50 50 111**



## 10 Maintenance

In this chapter you will find important information about

- Inspection
- Maintenance
- Repair

of the CNC machine.

### ATTENTION!

**Properly performed regular maintenance is an essential prerequisite for**

- **operational safety,**
- **failure-free operation,**
- **long service life of the CNC machine and**
- **the quality of the products which you manufacture.**

Installations and equipment from other manufacturers must also be in good order and condition.



### ENVIRONMENTAL PROTECTION

**During work on the cooling lubricant equipment please make sure that**

- **collecting containers with sufficient capacity for the amount of liquid to be collected are used.**
- **Liquids and oils should not be split on the ground.**

Clean up any spilt liquid or oils immediately using proper oil-absorption methods and dispose of them in accordance with current environmental protection regulations.



### Collect leakages

Do not re-introduce liquids spilt outside the system during repair or as a result of leakage from the reserve tank; collect them in a collecting container for disposal.

### Disposal

Never dump oil or other environmentally hazardous substances which are harmful to the environment in water inlets, rivers or channels. Used oils must be delivered to a collection centre. Please consult your supervisor for further information on your nearest collection point.

## 10.1 Operating material

### 10.1.1 Machine lubricants

Only use appropriate lubricants which guarantee a safe operation of the machine.

Recommended lubricant class: ISO VG68

Recommended lubricant: guideway oil Mobil Vactra (Oil No. 2)

### 10.1.2 Cooling lubricants

In order to avoid interferences during operation the water-mixed cooling lubricant and the slide-way oil or grease need to be compatible.

### INFORMATION

The CNC milling machine is lacquered with a **one-component paint**. Take this criterion into account when selecting your cooling lubricant.

The company Optimum Maschinen Germany GmbH does not assume any guarantee for subsequent damages due to unsuitable cooling lubricants.

The flashpoint of the emulsion must be higher than 140°C.







When using non-water-miscible cooling lubricants (oil content > 15%) with a flashpoint, ignitable aerosol air mixtures might develop. There is a potential danger of explosion.

#### ATTENTION!

**Only the correct selection of an appropriate combination of cooling lubricants and slideway oils as well as the proper care and maintenance of the cooling lubricant can ensure that no problems such as stick-slip effects or deposits are resulting.**

The selection of cooling lubricants and slideway oils, lubricating oils or greases as well as their care are being determined by the machine operator or operating company.

Therefore, Optimum Maschinen Germany GmbH cannot be held liable for machine damages caused by unsuitable coolants and lubricants as well as by inadequate maintenance and servicing of the coolant. In case of problems with the cooling lubricant and the slideway oil or grease, please contact your mineral oil supplier.



#### ATTENTION!

**For safe functioning of the CNC-machine the cooling lubricant needs to be checked at least weekly also during down times with regard to its concentration, ph-value, bacteria and fungal decay.**



#### ATTENTION!

**Cooling lubricants and oils for the mechanic and pneumatic system of the machine have to be adjusted referring the water added initially, the cooling lubricant emulsion and the machining task.**



We would like to ask you to have the following machine-related properties of the cooling lubricant confirmed in writing by the manufacturer of the cooling lubricant.

- The products must comply with the provisions of the current statutory regulations and the employers' liability insurance association.
- Request documentation for the products such as the product description VKIS and EC safety data sheet from the cooling lubricants manufacturer. The EC safety data sheet gives you information about the water hazard class.

They need to be environmentally friendly and workplace-friendly. Thus, they need to be free of nitrite, PCB, chlorine and nitrosatable diethanolamine (DEA), according to TRGS 611.

- The manufacturer should be able to provide a certificate concerning skin tolerance.
- The mineral oil content according to DIN 51417 should be at least 40% in the concentrate.
- If possible, it should be universally applicable for all chippings and materials.
- Long service life of the emulsion e.g. long-term stable and resistant to bacteria.
- Safe corrosion protection according to DIN 51360/2.
- Re-emulsifiable and non-adhesive according to VKIS sheet 9: Adhesive and residue behaviour.
- It should not attack the varnish of the machine according to VDI 3035.
- It should not attack any machine elements (metals, elastomers).
- Low foaming behaviour of the emulsion.
- It should be as finely dispersed as possible in order to avoid clogging in the needle slot screen.

## 10.2 Safety

#### WARNING!

**The consequences of incorrect maintenance and repair work may include:**

- **Severe injuries of persons working on the CNC machine,**
- **Damage to the CNC machine.**





**Only qualified personnel should carry out maintenance and repair work on the CNC machine.**

## **Validation**

Check and maintain all safety-relevant stop, control and measuring devices (validation).

## **Documentation**

Record all tests and works in a operator's log resp. log book.

### **10.2.1 Preparation**

#### **WARNING!**

**Only carry out work on the CNC machine, if the main switch is switched off and secured against restarting by means of a padlock.**

☞ "Disconnecting and securing the CNC machine" on page 17

Attach a warning sign.



### **10.2.2 Restarting**

Before restarting, run a safety check.

☞ "Safety check" on page 14

#### **WARNING!**

**Before starting the CNC machine, you must check that there is no danger for persons and that the CNC machine is not damaged.**





## 10.3 Inspection and maintenance

The type and level of wear depends to a large extent on the individual usage and operating conditions. Any indicated intervals therefore are only valid for the corresponding approved conditions.

Interval	Where?	What?	How?	Check ✓
Start of work, after each maintenance or repair work	CNC milling machine	☞ "Safety check" on page 14		
	Sight window	Cleaning	Clean the polycarbonate sight windows using a suitable cleaning agent.	
	Compressed air supply	Draining	☞ "Drain the compressed air service unit" on page 166	
Start of work, Daily after every maintenance or repair work	Lubricant tank Slideway oil	Fill level control	Manual  Check the quantity and refill the lubricant tank of the central lubrication system. See also ☞ "Cooling lubricants" on page 160 3.5 "Refill central lubrication system" on page 33	
	Oil collecting tank	Empty	Check the oil level in the oil collecting gutter. Empty the oil collecting tank.	
Every week		Oiling Lubricate	Oil all bare steel surfaces. Use an acid-free oil, e.g. weapon oil or motor oil.	



Interval	Where?	What?	How?	Check ✓
Every week	Cooling lubricant tank	Fill level status control	Check for liquid level, concentration, pH value, bacteria and fungal decay.	
Weekly	Cooling lubricant	Measure pH value	Check the pH value. If required, replace the cooling lubricant.	
Every week	Drip feed lubricator Compressed air cylinder	Fill level control	Check the liquid level and refill if required.	
Every month	Gear belt Spindle head	Check Readjusting Replacing	Check the drive belt on wear and excessive clearance. If required, readjust the gear belt and/or replace it.	
every six months	central lubrication system	Check Replace	The suction filter should be cleaned regularly and replaced if necessary. Spare parts  "Zentralschmierung" on page 178 "Refill central lubrication system" on page 33	
Annually	Clutch Coupling Drive Z-, Y-, X-axis	Check Replace	Check if coupling is worn and check coupling slack. check If necessary, replace the coupling.	



Interval	Where?	What?	How?	Check ✓
Every 1000 operating hours	Wipers on the slides	Check Replace	Check the wipers on the slides. Replace if damaged immediately.	
	Electrical cabinet	Cleaning	☞ "Cleaning the electrical cabinet" on page 166	
	Servo motors	Inspection	Connections on the servo motors.	
As required,	Chip collection tray	Cleaning	Clean the chip collection tray of the cooling lubricant equipment.	
60 months	Protective covers	Replacing the sight window	☞ "Cleaning and replacing of the polycarbonate windows" on page 165	

## 10.3.1 Cleaning and replacing of the polycarbonate windows

### WARNING!

The polycarbonate sight windows are part of the safety device on your CNC machine.

**Damaged, scratched or even broken polycarbonate sight windows must be replaced immediately.**

A soft cloth should be used to clean the machine safety glass. The following detergents have been tested and approved by us:

Hahnerol glass cleaner (Hahnerol), Sidolin Streak Free (Henkel), Active Window Cleaner (Neumann).

We recommend that the polycarbonate sight windows in the door is changed 60 months after commissioning of the CNC machine.

In the following cases, an immediate replacement is strongly recommended:

- plastic deformation (distortion) by previous impact stress,
- cracks,
- Damage to the edge seal,
- immersion of lubricating coolant in the composite structure,
- destroyed or damaged safety screen (coating) on the workspace or operator's side.





## 10.3.2 Cleaning the electrical cabinet

Although the electrical cabinet is constructed to shut off external air, foreign particles such as dust and dirt may enter the cabinet when the door is open.

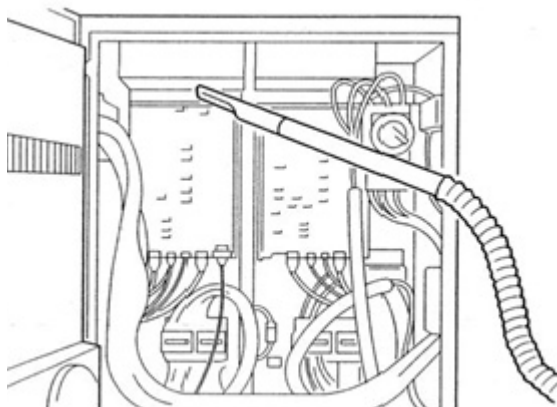
Accumulation of foreign particles on the printed circuit boards or other electronic components could cause machine malfunction.

Clean the inside of the electrical cabinet regularly.

Remove dust inside electrical cabinet with a vacuum cleaner. Do not use compressed air to clean the electrical cabinet.

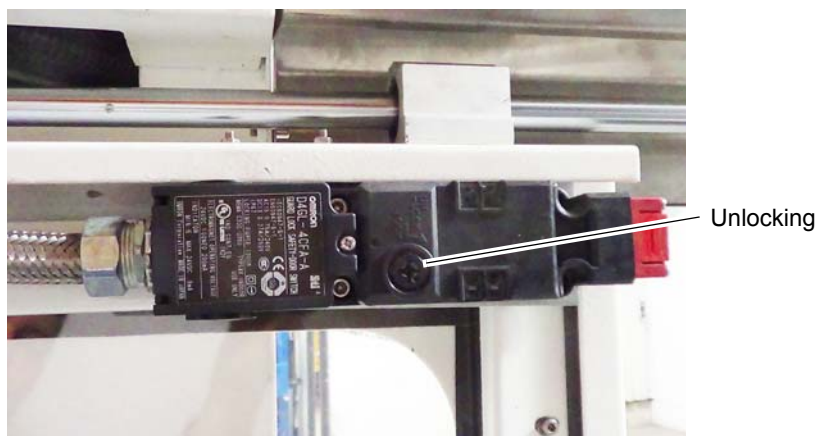
Never touch circuit boards or parts around the connector. Also avoid hitting these parts with the vacuum.

We recommend that the electrical cabinet is cleaned every 1000 operating hours .



## 10.4 Interlock switch sliding door

The interlock switch can be mechanically unlock for repair and maintenance purposes. Perform a safety check if you have set the interlock switch back to the ground state. ➡ "Safety check" on page 14



Img. 10-1: Interlock switch sliding door

### 10.4.1 Drain the compressed air service unit

The filtration of water and other impurities in the compressed air is automatic when compressed air flows through the maintenance unit.

