



Operating manual

Version 1.1.6

CNC milling machine

OPTimill[®]
F 80

Part no. 3501081





Table of contents

1	Safety	
1.1	Representation conventions	6
1.2	Rating plate	6
1.3	Safety instructions (warning notes)	6
1.3.1	Classification of hazards	6
1.3.2	Other pictograms	7
1.4	Intended use	8
1.5	Reasonably foreseeable misuses	8
1.5.1	Avoiding misuses	9
1.6	Possible dangers caused by the CNC-machine	9
1.7	Industrial security	10
1.8	Qualification of the staff	10
1.8.1	Target group	10
1.8.2	Authorized staff	11
1.9	Operator positions	12
1.10	Safety devices	12
1.10.1	Lockable main switch	13
1.10.2	Emergency stop push button	13
1.10.3	Control technical protection	14
1.10.4	Polycarbonate windows	15
1.10.5	Prohibition, warning and mandatory signs	15
1.11	Safety check	15
1.12	Personnel protective equipment	16
1.13	Safety during operation	16
1.14	Safety during maintenance	17
1.15	Disconnecting and securing the CNC-machine	17
1.15.1	Using lifting equipment	17
1.15.2	Mechanical maintenance work	18
1.16	Releasing trapped persons	18
1.16.1	Sliding door interlock switch	18
1.17	Accident report	18
1.18	Electrical system	18
1.19	Clamping devices for workpieces and tools	18
1.20	Environmental protection and water conservation	19
2	Technical specification	
2.1	Electrical connection	20
2.2	Milling spindle	20
2.3	Lubricant/coolant system	20
2.4	Compressed air	20
2.4.1	Compressed air service unit	20
2.5	Dimensions	20
2.6	Working area, traverse path	21
2.7	Accuracy	21
2.8	Torques of servo motors	21
2.9	Milling table	21
2.10	Tool changer	21
2.11	Environmental conditions - operation	21
2.12	Environmental conditions - storage	21
2.13	Spindle power Advanced	22
2.14	Milling table dimensions	22
2.15	Emissions	22
2.16	Tools and tool holding fixtures	23
3	Delivery, interdepartmental transport, assembly and commissioning	
3.1	Notes on transport, installation, commissioning	24
3.1.1	General risks during internal transport	24
3.2	Scope of delivery	25
3.3	Load attachment points	25
3.4	Installation and assembly	26
3.4.1	Requirements to the installation site	26
3.5	Installation plan	27
3.5.1	Machine mounting	29
3.5.2	Aligning the machine	29
3.6	Installation and assembly	29
3.6.1	Mounting the coolant / lubricant tank	29



3.6.2	Assembly of the coolant / lubricant pumps	30
3.6.3	Corrosion protection	30
3.6.4	Connection compressed air supply	30
3.7	First commissioning	32
3.7.1	Fill up coolant	32
3.8	Refill central lubrication system	32
3.9	Functional test and controls	32
3.9.1	Warming up the machine	33
3.9.2	Save data function	33
3.9.3	Electrical connection	34
3.9.4	Regulated drives in connection with residual current devices	35
3.9.5	Protection from Dangerous Shock Currents, use of ELCBs	35
3.9.6	Current in the protective earth conductor - Leakage current	35
3.9.7	When the ELCB triggers	36
4	General information about CNC	
4.1	Compensation of geometry	37
4.2	Coordinate systems on CNC-machine tools	37
4.2.1	Cartesian coordinate system	37
4.2.2	Polar coordinate system	38
4.2.3	Machine coordinate system	38
4.2.4	Workpiece coordinate system	38
4.2.5	Rotary axes and secondary axes	39
4.3	NC mathematics	39
4.3.1	Basics of the coordinate calculation	39
4.3.2	Parameters of a triangle	39
4.3.3	Angle on a triangle	39
4.4	Trigonometric functions	40
5	User interface, machine control panel	
5.1	Screen arrangement	41
5.2	Elements on the PPU (Panel Processing Unit) front	42
5.2.1	Key combination	46
5.3	Elements on the MCP	47
5.4	Protection levels	52
5.4.1	Passwords	53
5.4.2	Change passwords	53
6	Operation	
6.1	Safety	54
6.2	Control and indicating elements	54
6.3	Operational modes	55
6.4	Programming	55
6.5	Operation of the machine	56
6.5.1	Turning on the CNC- machine	56
6.5.2	Reference point approach after turning on	57
6.5.3	Using the electronic handwheel	57
6.5.4	Inserting a tool	58
6.5.5	Manual tool change	58
6.5.6	Tool changer	59
6.5.7	Machine parameters of the tool changer	59
6.5.8	Turning off the CNC- machine	59
6.6	Operational modes	59
6.7	Programming	59
6.8	Start program	60
6.9	Central lubrication system	60
6.10	Data interfaces and current collection	60
6.11	Selecting the speed	60
6.11.1	Standard values for cutting speeds	62
6.11.2	Standard values for speeds with HSS – Eco – twist drilling	62
6.11.3	Clamping workpiece and setting workpiece datum	64
6.12	DRB central lubrication system	66
6.12.1	Technical specification	66
6.12.2	Function	67
6.12.3	Adjusting the amount of lubrication oil at the central lubrication	68
6.12.4	WGKX-4 and WGKX-5 Controller	68
6.12.5	Setting the operating mode on WGKX-1	69
6.12.6	Setting the running time and interval time	69



7	M - Code list, M functions	
7.1	M-function for milling machines according to PAL	70
7.2	G functions to PAL	71
8	Notes, messages and error messages	
8.1	Malfunctions on the machine	72
8.2	Problem- solving	74
8.2.1	Tool changer problem	74
9	SINUMERIK 808 D Advanced	
10	Maintenance	
10.1	Operating material	76
10.1.1	Machine lubricants	76
10.1.2	Cooling lubricants	76
10.2	Safety	77
10.2.1	Preparation	78
10.2.2	Restarting	78
10.3	Inspection and maintenance	78
10.3.1	Cleaning and replacing of the polycarbonate windows	81
10.3.2	Cleaning electrical cabinet	82
10.4	Interlock switch sliding door	82
10.4.1	Drain compressed air service unit	82
10.5	Recommended wear and tears parts	83
10.6	Repair	83
10.6.1	Customer service technician	83
10.7	Cooling lubricants and tanks	84
10.7.1	Inspection plan for water-mixed cooling lubricants	85
11	Ersatzteilzeichnungen - Spare part drawings	
11.1	F80 Advanced (20.04.2020) - Schaltplan - Wiring diagram	100
12	Brief instruction 808D Milling	117
13	Annex	
13.1	Copyright	206
13.2	Changes	206
13.3	Product follow-up	206
13.4	Change information manual	206
13.5	Terminology/Glossary	207
13.6	Re-export	208
13.7	Storage	208
13.8	Liability claims for defects / warranty	209
13.9	Advice for disposal / Options of re-use	209
13.9.1	Decommissioning	210
13.9.2	Disposal of the packaging of the new machine	210
13.9.3	Disposal of the used machine	210
13.9.4	Disposal of electrical and electronic components	210
13.9.5	Disposal of lubricants and cooling lubricants	211



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

Internet: www.optimum-maschinen.com



1 Safety

This part of the operating instructions

- explains the meaning and use of the warning notices 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,
- legal regulations 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. Please find the serial number on the rating plate.

1.1 Representation conventions

gives additional advices

calls on you to act

○ enumerations

1.2 Rating plate

DE Fräsmaschine EN Milling machine FR Fraiseuse ES Fresadora IT Fresatrice CS Frézka DA Freamaschine EL Φρεζοβρανο FI Porajyrsin HU Marógép NL Freamachine PL Frezarka PT Máquina freadora RO Mașină de frezat RU Фрезерный станок SK Frézka SL Frezalni stroj SV Fräsmaskiner TR Freze Tezgahı	OPTIMUM® MASCHINEN - GERMANY F 80 Optimum Maschinen Germany GmbH Dr.-Robert-Pfleger-Str. 26 96103 Hallstadt / Deutschland NO 3501081 3,7 KW SN 400 V ~50 Hz 10000 min⁻¹ Year www.optimum-maschinen.de CE
--	--

1.3 Safety instructions (warning notes)

1.3.1 Classification of hazards

We classify the safety instructions into different levels. The table below gives an overview of the classification of symbols (pictograms) and signal words for the specific danger and its (possible) consequences.

Pictogram	Signal word	Definition/Consequences
	DANGER!	Imminent danger that will cause severe injury or death to the staff.
	WARNING!	A danger that might cause severe injury to the staff or can lead to death.
	CAUTION!	Danger of unsafe procedure that might cause injury to the staff or property damages.

F80_GB_1.fm

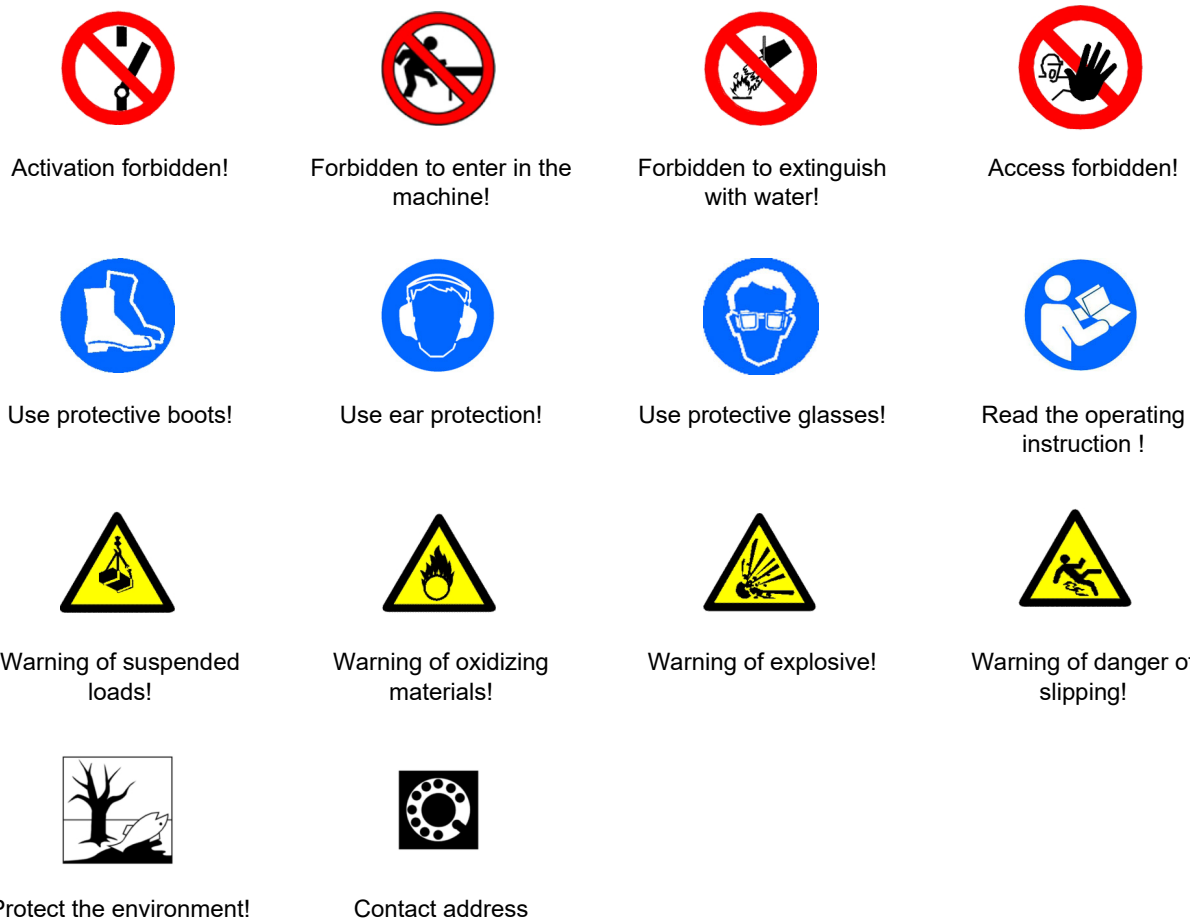


Pictogram	Signal word	Definition/Consequences
	ATTENTION!	Situation that could cause damage to the CNC-machine and products and other types of damage. No risk of injury to the staff.
	INFORMATION	Application tips and other important or useful information and notes. No dangerous or harmful consequences for the staff or objects.

In case of specific dangers, we replace the pictogram by



1.3.2 Other pictograms





1.4 Intended use

WARNING!

In the event of improper use of the CNC-machine

- **there may be a risk to the staff,**
- **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 76

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 approval of the company Optimum Maschinen Germany GmbH then the CNC-machine is being used improperly. Intended use

We do not take any liability for damages caused by intended use.

We expressly point out that the guarantee or CE conformity will expire due to any constructive technical or procedural changes which had not been performed by the company Optimum Maschinen Germany GmbH.

It is also part of intended use that

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

WARNING!

Heaviest injuries through improper use.

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



1.5 Reasonably foreseeable misuses

Any other use as the one determined under the "Intended use" or any use beyond the described use shall be deemed as not in conformity and is forbidden.

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 misuses it is necessary to read and understand the operating instructions before the first commissioning.

The operating staff has to be qualified.



1.5.1 Avoiding misuses

- 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. It has been designed and built using the latest technological advances.

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 for the staff resulting from these hazards.

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

INFORMATION

All staff involved in assembly, commissioning, operation and maintenance, must

- be duly qualified,
- strictly follow these operating instructions.

During improper use

- there may be a risk to the staff,
- there may be a risk 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 Industrial security

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement - and continuously maintain - a holistic, state-of-the-art industrial security concept. Siemens products and solutions only represent one component of such a concept.

The customer is responsible for preventing unauthorized access to its plants, systems, machines and networks. Systems, machines and components should only be connected to the enterprise network or the internet if and to the extent necessary and with appropriate security measures (e.g. use of firewalls and network segmentation) in place.

Additionally, Siemens' guidance on appropriate security measures should be taken into account. For more information about industrial security, please visit:

Industrial security (<http://www.siemens.com/industrialsecurity>).

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends to apply product updates as soon as available and to always use the latest product versions. Use of product versions that are no longer supported, and failure to apply latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed at: Industrial security (<http://www.siemens.com/industrialsecurity>).

WARNING!

Danger to life as a result of unsafe operating states resulting from software manipulation. Software manipulations (e.g. viruses, trojans, malware or worms) can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.



1.8 Qualification of the staff

1.8.1 Target group

This manual is addressed to

- the operating company,
- operators having sufficient specialist knowledge,
- the maintenance staff.

Therefore, the warning notes refer to both, operation and maintenance staff 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 staff.

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

Operator

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

Electrical specialist

Due to his professional training, knowledge and experience as well as his knowledge of respective standards and regulations the electrical specialist is able to perform works on the electrical system and to recognise and avoid any possible dangers himself.

The electrical specialist is specially trained for the working environment in which he is working and knows the relevant standards and regulations.

Specialist staff

Due to its professional training, knowledge and experience as well as his knowledge of relevant regulations the specialist staff is able to perform the assigned tasks and to recognise and avoid any possible dangers himself.

Instructed persons

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

1.8.2 Authorized staff

INFORMATION

For working on the CNC-machine sufficient expertise is required. No one must work on the machine without having the necessary education, not even for a short while.

As aid for training and operation we recommend to use the CNC-Software SinuTrain.

SinuTrain made by Siemens is the perfect software-supplement for the CNC-maschine F80 of Optimum Maschinen Germany GmbH

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 staff, objects and the environment.

Only authorized staff may operate the CNC-machine !





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

The operating company must

- train the staff,
- instruct the staff in regular intervals (at least once a year) on
 - all safety standards that apply to the CNC-machine,
 - operation of the CNC-machine,
 - accredited technical guidelines,
 - possible emergency situations,
- check staff's state of knowledge,
- document training/instruction in a operation book,
- require staff to confirm participation in training/instructions by means of a signature,
- check whether the staff is working safety- and risk-conscious and observe the operating instructions.

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

Obligations of the
operator

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

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

Additional
requirements
regarding the
qualification

1.9 Operator positions

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

1.10 Safety devices

Use the CNC-machine only with properly functioning safety devices.

Stop the CNC-machine immediately if there is a failure on the safety device or if it is not functioning for any reason.

It is your responsibility!

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

- have removed the cause of the failure,
- have verified that there is no danger resulting for the staff or objects.

WARNING!

If you bypass, remove or deactivate a safety device in any other way, you are endangering yourself and other staff 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,
- a fatal electrocution,
- seizing of clothes.

The CNC-machine includes the following safety devices:





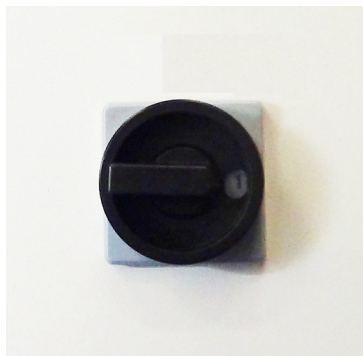
- a lockable main switch,
- One emergency stop push-button on the machine control panel, the milling head and on the electronic handwheel,
- A locked, separating protective equipment around the CNC-milling machine with sight windows made of break-proof Makrolon.
- Locking switch on the separating safety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

1.10.1 Lockable main switch

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

When the main switch is switched-off, the current supply is interrupted.

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.

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



1.10.2 Emergency stop push button

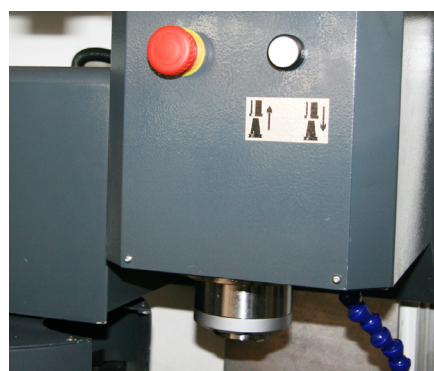
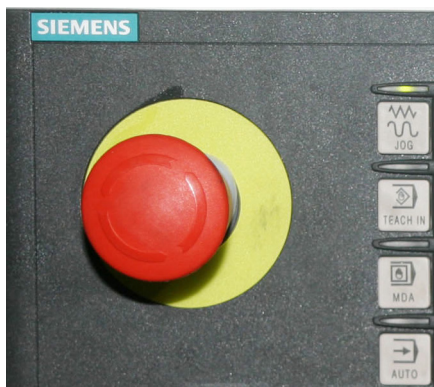
ATTENTION!

The emergency stop push button immediately stops the operation of the CNC-machine.

Press the emergency stop push 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, turn the knob of the particular push button to the right in order to restart the machine.





Img. 1-2: Emergency-stop push button

1.10.3 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,
- a fatal electrocution,
- seizing 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.





1.10.4 Polycarbonate windows

Polycarbonate windows which have a safety-critical function with respect to ejected parts, must be visual inspected by the customer responsible personnel at regular intervals to guarantee the operational safety of the CNC-machine.

Polycarbonate windows are subject to an aging process and are classified as wear parts.

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

A longer exposure from polycarbonate windows by cutting fluids can lead to accelerated aging. Also from the operator side can coolant agent, detergents, fats and oils or other corrosive substances cause a deterioration of the polycarbonate windows. The result is a reduced parts retentivity of the polycarbonate windows.

Cleaning and replacing of the polycarbonate windows on page 81

1.10.5 Prohibition, warning and mandatory signs

INFORMATION

All warning and mandatory signs must be legible. Check them regularly.



1.11 Safety check

Check the CNC-machine at least once per shift. Inform the person responsible immediately of any damage, defect or change in 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 every maintenance and repair work.

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

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

INFORMATION

Use the following table for organizing the checks.



General check		
Equipment	Check	OK
Protective housing	Switching function, firmly bolted and not damaged	
Signs, Markings	Installed and legible	
Sight window	Check for mechanical damage (scratches, cracks). Polycarbonate windows on page 15	
Date:	Checked by (signature):	



Functional test		
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):	

1.12 Personnel protective equipment

For some works you need personnel protective equipment as protective equipment.

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

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

Use 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 prescribed personnel protective equipment is available at the working place.



CAUTION!

Dirty or contaminated personnel protective equipment can cause diseases. Clean it each time after use and once a week.



1.13 Safety during operation

WARNING!

Before activating the CNC-machine assure yourself that this will neither endanger other persons nor cause damage to equipment.



Avoid any unsafe working practices:

- 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: Disorder
- Red light: Emergency stop Button actuated



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 result in a fire and/or an explosion (deflagration).

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

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

Only the following extinguishing agents are admitted:

- solid extinguishing agent of the fire class D (fires of metals)
- dry covering salts for magnesium
- a mixture of sand and cast chips
- argon (Ar) or nitrogen (N₂)

If fine mist and smoke is generated in the working room, suction units need to be provided in order to avoid the accumulation of ignitable mixtures and emissions.

We specially point out the specific dangers when working with and on the CNC-machine.

**1.14 Safety during maintenance**

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

Report all safety relevant changes and performance details of the CNC-machine. Document all changes, have the operating instructions updated accordingly and train machine operators.

1.15 Disconnecting and securing the CNC-machine

Switch off the CNC-machine by turning off the main switch before starting any maintenance and 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 tensions 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 the CNC-machine by turning off the main switch due to required works (e.g. functional control).

1.15.1 Using lifting equipment**WARNING!**

The use of unstable lifting and load suspension gear 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 competent supervisory authority, responsible for your company.

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

Fasten the loads properly.

Never walk under suspended loads!



1.15.2 Mechanical maintenance work

Remove or install protection safety devices before starting any maintenance work and re-install them once the work has been completed. This includes:

- Covers,
- Safety indications and warning signs,
- Earth (ground) connections.

If you remove protective or safety devices, re-fit them immediately after the completing the work. Check if they are working properly!

1.16 Releasing trapped persons

1.16.1 Sliding door interlock switch

 Interlock switch sliding door on page 82

The interlock switch can be mechanically unlocked for maintenance and servicing purposes. Carry out a safety check when you have reset the locking switch to its default state.  Safety check on page 15

1.17 Accident report

Inform your superiors 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 faster the causes can be eliminated.

INFORMATION

We point out the specific dangers when performing works with and on the CNC-machine when describing such works.



1.18 Electrical system

Have the machine and/or the electrical equipment checked regularly, at least every six months. 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. Immediately disconnect the CNC-machine if there are any anomalies in the power supply!  Maintenance on page 76

1.19 Clamping devices for workpieces and tools

ATTENTION!

Attention when taking over the existing clamping devices. Please check critically if the clamping device is appropriate for your CNC-machine.

- Only use clamping devices which have a complete inherent stiffness.
- Contact the manufacturer of the clamping device regarding the reuse of the clamping devices after damages on the clamping devices 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 §19g of the Water Resources Law.

When operating, decommissioning or disassembling the CNC-machine or parts hereof, please follow the requirements of the Water Resources Law. Please find detailed information about this topic in the regulation about devices to treat materials which are hazardous to water.





2 Technical specification

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

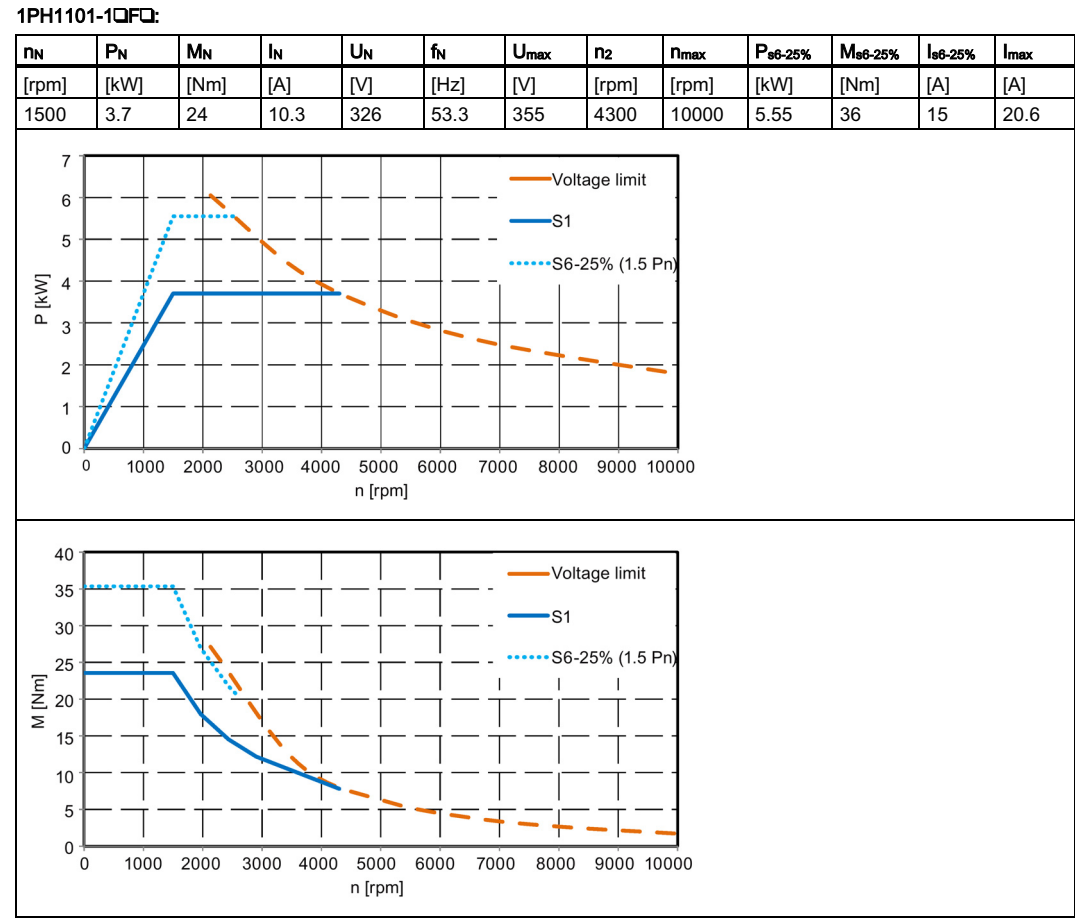
2.1 Electrical connection	
Total connection	3 x 400V ~ 50Hz 14 KVA
2.2 Milling spindle	
Power of spindle drive S1 operation	3.7 kW
Power of spindle drive S6-30 % operation	5.5 kW
Torque S1 operation	24 Nm
Torque S6-30 % operation	36 Nm
Speed	50 - 10000 min ⁻¹
Spindle seat	ISO 40 / BT 40 x 45°
2.3 Lubricant/coolant system	
Power of the cooling lubricant pump [W]	450
Tank capacity [litre]	80
2.4 Compressed air	
min. connection [MPa]	0.6 (6 bar)
Air consumption	0.1 - 0.15 m ³ / h at 0.6 MPa
Recommended compressed air quality	ISO 8573.1 class 2
Compressed air consumer	• Tool changer forward / backward • Tool clamp / unclamp
2.4.1 Compressed air service unit	
Drainage filters	40 µm
Housing	Polycarbonate
max. pressure	1.5 MPa
Recommended oil lubricator for compressed air	ISO VG32
Quantity of oil Lubricator for compressed air	90ml
Dewatering	manual
Compressed air connection thread	3/8"
2.5 Dimensions	
Height	Height during operation up to 2.75 metres Height on transport pallet 2.49 metres
Depth [mm]	 Installation plan on page 27
Width [mm]	
Total gross weight [kg]	2200



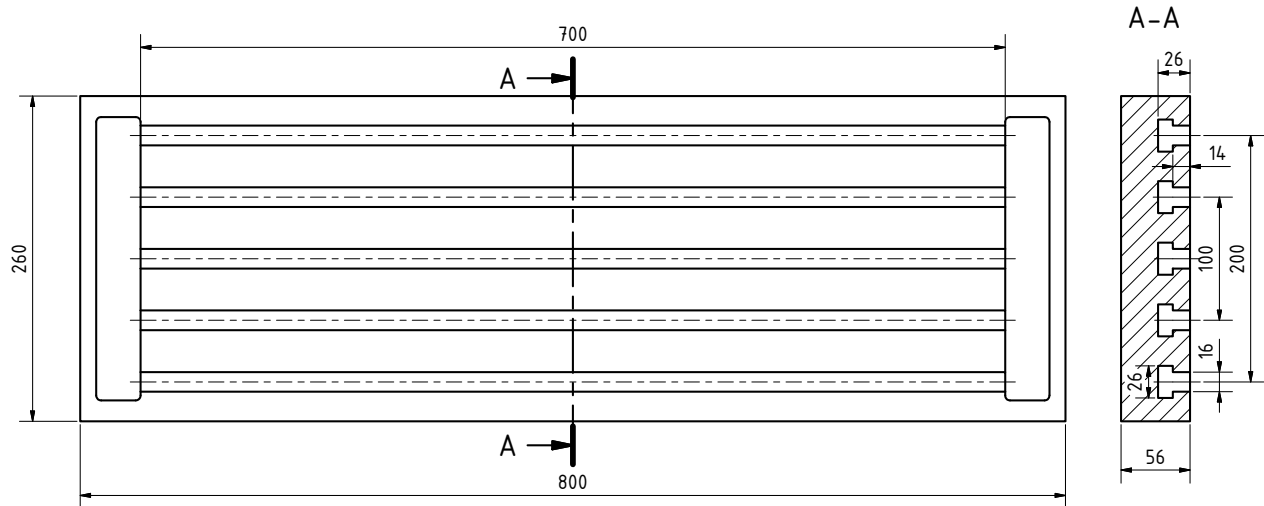
2.6 Working area, traverse path	
X axis [mm]	400
Y axis [mm]	230
Z axis travel [mm]	400
Z axis Distance spindle - milling table [mm]	100 to 550
2.7 Accuracy	
Repeat accuracy	± 0.01mm
Positioning accuracy	± 0.01mm
2.8 Torques of servo motors	
X axis	6 Nm
Y axis	6 Nm
Z axis	11 Nm
2.9 Milling table	
Table length [mm]	800
Table width [mm]	260
T - slot size / number / distance [mm]	16 / 5 / 50
Max. load [kg]	200
max. feedrate [mm/min.]	20000
2.10 Tool changer	
Max. number of tool places	12
max. tooling diameter [mm]	90
Max. tool length [mm]	300
Max. tool weight [kg]	8
Tool change time, chip to chip [s]	15.33
Tool change time, tool to tool [s]	7
2.11 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
2.12 Environmental conditions - storage	
Temperature	5 - 45 °C



2.13 Spindle power Advanced



2.14 Milling table dimensions



2.15 Emissions

The generation of noise emitted by the CNC machine is less than 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 protectors.

We generally recommend the use of noise and ear protection.



2.16 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 rpm - G 6.3
- from a speed of 6000 rpm - G 2.5

according to DIN / ISO 1940.



INFORMATION

Difference SK40 - BT40. The gripper groove for the tool changer is in a slightly different position.

With SK30, the collar is narrower and the gripper groove is closer to the taper. With BT40, the collar is higher and the gripper groove is about 2 - 3 mm further down.

The equipment of the gripper arm of the tool changer must match the SK40 or BT40 tools.





3 Delivery, interdepartmental transport, assembly and commissioning

3.1 Notes on transport, installation, commissioning

Improper transport, installation and commissioning is liable to accidents and can cause damage or malfunctions to the machine for which we do not assume any liability or guarantee.

Transport the scope of delivery secured against shifting or tilting with a sufficiently dimensioned industrial truck or a crane to the installation site.

WARNING!

Severe or fatal injuries may occur if 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.



Note the total weight of the machine. The weight of the machine is indicated in the "Technical data" of the machine. When the machine is unpacked, the weight of the machine can also be read on the rating plate.

Only use transport devices and load suspension gear that can hold the total weight of the machine.

WARNING!

The use of unstable lifting and load suspension equipment that might break under load can cause severe injuries or even death. Check that the lifting and load suspension gear has sufficient load-bearing capacity and that it is in perfect condition.



Observe the accident prevention regulations issued by your Employers Liability Insurance Association or other competent supervisory authority, responsible for your company. Fasten the loads properly.

3.1.1 General risks during internal transport

WARNING: TILTING DANGER!

The machine may be lifted unsecured by a maximum of 2 cm.

Employees must be outside the danger zone, i.e. the reach of the load.

Warn employees and advise them of the hazard.

Machines may only be transported by authorized and qualified persons. Act responsibly during transport and always consider the consequences. Refrain from daring and risky actions.

Gradients and descents (e.g. driveways, ramps and the like) are particularly dangerous. If such passages are unavoidable, special caution is required.

Before starting the transport check the transport route for possible danger points, unevenness and faults.

Danger points, unevenness and disturbance points must be inspected before transport. The removal of danger spots, disturbances and unevenness at the time of transport by other employees leads to considerable dangers.

Careful planning of interdepartmental transport is therefore essential.





3.2 Scope of delivery

INFORMATION

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

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. We would like to ask you to inform us about any other claims within six days upon receipt of the delivery.

Check if all parts are firmly seated.

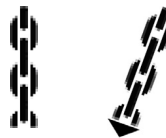


3.3 Load attachment points

○ Centres of gravity



○ Load suspension point (Marking of positions for the load suspension point)



○ Prescribed transportation position (Marking of the top surface)

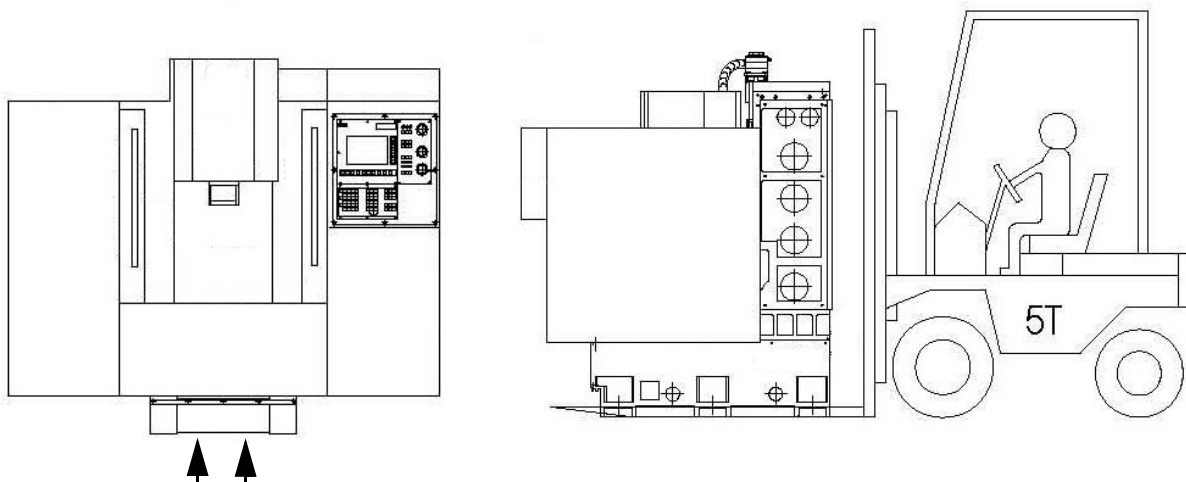


○ Means of transport to be used

○ Weights

- ➔ Check the substructure. The substructure has to bear the load.
- ➔ Dismount the side 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.
- ➔ 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 the load attachment does not cause damage to components or paint.

📖 Machine mounting on page 29



Img.3-1: Transporting by fork-lift truck

3.4 Installation and assembly

3.4.1 Requirements to 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 hindered. 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.

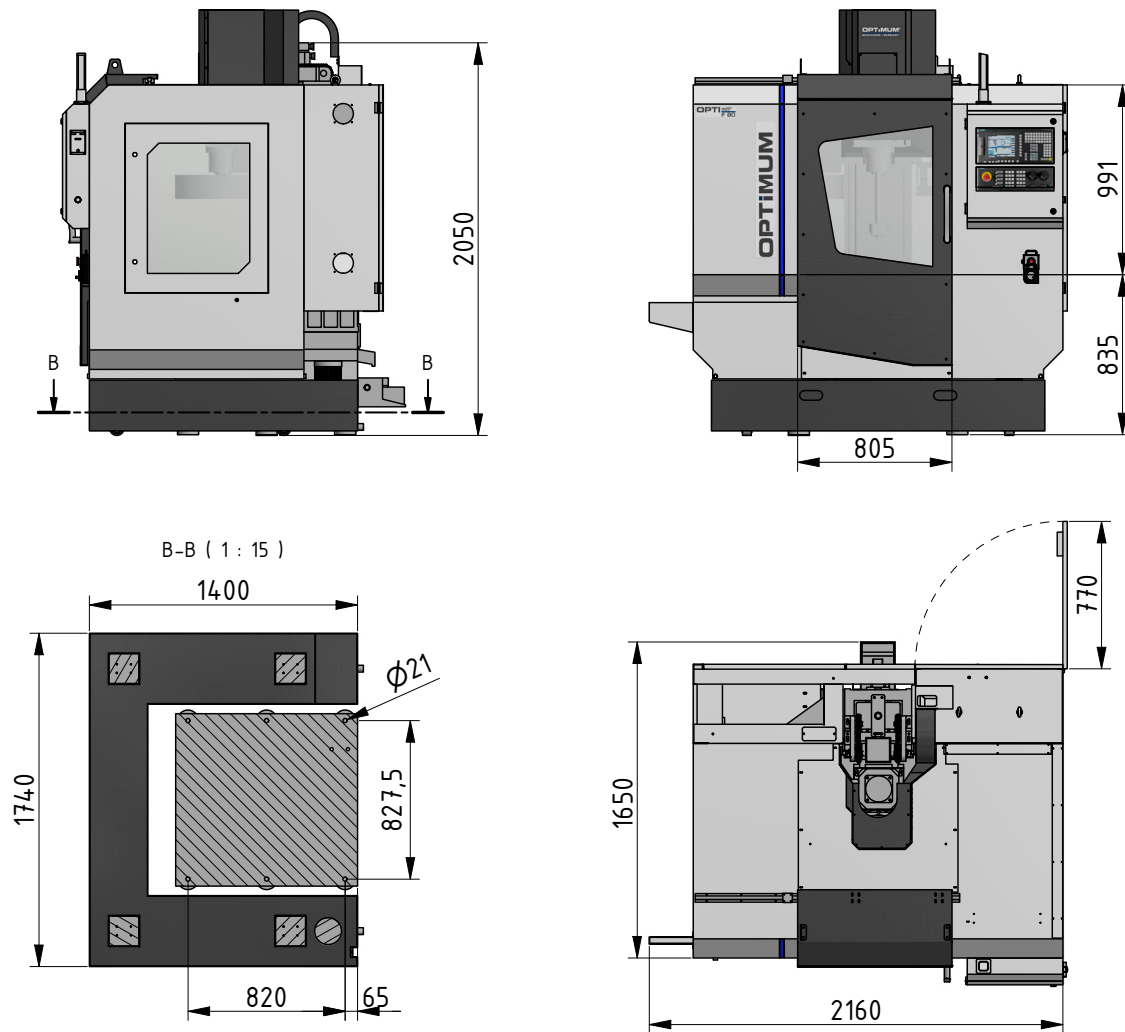
INFORMATION

The main switch of the CNC-machine must be easily accessible.





3.5 Installation plan

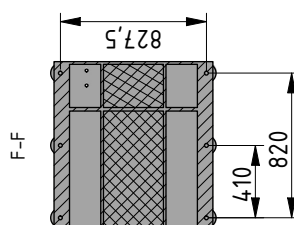
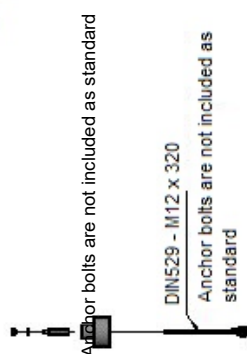
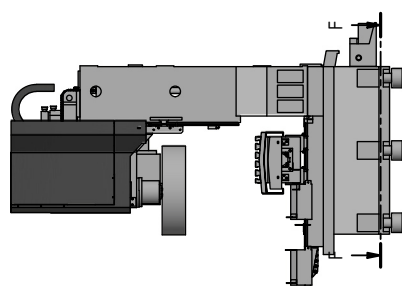
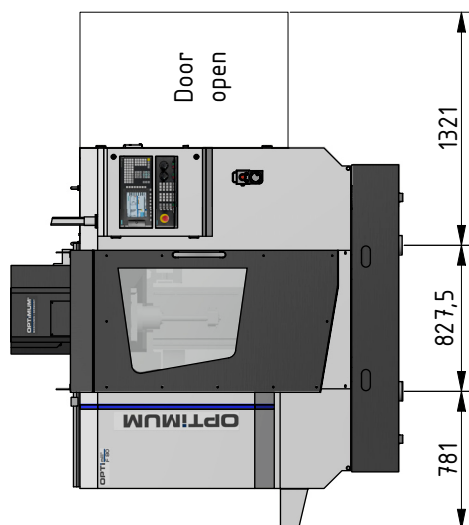
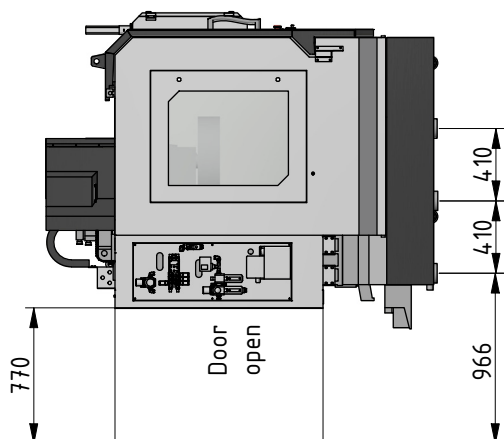


Img.3-2: Installation plan



Anchored assembly

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



Img.3-3: Drawing of the anchoring



3.5.1 Machine mounting

Anchoring-free assembly

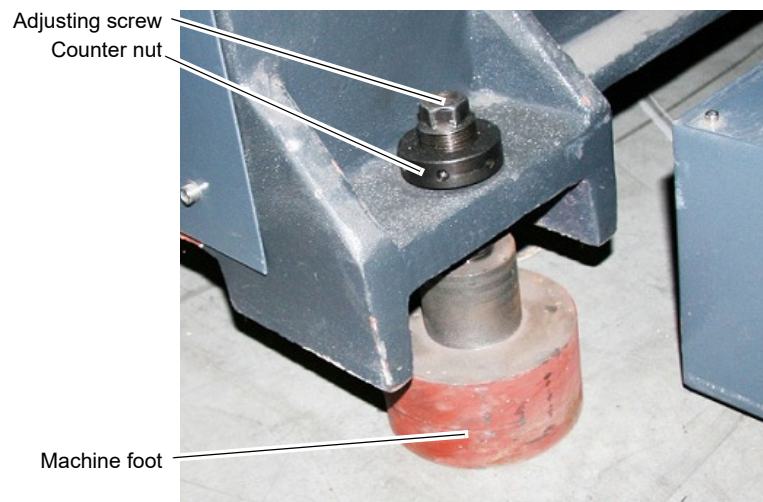
- If required, use levelling vibration- damping elements for the substructure.
- Align the CNC-machine with a machine spirit level.
- Check the alignment of the machine after a few days of usage.

3.5.2 Aligning the machine

- Align the CNC machine on the milling table with a machine spirit level. Use the set screws in order to perform the required height levelling.  Machine mounting on page 29
- The slope deviation of all levels must not exceed 0.03/1000mm.

3.6 Installation and assembly

- Position the machine feet included in the delivery volume below the adjusting screws of the machine substructure.
- Align the CNC-machine with a machine spirit level.



Img. 3-4: Machine feet

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.

3.6.1 Mounting the coolant / lubricant tank

- Push the coolant / lubricant tank below the CNC machine.

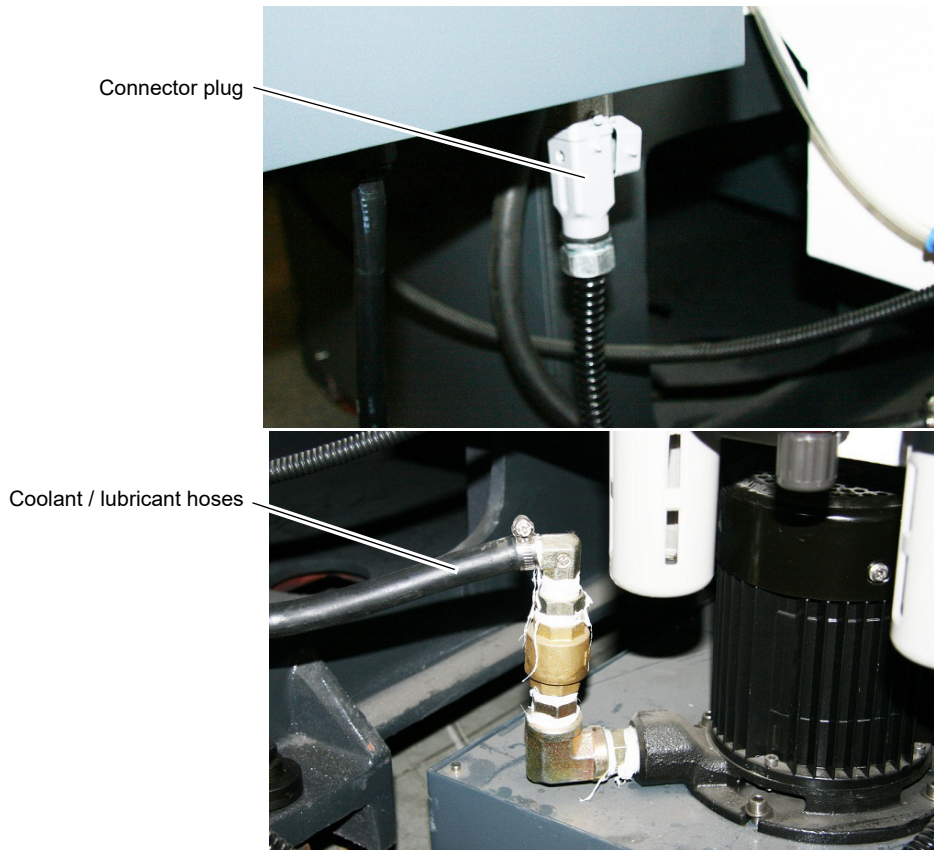


Img. 3-5: Mounting the coolant / lubricant tank

- ➔ Make sure that the coolant / lubricant tank is correctly aligned underneath the CNC machine.

3.6.2 Assembly of the coolant / lubricant pumps

- ➔ Connect the connector plugs of the coolant / lubricant pumps and of the chip conveyor, as well as the coolant / lubricant hoses of the coolant / lubricant tank. Make sure that the plug connections are correctly connected.



Img.3-6: Connector plug

3.6.3 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.

3.6.4 Connection compressed air supply

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



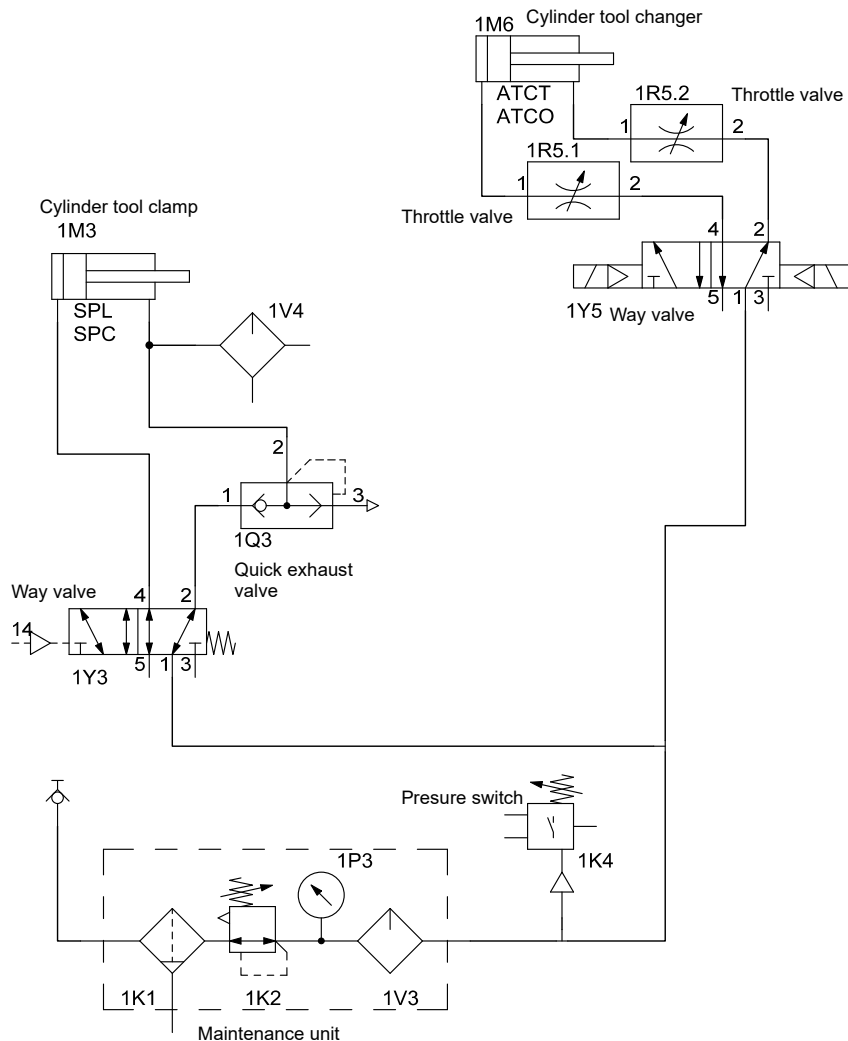
Img.3-7: Compressed air maintenance unit

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.



