

Grizzly ***Industrial, Inc.***®

MODEL G0877 **10" X 31" ENCLOSED CNC MILL** **w/AUTO TOOL CHANGER** **OWNER'S MANUAL** *(For models manufactured since 02/19)*





WARNING!

This manual provides critical safety instructions on the proper setup, operation, maintenance, and service of this machine/tool. Save this document, refer to it often, and use it to instruct other operators.

Failure to read, understand and follow the instructions in this manual may result in fire or serious personal injury—including amputation, electrocution, or death.

The owner of this machine/tool is solely responsible for its safe use. This responsibility includes but is not limited to proper installation in a safe environment, personnel training and usage authorization, proper inspection and maintenance, manual availability and comprehension, application of safety devices, cutting/sanding/grinding tool integrity, and the usage of personal protective equipment.

The manufacturer will not be held liable for injury or property damage from negligence, improper training, machine modifications or misuse.



WARNING!

Some dust created by power sanding, sawing, grinding, drilling, and other construction activities contains chemicals known to the State of California to cause cancer, birth defects or other reproductive harm. Some examples of these chemicals are:

- **Lead from lead-based paints.**
- **Crystalline silica from bricks, cement and other masonry products.**
- **Arsenic and chromium from chemically-treated lumber.**

Your risk from these exposures varies, depending on how often you do this type of work. To reduce your exposure to these chemicals: Work in a well ventilated area, and work with approved safety equipment, such as those dust masks that are specially designed to filter out microscopic particles.

Table of Contents

0.1	Machine data sheet	3			
1	Safety		6	Operation	
1.1	Safety instructions for machinery	6	6.1	Safety	54
1.2	Additional safety for CNC mills/lathes	8	6.2	Control and indicating elements	54
1.3	Safety instructions (warning notes)	9	6.3	Operational modes	55
1.4	Intended use	11	6.4	Programming	55
1.5	Reasonably foreseeable misuses	11	6.5	Operation of the machine	56
1.6	Possible dangers w/CNC machines	12	6.6	Operational modes	61
1.7	Qualification of the staff	13	6.7	Programming	61
1.8	Operator positions	14	6.8	Start program	62
1.9	Safety devices	15	6.9	Central lubrication system	63
1.10	Safety check	17	6.10	Data interfaces and current collection	63
1.11	Personnel protective equipment	18	6.11	Selecting the speed	63
1.12	Safety during operation	18	6.12	CNC Rotary Table cutting speed char	66
1.13	Safety during maintenance	19	7	M - Code list, M functions	
1.14	Disconnecting/securing CNC machine	19	7.1	M-function for milling machines	68
1.15	Accident report	20	7.2	G functions to PAL	70
1.16	Electrical system	20	8	Notes, messages and error messages	71
1.17	Clamping devices for workpieces and tools	20	9	SINUMERIK 808 D	74
1.18	Environmental protection	21	10	SINUMERIK 808 D Advanced	75
2	Power supply		11	Maintenance	
2.1	Availability	22	11.1	Operating material	77
2.2	Full-load current rating	22	11.2	Safety	78
2.3	Circuit information	22	11.3	Inspection and maintenance	79
2.4	Circuit requirements for 220V	22	11.4	Interlock switch sliding door	82
2.5	Connection type	23	11.5	Cooling lubricants and tanks	83
2.6	Grounding instructions	23	12	Brief instructions for 808 D milling	
2.7	Extension cords	23	12.1	Preparation	86
3	Assembly and commissioning		12.2	Switch on and referencing	92
3.1	Scope of delivery	24	12.3	Tool Setup	96
3.2	Transport	24	12.4	Workpiece setup	112
3.3	Installation and assembly	25	12.5	Create part program: part 1	121
3.4	Installation plan	26	12.6	Create part program: part 2	137
3.5	Installation and assembly	27	12.7	Simulate program	163
3.6	First commissioning	32	12.8	Test program	167
3.7	Refill central lubrication system	33	12.9	Machine pieces	171
3.8	Functional test and controls	35	12.10	Program restart	177
4	General information about CNC		12.11	Additional information: part 1	181
4.1	Compensation of geometry	37	12.12	Additional information: part 2	201
4.2	Coordinate systems on CNC-machines	37	12.13	Sample program	219
4.3	NC mathematics	39	12.14	ISO mode	234
4.4	Trigonometric functions	40	12.15	Appendix	253
5	User interface, machine control panel				
5.1	Screen arrangement	41			
5.2	Elements on the PPU front	47			
5.3	Elements on the MCP	47			
5.4	Protection levels	52			

Table of Contents (Cont.)

13 Wiring Diagrams

13.1	Wiring safety instructions	258
13.2	System wiring diagrams (1-6)	259
13.3	Main motor wiring diagram	262
13.4	X/Y/Z axes wiring diagram	263

14 Parts

14.1	Column	264
14.2	Milling head	266
14.3	Milling table	267
14.4	X-Axis	268

14.5	Y-Axis	269
14.6	Z-Axis	270
14.7	Tool changer	271
14.8	Housing	272
14.9	Stand & coolant tank	273
14.10	Pneumatic system	274
14.11	Labels & cosmetics	275

15	Warranty & returns	277
-----------	-------------------------------------	------------



MACHINE DATA SHEET

Customer Service #: (570) 546-9663 · To Order Call: (800) 523-4777 · Fax #: (800) 438-5901

MODEL G0877 10" X 31" ATC ENCLOSED CNC MILL

Product Dimensions:

Weight 3744 lbs.
Width (side-to-side) x Depth (front-to-back) x Height 77 x 68 x 87 in.
Footprint (Length/Width) 117 x 88 in.

Shipping Dimensions:

Type Wood Crate
Content Machine
Weight 4200 lbs.
Length x Width x Height 83 x 70 x 95 in.
Must Ship Upright Yes

Electrical:

Power Requirement 220V, 3-Phase, 60 Hz
Full-Load Current Rating 14.2A
Minimum Circuit Size 20A
Connection Type Permanent (Hardwire to Shutoff Switch)
Power Cord Included Yes
Power Cord Length 6-1/2 ft.
Power Cord Gauge 12 AWG

Motor:

Spindle

Horsepower 3 HP
Phase 3-Phase
Amps 6.4A
Speed 10,000 RPM
Type Servo
Power Transfer Belt Drive
Bearings Shielded & Permanently Lubricated

X-Axis

Horsepower 1 HP
Phase 3-Phase
Amps 2.1A
Speed 4,000 RPM
Type Servo
Power Transfer Direct Drive
Bearings Sealed & Permanently Lubricated

Y-Axis

Horsepower 1 HP
Phase 3-Phase
Amps 2.1A
Speed 4,000 RPM
Type Servo
Power Transfer Direct Drive
Bearings Sealed & Permanently Lubricated



Z-Axis

Horsepower	1.3 HP
Phase	3-Phase
Amps	3A
Speed	3,000 RPM
Type.....	Servo
Power Transfer.....	Direct Drive
Bearings	Sealed & Permanently Lubricated

Tool Changer

Horsepower	37 Watts
Phase	3-Phase
Amps	0.27A
Speed	1300 RPM
Type.....	Servo
Power Transfer.....	Gearbox
Bearings	Sealed & Permanently Lubricated

Coolant Pump

Horsepower	0.6 HP
Phase	3-Phase
Amps	0.8A
Speed	2900 RPM
Type.....	TEFC Induction
Power Transfer.....	Direct Drive
Bearings	Shielded & Permanently Lubricated

Main Specifications:

Operation Information

Max Distance Spindle to Column	12-1/2 in.
Max Distance Spindle to Table	18-5/8 in.
Longitudinal Table Travel (X-Axis)	15-5/8 in.
Cross Table Travel (Y-Axis)	9 in.
Vertical Head Travel (Z-Axis)	15-5/8 in.
Position Resolution.....	± 0.0008 in. (0.020 mm)
Repeat Position Resolution.....	± 0.0005 in. (0.013 mm)

Tool Changer Info

Type.....	Turntable Style
Capacity.....	10 Tools
Maximum Tool Weight.....	13 lbs.
Maximum Tool Diameter with Adjacent Tool	4 in.
Tool Change Average Time	6 Seconds

Table Info

Table Length	31-1/2 in.
Table Width	10-1/4 in.
Table Thickness	2-3/4 in.
Table Height (from Floor/Base).....	37-1/2 in.
Table Weight Capacity	330 lbs.
Number of T-Slots	5
T-Slot Size.....	5/8 in.
T-Slots Centers	2 in.
X-Axis Rapid Feed Rate.....	400 IPM
Y-Axis Rapid Feed Rate.....	400 IPM
Z-Axis Rapid Feed Rate.....	400 IPM



Spindle Info

Spindle Taper BT30
Pull Stud BT30 x 45°
Spindle Speed Range 50 - 10,000 RPM
Spindle Bearings (Size & Type) Angular Contact Bearings

Fluid Capacities

Coolant Capacity 42 Gallons

Construction

Spindle Housing/Quill Steel
Table Cast Iron
Head Cast Iron
Column/Base Cast Iron
Base Cast Iron
Stand Cast Iron
Paint Type/Finish Enamel

Other Specifications:

Country of Origin China
Warranty 1 Year
Approximate Assembly & Setup Time 2 Hours
Serial Number Location Machine ID Label
ISO 9001 Factory Yes

Features:

- Siemens Sinumerk® CNC Controller
- Servo Motors with Ball Screws on All Axes
- Built-In Pendant/Hand Controller
- 3 HP Spindle Motor
- 50–10,000 RPM Spindle Speeds
- Hands-Free Tool Changes with 10-Position Automatic Tool Changer (ATC)
- Rapid Table Speeds of Up to 400 in./min.
- Heavy-Duty BT30 Spindle



SAFETY

For Your Own Safety, Read Instruction Manual Before Operating This Machine

The purpose of safety symbols is to attract your attention to possible hazardous conditions. This manual uses a series of symbols and signal words intended to convey the level of importance of the safety messages. The progression of symbols is described below. Remember that safety messages by themselves do not eliminate danger and are not a substitute for proper accident prevention measures. Always use common sense and good judgment.



Indicates an imminently hazardous situation which, if not avoided, **WILL** result in death or serious injury.



Indicates a potentially hazardous situation which, if not avoided, **COULD** result in death or serious injury.



Indicates a potentially hazardous situation which, if not avoided, **MAY** result in minor or moderate injury. It may also be used to alert against unsafe practices.

NOTICE

Alerts the user to useful information about proper operation of the machine to avoid machine damage.

Safety Instructions for Machinery

WARNING

OWNER'S MANUAL. Read and understand this owner's manual **BEFORE** using machine.

TRAINED OPERATORS ONLY. Untrained operators have a higher risk of being hurt or killed. Only allow trained/supervised people to use this machine. When machine is not being used, disconnect power, remove switch keys, or lock-out machine to prevent unauthorized use—especially around children. Make your workshop kid proof!

DANGEROUS ENVIRONMENTS. Do not use machinery in areas that are wet, cluttered, or have poor lighting. Operating machinery in these areas greatly increases the risk of accidents and injury.

MENTAL ALERTNESS REQUIRED. Full mental alertness is required for safe operation of machinery. Never operate under the influence of drugs or alcohol, when tired, or when distracted.

ELECTRICAL EQUIPMENT INJURY RISKS. You can be shocked, burned, or killed by touching live electrical components or improperly grounded machinery. To reduce this risk, only allow qualified service personnel to do electrical installation or repair work, and always disconnect power before accessing or exposing electrical equipment.

DISCONNECT POWER FIRST. Always disconnect machine from power supply **BEFORE** making adjustments, changing tooling, or servicing machine. This prevents an injury risk from unintended startup or contact with live electrical components.

EYE PROTECTION. Always wear ANSI-approved safety glasses or a face shield when operating or observing machinery to reduce the risk of eye injury or blindness from flying particles. Everyday eyeglasses are **NOT** approved safety glasses.



WARNING

OWNER'S MANUAL. Read and understand this owner's manual BEFORE using machine.

TRAINED OPERATORS ONLY. Untrained operators have a higher risk of being hurt or killed. Only allow trained/supervised people to use this machine. When machine is not being used, disconnect power, remove switch keys, or lock-out machine to prevent unauthorized use—especially around children.

WEARING PROPER APPAREL. Do not wear clothing, apparel or jewelry that can become entangled in moving parts. Always tie back or cover long hair. Wear non-slip footwear to reduce risk of slipping and losing control or accidentally contacting cutting tool or moving parts.

MENTAL ALERTNESS REQUIRED. Full mental alertness is required for safe operation of machinery. Never operate under the influence of drugs or alcohol, when tired, or when distracted.

ELECTRICAL EQUIPMENT INJURY RISKS. You can be shocked, burned, or killed by touching live electrical components or improperly grounded machinery. To reduce this risk, only allow qualified service personnel to do electrical installation or repair work, and always disconnect power before accessing or exposing electrical equipment.

DISCONNECT POWER FIRST. Always disconnect machine from power supply BEFORE making adjustments, changing tooling, or servicing machine. This prevents an injury risk from unintended startup or contact with live electrical components.

HAZARDOUS DUST. Dust created by machinery operations may cause cancer, birth defects, or long-term respiratory damage. Be aware of dust hazards associated with each workpiece material. Always wear a NIOSH-approved respirator to reduce your risk.

EYE PROTECTION. Always wear ANSI-approved safety glasses or a face shield when operating or observing machinery to reduce the risk of eye injury or blindness from flying particles. Everyday eyeglasses are NOT approved safety glasses.

HEARING PROTECTION. Always wear hearing protection when operating or observing loud machinery. Extended exposure to this noise without hearing protection can cause permanent hearing loss.

REMOVE ADJUSTING TOOLS. Tools left on machinery can become dangerous projectiles upon startup. Never leave chuck keys, wrenches, or any other tools on machine. Always verify removal before starting!

USE CORRECT TOOL FOR THE JOB. Only use this tool for its intended purpose—do not force it or an attachment to do a job for which it was not designed. Never make unapproved modifications—modifying tool or using it differently than intended may result in malfunction or mechanical failure that can lead to personal injury or death!

GUARDS & COVERS. Guards and covers reduce accidental contact with moving parts or flying debris. Make sure they are properly installed, undamaged, and working correctly BEFORE operating machine.

UNATTENDED OPERATION. To reduce the risk of accidental injury, turn machine **OFF** and ensure all moving parts completely stop before walking away. Never leave machine running while unattended.

MAINTAIN WITH CARE. Follow all maintenance instructions and lubrication schedules to keep machine in good working condition. A machine that is improperly maintained could malfunction, leading to serious personal injury or death.

DAMAGED PARTS. Regularly inspect machine for damaged, loose, or mis-adjusted parts—or any condition that could affect safe operation. Immediately repair/replace BEFORE operating machine. For your own safety, DO NOT operate machine with damaged parts!

EXPERIENCING DIFFICULTIES. If at any time you experience difficulties performing the intended operation, stop using the machine! Contact our Technical Support at (570) 546-9663.



Additional Safety for CNC Mills/Lathes

WARNING

You can be seriously injured or killed by getting clothing, jewelry, or long hair entangled with rotating cutter/spindle. You can be severely cut or have fingers amputated from contact with rotating cutters. You can be blinded or struck by broken cutting tools, metal chips, workpieces, or adjustment tools thrown from the rotating spindle with great force. To reduce your risk of serious injury when operating this machine, completely heed and understand the following:

UNDERSTAND ALL CONTROLS. Make sure you understand the function and proper use of all controls before starting. This will help you avoid making mistakes that result in serious injury.

AVOIDING ENTANGLEMENT. DO NOT wear loose clothing, gloves, or jewelry, and tie back long hair. Keep all guards in place and secure. Always allow spindle to stop on its own. DO NOT stop spindle using your hand or any other object.

EYE INJURIES. Operator and bystanders MUST wear ANSI-approved safety glasses to help protect eyes from thrown metal shards and chips.

USE CORRECT SPINDLE SPEED. Follow recommended speeds and feeds for each size and type of cutting tool. This helps avoid tool breakage during operation and ensures best cutting results.

INSPECT CUTTING TOOL. Inspect cutting tools for sharpness, chips, or cracks before each use. Replace dull, chipped, or cracked cutting tools immediately.

UNATTENDED MACHINE. Operator MUST be present to immediately stop machine in case of malfunction to prevent injury to bystanders and machine damage.

POWER DISRUPTION. In event of power outage during operation, turn spindle switch **OFF** to avoid a possible sudden startup once power is restored.

SECURE WORKPIECE TO TABLE. Clamp workpiece to table or secure in a vise mounted to table, so workpiece cannot unexpectedly shift or spin during operation. NEVER hold workpiece by hand during operation.

DISCONNECT POWER FIRST. To reduce risk of electrocution or injury from unexpected startup, make sure mill/drill is turned **OFF**, disconnected from power, and all moving parts have come to a complete stop before changing cutting tools or starting any inspection, adjustment, or maintenance procedure.

CLEAN MACHINE SAFELY. Metal chips or shavings can be razor sharp. DO NOT clear chips by hand or compressed air that can force chips farther into machine—use a brush or vacuum instead. Never clear chips while spindle is turning.

PROPERLY MAINTAIN MACHINE. Keep machine in proper working condition to help ensure that it functions safely and all guards and other components work as intended. Perform routine inspections and all necessary maintenance. Never operate machine with damaged or worn parts that can break or result in unexpected movement during operation.

SAFE OPERATING LOCATION. DO NOT place machine where it can be exposed to rain or moisture. Exposure to water creates a shock hazard and will reduce life of machine.



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.
	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



general danger



by a warning of



injuries to hands,



hazardous electrical
voltage,



rotating parts.

1.3.2 Other pictograms



Activation forbidden!



Forbidden to enter in the
machine!



Forbidden to extinguish
with water!



Access forbidden!



Use protective boots!



Use ear protection!



Use protective glasses!



Read the operating
instruction !



Warning of suspended
loads!



Warning of oxidizing
materials!



Warning of explosive!



Warning of danger of
slipping!



Protect the environment!



Contact address



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. See "Cooling lubricants" on page 85.

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

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

If the CNC-machine is used in any way other than described above, modified without the Intended use approval of the company Grizzly Industrial, Inc. then the CNC-machine is being used improperly.

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 Grizzly Industrial, Inc.

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 Qualification of the staff

1.7.1 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' 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.



1.7.2 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-machine G0877 by Grizzly Industrial, Inc.

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.

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.

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.

1.8 Operator positions

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



Obligations of the operating company

Obligations of the operator

Additional requirements regarding the qualification



1.9 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.

1.9.1 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.



1.9.2 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

1.9.3 Control technical protection

WARNING!

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

- injuries due to tools, workpieces or fragments hereof which are flying off at high speed,
- contact with rotating parts,
- 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.



1.9.4 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

1.9.5 Prohibition, warning and mandatory signs

INFORMATION

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



1.10 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). 	
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):	

1.11 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.



1.12 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



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 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.

1.13 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.

1.14 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).



1.14.1 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!



1.14.2 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!

1.15 Accident report

Inform your superiors and Grizzly Industrial, Inc. 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.



1.16 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

1.17 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.



1.18 Environmental protection and water conservation

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

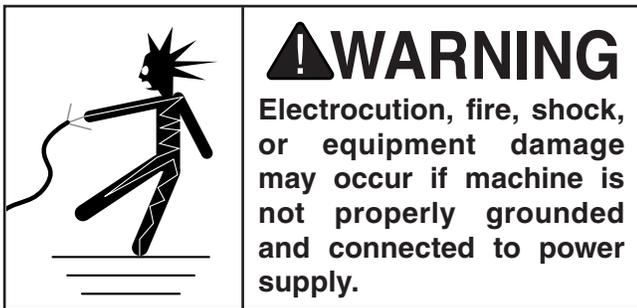
When operating, decommissioning or disassembling the CNC-machine or parts hereof, please follow the requirements of the Water Resources Law. Please find detailed information about this topic in the regulation about devices to treat materials which are hazardous to water.



POWER SUPPLY

Availability

Before installing the machine, consider the availability and proximity of the required power supply circuit. If an existing circuit does not meet the requirements for this machine, a new circuit must be installed. To minimize the risk of electrocution, fire, or equipment damage, installation work and electrical wiring must be done by an electrician or qualified service personnel in accordance with all applicable codes and standards.



Full-Load Current Rating

The full-load current rating is the amperage a machine draws at 100% of the rated output power. On machines with multiple motors, this is the amperage drawn by the largest motor or sum of all motors and electrical devices that might operate at one time during normal operations.

Full-Load Current Rating at 220V .. 14.2 Amps

The full-load current is not the maximum amount of amps that the machine will draw. If the machine is overloaded, it will draw additional amps beyond the full-load rating.

If the machine is overloaded for a sufficient length of time, damage, overheating, or fire may result—especially if connected to an undersized circuit. To reduce the risk of these hazards, avoid overloading the machine during operation and make sure it is connected to a power supply circuit that meets the specified circuit requirements.

Circuit Information

A power supply circuit includes all electrical equipment between the breaker box or fuse panel in the building and the machine. The power supply circuit used for this machine must be sized to safely handle the full-load current drawn from the machine for an extended period of time. (If this machine is connected to a circuit protected by fuses, use a time delay fuse marked D.)

! CAUTION
For your own safety and protection of property, consult an electrician if you are unsure about wiring practices or electrical codes in your area.

Note: *Circuit requirements in this manual apply to a dedicated circuit—where only one machine will be running on the circuit at a time. If machine will be connected to a shared circuit where multiple machines may be running at the same time, consult an electrician or qualified service personnel to ensure circuit is properly sized for safe operation.*

Circuit Requirements for 220V

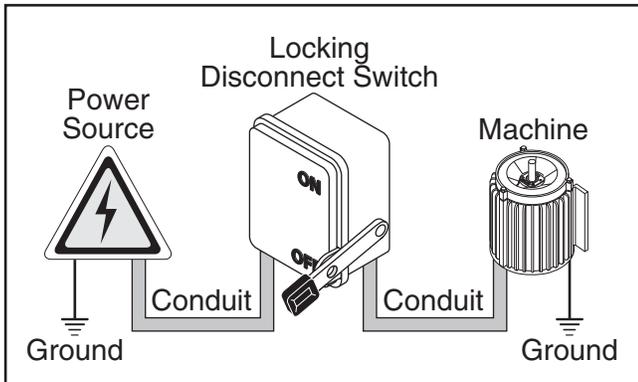
This machine is prewired to operate on a power supply circuit that has a verified ground and meets the following requirements:

Nominal Voltage 220V, 230V, 240V
Cycle 60 Hz
Phase 3-Phase
Power Supply Circuit 20 Amps



Connection Type

A permanently connected (hardwired) power supply is typically installed with wires running through mounted and secured conduit. A disconnecting means, such as a locking switch (see following figure), must be provided to allow the machine to be disconnected (isolated) from the power supply when required. This installation must be performed by an electrician in accordance with all applicable electrical codes and ordinances.



Typical setup of a permanently connected machine.

Grounding Instructions

In the event of a malfunction or breakdown, grounding provides a path of least resistance for electrical current to reduce the risk of electric shock. A permanently connected machine must be connected to a grounded metal permanent wiring system; or to a system having an equipment-grounding conductor. All grounds must be verified and rated for the electrical requirements of the machine. Improper grounding can increase the risk of electric shock!

!WARNING

Serious injury could occur if you connect machine to power before completing setup process. DO NOT connect to power until instructed later in this manual.

Extension Cords

Since this machine must be permanently connected to the power supply, an extension cord cannot be used.



3 Assembly and commissioning

INFORMATION

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



3.1 Scope of delivery

Compare the delivery volume with the attached packing list.

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

3.2 Transport

WARNING!

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



- 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

WARNING!

Use of unstable lifting equipment and load-suspension gears that break under load can cause very serious injury or even death.

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

Observe the accident prevention regulations.

Fasten the loads properly.

Never walk under suspended loads!

→ 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.

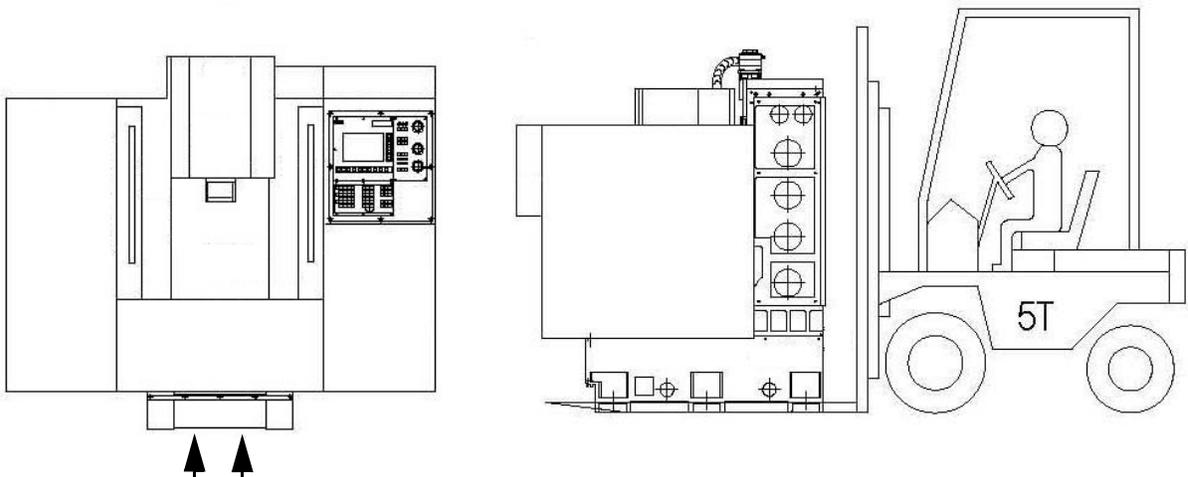


- Disassemble the clamping bolts which are used to fix the machine on the pallet.
- Lift the CNC-machine carefully from the pallet of the transportation box by means of a crane or a fork-lift truck.
- Bring the CNC-machine with an appropriate handling device, e.g. electric pallet truck or fork-lift truck at their firm position.
- Make sure that the load attachment does not cause damage to components or paint.

WARNING!

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

👉 "Machine mounting" on page 26



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

3.3 Installation and assembly

3.3.1 Requirements to the installation site

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

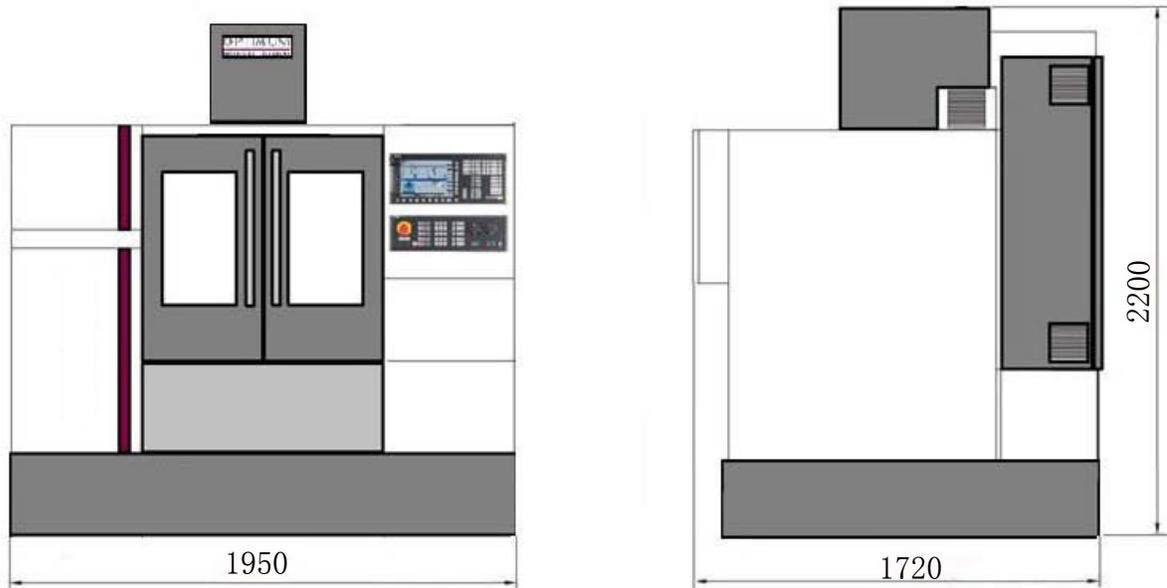
The working area for operating, maintenance and repair must not be hindered. Follow the prescribed safety areas and escape routes according to environmental conditions for the operation of the CNC-machine.