If the water level of the filter housing exceeds the maximum limit, the water enters the pneumatic equipment and causes damage.

Check the water level daily and drain the water level in the filter if necessary.



## 10.5 Cooling lubricants and tanks

### CAUTION!

**The cooling lubricant can cause diseases. Avoid direct contact with cooling lubricant or parts covered in cooling lubricant.**



Cooling lubricant circuits and tanks for water-cooling lubricant mixtures must be completely emptied, cleaned and disinfected as needed, but at least once per year or every time the cooling lubricant is replaced.

If fine chips and other foreign matters are accumulated in the coolant tank, the machine can no longer be correctly supplied with coolant. Furthermore, the lifetime of the coolant pump is reduced.

When processing cast iron or similar materials generating fine chips, cleaning the coolant tank more often is recommended.

**The cooling lubricant must be replaced, the cooling lubricant circuit and tank emptied, cleaned and disinfected if**

- the pH value drops by more than 1 based on the value during initial filling. The maximum permissible pH value during initial filling is 9.3
- there is a perceivable change in the appearance, odour, floating oil or increase of the bacteria to more than 10/6/ml
- there is an increase in nitrite content to more than 20 ppm (mg/l) or nitrate content to more than 50 ppm (mg/l)
- there is an increase in the N-nitrosodiethanolamine (NDELA) to more than 5 ppm (mg/a)

### CAUTION!

**Comply with the manufacturer's specifications for mixture ratios, hazardous substances, e.g. system cleaners, including their permissible minimum use times.**



### CAUTION!

**Since the cooling lubricant escapes under high pressure, pumping out the coolant by using the existing cooling lubricant pump via a pressure hose into a suitable tank is not recommended.**



## ENVIRONMENTAL PROTECTION

**During work on the cooling lubricant equipment please make sure that**

- **collector tanks are used with sufficient capacity for the amount of liquid to be collected.**
- **liquids and oils should not be spilled on the ground.**



Clean up any spilled liquid or oils immediately using proper oil-absorption methods and dispose of them in accordance with current statutory environmental regulations.

### Collect leakages

Do not re-introduce liquids spilled outside the system during repair or as a result of leakage from the reserve tank, instead collect them in a collecting container for disposal.

### Disposal

Never dump oil or other substances which are harmful to the environment into water inlets, rivers or channels. Used oils must be delivered to a collection centre. Consult your supervisor if you do not know where the collection centre is.





## 10.5.1 Inspection plan for water-mixed cooling lubricants

Company: No.: Date: used cooling lubricant			
size to be checked	Inspection methods	Inspection intervals	Procedure and comment
noticeable changes	Appearance, odour	daily	Find and rectify causes, e.g. skim off oil, check filter, ventilate cooling lubricant system
pH value	Laboratory techniques electrometric with pH meter (DIN 51369) Local measurement method: with pH paper (Special indicators with suitable measuring range)	weekly <sup>1)</sup>	if pH value decreases > 0.5 based on initial filing: Measures in accordance manufacturer's recommendations > 1.0 based on initial filing: Replace cooling lubricant, clean cooling lubricant circulation system
Usage concentration	Manual refractometer	weekly <sup>1)</sup>	Method results in incorrect values with tramp oil content
Base reserve	Acid titration in accordance with Manufacturer's recommendation	as required	Method is independent of tramp oil content
Nitrite content	Test sticks method or laboratory method	weekly <sup>1)</sup>	> 20 mg/L nitrite: Replace cooling lubricant or part or inhibiting additives; otherwise NDELA (N-nitrosodiethanolamine) in the cooling lubricant system and in the air must be determined > 5 mg/L NDELA in the cooling lubricant system: Replacement, clean and disinfect cooling lubricant circulation system, find nitrite source and, if possible, rectify.
Nitrate/nitrite content of the preparation water, if this is not removed from the public grid	Test sticks method or laboratory method	as required	Use water from the public grid if there is water from the public grid has > 50 mg/l nitrate: Inform the waterworks

<sup>1)</sup> The specified inspection intervals (frequency) are based on continuous operation. Other operational conditions can result in other inspection intervals; exceptions are possible in accordance with Sections 4.4 and 4.10 of the TGS 611.

Editor:

Signature:



## 11 Ersatzteile - Spare parts - F3

### 11.1 Ersatzteilbestellung - Ordering spare parts

Bitte geben Sie folgendes an - *Please indicate the following :*

- Seriennummer - *Serial No.*
- Maschinenbezeichnung - *Machines name*
- Herstellungsdatum - *Date of manufacture*
- Artikelnummer - *Article no.*

Die Artikelnummer befindet sich in der Ersatzteilliste. *The article no. is located in the spare parts list.*

Die Seriennummer befindet sich am Typschild. *The serial no. is on the type plate.*

Wenn Sie Ersatzteile anfragen, die unterschiedliche Längen aufweisen können, geben Sie bitte die Spitzenweite der Maschine an. *When requesting spare parts which might vary in length, indicate the distance between centres of the machine.*

### 11.2 Elektrische Ersatzteile - Electrical spare parts

### 11.3 Schaltplan - Wiring diagram

Der Schaltplan mit Ersatzteilliste befindet sich im Schaltschrank der CNC Maschine.

*The circuit diagram and spare parts list is located in the control cabinet of the CNC machine.*