Pneumatic scheme F80



F80_GB_3.fm



3.7 First commissioning

3.7.1 Fill up coolant

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 76



ATTENTION!

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



INFORMATION

Use a water soluble environmentally compliant drilling emulsion as cooling lubricant procured from the specialised trade.

Make sure that the cooling lubricant is properly absorbed.

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.

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

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

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



3.8 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 lubrication system is depending on the model in the working area on the column of the spindle or at other locations.

→ Fill the tank with lubricating oil via the filler hole.
 Cooling lubricants on page 76



3.9 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:

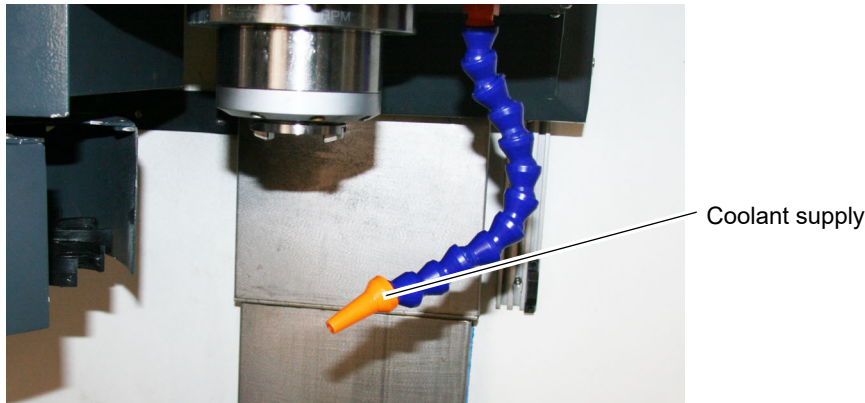
-  Turning on the CNC- machine on page 56
- Close the sliding door and turn on the coolant supply.



F80_GB_3.fm



→ Check if cooling agent flows.



Img.3-8: Coolant supply

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.



3.9.1 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 such as 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.



3.9.2 Save data function

The CNC control only provides a "transient working memory" losing its contents at the latest after 3 weeks.

For this reason it is imperatively necessary to perform a data backup after commissioning the machine respectively the machine control.

The "Save data" function saves the contents of the volatile memory into a nonvolatile memory area.

Requirement: There is no program currently executing.

Do not carry out any operator actions while the data backup is running!

The NC and PLC data are backed up.

To call the saved data, follow these steps:

1. Press the <SELECT> key while the control system is booting.
2. In the setup menu, select "Reload saved user data".
3. Press <INPUT>.



INFORMATION

Data that have been backed up can be called again by selecting "System" > "Start-up" > "Power-up with saved data"!



3.9.3 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.

Due to the design, the leakage current 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.



3.9.4 Regulated drives in connection with residual current devices

Speed-controlled drives are one of the standard equipment in machine and plant construction and perform various tasks. Compared to a simple motor, the electronic rectifiers or converters require some special features for the necessary safety measures for electrical safety. Depending on the application, the use of a fault current protection device, differential current monitoring or insulation monitoring can make more sense.

For electrical safety, DIN VDE 0100-410 (VDE 0100 part 410): 1997-01 "Erection of heavy current installations up to 1000V" is a basic standard. It describes both, the admissible net forms and the necessary protective measures against dangerous body currents. Based on this standard DIN EN 50178 (VDE 0160): 1998-04 "Equipping of heavy current systems with electronic equipment" specifies the protective measures to be applied to controlled drives in more detail. It calls for: "In the case of electronic equipment, the protection of persons against dangerous body currents must be carried out in such a way that a single fault does not cause any danger."

Regulated drives with residual current devices

The TN-S system is the most common network form for the operation of controlled drives. This is done, among other things, for EMV reasons and to avoid vagabonding currents. In accordance with DIN VDE 0100-410 (VDE 0100-410): 1997-01, fault current protective devices (ELCB) can be used as a protective measure against dangerous body currents. According to DIN VDE 0100-482 (VDE 0100 part 482): 2003-06 "Electrical installations of buildings", cables and wiring systems in fire-endangered plants must be protected by ELCBs with a rated differential current of 300 mA. According to IEC 60755, ELCBs differ in the type of fault currents they can detect. In conjunction with electronic devices currents with DC components may occur.

3.9.5 Protection from Dangerous Shock Currents, use of ELCBs

To achieve increased safety in all installation systems, and in power supply ranges for which the installation provisions stipulate or recommend the ELCB devices.

Measure for "Protection from Dangerous Shock Currents", as regulated in DIN VDE 0100 Part 410. All measures are to be mentioned:

- Protection from indirect contact – as protection against fault by shutting down in the event of inadmissibly high contact voltage by short circuit shock on the operating resource.
- Protection from direct contact – as additional protection by shutting down in the event of contact with a live conductor. Dangerous shock currents are shut down within the shortest possible time, if the rated fault current of the circuit breaker is 30 mA (e.g. Domestic environment), for a personal protection system 10 mA (e.g Bathroom).
- Fire prevention – Prevention of the origination of electrically-ignited fires if the rated fault current of the circuit breaker is 300 mA. Operating premises at risk of fire to VdS 2033: 2002-02 300 mA (e.g. Factory halls).

3.9.6 Current in the protective earth conductor - Leakage current

With EMC filters in frequency converters, the leakage current is always greater than 3.5 mA due to physics. Some types of frequency converters also achieve a leakage current of up to 300mA.

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.

Preferably, machines with frequency converters are therefore to be permanently connected to a terminal box, otherwise an additional fixed earth connection is required, which is not routed over the plug, and must correspond to at least the cross-section of the cable in the plug.

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:



To avoid an operating fault, you need an AC/DC-sensitive ELCB. Be absolutely sure which leakage current security is necessary for dangerous body currents, as regulated in DIN VDE 0100 part 410, at your mains connection.

3.9.7 When the ELCB triggers

- Pulse current - sensitive ELCB type A
ELCB type A independent of rated voltage, for triggering when changing fault currents and pulsing DC fault currents. 
- AC/DC - sensitive ELCB type B
ELCBs of series type B also accept the detection of smooth AC fault currents as well as the detection of fault current shapes of type A; they are therefore suitable for all the circuits mentioned. ELCBs of this series therefore detect all types of fault current according to the triggering characteristic B, i.e. both smooth DC fault currents and also all AC fault currents of all frequencies and mixed frequencies up to 1 MHz are detected and switched off reliably in the event of a fault.  
- Alternating current - sensitive ELCBs of type AC (only alternating current) are unsuitable for frequency converters. Alternating current - sensitive ELCBs of type AC are not customarily used and are no longer permitted in Germany. 

Type B must be used with 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.



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 (α) to the defined axis. The angle (α) 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.

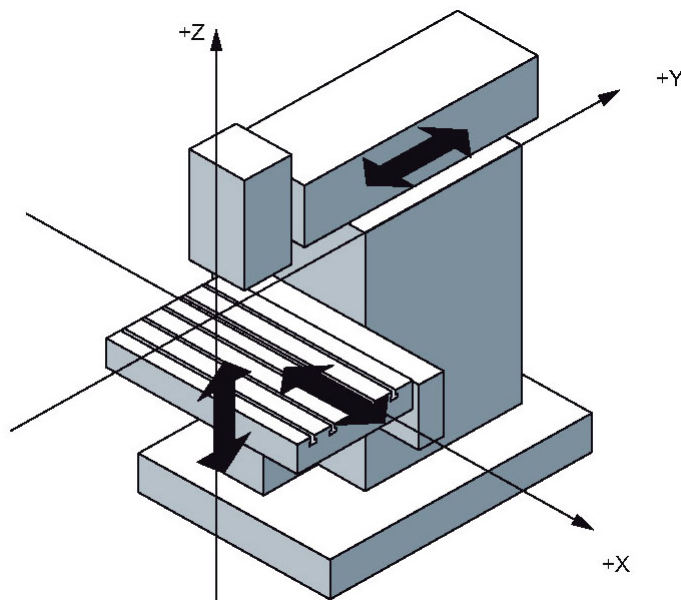
Turns from +X to +Y are created in +Z direction, which result in a right-handed screw.

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

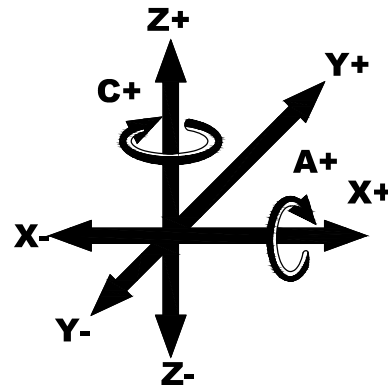
Rotary axis: A B C

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.



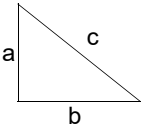
On a rectangular triangle the right angle is described by an quarter circle and a point in the angle.

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:

$$a^2 + b^2 = c^2$$


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:

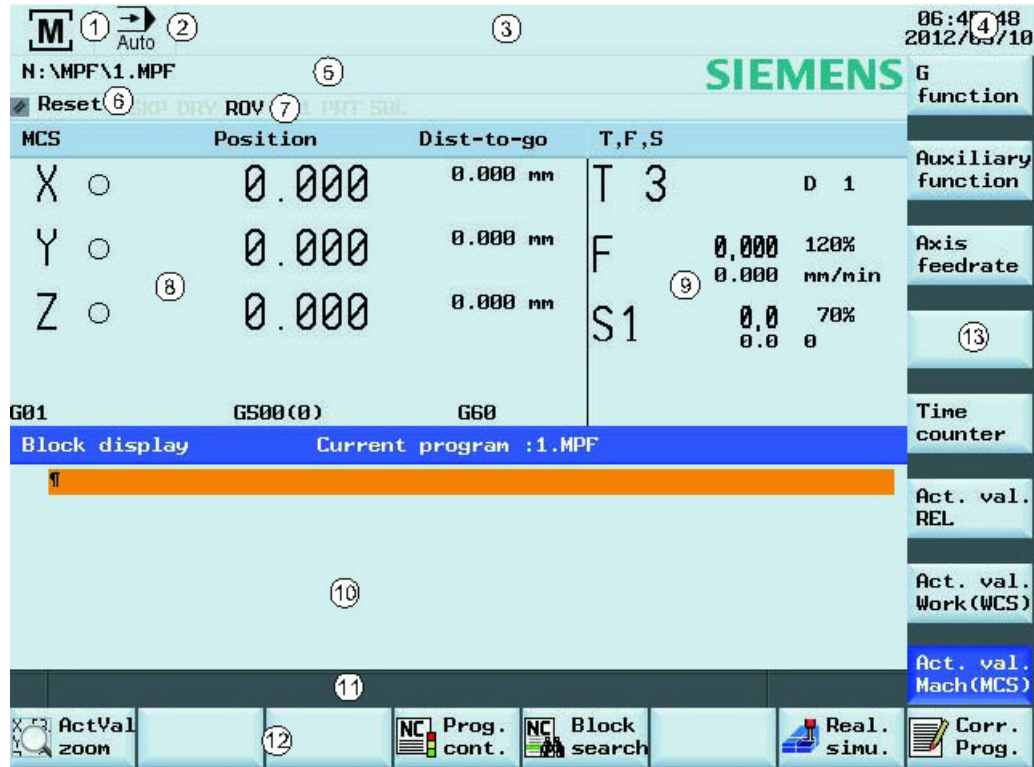
$\text{opposite leg} = \text{adjacent leg} \times \tan \alpha$



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 layout

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

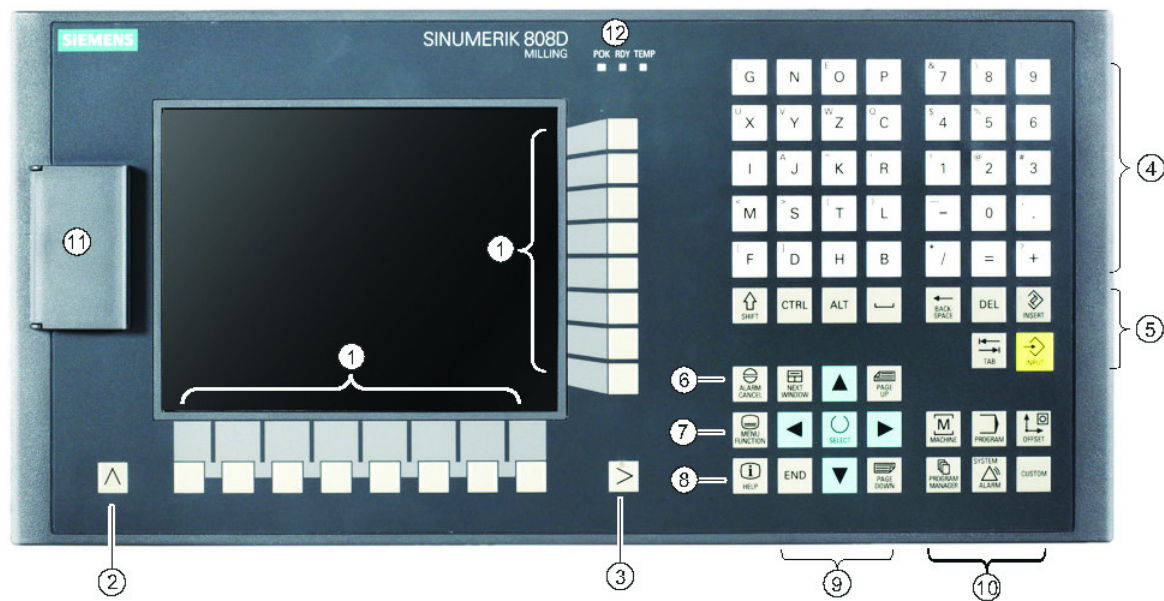
- ⑧ Actual value window
- ⑨ T, F, S window
- ⑩ Operating window with program block display

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

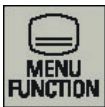
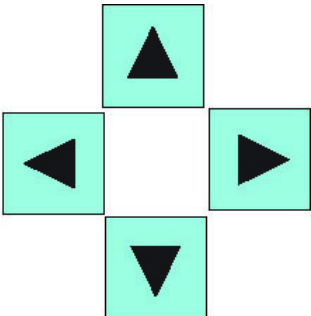
Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Description	
①		Vertical and horizontal softkeys Calls specific menu functions
②		Return key Returns to the next higher-level menu.
③		Menu extension key No function is assigned to this key. Reserved for future use.
④		Alphabetic and numeric keys 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
		Backspace key Deletes a character selected to the left of the cursor.
		Delete key Deletes the selected file or character.
		Insert key
		Tab key • Indents the cursor by several characters. • Toggles between the input field and the selected program name.
		Input key • Confirms your entry of a value. • Opens a directory or program.
⑥		Alarm cancel key Cancels alarms and messages that are marked with this symbol

F80_GB_5.fm



Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
		Description
⑦		Menu function key Opens the wizard main screen.
⑧		Help key Calls the context-sensitive help for the selected window, alarm, message, machine data, setting data, or end-user wizard.
⑨	Cursor keys	
		Cursor keys up/down/left/right keys
		Next window key No function is assigned to this key. Reserved for future use.
		End key Moves the cursor to the end of a line.
		Page up key Scrolls upwards on a menu screen
		Page down key Scrolls downwards on a menu screen
		Selection key • Toggles between entries in the input field. • Enters the "Set-up menu" dialog at NC start-up.



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: • Connects to an external USB memory sticker to transfer data between the USB sticker and the CNC. • Connects to an external USB keyboard for use as an external NC keyboard.
⑫	Status LEDs 	LED "POK" Lights up green: The power supply for the CNC is switched on.
		LED "RDY" Lights up green: The CNC is ready for operation.
		LED "TEMP" 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> + 	Opens the "Program" operating area
<ALT> + <M>	Opens the "Alarm" operating area
<ALT> + <N> <SHIFT> + 	Opens the "System" operating area
<ALT> + <H>	Calls the online help system.
<ALT> + <L>	Enables input of lowercase letters.
<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> + 	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 Elements on the MCP



Img.5-3: Machine Control Panel

Elements on the MCP	
Element	Function
	Drive voltage Push-button with indicator light - Indicator light ON, drive voltage activated - Indicator light OFF, drive voltage deactivated

Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
①		<Emergency stop button> Activate the button in situations where - life is at risk. - there is the danger of a machine or workpiece being damaged. All drives will be stopped with the greatest possible braking torque.
②		Handwheel key (with an LED status indicator) Controls the axis movement with external handwheels.

F80_GB_5.fm



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

	Element	Description
③		Tool number display 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)	
		Program test key 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.
		Conditional stop key Stops the program at every block in which miscellaneous function M01 is programmed.
		Rapid override key Adjusts axis feedrate override
		Single block key Activates single block execution mode
⑥	User-defined keys (all with LED status indicators)	

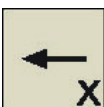


Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
		Lamp control key Pressing this in any operating mode switches on/off the lamp. LED on: The lamp is switched on. LED off: The lamp is switched off.
		Coolant control key 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.
		Safety door control key 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.
		Clock wise magazine rotation (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.
		Reference point approach of the magazine (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.
		Counterclockwise magazine rotation (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.
		Forward rotation of the chip conveyor (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. INFORMATION The F80 does not have a chip conveyor. Therefore, the key has no function. 

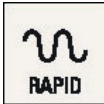
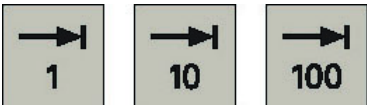
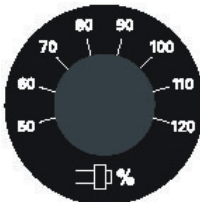


Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"

	Element	Description
		Reverse rotation of the chip conveyor (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. INFORMATION The F80 does not have a chip conveyor. Therefore, the key has no function.
		User-defined keys
		Position of tool changer The <K9> key lights up when the tool changer is extended. In the reference mode, the key can be pressed to move the tool changer out. The milling spindle must be located above the tool changer.
		Position of tool changer The <K10> key lights up when the tool changer is in the basic position. In the reference mode, the key can be pressed to move the tool changer to the basic position. The milling spindle must be located above the tool changer.
		Manual tool change Enables or disables the manual tool change. The manual tool change is only possible with the sliding door open.
⑦	Axis traversing keys	
		X axis key Traverses the X axis in the positive direction.
		X axis key Traverses the X axis in the negative direction.
		Z axis key Traverses the Z axis in the negative direction.



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

	Element	Description
		Z axis key Traverses the Z axis in the positive direction.
		Y axis key Traverses the Y axis in the positive direction.
		Y axis key Traverses the Y axis in the negative direction.
		Rapid traverse overlay key 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
		Spindle speed override switch Makes the spindle rotate at the specified speed override.

F80_GB_5.fm



Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
⑨	Program state keys	
		Cycle stop key Stops the execution of NC programs
		Cycle start key Starts the execution of NC programs
		Reset key Resets NC programs Cancels alarms that meet the cancel criterion.
⑩		Feedrate override switch 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 machine data.



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.

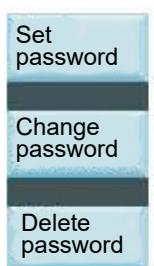


5.4.2 Change passwords

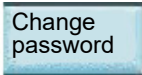
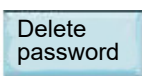
Step 1

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

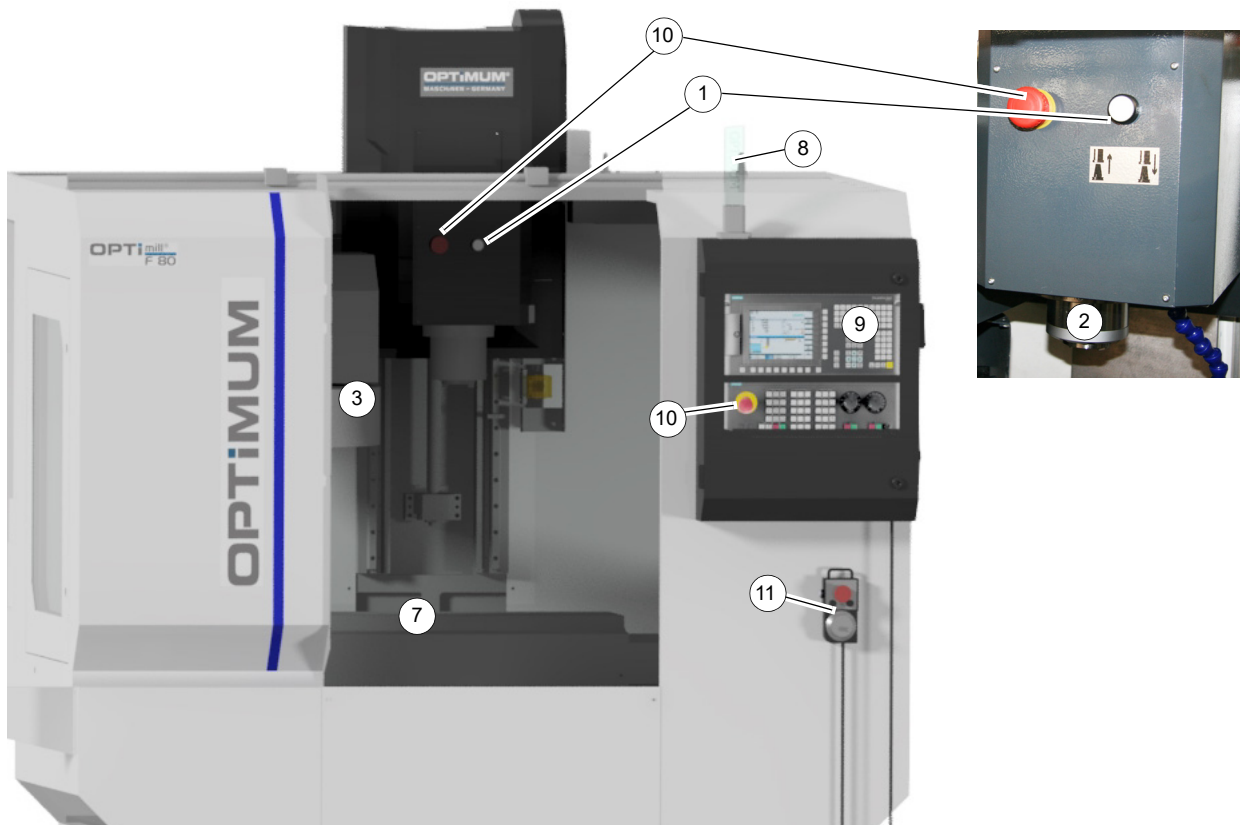
Commission the CNC-machine only under the following conditions:

- 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 16

6.2 Control and indicating elements



Img.6-1: F80

No.	Designation
1	Touch-switch to release and clamp the tool.
2	Milling spindle
3	Tool change system incl. magazine disk
7	Milling table
8	Signal lamp (if the signal lamp is illuminated --> CNC- program is running)
9	Machine Control Panel
10	Emergency-stop push button
11	Handwheel for manual travelling with emergency stop push button and acknowledgement button



Signal lamp

	Pos. No	Colour	Designation
	①	Red	Lights up when activity the emergency stop push button
	②	Orange	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 allowed in the operating mode "JOG" and in the operating mode "MDA". 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. 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:

By means of the CAD/CAM program (for instance a 3D-CAD program including downstream co-processor) construction data are transmitted (semi-)automatically to an executable program. For this kind of programming a 3D model is designed using a PC. By means of an operating sequence which is predefined by the user the motion-sequence of the machine are being calculated. These programs are accessing 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 complete programs within shortest time without having any knowledge of the individual program commands and its 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 13
- Please wait until the control is completely started.
- Press the pushbutton "Drive control ON" (1).
- Unlock the "Emergency-Stop push button" (2).
 - on the spindle head (not visible in the picture),
 - on the MCP,
 - on the electronic handwheel (not visible in the picture),
- 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

Reference point travel is not required, your machine is configured with ABS encoders (808D ADVANCED).



6.5.3 Using the electronic handwheel

The electronic handwheel can always be used when

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



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

	Pos. No	Designation	Description
	①	Rotary switch	Selector switch to control the individual axis.
	②	Rotary switch	Selector switch for the feed speed (3 stages).
	③	Hand wheel	Handwheel to travel the individual axes.
	④	<Emergency stop button>	The emergency stop push button switches off the CNC- machine.
	⑤	Push button resp. acknowledgement button	In the setting mode it is necessary to actuate the acknowledgement button in order to expressly allow the movement of the individual axes.

WARNING!

A manual movement of axes with open door is not possible. The F80 has no acknowledgement button to allow movement of axes with open slide door.

The lock switch on the slide door may only be unlocked for maintenance and repair work.





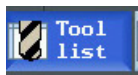
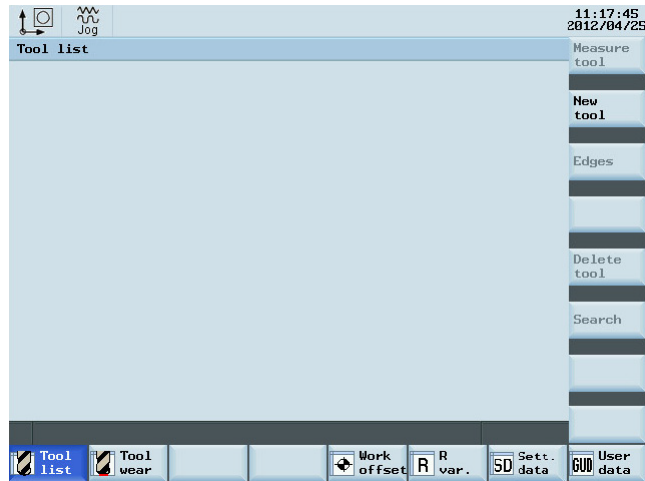
6.5.4 Inserting a tool

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.
- Check that the LED on the pushbutton <K11>lights up.
Press - if necessary - the pushbutton <K11>. The manual tool change is thereby enabled again.
- Press the pushbutton "manual tool change" to loosen or to clamp the tool.



Pushbutton manual tool change

Img.6-3: Spindle head



6.5.6 Tool changer

The tool disc is rotated by an electric motor and the tool is replaced by a pneumatic cylinder.

The tool change is called up in your program. Load tool into Spindle on page 124

If the machine is left idle for a long time without compressed air, the tool changer can move to an undefined position. The pneumatic cylinder has an end position damping and can thereby move out of the end position.

see also User-defined keys on page 50

see also Tool changer problem on page 74

6.5.7 Machine parameters of the tool changer

INFORMATION



For service personnel only.

- 14514 [1] The tool change ready position of the Z axis.
- 14514 [2] The tool change position of the Z axis.
- 14514 [3] The speed of reaching the tool change position in the Z axis.
- 14514 [4] The speed of reaching the tool change ready position in the Z axis.
- 14510 [20] The maximum number of tools.

6.5.8 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 allowed in the operating mode "JOG" and in the operating mode "MDA". 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. 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:

By means of the CAD/CAM program (for instance a 3D-CAD program including downstream co-processor) construction data are transmitted (semi-)automatically to an executable program. For this kind of programming a 3D model is designed using a PC. By means of an operating sequence which is predefined by the user the motion-sequence of the machine are being calculated. These programs are accessing 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



complete programs within shortest time without having any knowledge of the individual program commands and its 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 Start 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 programs you have to close the sliding door of the separating protective equipment.

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



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 solids 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 interface of the control within 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 nearby the drive apparatuses of the NC axis or the frequency converter itself. Errors occur during data transfer due to electromagnetic radiation (EMC-problems).

The control is equipped with the following data interfaces. They are located laterally on the control panel of the CNC-maschine:

- RJ45 plug- in connection
- 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⁻¹ (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 conventional 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] Peripheral and side milling cutters	Steel 10 - 25 m/min	Grey cast iron 10 - 22 m/min	Al alloy age-hardened 150 - 350 m/min
	Speed [min ⁻¹]		
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 ⁻¹]		
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 ³⁾
		2	3	4	5	6	7	8	9	10	
Steel, unalloyed, up to 600 N/mm ²	n ¹⁾	5600	3550	2800	2240	2000	1600	1400	1250	1120	E
	f ²⁾	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 ²	n ¹⁾	3150	2000	1600	1250	1000	900	800	710	630	E/oil
	f ²⁾	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 ²	n ¹⁾	2500	1600	1250	1000	800	710	630	560	500	Oil
	f ²⁾	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 ² e.g. X5CrNi1810	n ¹⁾	2000	1250	1000	800	630	500	500	400	400	Oil
	f ²⁾	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

High temperatures are generated at the tip of the tool by the occurring friction heat. 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 environmentally compliant emulsion as cooling agent procured from the specialized trade.



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.





6.11.3 Clamping workpiece and setting workpiece datum

ATTENTION!

Clamp your workpiece safely and firmly on the milling table using appropriate clamping devices. When setting and using already created programs observe the safety clearance in order to avoid collisions with the selected clamping means.

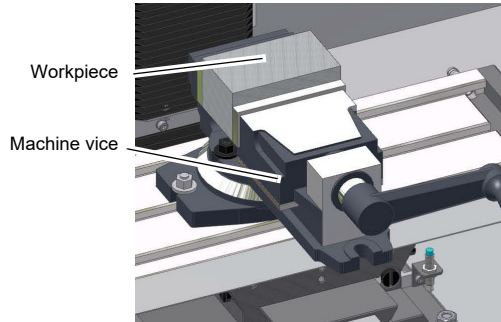


WARNING!

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

- Use a 3D calliper or another appropriate measuring device in order to determine the workpiece zero point.
- Travel the X and Y axis using the handwheel until the 3D measuring probe indicates the value "0".



Img.6-5: Determine workpiece datum X axis resp. Y axis

Measuring the Z axis by means of a pressure sensor

- Change a tool with a length of "0" listed in the tool list
- Travel the spindle (Z axis) to the pressure sensor (by means of the handwheel) until the indicated value is "0".



Img.6-6: Workpiece datum Z axis with pressure sensor

A resulting offset of the machine zero point to the workpiece zero point is determined in the X, Y and Z axes and entered in the adjustable zero offset.

This offset is activated in the NC programme during programme run, for example with a programmed G54 command.

Further information on this can be found in the Programming and Operating Manual for the Siemens 808D controller under the Setup menu item.



6.12 DRB central lubrication system

6.12.1 Technical specification

Type	Code	Nominal pressure	Displacement	Power	Voltage	Capacity	Control systems	Distributor
		MPa	ml / cycle	W	VAC	L		
DRB-215A	12215-1A	2.5	0.08	15	220V	2	none	BJ
	12215-2A							DPQ
	12215-3A						WGKX-3	BJ
	12215-5A						WGKX-4	BJ
	12215-6A						WGKX-5	DPQ
DRB-225A	12225-1A	2.5	0.08	15	220V	4	none	BJ
	12225-2A							DPQ
	12225-3A						WGKX-4	BJ
	12225-4A						WGKX-5	DPQ
DRB-215Y	12215-1Y	2.5	0.08	15	220V	2	none	BJ
	12215-2Y							DPQ
	12215-3Y						WGKX-1	BJ
	12215-4Y							DPQ
DRB-315Y	12215-1Y				380V		none	BJ
	12315-2Y							DPQ
DRB-225Y	12225-1Y			25	220V	4	none	BJ
	12225-2Y							DPQ
	12225-3Y						WGKX-1	BJ
	12225-4Y							DPQ
DRB-325Y	12325-1Y				380V		none	BJ
	12325-2Y							DPQ

A, stands for the cover of the electrical box made of plastic, Y for the cover of the electrical box made of die-cast.
Distribution Note: BJ - measuring unit (external pressure switch) ; DPQ - quantity distributor

Factory setting	Make a note of the factory setting before making a change
Running time [seconds]	
Interval Time [minutes]	
Operating mode WGKX-1	C000
Pressure setting	2.5 MPa

- Running time = time during which the pump supplies the lubrication points.
- Interval Time = Pause between runtimes.
- Pressure setting = mechanically preset setting on the overflow valve.



6.12.2 Function

Your machine is equipped with a central lubrication system. During operation, the supply points are automatically supplied with oil at intervals via the integrated controls.

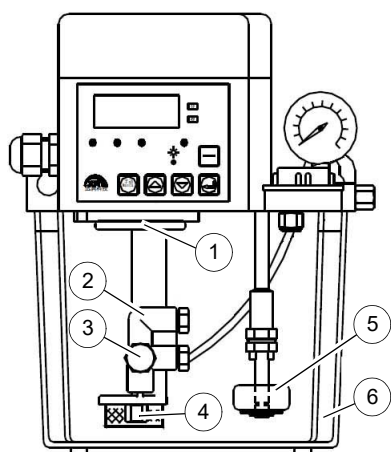
The lubricating system is used to maintain an oil film on the slideways, the bearings, the ledges and the ball screws and to reduce their wear.

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.

The system consists of lubrication with a pump unit, a distributor system, feed pipes and a float switch.

Your CNC machine is equipped with a DPQ - quantity distributor.

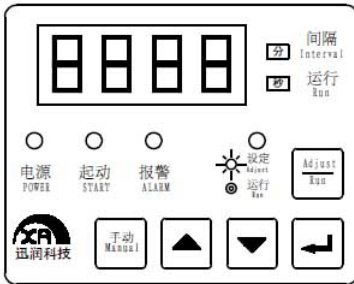
- The supply of the lubrication points by the pump is carried out at specified time intervals by the integral controls.
- An overflow valve ensures an even flow of oil to the machine's distribution system. The distribution system in the machine ensures the correct amount of oil to each consumer.
- A float switch monitors the level and gives a signal if the level is too low and also an acoustic signal.



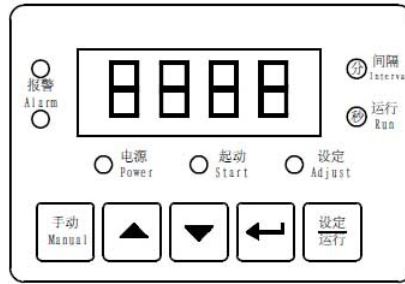
- ① Single-phase drive motor
- ② Gear pump
- ③ Pressure relief valve
- ④ Overflow valve
- ⑤ Float switch
- ⑥ Reservoir

6.12.3 Adjusting the amount of lubrication oil at the central lubrication

WGKX-1 Controller

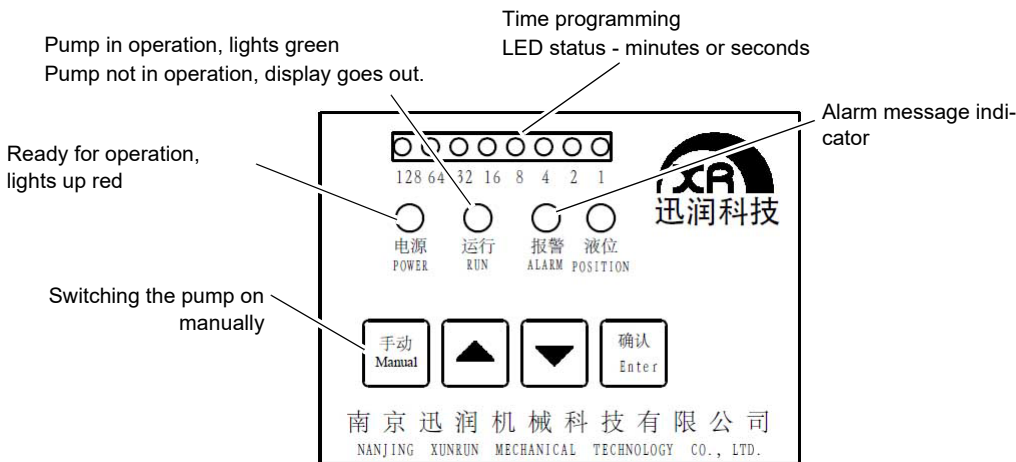


DRB-215Y Control panel



DRB-225Y Control panel

6.12.4 WGKX-4 and WGKX-5 Controller



Setting of parameters

Depending on the control system used, the CNC machine is delivered with the operating mode C000.

Three operating modes C000, C001 and C002 can be selected on the WGKX-1 controller for different applications.

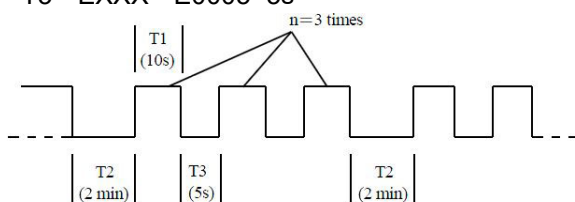
Operating mode C000 = pumps as running time and interval time

Operating mode C001 = pumps as running time with external pressure switch. When the pressure has reached the set value in the set time, the pressure switch is activated, the motor stops after 4 seconds. If the pressure has not reached the set value in the set time, the buzzer sounds and "EEEE" flashes on the display. This indicates that leakage oil is escaping in the lubrication line.

Operating mode C002 = cycle operating mode.

Example cycle operating mode C002:

- Running time T1=10s
- Interval time T2=2min
- n="d XXX"=d003=3 times
- T3="EXXX"=E0005=5s





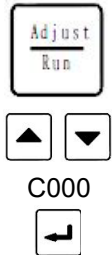
6.12.5 Setting the operating mode on WGKX-1

If the **WGKX-1** control unit is used on your machine.

The central lubrication operating mode for your CNC machine must be set to C000.

If necessary, the operating mode on the WGKX-1 controller is changed as follows.

- ➔ Disconnect the complete machine or only the central lubrication system from the power supply.
- ➔ Restore power to the complete machine or only to the central lubrication system and press the <Adjust/Run> key. At this moment the adjustment lamp lights up and the interval lamp flashes.
- ➔ Use the cursor buttons to select the "C000" operating mode and confirm with the <Enter button>.



6.12.6 Setting the running time and interval time

- Determine the status of the setting.
Press the "Enter" key and ▲ or ▼ simultaneously, the LED and the running light will now flash.
- 1. Changing the Interval Time (minutes) setting.
Press the buttons ▲ ▼ to change the interval time and confirm the change with "Enter".
- The set interval time in minutes is shown by the LEDs.
- Press the "Enter" key.

○ ○ ○ ○ ○ ○ ○ ○ LED
128 64 32 16 8 4 2 1 minutes

○ ○ ⊗ ○ ⊗ ⊗ ○ ○ LED on = 32 + 8 + 4 = 44
128 64 32 16 8 4 2 1 minutes

- 2. Changing the setting of the running time (seconds).
Press the buttons ▲ ▼ to change the runtime and confirm the change with "Enter".
- The set running time is now displayed in seconds by the LEDs.
- Press the "Enter" key.

All parameters are stored by pressing the "Enter" key.

- The setting for the WGKX-4 controller is completed and the controller starts to operate.

In the sequence of operations, the WGKX-5 controller now expects the input of the operating mode setting. The operating mode is set by selecting the state of the yellow LED and confirmed with the "Enter Button".

Selection of the operating modes on WGKX-5:

- without pressure alarm - yellow LED is permanently on The central lubrication system emits an acoustic signal, the alarm LED lights up if the reservoir level is too low.
- with pressure alarm - yellow LED flashes The central lubrication system emits an acoustic signal and the alarm LED lights up if the pressure is not reached within the specified time.



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	Positioning the tool magazine to spindle position.
M22	Positioning the tool magazine to original position.
M23	
M24	
M25	Clamping tool
M26	Releasing tool
M27	
M28	

No.	Function
M29	
M30	
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	

M-functions_808-milling-machines_GB.fm



No.	Function
M62	
M63	
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	

No.	Function
M97	
M98	
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

8.1 Malfunctions on the machine

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	User alarm 04
700005	Main power three phase direction error
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
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



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



8.2 Problem- solving

8.2.1 Tool changer problem

CAUTION!

The milling spindle must be located above the tool changer, otherwise the machine will damage the machine as described below.



Cause:

The tool changer can not be moved in the reference mode with the user-defined keys <K9> and <K10> although the milling spindle is located above the tool changer.

Solution:

- ➔ Move the milling spindle (milling head) upwards, above the tool changer.
- ➔ Operate the pneumatic valve of the tool changer manually by means of the red pushbuttons.



Img.8-1: pneumatic valve - tool changer



9 SINUMERIK 808 D Advanced

The complete Sinumerik 808D documentation consists of the following manuals, which can be downloaded from the Siemens website.

<https://support.industry.siemens.com/cs/products?dtp=Manual&mfn=ps&pnid=14581&lc=en-WW>

Manuals are strictly necessary for the operator, CNC programmer and also for the maintenance and repair personnel of the 808D Advanced.

- Programming and operating manual (milling)
6FC5398-4DP10-0AA1 - (808D_ADVANCED_OPM_0114.....pdf)
- Programming, turning and milling in ISO code
6FC5398-0DP40-0AA0 - (SINUMERIK_808D_ADVANCED.....pdf)

Manuals that are required for the maintenance and repair personnel.

- Service manual (808_TGSH_0712....pdf)
- Diagnostic manual (808D_Diagnostics_Manual....pdf)
- Function manual (808D_Function_Manual_0512_.....pdf)
- Commissioning manual (808_TGIH_0712_.....pdf)

Additional useful manuals and information

- Training Manual Operation and Programming Milling (808_TGOPM_0712....pdf)
- PLC manual for subroutines (SINUMERIK_808D_PLC_Subroutines_.....pdf)
Describes, for example, how to establish a connection with 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 manual
- Parameter Manual

The operator's manual can be downloaded in PDF format - also in other languages - from the Siemens website. With the date of issue of this operating manual, the programming and operating manual - 808D Advanced is currently available in German, English, Polish, Russian, Portuguese, Bulgarian, Korean and Chinese.

<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

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



Clean up any spill liquid or oils immediately using proper oil-absorption methods and dispose of them in accordance with current legal requirements on the environment.

Collect leakages

Do not re-introduce liquids split outside the system during repair or as a result of leakage from the reserve tank: collect them in a collecting container to be disposed of.

Disposal

Never dump oil or other substances which are harmful for the environment in 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.

10.1 Operating material

10.1.1 Machine lubricants

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

Recommended lubricant class: ISO V668

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

10.1.2 Cooling lubricants

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

INFORMATION

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

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

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





When using non water-mixable cooling lubricants (oil content > 15%) with flashing point 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 which are 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 supplier for mineral oils.



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 need to comply with the current regulations of the law and of the employers liability insurance association.
- Request your manufacturer for cooling lubricants to submit you the documentation for the products such as the product description VKIS and EC safety data sheet. The EC safety data sheet gives you information about the water-hazard class.

They need to be environmentally friendly and working place-friendly. Thus, they need to be free from nitrite, PCB, chlorine and nitrosinable diethanolamin (DEA), according to TRGS 611.

- The manufacturer should be able to submit a certificate concerning skin-tolerance.
- The mineral 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 not sticking 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 disperse as possible in order to avoid clogging-up at 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 staff 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 15

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. For this reason, all the intervals are only valid for the authorised conditions.

Interval	Where?	What?	How?	Check ✓
Start of work, after each maintenance or repair work	CNC- milling machine	🔧 Safety check on page 15		
	Sight window	Cleaning	Clean the sight windows made of polycarbonate using a suitable cleaning agent.	
	compressed air supply	Draining	🔧 Drain compressed air service unit on page 82	

F80_GB_10.fm



Interval	Where?	What?	How?	Check ✓
Start of work, Daily after every maintenance or repair work	Lubricant tank Slideway oil	Filling level control	Manually Check the quantity and refill the lubricant tank of the central lubrication system. Refer to Cooling lubricants on page 76	
	Oil collecting tank	Empty	Check the oil level in the oil collecting gutter. Empty the oil collecting tank.	
Every week	CNC- milling machine	Save data function on page 33		
Every week		Oiling Lubricating	Oil all bare steel surfaces. Use an acid-free oil, e.g. weapon oil or motor oil. Press the push-button of the central lubricating system.	
Every week	Cooling lubricant tank	Status control of filling level	Check for liquid level, concentration, pH value, bacteria and fungal decay.	
every week	Cooling lubricant	pH value measuring	Check the ph-value.If required, replace the cooling lubricant.	
Every week	Drip feed lubricator Compressed air cylinder	Filling 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.	

F80_GB_10.fm



Interval	Where?	What?	How?	Check ✓
Annually	Clutch Coupling Drive Z-, Y-, X-axis	Check Replacing	Check if coupling is worn and check coupling slack. If necessary, replace the coupling.	
Annually	Pressure intensifier for tool change	Checking Refill	The pressure intensifier for clamping and releasing the tool (compressed air cylinder with oil reservoir) is equipped with an oil reservoir on the side. The fill level of the oil reservoir must be checked and topped up if necessary.	
Every 1000 operating hours	Wipers on the slides	Check Replacing	Check the wipers on the slides. Replace if damaged immediately.	
	Switch cabinet	Cleaning	👉 Cleaning electrical cabinet on page 82	
	Servo motors	Inspection	Connections on the servo motors.	
Every six months	Tool changer	Check the tool change function. Positioning	By manual changing	
As required	Chip collection tray	Cleaning	Clean the chip collection tray of the cooling lubricant equipment.	



Interval	Where?	What?	How?	Check ✓
60 months	Protective covers	Replacing the sight window	☞ Cleaning and replacing of the polycarbonate windows on page 81	
the latest after 3 years		The service life off the interlock switch on the sliding door may have been reached due to the operating conditions. Replacement is required.	By the service technicians ☞ Customer service technician on page 83	
after 3 years		The service life off the interlock switches on the side doors may have been reached due to the operating conditions. Replacement is recommended to ensure further, fault-free operation.	By the service technicians ☞ Customer service technician on page 83	

10.3.1 Cleaning and replacing of the polycarbonate windows

WARNING!



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

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

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

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

We recommend to replace the polycarbonate windows of the door after 60 months of 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 cooling agent in the composite structure,
- destroyed or damaged window (coating) on the workspace or operator position.

10.3.2 Cleaning 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.

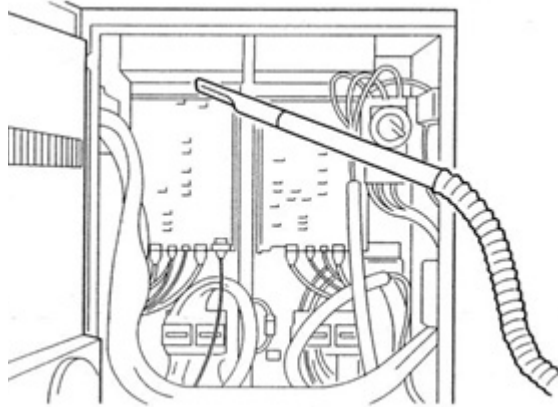
Accumulation of foreign particles on the printed circuit boards or other electronic components could cause machine malfunction.

Clean the inside of the electrical cabinet regularly.