INFORMATION

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



3.4 Installation plan



Img.3-2, 3-3: Installation plan

3.4.1 Machine mounting

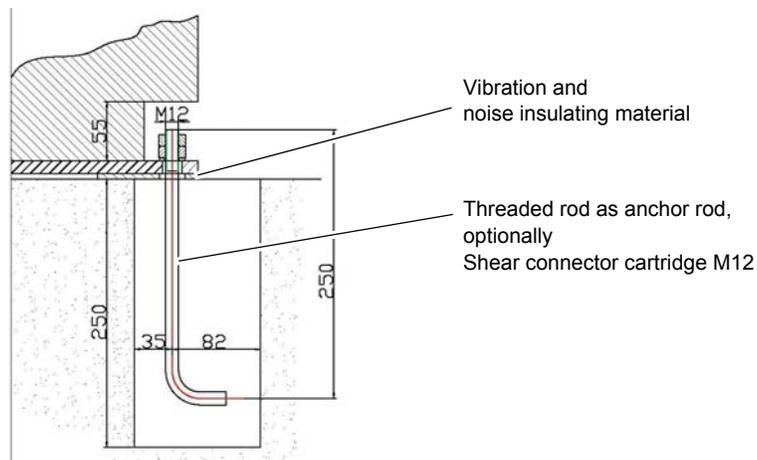
Anchoring-free assembly

- 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.



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.



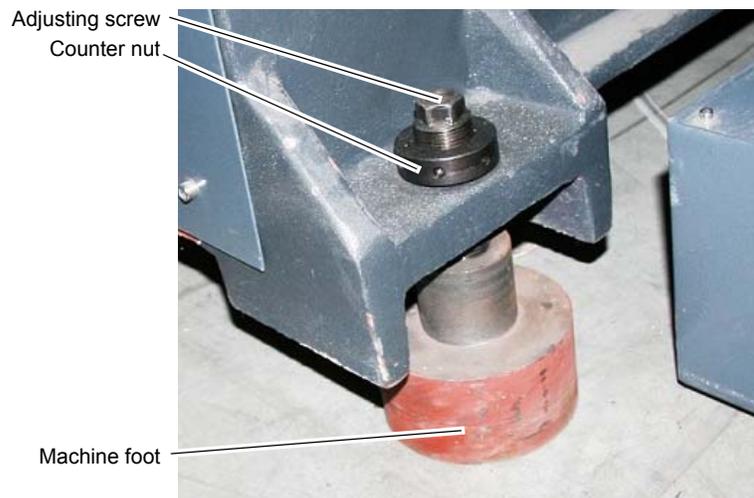
Img.3-4: Drawing of the anchoring

3.4.2 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 26
- The slope deviation of all levels must not exceed 0.03/1000mm.

3.5 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.3-5: Machine feet

ATTENTION!

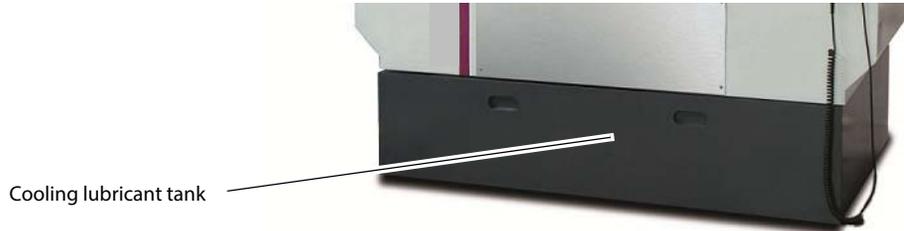
An insufficient rigidity of the substructure leads to superposition of vibrations between the CNC machine and the substructure (natural frequency of the components). Critical speeds and moves in the axis with displeasing vibrations are rapidly achieved in case of insufficient rigidity of the whole system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



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

3.5.1 Mounting the coolant / lubricant tank

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

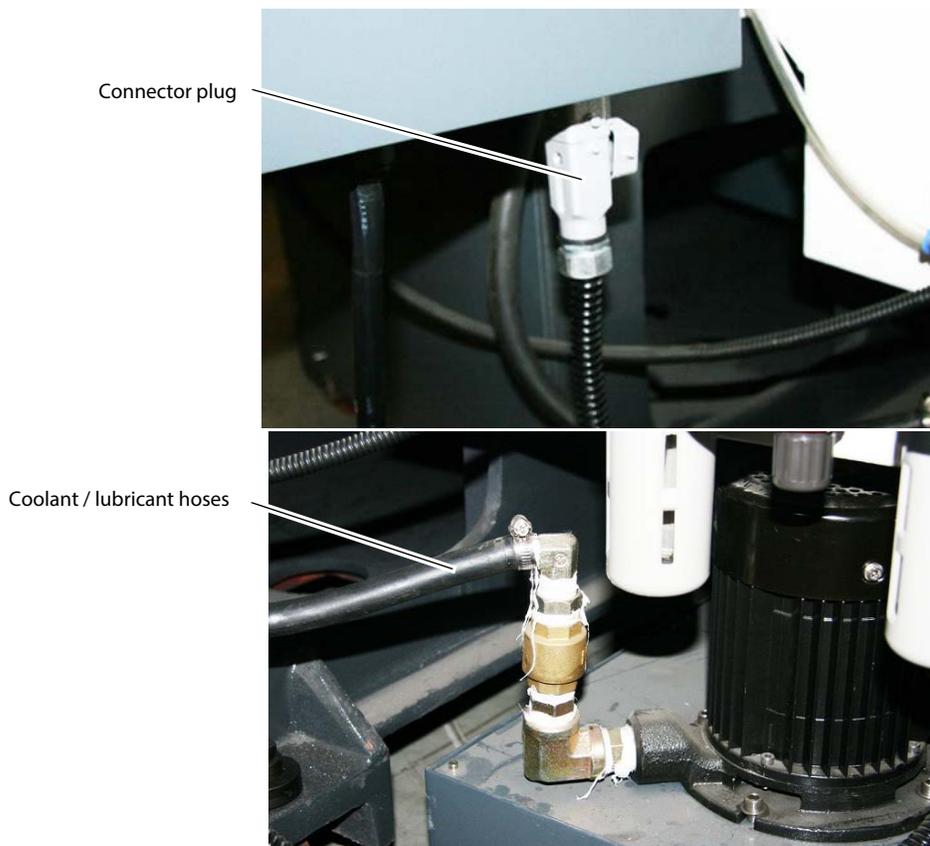


Img.3-6: Mounting the coolant / lubricant tank

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

3.5.2 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.3-7: Connector plug

3.5.3 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.



3.5.4 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.

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.

3.5.5 Current in the Protective Earth Ground Wire

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

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

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

Type AC should never be used in converters.

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

Type B must be used for 3-phase converters.



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

Line systems

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

Prohibited operation

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

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

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

Permissible line supplies

Operation on TN and TT line systems

TN line system

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

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

TT system

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

3.5.6 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.3-8: Compressed air maintenance unit

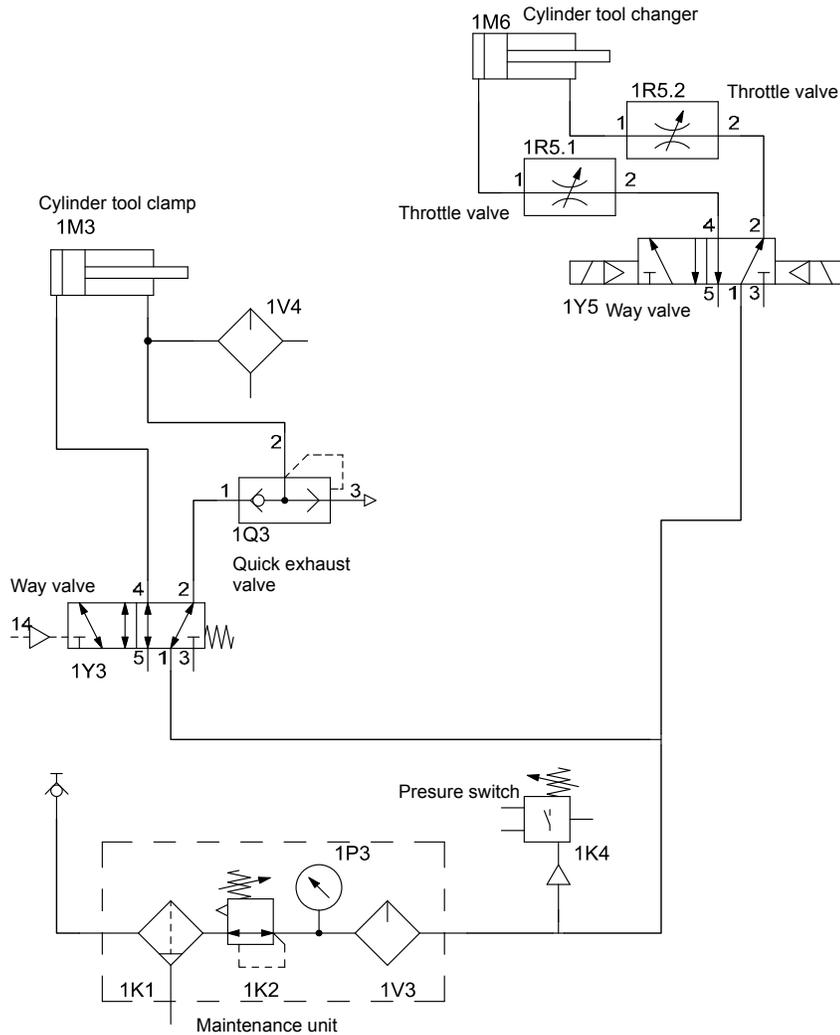


ATTENTION!

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



Img. 3-9: Pneumatic drawing



3.6 First commissioning

3.6.1 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 85



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 Grizzly Industrial, Inc. 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.



3.7 Refill central lubrication system

The CNC-machine is equipped with a central lubrication system.

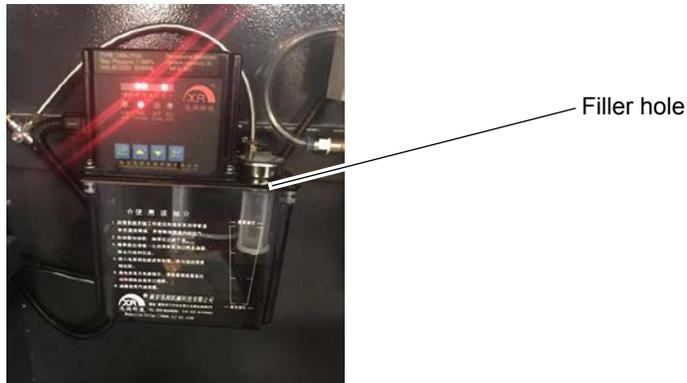
INFORMATION

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 85



Img. 3-10: Central lubrication system

Type:	DRB 215A
Pressure	2.5 MPa
Power	15 watt
Connection	220V ~ 50Hz
Filling capacity	2 liter
Flow rate	0.08 ml / cycle



Setting the jumpers

A. Interval time

Enter the status of adjustment first time, interval time is shown by the luminotron. The adjustable range: 1~255minutes. Press ▲ or ▼ to change the parameter value.

Press the "Enter" button, enter the adjustment of runtime.

For example:

1+4+16=21min
128 64 32 16 8 4 2 1

1+2+16+32+64=115min
128 64 32 16 8 4 2 1

B. Runtime

After enter the adjustment of runtime, runtime flashes by the luminotron (the run lamp lights). The adjustable range:1~255 seconds. Press ▲ or ▼ to change the parameter value. All the parameter values are stored by pressing the "Enter" button. The WGKX-4(5A) controller starts to run, but the WGKX-5(6A) controller switch to enter the adjustment of operation mode.

For example:

1+2+4+8=15s
128 64 32 16 8 4 2 1

2+16+32+128=178s
128 64 32 16 8 4 2 1

INFORMATION

The factory setting is 120 minutes as a break between lubrication cycles and 30 seconds for the duration of the lubricating cycle.



3.8 Functional test and controls

Rotation coolant pump

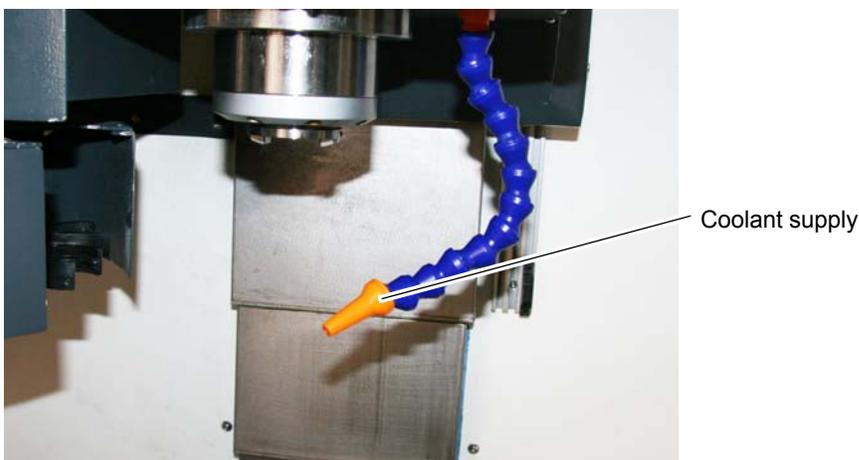
- Check the direction of rotation of the CNC-machine. There must be a right-handed rotating field. If the direction of rotation is wrong, exchange two of the three phase conductors. At this, the correct direction of rotation only refers to the drive of the cooling lubricant pumps.

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.
- Check if cooling agent flows.



Img.3-12: 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.



3.8.1 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.



3.8.2 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"!



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.

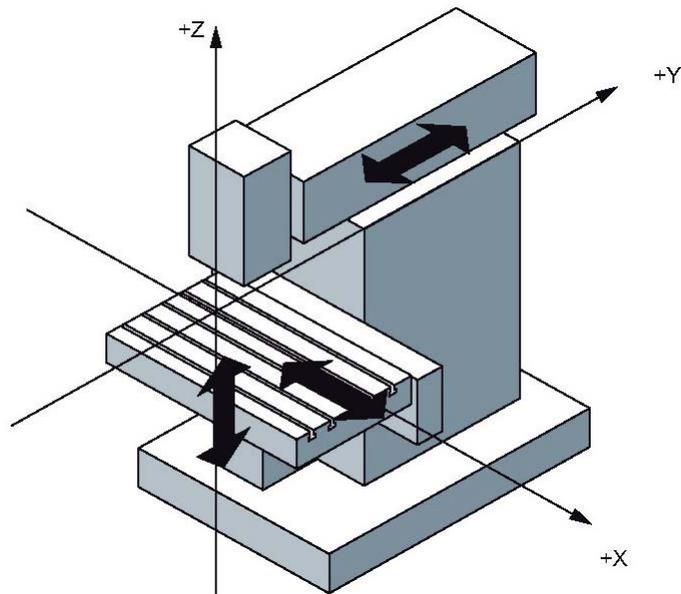
Turns from +X to +Y are created in +Z direction, which result in a right-handed screw.

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.

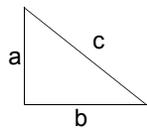
In a rectangular triangle it applies:

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 B.C.) 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:

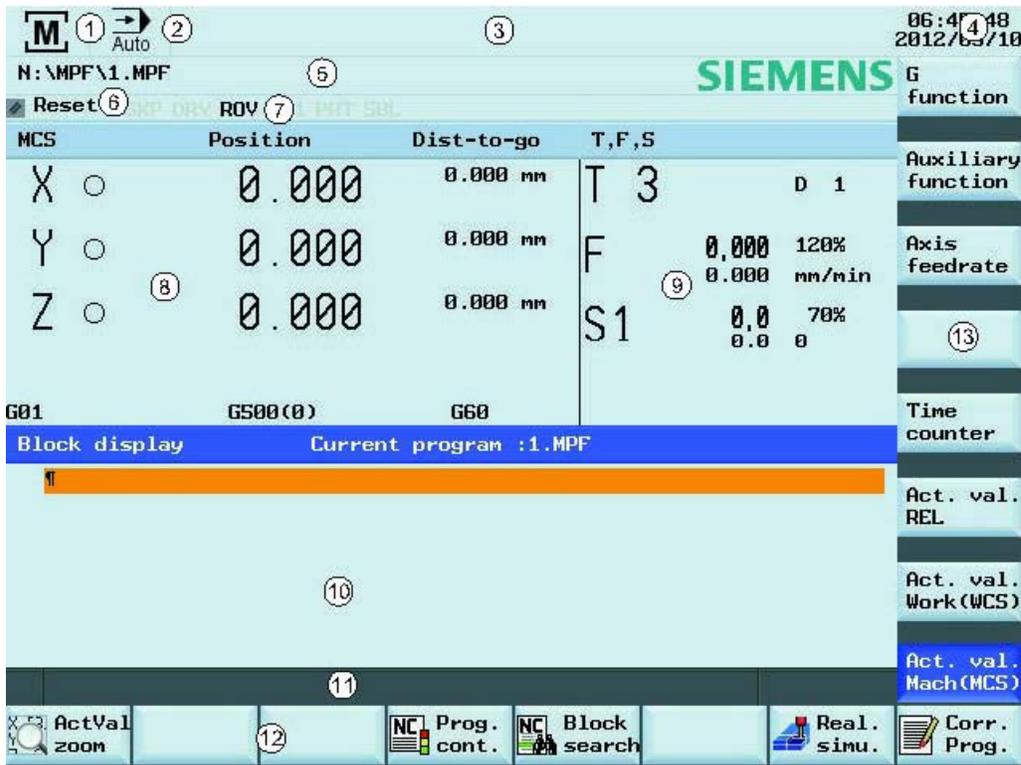
$\text{opposite leg} = \text{adjacent leg} \times \tan \alpha$



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

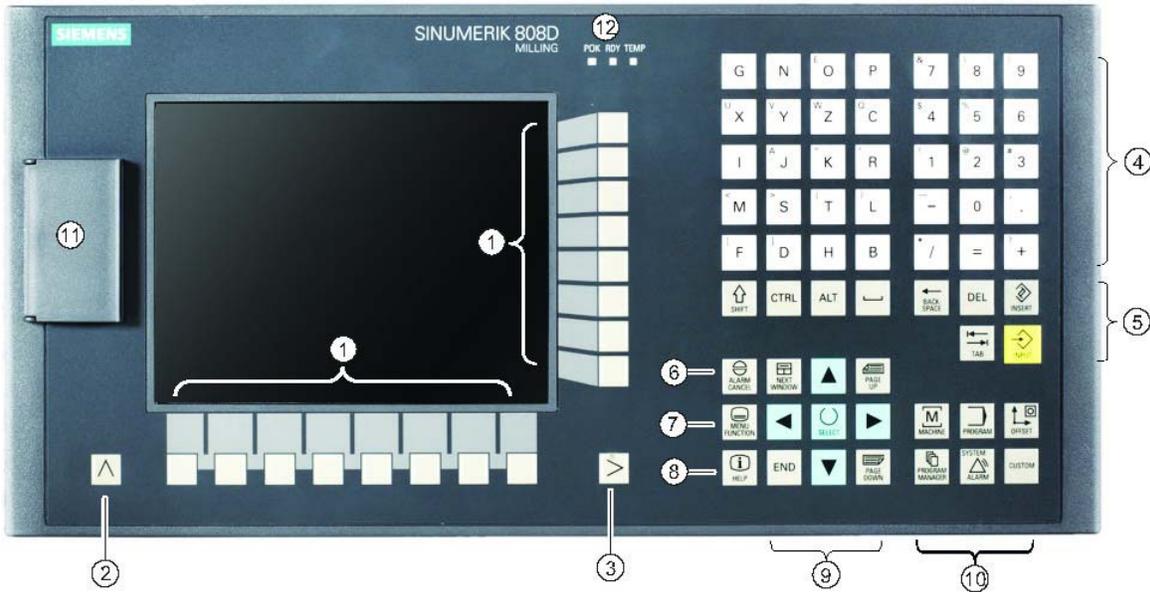
- ⑧ Actual value window
- ⑨ T, F, S window
- ⑩ Operating window with program block display

Tip and softkey area

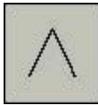
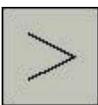
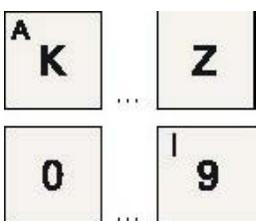
- ⑪ Information line
- ⑫ Horizontal softkey bar
- ⑬ Vertical softkey bar



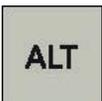
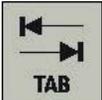
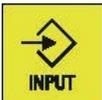
5.2 Elements on the PPU (Panel Processing Unit) front



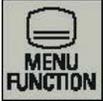
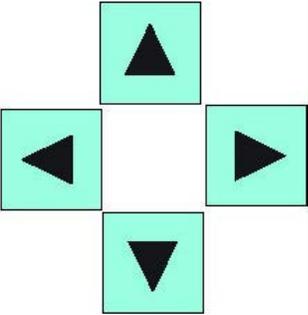
Img. 5-2: Panel Processing Unit

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 <ul style="list-style-type: none"> • Indents the cursor by several characters. • Toggles between the input field and the selected program name.
		Input key <ul style="list-style-type: none"> • 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



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 <ul style="list-style-type: none"> • 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
	 <p>Opens the "Machine" operating area</p>
	 <p>Opens the "Program" operating area</p>
	 <p>Opens the "Offset" operating area</p>
	 <p>Opens the "Program" operating area</p>
	 <p>Pressing this key opens the "Alarm" operating area. Holding down <SHIFT> while you press this key opens the "System" operating area.</p>
	 <p>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.</p>
⑪	<p>USB interface</p> <p>Connects to a USB device</p> <p>Examples:</p> <ul style="list-style-type: none"> • 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	<p>Status LEDs</p>  <p>LED "POK" Lights up green: The power supply for the CNC is switched on.</p> <p>LED "RDY" Lights up green: The CNC is ready for operation.</p> <p>LED "TEMP" Unlit: The CNC temperature is within the specified range. Lights up orange: The CNC temperature is out of range.</p>



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



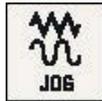
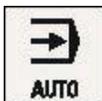
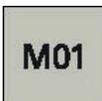
5.3 Elements on the MCP



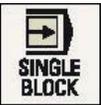
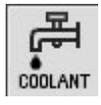
Img. 5-3: Machine Control Panel

Elements on the MCP	
Element	Function
	<p>Drive voltage Push-button with indicator light</p> <ul style="list-style-type: none"> Indicator light ON, drive voltage activated Indicator light OFF, drive voltage deactivated
	<p>Manual tool change Enables or disables the manual tool change. The manual tool change is only possible with the sliding door open.</p>

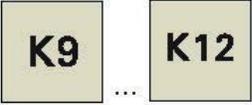
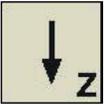


Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
①		<p><EMERGENCY STOP button></p> <p>Activate the button in situations where</p> <ul style="list-style-type: none"> • life is at risk. • there is the danger of a machine or workpiece being damaged. <p>All drives will be stopped with the greatest possible braking torque.</p>
②		<p>Handwheel key (with an LED status indicator)</p> <p>Controls the axis movement with external handwheels.</p>
③		<p>Tool number display</p> <p>Displays the current tool number</p>
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)	
		<p>Program test key</p> <p>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.</p>
		<p>Conditional stop key</p> <p>Stops the program at every block in which miscellaneous function M01 is programmed.</p>

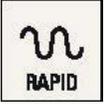
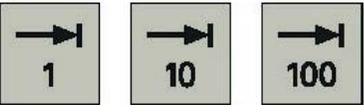
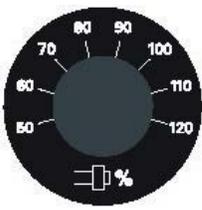


Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
		Rapid override key Adjusts axis feedrate override
		Single block key Activates single block execution mode
⑥	User-defined keys (all with LED status indicators)	
		Lamp control key Pressing 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 off: 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.

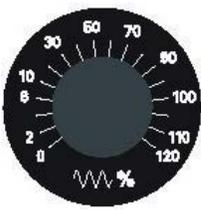


Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
		<p>Forward rotation of the chip conveyor (active only in JOG mode)</p> <p>Pressing this key in any operating mode starts the forward rotation of the chip conveyor.</p> <p>LED on: The chip conveyor starts forward rotation. LED off: The chip conveyor stops rotation.</p> <p>INFORMATION </p> <p>The machine does not have a chip conveyor. Therefore, the key has no function.</p>
		<p>Reverse rotation of the chip conveyor (active only in JOG mode)</p> <p>Keeping pressing this key in any operating mode rotates the chip conveyor in reverse order.</p> <p>Releasing the key changes the chip conveyor to the previous forward rotation or stop state.</p> <p>LED on: The chip conveyor starts reverse rotation. LED off: The chip conveyor stops reverse rotation.</p> <p>INFORMATION </p> <p>The machine does not have a chip conveyor. Therefore, the key has no function.</p>
		<p>User-defined keys</p> <p>INFORMATION </p> <p>The <K9> button : Mag go to spindle The <K10> button : Mag back home The <K11> button is associated with the manual tool change.</p>
	Axis traversing keys	
		<p>X axis key</p> <p>Traverses the X axis in the positive direction.</p>
		<p>X axis key</p> <p>Traverses the X axis in the negative direction.</p>
		<p>Z axis key</p> <p>Traverses the Z axis in the negative direction.</p>
		<p>Z axis key</p> <p>Traverses the Z axis in the positive direction.</p>



Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
	Element	Description
		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.
⑨	Program state keys	



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

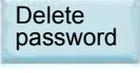
Step 1

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

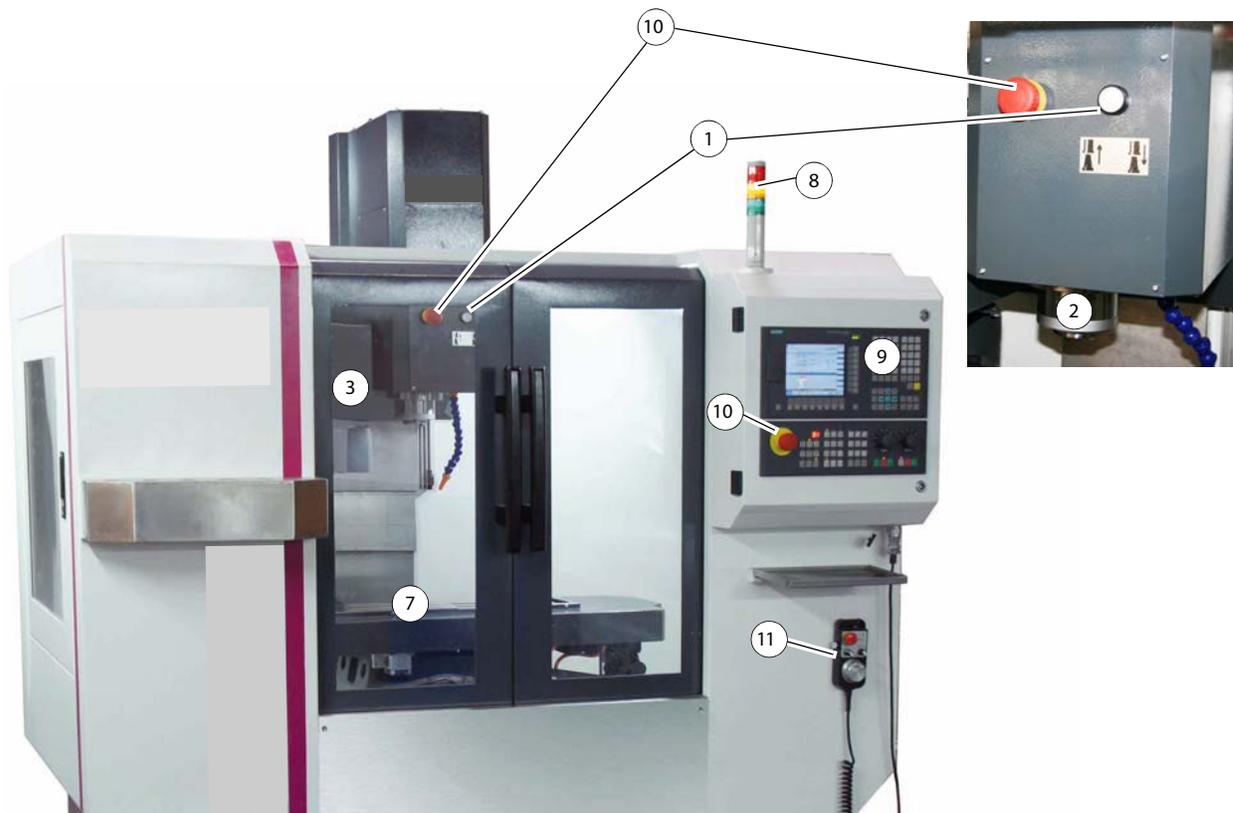
Commission the CNC-machine only under the following conditions:

- 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.

☞ "Additional Safety" on page 8

6.2 Control and indicating elements



Img.6-1: Controls

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 15
- 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

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

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

After turning on, the machine must first be referenced. Without existing reference points (machine zero points) you cannot start and run programs in the control.