## 11.4 Gehäuse - Housing

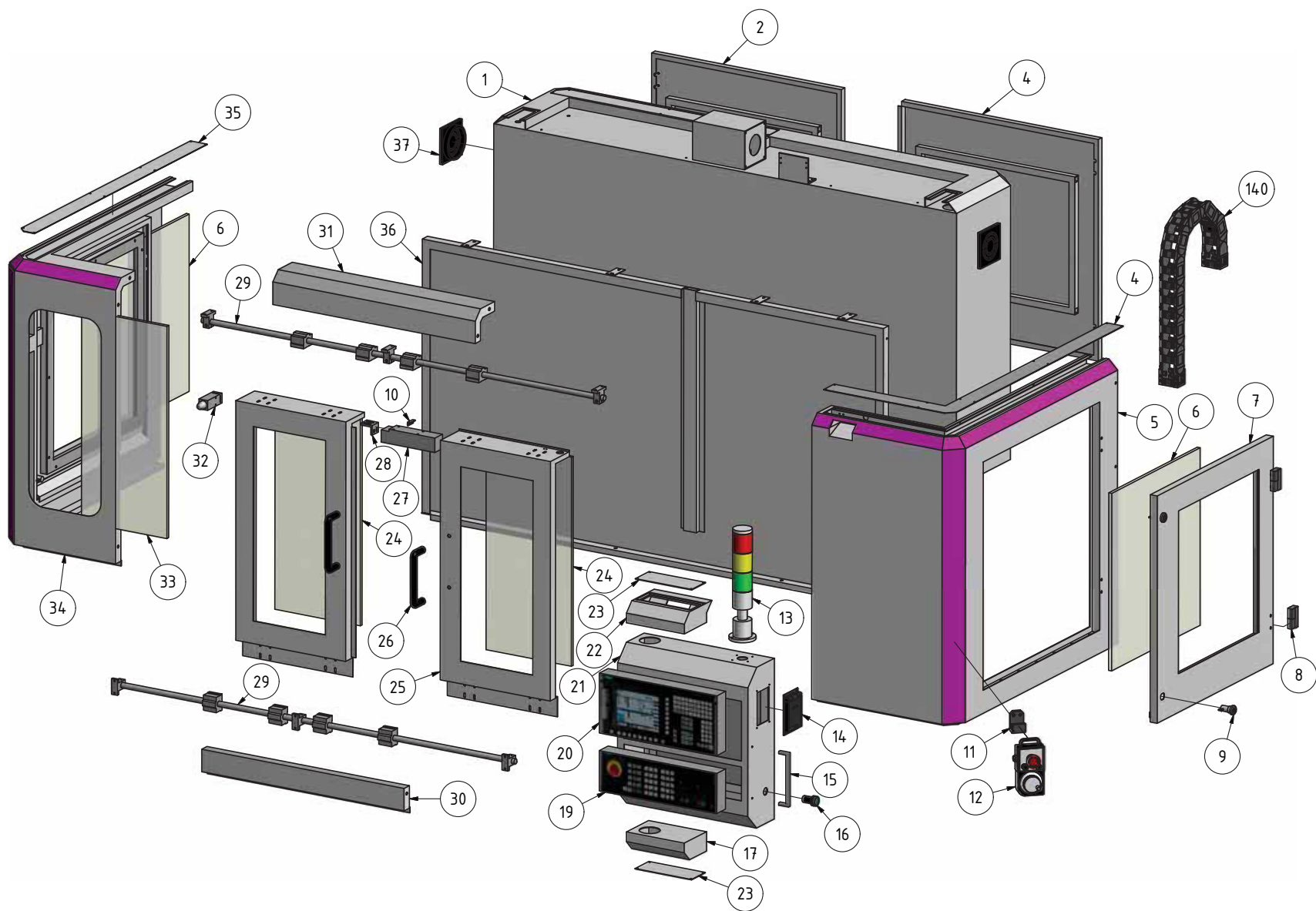


Abb.11-1: Gehäuse - Housing

Ersatzteile - Spare parts - F3  
Version 1.0 2015-05-27



F3 DE | GB

## 11.6 Fräskopf - Milling head

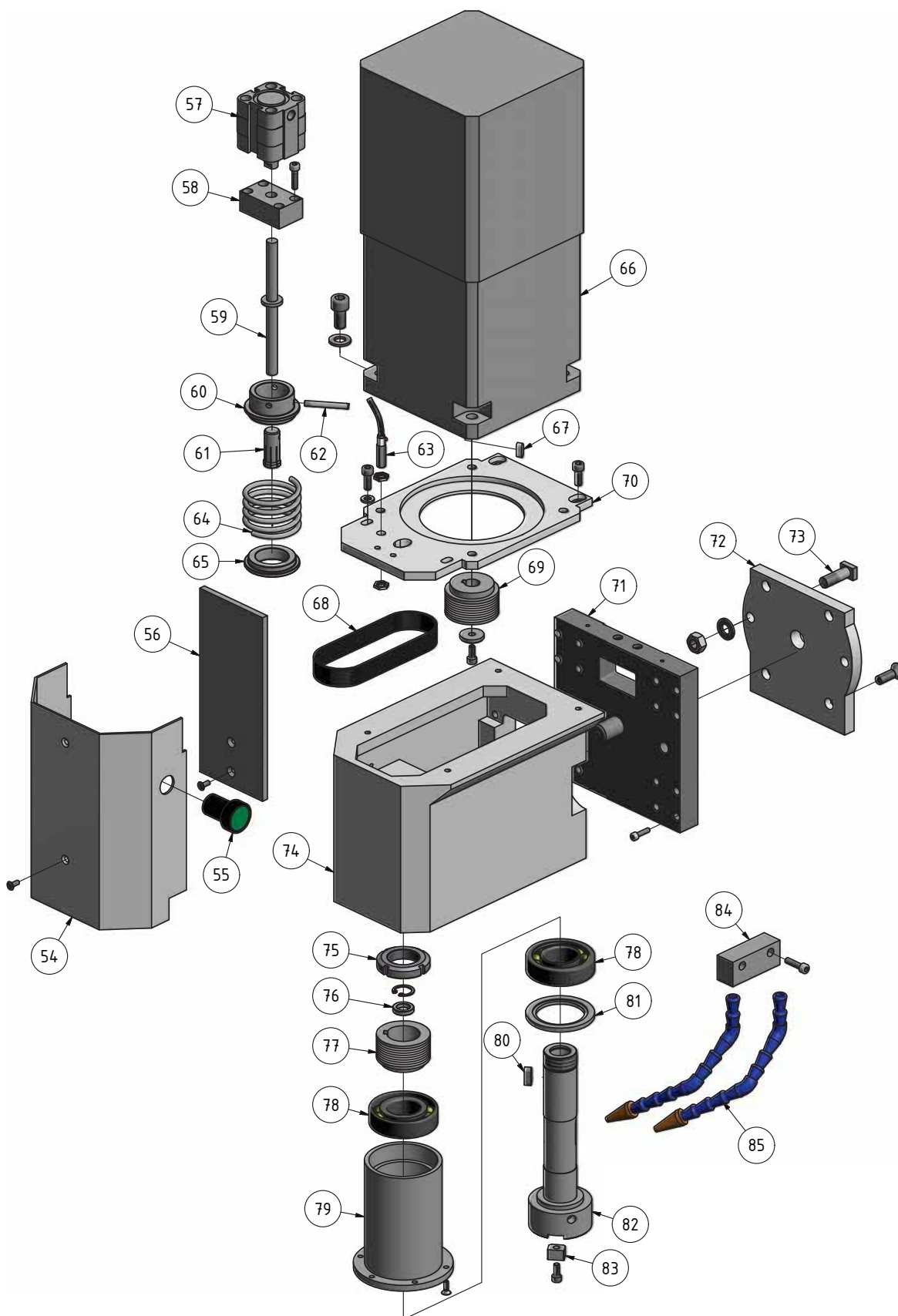


Abb. 11-3: Fräskopf - Milling head

## 11.7 Frästisch - Milling table

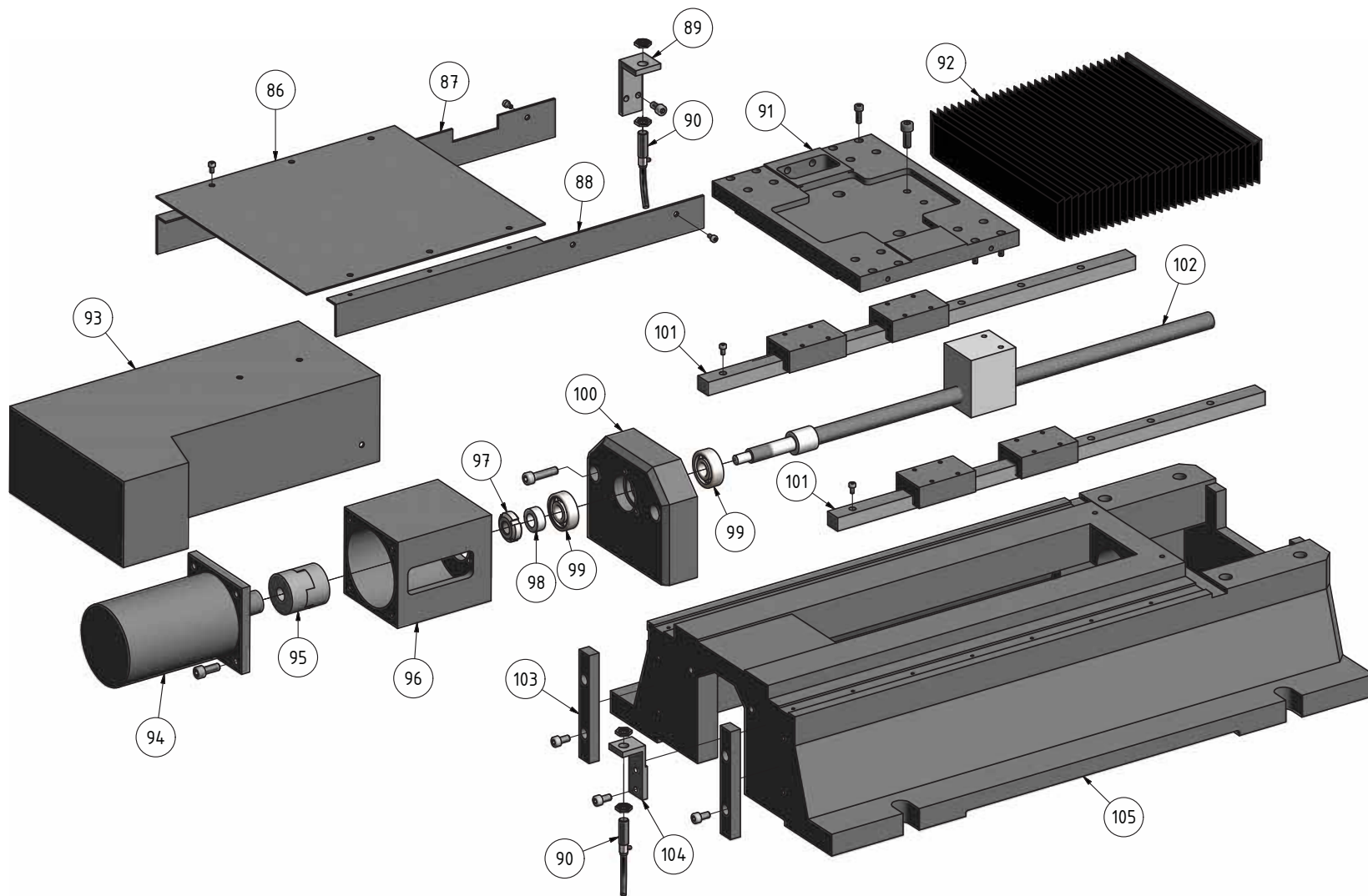


Abb.11-4: Frästisch - Milling table





## 11.8 Frästisch - Milling table

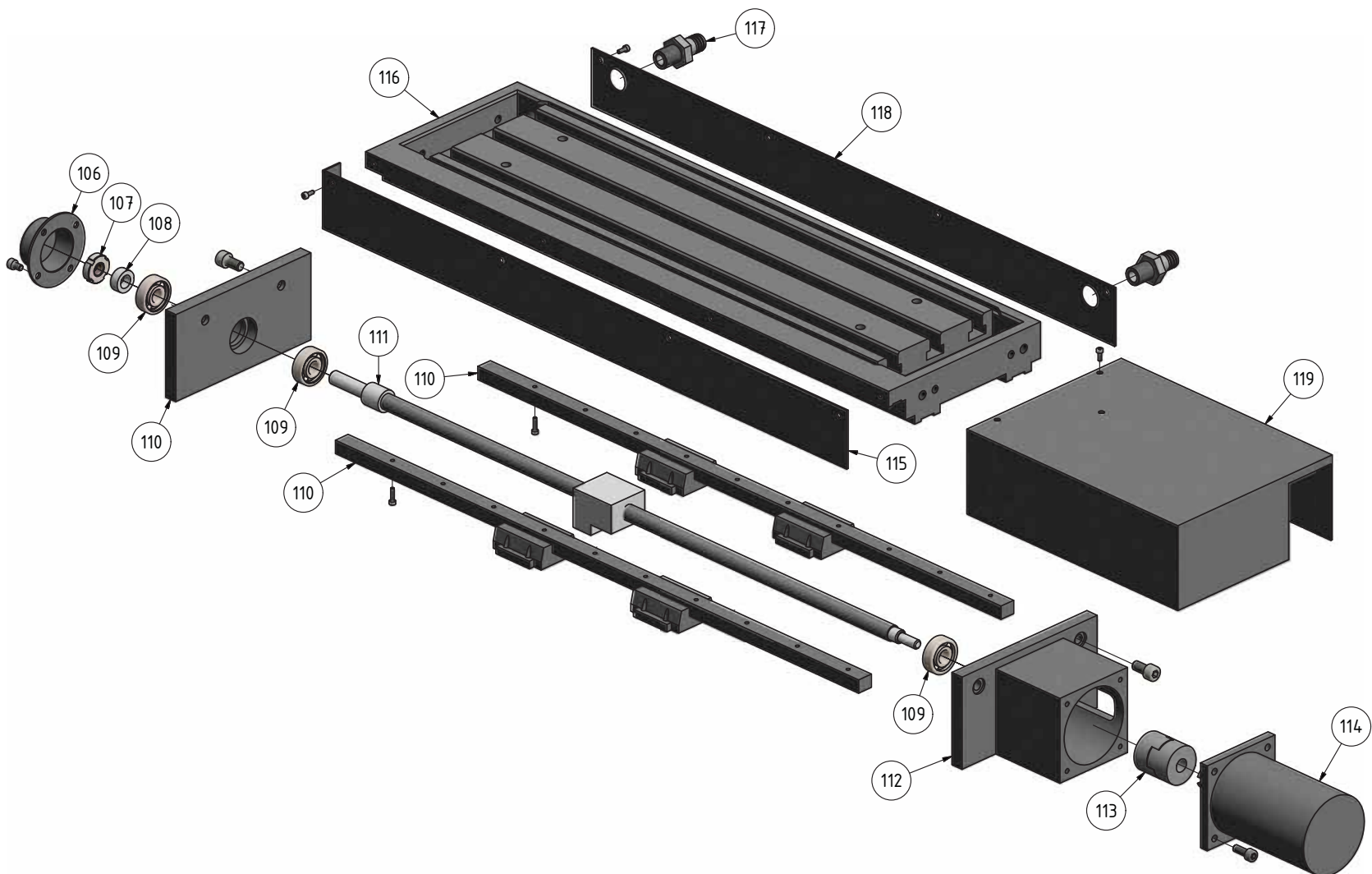


Abb. 11-5: Frästisch - Milling table





## 11.9 Säule - Column

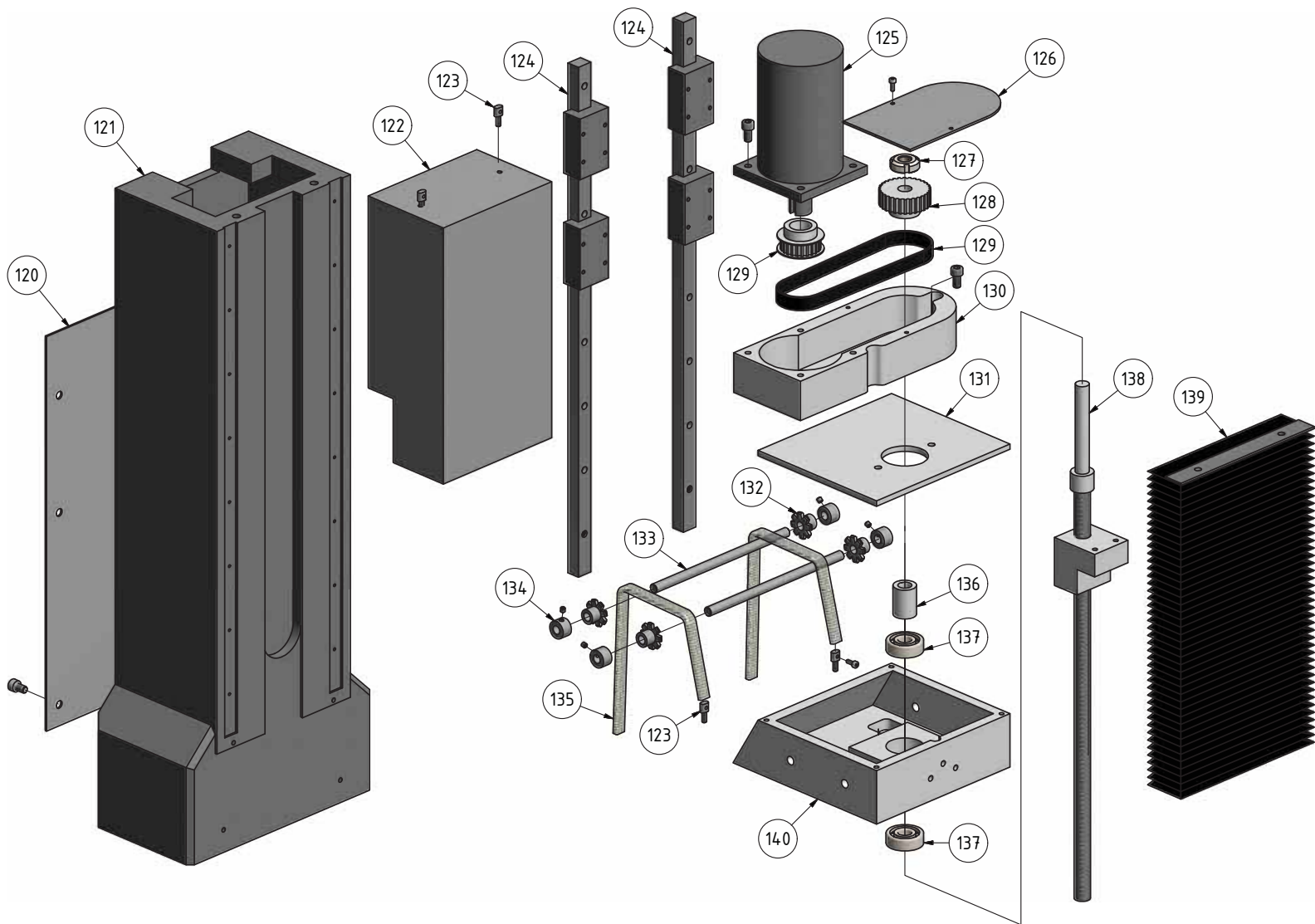


Abb.11-6: Säule - Column



## Teileliste - Parts list - F3

Pos. Pos.	Bezeichnung	Description	Menge	Grösse Size	Artikelnummer
			Qty.		Item no.
1	Schaltschrank	Switch cabinet	1		0350041001
2	Tür A	Door A	1		0350041002
3	Tür B	Door B	1		0350041003
4	Abdeckung	Cover	1		0350041004
5	Wand rechts	Wall right	1		0350041005
6	Glasscheibe	Glass pane	2		0350041006
7	Tür rechts	Door right	1		0350041007
8	Scharnier	Hinge	6		0350041008
9	Schloss	Lock	4		0350041009
10	Winkel	Angle	1		0350041010
11	Haken	Hook	1		0350041011
12	Handsteuerung	Hand control	1		0350041012
13	Kontrolleuchte	Control light	1		0350041013
14	Schnittstelle	Cut surface	1		0350041014
15	Griff	Grip	1		0350041015
16	Taster	Button	1		0350041016
17	Kasten	Box	1		0350041017
18	Abdeckung	Cover	1		0350041018
19	Steuerung A	Control A	1		0350041019
20	Steuerung B	Control B	1		0350041020
21	Gehäuse Steuerung	Control housing	1		0350041021
22	Kasten	Box	1		0350041022
23	Abdeckung	Cover	1		0350041023
24	Glasscheibe	Glass pane	2		0350041024
25	Schiebetür A	Sliding door A	1		0350041025
26	Griff	Gripp	2		0350041026
27	Sicherheitsschalter	Safety switch	1		0350041027
28	Winkel	Angle	1		0350041028
29	Führungsstange	Guide rod	2		0350041029
30	Abdeckung	Cover	1		0350041030
31	Abdeckung	Cover	1		0350041031
32	Sicherheitsschalter	Safety switch	1		0350041032
33	Glasscheibe	Glass pane	1		0350041033
34	Wand links	Wall left	1		0350041034
35	Abdeckung	Cover	1		0350041035
36	Rückwand	Back wall	1		0350041036
37	Lüfter	Fan	4		0350041037
38	Schiberkasten	Steam chest	1		0350041038
39	Halter	Holder	1		0350041039
40	Sieb	Filter	1		0350041040
41	Abdeckung	Cover	1		0350041041
42	O-Schraube	O-Screw	1		0350041042
43	Kühlmittelpumpe	Coolant pump	1		0350041043
44	Kühlmittel-tank	Coolant tank	1		0350041044
45	Gewindebuchse	Screw socket	6		0350041045
46	Maschinenfuß	Machine foot	6		0350041046
47	Tür rechts	Door right	1		0350041047
48	Tür links	Door left	1		0350041048
49	Schloss	Lock	2		0350041049
50	Unterbau	Substruction	1		0350041050
51	Abdeckung	Cover	1		0350041051
52	Abdeckung	Cover	1		0350041052
53	Sieb	Filter	2		0350041053
54	Abdeckung	Cover	1		0350041054
55	Taster	Button	1		0350041055
56	Platte	Plate	1		0350041056
57	Pneumatikzylinder	Pneumatic cylinder	1		0350041057
58	Platte	Plate	1		0350041058
59	Stange	Rod	1		0350041059
60	Buchse	Bushing	1		0350041060
61	Werkzeughalter	Tool holder	1		0350041061
62	Gewindestift	Grub screw	1		0350041062
63	Drehzahlsensor	Rotation speed sensor	1		0350041063
64	Feder	Spring	1		0350041064
65	Ring	Ring	1		0350041065