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

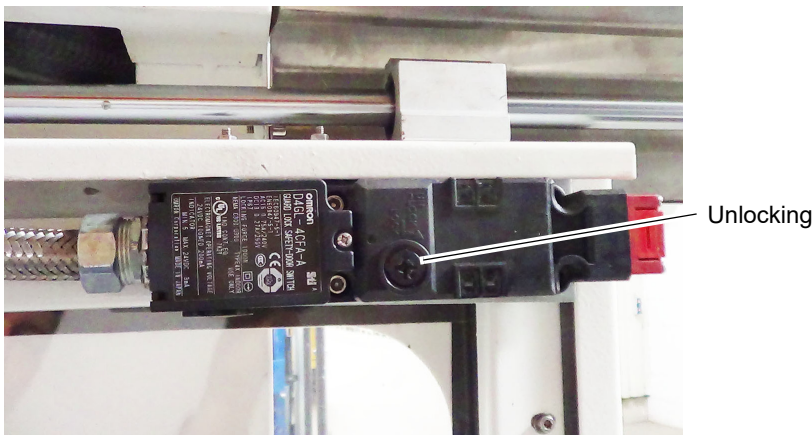
Never touch printed circuit boards or parts around the connector. Also avoid shock on these parts.

It is recommended to clean the electrical cabinet 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 15



Img. 10-1: Interlock switch sliding door

10.4.1 Drain 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 cause damage.

Check the water level daily and drain the water level in the filter when needed.



10.5 Recommended wear and tears parts

Polycarbonate viewing panes
Interlock switch on the sliding door
Interlock switches on the side doors
Stripper on the guide tracks

10.6 Repair

10.6.1 Customer service technician

Request the assistance of an authorised customer service technician for any repair work. Contact your specialist dealer if you do not have the contact details for the customer service team or contact Stürmer Maschinen GmbH in Germany who can provide you with the contact information for a specialist dealer. Optionally, the

Stürmer Maschinen GmbH

Dr.-Robert-Pfleger-Str. 26

D- 96103 Hallstadt

can provide a customer service technician, however, the request for a customer service technician can only be made via your specialist dealer.

If the repairs are carried out by qualified technical personnel, they must follow the indications given in these operating instructions.

Optimum Maschinen Germany GmbH accepts no liability nor does it guarantee against damage and operating malfunctions resulting from failure to observe these operating instructions.

For repairs only use

- faultless and suitable tools,
- original parts or series components expressly authorized by Optimum Maschinen Germany GmbH.



10.7 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.

Limit values

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.



10.7.1 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 ¹⁾	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 ¹⁾	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 ¹⁾	> 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

¹⁾ 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:

Schmierstoffe Lubricant Lubrifiant	Viskosität Viskosity Viscosité ISO VG DIN 51519 mm²/s (cSt)	Kennzeich- nung nach DIN 51502							
Getriebeöl Gear oil Huile de réducteur	VG 680	CLP 680	Aral Degol BG 680	BP Energol GR-XP 680	SPARTAN EP 680	Klüberoil GEM 1-680	Mobilgear 636	Shell Omala S2 GX 680	Meropa 680
	VG 460	CLP 460	Aral Degol BG 460	BP Energol GR-XP 460	SPARTAN EP 460	Klüberoil GEM 1-460	Mobilgear 634	Shell Omala S2 GX 460	Meropa 460
	VG 320	CLP 320	Aral Degol BG 320	BP Energol GR-XP 320	SPARTAN EP 320	Klüberoil GEM 1-320	Mobilgear 632	Shell Omala S2 GX 320	Meropa 320
	VG 220	CLP 220	Aral Degol BG 220	BP Energol GR-XP 220	SPARTAN EP 220	Klüberoil GEM 1-220	Mobilgear 630	Shell Omala S2 GX 220	Meropa 220
	VG 150	CLP 150	Aral Degol BG 150	BP Energol GR-XP 150	SPARTAN EP 150	Klüberoil GEM 1-150	Mobilgear 629	Shell Omala S2 GX 150	Meropa 150
	VG 100	CLP 100	Aral Degol BG 100	BP Energol GR-XP 100	SPARTAN EP 100	Klüberoil GEM 1-100	Mobilgear 627	Shell Omala S2 GX100	Meropa 100
	VG 68	CLP 68	Aral Degol BG 68	BP Energol GR-XP 68	SPARTAN EP 68	Klüberoil GEM 1-68	Mobilgear 626	Shell Omala S2 GX 68	Meropa 68
	VG 46	CLP 46	Aral Degol BG 46	BP Bartran 46	NUTO H 46 (HLP 46)	Klüberoil GEM 1-46	Mobil DTE 25	Shell Tellus S2 MX 46	Anubia EP 46
	VG 32	CLP 32	Aral Degol BG 32	BP Bartran 32	NUTO H 32 (HLP 32)	Klübersynth GEM 4- 32 N	Mobil DTE 24	Shell Tellus S2 MX 32	Anubia EP 32
Hydrauliköl Hydraulic oil Huile hydraulique	VG 32	CLP 32	Aral Vitam GF 32	BP Energol HLP HM 32	NUTO H 32 (HLP 32)	LAMORA HLP 32	Mobil Nuto HLP 32	Shell Tellus S2 M 32	Rando HD HLP 32
	VG 46	CLP 46	Aral Vitam GF 46	BP Energol HLP HM 46	NUTO H 46 (HLP 46)	LAMORA HLP 46	Mobil Nuto HLP 46	Shell Tellus S2 M 46	Rando HD HLP 46
Getriebefett Gear grease Graisse de réducteur		G 00 H-20	Aral FDP 00 (Na-verseift) Aralub MFL 00 (Li-verseift)	BP Energ grease PR-EP 00	FIBRAX EP 370 (Na-verseift)	MICROLUB E GB 00	Mobilux EP 004	Shell Alvania GL 00 (Li-verseift)	Marfak 00

oil-compare-list.fm

Spezialfette, wasserabweisend Special greases, water resistant Graisses spéciales, déperlant			Aral Aralub	Energrease PR 9143		ALTEMP Q NB 50 Klüberpaste ME 31-52	Mobilux EP 0 Mobil Greaserex 47		
Wälzlagerfett Bearing grease Graisse de roulement		K 3 K-20 (Li-verseift)	Aralub HL 3	BP Energrease LS 3	BEACON 3	CENTOPLE X 3	Mobilux 3	Shell Alvania R 3 Alvania G 3	Multifak Premium 3
Öle für Gleitbahnen Oils for slideways Huiles pour glissières	VG 68	CGLP 68	Aral Deganit BWX 68	BP Maccurat D68	ESSO Febis K68	LAMORA D 68	Mobil Vactra Oil No.2	Shell Tonna S2 M 68	Way lubricant X 68
Öle für Hochfrequenzspindeln Oils for Built-in spindles Huiles pour broches à haute vitesse	VG 68		Deol BG 68	Emergol HLP-D68	Spartan EP 68		Drucköl KLP 68-C	Shell Omala 68	
Fett für Zentralschmierung (Fließfett) Grease for central lubrication Graisse pour lubrification centrale	NLGI Klasse 000 NLGI class 000		ARALUB BAB 000	Grease EP 000	Shell Gadus S4 V45AC	CENTOPLE X GLP 500	Mobilux EP 023		Multifak 264 EP 000
Fett für Hochfrequenzspindeln Grease for Built-in spindles Graisse pour broches à haute vitesse	METAFLUX-Fett-Paste (Grease paste) Nr. 70-8508 METAFLUX-Moly-Spray Nr. 70-82 Techno Service GmbH ; Detmolder Strasse 515 ; D-33605 Bielefeld ; (++49) 0521- 924440 ; www.metaflux-ts.de								
Kühlschmiermittel Cooling lubricants Lubrifiants de refroidissement			Aral Emusol	BP Sevora	Esso Kutwell		Mobilcut	Shell Adrana	Chevron Soluble Oil B

11 Ersatzteilzeichnungen - Spare part drawings

A Säule - Column

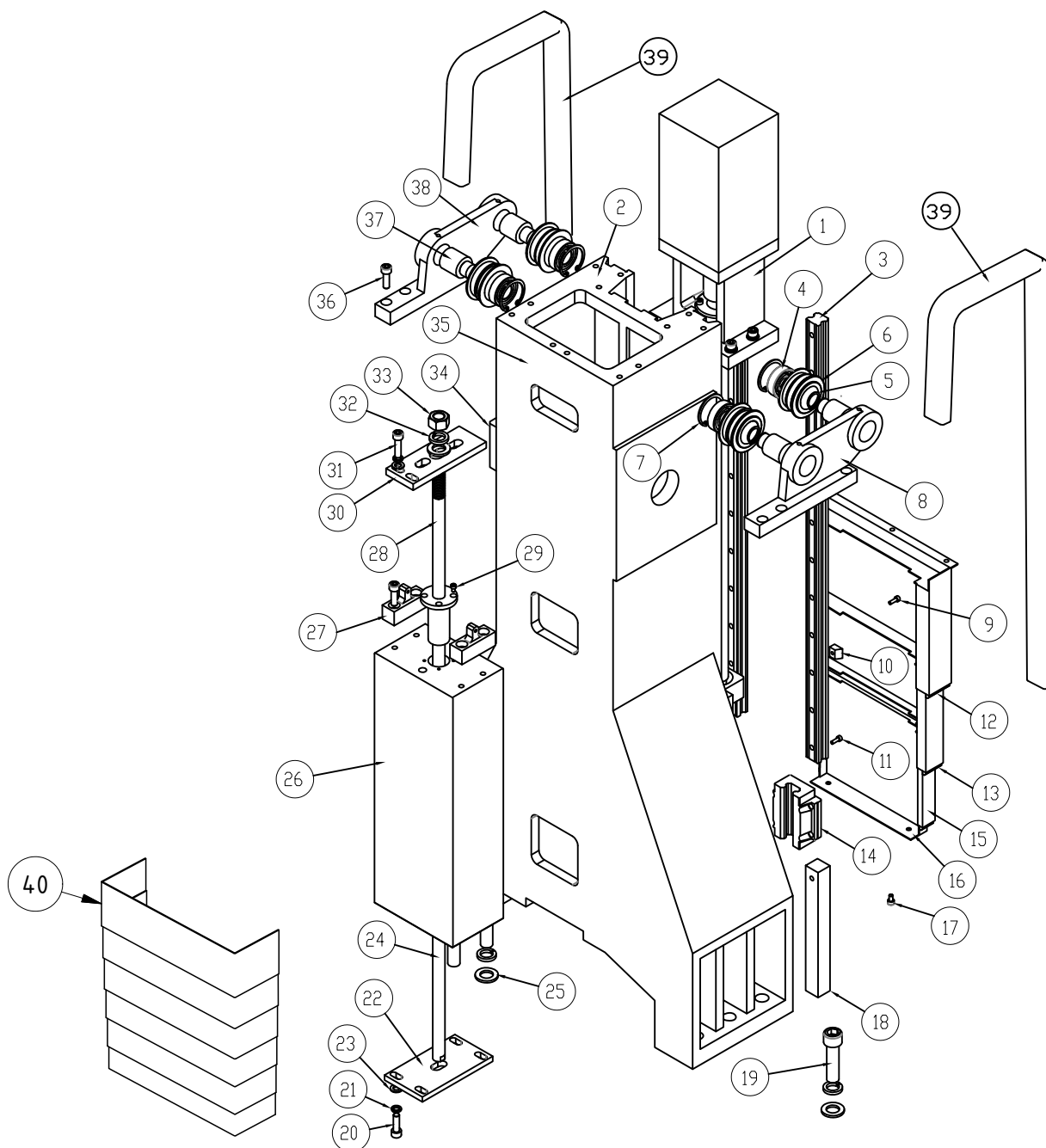


Abb. 11-1: F80 column

F80_parts.fm

F80 - Säule - Column					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Lagerbock	Bearing block	1		03501080101
2	Säule	Column	1		03501080102
3	Linearführung	Lineare guide	2		03501080103
4	Kugellager	Ball bearing	4	6004	0406004.2R
5	Sicherungsring	Retaining ring	4		03501080105
6	Führungsrolle	Guide roll	4		03501080106
7	Sicherungsring	Retaining ring	4	42	
8	Halter	Holder	1		03501080108
9	Innensechskantschraube	Socket head crew	26	M4x12	
10	Platte	Plate	26		035010801
11	Innensechskantschraube	Socket head crew	32	M6x25	
12	Abdeckung A	Cover A	1		03501080112
13	Abdeckung B	Cover B	1		03501080113
14	Schlitten Linearführung	Lineare guide slide	4		03501080114
15	Abdeckung C	Cover C	1		03501080115
16	Platte	Plate	1		03501080116
17	Innensechskantschraube	Socket head crew	5	M5x8	
18	Platte	Plate	2		03501080118
19	Innensechskantschraube	Socket head crew	6	M16x60	
20	Innensechskantschraube	Socket head crew	4	M8x12	
21	Federring	Spring ring	4	8	
22	Platte	Plate	1		03501080122
23	Scheibe	Washer	8	4	
24	Führung	Guide	1		03501080124
25	Scheibe	Washer	7	16	
26	Gegengewicht	Balance weight	1		03501080126
27	Block	Block	2		03501080127
28	Welle	Shaft	1		03501080128
29	Innensechskantschraube	Socket head crew	4	M4x6	
30	Platte	Plate	1		03501080130
31	Innensechskantschraube	Socket head crew	2	M8x30	
32	Federring	Spring ring	2	16	
33	Sechskantmutter	Hexagon nut	2	M16x60	
34	Block	Block	1		03501080134
35	Scheibe	Washer	6	6	
36	Innensechskantschraube	Socket head crew	14	M8x25	
37	Welle	Shaft	4		03501080137
38	Halter	Holder	2		03501080138
39	Kette Gegengewicht	Balance chain	2		03501080139
40	Lamellenabdeckung	Cover	1		03501080140

B Fräskopf - Milling head

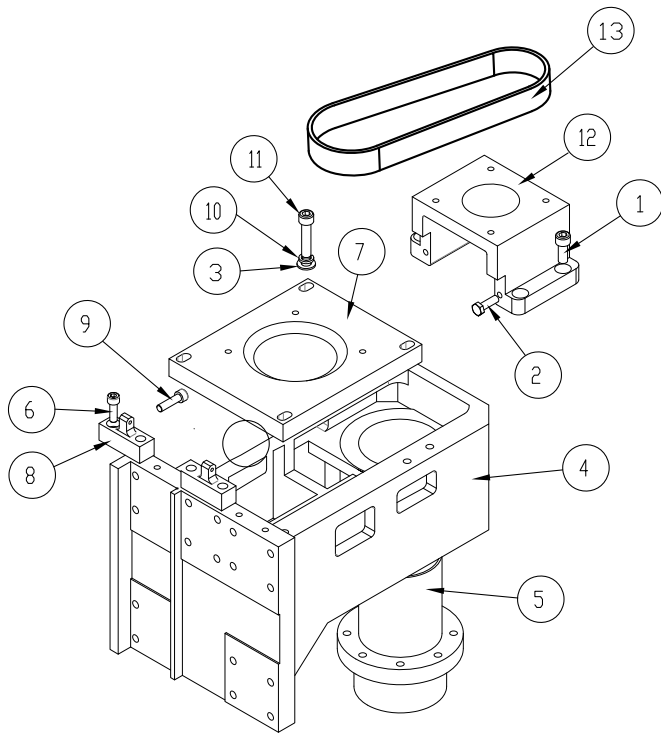


Abb.11-2: F80 milling head

F80 - Fräskopf - Milling head					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Innensechskantschraube	Socket head screw	4	M10x25	
2	Sechskantschraube	Hexagon screw	2	M8x25	
3	Scheibe	Washer	7	10	
4	Gehäuse	Housing	1		03501080204
5	Frässpindel	Milling spindle	1		03501080205
6	Innensechskantschraube	Socket head screw	12	M8x25	
7	Platte	Plate	1		03501080207
8	Platte	Plate	2		03501080208
9	Innensechskantschraube	Socket head screw	20	M8x30	
10	Federring	Spring ring	7	10	
11	Innensechskantschraube	Socket head screw	7	M10x45	
12	Motorhalter	Motor holder	1		03501080212
13	Antriebsriemen	Drive belt	1	Zahnriemen 5GT-550-30	0395GT55030

C Frästisch - Milling table

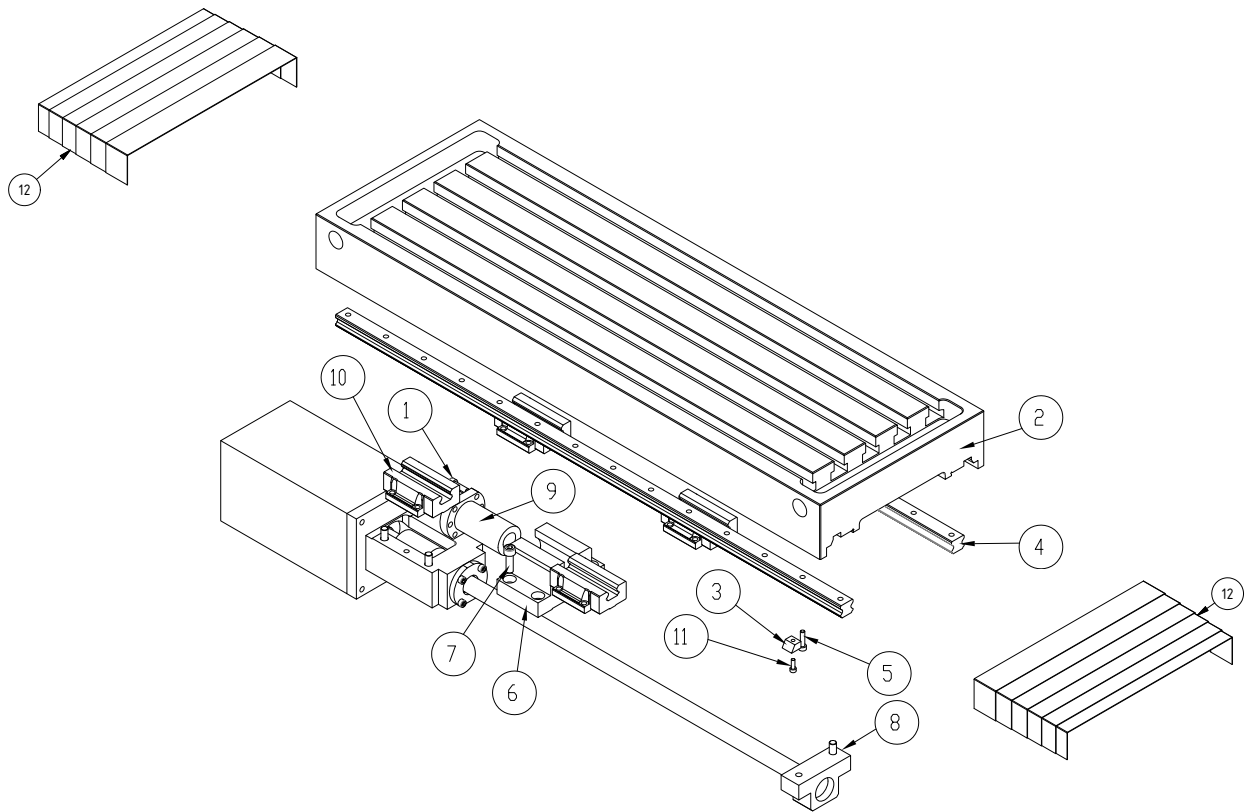


Abb.11-3: F80 milling table

F80 - Frästisch - Milling table					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Innensechskantschraube	Socket head screw	6	M6x20	
2	Frästisch	Milling table	1		03501080302
3	Platte	Plate	28		03501080303
4	Linearführung	Lineare guide	2		03501080304
5	Innensechskantschraube	Socket head screw	28	M5x20	
6	Halter	Holder	1		03501080306
7	Innensechskantschraube	Socket head screw	4	M8x25	
8	Halter	Holder	1		03501080308
9	Spindelmutter	Spindle nut	1		03501080309
10	Schlitten Linearführung	Lineare guide slide	4		03501080310
11	Innensechskantschraube	Socket head screw	28	M4x12	
12	Lamellenabdeckung	Cover	2		03501080312

D X-Achse - X axis

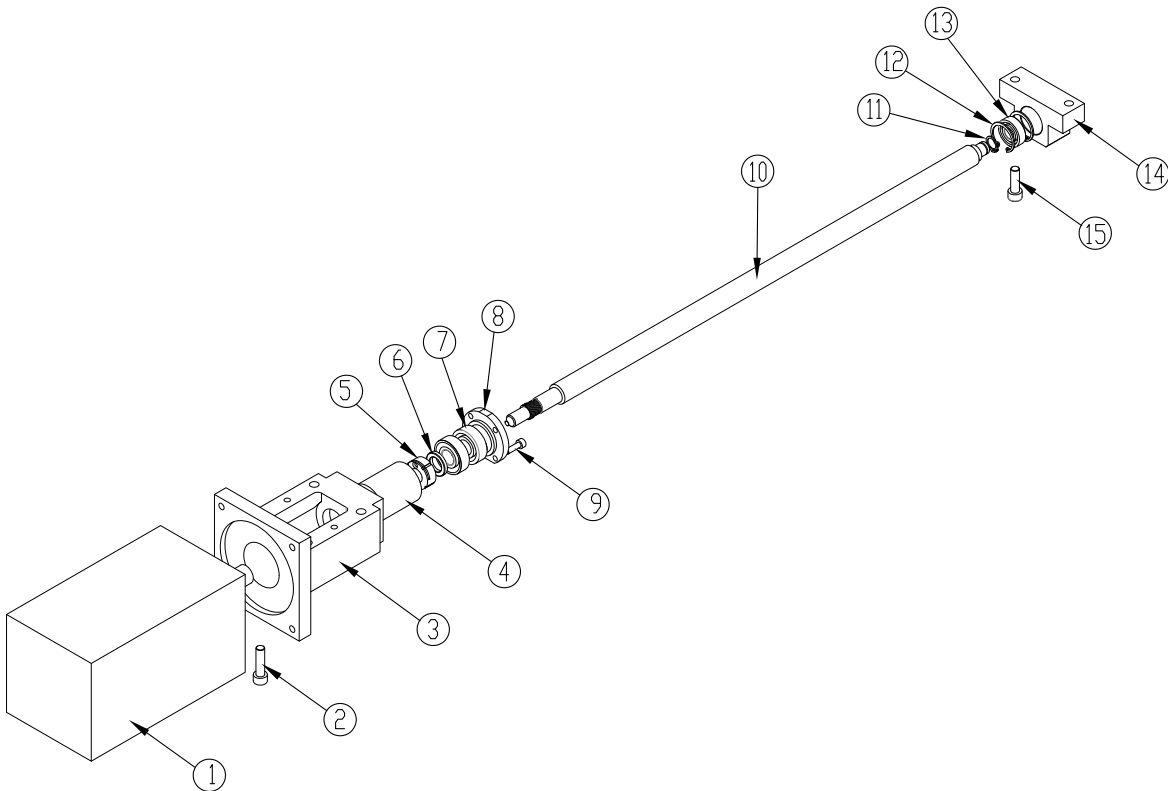


Abb. 11-4: F80 x axis

F80 - X-Achse - X axis					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Motor	Motor	1		03501080401
2	Innensechskantschraube	Socket head screw	4	M8x30	
3	Lagerbock	Bearing block	1		03501080403
4	Kupplung	Clutch	1		03501080404
5	Klemmmutter	Clamping nut	1		03501080405
6	Ring	Ring	1		03501080406
7	Kugellager	Ball bearing	2	7202	0707202
8	Flansch	Flange	1		03501080408
9	Innensechskantschraube	Socket head screw	4	M5x16	
10	Spindel	Spindle	1		03501080410
11	Sicherungsring	Retaining ring	1	12	
12	Sicherungsring	Retaining ring	2	28	
13	Kugellager	Ball bearing	1	6001	0406001
14	Lagerbock	Bearing block	1		03501080414
15	Innensechskantschraube	Socket head screw	2	M8x25	

E Y-Achse - Y axis

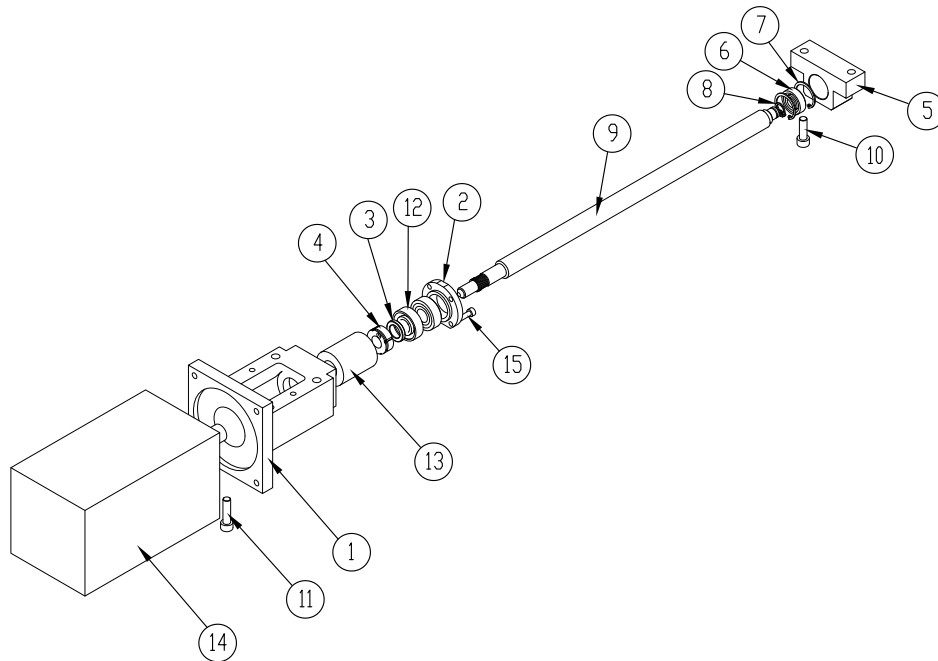


Abb.11-5: F80 Y axis

F80 - Y-Achse - Y axis					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Lagerbock	Bearing block	1		03501080501
2	Flansch	Flange	1		03501080502
3	Ring	Ring	1		03501080503
4	Klemmmutter	Clamping nut	1		03501080504
5	Lagerbock	Bearing block	1		03501080505
6	Kugellager	Ball bearing	1	6001	0406001
7	Sicherungsring	Retaining ring	2	28	
8	Sicherungsring	Retaining ring	1	12	
9	Spindel	Spindle	1		03501080509
10	Innensechskantschraube	Socket head screw	2	M8x25	
11	Innensechskantschraube	Socket head screw	4	M8x30	
12	Kugellager	Ball bearing	2	7202	0407202
13	Kupplung	Clutch	1		03501080513
14	Motor	Motor	1		03501080514
15	Innensechskantschraube	Socket head screw	4	M5x16	

F **Z-Achse - Z axis**

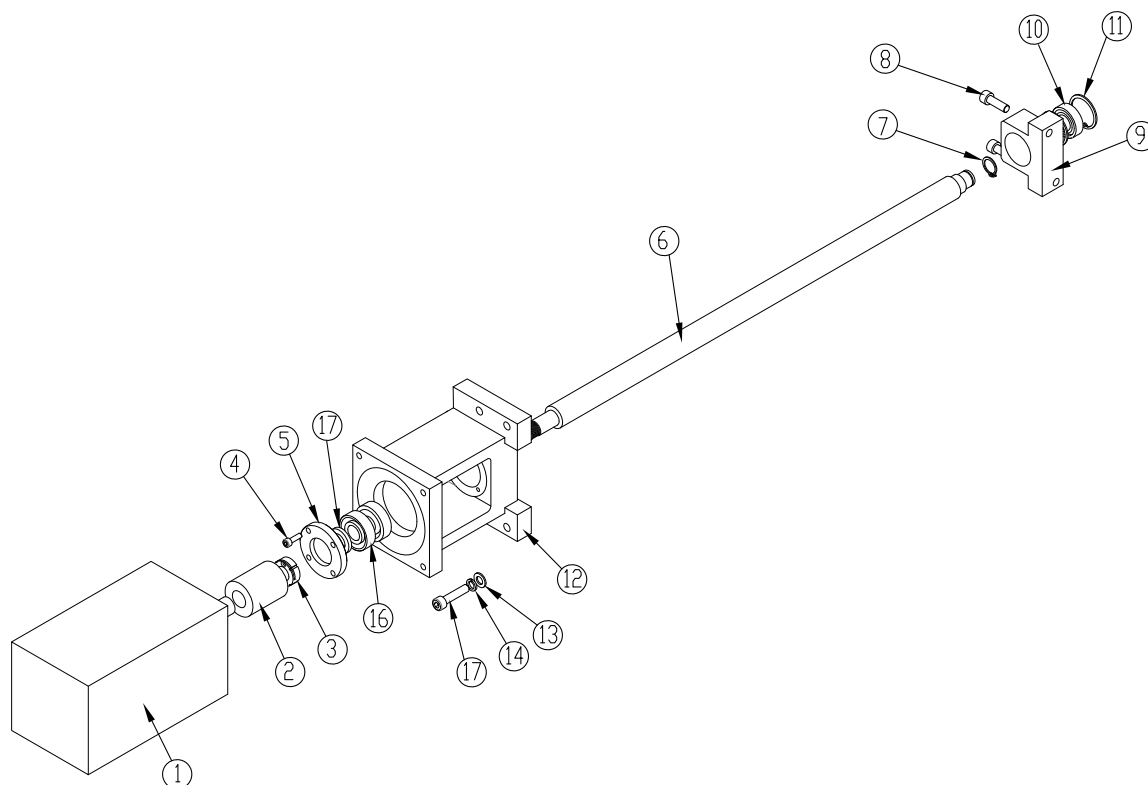


Abb.11-6: F80 Z axis

F80 - Z-Achse - Z axis					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Motor	Motor	1		03501080601
2	Kupplung	Clutch	1		03501080602
3	Klemmmutter	Clamping nut	1		03501080603
4	Innensechskantschraube	Socket head screw	4	M5x16	
5	Flansch	Flange	1		03501080605
6	Spindel	Spindle	1		03501080606
7	Sicherungsring	Retaining ring	1	17	
8	Innensechskantschraube	Socket head screw	2	M8x25	
9	Lagerbock	Bearing block	1		03501080609
10	Kugellager	Ball bearing	1	6003	0406003
11	Sicherungsring	Retaining ring	2	35	
12	Lagerbock	Bearing block	1		03501080612
13	Scheibe	Washer	4	8	
14	Federring	Spring ring	4	8	
15					03501080615
16	Kugellager	Ball bearing	2	7203	0407203
17	Innensechskantschraube	Socket head screw	4	M8x35	

G Werkzeugwechsler - Tool changer

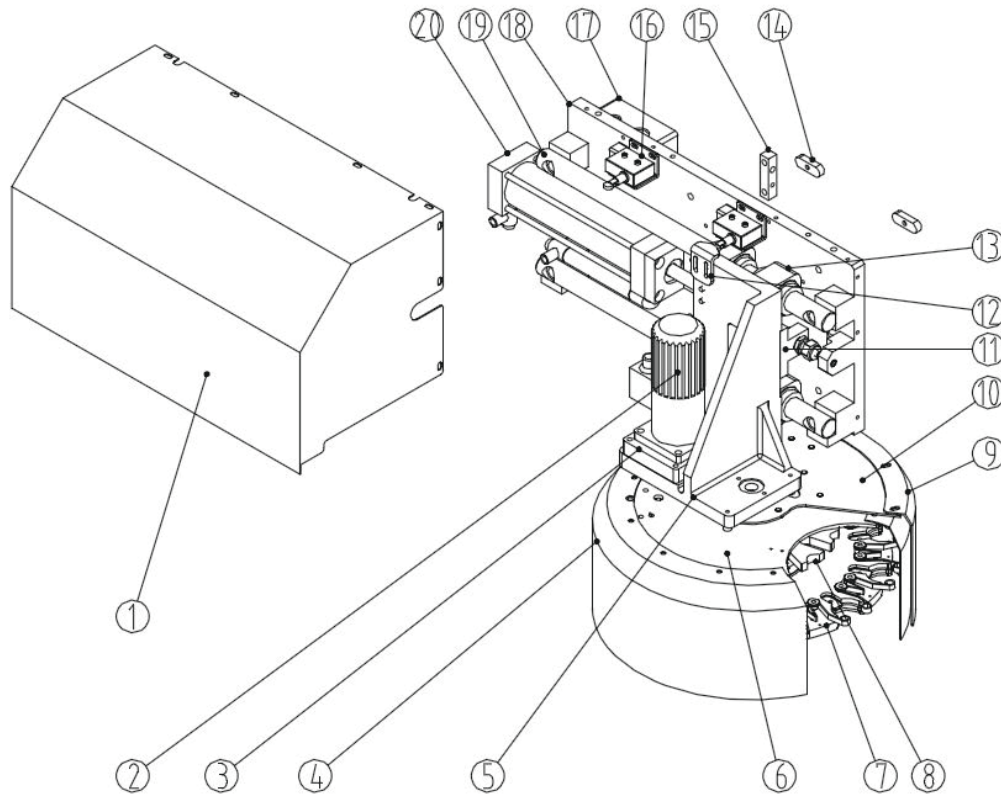


Abb. 11-7: F80 tool changer

F80 - Werkzeugwechsler - Tool changer					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Abdeckung	Cover	1		03501080901
2	Motor	Motor	1		03501080902
3	Motorplatte	Motor plate	1		03501080903
4	Abdeckung	Cover	1		03501080904
5	Halter	Holder	1		03501080905
6	Platte	Plate	1		03501080906
7	Werkzeugaufnahme	Tool holder	1		03501080907
8	Platte	Plate	1		03501080908
9	Abdeckung	Cover	1		03501080909
10	Platte	Plate	1		03501080910
11	Führung	Guide	1		03501080911
12	Endanschlag	Limit stop	1		03501080912
13	Führung	Guide	1		03501080913
14	Platte	Plate	1		03501080914
15	Klotz	Block	1		03501080915
16	Endschalter	End switch	1		03501080916
17	Verteiler	Manifold	1		03501080917
18	Grundplatte	Base plate	1		03501080918
19	Führungsstange	Guide rod	1		03501080919
20	Pneumatikzylinder	Pneumatic cylinder	1		03501080920

F80_parts.fm

L Gehäuse V2 - Housing V2

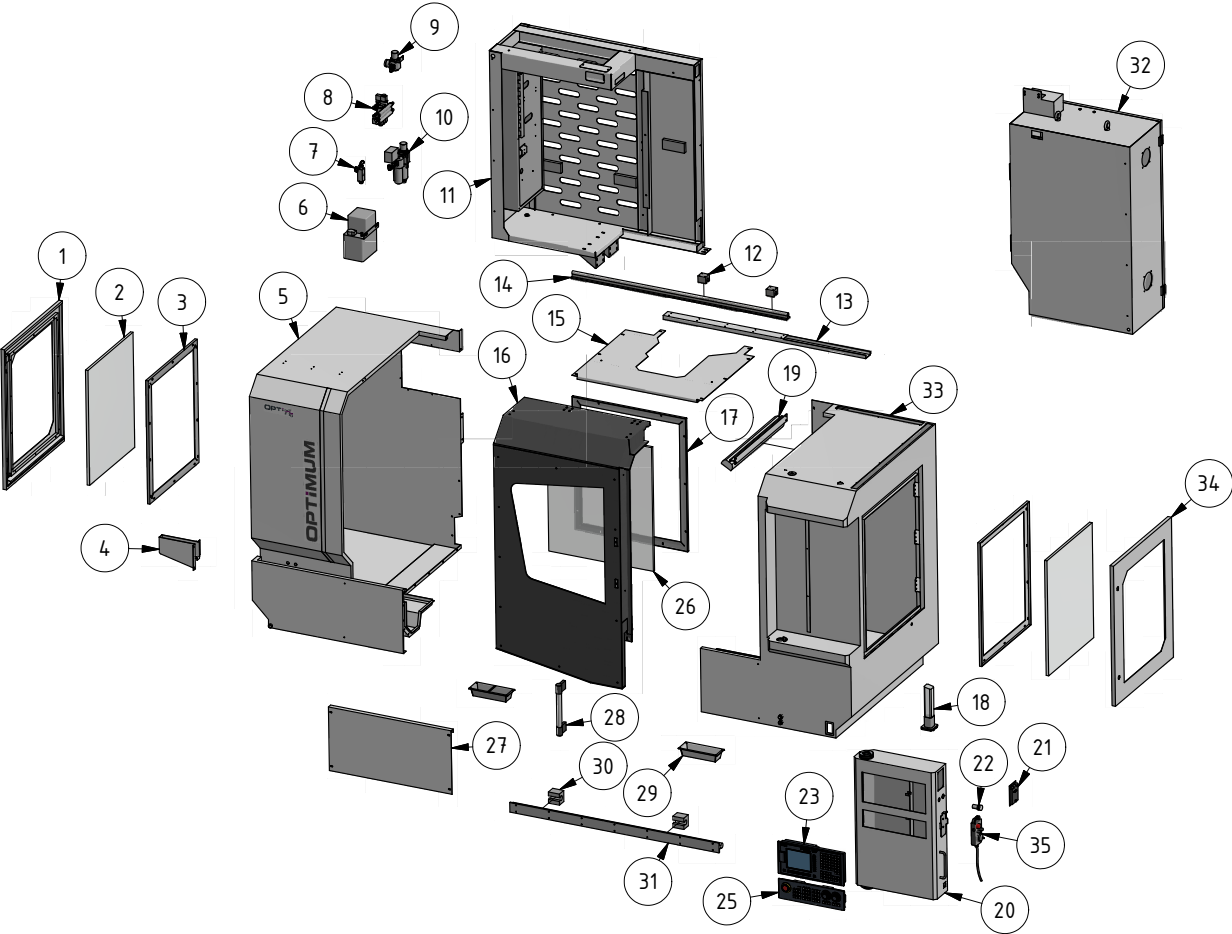


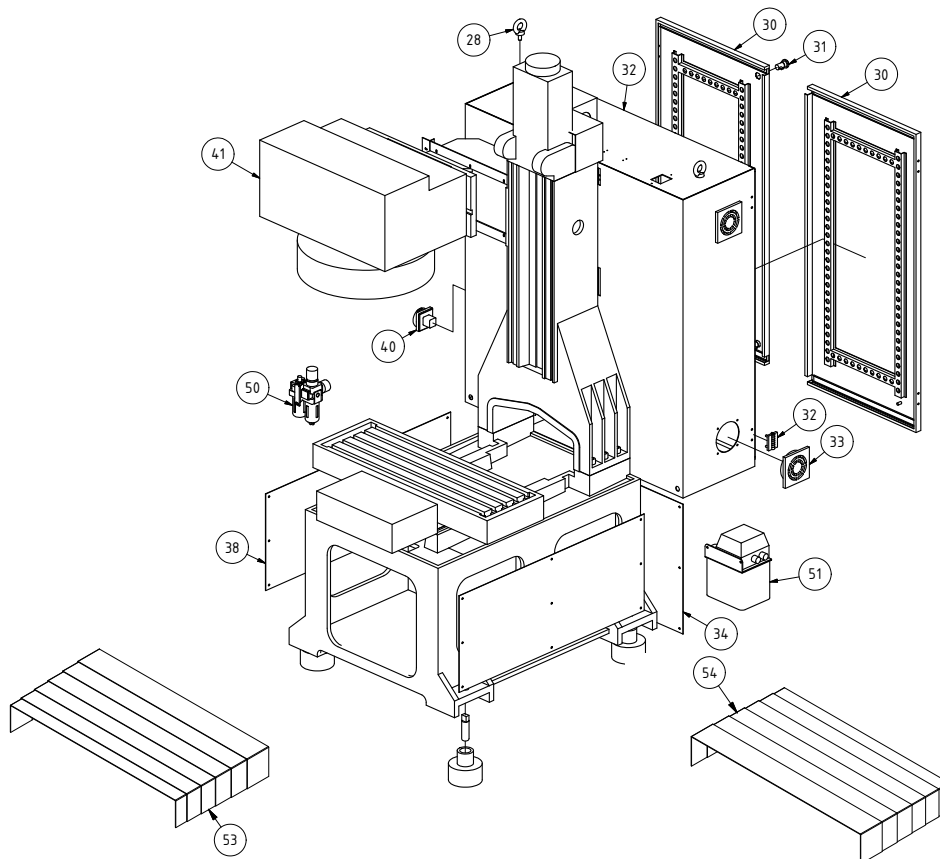
Abb.11-8: F80 housing

F80 - Gehäuse V2 - Housing V2					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
1	Linke Seitentür	Left side door	1		035010810801
2	Sicherheitsglas	Safety glass	2		035010810802
3	Sicherheitsglashalter	Safety glass holder	2		035010810803
4	Abtropfschale	Drainer tray	1		035010810804
5	Linke Umhausung	Left side cover	1		035010810805
6	Ölpumpe	Oil pump	1		035010810806
7	Anschlussstück	Connecting block	1		035010810807
8	Anschlussverteiler	Connection distributor	1		035010810808
9	Druckregler	Pneumatic regulator	1		035010810809
10	Wartungseinheit	Service unit	1		035010810810
11	Hintere Abdeckung	Rear cover	1		035010810811
12	Führungsblock Maschinentür	Guide block machine door	2		035010810812
13	Verbindungsstrebe	Connecting strut	1		035010810813
14	Führungsschne Maschinentür (oben)	Machine door guide rail (top)	1		035010810814
15	Obere Abdeckung	Upper cover	1		035010810815
16	Maschinentür	Machine door	1		035010810816
17	Sicherheitsglashalter	Safety glass holder	1		035010810817
18	Signallampe	Signal lamp	1		035010810818
19	Arbeitsleuchte	Machine light	1		035010810819
20	Gehäuse Steuerung	Housing control	1		035010810820
21	Schnittstelle	Interface	1		035010810821
22	Taster	Button	1		035010810822
23	Siemens Steuerung A	Siemens control	1		035010810823
25	Siemens Steuerung B	Siemens control B	1		035010810825

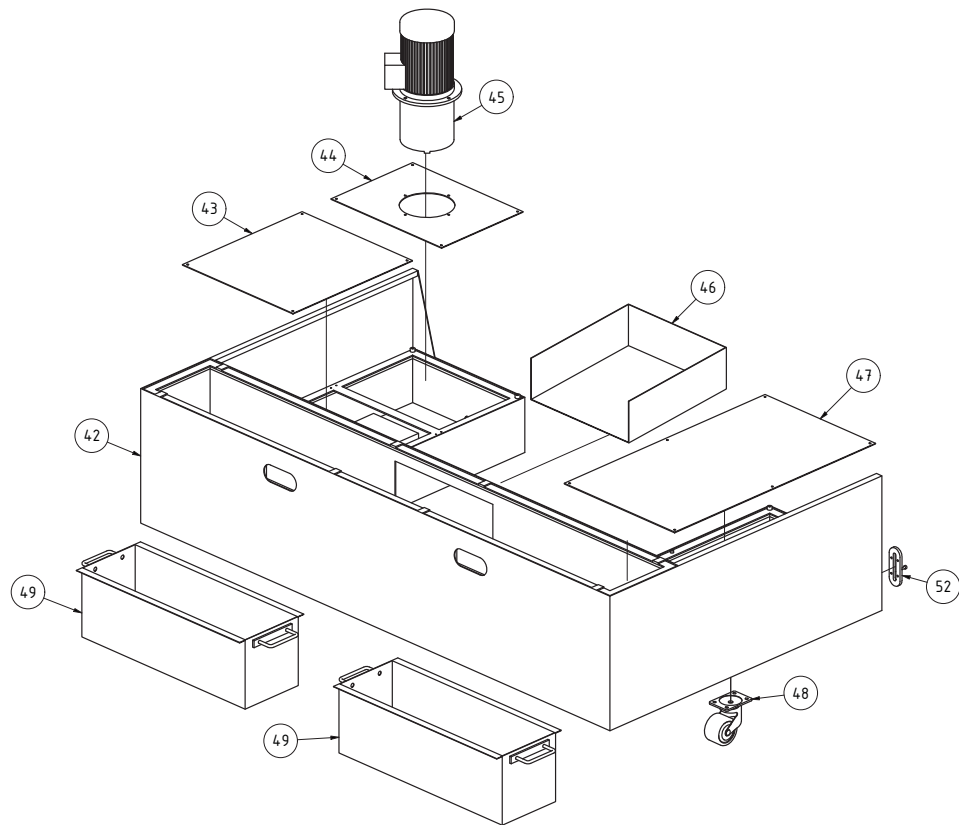
F80_parts.fm

26	Sicherheitsglas	Safety glass	1		035010810826
27	Abdeckung unten	Lower cover	1		035010810827
28	Türgriff	Door handle	1		035010810828
29	Sieb	Filter	1		035010810829
30	Führungsblock Maschinentür	Guide block machine door	2		035010810830
31	Führungsschne Maschinentür (unten)	Machine door guide rail (bottom)	1		035010810831
32	Schaltschrank	Electrical cabinet	1		035010810832
33	Rechte Umhausung	Right side cover	1		035010810833
34	Rechte Seitentür	Right side door	1		035010810834
35	Elektrisches Handrad	Electric handwheel	1		035010810835