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

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

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

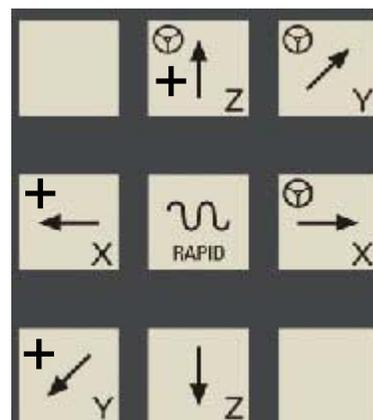


As long as the axes are not referenced, the symbol (circle) between the axis and the corresponding value is displayed.

MCS		Reference point	
X	○	0.000	mm
Y	○	0.000	mm
Z	○	0.000	mm

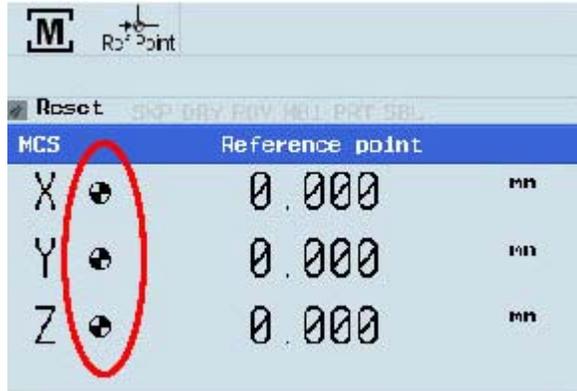
→ The axes are referenced with the "axis traversing keys."

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

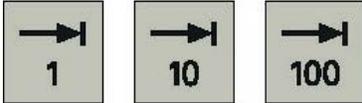


→ Traverse each axis to the machine zero point until the referenced symbol is shown on the respective axis.

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

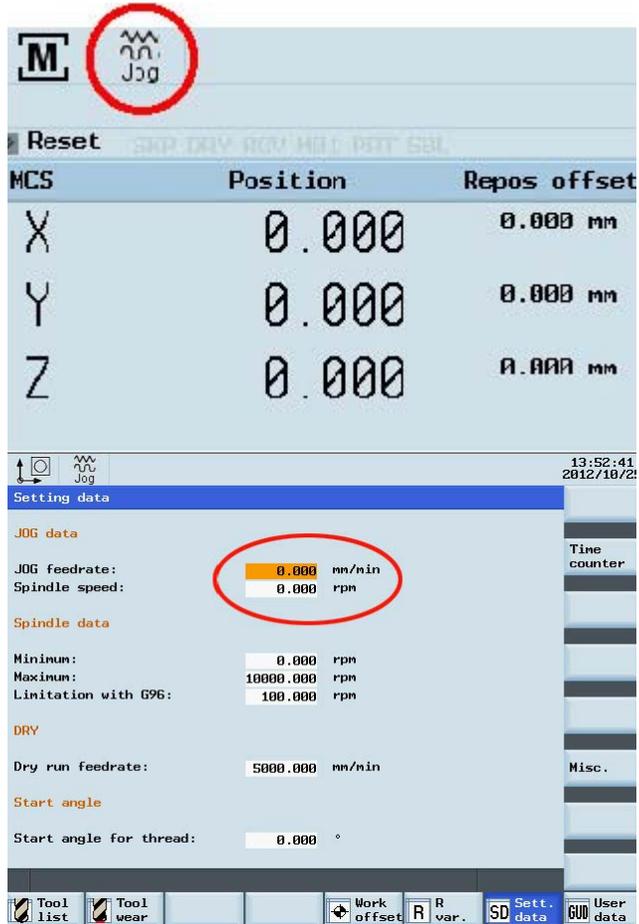


After returning to the "JOG" mode, the axes can be moved manually. Use the Increment button to specify the increment, or press the <JOG> button again to stop the increment again.



→ If possible, read the upcoming messages on the display, remove the error messages, such as e.g. insufficient compressed air,

→ Default values for example should apply for the feed rate in the "JOG" mode, be entered on the side of the standard values. Press the button <OFFSET> to get to the settings.



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



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 machine 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.



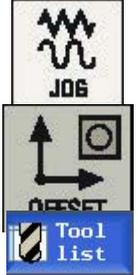
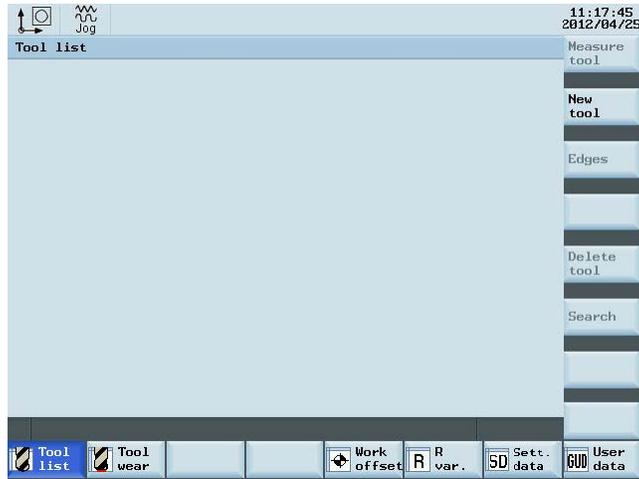
6.5.4 Inserting a tool

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 Clamping workpiece and setting workpiece datum

ATTENTION!

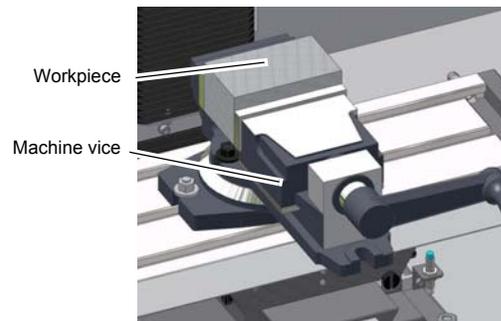
The workpiece is always to be fixed by a machine vice, jaw chuck or by another appropriate clamping tool such as for the clamping claws.

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

WARNING!

Risk of injury caused by workpieces flying off.

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



Img.6-3: Clamping the workpiece



6.5.6 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.



Img.6-4: Spindle head

6.5.7 Turning off the CNC- machine

→ Push the EMERGENCY STOP push button.

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



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 "Manual mode" of the operating instructions of "SINUMERIK 808D".

Automatic mode

Refer to "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 an d 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-machine:

- 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.

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.



6.12 CNC Rotary Table cutting speed chart

The direct instruction system with the F-code is used for instructions of speeds of CNC rotary table (moving in degrees per minute), as follows.

Example:

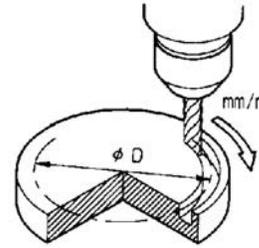
F100 = 100 °/min = 0.28 R.P.M

Relation between a feed of tool and a circular arc dia.

D at a time of cutting is listed in the following chart.

How to use the chart:

When an outer periphery of work D = 160 mm is to be cut by end milling at a speed of 110 mm/min. F value obtained as f = 80 from 111 mm / min. corresponding to D = 160 mm at the top-most line.



F	RPM	30	40	50	60	80	100	120	140	160	180	200	230	250	300	400	500	ØD	F
F 10	0.03	3	4	5	6	8	9	11	13	15	17	19	22	24	28	38	47	F 10	
F 20	0.06	6	8	9	11	15	19	23	26	30	34	38	43	47	57	75	94	F 20	
F 30	0.08	8	10	13	15	20	25	30	35	40	45	50	58	63	75	100	125	F 30	
F 40	0.11	10	14	17	21	28	35	42	48	55	62	69	80	87	104	138	173	F 40	
F 50	0.14	13	18	22	26	35	44	53	62	70	79	88	101	110	132	176	220	F 50	
F 60	0.16	15	20	25	30	40	50	60	70	80	91	101	116	126	151	201	252	F 60	
F 70	0.19	18	24	30	36	48	60	72	84	96	107	119	137	149	179	239	299	F 70	
F 80	0.22	21	28	35	41	55	69	83	97	111	124	138	159	173	207	276	346	F 80	
F 90	0.25	24	31	39	47	63	79	94	110	126	141	157	181	196	236	314	393	F 90	
F100	0.28	26	35	44	53	70	88	106	123	141	158	176	202	220	264	352	440	F100	
F110	0.31	29	39	49	58	78	97	117	136	156	175	195	224	244	292	390	487	F110	
F120	0.33	31	41	52	62	83	104	124	145	166	187	207	239	259	311	415	519	F120	
F130	0.36	34	45	57	68	90	113	136	158	181	204	226	260	283	339	452	566	F130	
F140	0.39	37	49	61	74	98	123	147	172	196	221	245	282	306	368	490	613	F140	
F150	0.42	40	53	66	79	106	132	158	185	211	237	264	303	330	396	528	660	F150	
F160	0.44	41	55	69	83	111	138	166	193	221	249	276	318	346	415	553	691	F160	
F170	0.47	44	59	74	89	118	148	177	207	236	266	295	340	369	443	591	739	F170	
F180	0.50	47	63	79	94	126	157	189	220	251	283	314	361	393	471	628	786	F180	
F190	0.53	50	67	83	100	133	167	200	233	266	300	333	383	416	500	666	833	F190	
F200	0.55	52	69	86	104	138	173	207	242	276	311	346	397	432	518	691	864	F200	
F210	0.58	55	73	91	109	146	182	219	255	292	328	364	419	456	547	729	911	F210	
F220	0.61	57	77	96	115	153	192	230	268	307	345	383	441	479	575	766	958	F220	
F230	0.64	60	80	101	121	161	201	241	282	322	362	402	463	503	603	804	1006	F230	
F240	0.67	63	84	105	126	168	211	253	295	337	379	421	484	526	632	842	1053	F240	
F250	0.69	65	87	108	130	173	217	260	304	347	390	434	499	542	650	867	1084	F250	
F260	0.72	68	90	113	136	181	226	271	317	362	407	452	520	566	679	905	1131	F260	
F270	0.75	71	94	118	141	188	236	283	330	377	424	471	542	589	707	942	1178	F270	
F280	0.77	73	97	121	145	194	242	290	339	387	435	484	556	605	726	968	1210	F280	
F290	0.81	76	102	127	153	204	255	305	356	407	458	509	585	636	764	1018	1273	F290	
F300	0.83	78	104	130	156	209	261	313	365	417	469	522	600	652	782	1043	1304	F300	
F310	0.86	81	108	135	162	216	270	324	378	432	486	540	624	676	811	1081	1351	F310	
F320	0.90	85	113	141	170	226	283	339	396	452	509	565	650	707	848	1131	1414	F320	
F330	0.92	87	116	145	173	231	289	347	405	462	520	578	665	723	867	1156	1445	F330	
F340	0.94	89	118	148	177	236	295	354	413	472	531	591	679	738	886	1181	1476	F340	
F350	0.97	91	122	152	183	244	305	366	427	488	548	609	701	762	914	1219	1524	F350	
F360	1.00	94	126	157	189	251	314	377	440	503	566	628	723	786	943	1257	1571	F360	
F370	1.03	97	129	162	194	259	324	388	458	518	582	647	744	809	971	1294	1618	F370	
F380	1.06	100	133	167	200	266	333	400	466	533	599	666	766	833	999	1332	1665	F380	
F390	1.08	102	136	170	204	271	339	407	475	543	611	679	780	848	1018	1357	1697	F390	
F400	1.11	105	139	174	209	279	349	418	488	558	628	697	802	872	1046	1395	1744	F400	
F410	1.14	107	143	179	215	286	358	430	501	573	645	716	824	895	1074	1432	1791	F410	
F420	1.17	110	147	184	221	294	368	441	515	588	662	735	845	919	1103	1470	1838	F420	
F430	1.19	112	150	187	224	299	374	449	523	598	673	743	860	935	1121	1495	1869	F430	
F440	1.22	115	153	192	230	307	383	460	537	613	690	767	882	958	1150	1533	1917	F440	
F450	1.25	118	157	196	236	314	398	471	550	628	707	785	908	982	1178	1571	1964	F450	
F460	1.28	121	161	201	241	322	402	483	563	643	724	804	925	1005	1206	1608	2011	F460	
F470	1.31	123	165	206	247	329	412	494	576	658	741	823	946	1029	1235	1646	2058	F470	
F480	1.33	125	167	209	251	334	418	501	585	668	752	836	961	1045	1253	1671	2089	F480	
F490	1.36	128	171	214	256	342	427	513	598	684	769	855	983	1068	1282	1709	2187	F490	
F500	1.39	131	175	218	262	349	437	524	611	699	786	873	1004	1092	1310	1747	2184	F500	
		30	40	50	60	80	100	120	140	160	180	200	230	250	300	400	500		

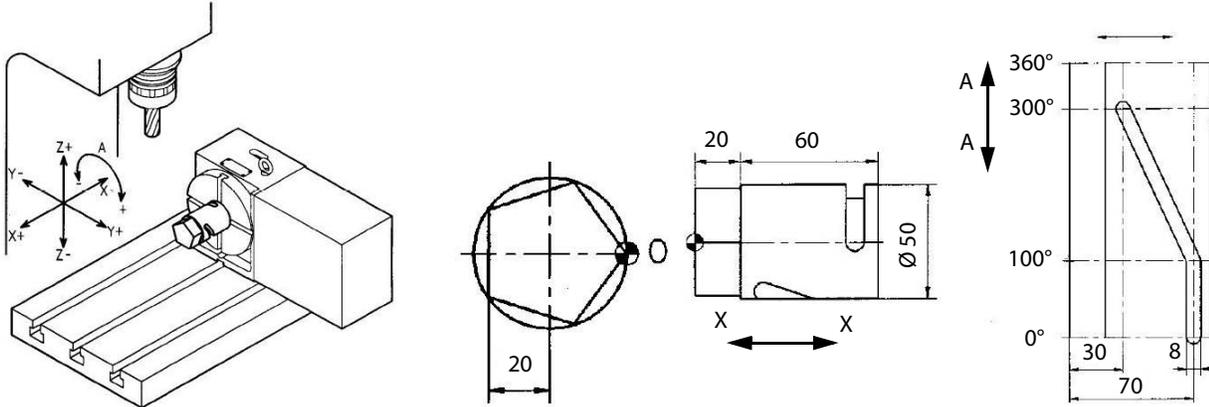


6.12.1 Example of programming

The workpiece is 50mm in diameter.

1 Cutting 5 face by 20 mm at end of the workpiece.

2 Helix cutting.



NC program 1	Description	NC program 2	Description
N1		M10	
T1 M06	End mill \varnothing 25mm	G01 Y-12. F100	Coolant off
M11	Rotary table unclamp	Y-25. F2000	
G90 G00 A36. M10	Rotary +36°	Z5. M09	Spindle stop
G90 G45 G00 X7.5 Y25	Rotary table clamp	M11	
G43 Z100. S500 H1		G00 A0.	End mill \varnothing 8mm
M03		G90 G00 Z100. M05	
Z5		G91 G28 Z0. G49	
G01 Z-5. F100 M08		N2	
Y-12.		T2 M06	
Y-25. F2000		G90 G54 G00 X70. Y0. G43	
M11		Z100. S3000 H2	
G00 A108. M10		M03	
G01 Y12. F100		G01 Z-6. F100 M08	
Y25. F2000		A100.	
M11		X30. A300.	
G100 A180. M10		G90 G00 Z5. M05	
G01 Y-12. F100		M09	
Y-25. F2000		M10	
M11		G91 G28 Z0. G49	
G00 A252. M10		M30	
G01 Y12. F100			



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	Tool magazine reaching to the spindle position
M22	Tool magazine reaching to the original position
M23	
M24	
M25	Clamp workpiece
M26	Loosen workpiece
M27	
M28	
M29	
M30	

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



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

No.	Function
M99	



7.2 G functions to PAL

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



8 Notes, messages and error messages

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

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

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

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

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



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



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



9 SINUMERIK 808 D

The Sinumerik 808D complete documentation consists of the manuals listed below, which are at the end of this manual or accompanying the machine separately.

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

- Programming and operating manual (milling)
 - Part 1 Milling operation (808D_OPM_Operation_0512....pdf)
 - Part 2 Programming milling (808D_OPM_Programming_Siemens_0512....pdf)
 - Part 3 Programming milling in ISO code (808D_OPM_Programming_ISO_0512_....pdf)
- Programming and operating manual (milling)
6FC5398-4DP10-0AA1 - (808D_ADVANCED_OPM_0114.....pdf)
- Programming milling in ISO code
Programmieren Drehen und Fräsen im ISO Code
6FC5398-0DP40-0AA0 - (SINUMERIK_808D_ADVANCED.....pdf)

Manuals that are required for the maintenance and repair personnel.

- Service (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 subroutines (SINUMERIK_808D_PLC_Subroutines_ pdf)
Describes e.g. making a connection to the RS232 interface.
- Online help for programming and operation (milling)

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

- Mechanical installation manual
- Electrical installation manual
- Parameter Manual

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

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

For any questions regarding the CNC control, please contact:

Siemens AG, A&D techsupport

Phone (+49) 0180 50 50 222

mailto: techsupport@ad.siemens.de

Siemens AG Hotline, Helpline

Phone (+49) 0180 50 50 111



10 SINUMERIK 808 D Advanced

The Sinumerik 808D complete documentation consists of the manuals listed below, which are at the end of this manual or accompanying the machine separately.

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

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

Manuals that are required for the maintenance and repair personnel.

- Service (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 subroutines (SINUMERIK_808D_PLC_Subroutines_. pdf)
Describes e.g. making a connection to the RS232 interface.
- Online help for programming and operation (milling)

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

- Mechanical installation manual
- Electrical installation manual
- Parameter Manual

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

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

For any questions regarding the CNC control, please contact:

Siemens AG, A&D techsupport

Phone (+49) 0180 50 50 222

mailto: techsupport@ad.siemens.de

Siemens AG Hotline, Helpline

Phone (+49) 0180 50 50 111



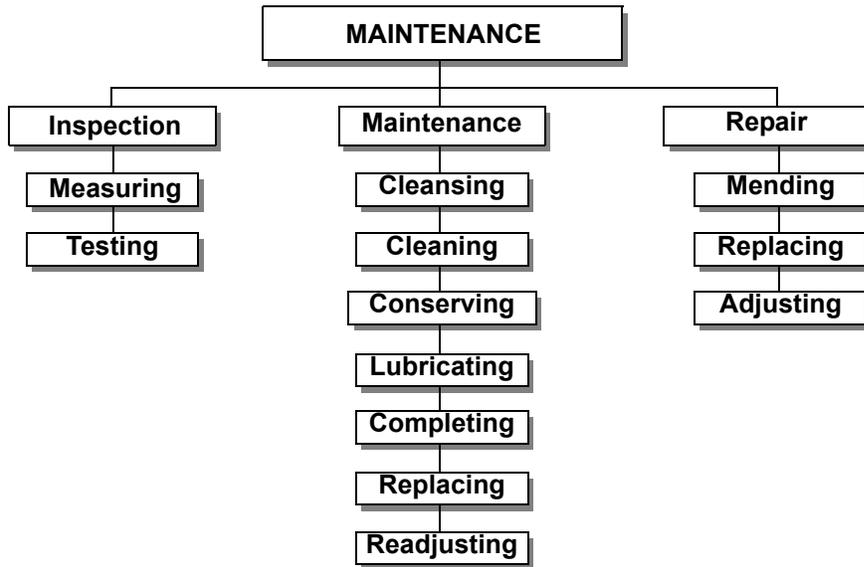
11 Maintenance

In this chapter you will find important information about

- Inspection
- Maintenance
- Repair

of the CNC-machine.

The diagram below shows you which tasks belongs to which term.



Img.11-1: Maintenance

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 spilt 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.



11.1 Operating material

11.1.1 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)

11.1.2 Cooling lubricants

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

INFORMATION

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

The company Grizzly Industrial, Inc. 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, Grizzly Industrial, Inc. 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 a safety data sheet. The 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).

- The manufacturer should be able to submit a certificate concerning skin-tolerance.



- The mineral content 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.
- Re-emulsifiable and not sticking.
- It should not attack the varnish of the machine.
- It should not attack any machine elements (metals, elastomeres).
- Low foaming behaviour of the emulsion.
- It should be as disperse as possible in order to avoid clocking-up at the needle slot screen.

11.2 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.

11.2.1 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 19

Attach a warning sign.

11.2.2 Restarting

Before restarting run a safety check.

☞ "Safety check" on page 17

WARNING!

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



11.3 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 17	
	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	
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 83	
	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 36	
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.	



Interval	Where?	What?	How?	Check ✓
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.	
Annually	Clutch Coupling Drive Z-, Y-, X-axis	Check Replacing	Check if coupling is worn and check coupling slack. If necessary, replace the coupling.	
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.	



Interval	Where?	What?	How?	Check ✓
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.	
60 months	Protective covers	Replacing the sight window	☞ "Cleaning and replacing of the polycarbonate windows" on page 81	

11.3.1 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.

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.



11.3.2 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.

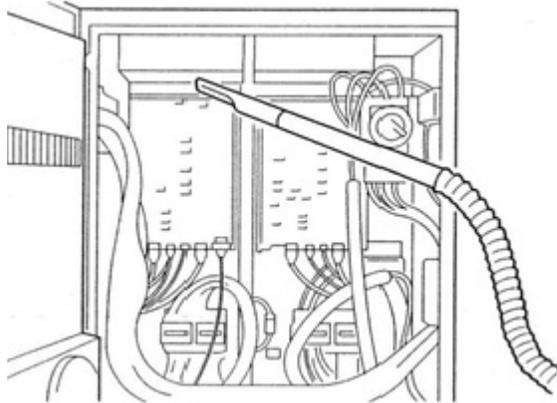
Accumulation of foreign particles on the printed circuit boards or other electronic components could cause machine malfunction.

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.

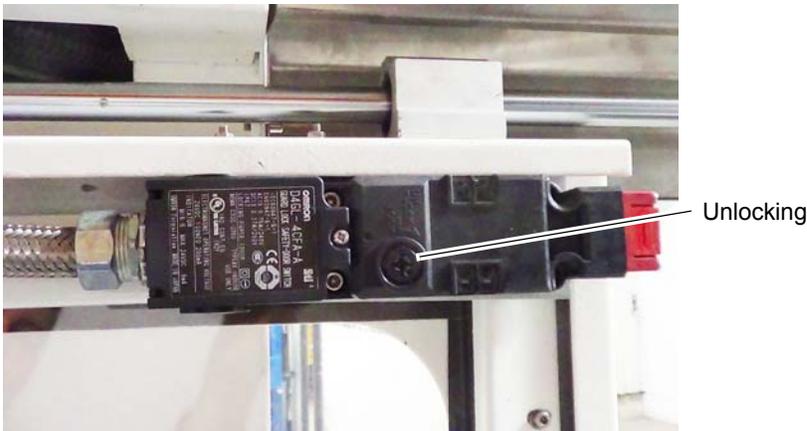
Never touch printed circuit boards or parts around the connector. Also avoid shock on these parts.

It is recommended to clean the electrical cabinet every 1000 operating hours.



11.4 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 17



Img. 11-2: Interlock switch sliding door

11.4.1 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.



11.5 Cooling lubricants and tanks

CAUTION!

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



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

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

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

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.



11.5.1 Inspection plan for water-mixed cooling lubricants

Company: No.: Date: used cooling lubricant			
size to be checked	Inspection methods	Inspection intervals	Procedure and comment
noticeable changes	Appearance, odour	daily	Find and rectify causes, e.g. skim off oil, check filter, ventilate cooling lubricant system
pH value	Laboratory techniques electrometric with pH meter 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.

Editor:

Signature:



11.5.2 Lubricants

Lubricant	Viscosity ISO VG					
Gear oil	VG 680	BP Energol GR-XP 680	SPARTAN EP 680	Mobilgear 636	Shell Omala 680	Meropa 680
	VG 460	BP Energol GR-XP 460	SPARTAN EP 460	Mobilgear 634	Shell Omala 460	Meropa 460
	VG 320	BP Energol GR-XP 320	SPARTAN EP 320	Mobilgear 632	Shell Omala 320	Meropa 320
	VG 220	BP Energol GR-XP 220	SPARTAN EP 220	Mobilgear 630	Shell Omala 220	Meropa 220
	VG 150	BP Energol GR-XP 150	SPARTAN EP 150	Mobilgear 629	Shell Omala 150	Meropa 150
	VG 100	BP Energol GR-XP 100	SPARTAN EP 100	Mobilgear 627	Shell Omala 100	Meropa 100
	VG 68	BP Energol GR-XP 68	SPARTAN EP 68	Mobilgear 626	Shell Omala 68	Meropa 68
	VG 46	BP Bartran 46	NUTO H 46 (HLP 46)	Mobil DTE 25	Shell Tellus S 46	Anubia EP 46
	VG 32	BP Bartran 32	NUTO H 32 (HLP 32)	Mobil DTE 24	Shell Tellus S 32	Anubia EP 32
Gear grease		BP Energrease PR-EP 00	FIBRAX EP 370 (Na-ver- seift)	Mobilux EP 004	Shell Alvania GL 00 (Li- verseift)	Marfak 00
Special greases, water resistant				Mobilux EP 0		
Bearing grease		BP Energrease LS 3	BEACON 3	Mobilux 3	Shell Alvania R 3 Alvania G 3	Multifak Premium 3



Brief Instructions for 808D Milling

Preparation



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

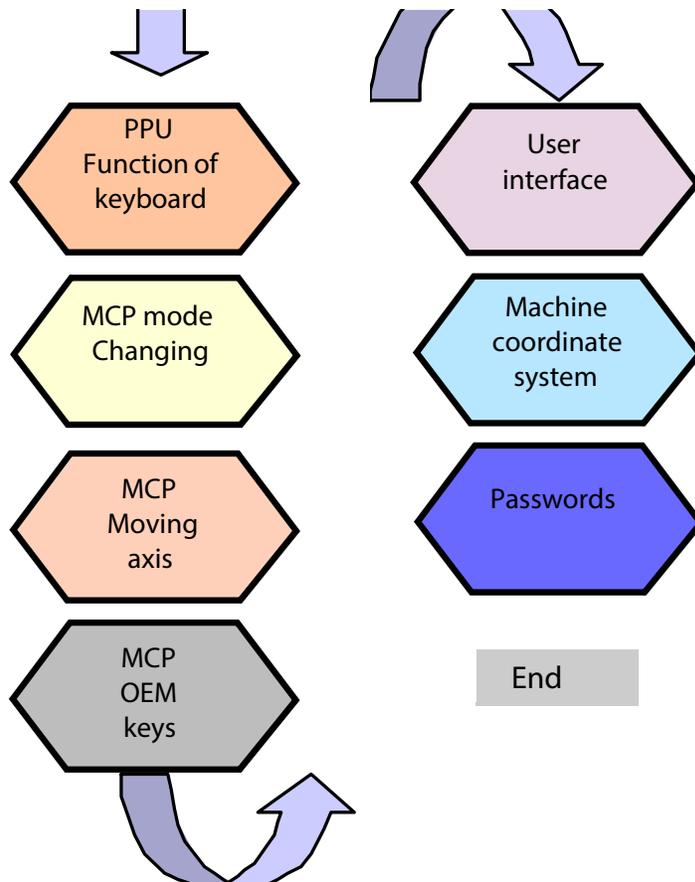
The information provided in this brief instruction contains merely general descriptions or characteristics of performance which in actual case of use do not always apply as described or which may change as a result of further development of the products.

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



Preparation

Basic Theory

PPU
Function of
keyboard

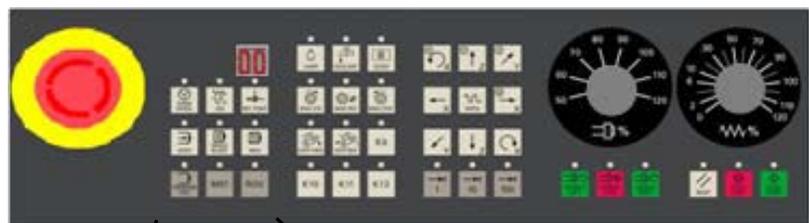


The 808D panel processing unit (PPU) is used to input data to the CNC and to navigate to operating areas of the system.