66	Motor	Motor	1		0350041066
67	Passfeder	Fitting key	1		0350041067



## Teilleiste - Parts list - F3

Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
68	Zahnriemen	Gear belt	1		0350041068
69	Riemenscheibe	Pulley	1		0350041069
70	Platte	Plate	1		0350041070
71	Halter	Holder	1		0350041071
72	Platte	Plate	1		0350041072
73	Schraube	Screw	2		0350041073
74	Gehäuse	Housing	1		0350041074
75	Nutmutter	Groove nut	1		0350041075
76	Ring	Ring	1		0350041076
77	Riemenscheibe	Pulley	1		0350041077
78	Kugellager	Ball bearing	2	7206	0350041078
79	Pinole	Sleeve	1		0350041079
80	Passfeder	Fitting key	1		0350041080
81	Ring	Ring	1		0350041081
82	Frässpindel	Mill spindle	1		0350041082
83	Nutenstein	Slot nut	2		0350041083
84	Klotz	Block	1		0350041084
85	Kühlmittelschlauch	Coolant hose	2		0350041085
86	Abdeckung	Cover	1		0350041086
87	Platte	Plate	1		0350041087
88	Platte	Plate	1		0350041088
89	Winkel	Angle	1		0350041089
90	Sensor	Sensor	3		0350041090
91	Platte	Plate	1		0350041091
92	Faltenbalg	Gaiter	1		0350041092
93	Abdeckung	Cover	1		0350041093
94	Motor Y-Achse	Motor Y-Axis	1		0350041094
95	Kupplung	Clutch	1		0350041095
96	Gehäuse	Housing	1		0350041096
97	Nutmutter	Groove nut	1		0350041097
98	Buchse	Bushing	1		0350041098
99	Kugellager	Ball bearing	2	6201	0350041099
100	Lagerbock	Bearing block	1		03500410100
101	Linearführung Y-Achse	Linear guides Y-Axis	2		03500410101
102	Kugelumlaufspindel	Ball screw	1		03500410102
103	Platte	Plate	2		03500410103
104	Winkel	Angle	1		03500410104
105	Maschinenfuß	Machine foot	1		03500410105
106	Abdeckung	Cover	1		03500410106
107	Nutmutter	Groove nut	1		03500410107
108	Buchse	Bushing	1		03500410108
109	Kugellager	Ball bearing	3	6201	03500410109
110	Linearführung X-Achse	Linear guides X-Axis	2		03500410110
111	Kugelumlaufspindel X-Achse	Ball screw X-Axis	1		03500410111
112	Lagerbock	Bearing block	1		03500410112
113	Kupplung	Clutch	1		03500410113
114	Motor X-Achse	Motor X-Axis	1		03500410114
115	Abdeckung	Cover	1		03500410115
116	Frästisch	Mill table	1		03500410116
117	Anschluss	Plug	2		03500410117
118	Abdeckung	Cover	1		03500410118
119	Abdeckung	Cover	1		03500410119
120	Abdeckung	Cover	1		03500410120
121	Säule	Column	1		03500410121
122	Gegengewicht	Balance	1		03500410122
123	Bolzen	Bolt	4		03500410123
124	Linearführung Z-Achse	Linear guides Z-Axis	2		03500410124
125	Motor Z-Achse	Motor Z-Axis	1		03500410125
126	Abdeckung	Cover	1		03500410126
127	Nutmutter	Groove nut	1		03500410127
128	Zahnscheibe	Gear wheel	1		03500410128
129	Zahnscheibe	Gear wheel	1		03500410129
130	Gehäuse	Housing	1		03500410130
131	Platte	Plate	1		03500410131
132	Zahnrad	Gear	4		03500410132
133	Welle	Shaft	2		03500410133
134	Buchse	Bushing	4		03500410134
135	Führungskette	Guide chain	2		03500410135
136	Buchse	Bushing	1		03500410136



## Teileliste - Parts list - F3

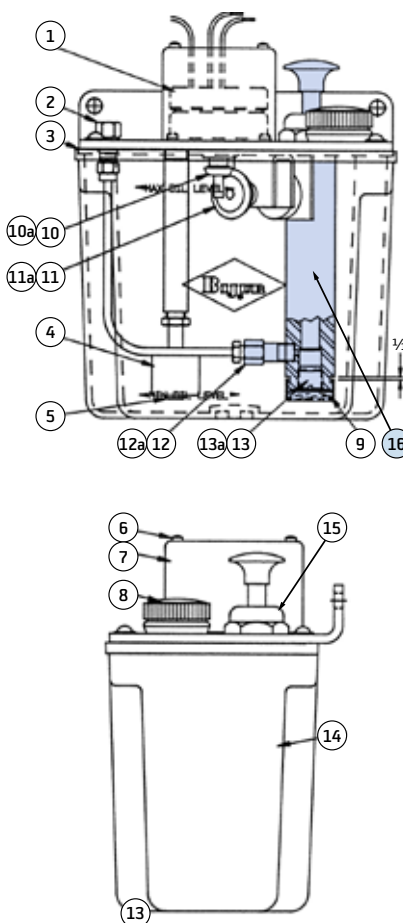
Pos.	Bezeichnung	Description	Menge Qty.	Grösse Size	Artikelnummer Item no.
137	Kugellager	Ball bearing	2	6201	03500410137
138	Kugelumlaufspindel Z-Achse	Ball screw Z-Axis	1		03500410138
139	Faltenbalg	Gaiter	1		03500410139
140	Energiekette	Energy chain	1		03500410140