I Maschinenständer - Machine stand

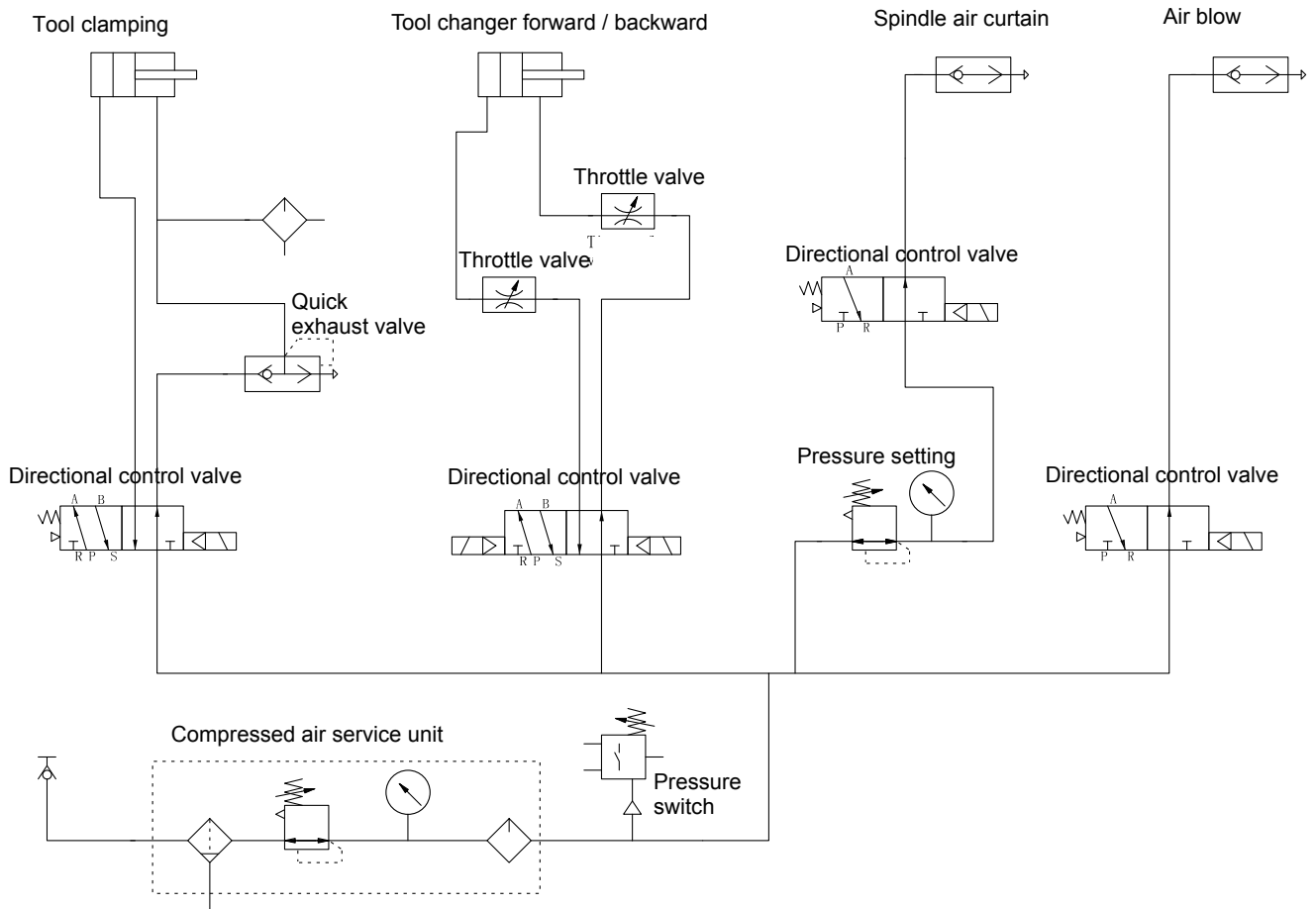


J Kühlmittelbehälter - Coolant tank

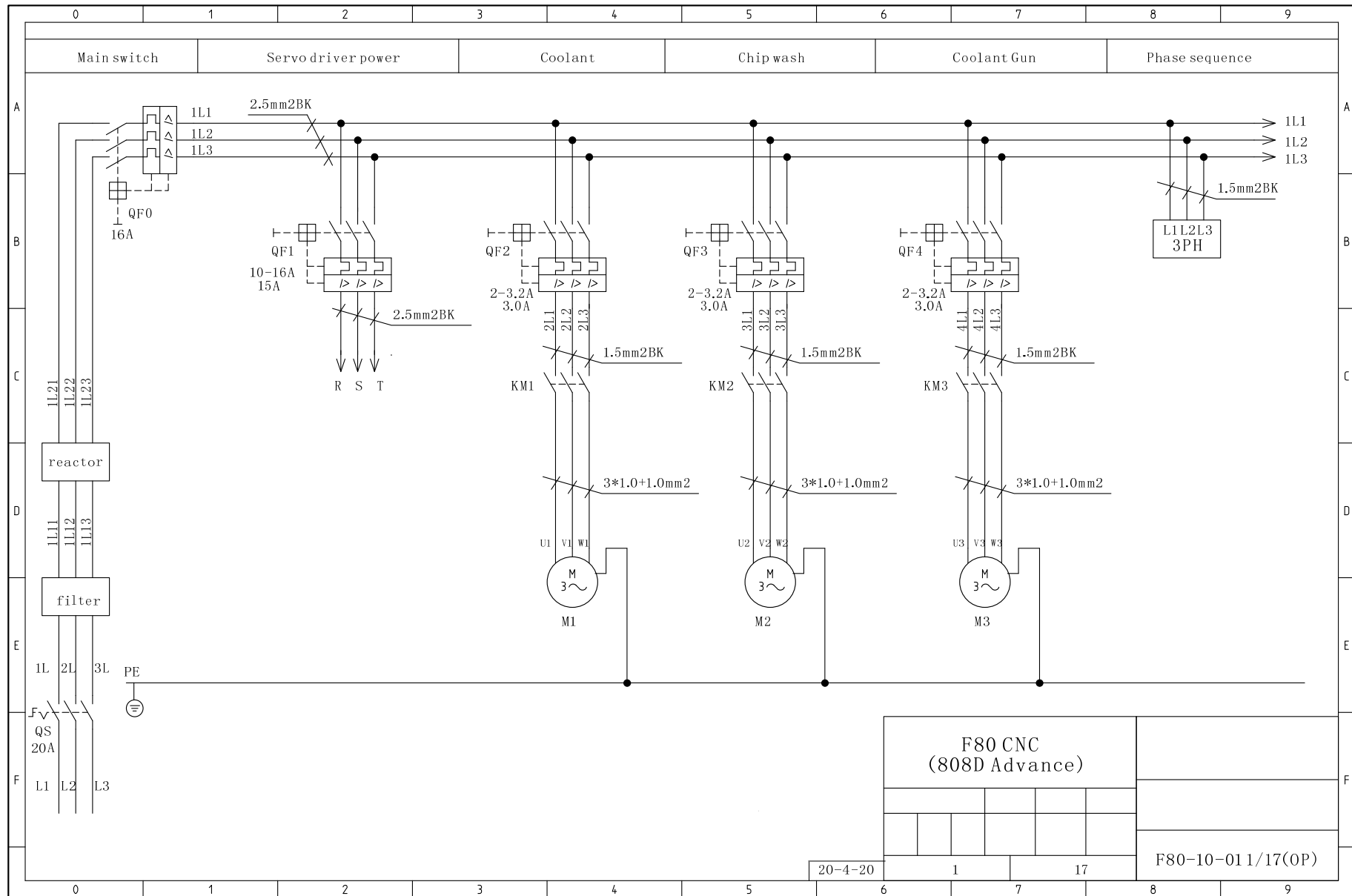


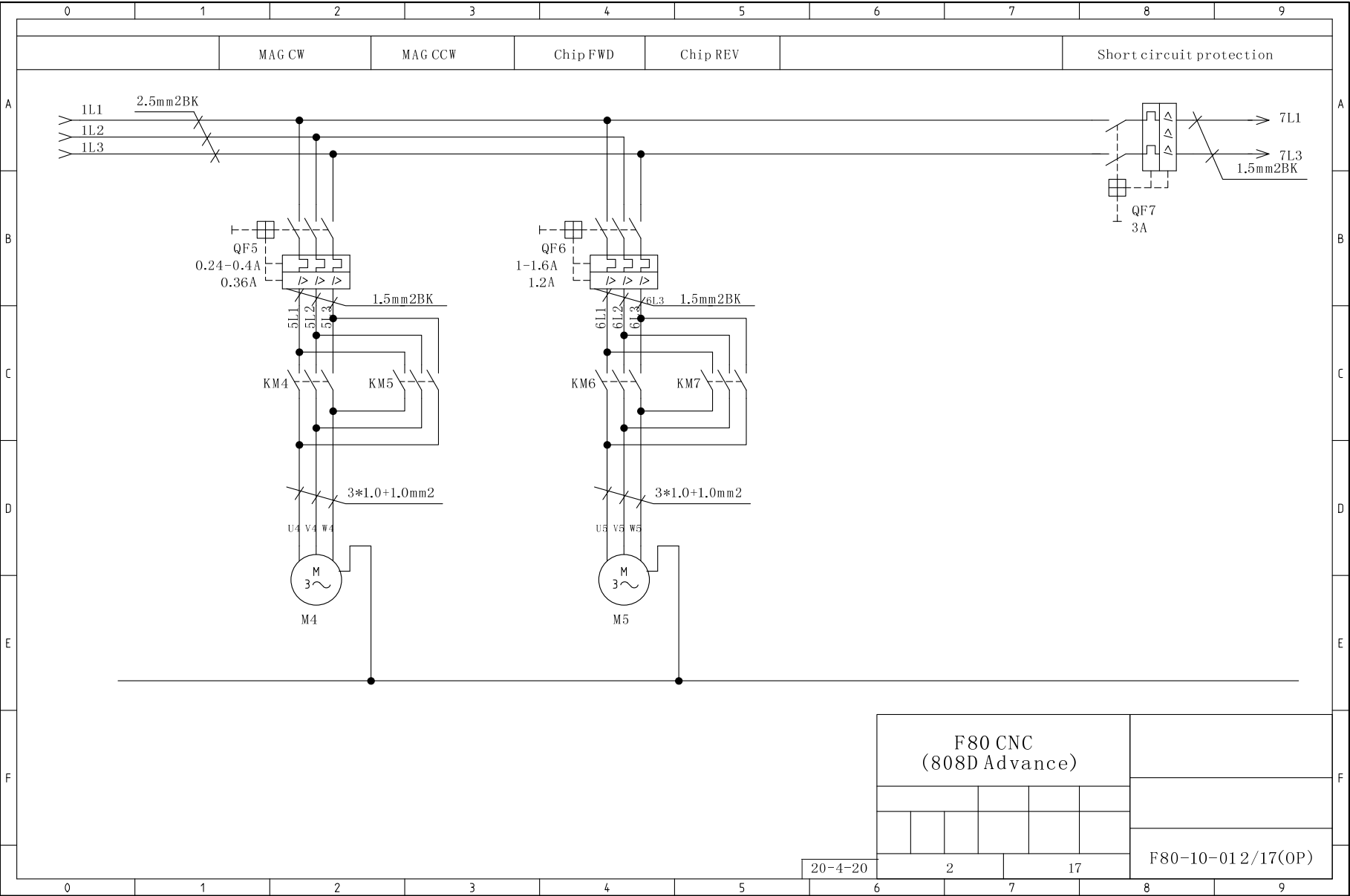
F80 - Maschinenständer, Kühlmittelbehälter - Machine stand, coolant tank					
Pos.	Bezeichnung	Description	Menge	Grösse	Artikelnummer
			Qty.	Size	Item no.
28	Augenschraube	Eye bolt	2		03501080828
30	Tür Schaltschrank	Door electrical cabinet	1		03501080830
31	Schloss	Lock	4		03501080831
32	Scharnier	Hinge	4		03501080832
33	Lüfter	Fan	4		03501080833
34	Abdeckung	Cover	1		03501080834
38	Abdeckung	Cover	2		03501080838
40	Hauptschalter	Main switch	1		03501080840
41	Werkzeugwechsler	Tool changer	1		03501080841
42	Unterbau	Substructure	1		03501080842
43	Abdeckung	Cover	1		03501080843
44	Abdeckung	Cover	1		03501080844
45	Kühlmittelpumpe	Coolant pump	1		03501080845
46	Abdeckung	Cover	1		03501080846
47	Abdeckung	Cover	1		03501080847
48	Rolle	Roll	4		03501080848
49	Sieb	Filter	2		03501080849
50	Wartungseinheit	Service unit	1		03501080850
51	Ölpumpe	Oil pump	1		03501080851
52	Schauglas	Sight glass	1		03501080852
53	Lamellenabdeckung	Cover	1		03501080853
54	Lamellenabdeckung	Cover	1		03501080854

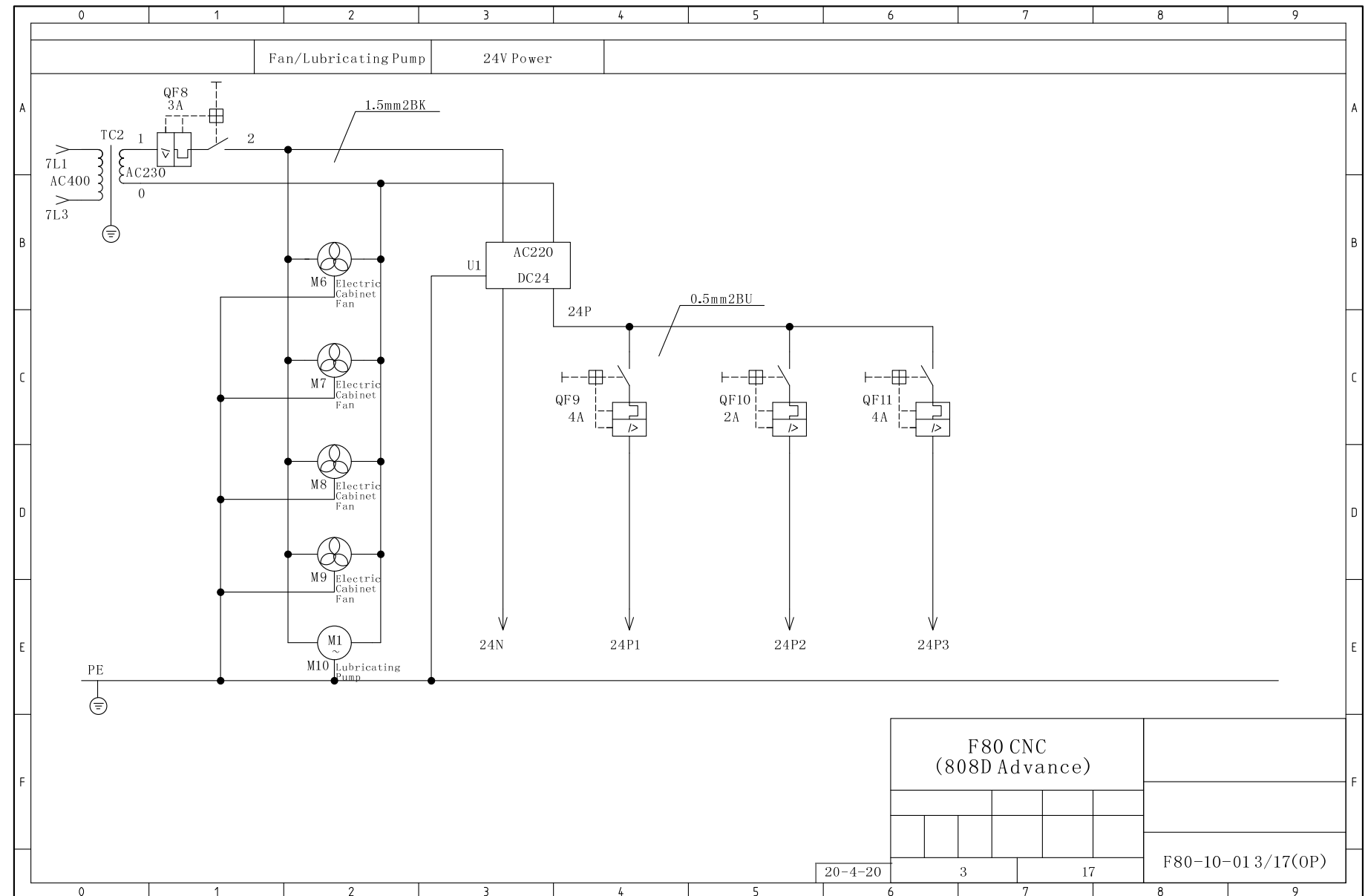
K Pneumatik - Pneumatic

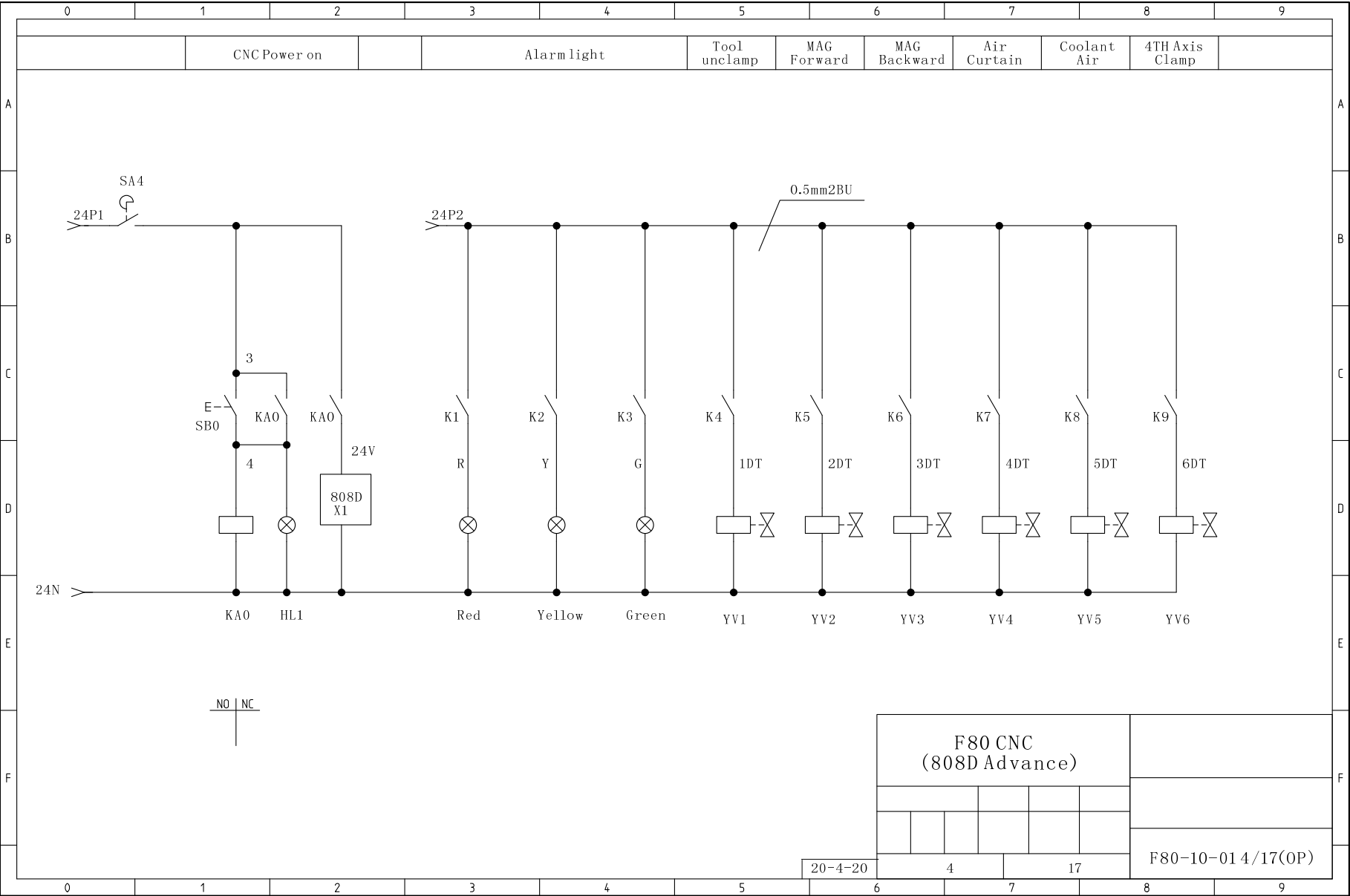


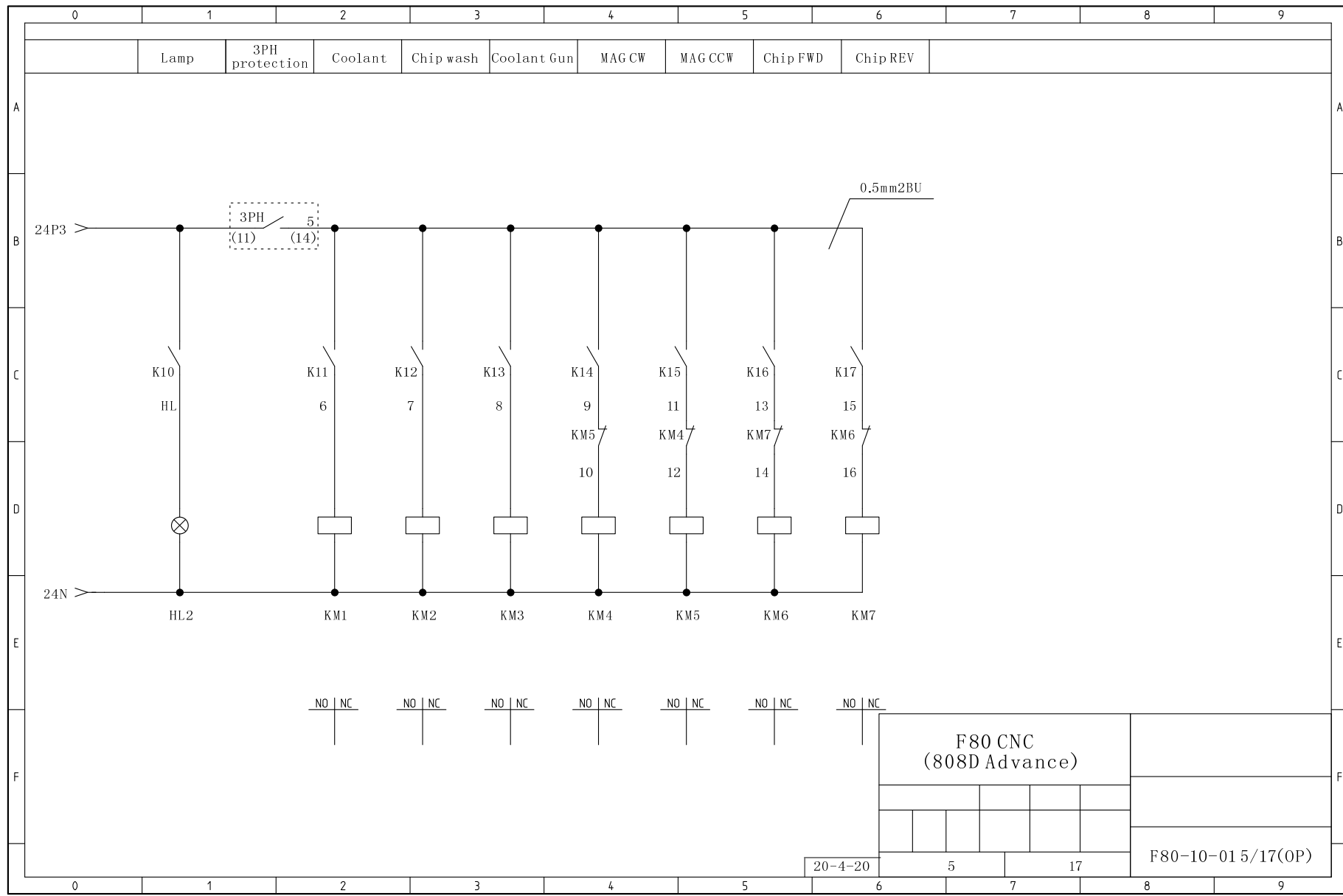
11.1 F80 Advanced (20.04.2020) - Schaltplan - Wiring diagram



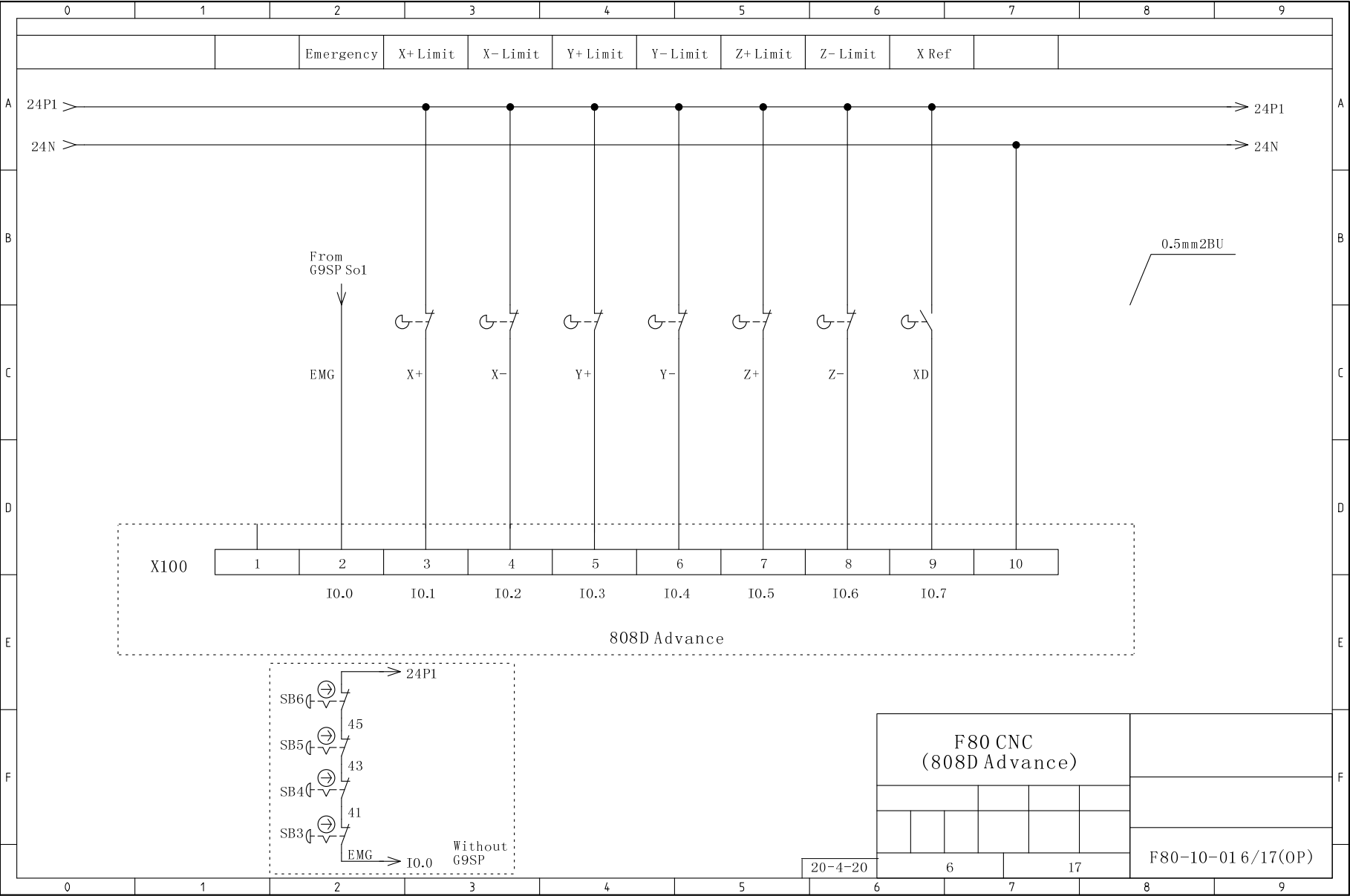


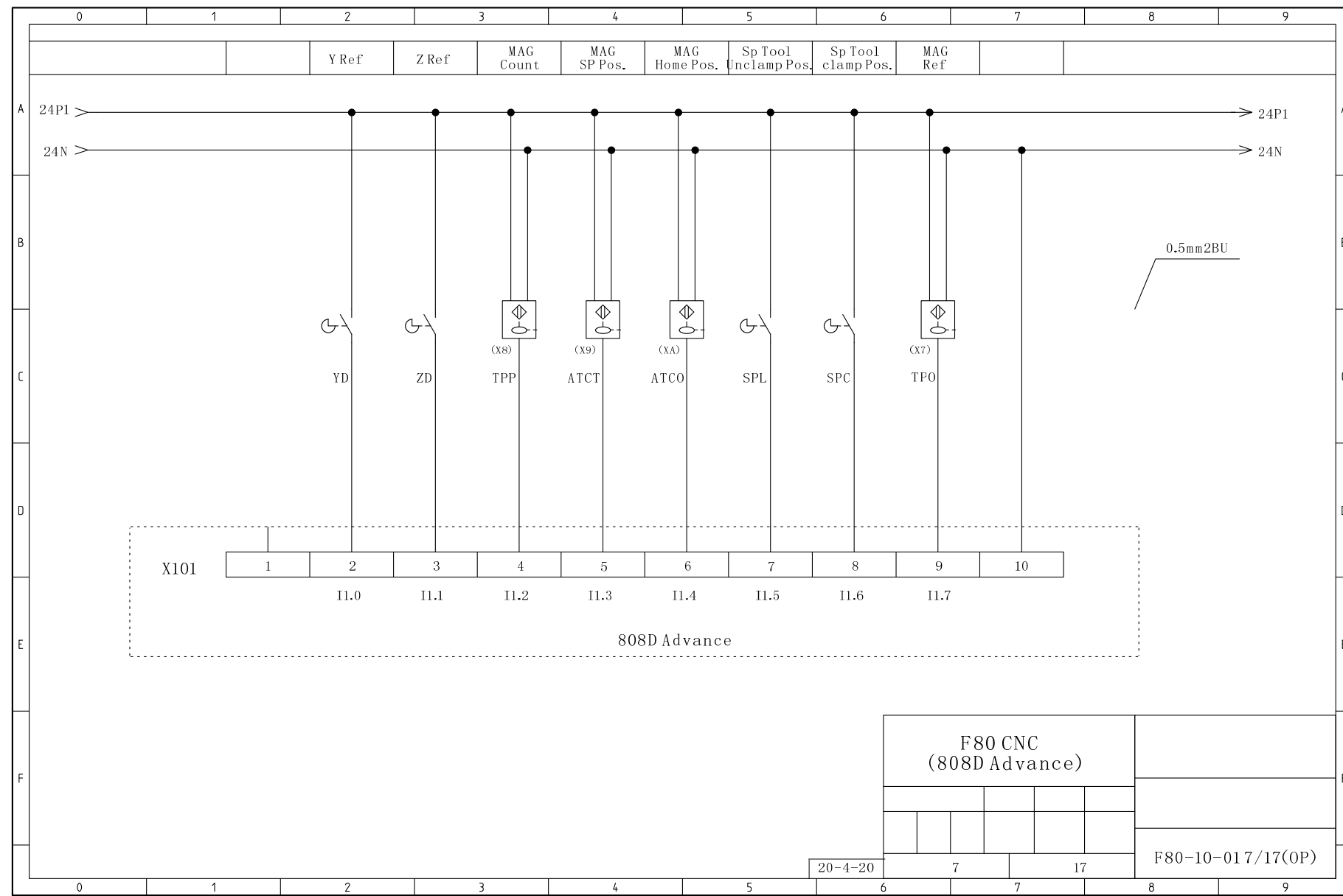


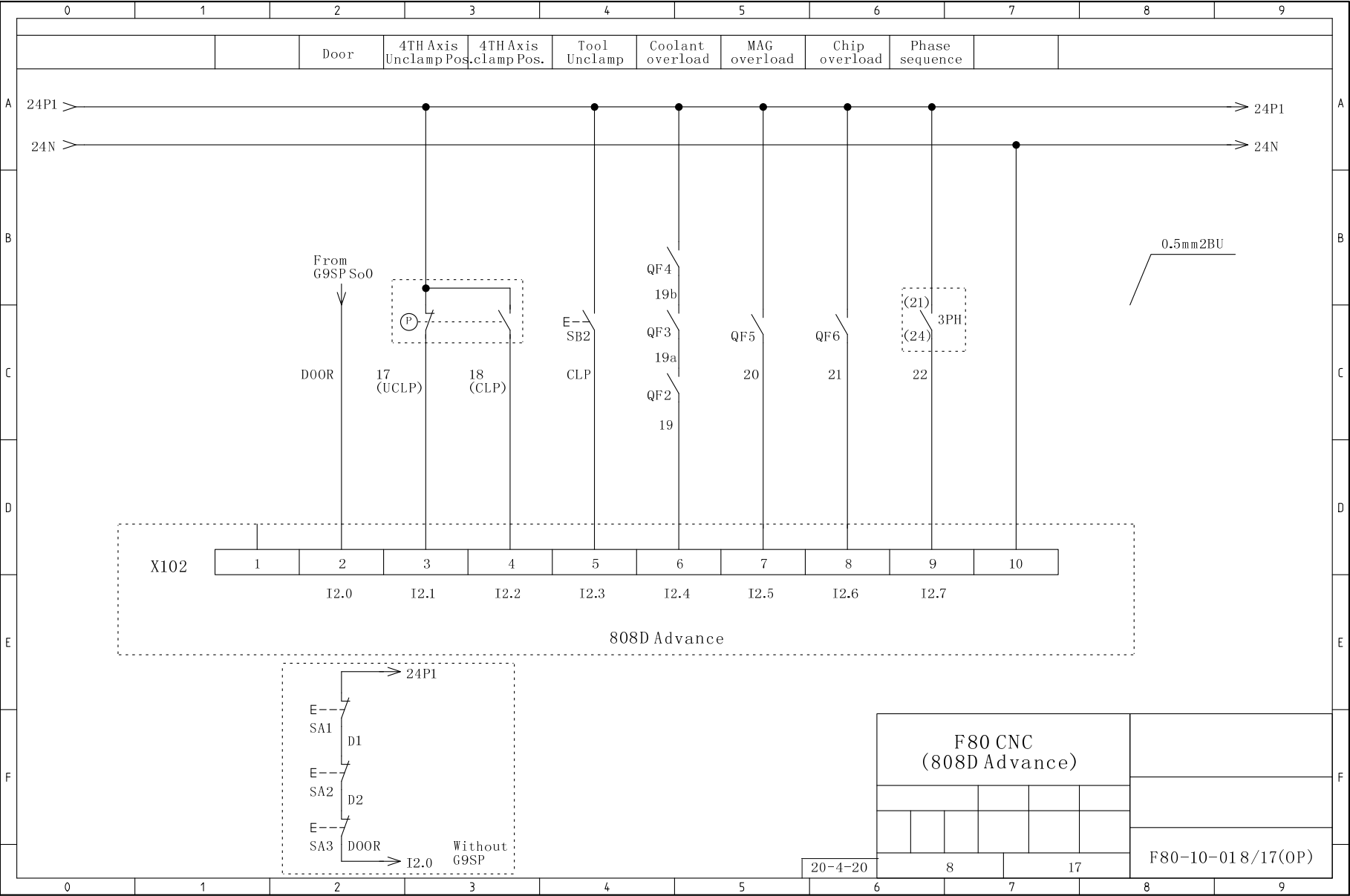


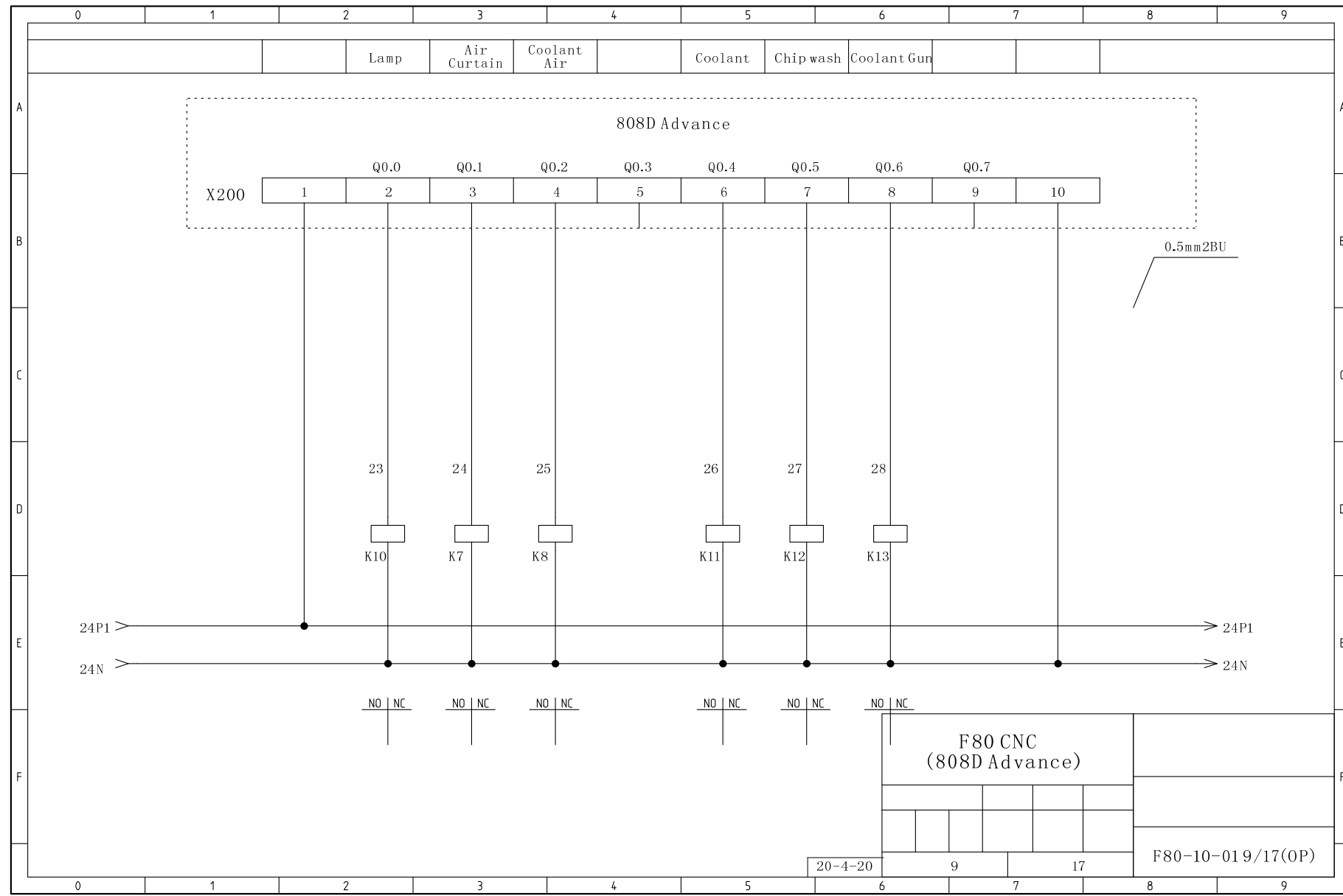


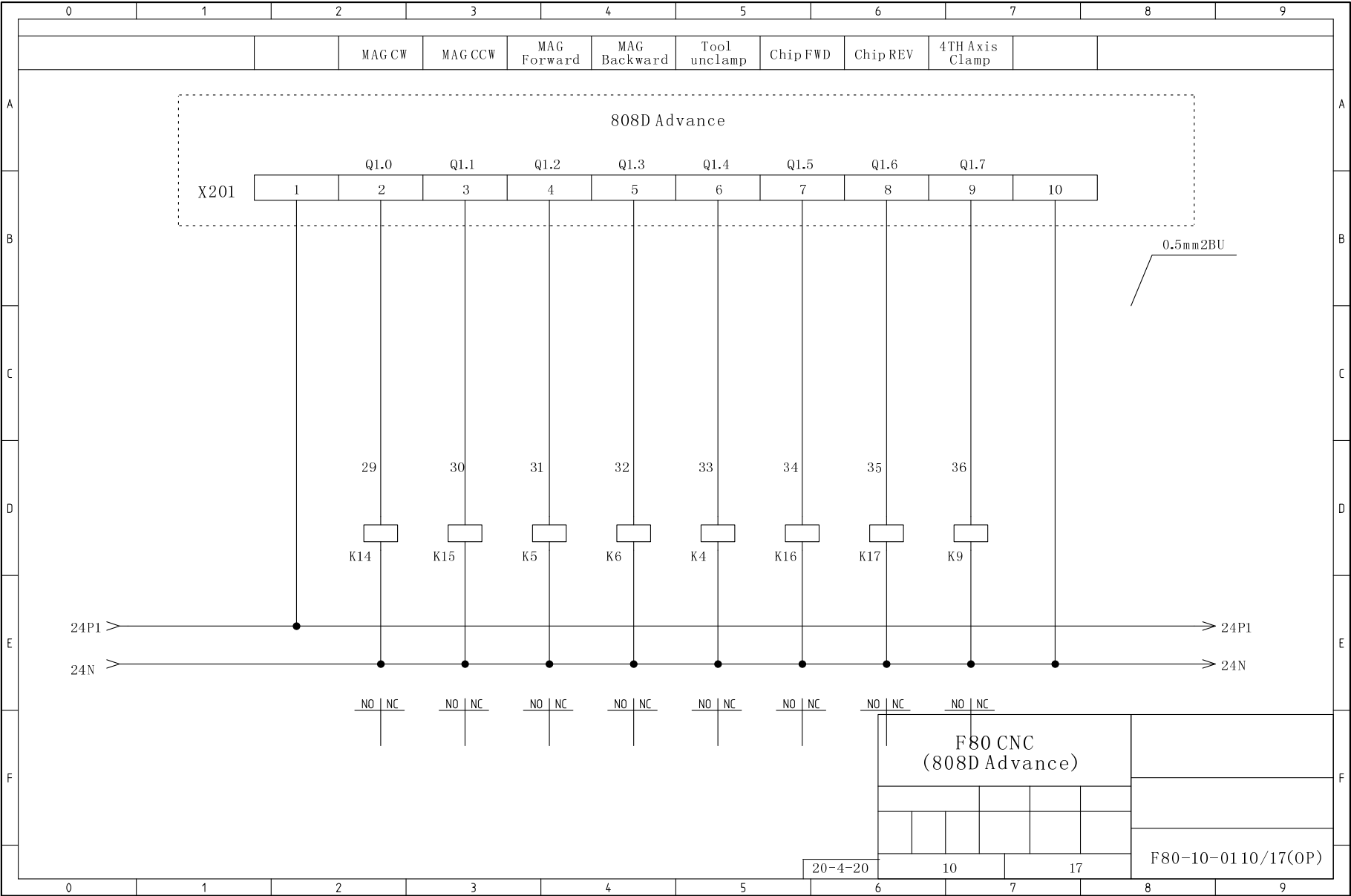
F80_advanced-wiring-diagram_20-04-2020.fm