Menu navigation

Operating area navigation

MCP mode
Changing

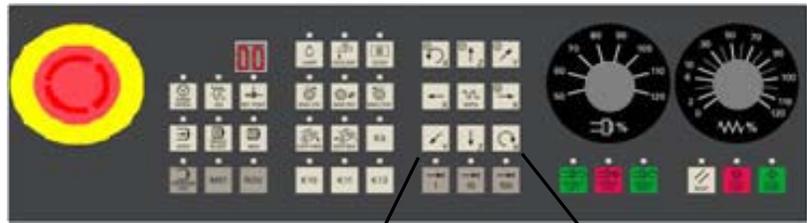


The 808D machine control panel (MCP) is used to select the machine operating mode :
JOG - MDA - AUTO

Mode navigation



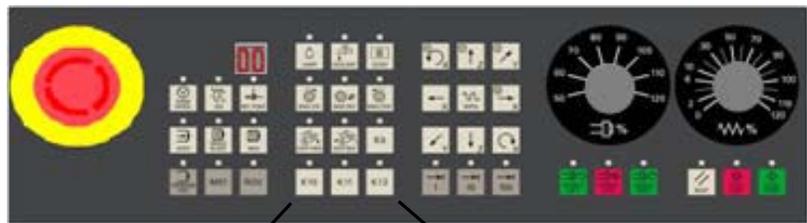
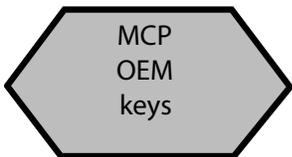
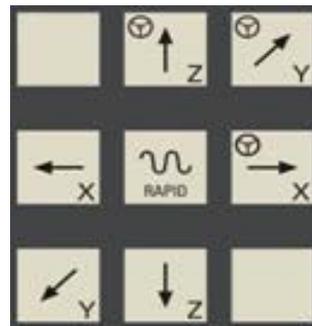
Preparation



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.



OEM keys

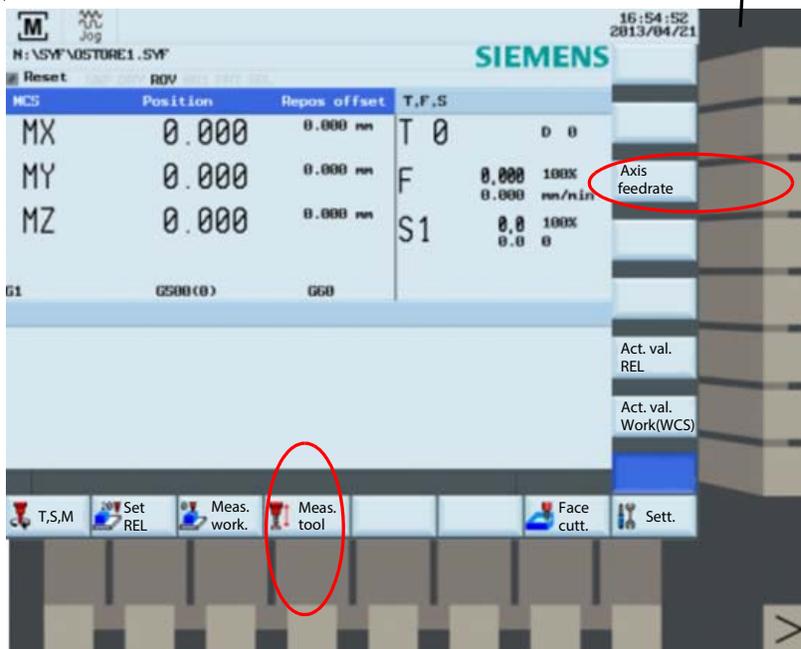
The 808D machine control panel (MCP) is used to control OEM machine functions.

The machine functions can be activated with the appropriate keys.



Preparation

User interface



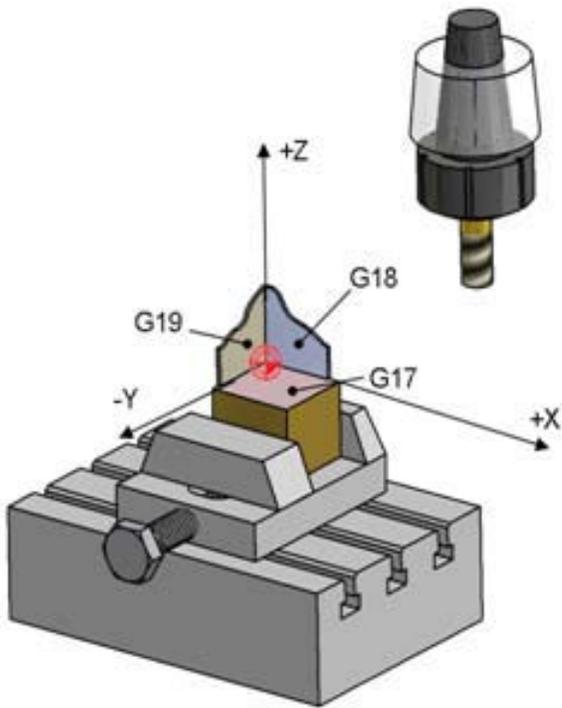
808D (PPU) has eight vertical softkeys (abbr. SKs) on the right of the screen. These SKs can be activated with the corresponding button (located on the right).

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



Preparation

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.



Preparation

SEQUENCE



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

Changing password

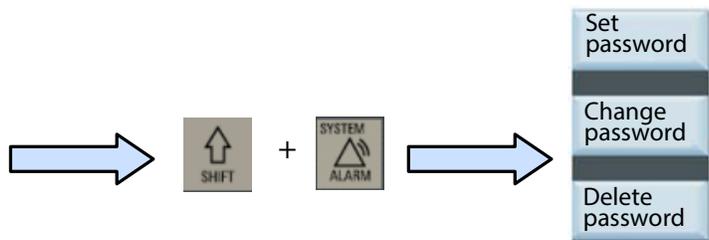
Customer's password = CUSTOMER
 Manufacturer's password = SUNRISE

Step 1



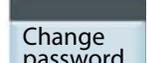
Usually the machine, operator does not need to change the password.

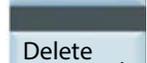
The service mode is opened with the appropriate key combination. In the service mode, the password can be activated and deactivated.



Step 2

- 

 Enter customer's or manufacturer's password
- 

 Change customer's or manufacturer's password
- 

 Delete customer's or manufacturer's password

End



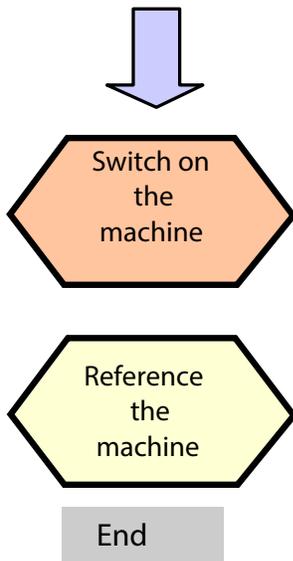
Switch on and referencing

Switch On and Referencing

Description

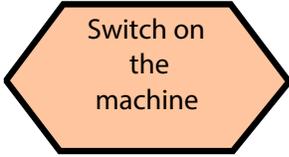
This unit describes how to switch the machine on and reference it.

Content



Switch on and referencing

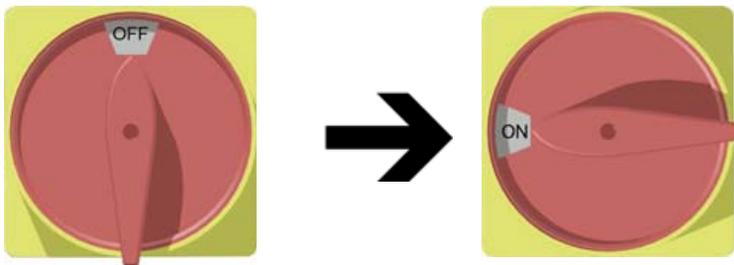
SEQUENCE



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!



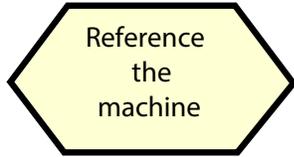
Release all the EMERGENCY STOP buttons on the machine!

End



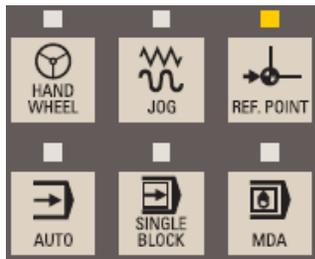
Switch on and referencing

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

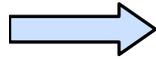
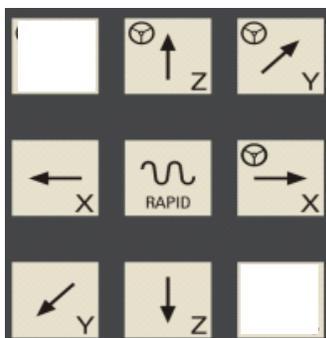


After power on, the machine will be in the reference point approach mode (default).

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.



Switch on and referencing

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.

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



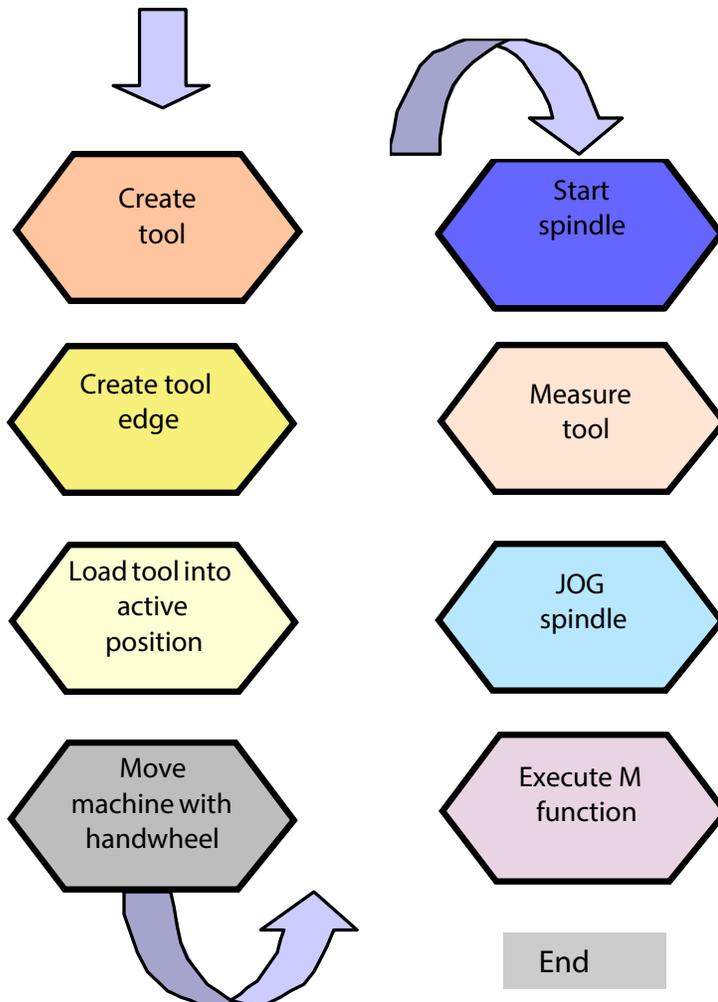
Tool setup

Tool Setup

Description

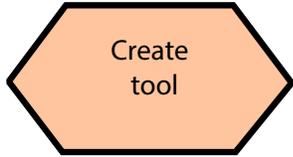
This unit describes how to create and set up tools.

Content



Tool setup

SEQUENCE



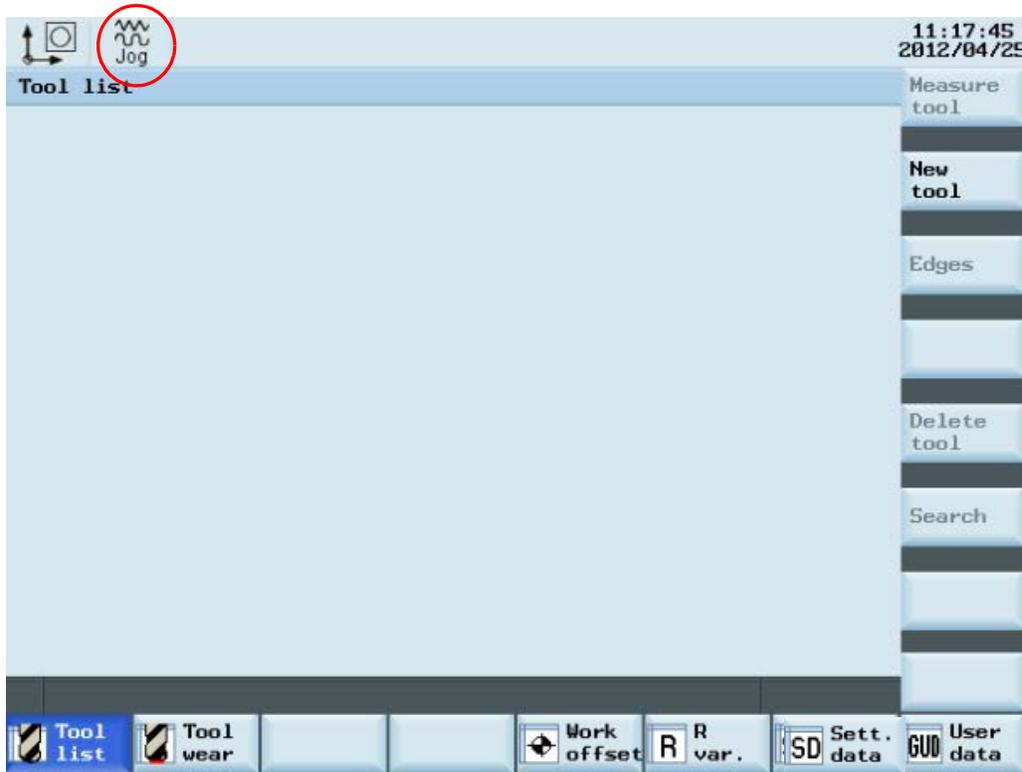
A tool must have been created and measured before executing the program.

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

Press "Offset" on the PPU.



Press the "Tool list" SK on the PPU.



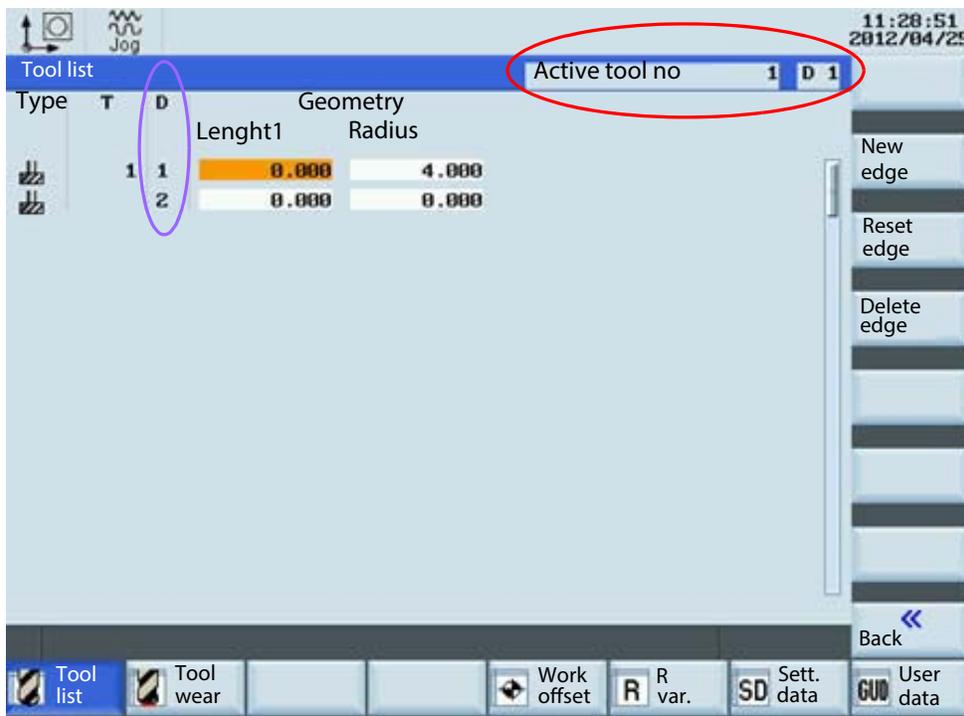
Tool setup

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!



Tool setup



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



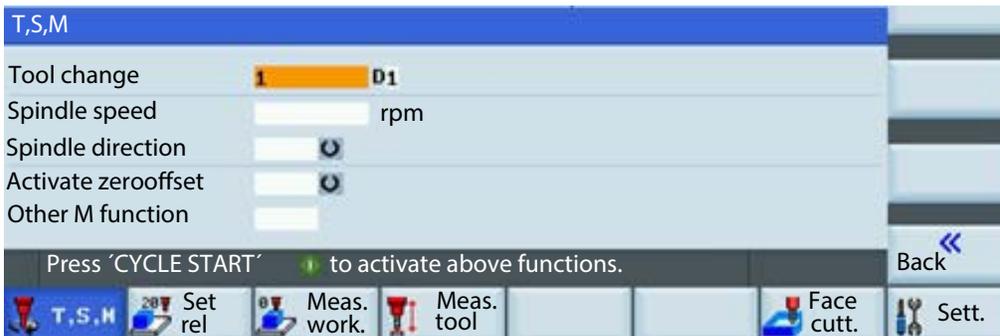
Press the "JOG" key on the MCP



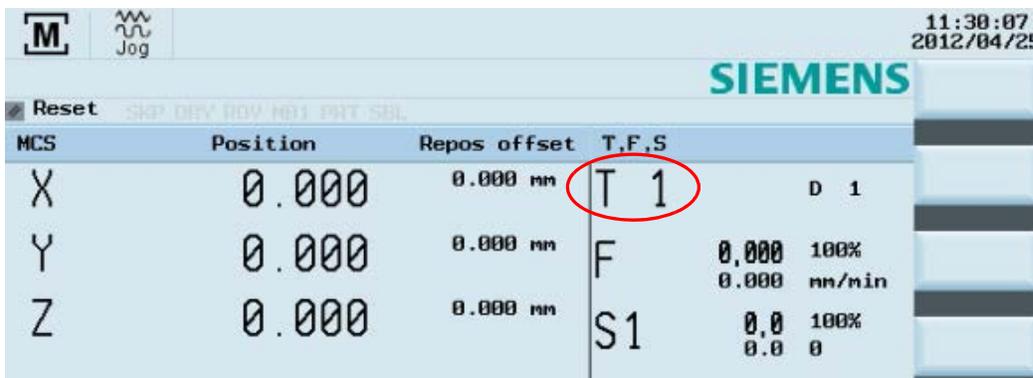
Press the "T.S.M" SK on the PPU



Enter tool number "1" in "T"



Press "CYCLE START" on the MCP



Press the "Back" SK on the PPU



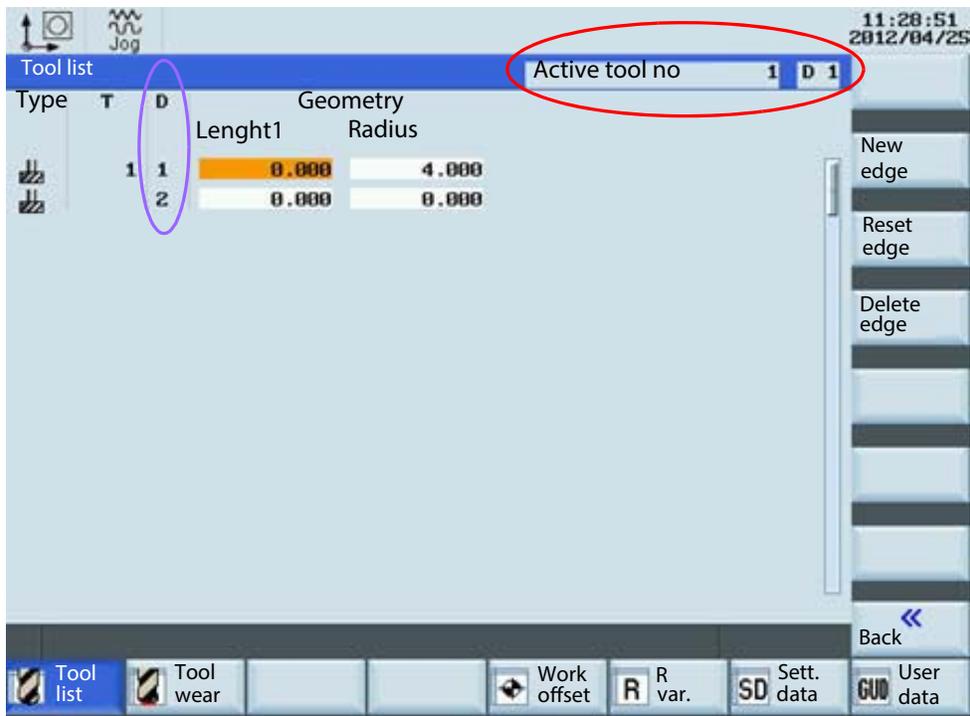
Tool setup

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!

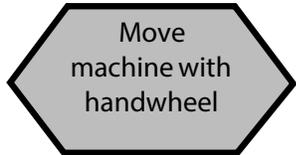


Tool setup

SEQUENCE

The tool are usually loaded manually into the spindle.

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



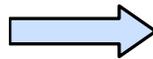
Make sure there is no obstruction when moving the tool to avoid a crash.

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

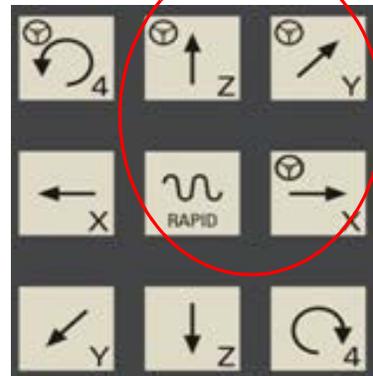
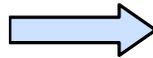
Press the "Machine" key on the PPU



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.

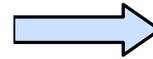


Tool setup

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"

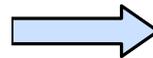


The handwheel increment is "0.100 mm"

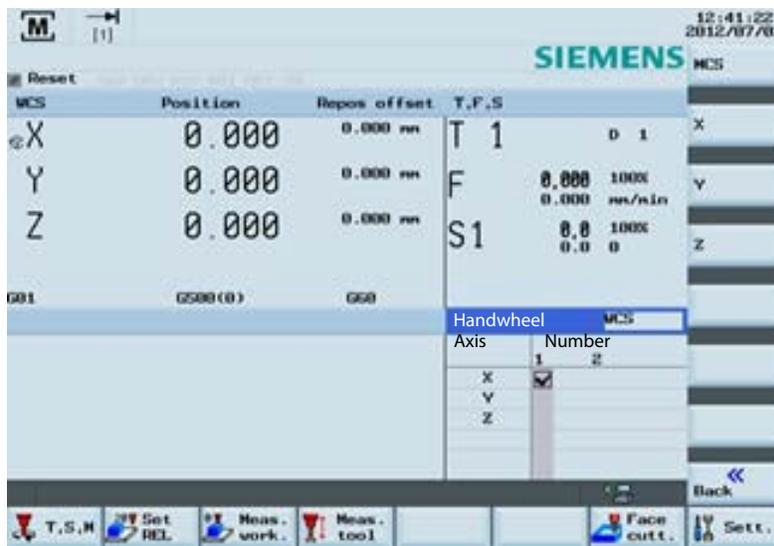


The selected axis can now be moved with the handwheel.

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 ✓



Tool setup

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



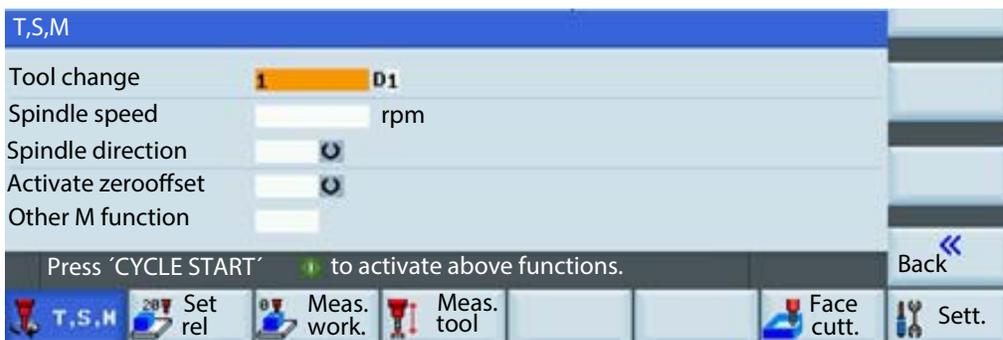
Press the "T.S.M" SK on the PPU



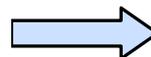
Enter "500" at "Spindle speed"



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

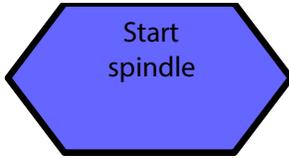


Press the "CYCLE START" key on the MCP



Tool setup

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



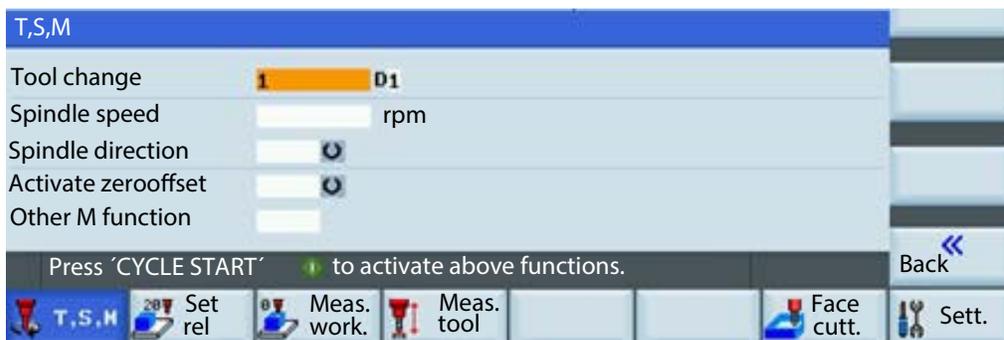
Press the "T.S.M" SK on the PPU



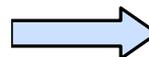
Enter "500" at "Spindle speed"



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



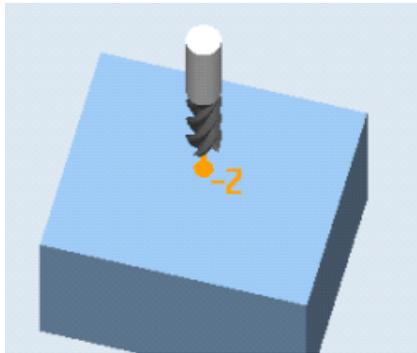
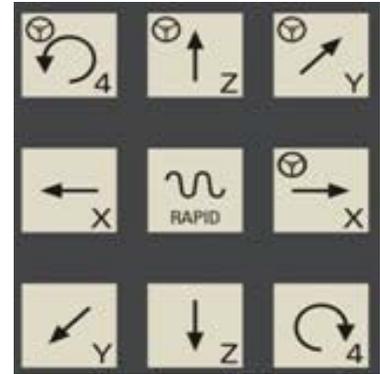
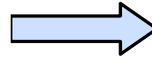
Press the "CYCLE START" key on the MCP



Tool setup

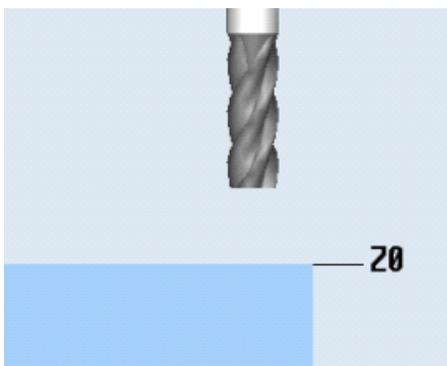
SEQUENCE

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



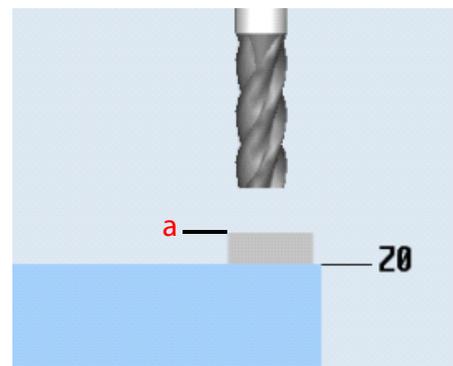
Note: The following text describes the required settings in the workpiece coordinate system
 "X / Y / Z" zero points as: "X0" / "Y0" / "Z0"

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



Move directly to zero point

or

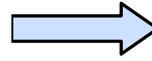


Use a setting block.

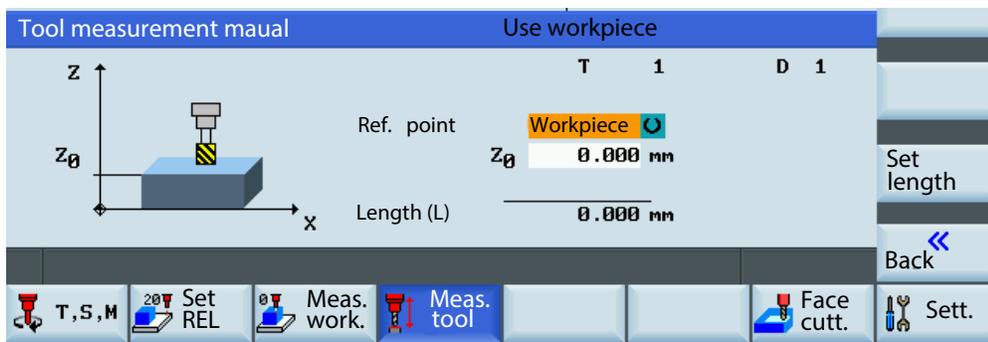


Tool setup

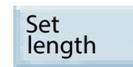
Use "SELECT" key to set the reference point as "workpiece" (In real measurement, the reference point can be set as either "workpiece" or "fixed point" if required.)