## 11.10 Zentralschmierung

### TM1 C2893

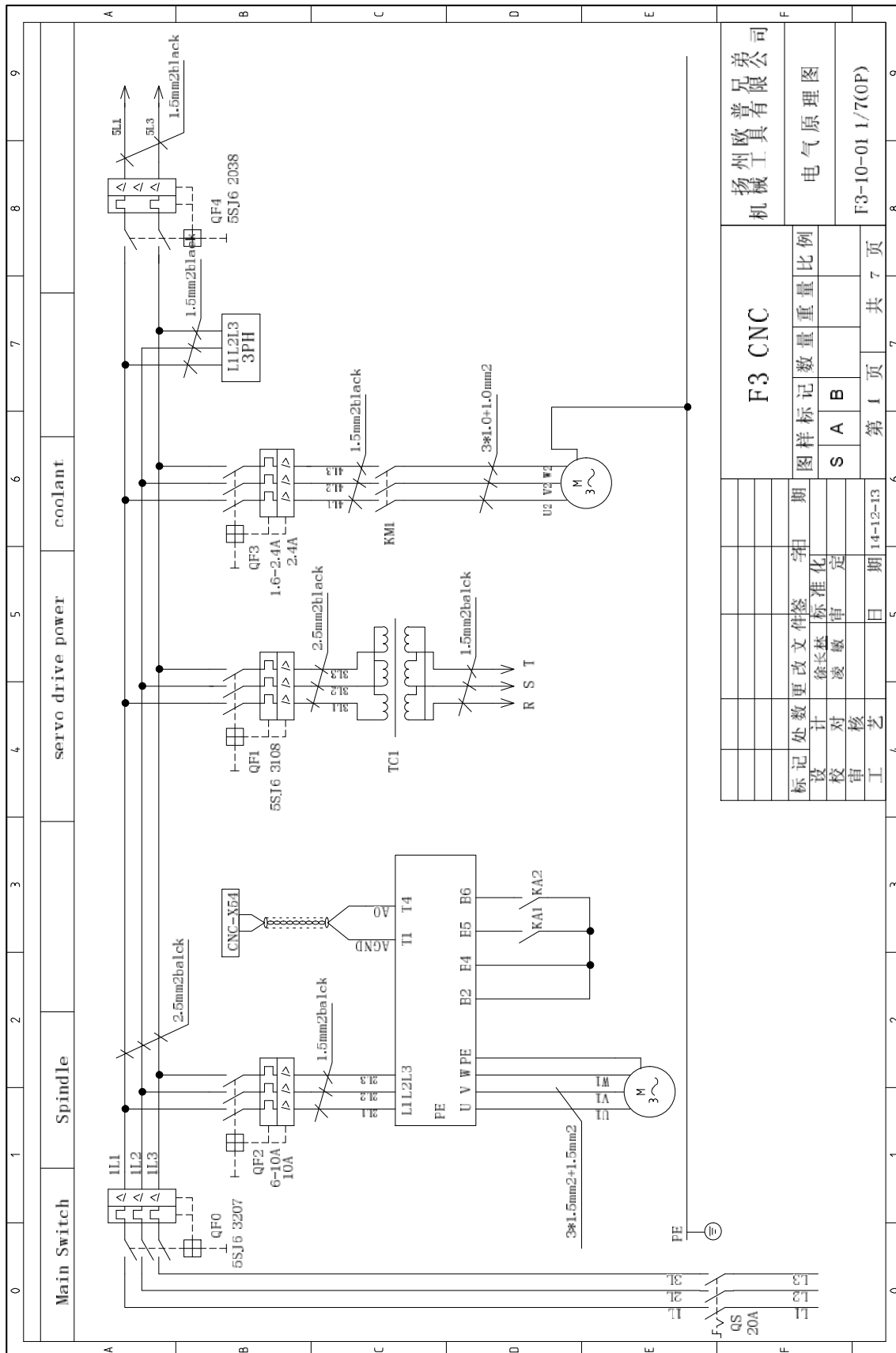
#### Ersatzteile

Teil	Artikelnr.	Beschreibung
1	variabel	Motor
2	B3108	Schottverschraubung
3	B8216	Behälterdichtung (0.475 l)
	21433	Behälterdichtung (1 Liter)
4	B8807	Füllstandschalter (0.475 l)
	B7760	Füllstandschalter (1 Liter)
5	H0402	O-Ring (Füllstandschalter)
6	HS195	Schraube, Abdeckkappe (4x)
7	B7827	Abdeckkappe (Motor)
8	B9230	Einfülldeckel
9	S178	Filter kompl.
10	B7824	Kegelrad
10a	HS209	Schrauben, Kegelrad (Set)
11	B7823	Kegelrad
11a	HS210	Schrauben, Kegelrad (Set)
12	B3344	Auslass-Rückschlagventil
12a	A3844	Auslass-Ventildichtung
13	B4359	Ansaugventil
13a	A4191	Ansaugventil-Dichtung
14	D3086C-KIT	Behälter (Kunststoff) (0.475 l)
	20324-KIT*	Behälter (Kunststoff) (1 Liter)
15	B7389	Vorspannung
16	B7378	Pumpe - kompl. (0.475 l)
	B8827	Pumpe - kompl. (1 Liter)

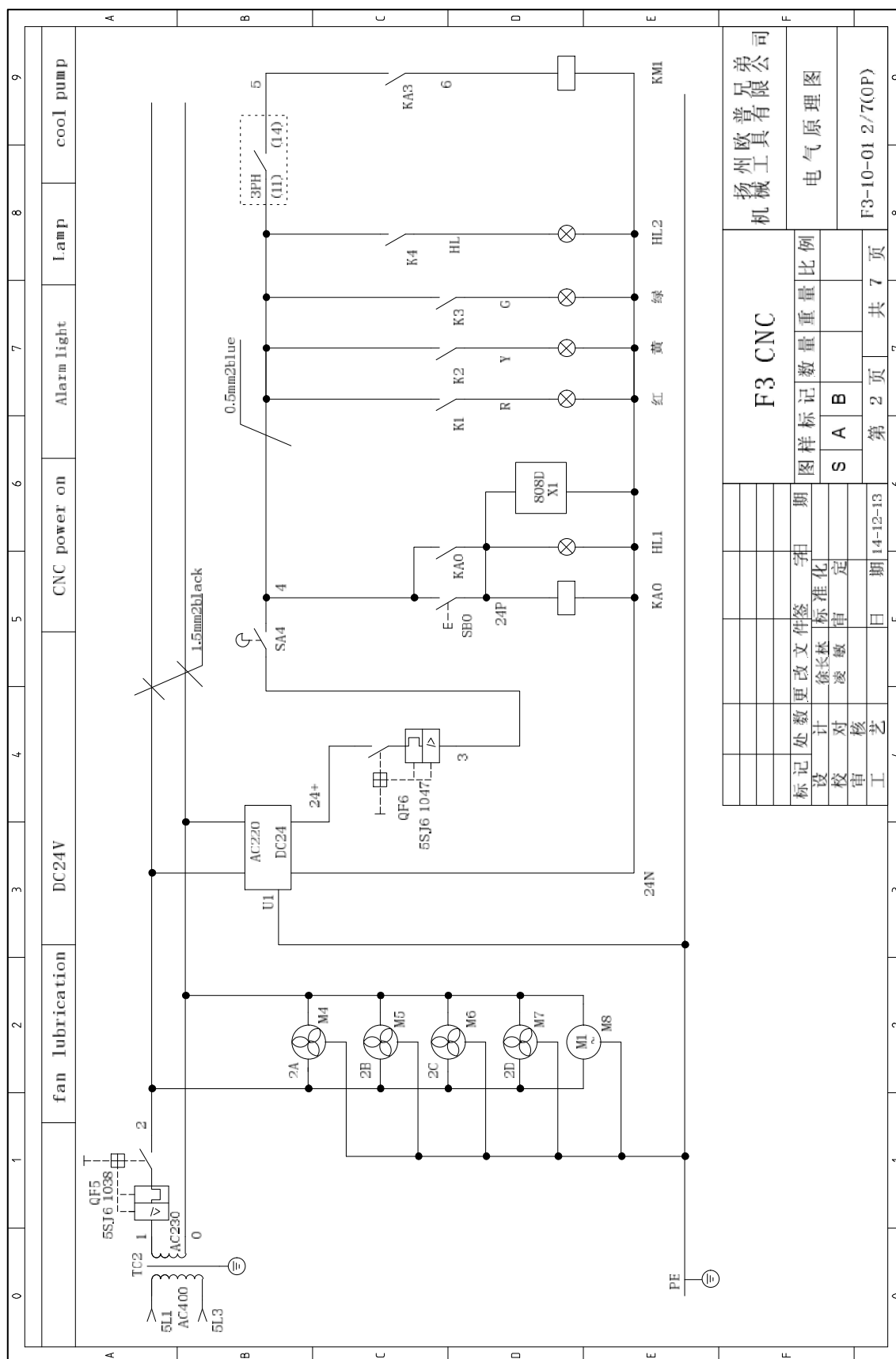




## 11.11 Schaltplan



F3 CNC		扬州欧普有限公司	
图样标记	数量	重量	比例
S	A	B	
第 1 页		共 7 页	
标记	处数	更改文件	日期
设计	校对	审核	日期
校核	工艺		14-12-13
F3-10-01 1/7(OP)			