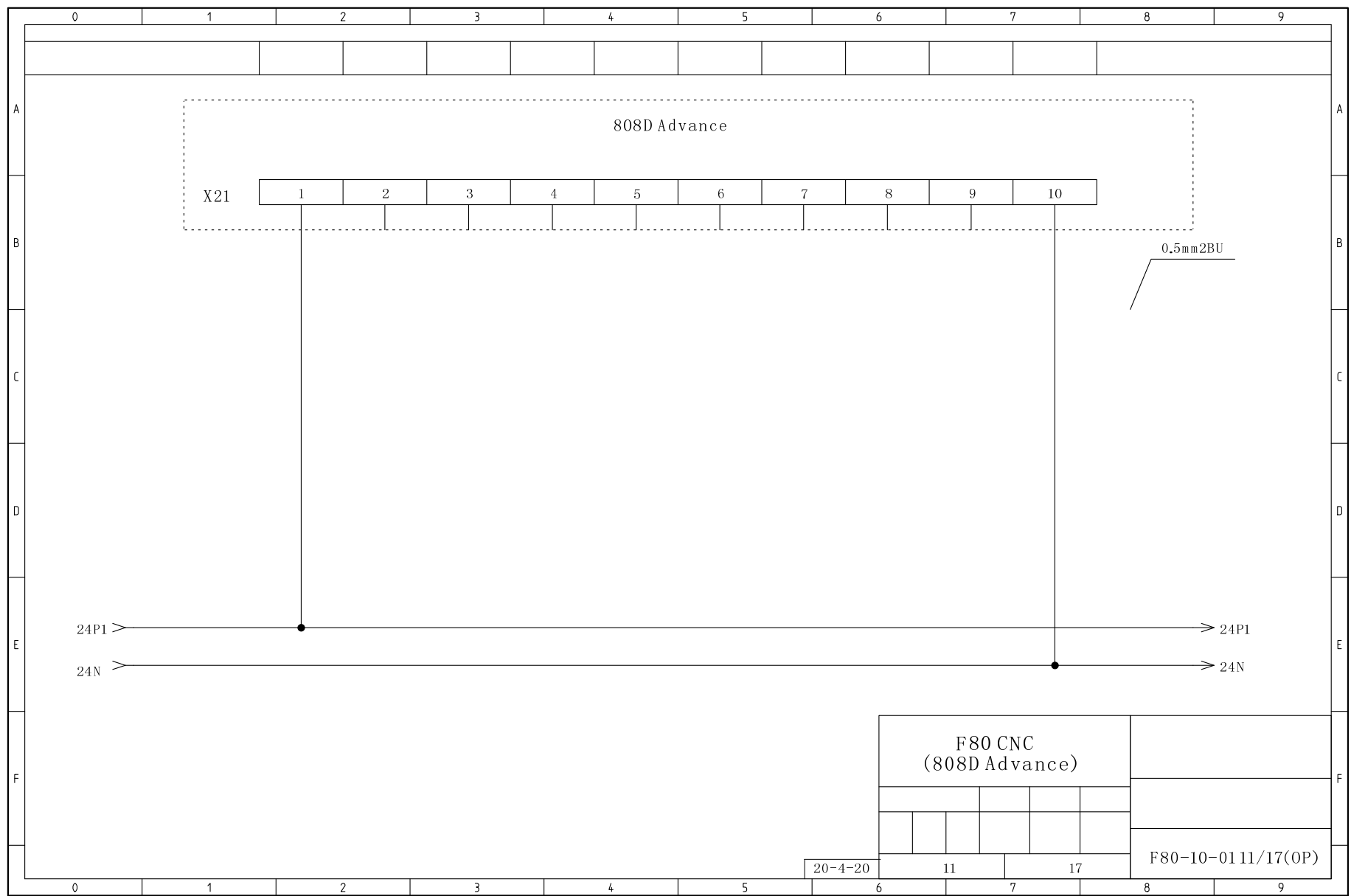


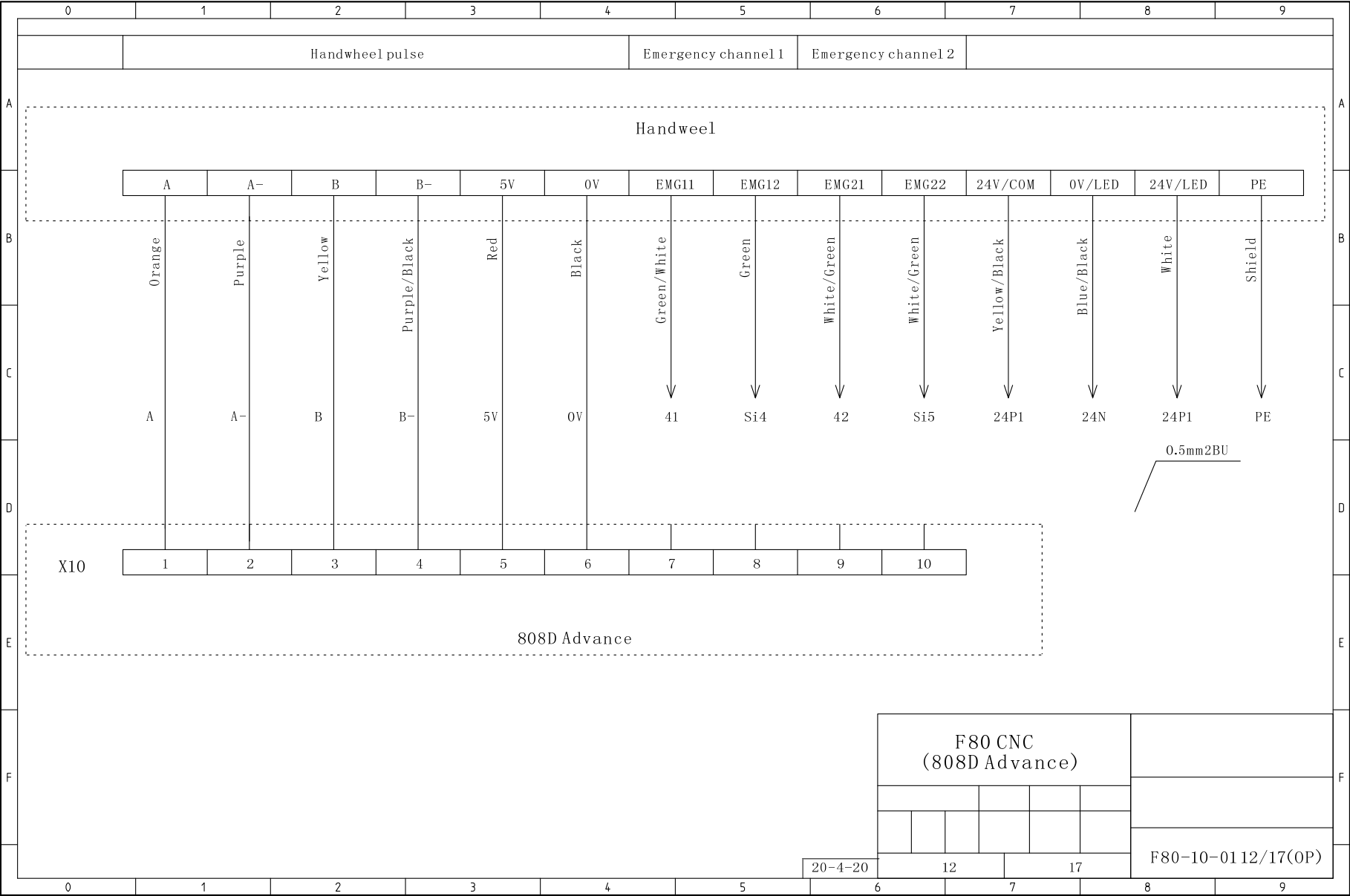


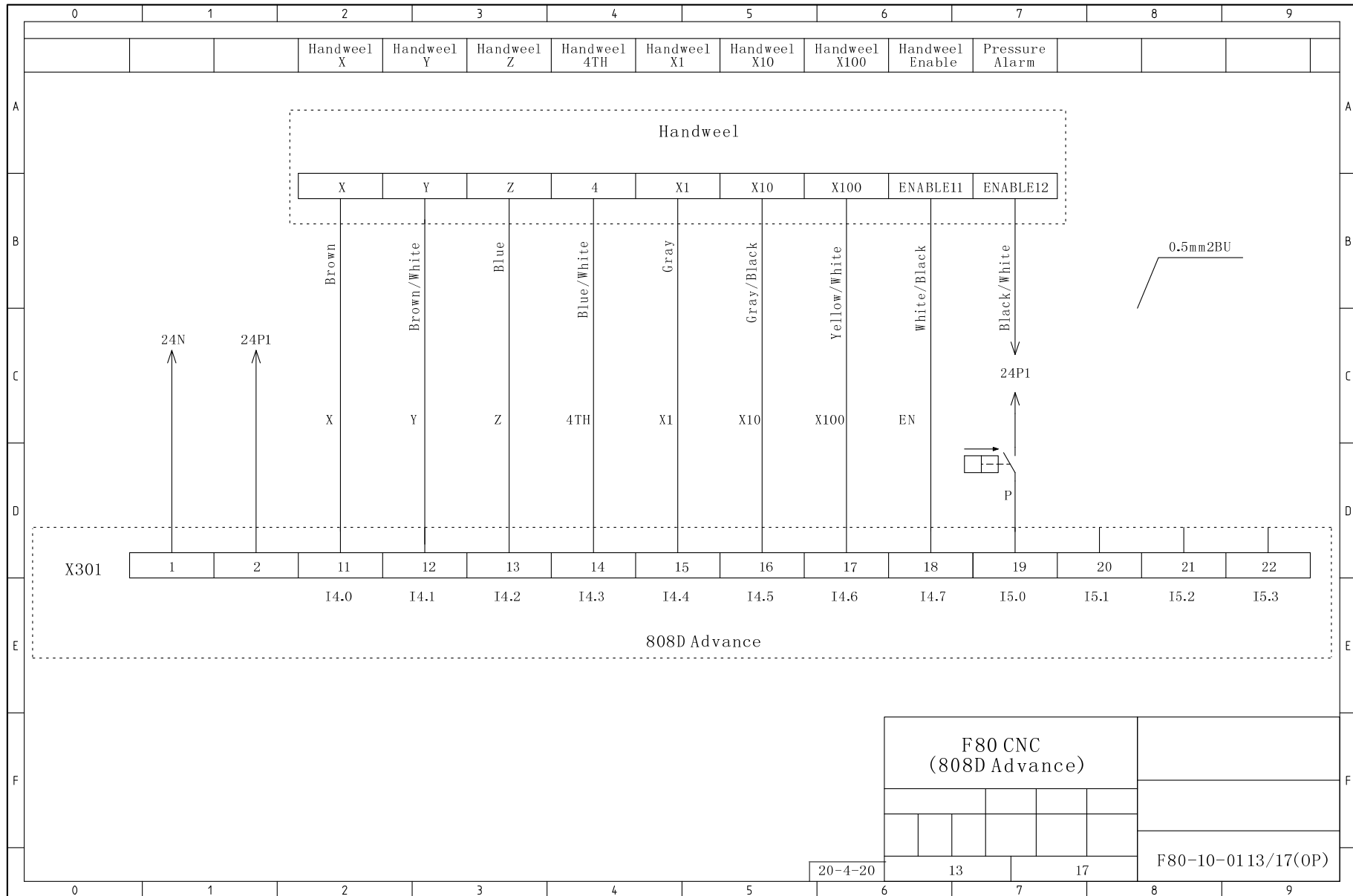


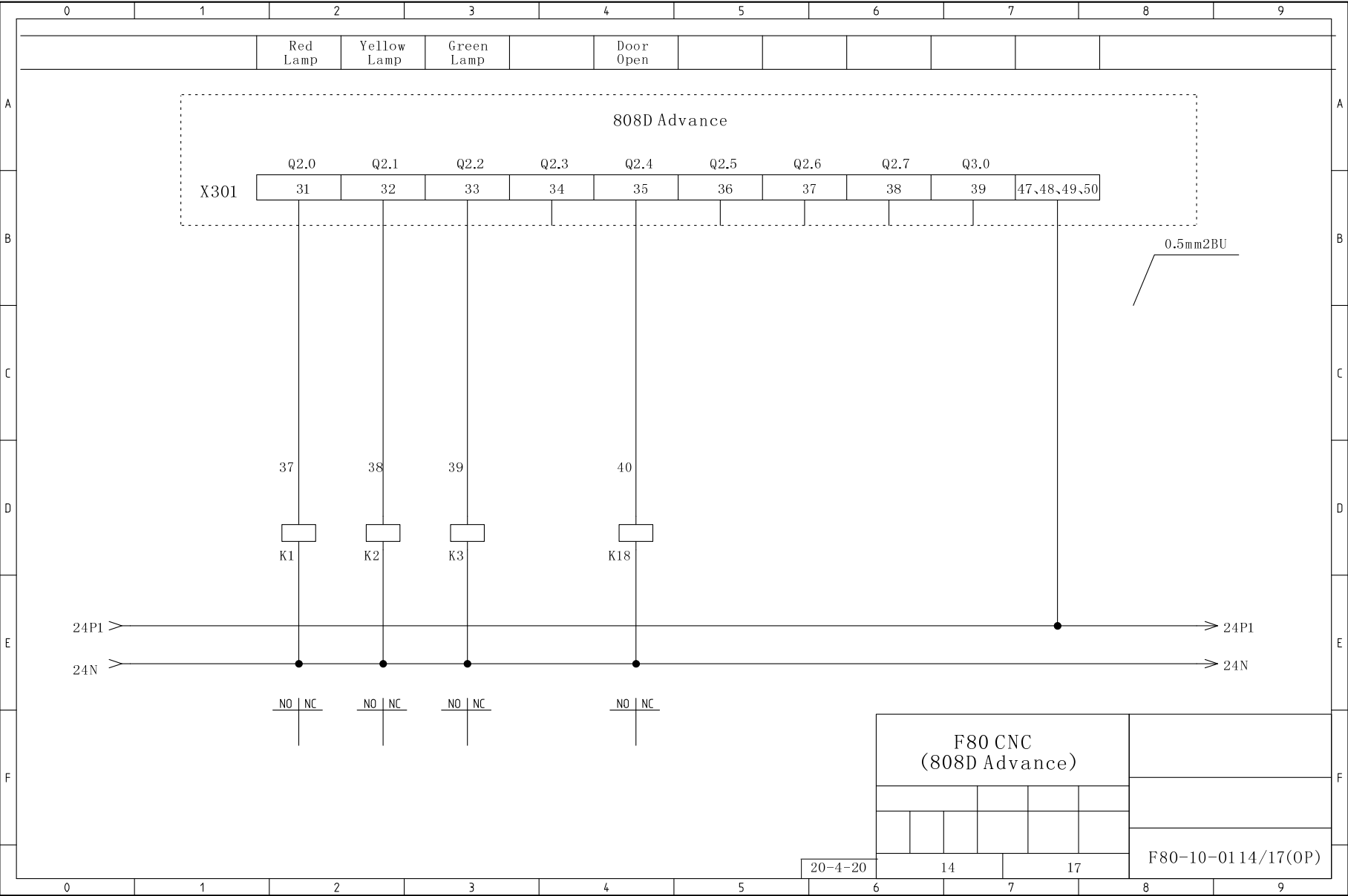


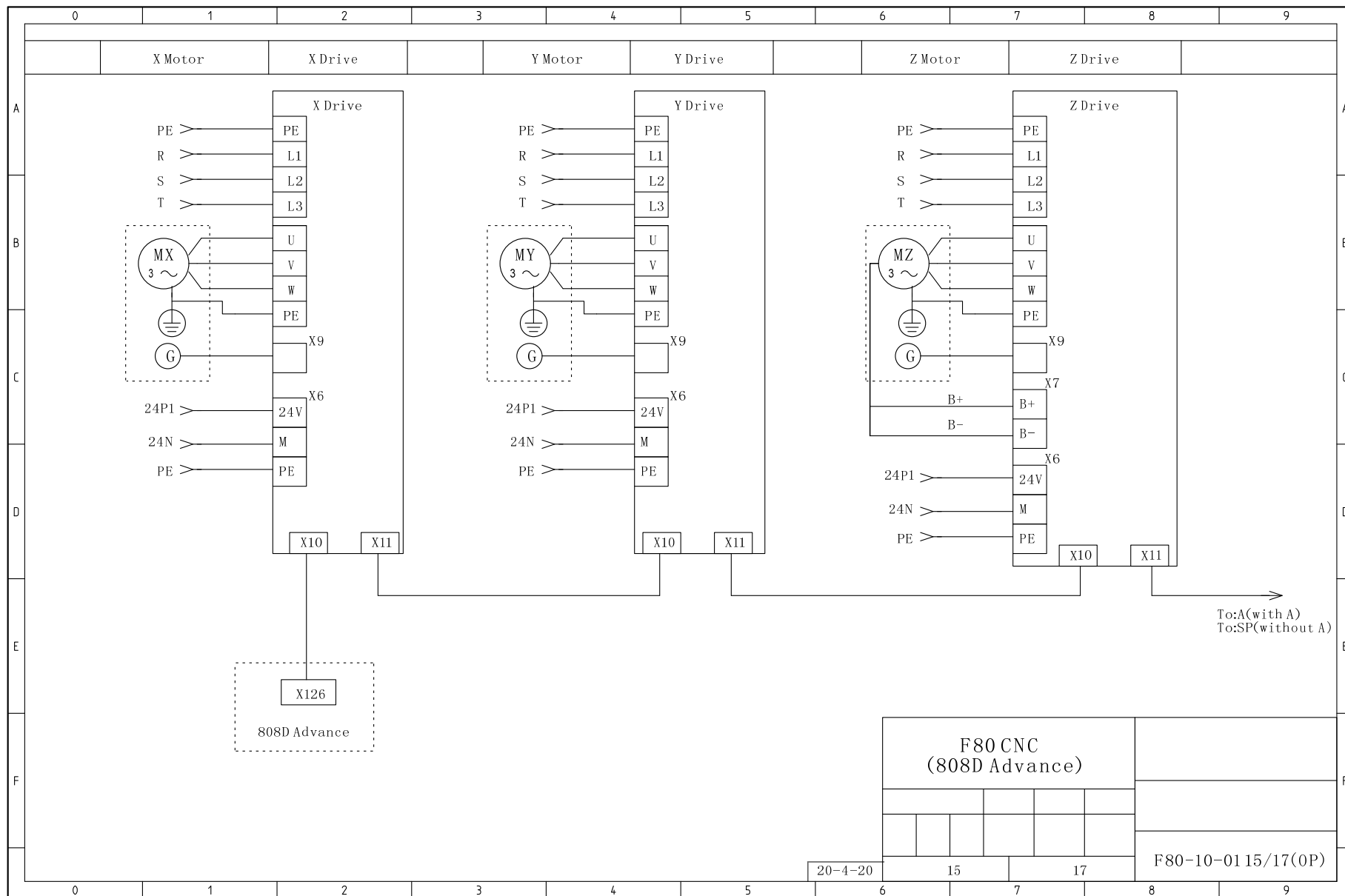


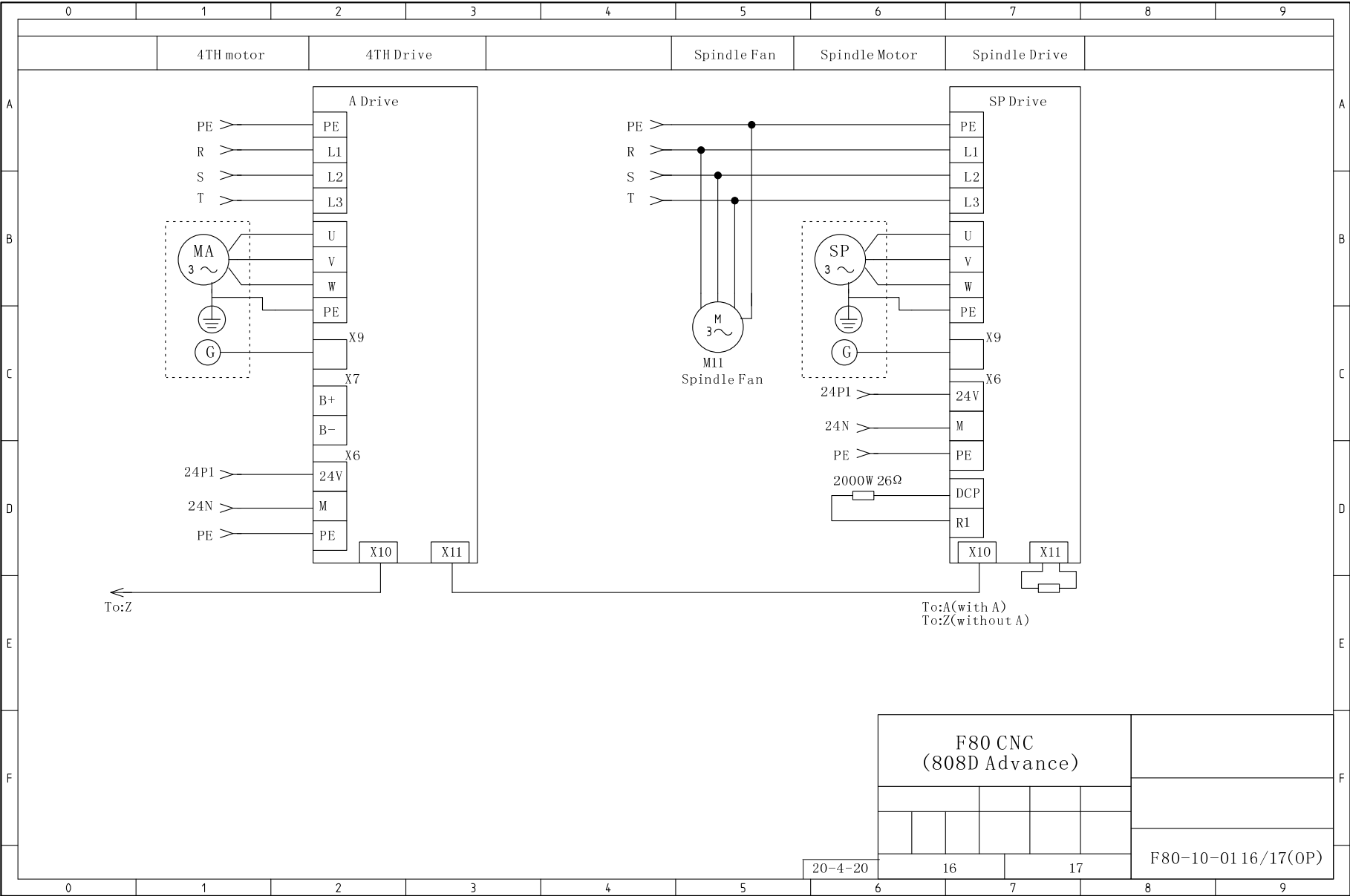


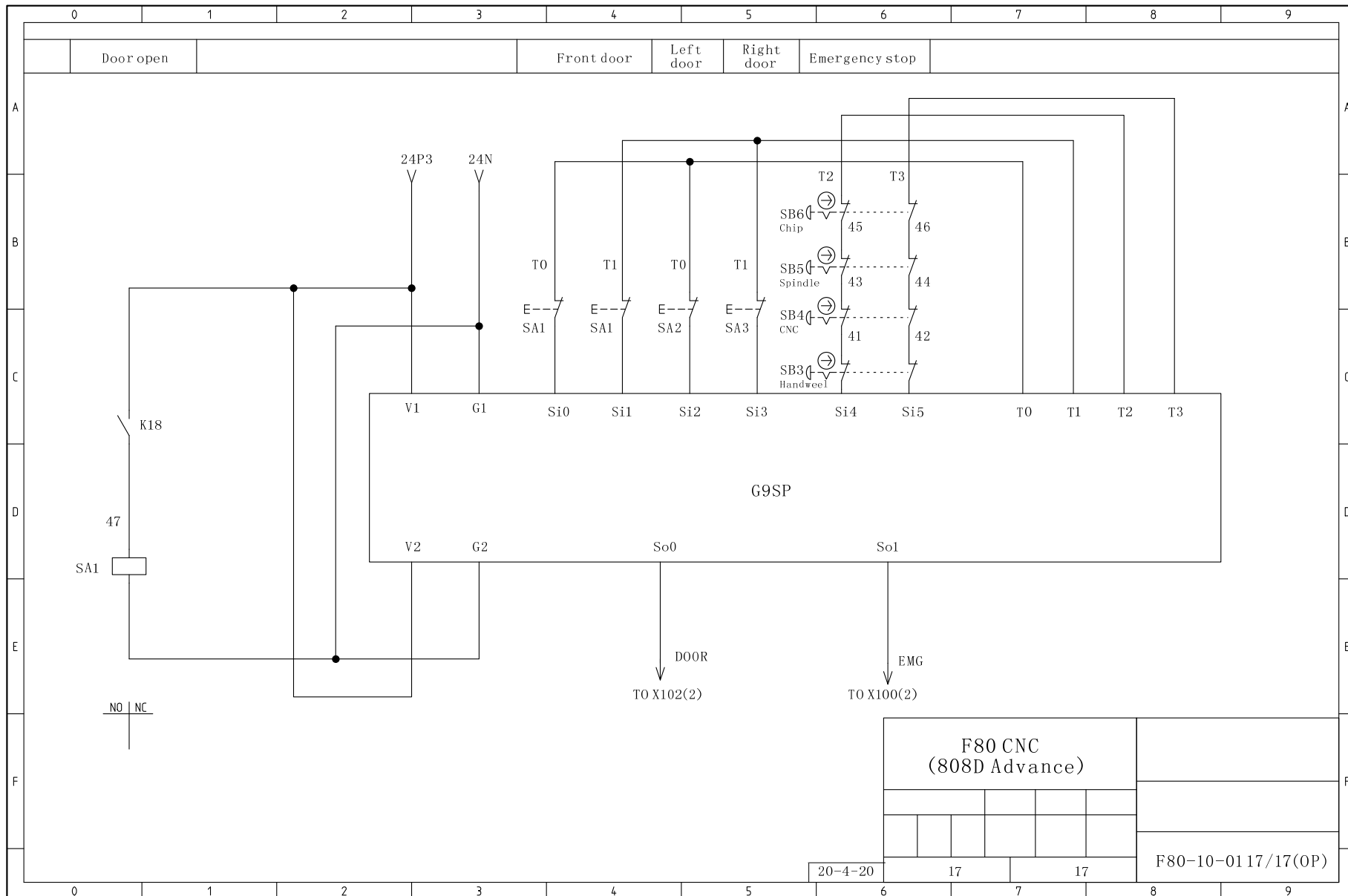












12 Brief instruction 808D Milling



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.

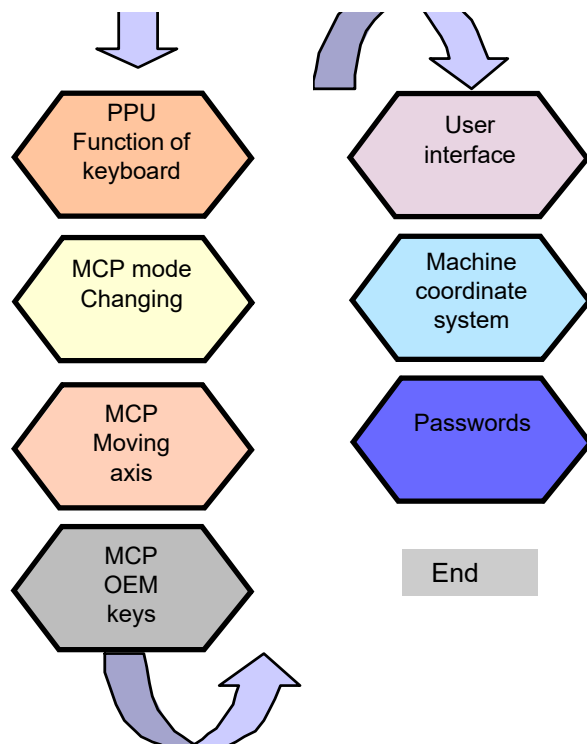
This brief instruction is not the replacement for Siemens 808D manuals and only serves as a reference for quickly find already-known operations and functions.

Preparation

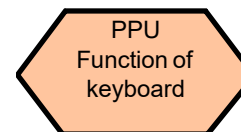
Description

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

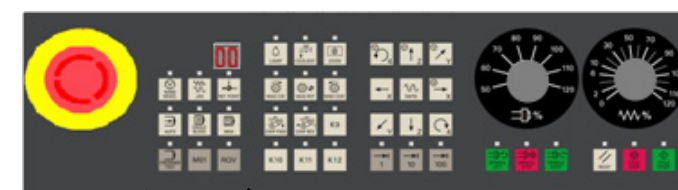
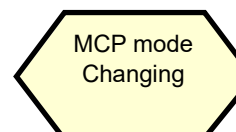


Basic Theory



Menu navigation

Operating area navigation



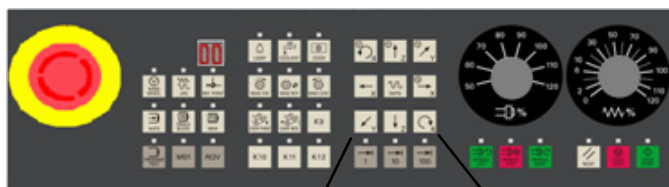
Mode navigation

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

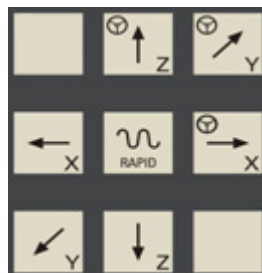


The 808D panel processing unit (PPU) is used to input data to the CNC and to navigate to operating areas of the system.

MCP
Moving
axis



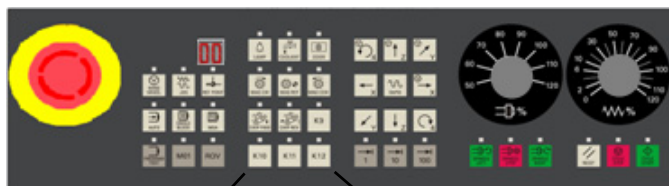
Axis movement



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



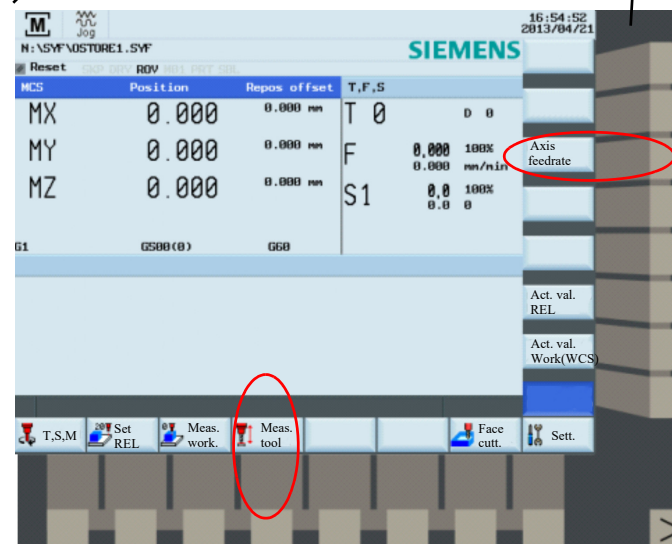
OEM keys



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

The machine functions can be activated with the appropriate keys.

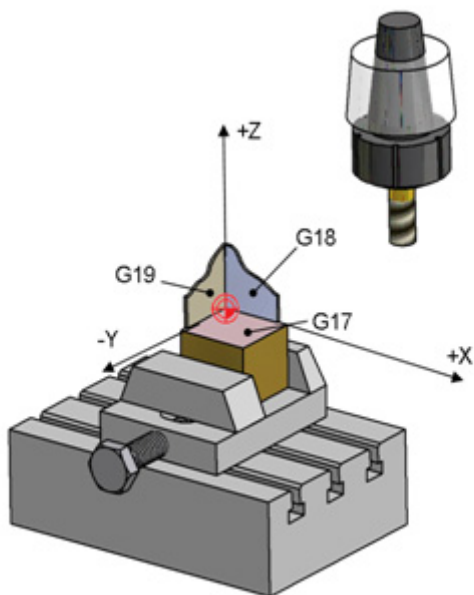
User
interface



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

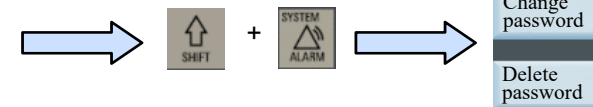
Changing password

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

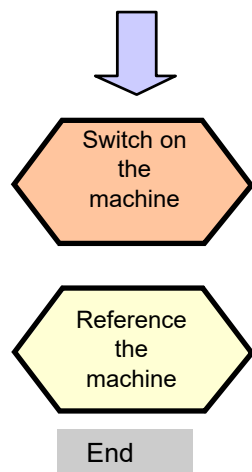
End

Switch On and Referencing

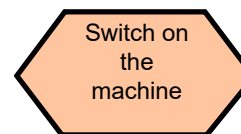
Description

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

Content



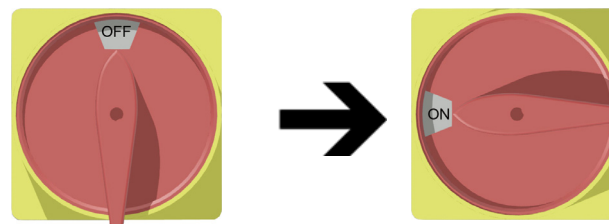
SEQUENCE



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

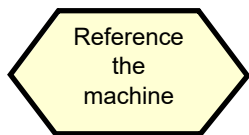
Make sure you perform the following operation!

Release all the emergency stop buttons on the machine!



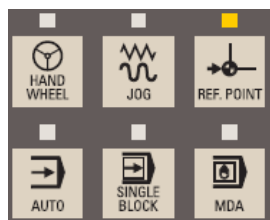
End

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

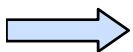
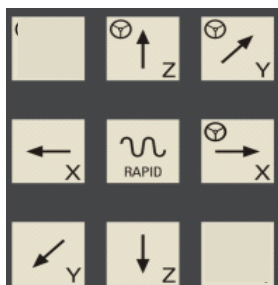


After power on, the machine will be in the reference point approach mode (default).

Ref Point		
N:\SYF\STORE1.SVF		
Reset	SKP DRY	ROV
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
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
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.

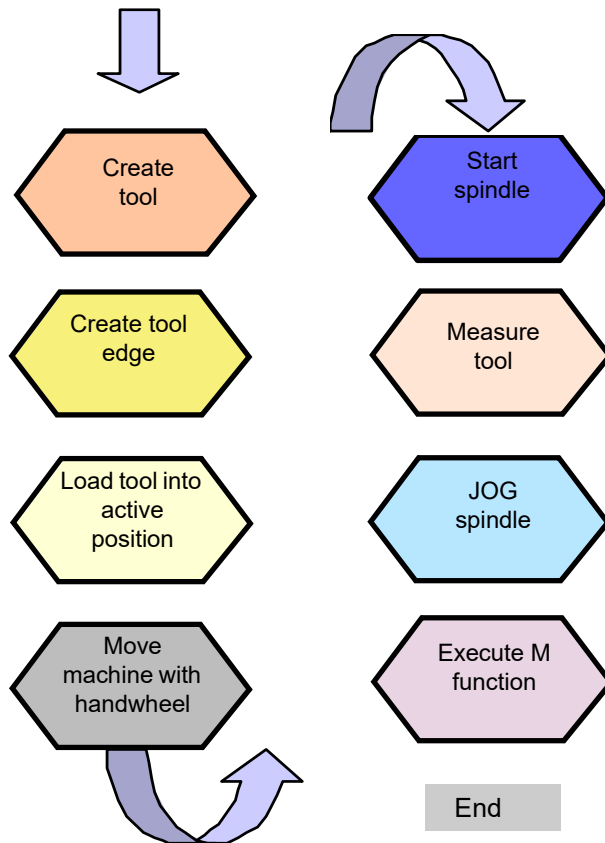
End

Tool Setup

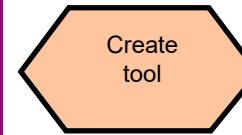
Description

This unit describes how to create and set up tools.

Content



SEQUENCE



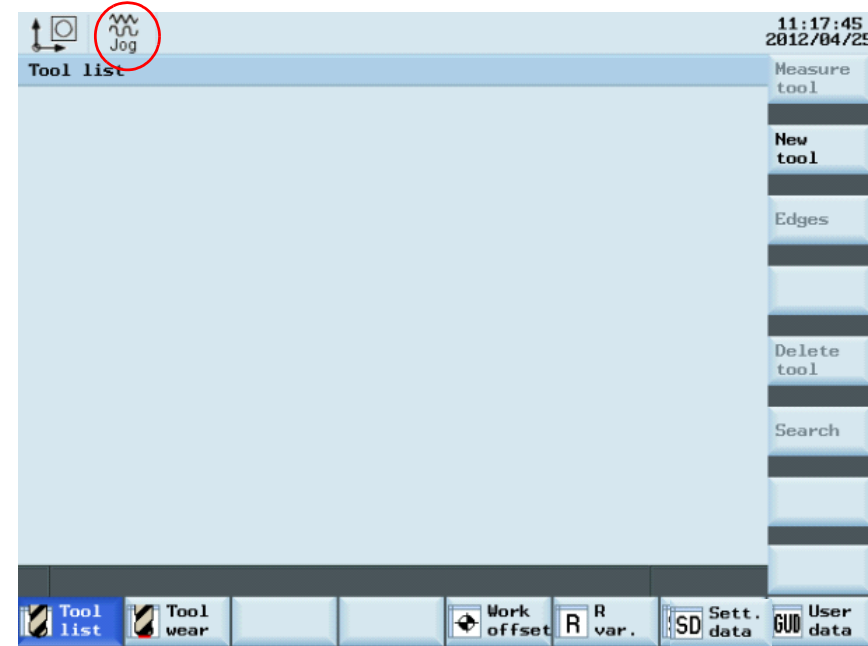
A tool must have been created and measured before executing the program.

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

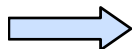
Step 2



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."

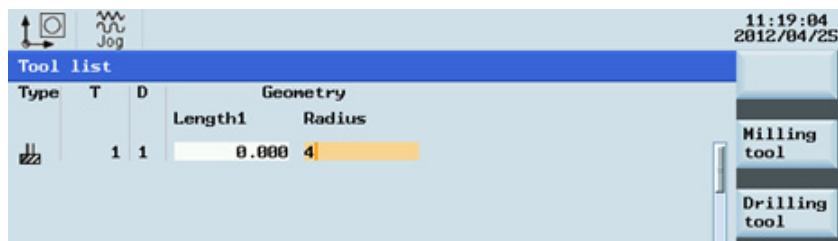


Press the "OK" SK on the PPU.



OK

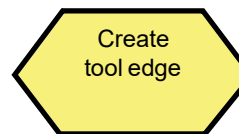
Enter the "Radius" of the milling tool.



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.



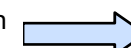
OFFSET

Press the "Tool list" SK on the PPU.

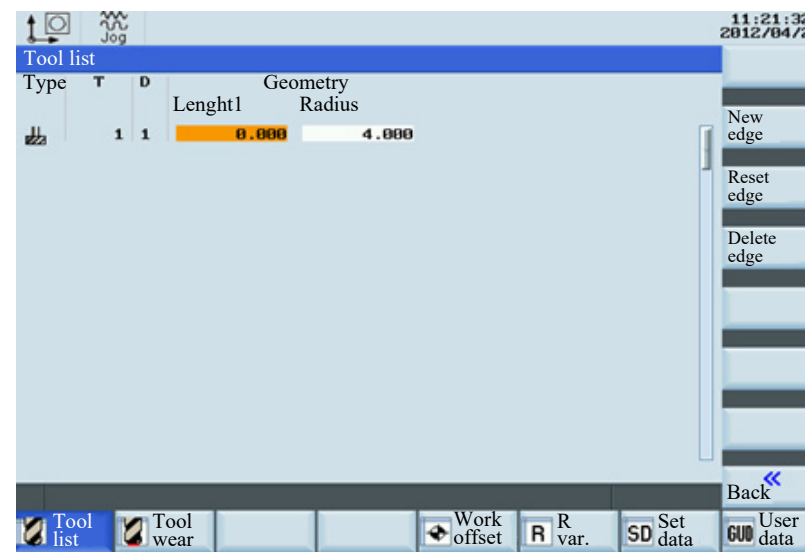


Tool list

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.



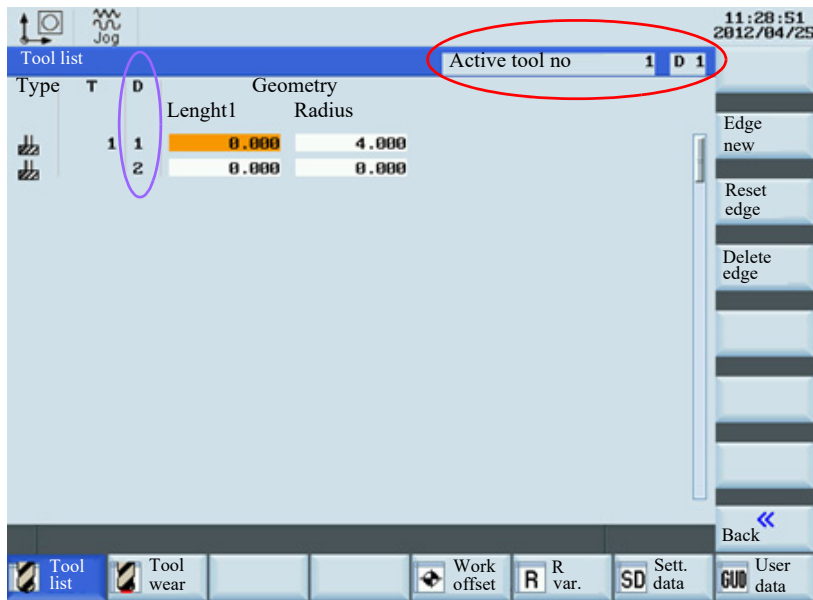
New edge

SEQUENCE

Step 2

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

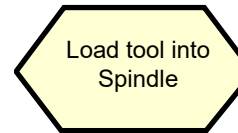
The red circle shows the actual active tool and tool edge, the purple circle shows how many tool edges have been created and the related data for each tool edge.



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!



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 →



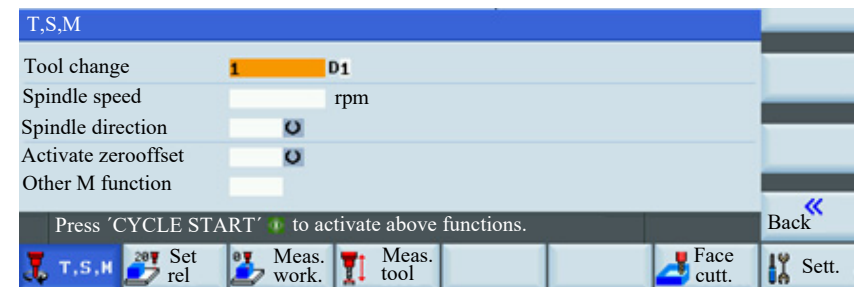
Press the "JOG" key on the MCP →



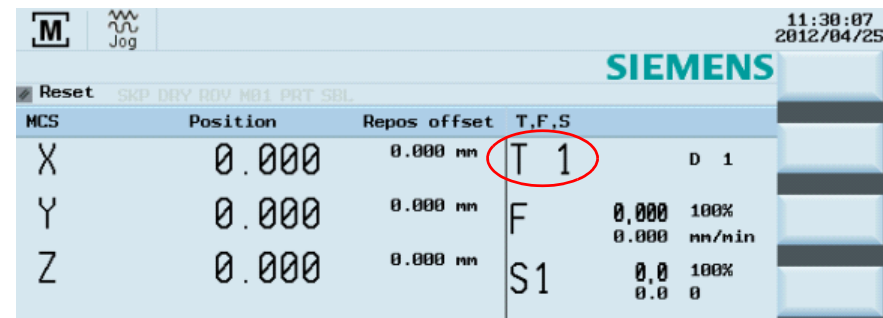
Press the "T.S.M" SK on the PPU →



Enter tool number "1" in "T" →



Press "CYCLE START" on the MCP →



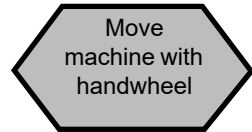
Press the "Back" SK on the PPU →



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.

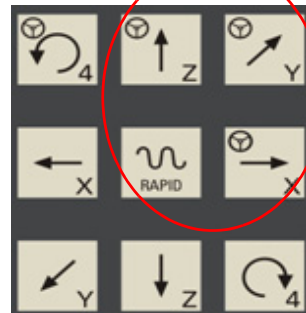
Press the “Machine” key on the PPU →



Press the “Handwheel” key on the MCP →



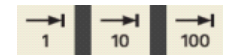
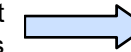
Select the axis you want to move with the appropriate keys. on the MCP →



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)



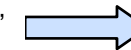
The handwheel increment is “0.001 mm” →



The handwheel increment is “0.010 mm” →



The handwheel increment is “0.100 mm” →

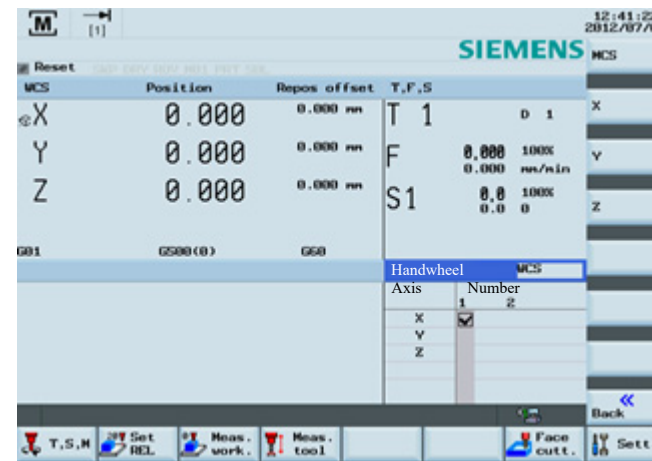


The selected axis can now be moved with the handwheel.

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



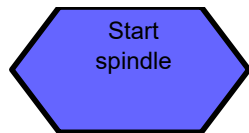
Notes: If set the MD14512[16]=80, the system will deactivate the function of MCP for selecting the axis of handwheel, the user will have to activate “Handwheel” function with PPU softkey.



Handwheel

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

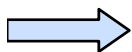
SEQUENCE



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

Start the spindle before adjusting tools as follows:

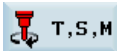
Press the "Machine" key on the PPU



Press the "JOG" key on the MCP



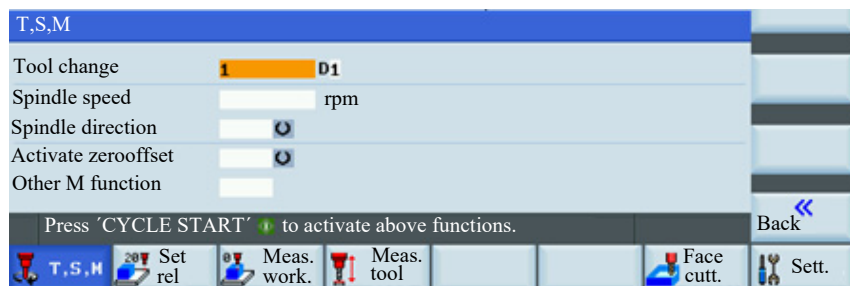
Press the "T.S.M" SK on the PPU



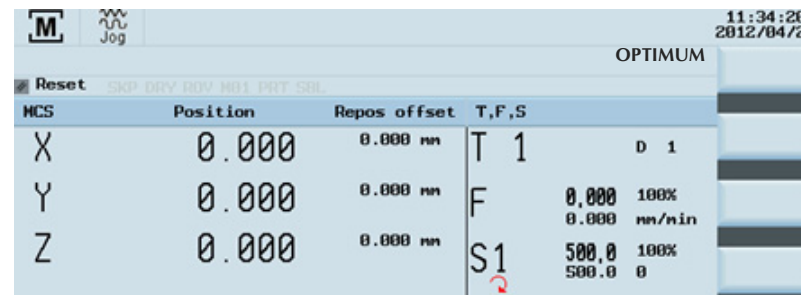
Enter "500" at "Spindle speed"



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



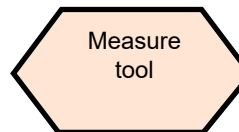
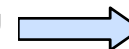
Press the "CYCLE START" key on the MCP



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

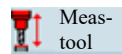
Press the "Machine" key on the PPU



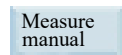
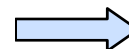
Press the "JOG" key on the MCP



Press the "Meas. tool" SK on the PPU

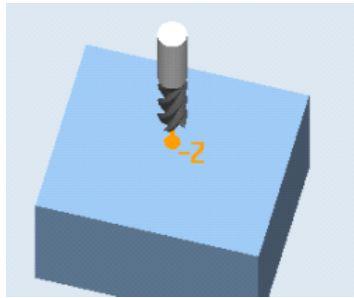
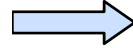


Press the "Measure manual" SK on the PPU



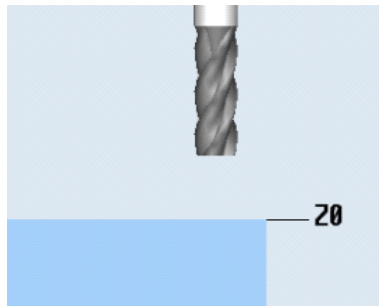
SEQUENCE

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

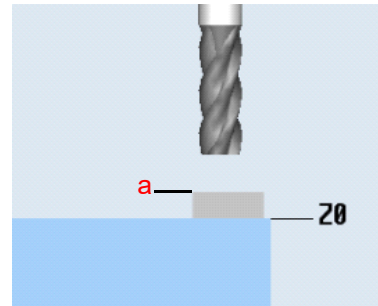


Note: The following text describes the required settings in the workpiece coordinate system
“X / Y / Z” zero points as: “X0” / “Y0” / “Z0”

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



or



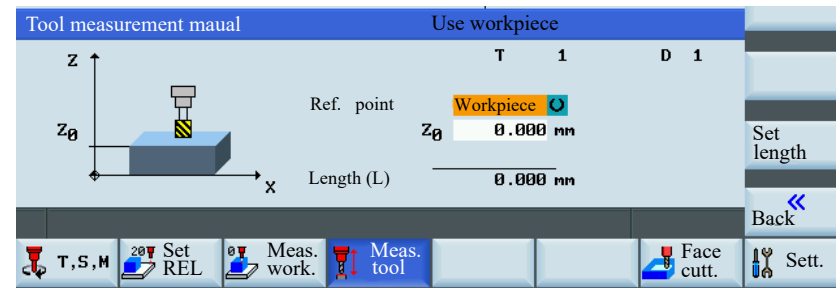
Use a setting block.

Move directly to zero point

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.

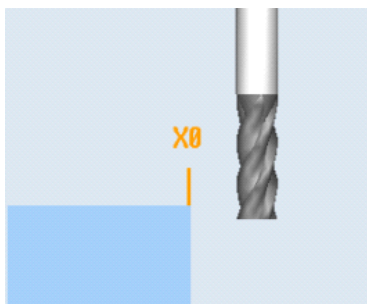
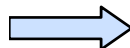
Step 2 Measure diameter

Press the “Diameter” SK on the PPU

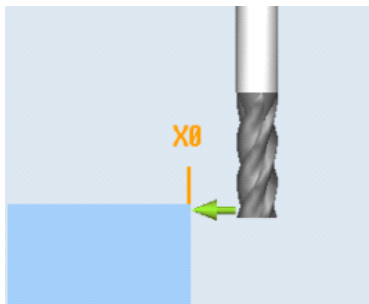


SEQUENCE

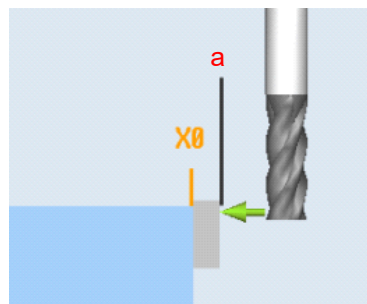
Press the axis keys on the MCP to move the tool to the set position.



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



or

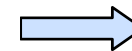


Move directly to zero point

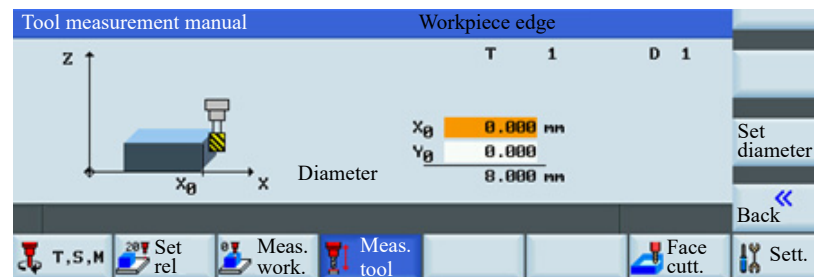
Use a setting block.

Enter "0" at "X0"

Enter "0" at "Y0"



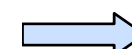
(This is the value of the width of a setting block if it is used. Select one of X0/Y0 according to requirement.)



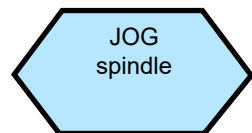
Press the "Set diameter" SK on the PPU



Press the "Back" SK on the PPU



SEQUENCE

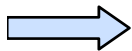


A tool must be loaded to the spindle.

Press the "Machine" key on the PPU



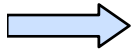
Press the "JOG" key on the MCP



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.

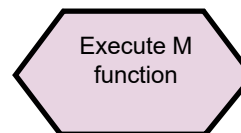


11:37:39
2012/04/25

SIEMENS

Reset SKP DRY ROV M01 PRT SBL

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 50.0 100% 50.0 0



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

Press the "Machine" key on the PPU



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



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



T,S,M	
Tool change	D1
Spindle speed	rpm
Spindle direction	
Activate zerooffset	
Other M function	8
Press 'CYCLE START' to activate above functions.	
Back	

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.

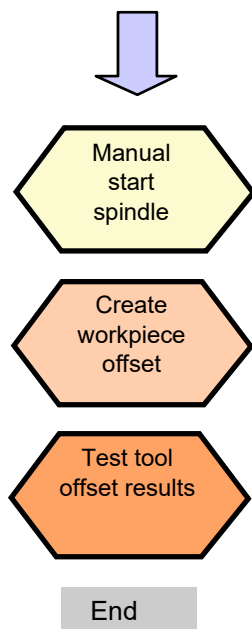


Workpiece Setup

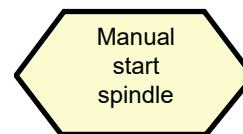
Description

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

Content



SEQUENCE



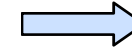
A tool must have been loaded into the spindle.

Before measuring, the spindle can be started as follows:

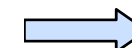
Press the “Machine” key on the PPU



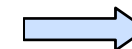
Press the “JOG” key on the MCP



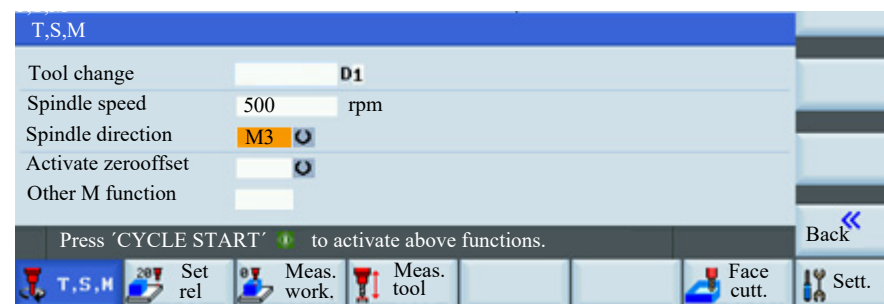
Press the “T.S.M” SK on the PPU.



Enter “500” at “Spindle speed”



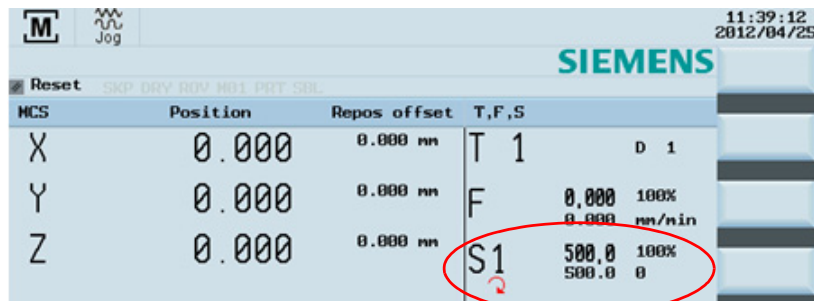
Select “M3” using the “Select” key on the PPU



Press the “CYCLE START” key on the MCP

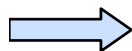


SEQUENCE

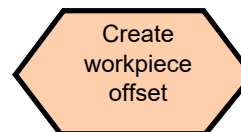
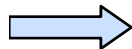


MCS	Position	Repos offset	T,F,S
X	0.000	0.000 mm	T 1 D 1
Y	0.000	0.000 mm	F 0.000 100% 0.000 mm/min
Z	0.000	0.000 mm	S1 500.0 100% 500.0 0

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.