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



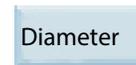
Press the "Set length" SK on the PPU



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

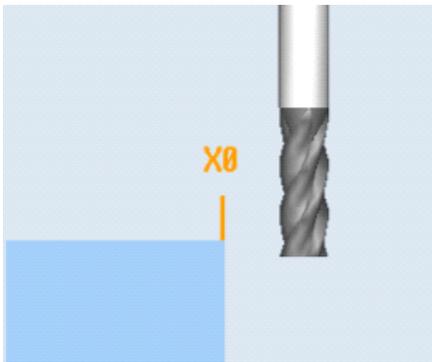
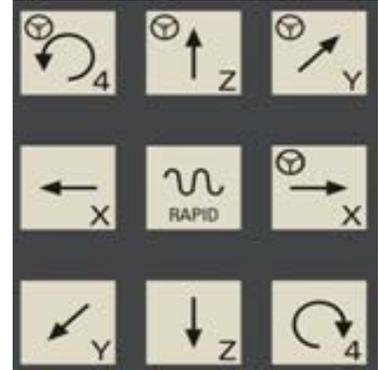
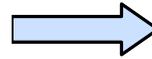
Press the "Diameter" SK on the PPU



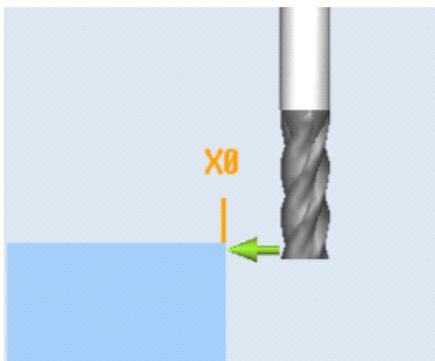
Tool setup

SEQUENCE

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

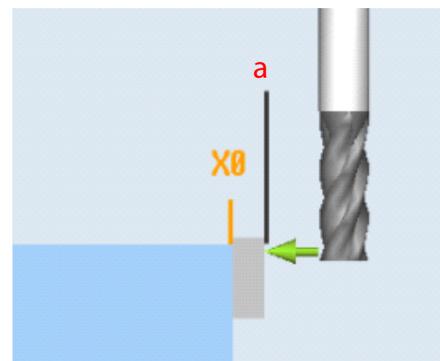


Press the "Handwheel" key on the MCP and position the tool at the location X0 or a of the workpiece.



Move directly to zero point

or



Use a setting block.

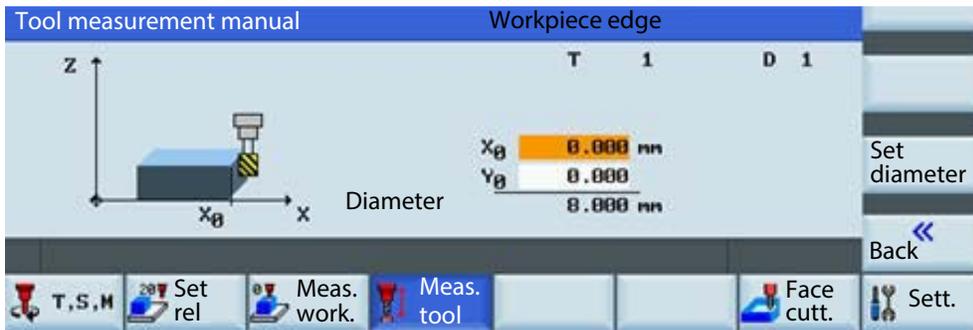


Tool setup

Enter "0" at "X0"

Enter "0" at "Y0"

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



Press the "Set diameter" SK on the PPU



Press the "Back" SK on the PPU



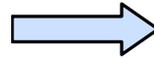
Tool setup

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.



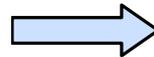
Press "Spindle left" on the MCP to start the spindle in the counter-clockwise direction.



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



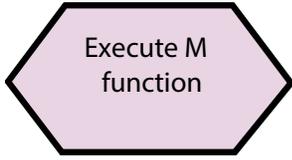
Press "Spindle right" on the MCP to start the spindle in the clockwise direction.



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



Tool setup



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

Press the "Machine" key on the PPU



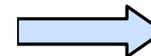
Press the "T.S.M" SK on the PPU.



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



Press "CYCLE START" on the MCP.



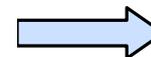
The coolant function button on MCP is active.



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



Press the "Back" SK on the PPU.



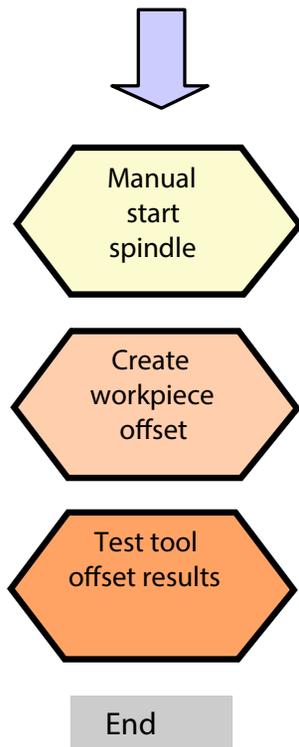
Workpiece setup

Workpiece Setup

Description

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

Content



Workpiece setup

SEQUENCE



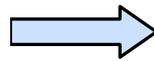
A tool must have been loaded into the spindle.

Before measuring, the spindle can be started as follows:

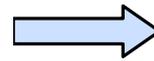
Press the "Machine" key on the PPU



Press the "JOG" key on the MCP



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



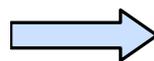
Enter "500" at "Spindle speed"



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



Press the "CYCLE START" key on the MCP



Workpiece setup

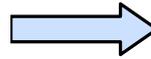
SEQUENCE

The image shows a Siemens CNC control screen with the 'Reset' menu open. The screen displays the following data:

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%
Z	0.000	0.000 mm	S1	500.0	100%
				500.0	0

The 'S1' parameter is circled in red. The screen also shows 'SIEMENS' and '11:39:12 2012/04/25'.

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



Press the "Back" SK on the PPU



Workpiece setup

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.



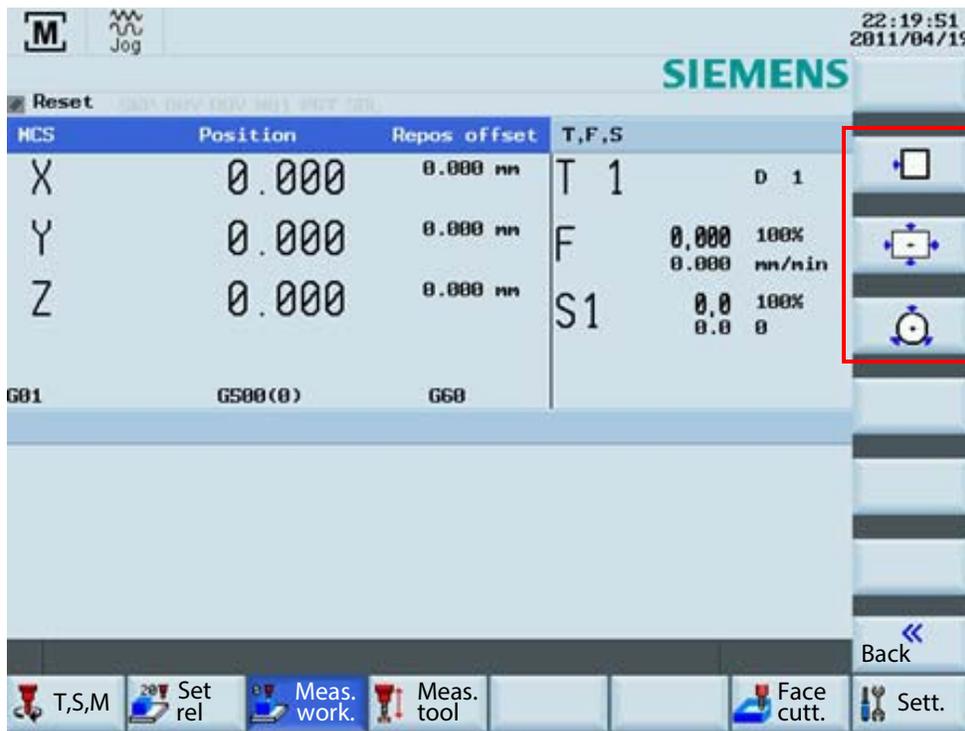
Press the "JOG" key on the MCP.



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.



Workpiece setup

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.

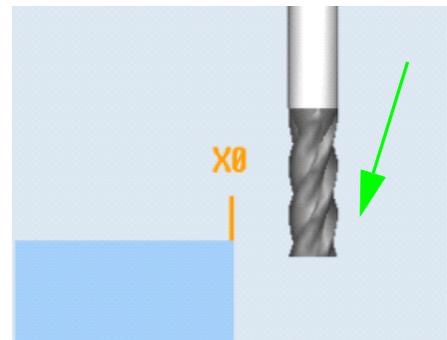
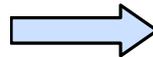
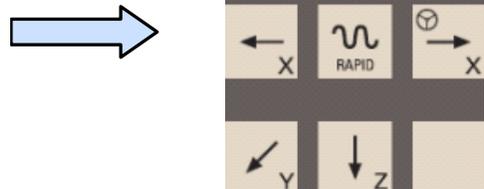
Press the corresponding SK of the first icon on the right-hand side of the PPU.



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

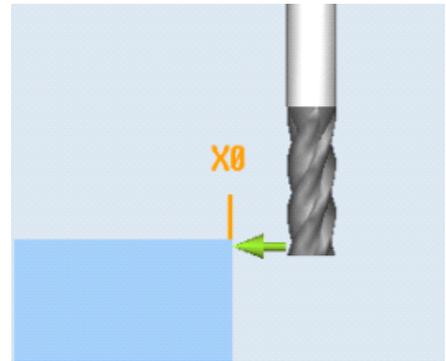


Press the axis traverse keys to move the tool to the required setting position in the X axis.



Workpiece setup

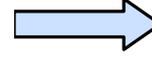
Press the "Handwheel" key on the MCP to position the tool at the X0 edge of the workpiece.



Select "Save in" Offset "G54" (or other offset).



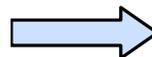
Select "Measuring direction" as "-".
(This value should be chosen according to realities)



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.



Workpiece setup

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.



Workpiece setup

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.

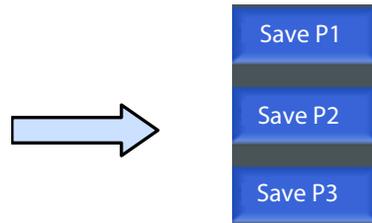


Observing the figure on the PPU, move the coordinate axis following the orange arrow to move the tool to the specified position and scratch the edge of the workpiece.

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.



Workpiece setup

SEQUENCE



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

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

Press the "Machine" key on the PPU



Press the "MDA" key on the MCP.



Press the "Delete file" SK on the PPU.

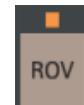


Enter the test program recommended on the right.
(can also be customized)



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).

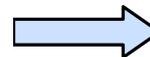


Note: The ROV function activates the feedrate override switch under the G00 function.



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.



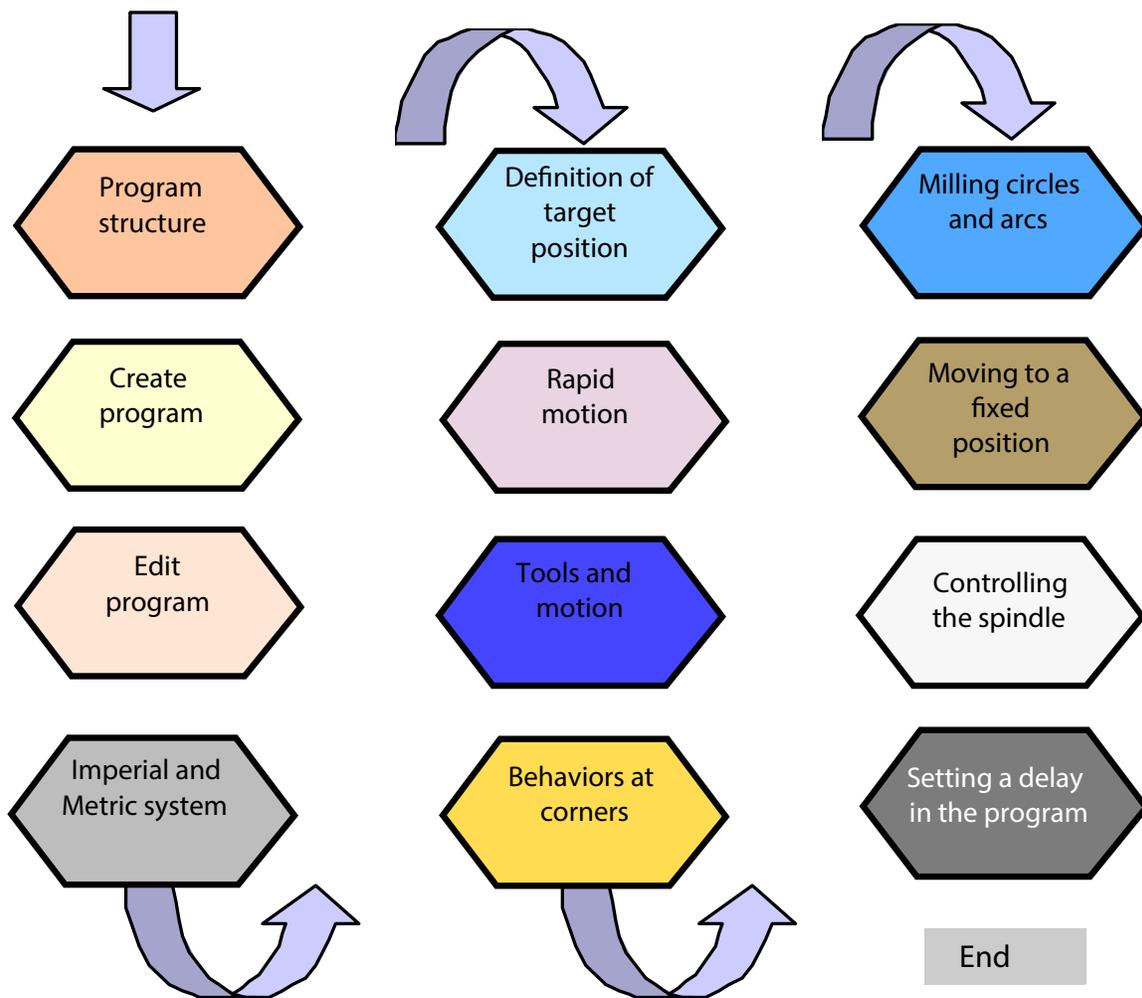
Create part program: part 1

Create Part Program Part 1

Description

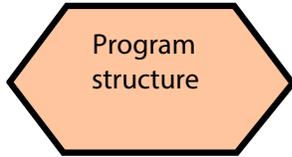
This unit describes how to create a part program, edit the part program and get to know the most important CNC commands required to produce a workpiece.

Content

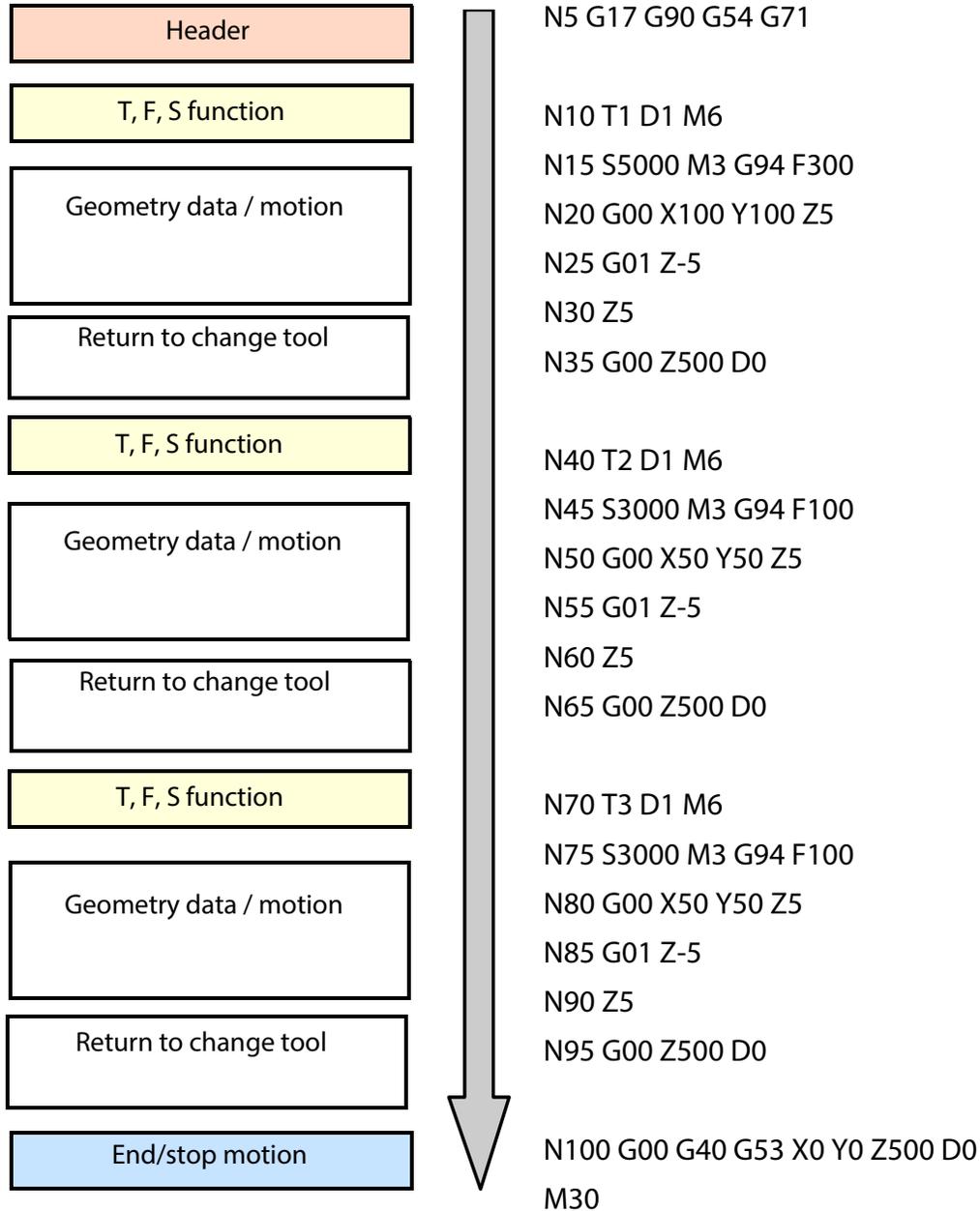


Create part program: part 1

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:



Create part program: part 1

SEQUENCE



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.



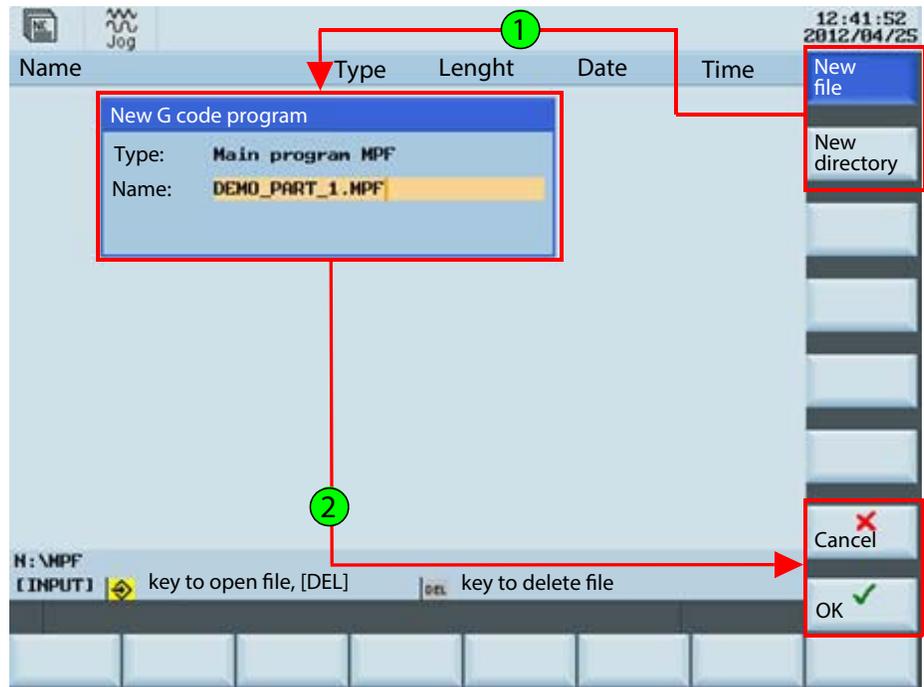
Create part program: part 1

Step 4

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

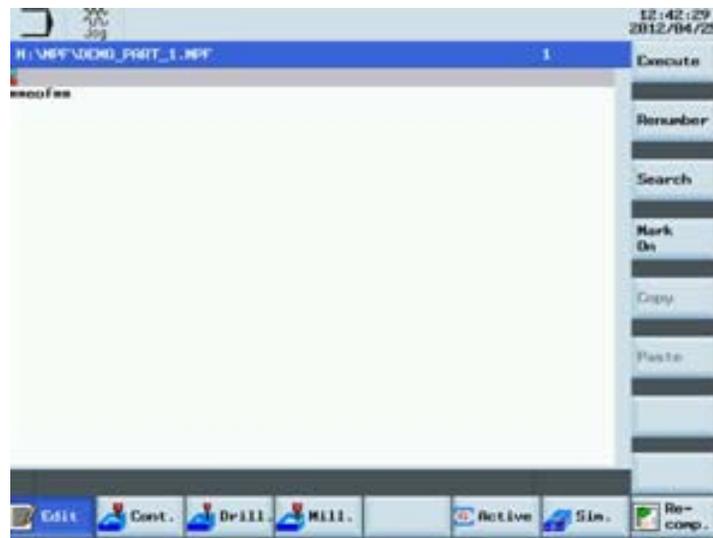
Choose "New" to create a program.

Choose "New directory" to create a file.



Step 5

Now the program is opened and can be edited.



The system will save it automatically after editing.

End

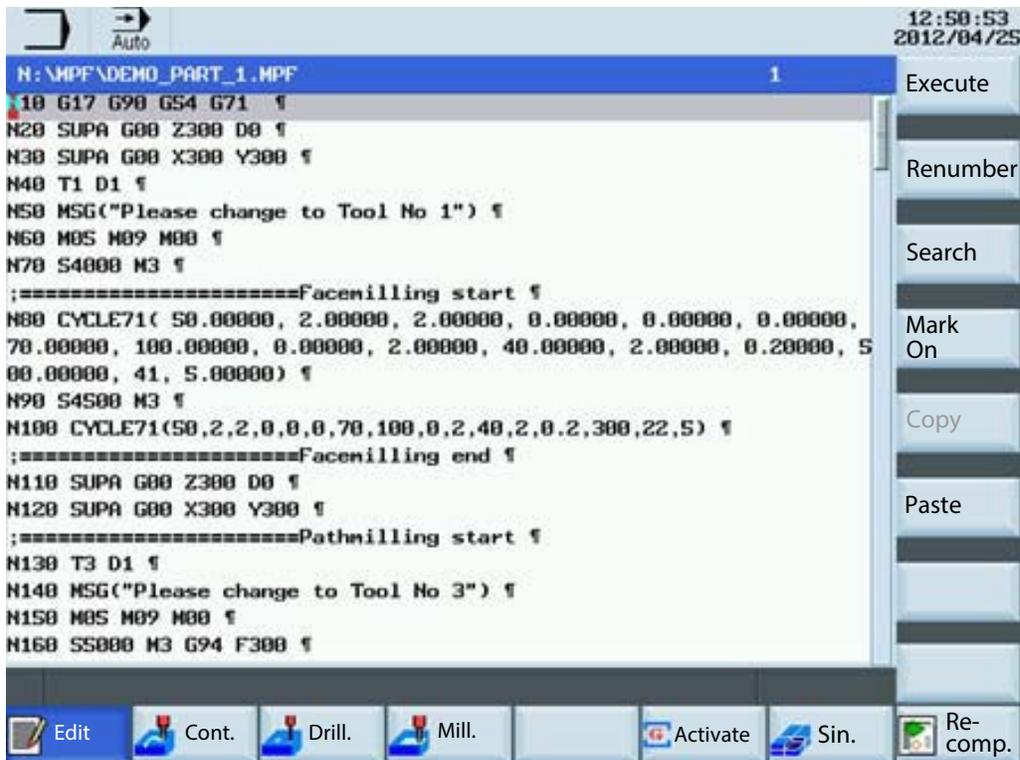
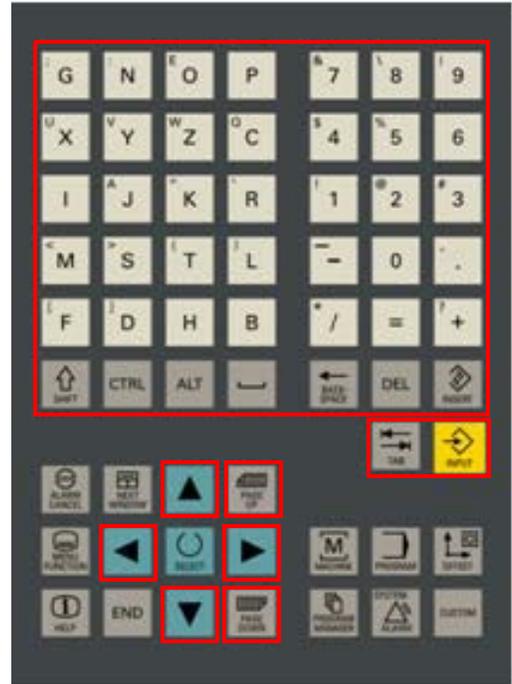


Create part program: part 1

Basic Theory



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



Create part program: part 1



G71

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

Header

N5 G17 G90 G54 G71

T, F, S function

N10 T1 D1 M6

N15 S5000 M3 G94 F300

Geometry data / motion

N20 G00 X 100 Y 100 Z5

N25 G01 Z- 5

N30 Z 5

Return to change tool

N35 G00 Z 500 D0

G70

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

Header

N5 G17 G90 G54 G70

T, F, S function

N10 T1 D1 M6

N15 S5000 M3 G94 F300

Geometry data / motion

N20 G00 X 3.93 Y 3.93 Z5

N25 G01 Z- 0.787

N30 Z 0.196

Return to change tool

N35 G00 Z 19.68 D0

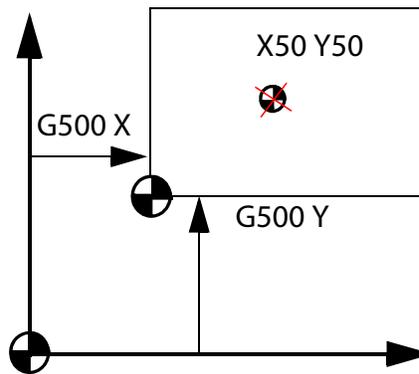


Create part program: part 1



G500

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



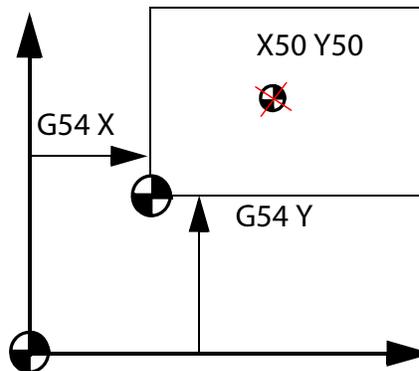
```
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X 50 Y 50 Z5
N25 G01 Z- 20
N30 Z 5
N35 G00 Z 500 D0
```

Or

G54 G55 G56 G57

G58 G59

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

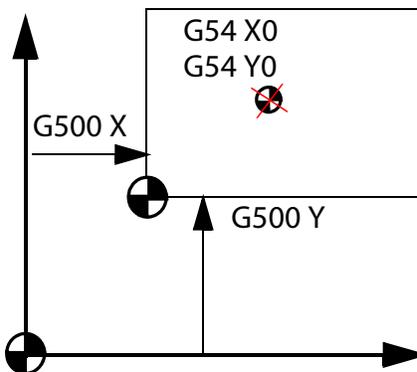


```
N5 G17 G90 G54 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X 0 Y 0 Z5
N25 G01 Z- 20
N30 Z 5
N35 G00 Z 500 D0
```

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 G 54 X 20 Y 20 Z5
N25 G01 Z- 20
N30 Z 5
N35 G00 G 53 Z500 D0
```



Create part program: part 1

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 X 100 Y 100 Z 5
```

```
N25 G01 Z -20
```

```
N30 Z 5
```

```
N35 G00 Z 500 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
```

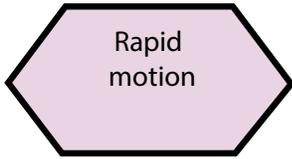
```
N25 G01 G91 Z-0.787
```

```
N30 Z 0.196
```

```
N35 G00 G90 Z19.68 D0
```

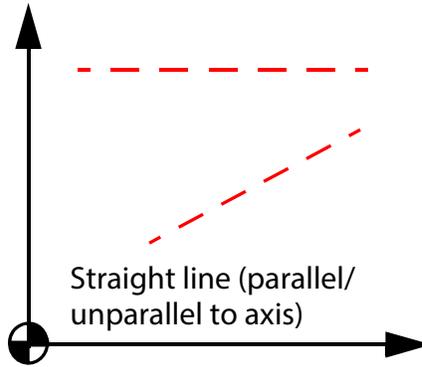


Create part program: part 1



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 X 50 Y 50 Z 5
N25 G01 Z -5
N30 Z 5
N35 G00 Z 500 D0
    
```



T1 D1 M06

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

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



```

N5 G17 G90 G54 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-20
N30 Z5
N35 G00 Z500 D0
    
```



Create part program: part 1

- 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

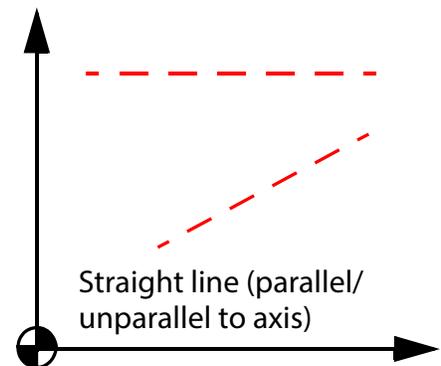
When G01 is activated in the program, the axis will traverse at the programmed feed rate in a straight line, according to the feed rate type defined by G94 or G95.