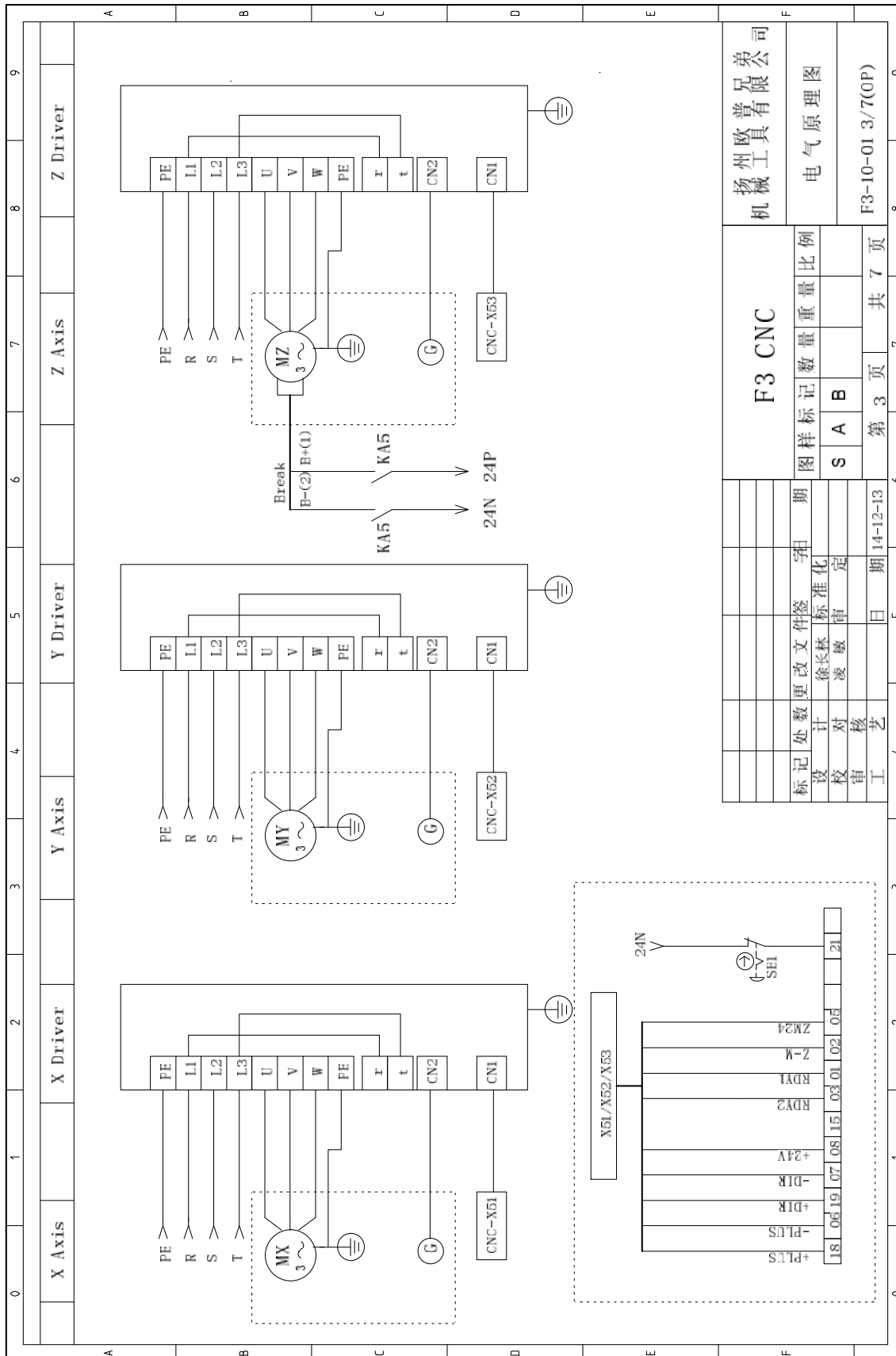


F3 CNC			
图样标记	数量	重量	比例
S	A	B	
第 2 页 共 7 页			
标记	处数	更改	文件
设计	校核	审核	标准
校核	审核	日期	14-12-13
工艺	日期	14-12-13	