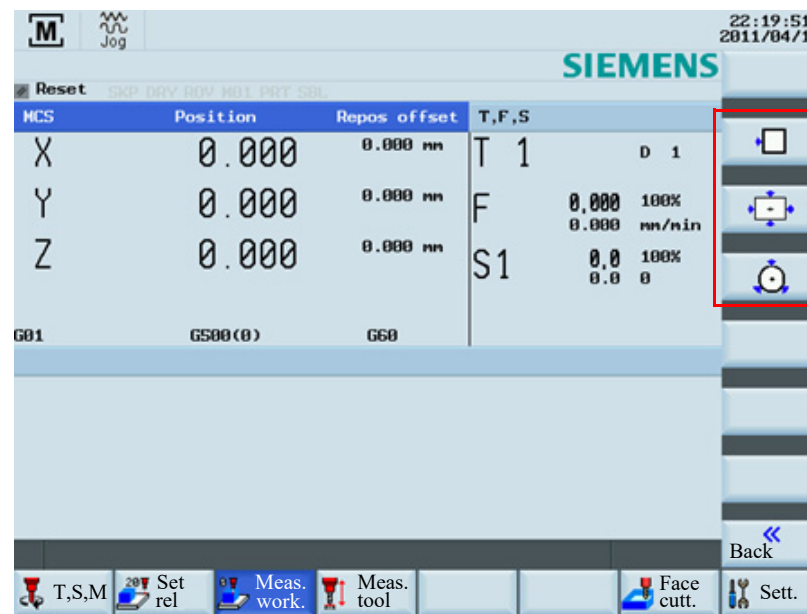
Press the "JOG" key on the MCP.



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.



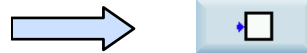
SEQUENCE

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.

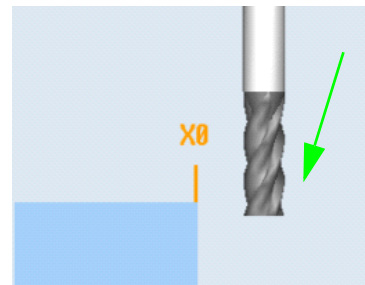
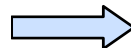
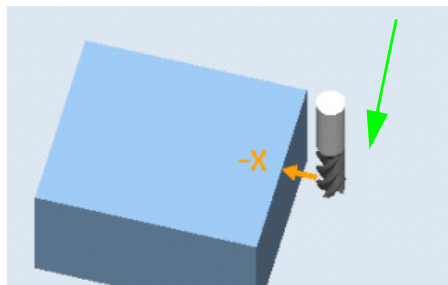
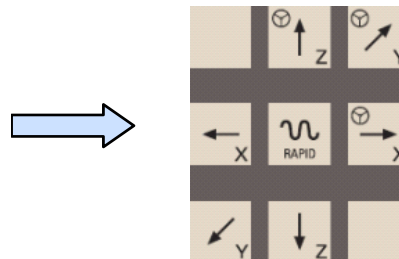
Press the corresponding SK of the first icon on the right-hand side of the PPU.



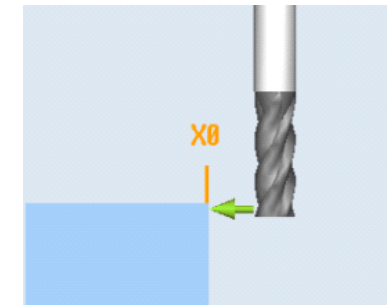
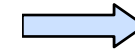
Press the appropriate SK to select the feed axis which needs to be set up.



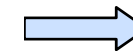
Press the axis traverse keys to move the tool to the required setting position in the X axis.



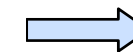
Press the “Handwheel” key on the MCP to position the tool at the X0 edge of the workpiece.



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



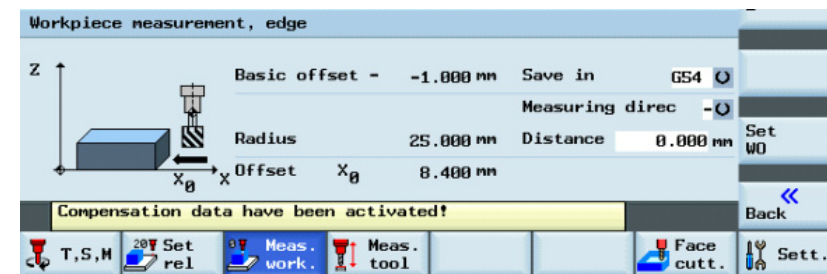
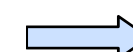
Select “Measuring direction” as “-”.
(This value should be chosen according to realities)



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.



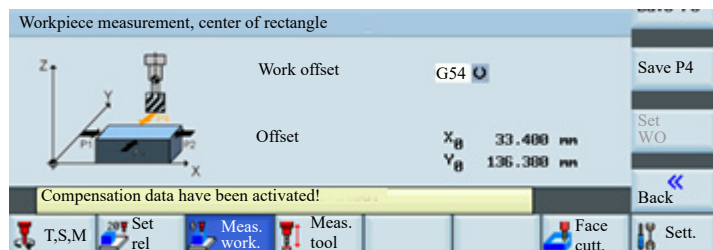
Repeat the process for positions 2, 3 and 4.
(When the setting is complete, the buttons will be shown in blue.)



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.

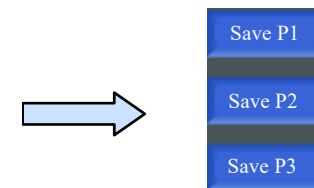


Observing the figure on the PPU, move the coordinate axis following the orange arrow to move the tool to the specified position and scratch the edge of the workpiece.

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



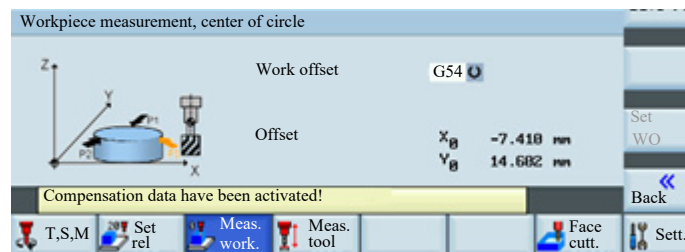
Repeat the process for positions 2 and 3.
(When the setting is complete, the buttons will be shown in blue.)



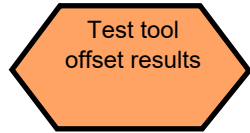
Press the "Set WO" SK on the PPU.



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



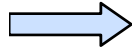
SEQUENCE



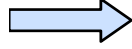
The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

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

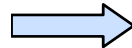
Press the "Machine" key on the PPU



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).

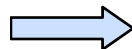


Note: The ROV function activates the feedrate override switch under the G00 function.



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

Press "CYCLE START" on the MCP.



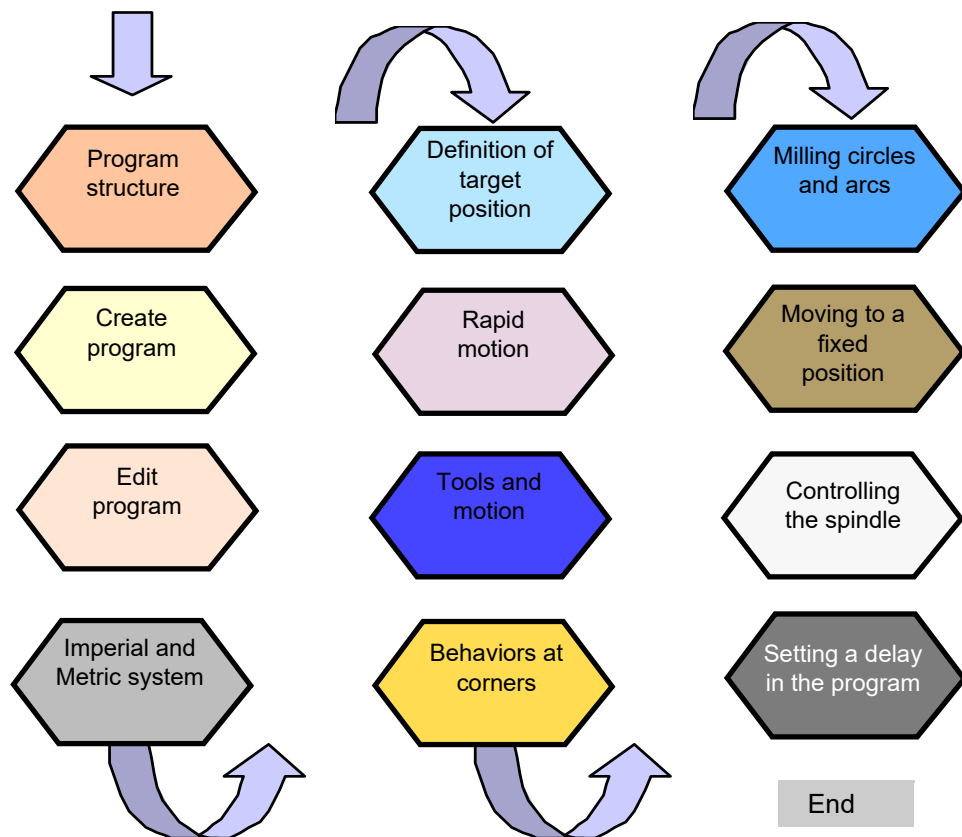
Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast. Observe whether the axis moves to the set position.

Create Part Program Part 1

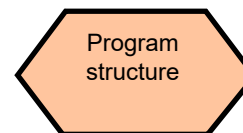
Description

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.

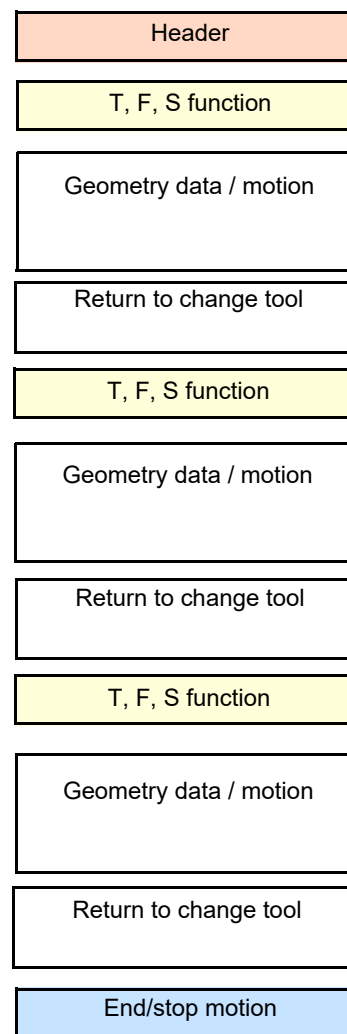
Content



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.



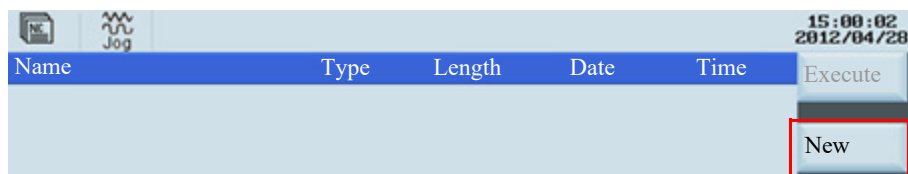
Step 2

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.

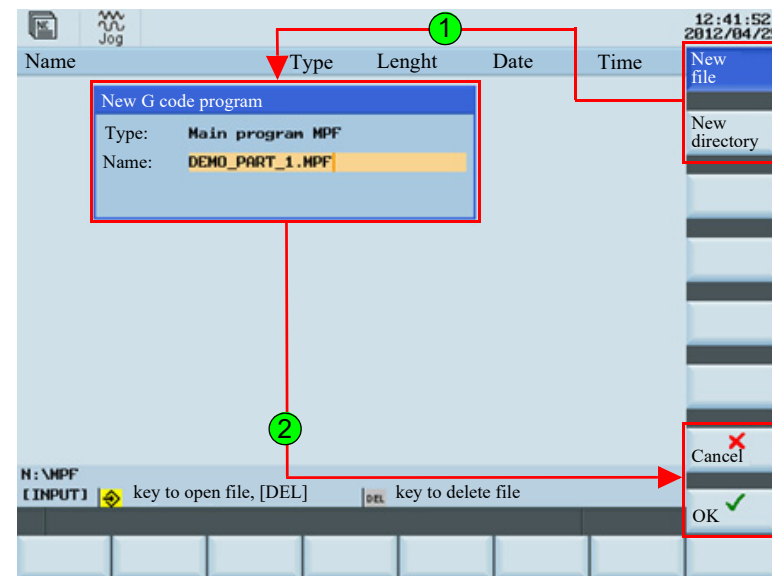


Step 4

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

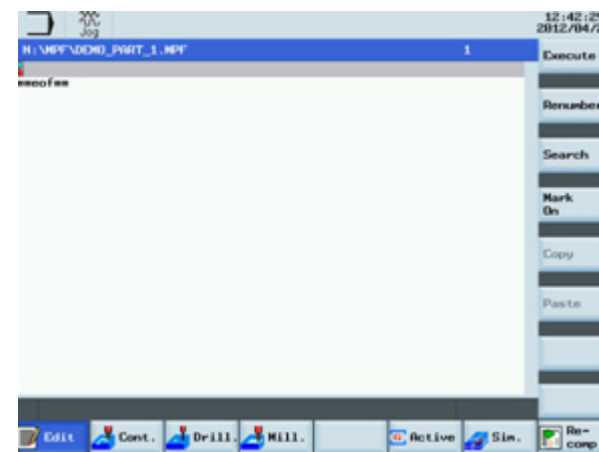
Choose “New” to create a program.

Choose “New directory” to create a file.



Step 5

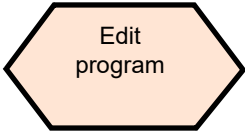
Now the program is opened and can be edited.



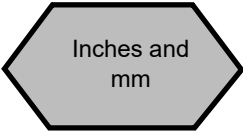
The system will save it automatically after editing.

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 X**100** Y**100** Z5

N25 G01 Z-5

N30 Z5

N35 G00 Z**500** 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

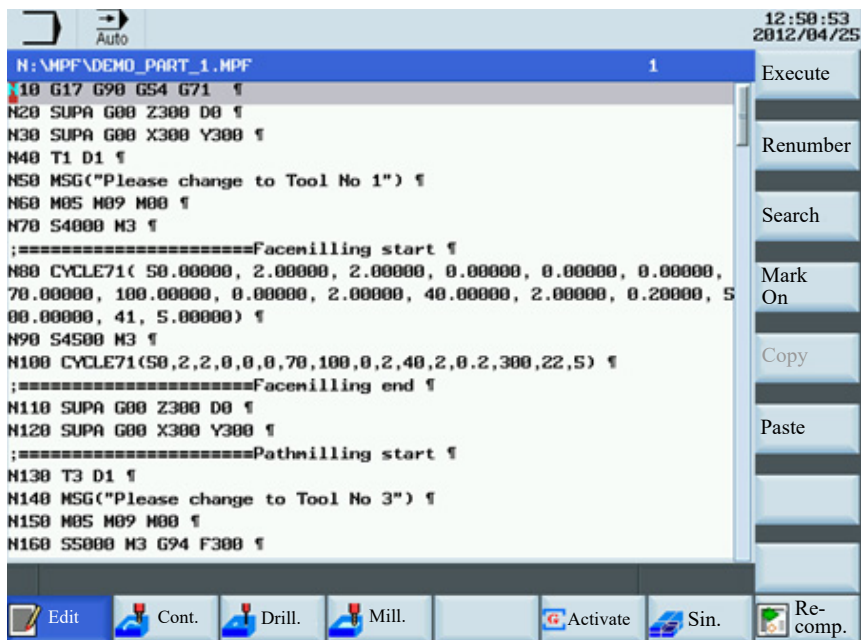
N10 T1 D1 M6

N15 S5000 M3 G94 F300

N20 G00 X**3.93** Y**3.93** Z5N25 G01 Z-**0.787**N30 Z**0.196**N35 G00 Z**19.68** D0

Geometry data / motion

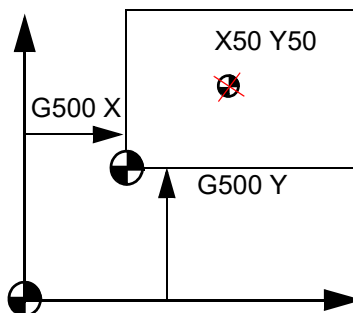
Return to change tool



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

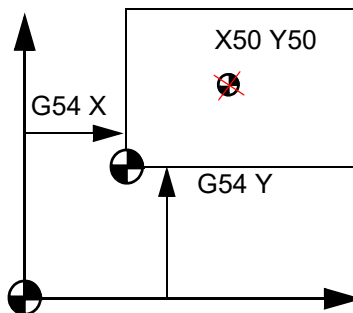
N30 **Z5**

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

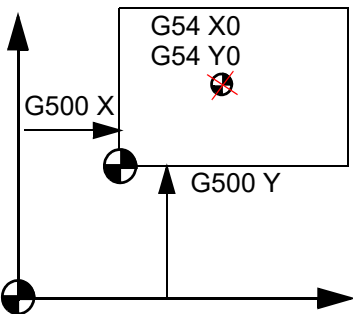
N30 **Z5**

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.

N5 G17 **G90** G54 G71

N10 T1 D1 M6

N15 S5000 M3 G94 F300

N20 G00 **X100 Y100 Z5**

N25 G01 Z-20

N30 **Z5**

N35 G00 **Z500** D0

G91

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

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

N5 G17 **G90** G54 G70

N10 T1 D1 M6

N15 S5000 M3 G94 F300

N20 G00 X3.93 Y3.93 Z0.196

N25 G01 **G91 Z-0.787**

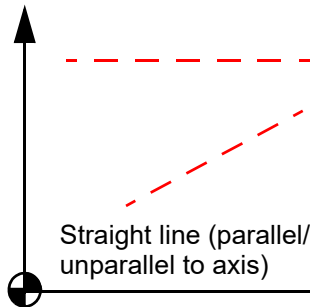
N30 **Z0.196**

N35 G00 **G90 Z19.68** D0

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

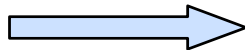
N35 G00 Z500 D0

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

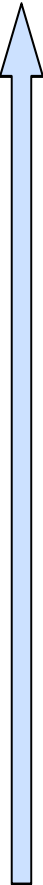
G40: Compensation of the radius can be deactivated



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



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



Arrow indicates the direction of tool motion along the contour.

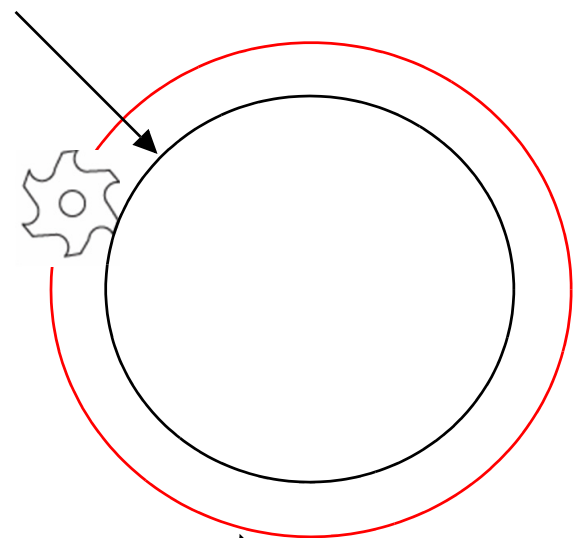
When traversing circular contours with cutter radius compensation, it should be decided whether the feed rate should be calculated along the contour of the workpiece or along the path defined by the center point of the cutting tool.

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

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

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

Contour feedrate with **CFC**



Direction for compensation, left of contour will be **G41**

Feedrate calculated when using tool center, inside or outside of the contour

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!

```

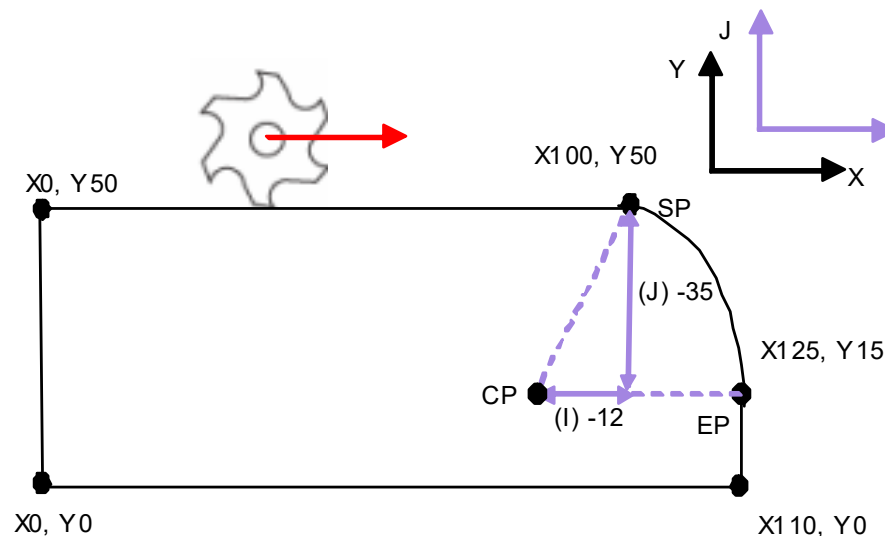
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X-20 Y-20 Z5
N25 G01 Z-5
N30 G41 X0 Y0
N35 Y50
N40 X100
N45 G02 X125 Y15 I-12 J-35
N50 G01 Y0
N55 X0
N60 G40 X-20 Y-20
N35 G00 Z500 D0
  
```

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

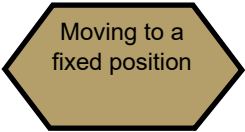
I = defined relative increment from start point to center point in X

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



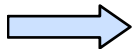
Moving to a fixed position

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

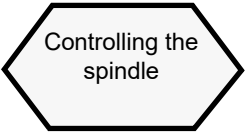


```
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 Z=0 ;reference point
```

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

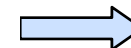
The following functions can be used to influence the operation of the spindle:

M3 accelerate to programmed speed clockwise

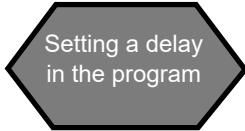
M4 accelerate to programmed speed counter-clockwise

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



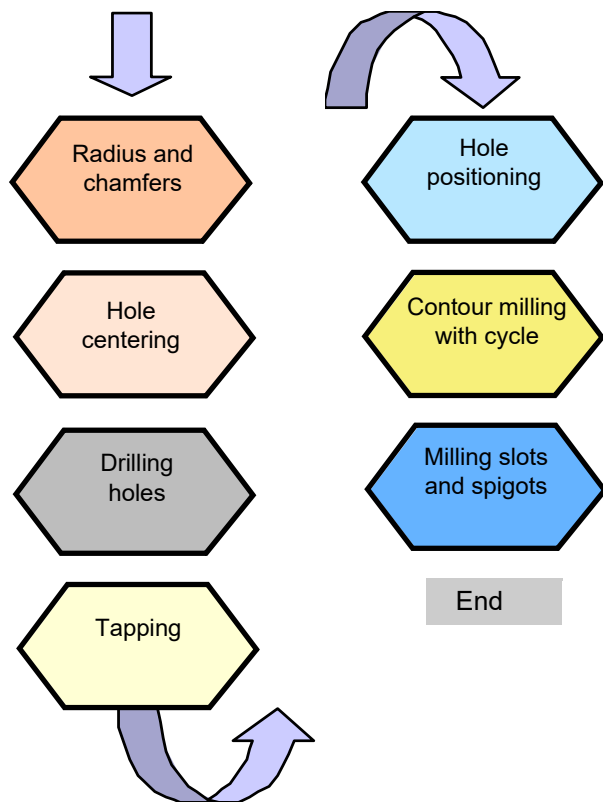
```
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 G04 F5
N35 Z5 M4
N40 M5
N45 M19
N35 G00 Z500 D0
```

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.

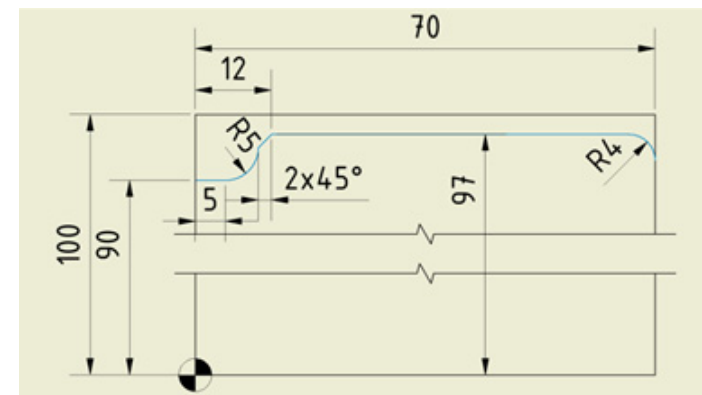
Content



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

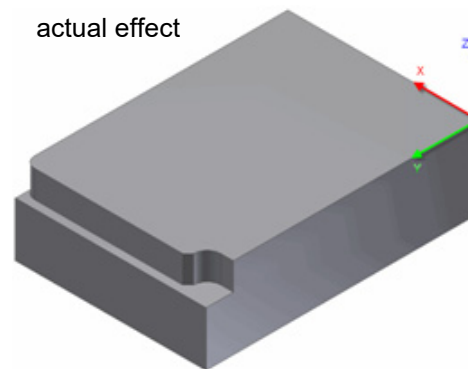
N115 G01 X70 RND=4

N120 G01 Y90

N125 G01 G40 X80

N130 G00 Z50

actual effect



Basic Theory

Hole
centring

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

CYCLE81: Without delay at current hole depth

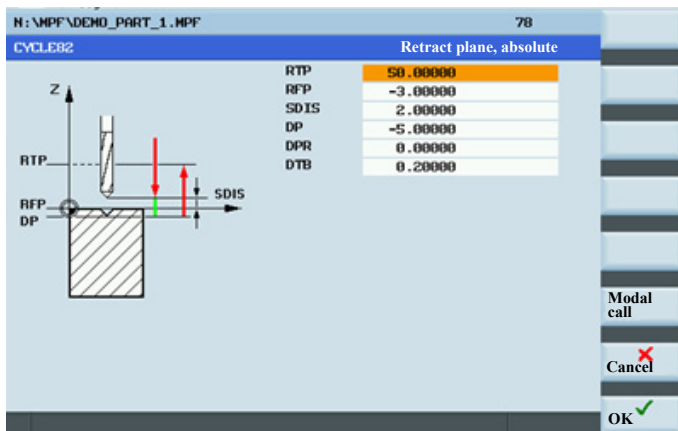
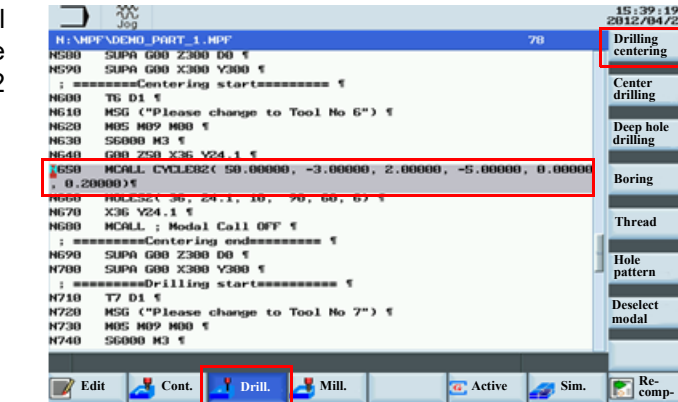
CYCLE82: With delay at current hole depth



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



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

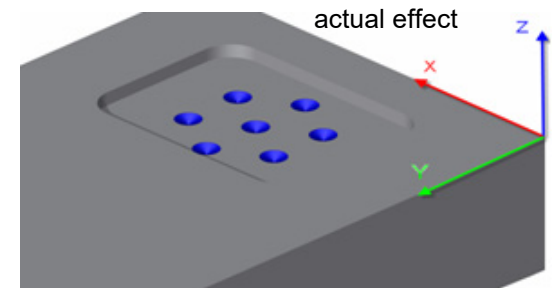
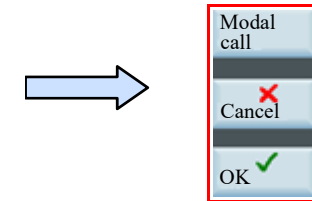


With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

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)

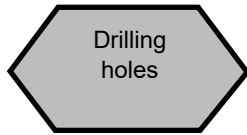
N330 X20 Y20 ; Hole will be centered

N335 X40 Y40 ; Hole will be centered

N340 MCALL

N345 X60 Y60 ; Hole will not be centered

Basic Theory



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

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

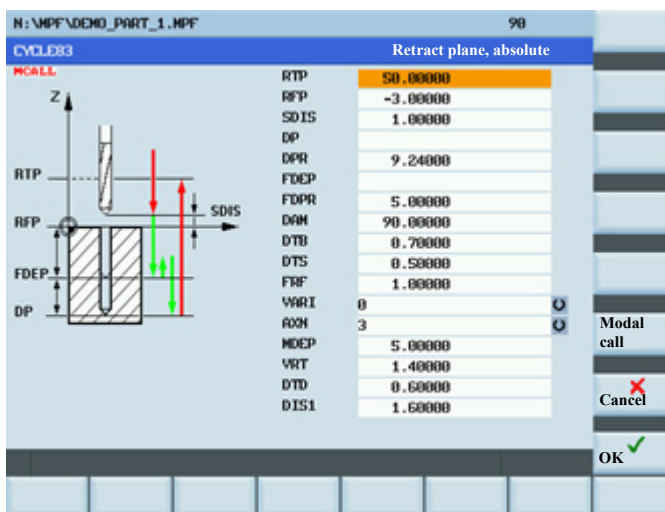
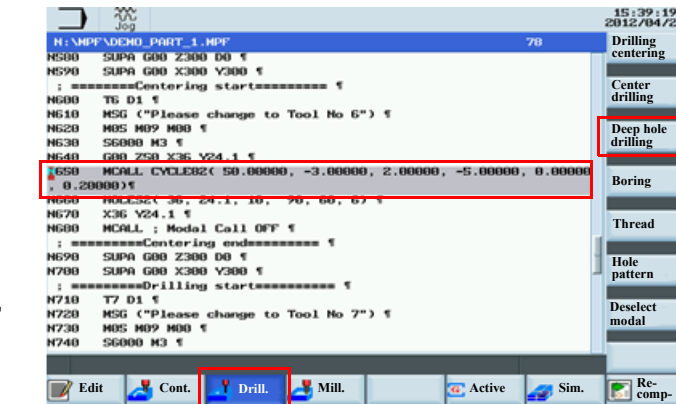
The cycle can be found and parameterized with the "Drill." SK.



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



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

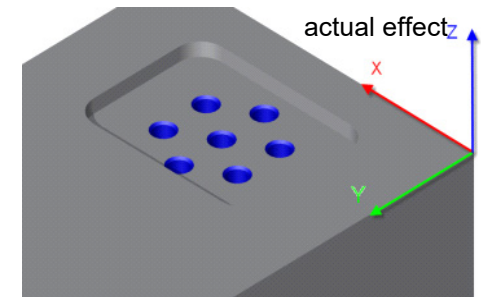
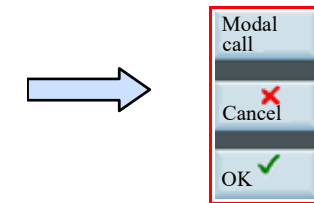


With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

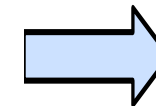
This will drill a hole at the current position.

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

The information is transferred as shown below.



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



For specific parameter commands, see the next page

N325 MCALL CYCLE83(50.00000, -3.00000, 1.00000, ,9.24000, ,5.00000, 90.00000, 0.70000, 0.50000, 1.00000, 0, 0, 5.00000, 1.40000, 0.60000, 1.60000)

N330 X20 Y20 ; Hole will be drilled

N335 X40 Y40 ; Hole will be drilled

N340 MCALL

N345 X60 Y60 ; Hole will not be drilled

Basic Theory



For descriptions of RTP, RFP, SDIS and DP, please see Page 144

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.

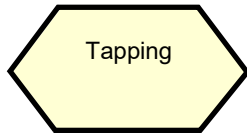
② DAM=0, drilling depth each time is same as the 1st drilling depth (FDPR),

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

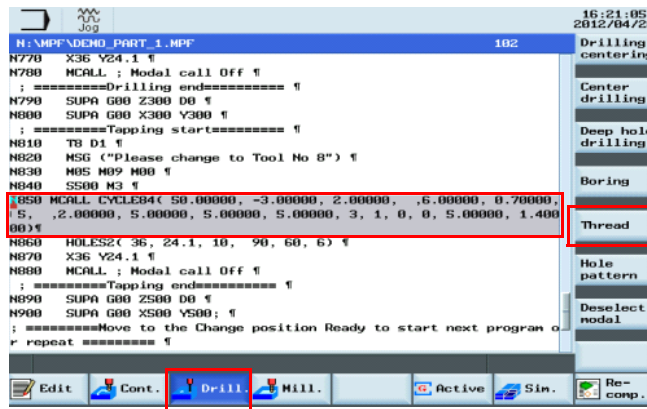
Feed times	Every feed depth/mm DAM=2	Actual depth/mm	Feed times	Every feed depth/mm DAM=0	Actual depth/mm
1.	FDPR=10	-10	1.	FDPR=10	-10
2.	FDPR-DAM=10-2=8	-18	2.	FDPR=10	-20
3.	(FDPR-DAM)-DAM=8-2=6	-24	3.	FDPR=10	-30
4.	(FDPR-2DAM)-DAM=6-2=4	-28	Remaining depth =10 < 2xFDPR, the remaining depth distribute 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.

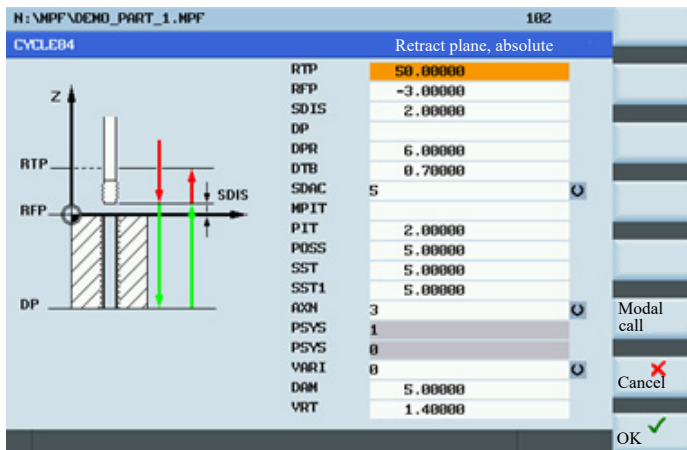
The cycles can be found and parameterized using the "Drill." SK.



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.



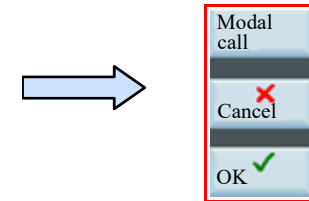
With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

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	



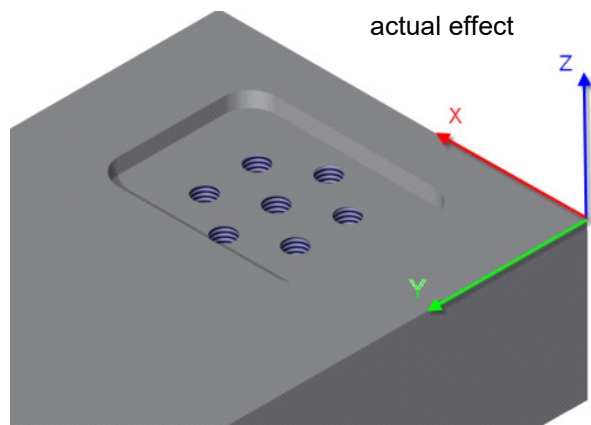
N325 MCALL CYCLE84(50.00000, -3.00000, 2.00000, ,6.00000, 0.70000, 5, ,2.00000, 5.00000, 5.00000, 5.00000, 3, 0, 0, 0, 5.00000, 1.40000)

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 144

For descriptions of AXH, VARI, DAM and VRT, please see Page 146

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.

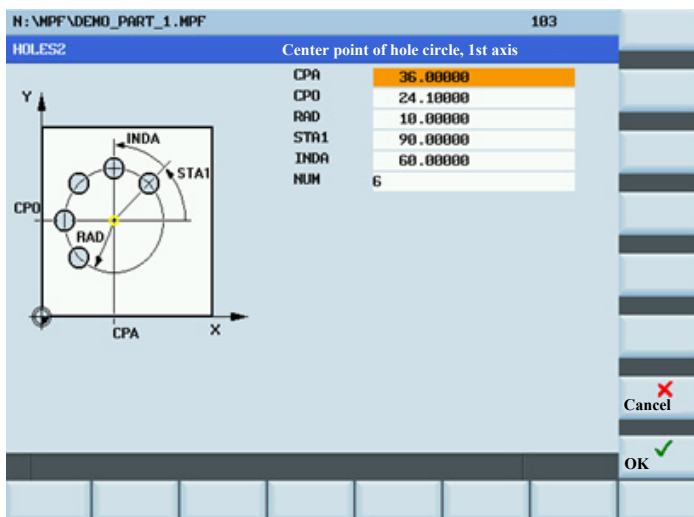
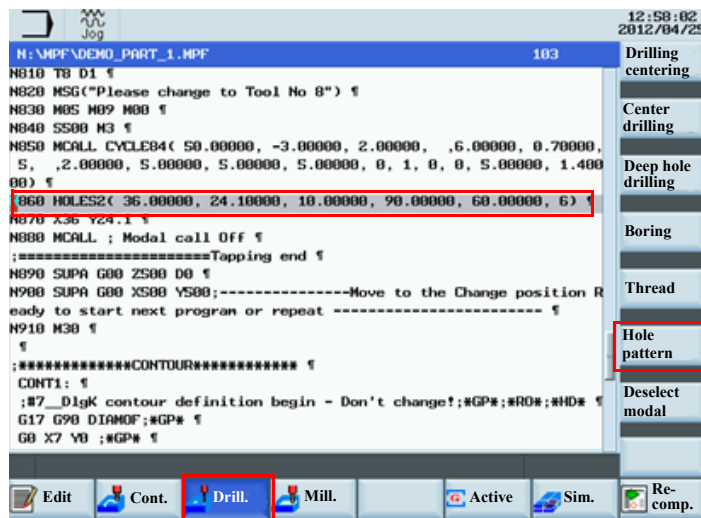
The cycles can be found and parameterized via the “Drill.” SK.



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

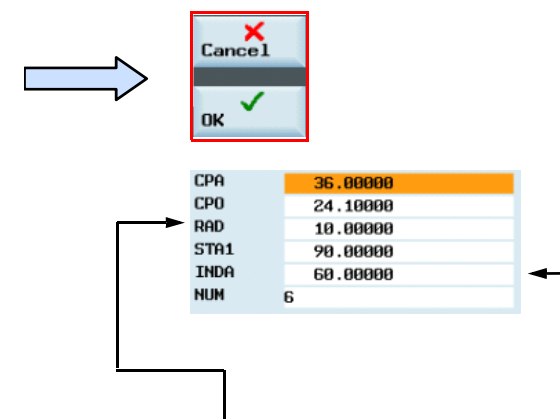


Select “Hole pattern” using the vertical SKs, and then select “Hole circle”, and parameterize the cycle according to requirement.



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

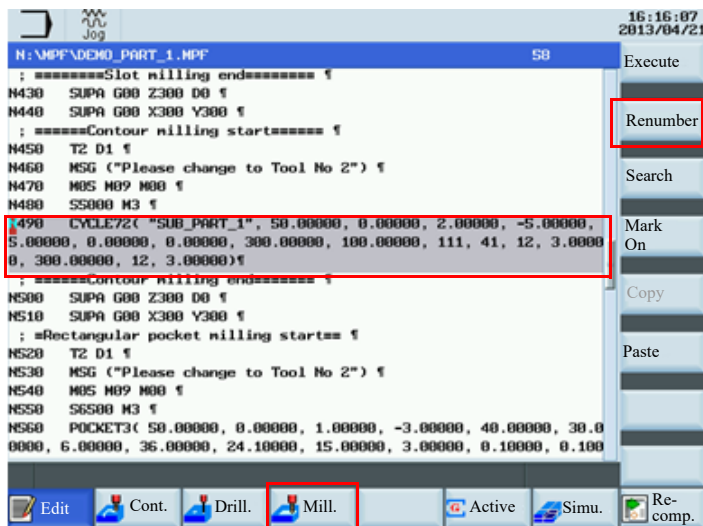
Parameters	Meanings
CPA=36	Center of hole circle horizontal coordinate is 36 (absolute value)
CPO=24.1	Center of hole circle horizontal coordinate is 24.1 (absolute value)
RAD=10	Circle radius is 10 mm
STA1=90	Angle between the circle and horizontal coordinate is 90°
INDA=60	Angle between the circles 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.

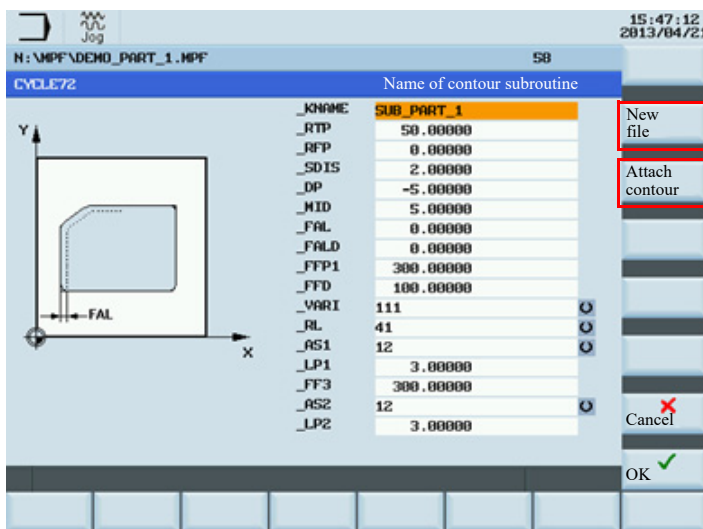
The cycle can be found and parameterized via the "Mill." SK.



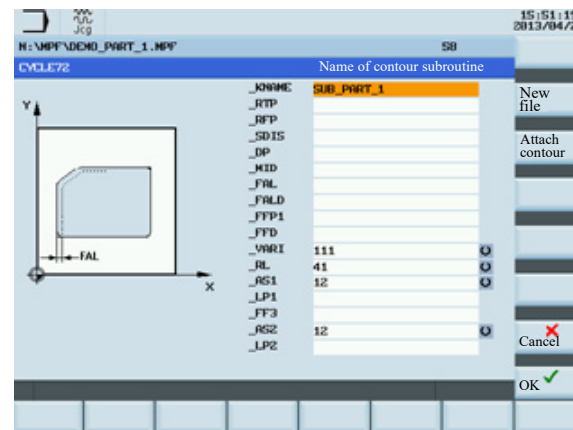
The "Contour milling" SK can be found in the vertical SKs on the right.

Contour
milling

The parameterization is performed as in this figure.

New
file

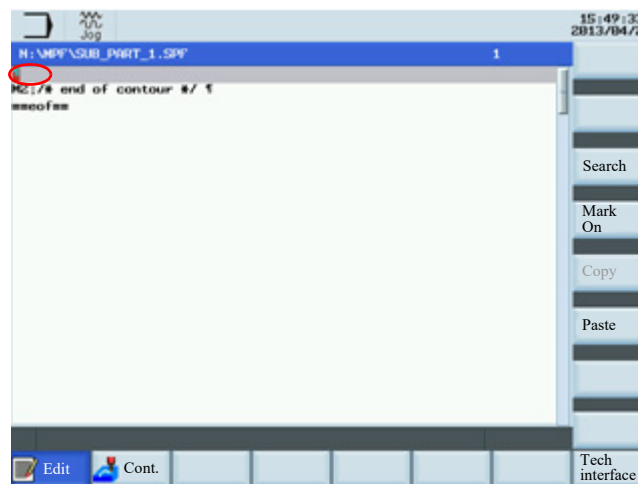
By selecting the "New file" SK, the contour turning data can be inserted into Sub Program File (.SPF). You can edit and change it when selected. The sequence is as follows:



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

Contour
milling

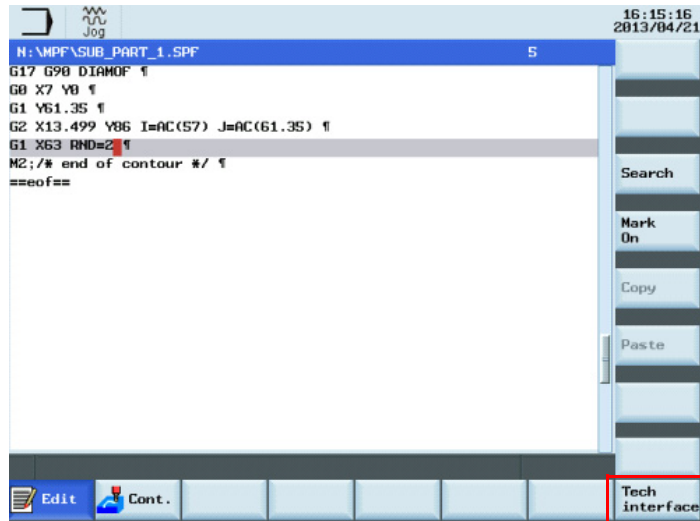
Press "New file" on the PPU to create contour information in the .SPF file. The cursor moves to the contour editing position automatically.

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:



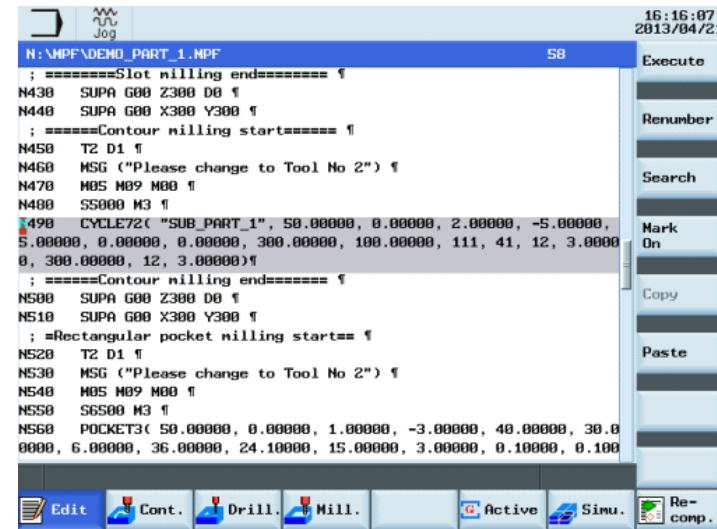
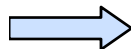
Enter appropriate coordinates based on the data from the technical drawing.

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.

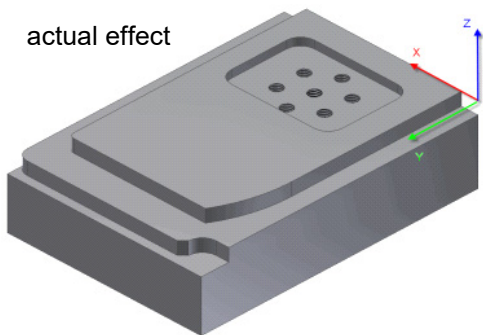


After all the settings take effect, the selected cycle and set data will be transferred to corresponding part program automatically (for further information, see next page).