```
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
```



Create part program: part 1

Basic Theory



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.



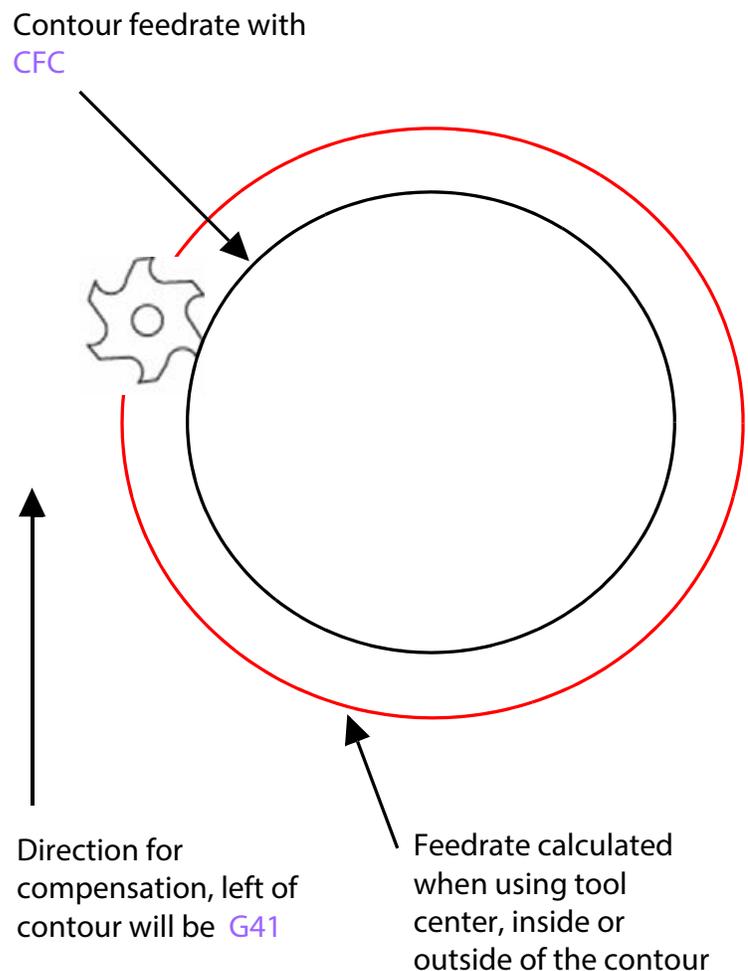
Create part program: part 1

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

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

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

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

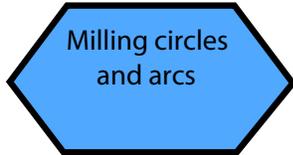


The result of the two commands will be such that the cutter goes very fast around a corner or slow on the contour.



Create part program: part 1

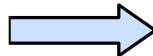
Basic Theory



The circle radius shown in the example on the right can be produced with the specified part program code.

When milling circles and arcs, you must define the circle center point and the distance between the start point / end point and the center point on the relative coordinate.

When working in the XY coordinate system, the interpolation parameters I and J are available.



```
N5 G17 G90 G500 G71
N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X-20 Y-20 Z 5
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 Z 500 D0
```

Note:

N45 can also be written as follows

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

Two common types of defining circles and arcs:

① :G02/G03 X_Y_I_J_;

② :G02/G03 X_Y_CR=_;

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

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

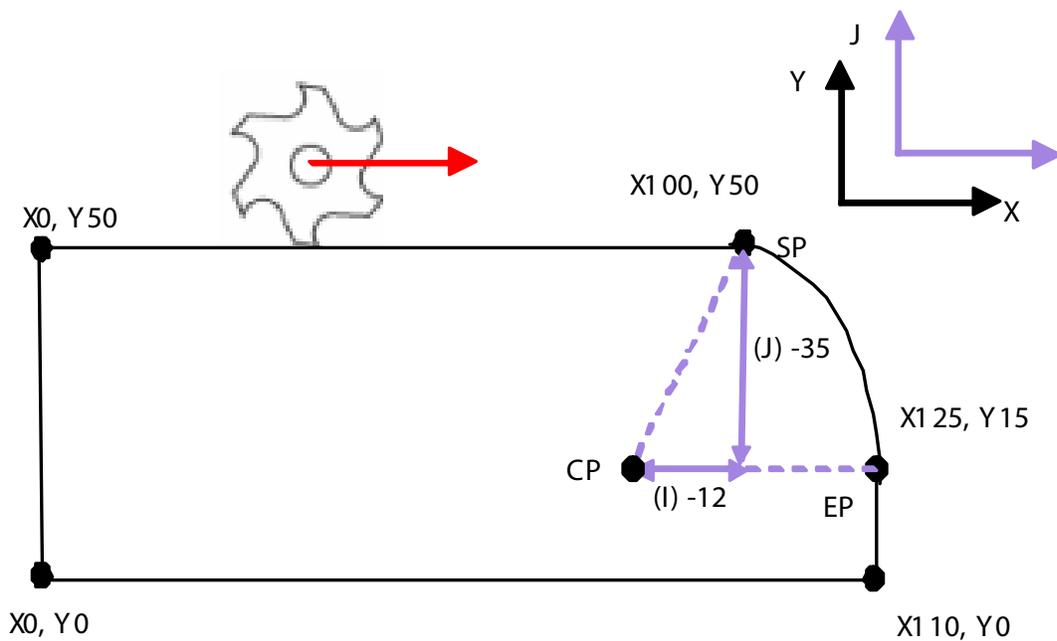


When milling circles, you can only use ① to define the program!



Create part program: part 1

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



Create part program: part 1

Basic Theory



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
```

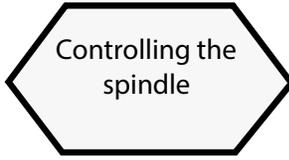
```
N30 Z5
```

```
N35 G74 Z=0 ;reference point
```

```
N40 G75 X=0 ;fixed point
```



Create part program: part 1



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
```

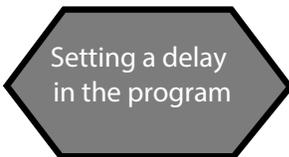
```
N30 M5
```

```
N35 Z5 M4
```

```
N40 M5
```

```
N45 M19
```

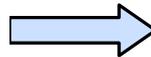
```
N50 G00 Z500 D0
```



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
```

```
N50 G00 Z500 D0
```



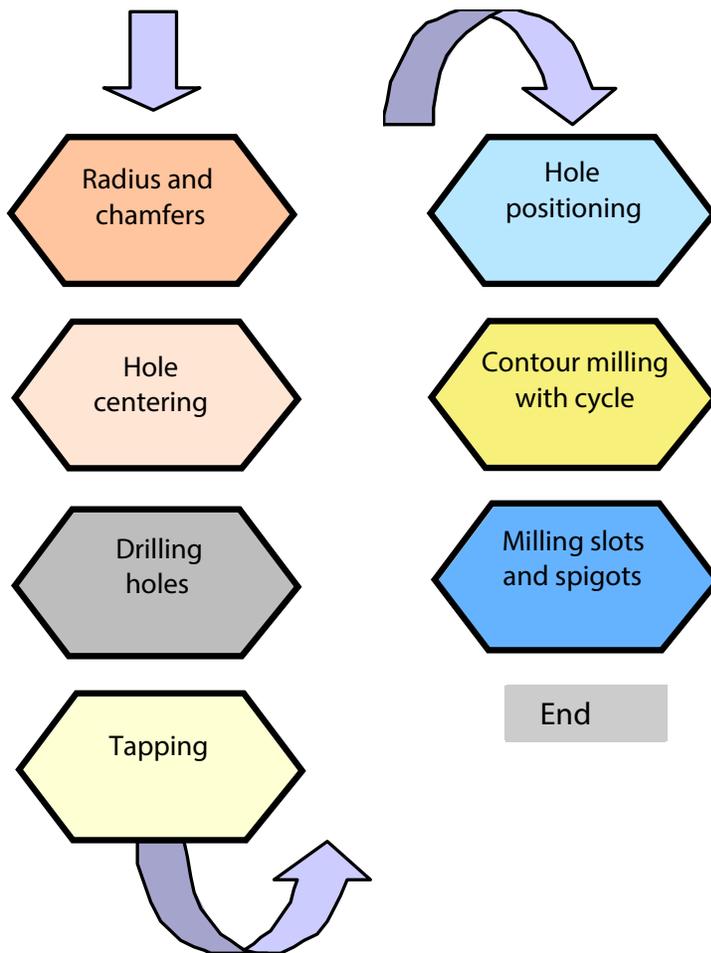
Create part program: part 2

Create Part Program Part 2

Description

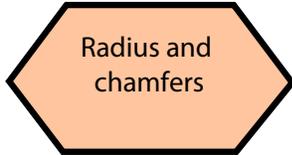
This unit describes how to create a part program, edit the part program and get to know the most important CNC commands required to produce a workpiece.

Content

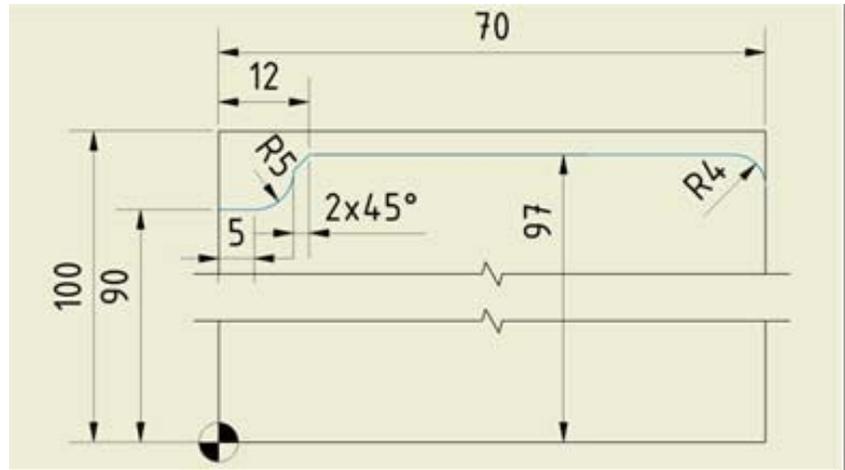


Create part program: part 2

Basic Theory



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

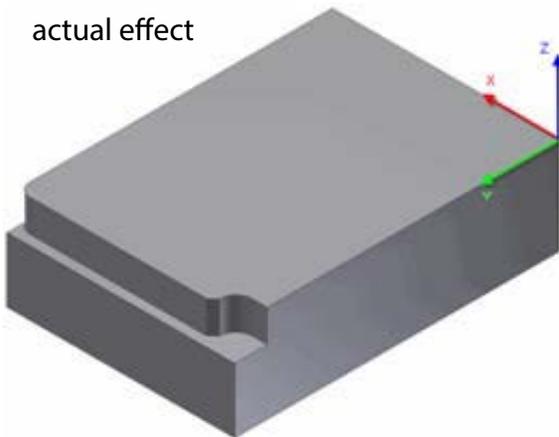
N115 G01 X70 RND=4

N120 G01 Y90

N125 G01 G40 X80

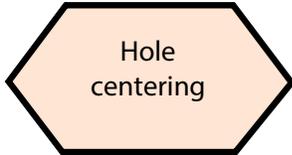
N130 G00 Z50

actual effect



Create part program: part 2

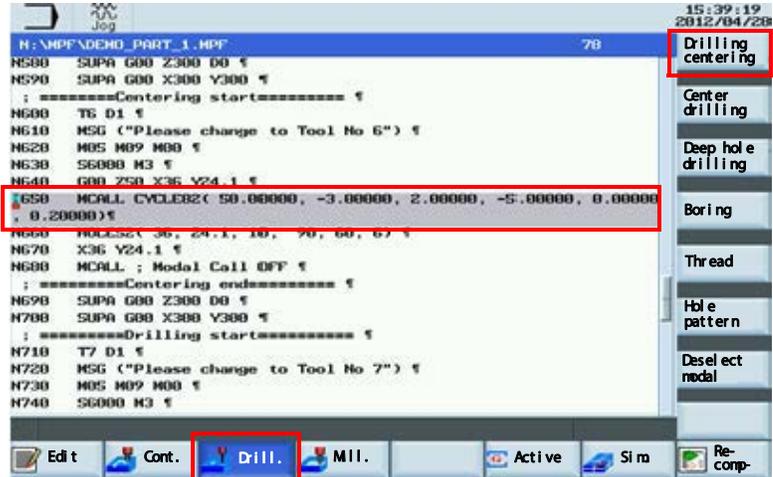
Basic Theory



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

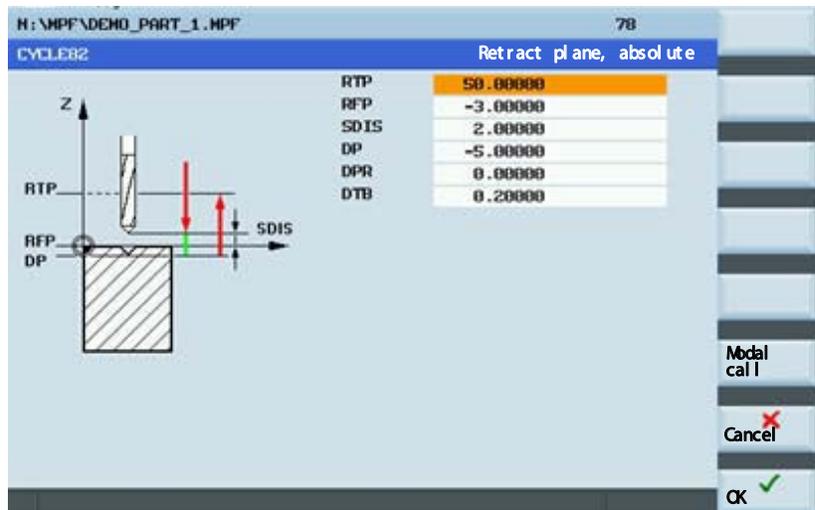
CYCLE82: With delay at current hole depth



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.



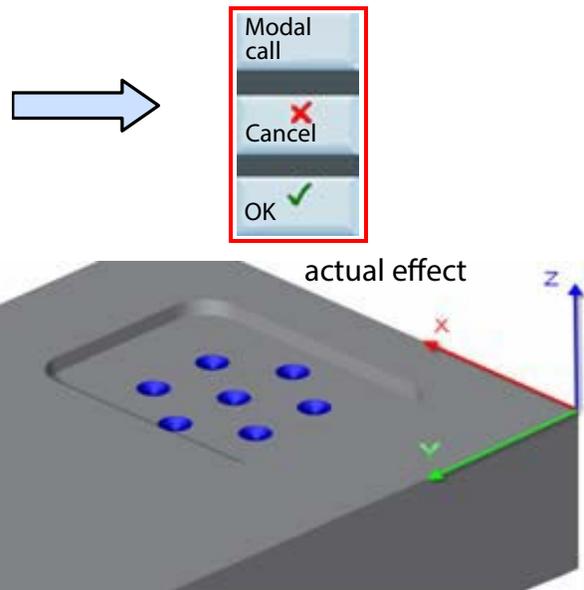
Create part program: part 2

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.



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

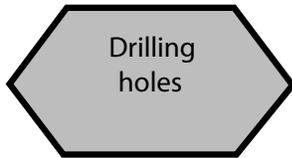
Parameters	Meanings
RTP=50	Coordinate value of turning position is 50 (absolute)
RFP=-3	Coordinate value of hole edge starting position under workpiece zero point surface is 3 (absolute)
SDID=2 (frequently used values 2~5)	Safety distance, feed path changes from quick feed to machine feed 2 mm away from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

```
N325 MCALL CYCLE82( 50.000, -3.000, 0.000, 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
```



Create part program: part 2

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.

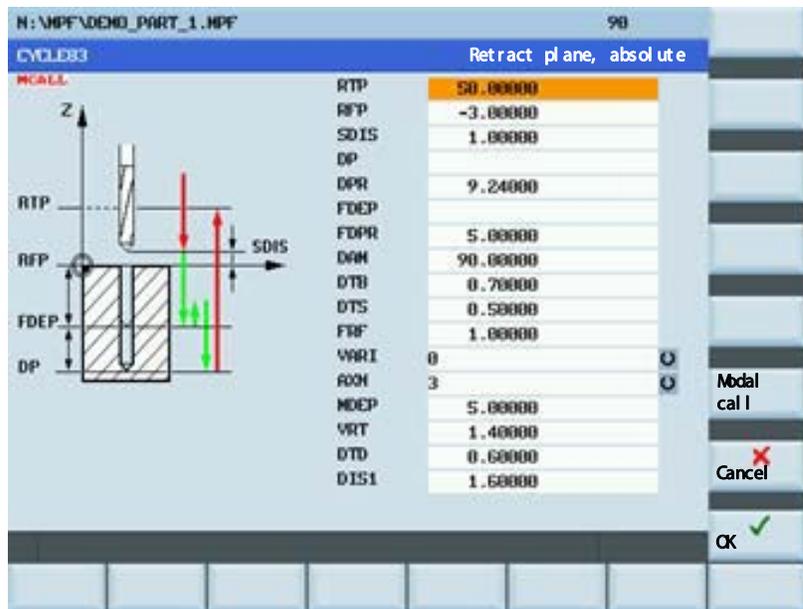
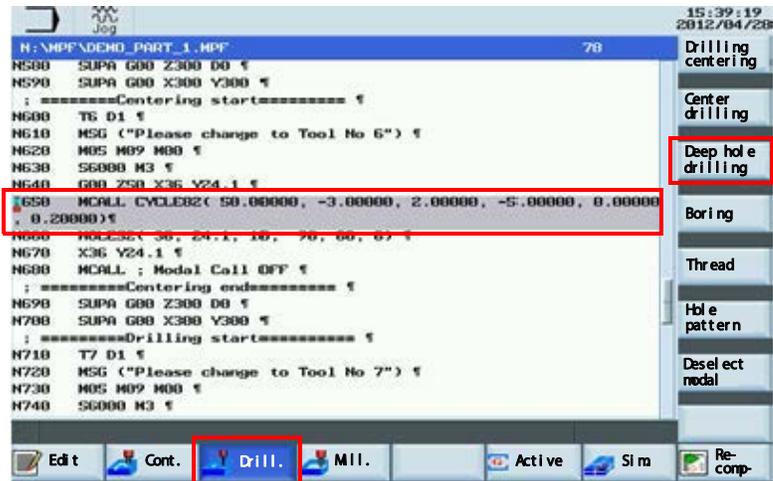
The cycle can be found and parameterized with the "Drill." SK.



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.



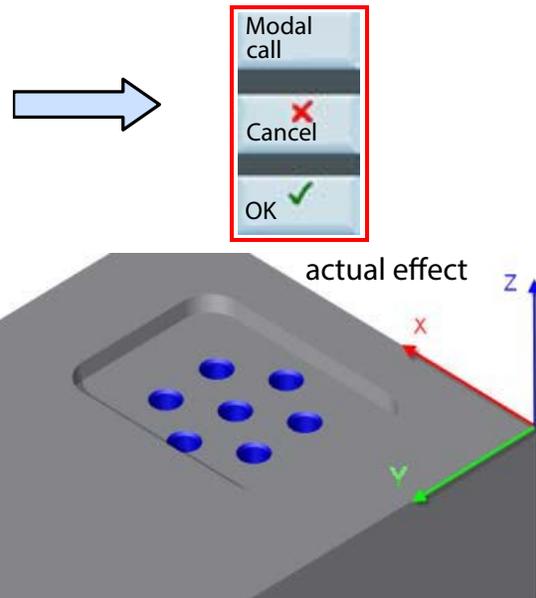
Create part program: part 2

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

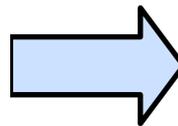
This will drill a hole at the current position.

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



Create part program: part 2

Basic Theory

	For descriptions of RTP, RFP, SDIS and DP, please see  Page 140	
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



Create part program: part 2

DAM parameter

① $DAM \neq 0$, 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 $\leq DAM$. 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 $< 2 \times FDPR$, 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 < 2x FDPR, the remaining depth distribute by the last two drilling		
5.	(FDPR-3DAM)-DAM =4-2=2	-30			
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.		



Create part program: part 2

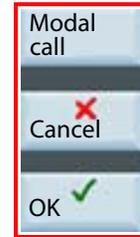
With the "OK" SK, the values and cycle call will be transferred to the part program as shown below.

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.



Create part program: part 2

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	

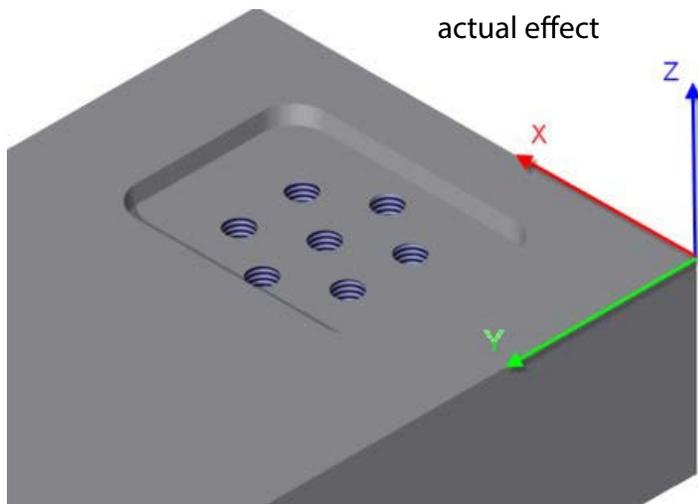
N325 MCALL CYCLE84(50.00000, -3.00000, 2.00000, ,6.00000, 0.70000, 5, ,2.00000, 5.00000, 5.00000, 5.00000, 3, 0, 0, 0, 5.00000, 1.40000)

N330 X20 Y20 ; Hole will be tapped

N335 X40 Y40 ; Hole will be tapped

N340 MCALL

N345 X60 Y60 ; Hole will not be tapped



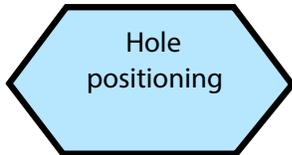
Create part program: part 2

	<p>For descriptions of RTP, RFP, SDIS and DP, please see  Page 140</p> <p>For descriptions of AXH, VARI, DAM and VRT, please see  Page 143</p>	
Parameters	Meanings	Remarks
DTB=0.7	Pause 0.7 s during final tapping to thread depth (discontinuous cutting)	
SDAC=5	Spindle state after cycle is M5	Enter values 3/4 -> M3/M4
PIT=2(Range of values:0.001~2000 mm)	Right hand thread with 2mm pitch	Evaluate value -> left hand thread
POSS=5	Spindle stops at 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
	<p>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.</p>	



Create part program: part 2

Basic Theory

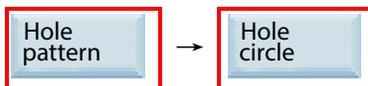


The easiest way to drill a series of holes is to use the pre-defined "Hole pattern" cycles.

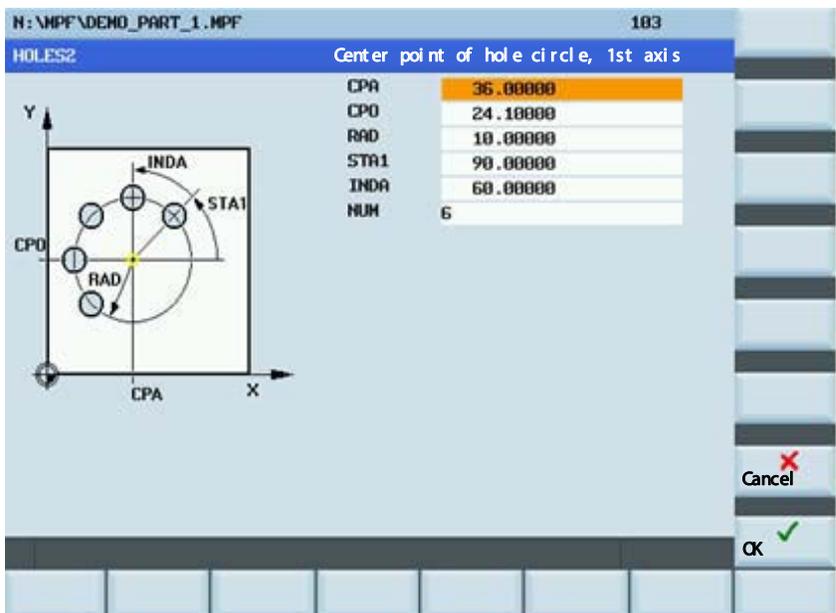
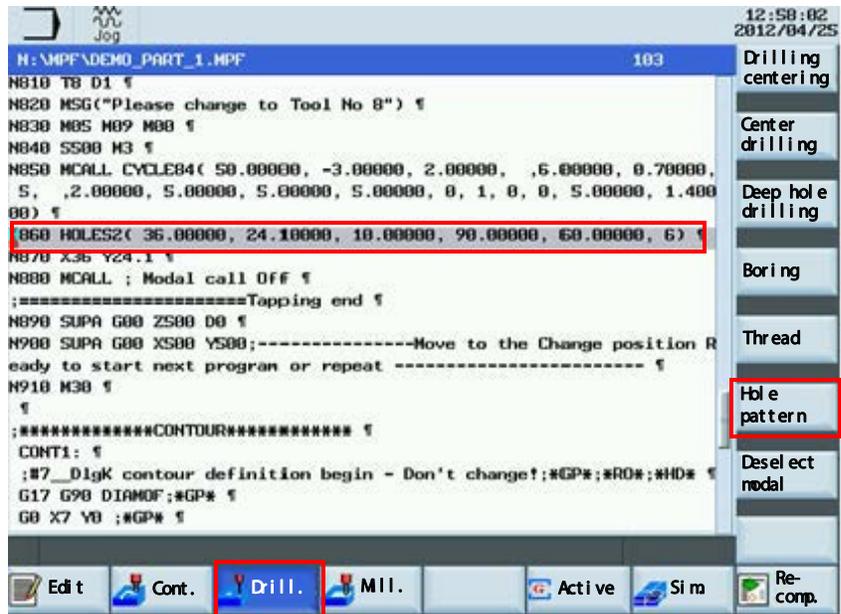
The cycles can be found and parameterized via the "Drill." SK.