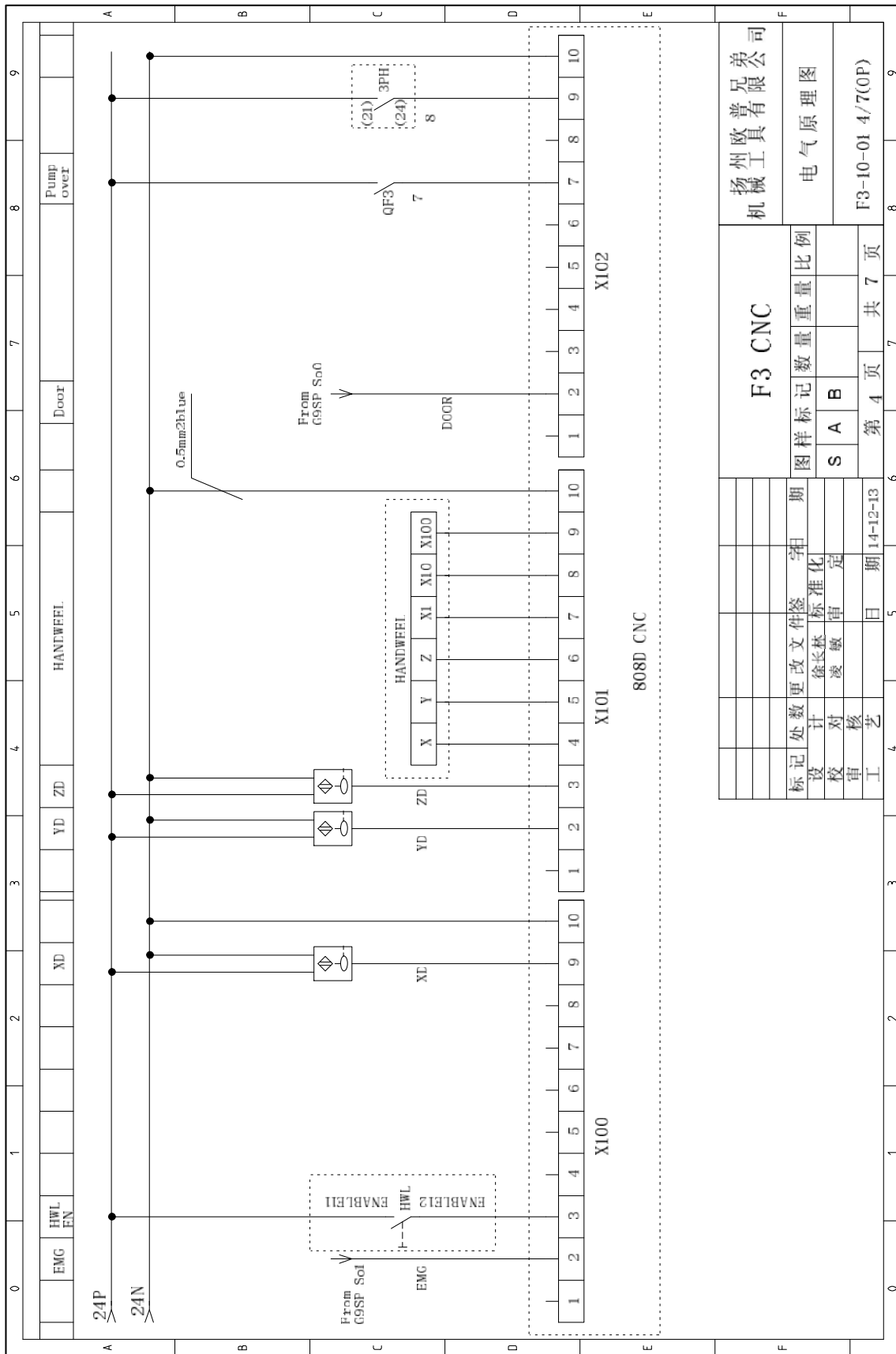
扬州欧普兄弟有限公司  
机械工具有限公司

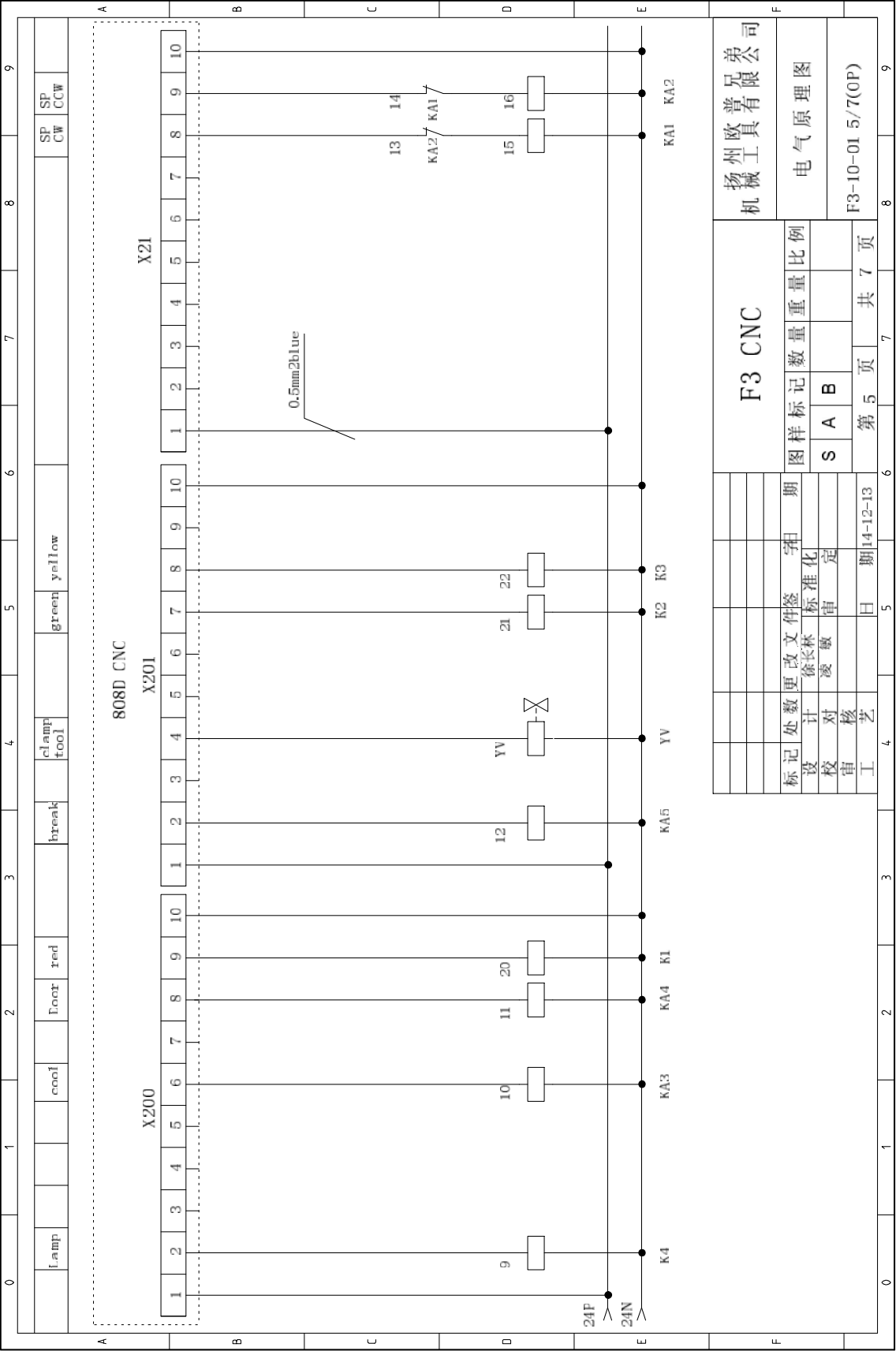
电气原理图

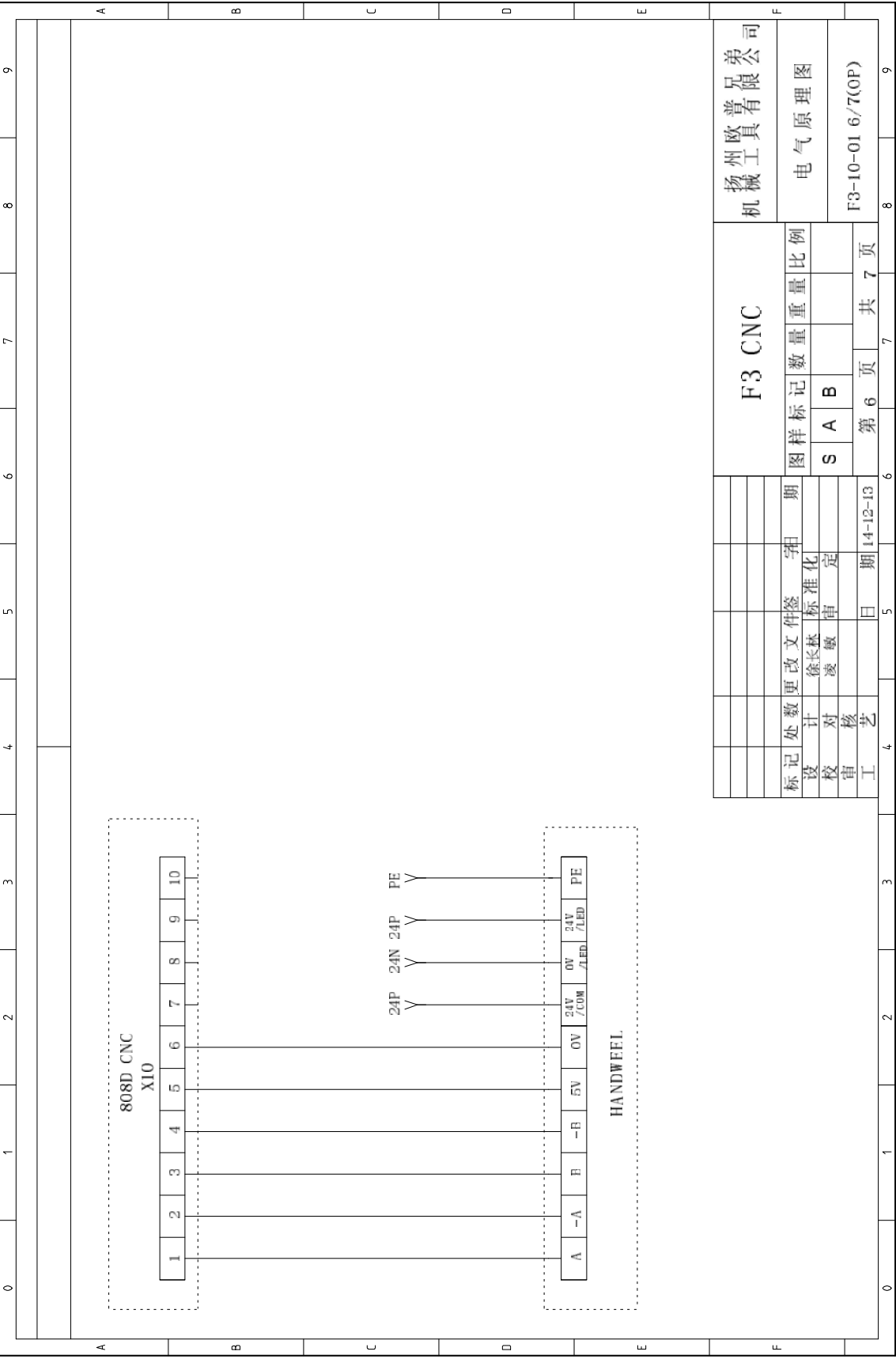
F3-10-01 2/7(OP)



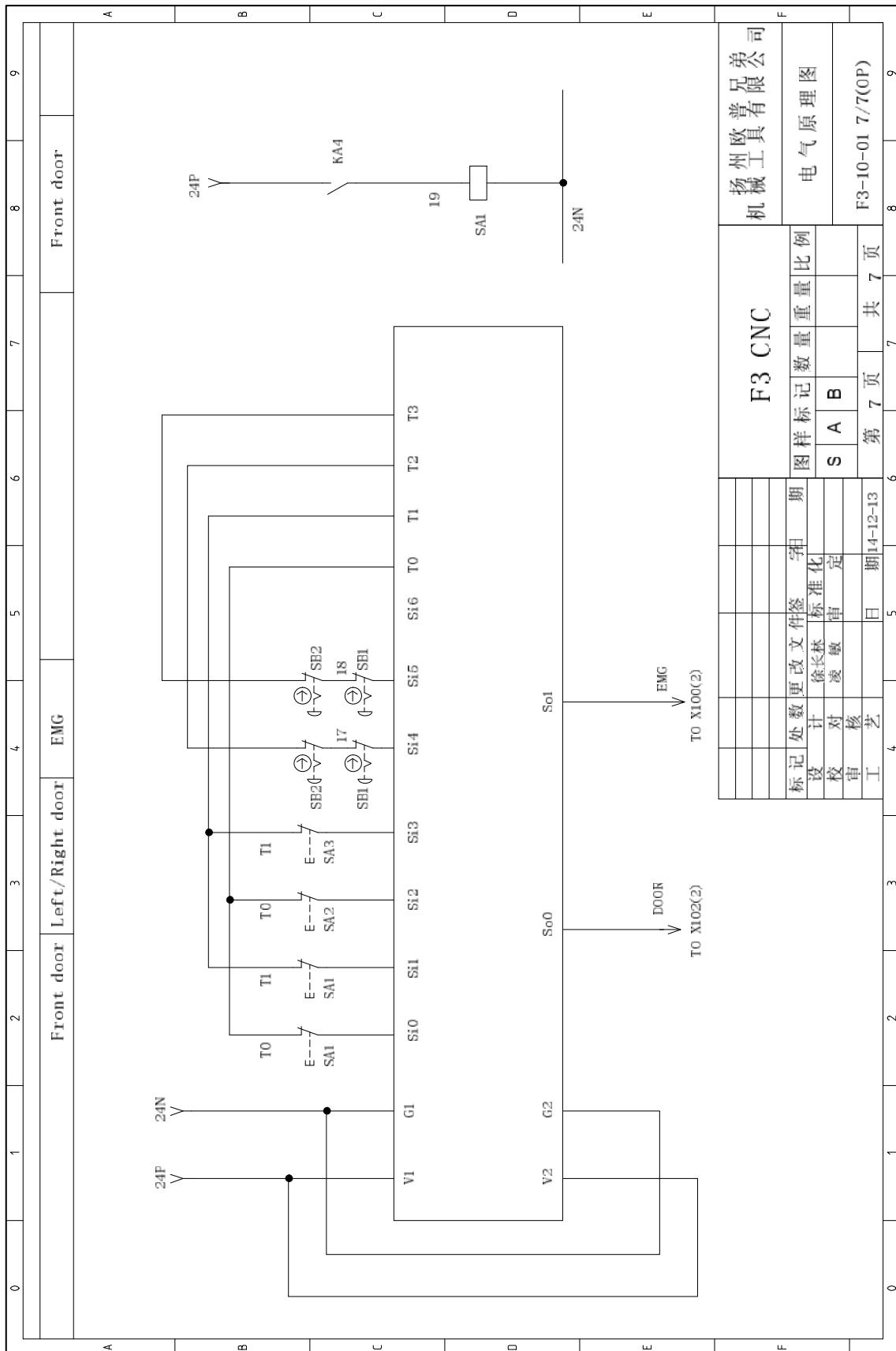








扬州欧普兄弟 机械工具有限公司				F3 CNC				电气原理图				F3-10-01 6/7(OP)			
图样标记				数量				重量				比例			
S				A				B				第 6 页			
第 6 页				共 7 页				F3-10-01 6/7(OP)				第 7 页			
标记				处数更改				文件				字			
设计				校				审				定			
工				艺				期				14-12-13			





## 12 Appendix

### 12.1 Copyright

This document is protected by copyright. All derived rights are reserved, especially those of translation, re-printing, use of figures, broadcast, reproduction by photo-mechanical or similar means and recording in data processing systems, either partial or total.

### 12.2 Changes

Any changes in the construction, equipment and accessories are reserved for reasons of enhancement. Therefore, no claims may be derived from the indications and descriptions. Errors excepted!

### 12.3 Storage

#### ATTENTION!

**In case of wrong and improper storage electrical and mechanical machine components might get damaged and destroyed.**

**Store packed and unpacked parts only under the intended environmental conditions.**

**Follow the instructions and information on the transport box:**



- **Fragile goods**  
(Goods require careful handling)



- **Protect against moisture and humid environment**



- **Prescribed position of the packing case**  
(Marking the top surface - arrows pointing up)



- **Maximum stacking height**

**Example: not stackable - do not stack further packing case on top of the first one.**



Consult Optimum Maschinen Germany GmbH if the CNC machine and accessories are stored for more than three months or are stored under different environmental conditions than those given here. ➡ "Environmental conditions - operation" on page 22



## 12.4 Terminology/Glossary

Term	Explanation
Cross table, milling table	Bearing surface, clamping surface for the workpiece with X- and Y-axis travel
CAD	C omputer A ided D esign
CAM	C omputer A ided M anufacturing
CAP	C omputer A ided P lanning
CNC	C omputerised N umerical C ontrol
NC	N umerical C ontrol
ATC	A utomatic T ool C hanger
Workpiece	piece to be milled, drilled or machined.
Spindle head	Milling head, upper part of the CNC milling machine
Spindle sleeve	Hollow shaft in which the milling spindle turns.
Milling spindle	Shaft activated by the motor
Tool	Milling cutter, drill bit, etc.
Machine control panel	Control panel, keyboard and screen of the CNC machine
Separating protective equipment	Cover, housing
Step motor	Synchronous motor where the rotor (turnable piece of the motor with a shaft) can be turned targeted around an angle if the triggered stator coils (non-turnable piece of the motor) is well selected.
Servo motor	A motor which can travel to different predetermined positions and remain there. In this case, the position is an angular position of a rotary motor; in case of a linear motor it is a linear position. The starting-up and maintaining of the predetermined position is performed by means of a control.

## 12.5 Product follow-up

We are required to perform a follow-up service for our products which extends beyond shipment.

We would be grateful if you could send us the following information:

- Modified settings
- Experiences with the CNC machine, which could be important to other users
- Recurring failures
- Difficulties with the documentation

Optimum Maschinen Germany GmbH

Dr.-Robert-Pfleger-Str. 26  
D-96103 Hallstadt

Technical hotline:

Fax +49 ( 0 ) 951 - 96555 - 888

email: [info@optimum-maschinen.de](mailto:info@optimum-maschinen.de)



## 12.6 Re-export

All contractual products and technical knowledge are delivered by Optimum Maschinen Germany GmbH in accordance with the currently applicable AWG/AWV/EC Dual-Use regulation as well as the US export regulations and are intended to be used and to remain in the delivery country, which had been agreed with the customer. If customers intend to re-export contractual products, they are obliged to comply with US, European and national export regulations. It is forbidden to re-export contract products contrary to these regulations.