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 144

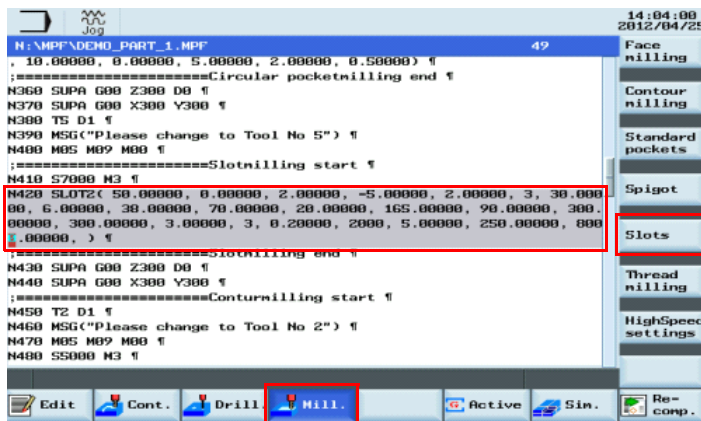
Parameters	Meanings	Remarks
KNAME= CONT1:CONT1_E	Set the name of the contour subprogram as "CONT1" ("CONT1_E" is automatically created)	The first two positions of the program name must be letters
MID=5	The maximal feed depth is 5 mm	
FAL=0	Finishing allowance at the contour side is 0 mm	
FALD=0	Finishing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

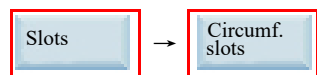
Milling slots and spigots

The easiest way to mill a slot is to use the SLOT2 cycle.

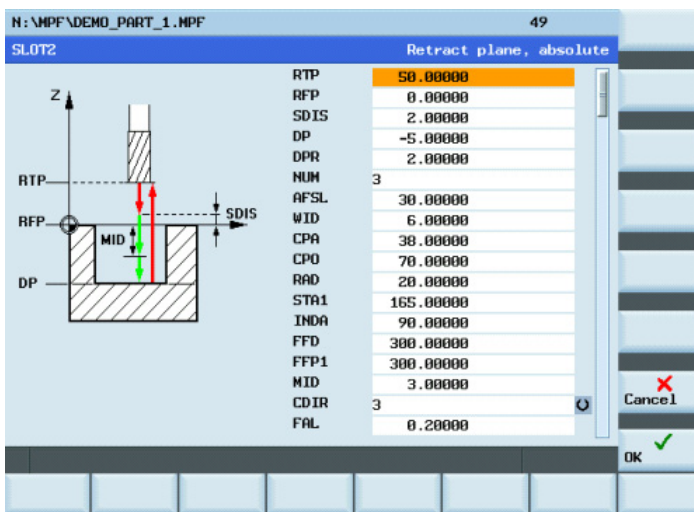
The cycle can be found and parameterized via the "Mill." SK.



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



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

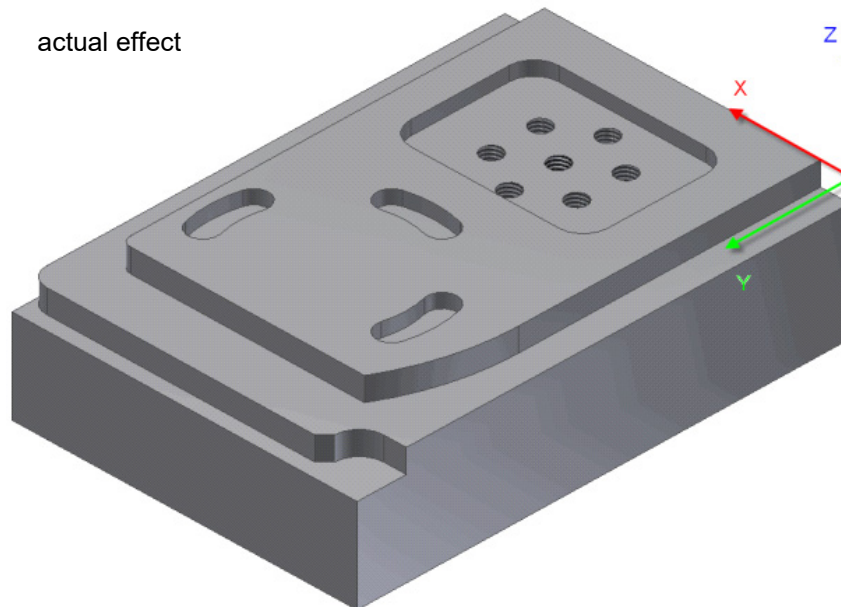


With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

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



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	

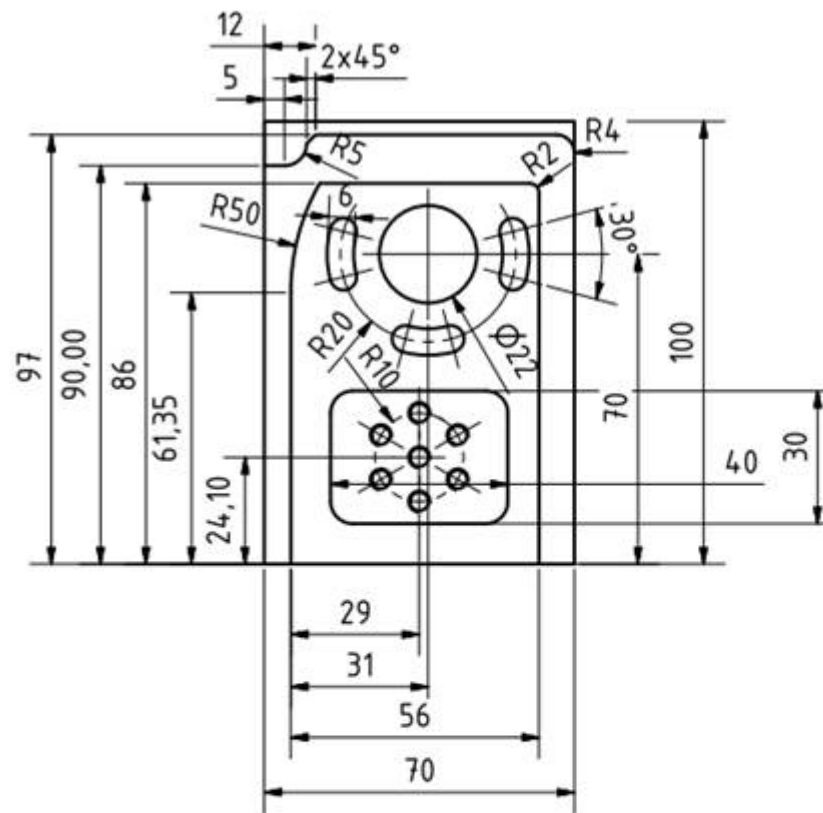
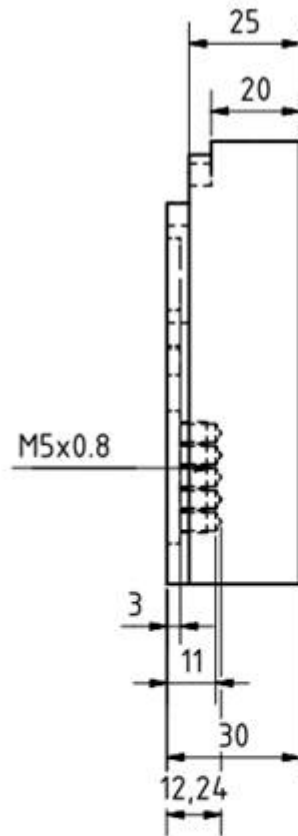
N210 SLOT2(
 50.00000, 0.00000, 2.00000, , 3.00000, 3, 30.00000, 6.00000,
 38.00000, 70.00000, 20.00000, 165.00000, 90.00000, 300.00000,
 300.00000, 3.00000, 3, 0.20000, 2000, 5.00000, 250.00000,
 3000.00000,)

For descriptions of RTP, RFP, SDIS, DP and DPR, please see Page 144

For descriptions of CPA, CPO and RAD, please see Page 149

For descriptions of FFD and FFP1, please see Page 152

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.	



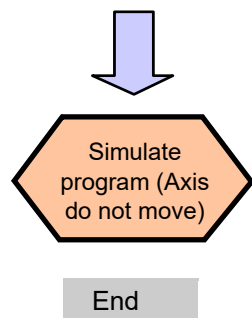
				Datum	Name		
				Geschieht			
				Kontrolliert			
				Norm			
						DEMO PART MILLING 1 A4	
Status	Änderungen	Datum	Name				

Simulate Program

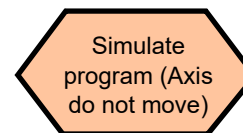
Description

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

Content



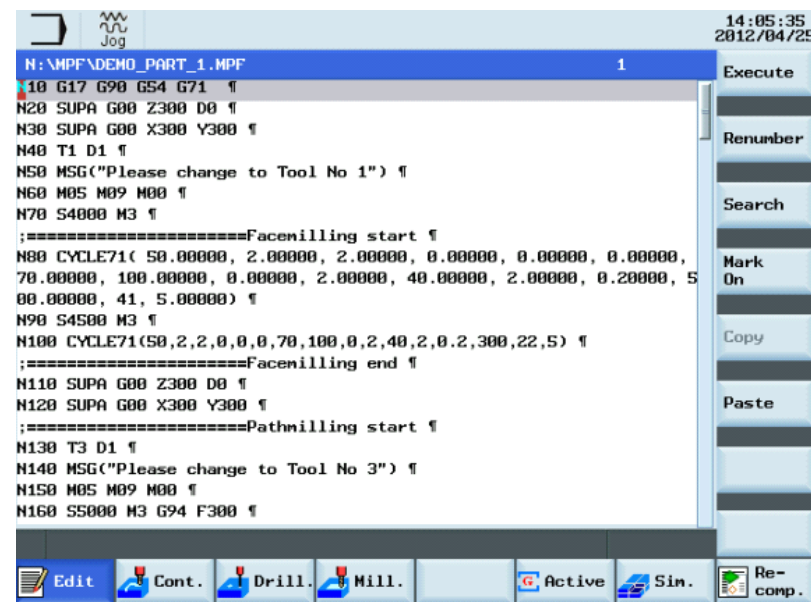
Basic Theory



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

Step 1

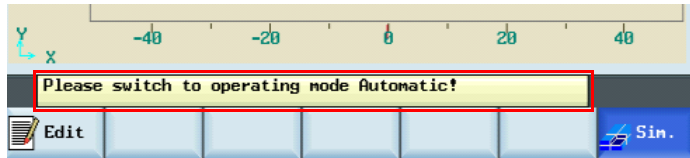
The part program must be opened using the "Program Manager" on PPU.



SEQUENCE

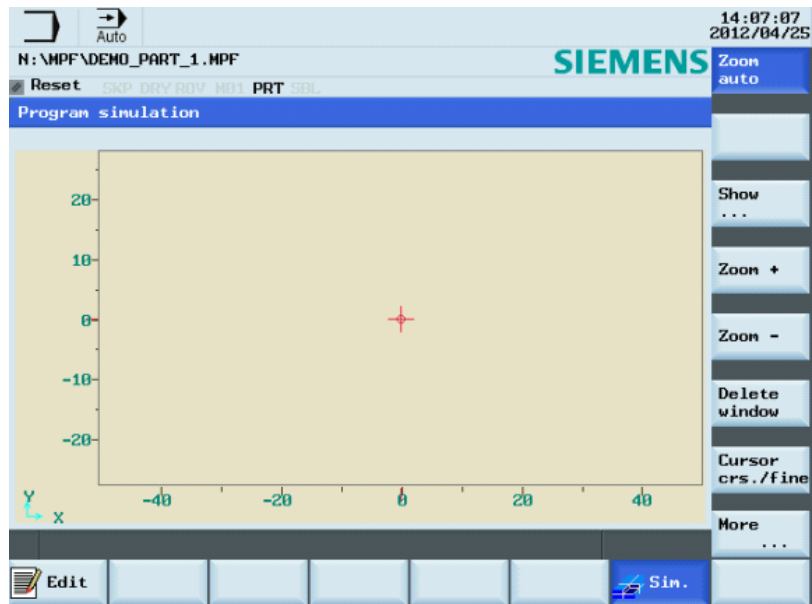
Step 2

Press the "Simu." SK on the PPU.



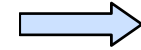
If the control is not in the correct mode, a message will be displayed at the bottom of the screen.

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

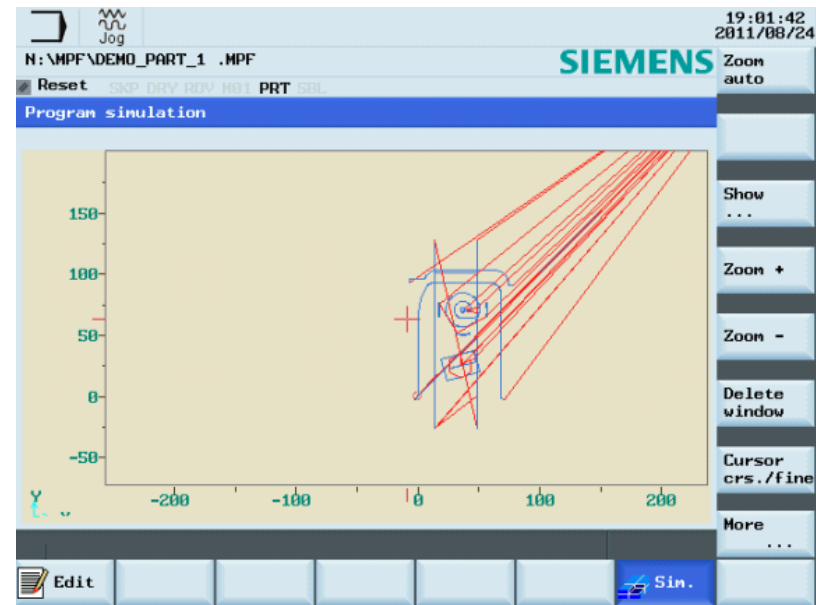


Step 3

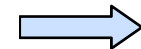
Press the "CYCLE START" key on the MCP.



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.



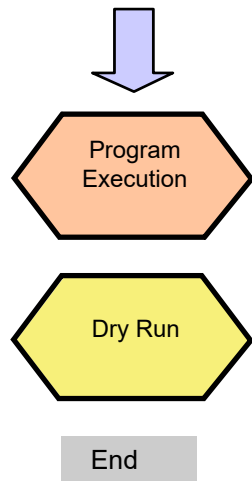
End

Test Program

Description

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

Content



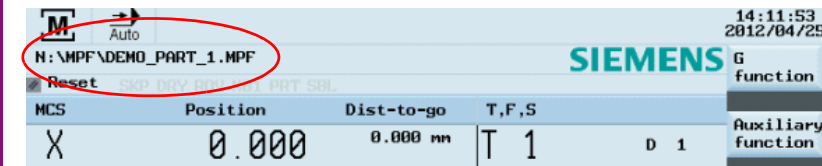
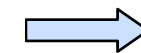
Basic Theory



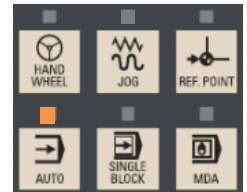
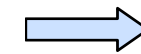
Before the part program can be loaded and executed in AUTO mode, it must be tested using the simulation function mentioned previously!



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!

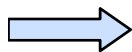
Note: The following operation is based on the finished “program execution” screen

Step 1

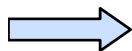


The data in the “Dry run feedrate” must first be set and checked!

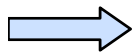
Press the “Offset” key on the PPU.



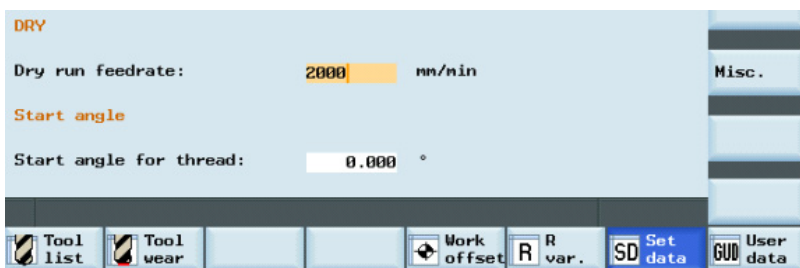
Press the “Sett. data” SK on the PPU.



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



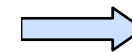
Enter the required feedrate in mm/min, enter “2000 ” in the example.



Press the “Input” key of the PPU.



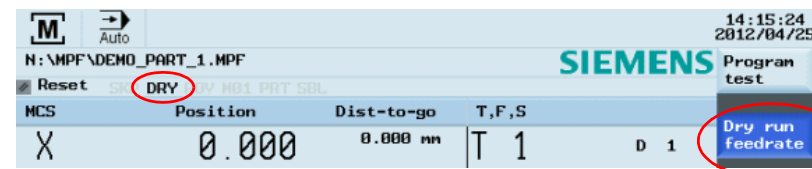
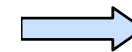
Press the “Machine” key on the PPU.



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

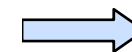


Make sure the feedrate override on the MCP is 0%.

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.



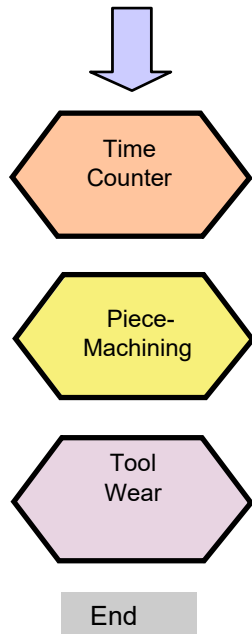
After finishing the dry run, please turn the changed offset back to the original value in order to avoid affecting the actual machining!

Machine Pieces

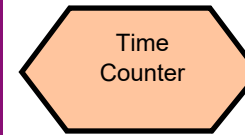
Description

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.



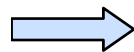
Time left 0000:00:00h

Step 2

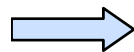


The "Time left" can only be counted after a successful cycle run of a part program!

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

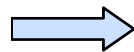


Enter the number of workpieces you require to be machined in "Required".



Required 45

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

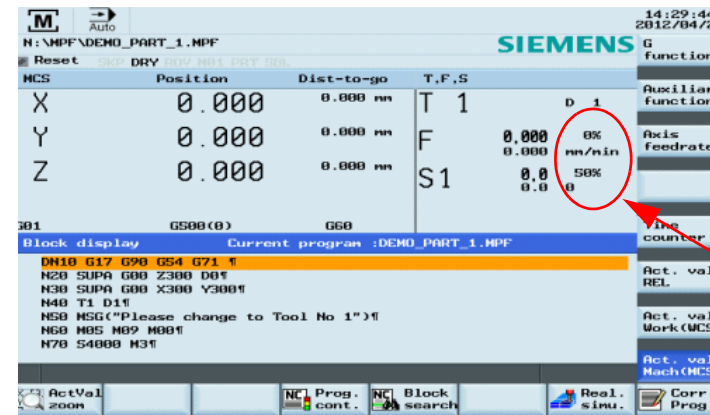


Actual 8

Block display	DEMO_PART_1.MPF	Part counter	Part timer
DN10 G17 G90 G54 G71		Cycle time	0000:00:06h
N20 SUPA G00 Z300 D01		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			



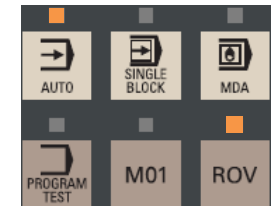
Make sure the program is correct before machining pieces!



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).

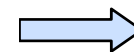


Notes: M01 function → program will stop at the position where there is M01 code.



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

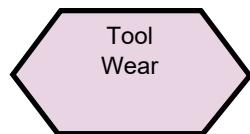
Press "Door" on the MCP to close the door of the machine. (If you don't use this function, just close the door on the machine manually.)



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



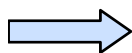
SEQUENCE



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

Step 1

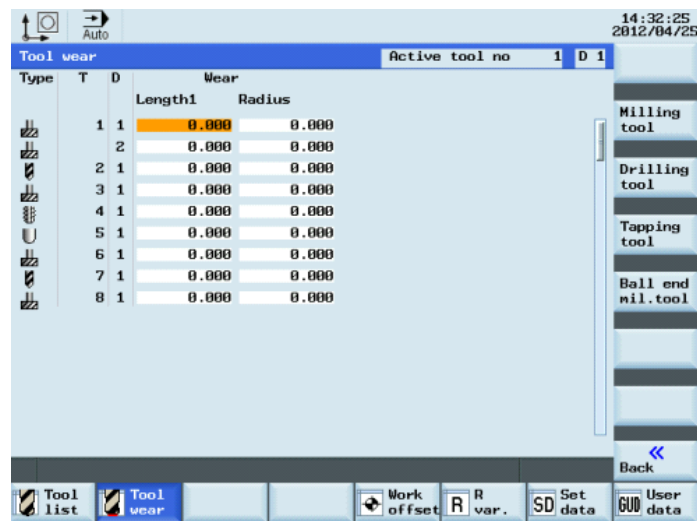
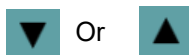
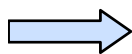
Press the "Machine" key on the PPU.



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.

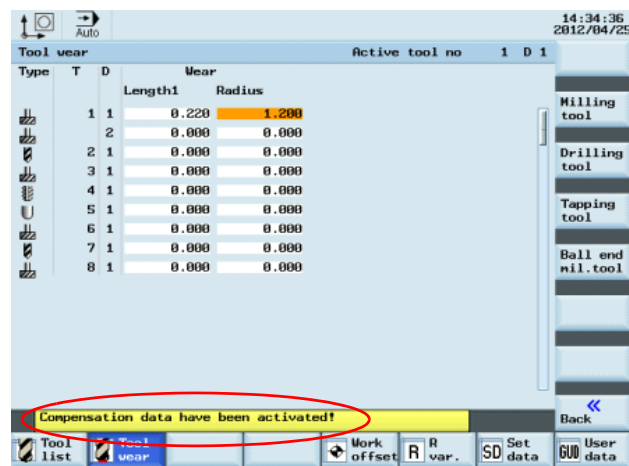
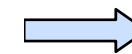


Set the tool radius wear parameter in "Radius", the sign determines the direction of wear compensation.

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.

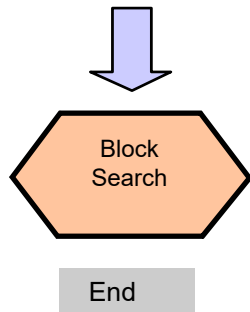


Program Restart

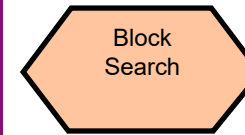
Description

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.



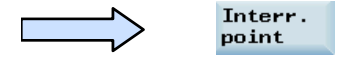
Press the "Auto" key on the MCP.



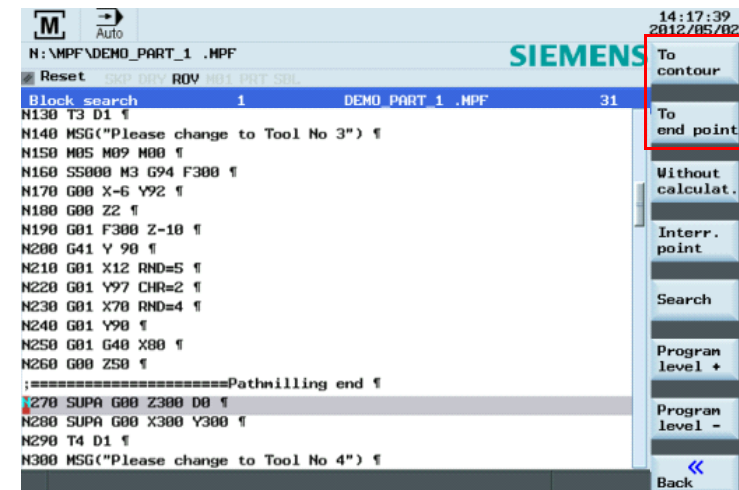
Press the "Block search" SK on the PPU.



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.

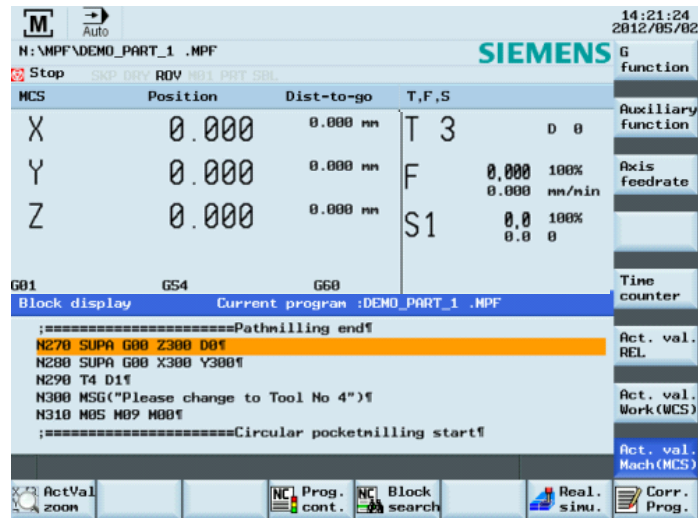
"To contour": The program will continue from the line before the breakpoint.

"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)

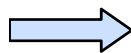


SEQUENCE



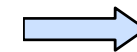
The feedrate override must always be set to 0%!
 Make sure the correct tool is selected before continuing!

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

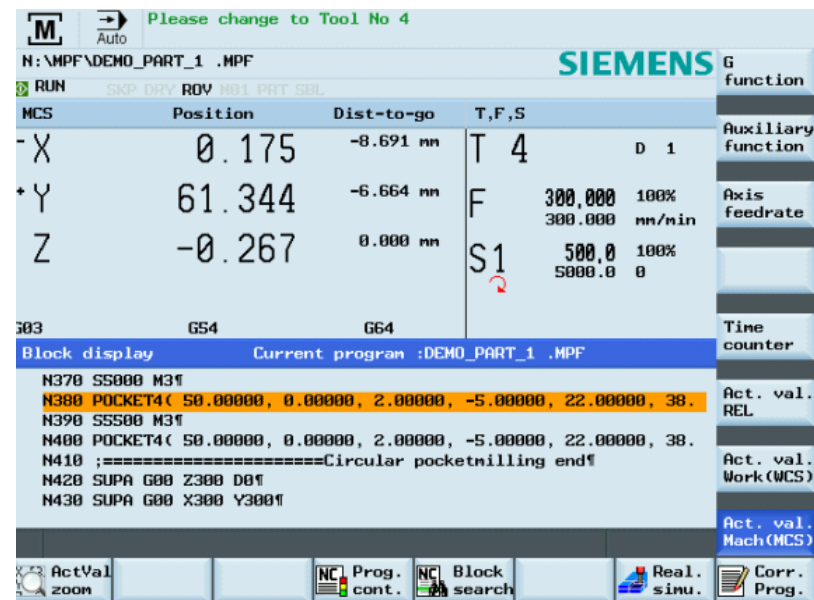


Alarm 010208 is shown at the top prompting to press the “CYCLE START” key to continue the program.

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



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

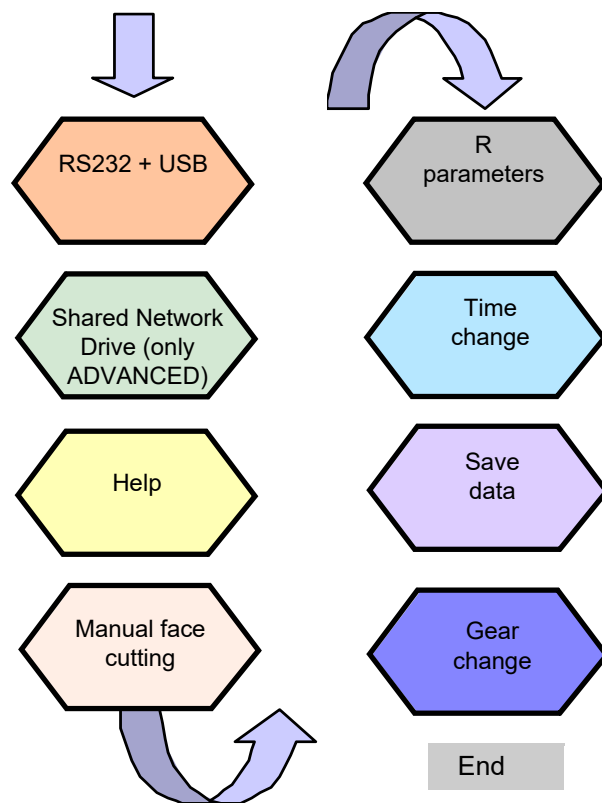


Additional Information Part 1

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.

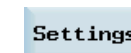
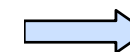
Press “Program Manager” on the PPU.



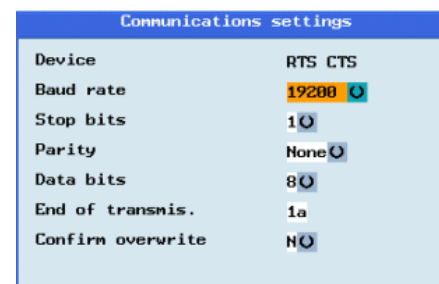
Press the “RS232” SK on the PPU.



Press the “Settings” SK on the PPU.



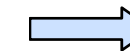
Adjust the parameters in “Communication settings” to match the settings of communication SW on PC.



Press the “Save” SK on the PPU.



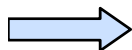
Press the “Back” SK on the PPU.



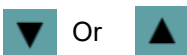
SEQUENCE

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

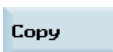
Press the "NC" SK on the PPU.



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.



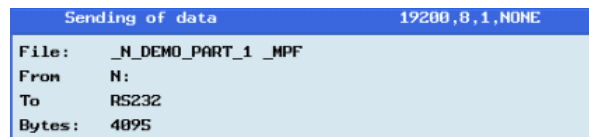
Check the interface setting and start the communication software to receive the program on PC.

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

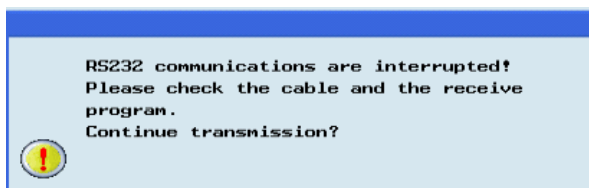
Press the "Send" SK on the PPU.



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.

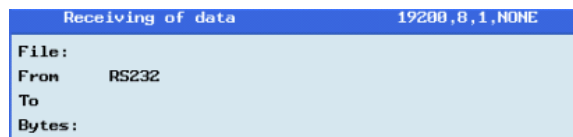
Press "Program Manager" on the PPU.



Press the "RS232" SK on the PPU.



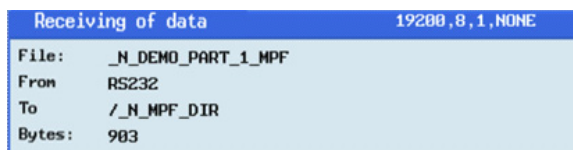
Press the "Accept" SK on the PPU.



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.



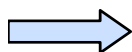
Press the "Copy" SK on the PPU.



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.



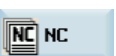
Use "Cursor + Select" to select the required part
program.
The selected program will be highlighted.



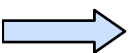
Press the "Copy" SK on the PPU.



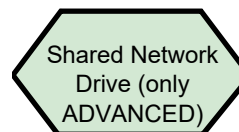
Press the "NC" SK on the PPU.



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:



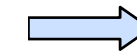
Press key:



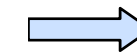
Press "Serv. Displ." SK



Press "Serv. Displ." SK

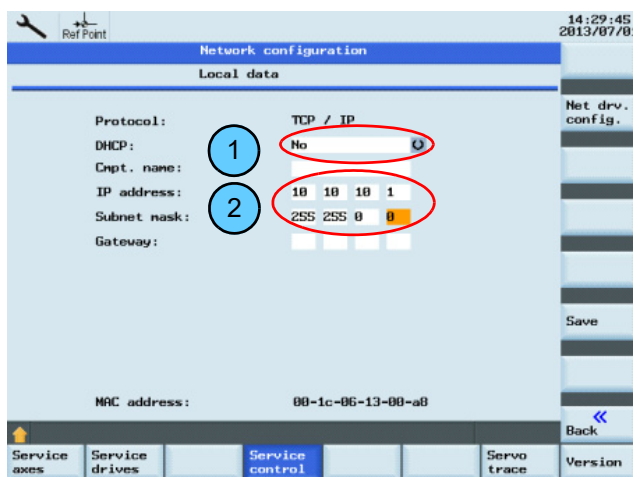


Press "Network Info" button to enter the "Local Configuration Data"



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

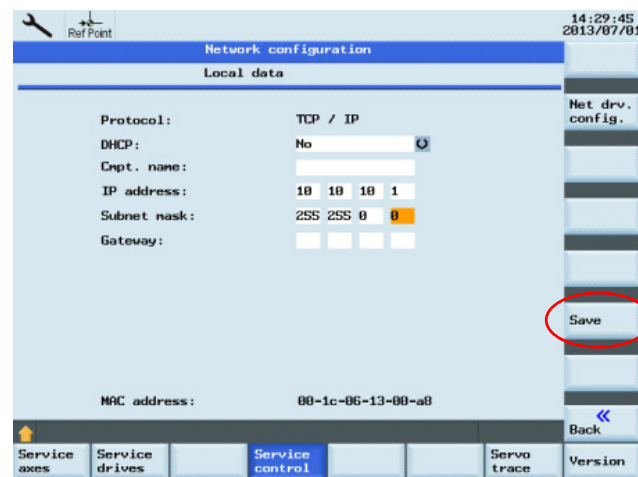
- ① DHCP is set to "No"
- ② IP address and subnet mask can be arbitrarily set according to need. (Right given only as an example)



"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.



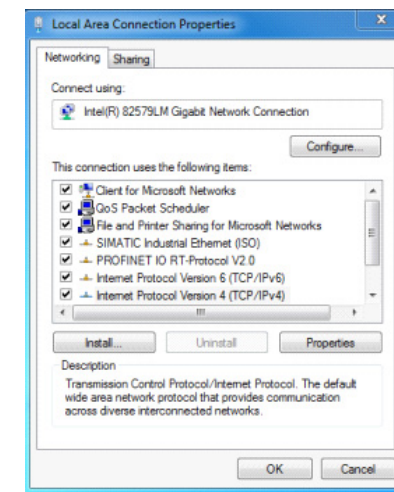
SEQUENCE

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".



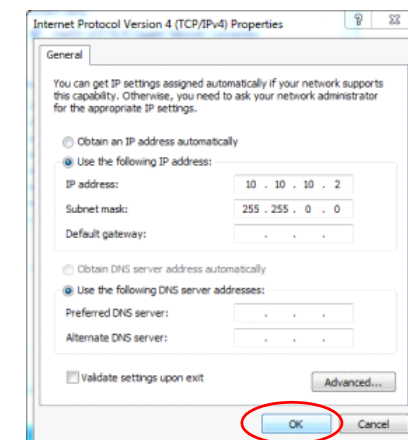
In the dialog box, select "Use the following IP address" and fill in the required IP address.

(Shown right only given as an example)

Select "OK" to complete the setup.

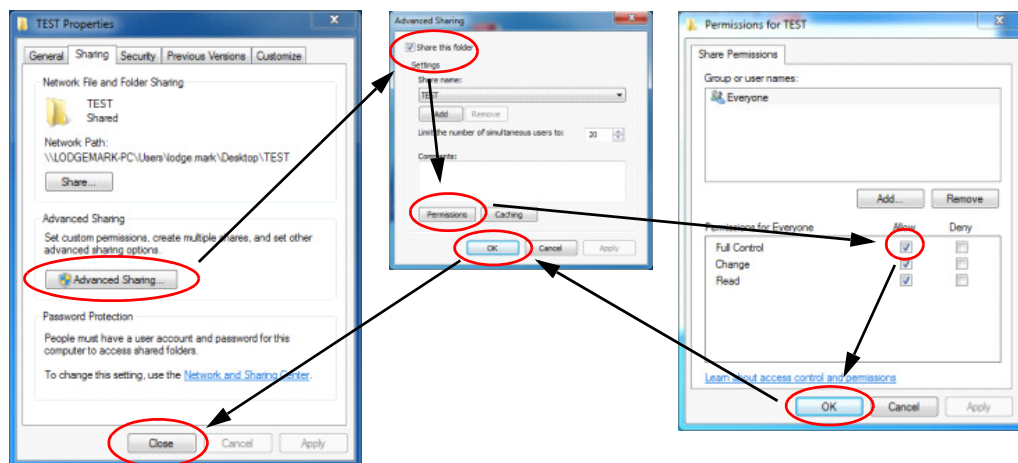
Note: The address "10.10.10.2" setting is based on the first step in the IP address of the PPU.

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."

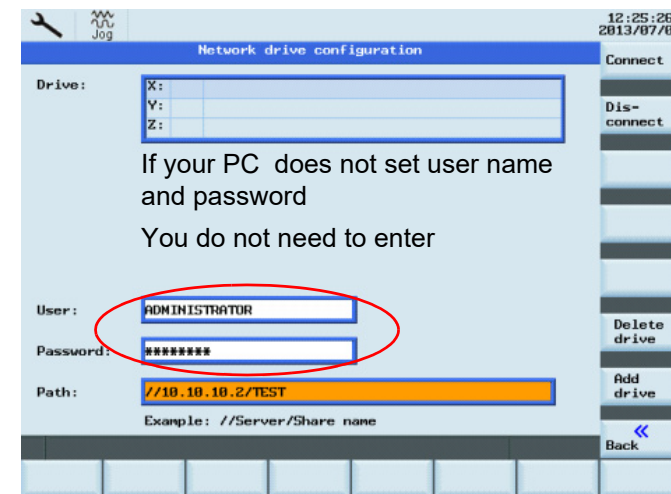
SEQUENCE

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

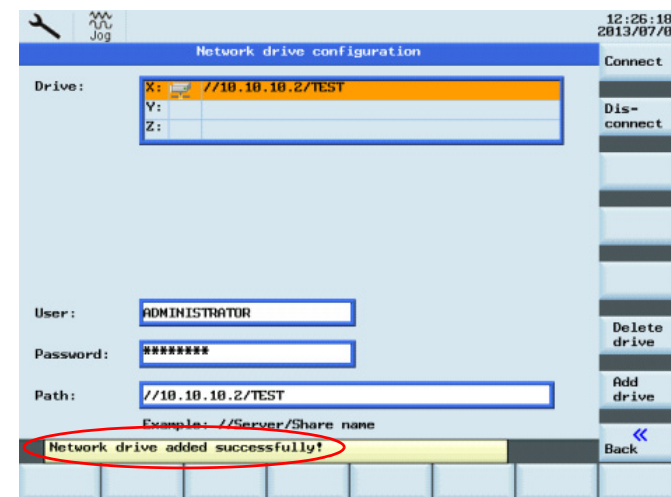
Note: Use "TAB" key to switching between different tasks boxes.



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.

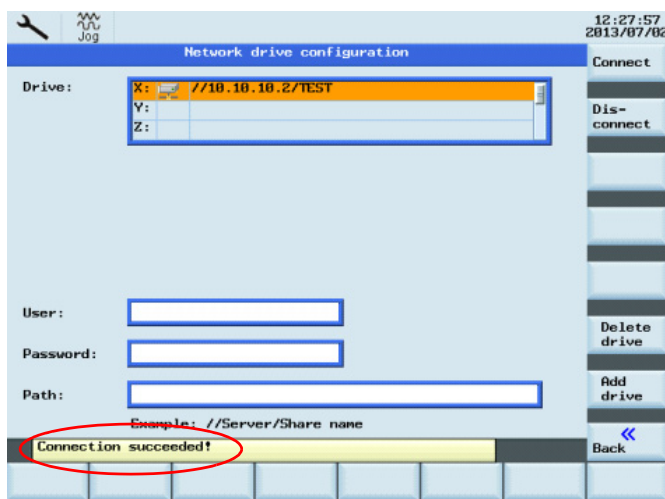


If the connection is lost select the drive path and press "Connect." SK

Connect

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

This will be shown with the text "Connection succeeded"

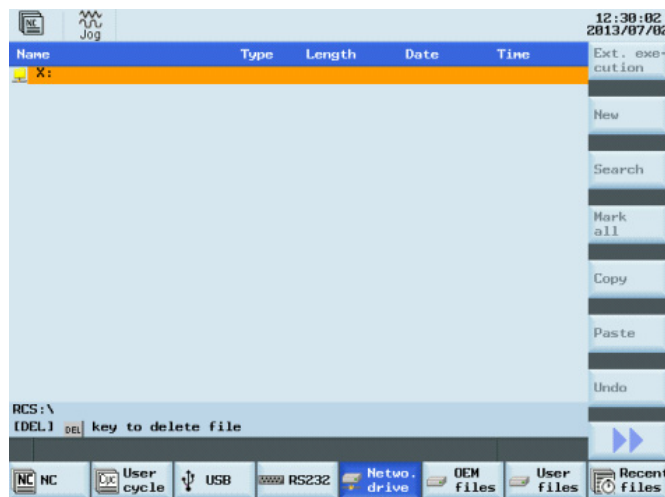


Press "Program Manager" Button

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



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

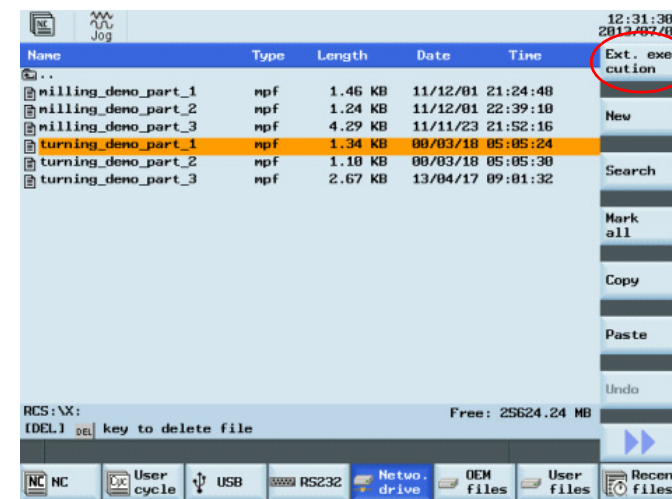


SEQUENCE

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

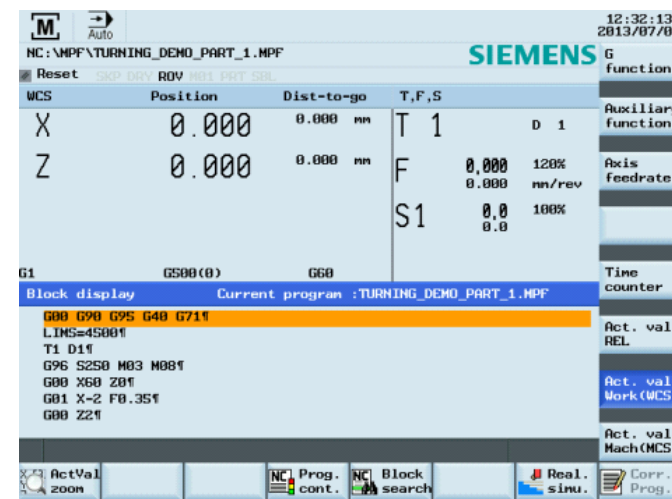
You can select the file you require to execute in AUTO mode, click "Exe. Execution".

Ext. exe-cution



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

Press the "Cycle Start" button for machining operation.



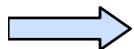
Note: You can also use the "Copy", "Paste" key to achieve "NC", "USB" and "Network Drive" moving files.

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.

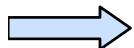


Press the "Help" key on the PPU.



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.

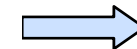


Press the "Sett." SK on the PPU.



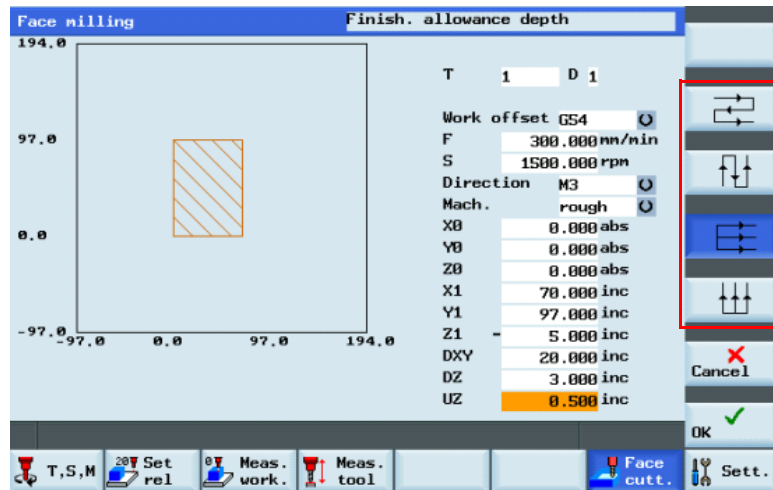
Enter appropriate values in "Retraction plane" and "Safety distance".

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



Step 2

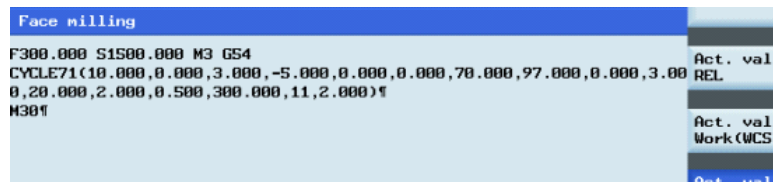
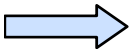
Press "Face cutt." SK on PPU.



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.



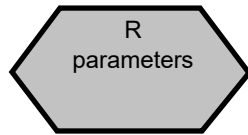
Make sure that the override value on the MCP is 0%!

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

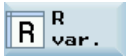
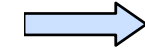
Programming the STOPRE command in a block will stop block preprocessing and buffering. The following block is not executed until all preprocessed and saved blocks have been executed in full. The preceding block is stopped in exact stop (as with G9).

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

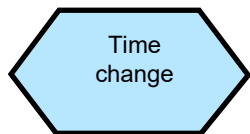
R variables	
R0	0.000000
R1	1.000000
R2	0.000000
R3	0.000000
R4	0.000000
R5	0.000000

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

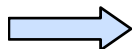
WCS	Position	Repos offset
X	3.000	0.000 mm
Y	0.000	0.000 mm
Z	10.000	0.000 mm

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.

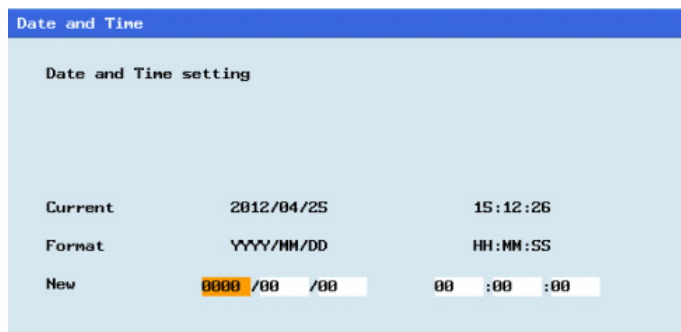


Make sure the password is set to the "CUSTOMER" access level.

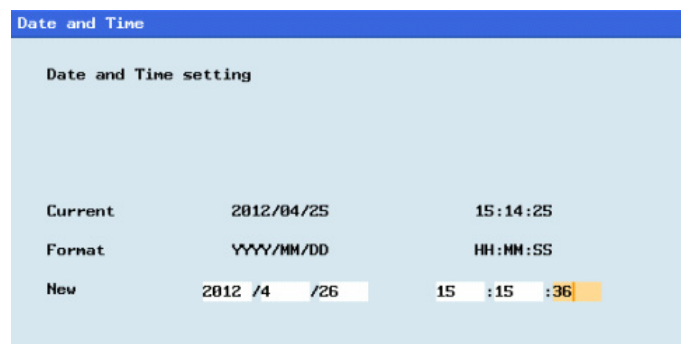
Press the "HMI" SK on the PPU.



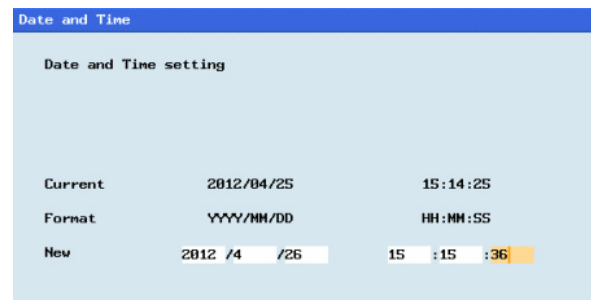
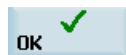
Press the "Date time" SK on the PPU.