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



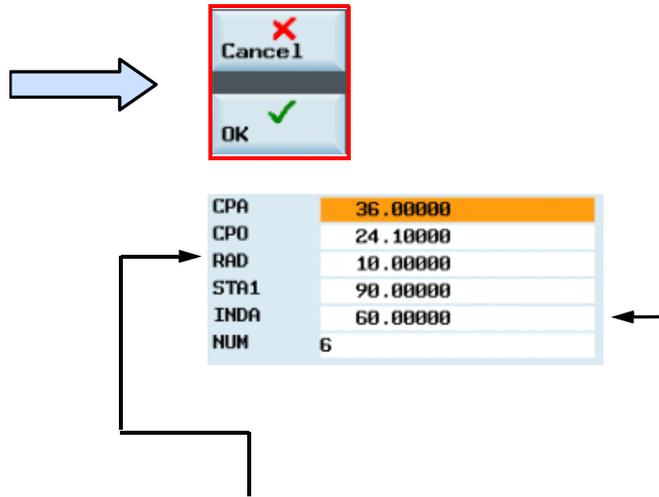
Select "Hole pattern" using the vertical SKs, and then select "Hole circle", and parameterize the cycle according to requirement.



Create part program: part 2

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

This will drill holes at the positions defined from within the cycle.



```
N325 MCALL CYCLE82( 50.00000, -3.00000, 2.00000, -5.00000, 0.00000, 0.20000)
```

```
N330 HOLES2( 36.00000, 24.10000, 10.00000, 90.00000, 60.00000, 6)
```

```
N335 X36 Y24.1
```

```
N340 MCALL ; Modal Call OFF
```

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



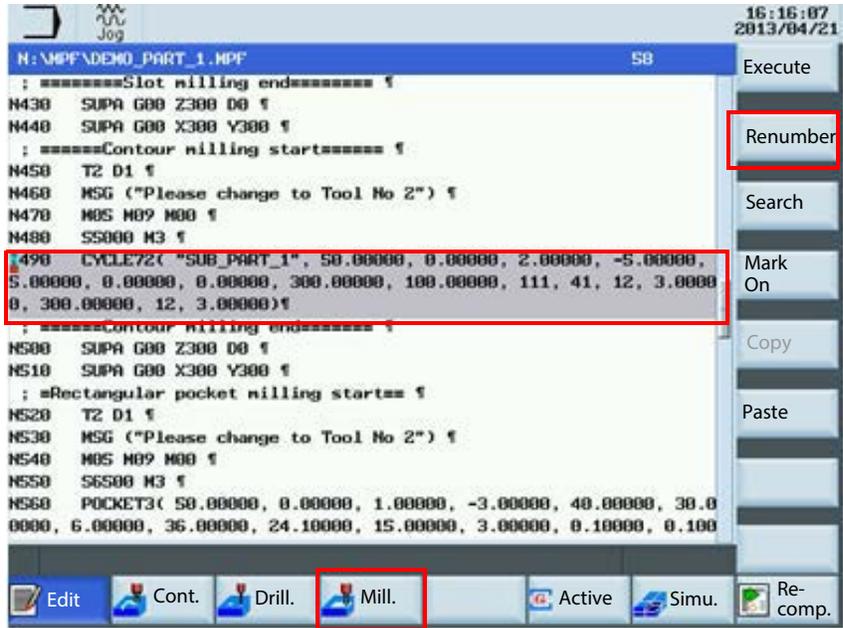
Create part program: part 2

Basic Theory



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

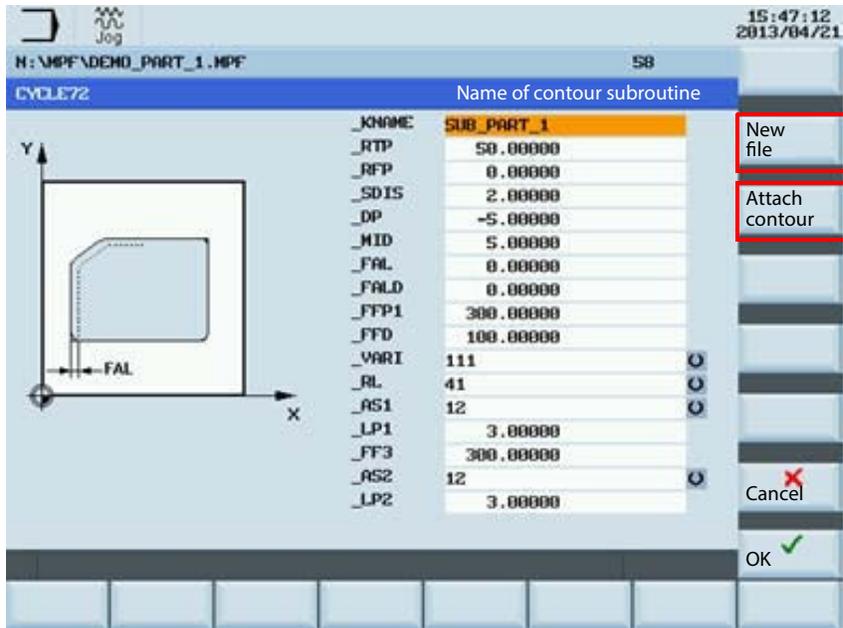
The cycle can be found and parameterized via the "Mill." SK.



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



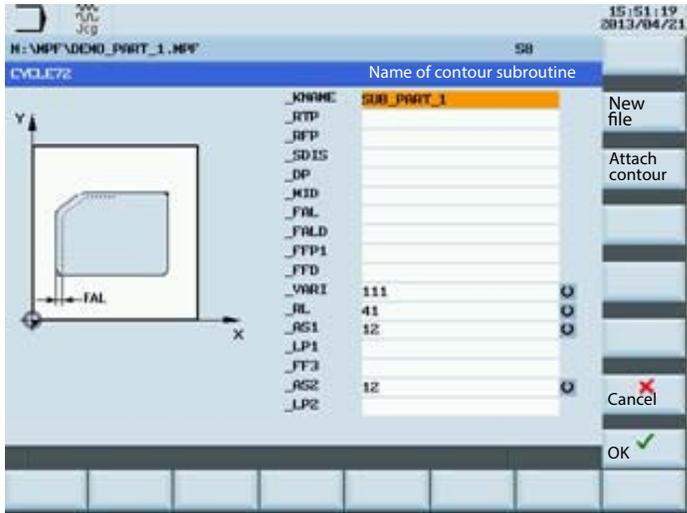
The parameterization is performed as in this figure.



Create part program: part 2

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:

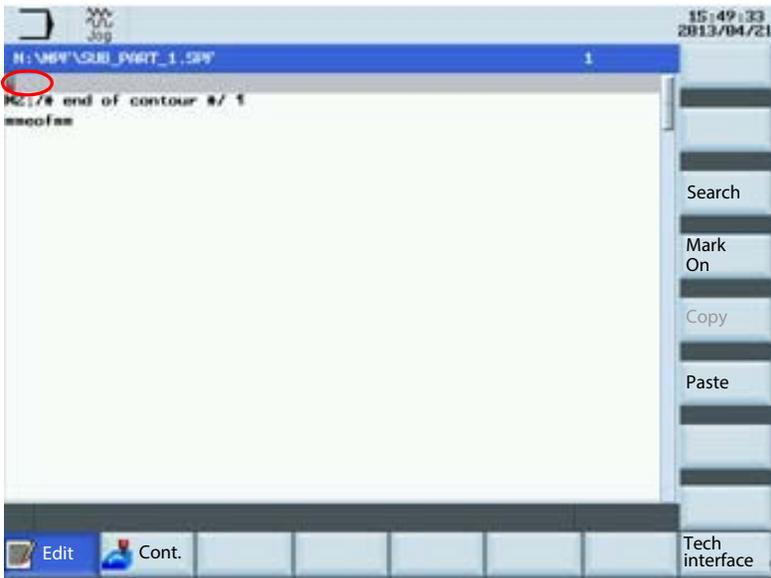


Enter the cycle data setting according to the former operations in the screen and enter the name of the contour subroutine.

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



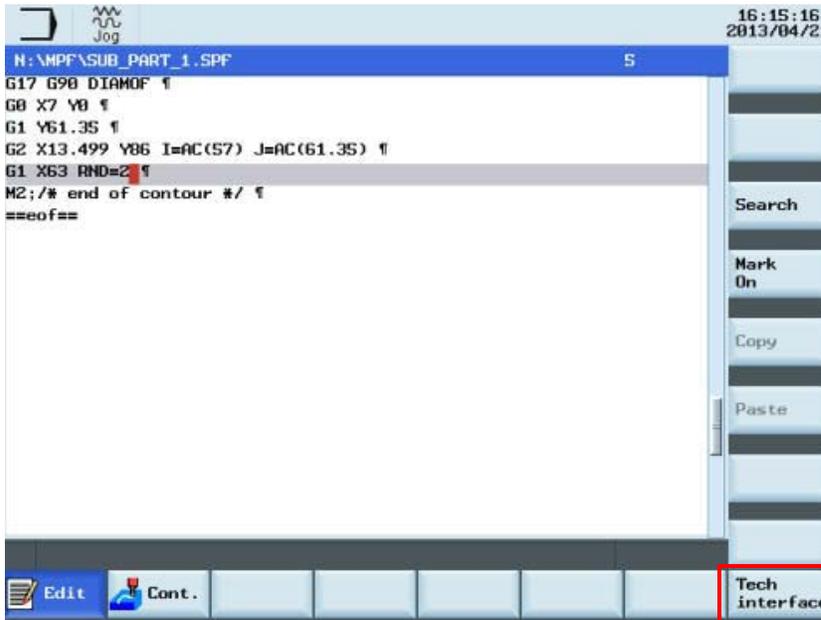
Make sure that the cursor has moved to the contour writing position (as shown in the figure).



Create part program: part 2

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.

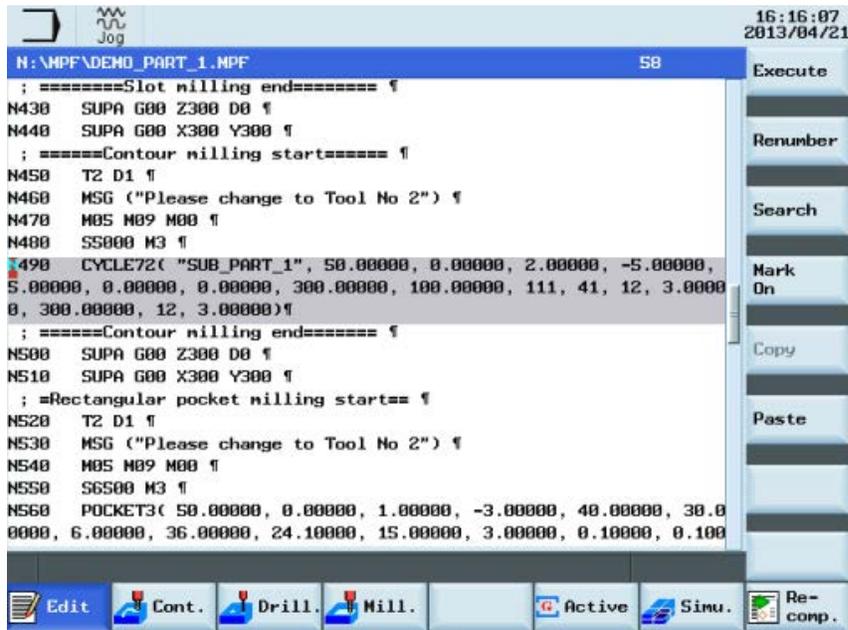
After completing the steps, the system will return to the edit interface. Press "Technical interface" on the PPU to return to the interface for setting the cycle data.



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



Create part program: part 2



```
N:\MPP\DEMO_PART_1.MPF 58
; =====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_1", 50.00000, 0.00000, 2.00000, -5.00000,
5.00000, 0.00000, 0.00000, 300.00000, 100.00000, 111, 41, 12, 3.0000
0, 300.00000, 12, 3.00000)¶
; =====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( 50.00000, 0.00000, 1.00000, -3.00000, 40.00000, 30.0
0000, 6.00000, 36.00000, 24.10000, 15.00000, 3.00000, 0.10000, 0.100
```

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).

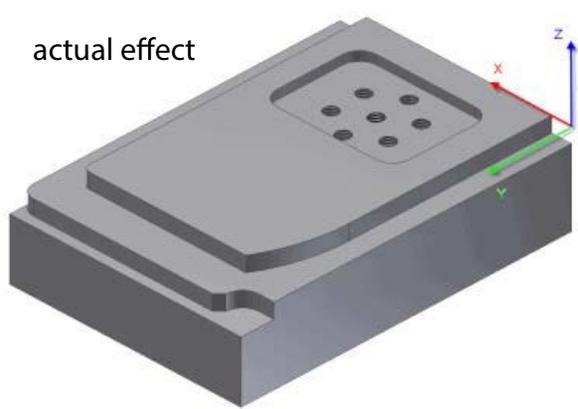


Create part program: part 2

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	<input type="checkbox"/>
_RL	41	<input type="checkbox"/>
_AS1	12	<input type="checkbox"/>
_LP1	3.00000	
_FF3	300.00000	
_AS2	12	<input type="checkbox"/>
_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)



Create part program: part 2

For descriptions of RTP, RFP, SDIS and DP, please see  Page 140		
Parameters	Meanings	Remarks
KNAME= CONT1:CONT1_E	Set the name of the contour subprogram as "CONT1" ("":CONT1_E" is automatically created)	The first two positions of the program name must be letters
MID=5	The maximal feed depth is 5 mm	
FAL=0	Finishing allowance at the contour side is 0 mm	
FALD=0	Finishing allowance at the bottom plane is 0 mm	
FFP1=300	Tool feed rate on plane is 300 mm/min	
FFD=100	Feed rate after inserting the tool in the material is 100 mm/min	
VARI=111	Use G1 to perform rough machining, and back to the depth defined by the RTP+SDIS at the completion of the contour	For other parameters, please refer to the standard manual
RL=41(absolute value)	PL=41 -> use G41 to make tool compensation on the left side of the contour	PL=40 -> G40, PL=42 -> G42
AS1=12	Approach the contour along the 1/4 circle on the path in space	For other parameters, please refer to the standard manual
LP1=3	The radius of the approaching circle is 20 mm	The length of the approaching path is along the line to approach
FF3=300	The feed rate during retraction of the path is 300 mm/min	
AS2=12	Return along the 1/4 circle on the path in space	Parameter explanations are the same as for AS1
LP2=3	The radius of the return circle is 20 mm	The length of the returning path is along the line to approach



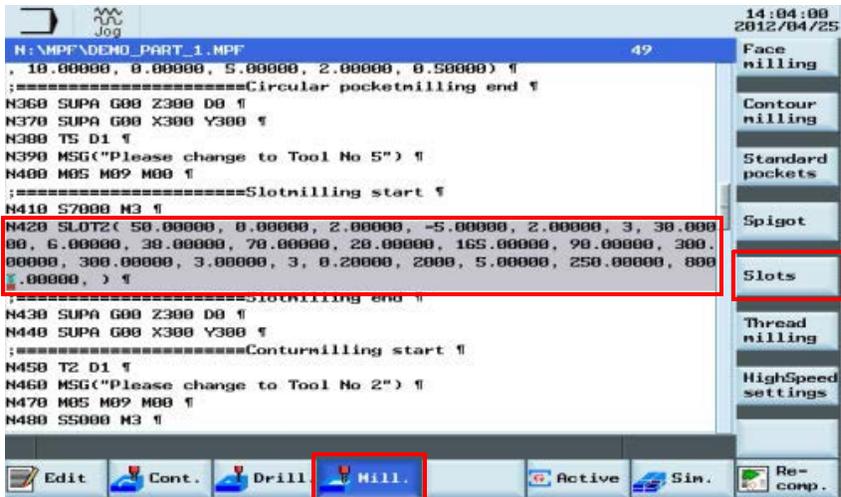
Create part program: part 2

Basic Theory

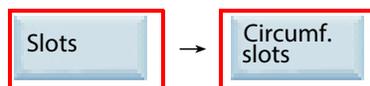


The easiest way to mill a slot is to use the SLOT2 cycle.

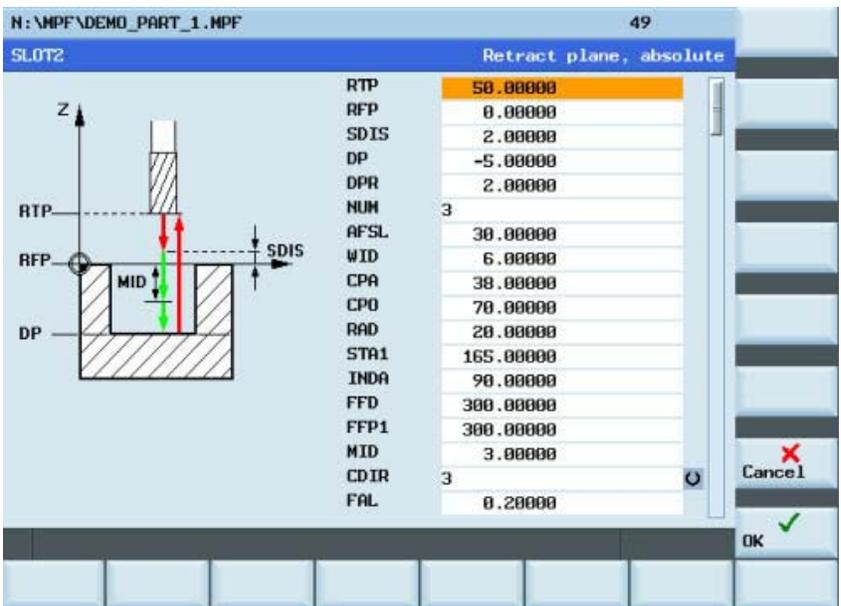
The cycle can be found and parameterized via the "Mill." SK.



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.



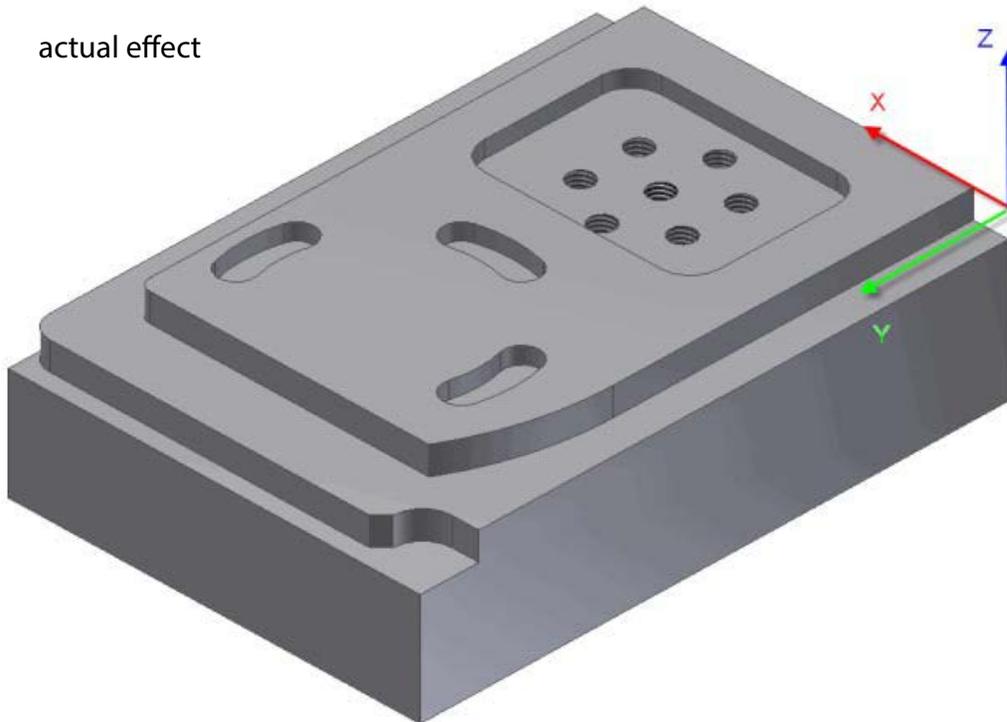
Create part program: part 2

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



Create part program: part 2

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	
	FPCP			

```
N210 SLOT2(
50.00000,0.00000,2.00000,, 3.00000,3,30.00000,6.00000,
38.00000,70.00000,20.00000,16 5.00000,90.00000,300.00000,
300.00000,3.00000,3,0.200 00,2000,5.00000,250.00000,
3000.00000,)
```

For descriptions of RTP, RFP, SDIS, DP and DPR, please see Page 140

For descriptions of CPA, CPO and RAD, please see Page 150

For descriptions of FFD and FFP1, please see Page 156

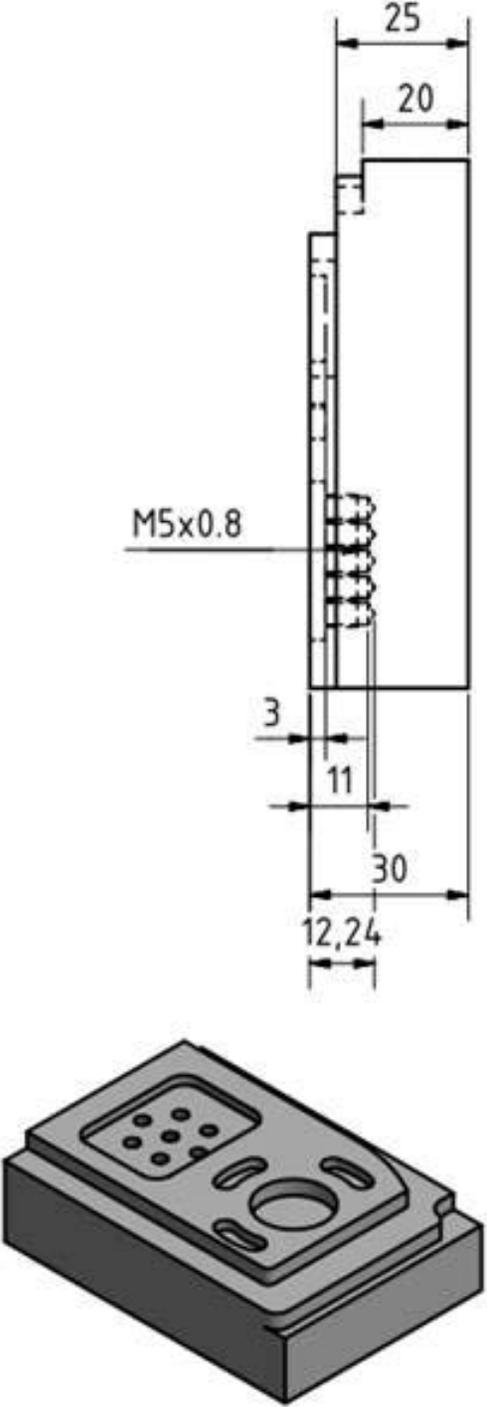


Create part program: part 2

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.	



Create part program: part 2



Technical drawing showing dimensions for a part:

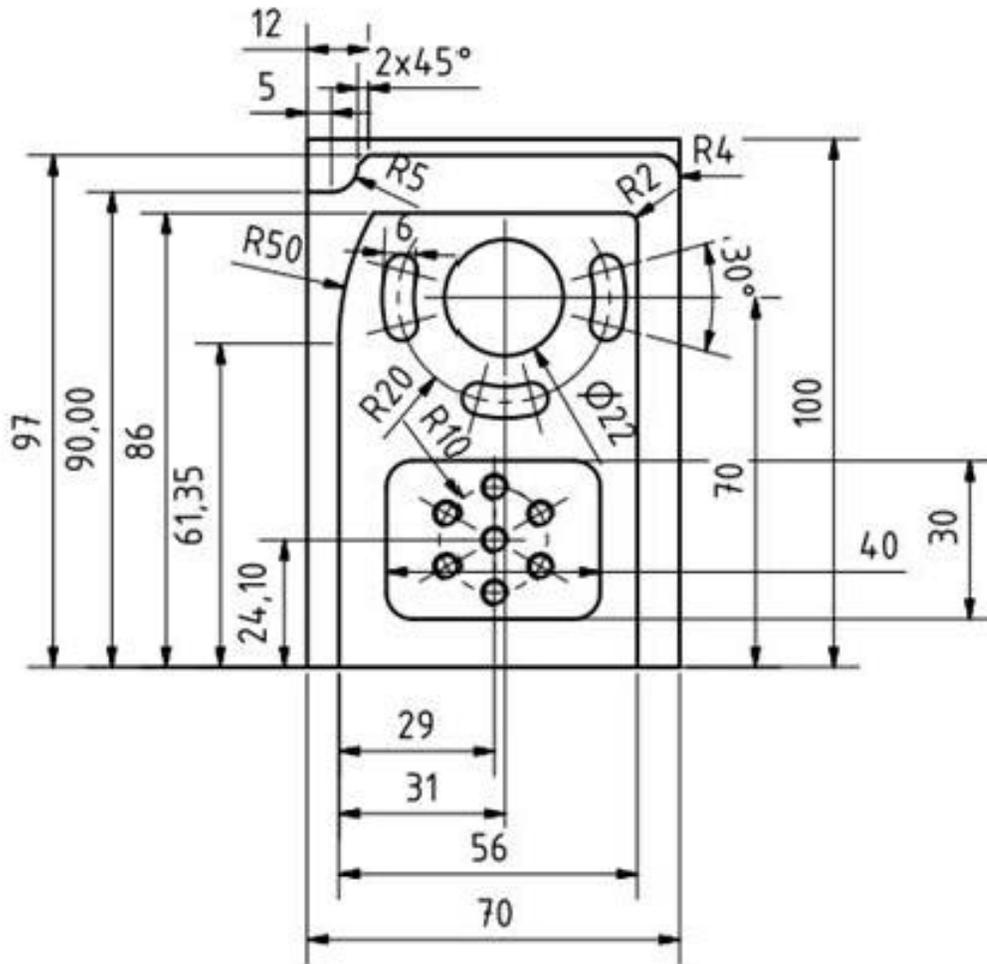
- Overall width: 25
- Inner width: 20
- Thread specification: M5x0.8
- Distance from bottom to start of thread: 3
- Distance from bottom to end of thread: 11
- Distance from bottom to end of main body: 30
- Distance from bottom to end of a specific section: 12,24

Isometric view of the part showing a rectangular block with a top surface featuring a grid of holes and a circular feature.

Status	Änderungen	Datum	Name	Datum
				Datum
				Gesichtet
				Kontrolliert
				Norm



Create part program: part 2



			Datum	Name	
		Gezeichnet			
		Kontrolliert			
		Norm			
					1
					A4
tungen	Datum	Name			



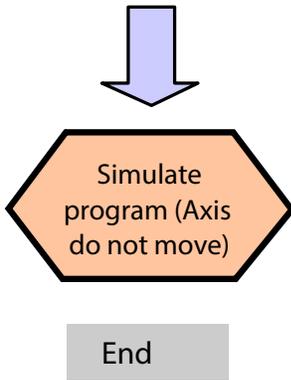
Simulate program

Simulate Program

Description

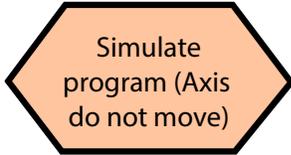
This unit describes how to simulate a part program before executing it in AUTO mode.

Content



Simulate program

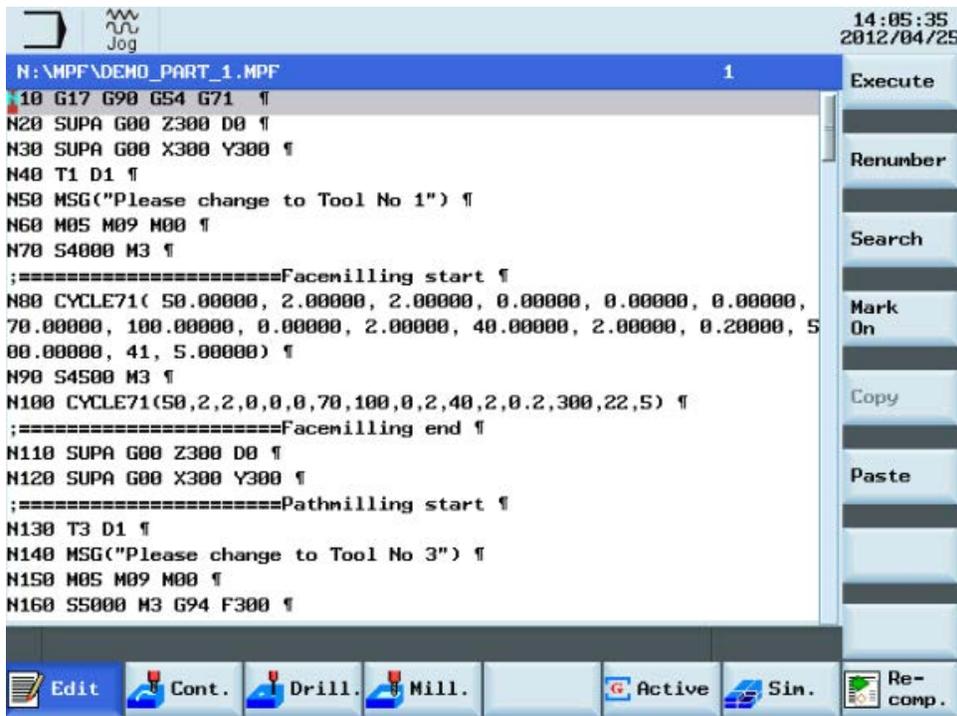
Basic Theory



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.