Customers are required to obtain information regarding the currently applicable regulations and provision (Office for Export, 65760 Eschborn/Taunus or US Department of Commerce, Office of Export Administration, Washington D. C. 20230). Regardless of whether the customer specifies the place of destination of the delivered contractual products, the customer shall be solely responsible for obtaining any and all necessary permits from the respective appropriate foreign trade authority before it exports such goods. Optimum Maschinen Germany GmbH is not obliged to supply any information and does not grant any warranty as to whether the contractual products comply with the corresponding export regulations. The customer is not entitled to requesting that Optimum Maschinen Germany GmbH adapt the contractual products to the corresponding export regulations.

Any further delivery of contractual products to the customer by third parties with or without knowledge of Optimum Maschinen Germany GmbH shall at the same time require the transfer of the export license conditions. The customer shall be completely liable in case of non-observance of the relevant conditions.

Without previous official approval, the customer is not allowed to deliver contractual products directly or indirectly to countries, which are subject to an US embargo or to deliver to any natural or juridical persons in such countries, including natural or juridical persons, who are listed on the US American, European or national Indexes (e.g.: "Specifically Designated Nationals and Blocked Persons", "Entity List/V" Denied Persons List"). Moreover, contractual products must not be supplied to natural or juridical persons who are in any way involved in the support, development, production or use of chemical, biological or nuclear weapons of mass destruction.





## 12.7 Liability Claims for Defects / Warranty

Beside the legal liability claims for defects of the customer towards the seller, the manufacturer of the product, OPTIMUM GmbH, Robert-Pfleger-Straße 26, D-96103 Hallstadt, does not grant any further warranties unless they are listed below or were promised in the framework of a single contractual provision.

- The processing of the liability claims or of the warranty is performed as chosen by OPTIMUM GmbH either directly or through one of its dealers.  
Any defective products or components of such products will either be repaired or replaced by components which are free from defects. Ownership of replaced products or components is transferred to OPTIMUM Maschinen Germany GmbH.
- The automatically generated original proof of purchase which shows the date of purchase, the type of machine and the serial number, if applicable, is the precondition in order to assert liability or warranty claims. If the original proof of purchase is not presented, we are not able to perform any services.
- Defects resulting from the following circumstances are excluded from liability and warranty claims:
  - Using the product beyond the technical options and proper use, in particular due to overstraining of the machine.
  - Any defects arising by one's own fault due to faulty operations or if the operating manual is disregarded.
  - Inattentive or incorrect handling and use of improper equipment
  - Unauthorized modifications and repairs
  - Insufficient installation and safeguarding of the machine
  - Disregarding the installation requirements and conditions of use
  - atmospheric discharges, overvoltage and lightning strokes as well as chemical influences
- The following items are also not subject to liability or warranty claims:
  - Wearing parts and components which are subject to a standard wear as intended such as e.g. V-belts, ball bearings, illuminants, filters, sealings, etc.
  - Non reproducible software errors
- Any services, which OPTIMUM GmbH or one of its agents performs in order to fulfil any additional warranty are neither an acceptance of the defects nor an acceptance of its obligation to compensate. Such services do neither delay nor interrupt the warranty period.
- Place of jurisdiction for legal disputes between businessmen is Bamberg.
- If one of the aforementioned agreements is totally or partially inoperative and/or invalid, a provision closest to the intent of the warrantor is considered agreed upon, which remains within the framework of the limits of liability and warranty which are specified by this contract.

## 12.8 Note regarding disposal / options to reuse:

Please dispose of your machine in an environmentally friendly way, not by disposing of the waste not in the environment, but by acting in a professional way.

Please neither throw away the packaging nor the used machine later on, but dispose of them according to the guidelines established by your city council/municipality or by the corresponding waste management enterprise.



## 12.8.1 Decommissioning

### CAUTION!

Used devices need to be decommissioned in a professional way in order to avoid later misuses and endangerment of the environment or persons.



- **Disconnect the plug from the power supply.**
- **Cut the connection cable.**
- **Remove all environmentally hazardous operating fluids from the used device.**
- **If applicable remove batteries and accumulators.**
- **Disassemble the machine if required into easy-to-handle and reusable assemblies and component parts.**
- **Dispose of machine components and operating fluids using the intended disposal methods.**

## 12.8.2 Disposal of new device packaging

All used packaging materials and packaging aids from the machine are recyclable and generally need to be supplied to the material reuse.

The packaging wood can be supplied to the disposal or the reuse.

Any packaging components made of cardboard box can be chopped up and supplied to the waste paper collection.

The films are made of polyethylene (PE) and the cushion parts are made of polystyrene (PS). These materials can be reused after reconditioning if they are passed to a collection station or to the appropriate waste management enterprise.

Only forward the packaging materials correctly sorted to allow direct reuse.

## 12.8.3 Disposing of the old device

### INFORMATION

Please take care in your interest and in the interest of the environment that all component parts of the machine are only disposed of in the intended and admitted way.



Please note that the electrical devices comprise a variety of reusable materials as well as environmentally hazardous components. Please ensure that these components are disposed of separately and professionally. In case of doubt, please contact your municipal waste management. If appropriate, call on the help of a specialist waste disposal company for the treatment of the material.

## 12.8.4 Disposal of electrical and electronic components

Please make sure that the electrical components are disposed of professionally and according to the statutory provisions.

The machine is composed of electrical and electronic components and must not be disposed of as household waste. According to the European Directive 2002/96/EC regarding electrical and electronic used devices and the implementation of national legislation, used power tools and electrical machines need to be collected separately and supplied to an environmentally friendly recycling centre.

As the machine operator, you should obtain information regarding the authorised collection or disposal system which applies for your company.

Please make sure that the electrical components are disposed of professionally and according to the legal regulations. Please only throw depleted batteries in the collection boxes in shops or at municipal waste management companies.



### 12.8.5 Disposal of lubricants and coolants

#### ATTENTION!

Please imperatively make sure to dispose of the used coolant and lubricants in an environmentally compatible manner. Observe the disposal instructions of your municipal waste management companies.



#### INFORMATION

Used coolant emulsions and oils should not be mixed up since it is only possible to reuse used oils without pre-treatment, if they have not been mixed.

The disposal instructions for used lubricants are made available by the manufacturer of the lubricants. If necessary, request the product-specific data sheets.



### 12.9 Disposal via municipal collection

Disposal of used electrical and electronic components

(Applicable in the countries of the European Union and other European countries with a separate collecting system for those devices).

The sign on the product or on its packing indicates that the product must not be handled as common household waste, but that it needs to be disposed of at a central collection point for recycling. Your contribution to the correct disposal of this product will protect the environment and the public health. Incorrect disposal constitutes a risk to the environment and public health. Recycling of material will help reduce the consumption of raw materials. For further information about the recycling of this product, please consult your District Office, the municipal waste collection station or the shop where you have bought the product.



### 12.10 RoHS, 2002/95/EC

The symbol on the product or on its packing indicates that this product complies with the European directive 2002/95/EC.



## EC - Declaration of Conformity

in accordance with the Machinery Directive 2006/42/EC Annex II 1.A

**The manufacturer / distributor:** Optimum Maschinen Germany GmbH  
Dr.-Robert-Pfleger-Str. 26  
D - 96103 Hallstadt

**hereby declares that the following product**

**Product designation:** F3  
**Type designation:** CNC milling machine  
**Serial number:** \_ \_ \_ \_ \_  
**Year of manufacture:** 20\_\_

CNC milling machine for craft and industrial plants which meets all the relevant provisions of the above mentioned Directive 2006/42/EC as well as the other directives applied (below) including their amendments in force at the time of declaration.

The following other EU Directives have been applied:

EMC Directive 2014/30/EC

Low Voltage Directive 2014/35/EC

The safety objective meet the requirement of EC Directive 2006/42/EC.

**The following harmonized standards were applied:**

EN ISO 12100:2010 - Safety of machinery - General principles for design - Risk assessment and risk reduction

DIN EN 60204-1 - Safety of machines - Electrical equipment of machines, Part 1 General requirements

EN 12417 - Machine tools - Safety - Machining centres

DIN 66025-1, Publication date:1983-01 - Numerical control of machines; general requirements

DIN 66025-2, Publication date:1988-09 - Industrial automation; numerical control of machines, preparatory and miscellaneous functions

Responsible for documentation: Kilian Stürmer, phone: +49 (0) 951 96555-800

**Address:** Dr.-Robert-Pfleger-Str. 26  
D - 96103 Hallstadt



**Kilian Stürmer**

Hallstadt, 19/05/2015



## Index

### A

Accident report .....	18
Accuracy .....	21
Alarms .....	156
Aligning the machine .....	28
Anchored assembly .....	28
Anchor-free assembly .....	28
Assembly	
anchored .....	28
AWG/AWV/EG-Dual-Use .....	188

### B

Brief instruction 808D ADVANCED Milling .....	64
---	----

### C

Central lubrication .....	33
Central lubrication system .....	33, 61
Cleaning	
Cooling lubricant tank .....	167
Commissioning .....	31
commissioning	
tool .....	58
Compressed air supply .....	31
Compressed air, .....	20
Connection	
compressed air .....	31
electrical .....	29
Cooling lubricant .....	160, 167
Cooling lubricants .....	160, 168
Coordinate systems .....	36
Copyright .....	186
Corrosion protection .....	28

### D

Data interfaces .....	61
Declaration of Conformity .....	192
Dimensions .....	21
DIN Code	
ISO Code	
60	
Disinfection	
Cooling lubricant tank .....	167
Disposal .....	191
Drain compressed	
air service unit .....	166
Dual-Use .....	188

### E

EC Declaration of Conformity .....	192
Electrical connection .....	20, 29
Electronics .....	18
Emergency-stop button .....	13
Environmental conditions .....	22
Error messages .....	156

### I

Insert tool .....	58
Inspection .....	163
Inspection plan	

for water-mixed cooling lubricants .....	168
Installation plan .....	27
Interlock switch sliding door .....	166

### K

Key combination .....	45
-----------------------	----

### L

Lubricant/coolant system .....	20
--------------------------------	----

### M

M - code list .....	153
M functions .....	153
Machine control panel .....	46
Main switch .....	12
Maintenance .....	167
Manual tool change .....	59
Mechanical maintenance work .....	17
Messages .....	156
Milling spindle .....	20
Milling table .....	21

### N

Notes .....	156
-------------	-----

### O

Obligations	
of the operating company .....	11
of the operator .....	11
Operating	
central lubrication system .....	61
Operating material .....	160
Operation .....	53
Operational modes .....	54
operator's log .....	162

### P

Panel Processing Unit .....	41
Passwords .....	51
Product follow-up .....	187
Programming .....	54
Prohibition, warning and mandatory signs .....	14
Protection levels .....	51
Protective	
equipment .....	15

### Q

Qualification of personnel	
Safety .....	10

### R

Re-export .....	188
Reference point approach .....	56
Requirements regarding the installation site .....	26

### S

Scope of delivery .....	25
Selecting the speed .....	61
SINUMERIK 808 D .....	159
Storage .....	26
Switching off the machine .....	60
Switching on the machine .....	55



## T

Tool	
commissioning .....	58
Tool changer .....	22
Tools .....	23
Torques of	
servo motors .....	21
Transport .....	25

## U

Using	
electronic handwheel .....	58
Using lifting equipment .....	17

## W

Working area	
traverse paths .....	21