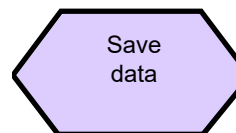
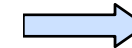
Enter a new "Date" and "Time".



Press the "OK" SK on the PPU.



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



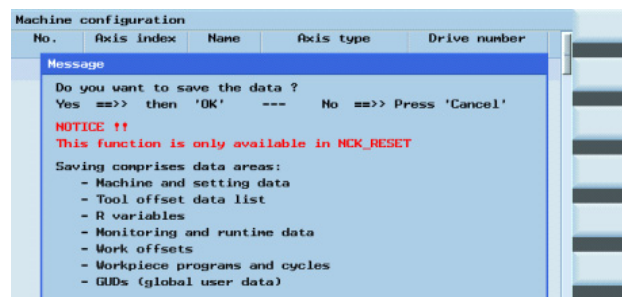
"Save data" enables the complete system to be backed up on the system CF card so that there is a system backup available to the operator.

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

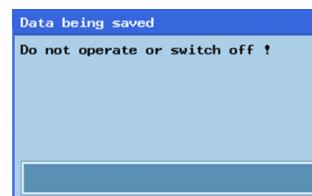
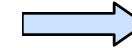


Make sure the password is set to the "CUSTOMER" access level.

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

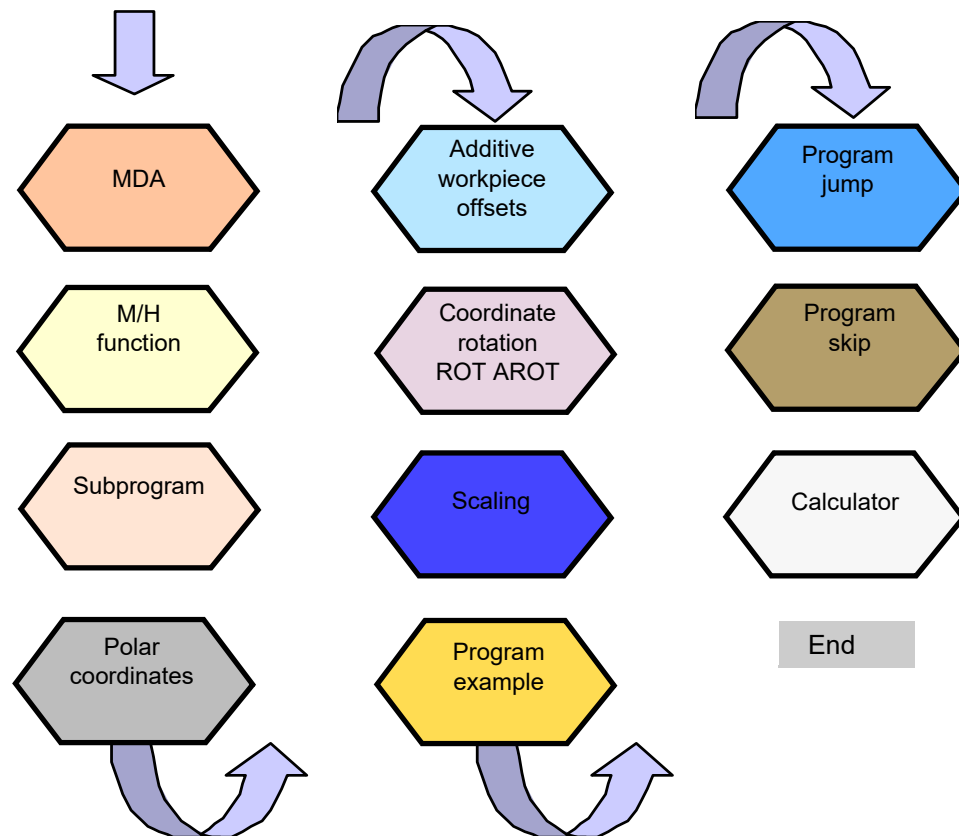
If the operator is manually selecting the gear stage in the part program, it is the operator's responsibility to select the correct gear stage according to the required speed.

Additional Information Part 2

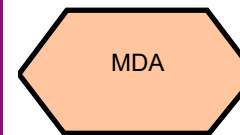
Description

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.

MDA		10:44:36 2012/04/28	
		SIEMENS	
Stop	MDI	MDI	G function
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
G00	G54	G60	
MDI - Block			
G0 G54 X10 Y10 Z50			
==eof==			
		Auxiliary function	
		Axis feedrate	
		Save file	
		Delete file	
		Act. val. REL	

SEQUENCE

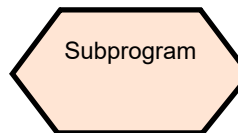


The M function initiates switching operations, such as "Coolant ON/OFF". Various M functions have already been assigned a fixed functionality by the CNC manufacturer. The M functions not yet assigned are reserved for free use of the machine tool manufacturer.

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

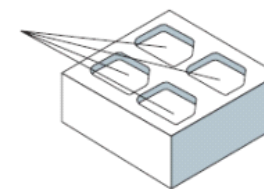
M codes and H functions created by the OEM should be backed up by the machine tool manufacturer.

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.

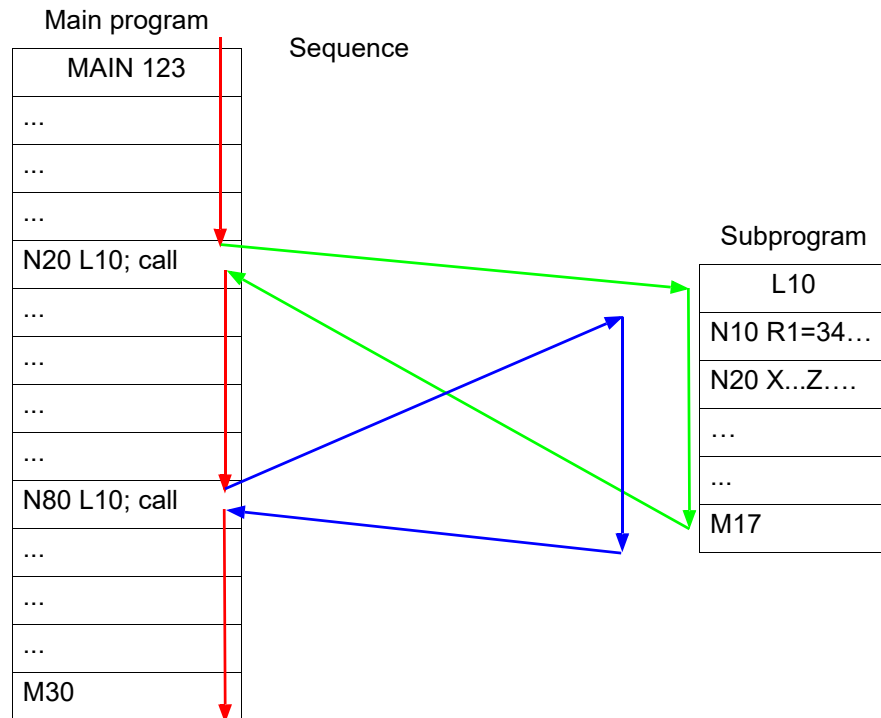
The subprogram should be given a unique name enabling it to be selected from several subprograms. When you create the program, the program name may be freely selected.

However, the following rule should be observed:

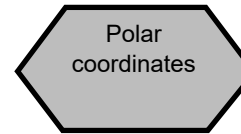
The name can contain letters, numbers and underscores and should be between 2 and 8 characters long.

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.

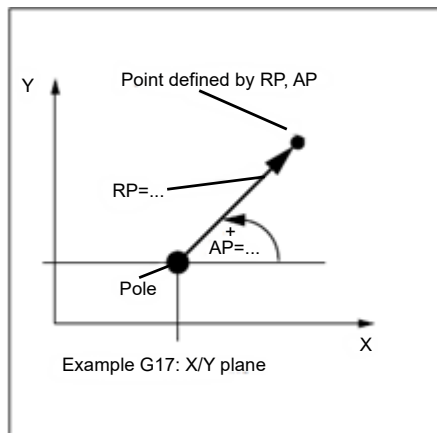
Basic Theory

- 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.

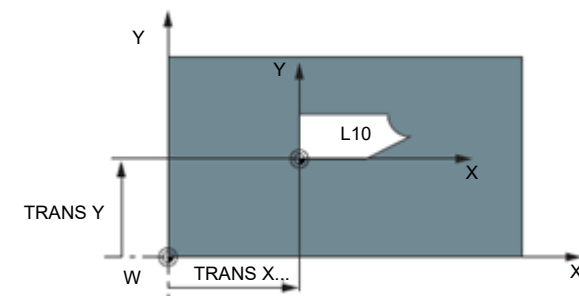
```

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
  
```

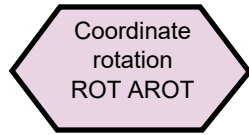
Programming example

```

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



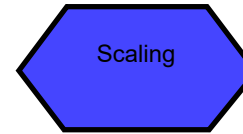
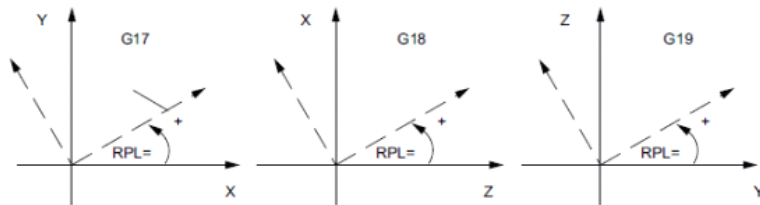
SEQUENCE



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



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)

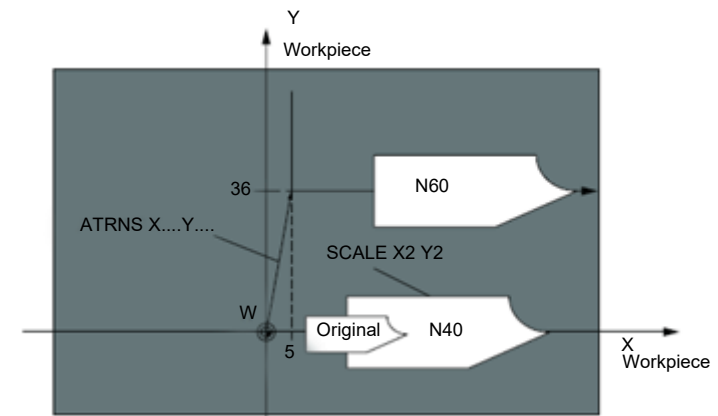
If a program contains SCALE or ASCALE, this must be programmed in a separate block.

Programming example

N10 G17

N20 SCALE X2.0 Y2.0 ; contour is enlarged two times in X and Y

L10 subprogram call

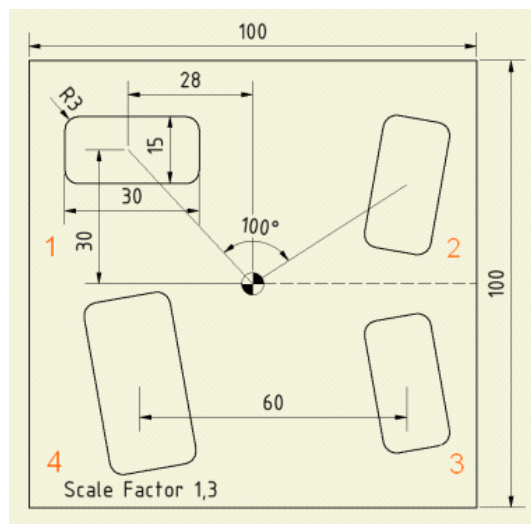


SEQUENCE

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

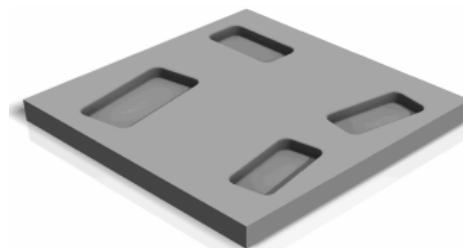
② Coordinate rotates 20°

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!



N10	SUPA G00 Z300 D0	N10	SUPA→cancel all settable offsets
N15	SUPA G00 X0 Y0	N15	
N20	G17 T1 D1	N20	coordinate plane G17, use tool 1
N25	MSG ("change to 1 tool")	N25	
N30	M5 M9 M00	N30	
N35	S5000 M3 G94 F300	N35	
N40	G00 X-28 Y 30	N40	
N45	G00 Z2	N45	
N50	LAB1:	N50	LAB1:milling start sign
N65	POCKET3(50, 0, 2, -5, 30, 15, 3, -28, 30, 0, 5, 0, 0, 300, 100, 0, 11, 5, , , 5, 3,)	N65	milling rectangular groove(depth 5 mm, length 30 mm, width 15 mm, corner radius 3 mm, groove datum coordinate (X-28,Y30), groove longitudinal axis and plane X axis clamping angle 0°)
N70	LAB2:	N70	LAB2:milling groove end sign
N75	M01	N75	
N80	ROT RPL=-100	N80	coordinate axis rotates 100° in positive direction
N85	REPEAT LAB1 LAB2 P1	N85	machining the same groove at the new position
N90	M01	N90	
N95	AMIRROR X=1	N95	along the new X axis to change the mirror image
N100	AROT RPL=-20	N100	coordinate axis rotates -20° in positive direction
N105	M01	N105	
N110	REPEAT LAB1 LAB2 P1	N110	machining the same groove at the new position
N115	AROT RPL=10	N115	coordinate axis rotates -10° in negative direction
N120	ATrans Y-60	N120	Y axis coordinate moves 60 in negative direction
N125	AROT RPL=-10	N125	
N130	ASCALE X1.3 Y1.3	N130	groove enlarged 1.3 times in the X,Y direction.
N135	REPEAT LAB1 LAB2 P1	N135	machining the same groove at the new position
N140	M30	N140	end

SEQUENCE



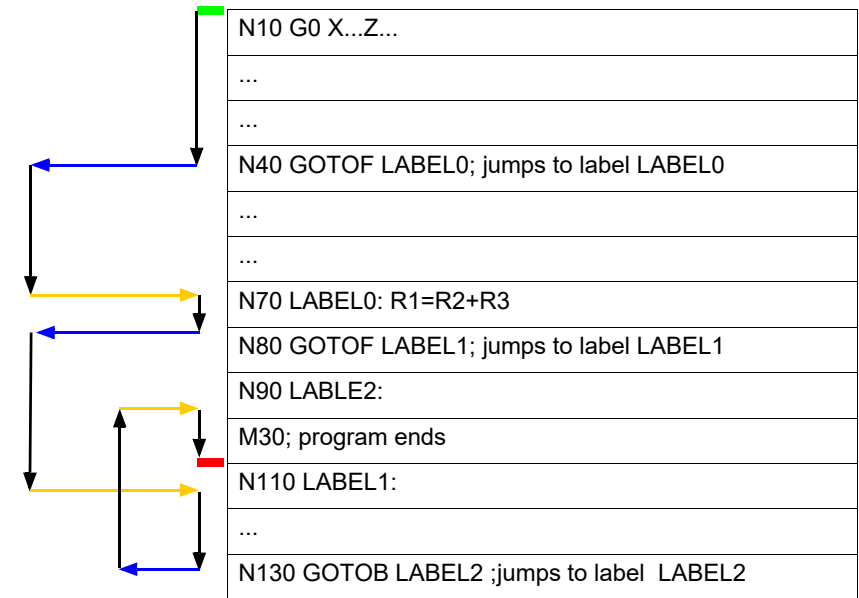
NC programs process their blocks in the sequence in which they were arranged when they were written. The processing sequence can be changed by introducing program jumps. The jump destination can be a block with a label or with a block number. This block must be located within the program. The unconditional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

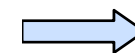
Using “;” code to add a remark to the N85 function, without any influence on the execution.

Method 2

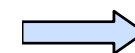
Press the “Machine” key on the PPU.



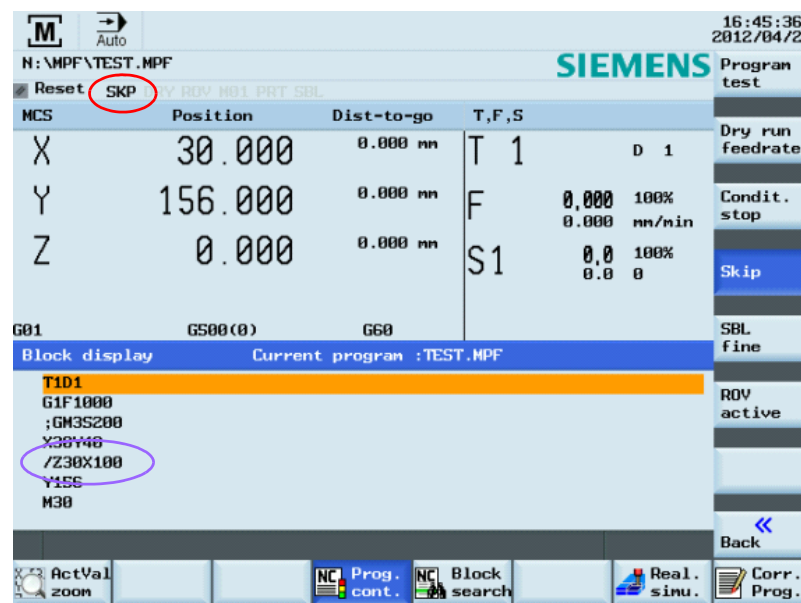
Press the “Auto” key on the MCP.



Press the “Prog cont.” SK on the PPU.



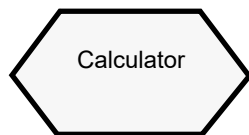
Press the “Prog cont.” SK on the PPU.



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

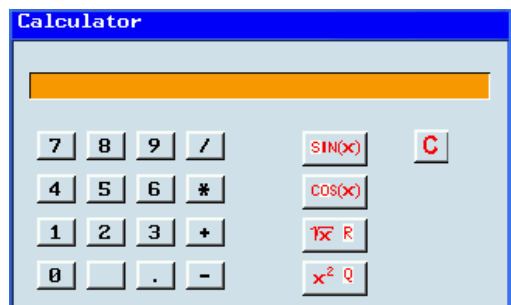
After activating “SKP”, using “/” at the beginning of the program string (shown in purple circle), the string will be skipped without influencing the execution.

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.

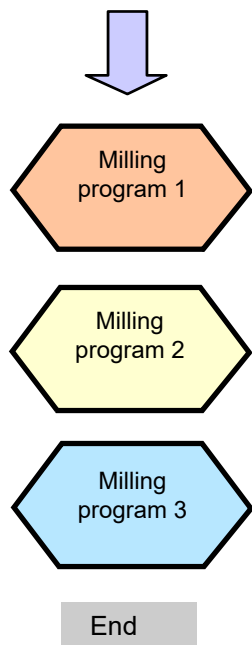
Use the “Accept” SK to enter the result in the input field at the current cursor position of the part program editor. The calculator will then close automatically.

Sample Program

Description

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

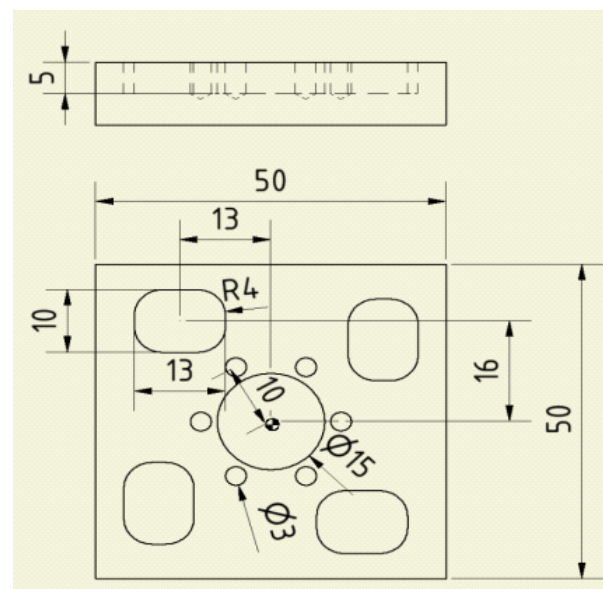
Content



Drawing



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

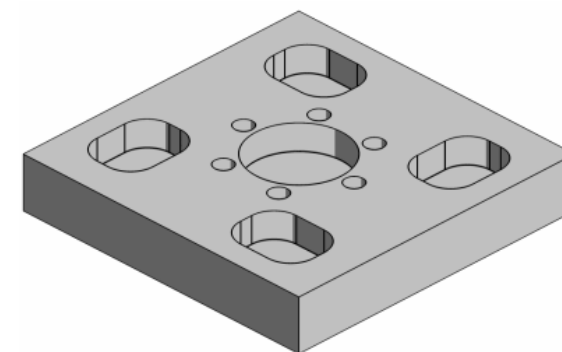


The zero point of the workpiece is located at the center point of the workpiece

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

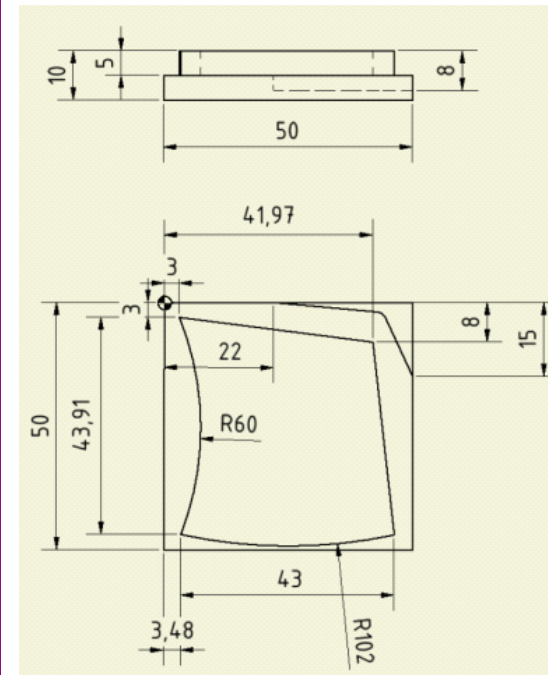
N390 circular line hole forms cycle
 command(circular center point coordinate(X0,Y0),
 radius 10 mm, angle between the line with first hole
 and circular center point and the X axis in positive
 direction is 45°, angle between the holes is 60°,
 circular hole number 6 →)

N400 cancel mode use
N410 M30

Drawing



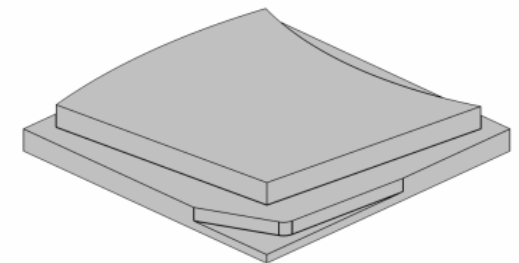
Make sure all the preparations and safety measures
have been performed before machining!



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 */

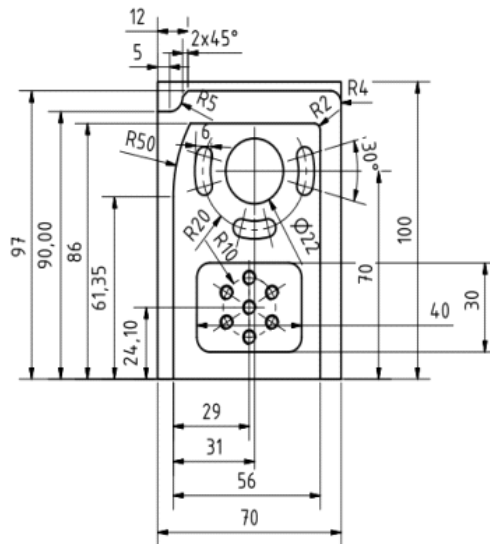
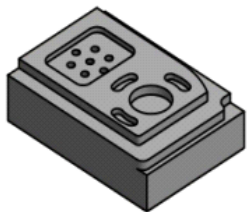
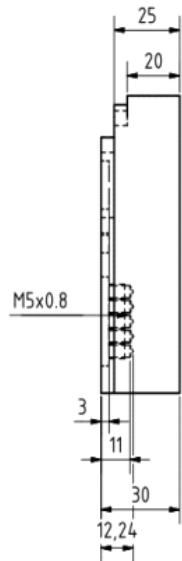
```

Drawing

Milling program 3



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



Tool information

T1 Milling tool D50

T2 Milling tool D12

T3 Milling tool D10

T4 Milling tool D16

T5 Milling tool D5

T6 Drilling tool D10

T7 Drilling tool D5

T8 Tap D6

Machining Process

```

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

```

```

N10
N20
N30
N40
N50 hint:change to tool 1
N60
N70
; =====Face milling start=====
N80 start point (X0,Y0), machining length: X → 70
mm, Y→ 100 mm, angle between vertical axis and X
axis is 0°, finishing allowance 0.2 mm, feedrate 500
mm/min, along the alternate direction parallel to the
Y axis to perform the finishing
N90
N100 repeat N80 contour process, the difference
in the feedrate is 300 mm/min along the single
direction parallel to the Y axis to perform the
finishing
; =====Face milling end=====
N110
N120
; =====Path milling start=====
N130
N140 hint:change to tool 3
N150
N160 feedrate 300 mm/min
N170
N180
N190
N200 left side radius compensation
N210 circle, milling radius is 5 mm
N220 incline, milling side length is 2 mm
N230
N240
N250 cancel tool radius compensation
N260
; =====Path milling end=====

```

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, 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

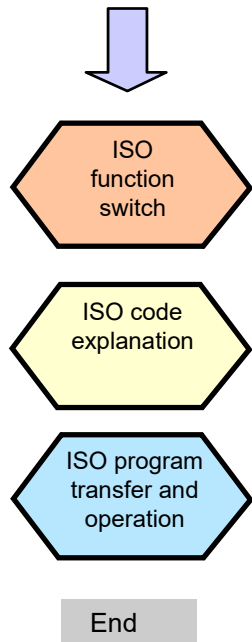
M2;/* end of contour */

ISO Mode

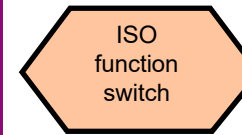
Description

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

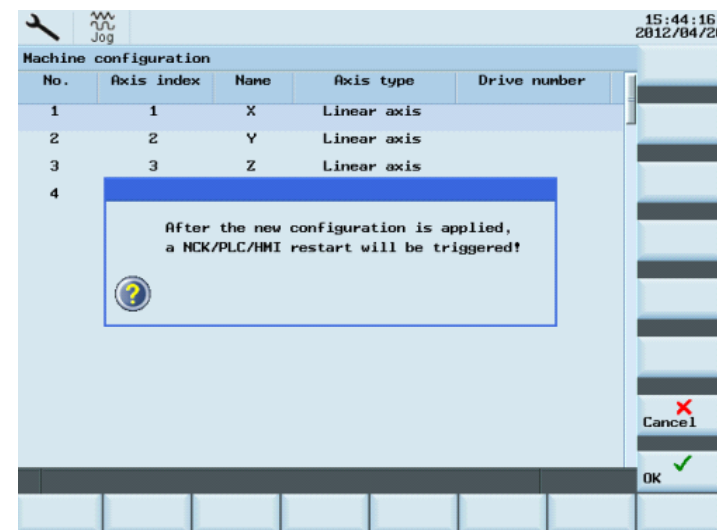
Press the “Shift” + “System - Alarm” keys on the PPU.
Input the manufacturer’s password (“SUNRISE”)



Press the “ISO mode” SK on the right.



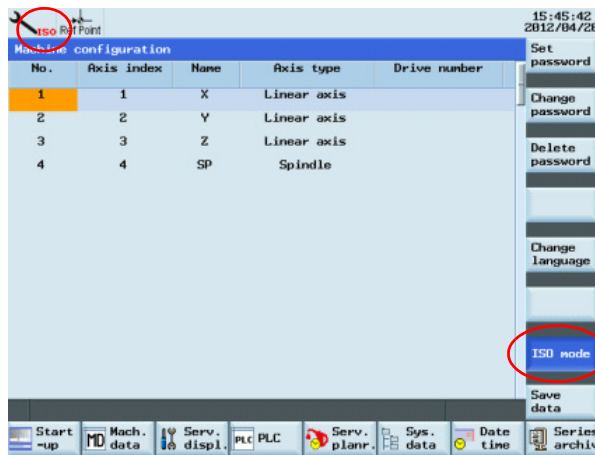
A dialog box appears prompting whether to activate the new setting. Select the “OK” SK to activate it.



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 H11
N25 MSG("Tool No 1 in use")F1
N35 S4000 M31
N40 CYCLE71( 50.00000, 2.00000, 2.00000, 0.00000, 0.00000, 0.0000
N45 S4500 M31
  
```



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

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

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

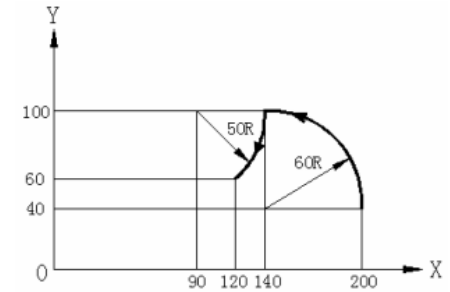
N5 G90 T1 M06
N10 M3 S2000; spindle rotation
N20 **G99** G81 X300 Y-250 Z-150
R-10 F120; **after orientation drilling, back to R point**
N30 X1000. ; **after orientation drilling, back to R point**
N40 **G04** X2.0 ; **delay 2 s**
N50 **G98** Y-550 ; **after orientation drilling, back to start point**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

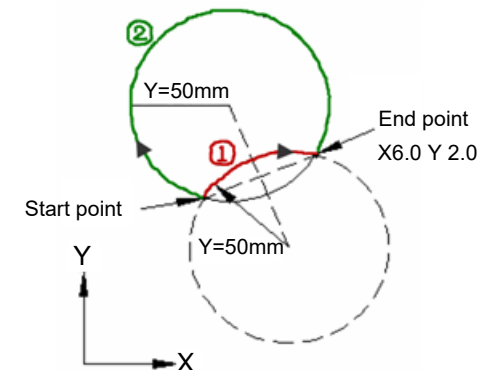
When specifying circle radii with parameter R

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

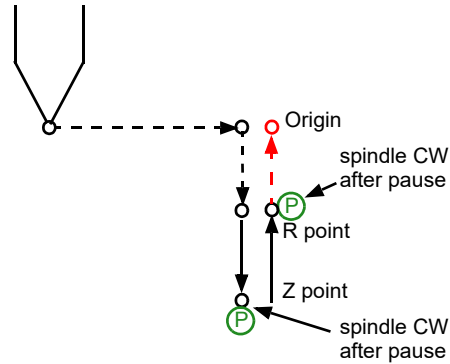
Motion process:

- 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:

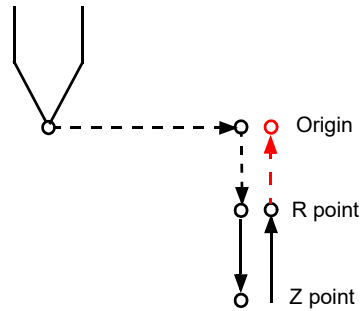
- ① Drilling motion (-Z) → cutting feed
- ② Motion at the bottom of the hole → none
- ③ Retraction motion (+Z) → cutting feed

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:

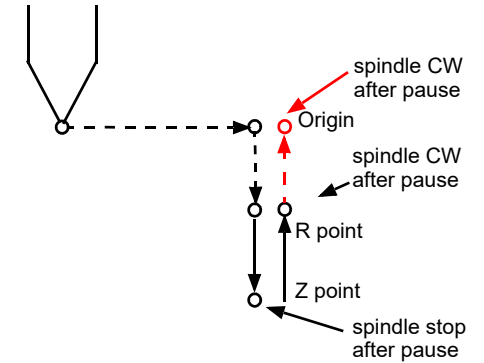
- ① Drilling motion (-Z) → cutting feed
- ② Motion at the bottom of the hole → spindle stop
- ③ Retraction motion (+Z) → fast feed

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:

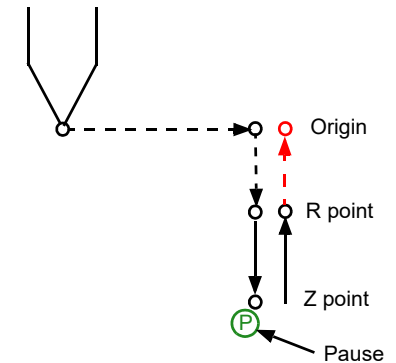
- ① Drilling motion (-Z) → cutting feed
- ② Motion at the bottom of the hole → pause
- ③ Retraction motion (+Z) → cutting feed

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

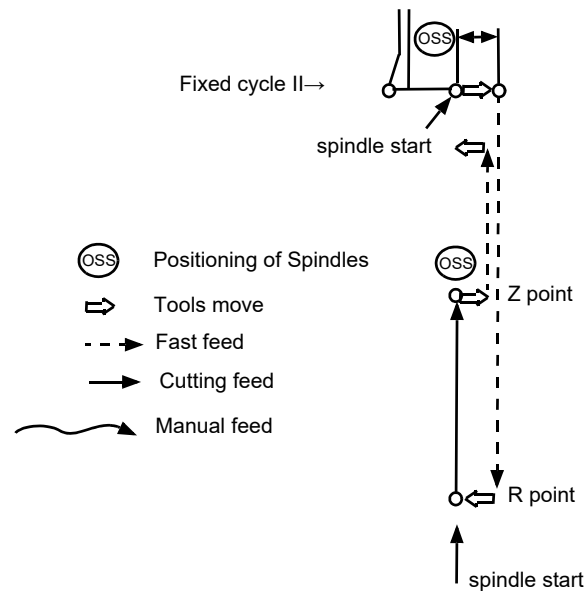
Common programming structures:

G87 X—Y—Z—R—Q—P—F—L

Motion process:

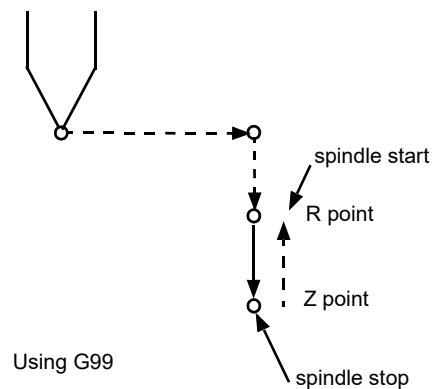
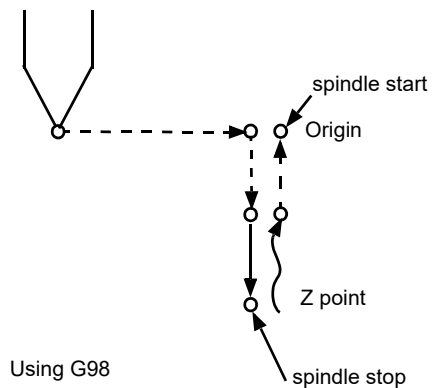
- ① Drilling motion (-Z) → cutting feed
- ② Motion at the bottom of the hole → spindle stops
- ③ Retraction motion (+Z) → manual operation or fast feed

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.



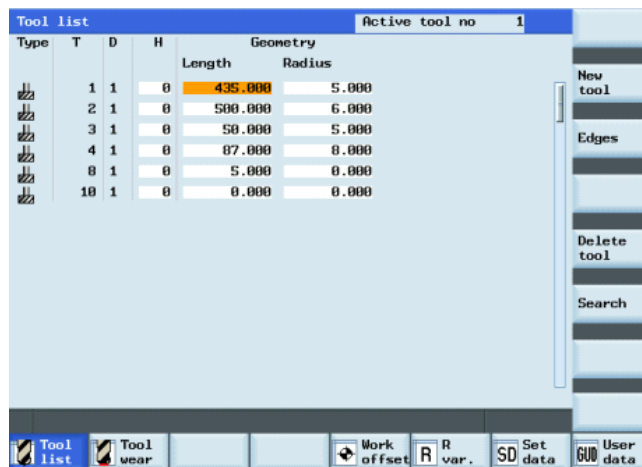
Programs in ISO mode in the 808D have their own rules. Suitable changes must be made at the appropriate positions so that you can run the ISO programs!

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;	O0001;Delete this line
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.

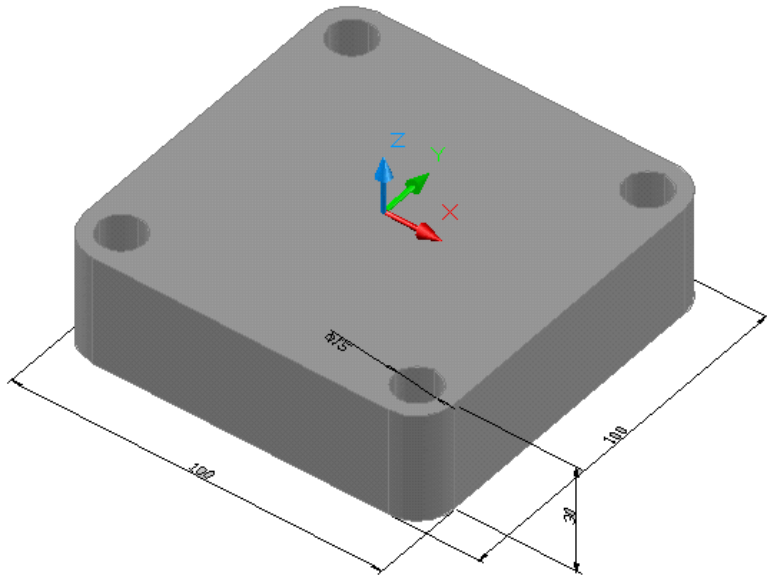
Basic Theory

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

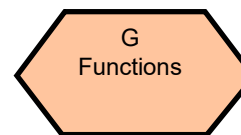
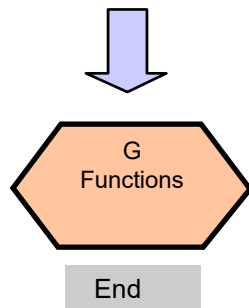
N160 G0G40G90Z60
N170 M09M05
N180 M30

; 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



12 Annex

12.1 Copyright

This document is copyright. All derived rights are also reserved, especially those of translation, re-printing, use of figures, broadcast, reproduction by photo-mechanical or similar means and recording in data processing systems, neither partial nor 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 Product follow-up

We have a follow-up service for our products which extends even after 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

12.4 Change information manual

Chapter	Short note	new version no.
3	Setting jumpers central lubrication	1.0.1
	Brief instruction 808D milling	1.0.2
	Enlargement Alarm Listing	
6.5.3	Electronic handwheel, sliding door	1.0.2
new	option, rotary table for F105 Advanced	1.0.3
11.5	Recommended wear and tears parts	1.0.4
11.3	Inspection and maintenance list	1.0.4
3	Central lubrication DRB-125A inserted	1.0.4
5 ; 6.5.7 ; 7.3	Tool changer, User-defined keys	1.0.5
15	Wiring diagram F105 Advanced	1.0.6
2	F80 and F105 variants, F105 Advanced spindle power	1.1.0
	Update circuit diagram F105 + parts list	1.1.1
	Update circuit diagram F105 + parts list	1.1.2
1.7	Industrial security	1.1.3
3	Interpartmental transport	1.1.4
2	Spindle motor power graph F80 + F105	1.1.5

F80_GB_14.fm



Chapter	Short note	new version no.
parts	new machine housing	1.1.6

12.5 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
MCP	M achine C ontrol P anel
PPU	P oly P ower U nit
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 will maintain there. In this case, the position is an angular position of a rotatory motor and in case of a linear motor it is a linear positioning. The starting-up and maintaining of the predetermined position is performed by means of a control.



12.6 Re-export

All contract products and technical knowledge are delivered by Optimum Maschinen Germany GmbH while ensuring the currently applicable AWG/AWV/ECDual-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 the customer intends to re-export contract products he is obliged to follow US American, European and national export regulations. It is forbidden to re-export contract products contrary to these regulations.

The customer must inform himself about the currently applicable regulations and terms (Office for Export, 65760 Eschborn/Taunus or US Department of Commerce, Office of Export Administration, Washington D. C. 20230). Regardless of the fact if the customer indicates the final place of destination of the delivered contract products the customer is obliged to gather the necessary authorizations of the correspondingly responsible competent authorities for foreign trade before exporting such products. Optimum Maschinen Germany GmbH is not obliged to give any information and does not submit any guarantee that the contract products comply with the corresponding export regulations. The customer cannot claim that Optimum Maschinen Germany GmbH adapts the contract products to the corresponding export regulations.

Any forwarding of contract products to the customer by third parties with or without knowledge of Optimum Maschinen Germany GmbH requires at the same time the transfer of the export authorization 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 contract 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 as well as 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 it is forbidden to deliver to natural or juridical persons who are in any relation with the support, development, production or use of chemical, biological or nuclear weapons of mass destruction in any way.

12.7 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 case:



- **Fragile goods (Goods require careful handling)**
- **Protect against moisture and humid environment**
- **Prescribed position of the packing case (Marking of the top surface - arrows pointing to the top)**



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.



12.8 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 have been promised in the framework of an individual contractual agreement.

- 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. The property of replaced products or components passes on to OPTIMUM 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
 - Non-authorised 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 as well not subject to the 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 servants performs in order to fulfil in the frame of an additional guarantee 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 among traders is Bamberg.
- If one of the above mentioned agreements is totally or partially inefficient and/or null, it is considered as agreed what is closest to the will of the warrantor and which remains in the framework of the limits of liability and warranty which are predefined by this contract.

12.9 Advice for disposal / Options of re-use

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 do not throw away the packaging and the used machine later on, but dispose of your material according to the guidelines established by your municipality or by the responsible waste management company.



12.9.1 Decommissioning

CAUTION!

Immediately decommission used machines in order to avoid later misuse and endangering of the environment or of persons.



- **Disconnect the machine from the mains.**
- **Cut the connecting cable into two.**
- **Remove all operating materials from the used device which are harmful to the environment.**
- **If required, remove the batteries and accumulators.**
- **If required, disassemble the machine into easy-to-handle and usable components and parts.**
- **Supply the machine components and operating materials to the provided disposal routes.**

12.9.2 Disposal of the packaging of the new machine

All used packaging materials and packaging aids of the machine are recyclable and generally need to be transported to the material recycling.

The packaging wood can be supplied to the disposal or reuse.

It is possible to crush any packaging material made of cardboard and supply it to the waste paper collection.

The films are made of polyethylene (PE) and the upholstery parts are made of polystyrene (PS). It is possible to reuse these materials after reconditioning, if you supply them to the collection station or to the responsible waste management company.

Supply the packaging material only correctly sorted, so that it is possible to directly supply it to the reuse.

12.9.3 Disposal of the used machine

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 electrical devices include a number of reusable materials as well as components which are harmful to the environment. Please help that these components are disposed of separately and professionally. In case of doubt, please contact your municipal waste management company. If necessary contact a specialised waste disposal centre to get help for the disposal.

12.9.4 Disposal of electrical and electronic components

Please make sure that the electrical components are disposed of professionally and according to the legal regulations.

The machine is composed of electrical and electronic components and must not be disposed of as household waste. According to the European directive 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.

Being the machine operator, you should gather information regarding the authorised collection or disposal system which applies for your company.

Please make sure that batteries and/or accumulators are disposed of in a professional way and according to the legal regulations. Please throw empty accumulators only into the collection boxes of retail markets or municipal waste disposal companies.



12.9.5 Disposal of lubricants and cooling lubricants

ATTENTION!

Please imperatively make sure that the used coolants and lubricants are disposed of in an environmentally friendly way. Observe the disposal advices of your municipal waste management companies.



INFORMATION

Do not mix up used coolant emulsions and oils, since only used oils which had not been mixed up are recyclable.

The disposal advices for the used lubricants are made available by the manufacturer for lubricants. If necessary, ask for the product-specific data sheets.





EC Declaration of Conformity

according to Machinery Regulation 2023/1230 Annex V Part A

The manufacturer / distributor Optimum Maschinen Germany GmbH
Dr.-Robert-Pfleger-Str. 26
D - 96103 Hallstadt, Germany

hereby declares that the following product

Product designation: Machining center

Type designation: F80

fulfils all the relevant provisions of the Machinery Regulation specified above and the additionally applied directives (in the following) - including the changes which applied at the time of the declaration.

Description:

Numerically controlled machine tool with vertical spindle arrangement and tool changing device for several machining processes.

The following additional EU Directives have been applied:

EMC Directive 2014/30/EU ; Restriction of the use of certain hazardous substances in electrical and electronic equipment 2015/863/EU

The following harmonized standards were applied:

EN ISO 16090-1: 2019-12 Machine tools safety - Machining centres, Milling machines, Transfer machines - Part 1: Safety requirements

EN ISO 12100: 2011-03 Safety of machinery - General principles for design - Risk assessment and risk reduction

EN 60204-1: 2019-06 Safety of machines - Electrical equipment of machines

EN 13849-1: 2016-06 Safety of machinery - Safety related parts of controls - Part 1: General design principles

EN 13849-2: 2013-02 Safety of machinery - Safety related parts of controls - Part 2: Validation

EN 50370-2: 2003-08 Electromagnetic compatibility (EMC) - Product family standard for machine tools - Part 2: Immunity

EN 55011: 2022-05 Industrial, scientific and medical equipment - Radio-frequency disturbance characteristics - Limits and methods of measurement - class A

Name and address of the person authorized to compile the technical file:

Kilian Stürmer, phone: +49 (0) 951 96555 - 800

Kilian Stürmer (CEO, General Manager)
Hallstadt, 2024-02-22



Index

A

Accident report	18
Alarme	74
Alarms	72
Aligning the machine	29
Anchored assembly	28
Anchoring-free assembly	29
Assembly	
anchored	28
of the coolant / lubricant pumps	30
AWG/AWV/EG-Dual-Use	208

B

Brief instruction 808D Milling	117
--------------------------------------	-----

C

Central lubrication	68
Central lubrication system	32, 60
Commissioning	32
tool	58
Compressed air supply	30
Connection	
compressed air	30
electrical	30
Control elements	54
Cooling lubricant	76
Cooling lubricants	76
Coordinate systems	37
Copyright	206
Corrosion protection	30
Customer service	83
Customer service technician	83

D

Data backup	33
Data interfaces	60
Declaration of conformity	212
DIN Code	60
Disinfection	
Cooling lubricant tank	84
Disposal	211
Drain compressed	
Air service unit	82
Dual-Use	208

E

Earth leakage switch	35
EC - declaration of conformity	212
Electrical connection	30
Electrical system	18
Electronic handwheel	57
EMERGENCY STOP push button	13
Error messages	72
Ersatzteile	88

F

Fehlermeldungen	74
Functional test	32

H

Handling	54
Hinweise	74

I

Indicating elements	54
Inserting a tool	57
Inspection	78
Installation plan	29
Interdepartmental transport	24
Interlock switch sliding door	82
ISO Code	60

K

Key combination	46
-----------------------	----

M

M - code list	70
M functions	70
Machine Control Panel	47
Main switch	13
Manual tool change	58
Mechanical maintenance work	18
Meldungen	74
Messages	72
Messages and error messages	72
Milling table	22
Montage	
verankert	28

N

Notes	72
-------------	----

O

Obligations	
of the operating company	12
of the operator	12
Operating material	76
Operational modes	55, 59
Operator's log	78

P

Panel Processing Unit	42
Passwords	52
Product follow-up	206
Programming	55
Prohibition, warning and mandatory signs	15
Protection levels	52
Protective	
equipment	16

Q

Qualification of the staff	
safety	10

R

Re-export	208
Reference point approach	57
Requirements regarding the installation site	26

S

Save data function	33
Scope of delivery	25
Selecting the speed	60
SINUMERIK 808 D	75
Specialist dealer	83
Storage	26

T

Tool	
commissioning	58
Tool changer	
manual operation	59
Tools	23
Transport	24, 25
Turning off the CNC- machine	59
Turning on the machine	56

U

Using	
electronic handwheel	57
Using lifting equipment	17

V

Verankerte Montage	28
--------------------------	----