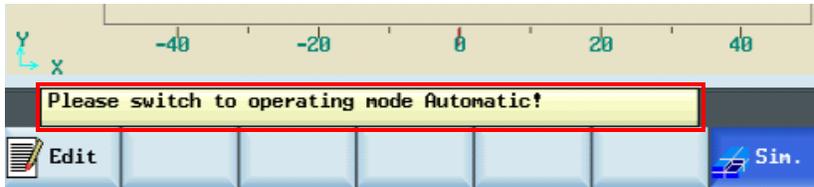


Simulate program

SEQUENCE

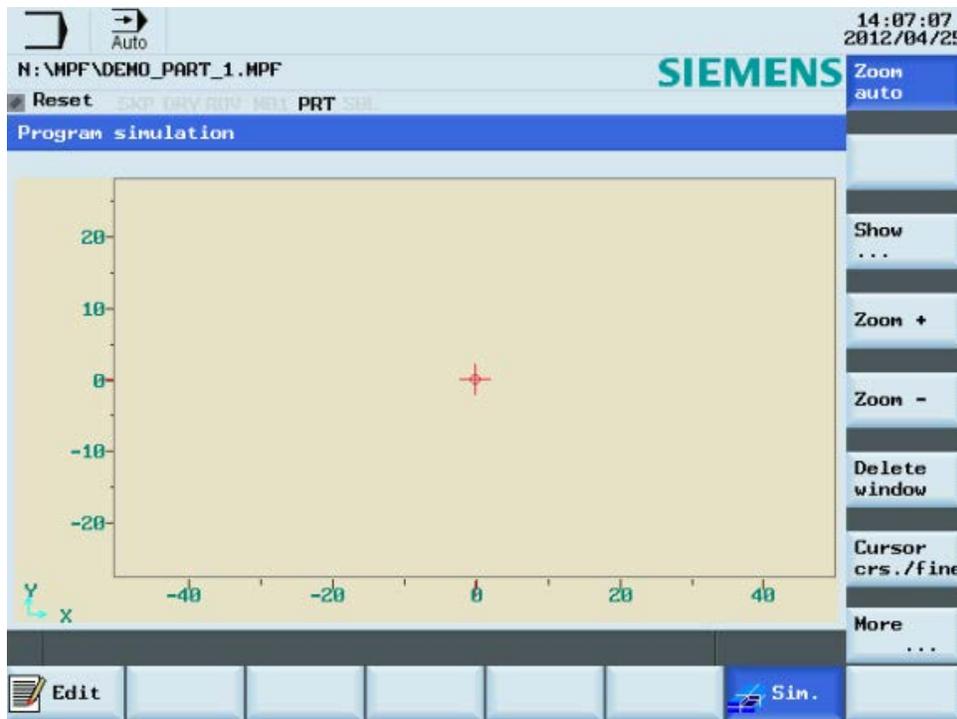
Step 2

Press the "Simu." SK on the PPU.



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

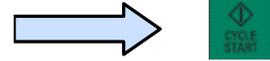
If this message is displayed at the bottom of the screen, press the "AUTO" mode key on the MCP.



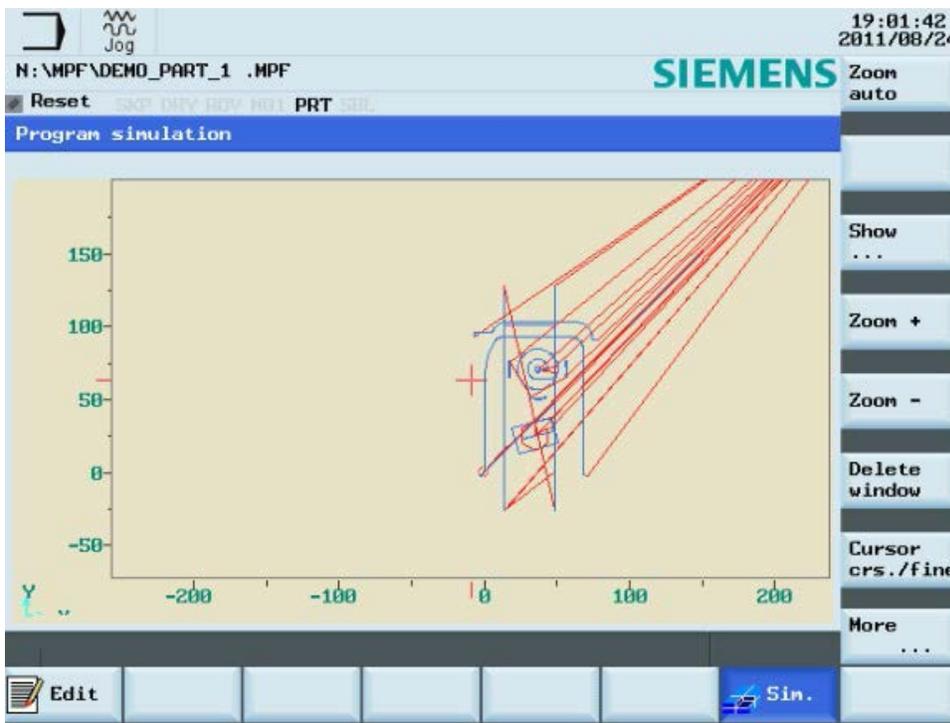
Simulate program

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



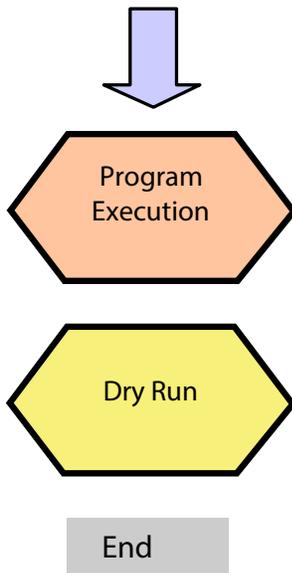
Test program

Test Program

Description

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

Content



Test program

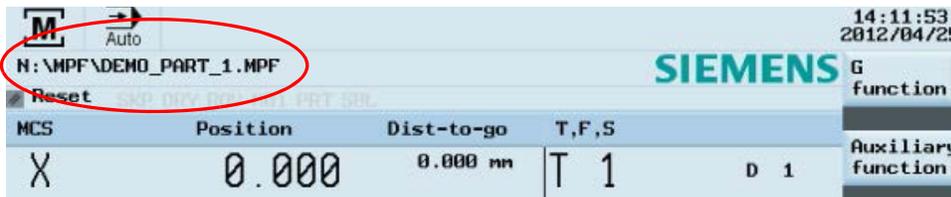
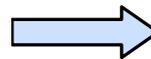
Basic Theory



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!



Test program

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.



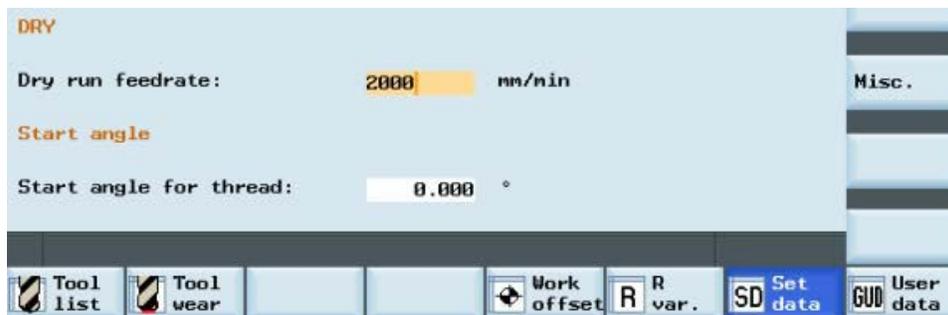
Press the "Sett. data" SK on the PPU.



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.

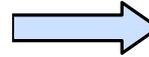


Press the "Input" key of the PPU.

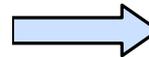


Test program

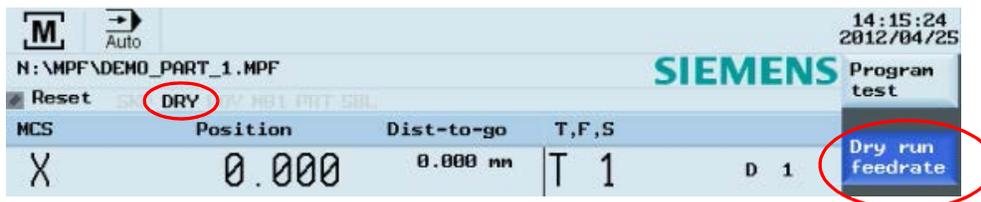
Press the "Machine" key on the PPU.



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.

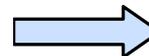


Step 2



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!



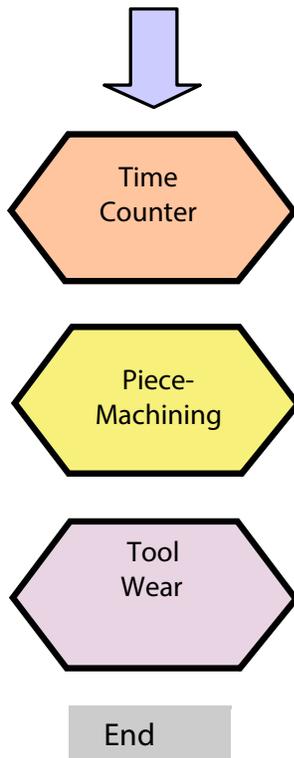
Machine pieces

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



Machine pieces

Basic Theory



Make sure the machine has been referenced before machining workpieces!

Step 1

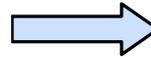
Press the "Machine" key on the PPU.



Press the "Auto" key on the MCP.



Press the "Time counter" SK on the PPU.



```
Block display          DEMO_PART_1.MPF
DN10 G17 G90 G54 G71 1
N20 SUPA G00 Z300 D01
N30 SUPA G00 X300 Y3001
N40 T1 D11
N50 MSG("Please change to Tool No 1")1
N60 M05 M09 M001
N70 S4000 M31
```

Time, counter		
Cycle time	0000:00:06h	
Time left	0000:00:00h	
Counter	No	↻



Machine pieces

SEQUENCE

“Cycle time” shows how long the program has been running.



Cycle time 0000:00:06h

“Time left” shows how much time remains before the program ends.



Time left 0000:00:00h

Step 2

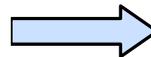


The “Time left” can only be counted after a successful cycle run of a part program!

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

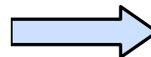


Enter the number of workpieces you require to be machined in “Required”.



Required 45

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



Actual 8

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

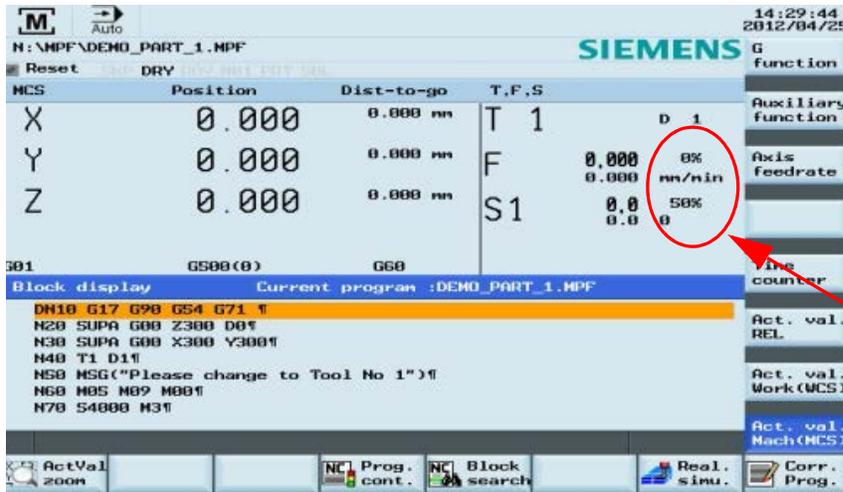
ActVal zoon	NC Prog. cont.	NC Block search	Real. sinu.
----------------	-------------------	--------------------	----------------



Machine pieces



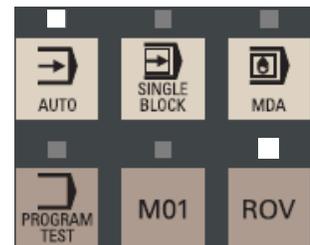
Make sure the program is correct before machining pieces!



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

Perform the relevant safety precautions!

Make sure that only "AUTO" mode and "ROV" mode are activated (or select the M01 function if required).



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



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

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



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



Machine pieces

SEQUENCE



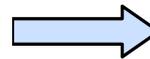
The tool wear compensation must distinguish the direction of compensation clearly!

Step 1

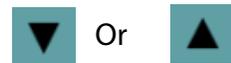
Press the "Machine" key on the PPU.



Press the "Auto" key on the MCP.



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



Tool wear				Active tool no 1 D 1	
Type	T	D	Wear		
			Length1	Radius	
	1	1	0.000	0.000	Milling tool
	2	1	0.000	0.000	
	2	1	0.000	0.000	Drilling tool
	3	1	0.000	0.000	
	4	1	0.000	0.000	Tapping tool
	5	1	0.000	0.000	
	6	1	0.000	0.000	Ball end mil. tool
	7	1	0.000	0.000	
	8	1	0.000	0.000	

SD Sett. data



Machine pieces

Step 2

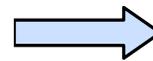
Set the tool length wear parameter of axis X in "Length X", the sign determines the direction of wear compensation.

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 work piece (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.



Type	T	D	Wear	
			Length1	Radius
Milling tool	1	1	0.220	1.200
	2		0.000	0.000
Drilling tool	2	1	0.000	0.000
	3	1	0.000	0.000
Tapping tool	4	1	0.000	0.000
	5	1	0.000	0.000
Ball end mil. tool	6	1	0.000	0.000
	7	1	0.000	0.000
	8	1	0.000	0.000



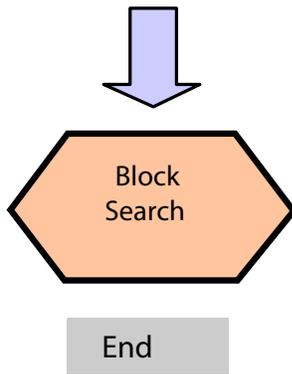
Program restart

Program Restart

Description

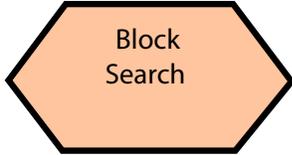
This unit describes how to restart the part program after a tool has been changed due to damage, or remachining has to be performed.

Content

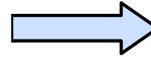


Program restart

Basic Theory



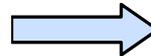
Press the "Machine" key on the PPU.



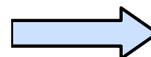
Press the "Auto" key on the MCP.



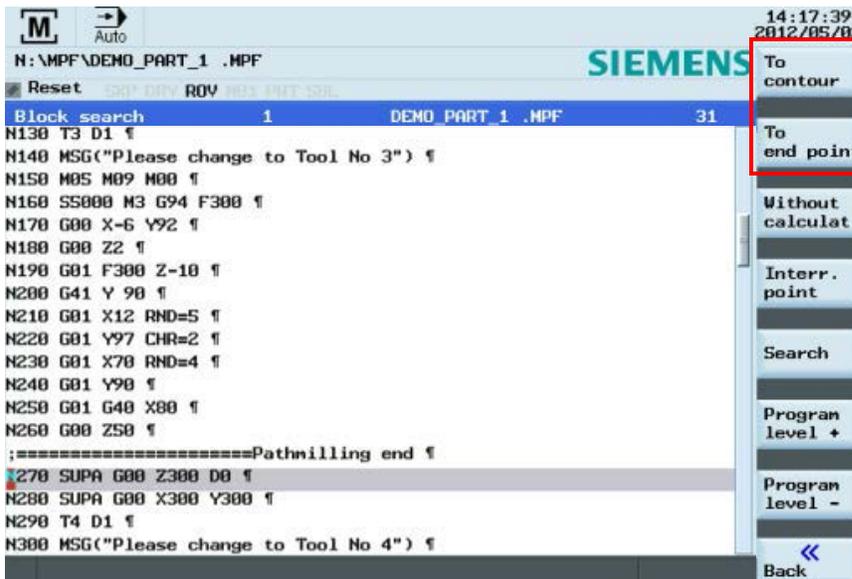
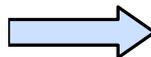
Press the "Block search" SK on the PPU.



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.

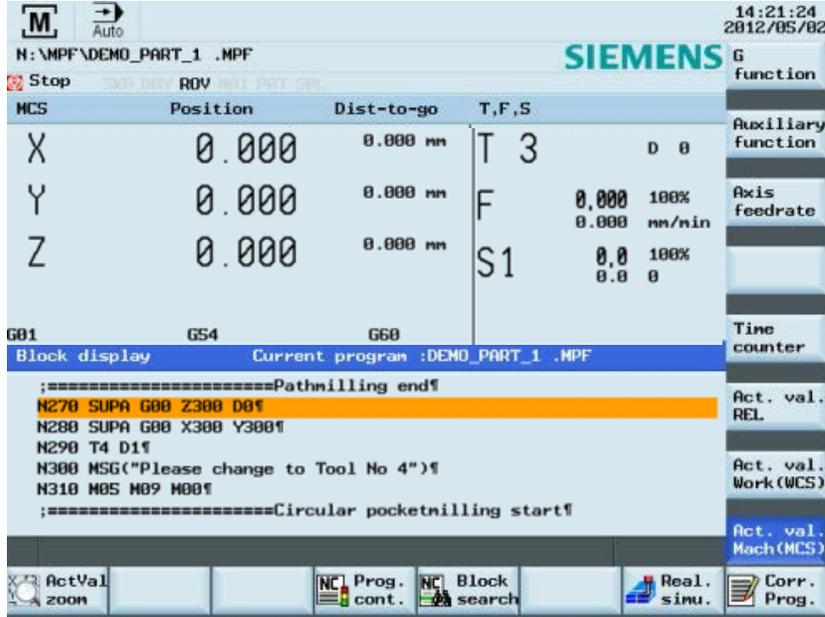
"To end point": The program will continue from the line with the breakpoint.

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



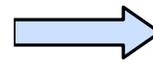
Program restart

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.

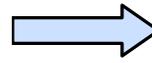


Alarm 010208 is shown at the top prompting to press the "CYCLE START" key to continue the program.



Program restart

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



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

The screenshot shows the Siemens CNC control interface. At the top, there is a status bar with 'M' in a box, 'Auto' mode, and a message 'Please change to Tool No 4'. Below this, the program path is 'N:\MPF\DEMO_PART_1.MPF'. The main display area is divided into several sections:

- SIEMENS G function:** Shows 'RUN' mode and 'SKIP DRV ROV NO1 PART 500'.
- Position and Dist-to-go:** A table showing current positions and distances to go for X, Y, and Z axes.
- T,F,S:** Shows tool number (T 4), feedrate (F 300.000), and spindle speed (S 1).
- Time counter:** Shows '303' and 'G54'.
- Block display:** Shows the current program 'DEMO_PART_1.MPF' and a list of program blocks (N370 to N430).
- Act. val.:** Shows 'REL' and 'Mach (MCS)'.

At the bottom, there are several icons and buttons: 'ActVal zoom', 'Prog. cont.', 'Block search', 'Real. sinu.', and 'Corr. Prog.'.

MCS	Position	Dist-to-go	T,F,S
-X	0.175	-8.691 mm	T 4
+Y	61.344	-6.664 mm	F 300.000 100%
Z	-0.267	0.000 mm	S 1 500.0 100%
			5000.0 0
303	G54	G64	



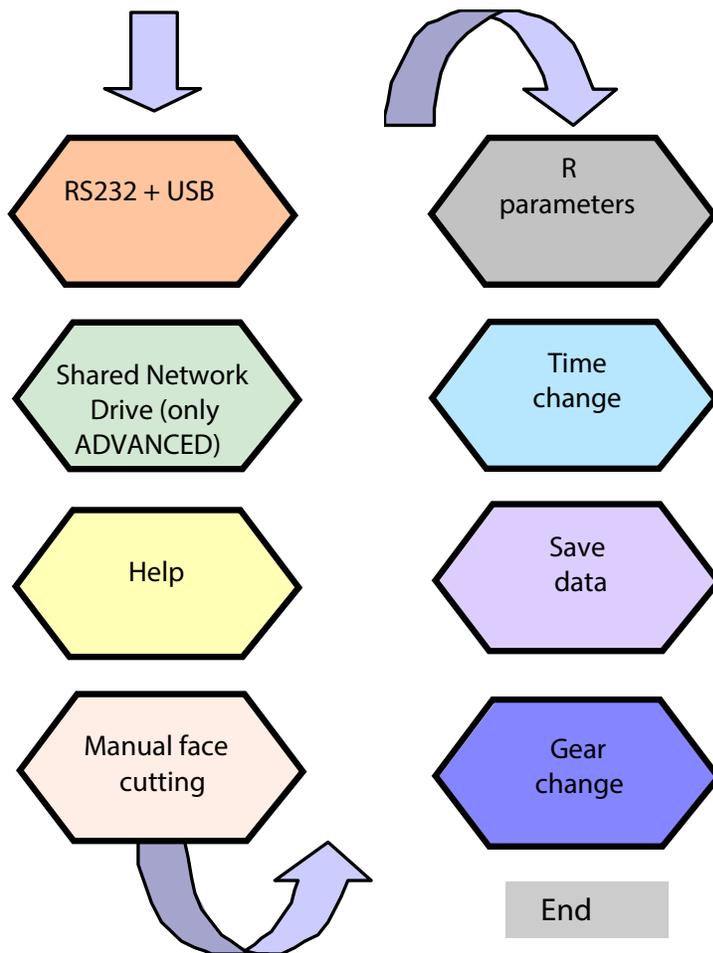
Additional information: part 1

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



Additional information: part 1

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.

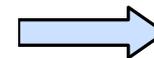


Settings

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



Press the "Save" SK on the PPU.



Save

Press the "Back" SK on the PPU.

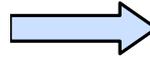


Additional information: part 1

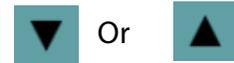
SEQUENCE

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.)

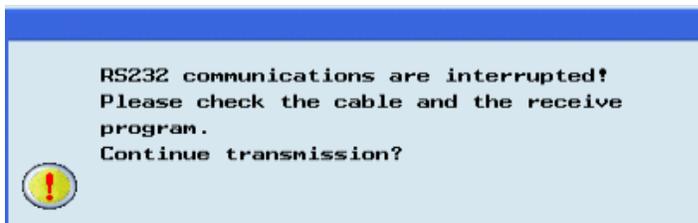
Press the "Send" SK on the PPU.



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.



Additional information: part 1

You can continue sending the part program.

Press the "OK" SK on the PPU.



Or you can abort the sending of the part program.

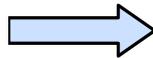
Press the "Cancel" SK on the PPU.



Step 3

Transfer a part program to the PPU from a PC.

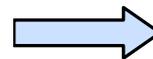
Press "Program Manager" on the PPU.



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.



Additional information: part 1

SEQUENCE



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

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.

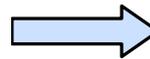
Press the “NC” SK on the PPU.



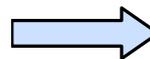
Use “Cursor + Select” to select the required part program.
The selected program will be highlighted.



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.

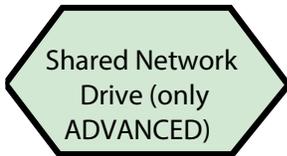


Press the “Paste” SK on the PPU.



Additional information: part 1

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

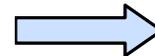
Press key:



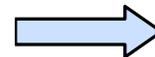
Press key:



Press "Serv. Displ." SK



Press "Serv. Displ." SK



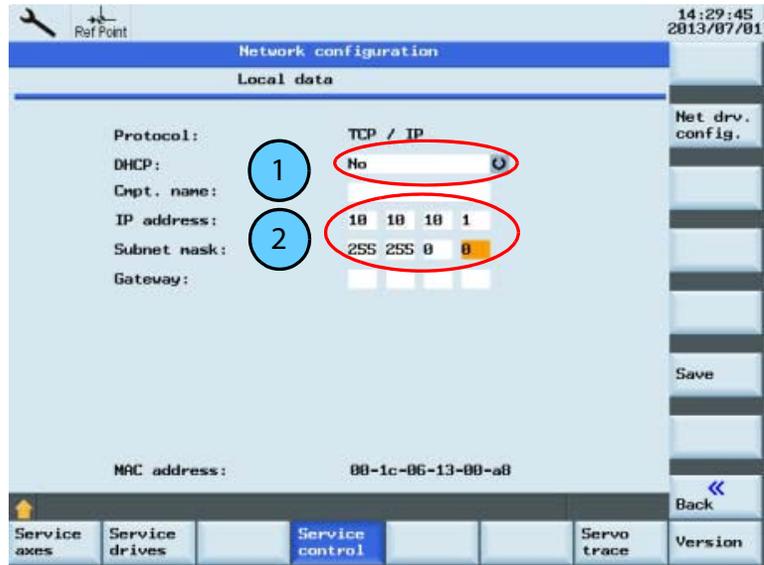
Press "Network Info" button to enter the "Local Configuration Data"



Additional information: part 1

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

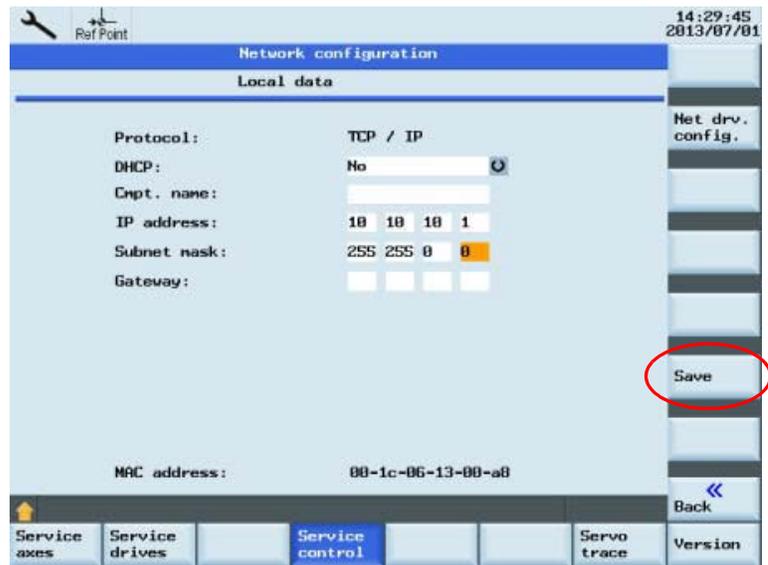
- ① DHCP is set to "No"
- ② IP address and subnet mask can be arbitrarily set according to need. (Right given only as an example)



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



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



Additional information: part 1

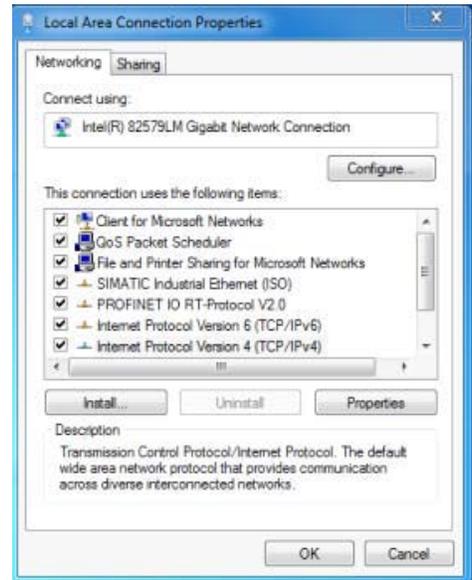
SEQUENCE

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)"

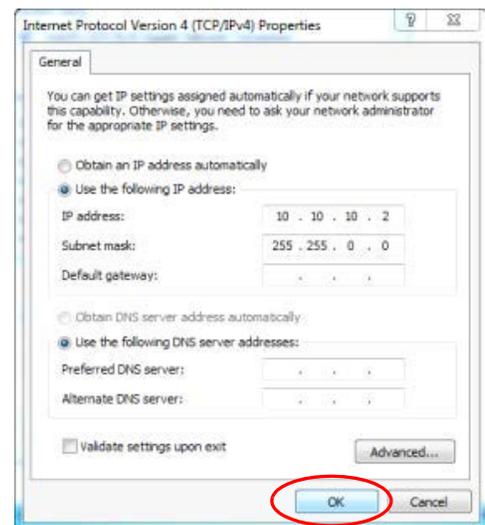
And double click "Properties".



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.

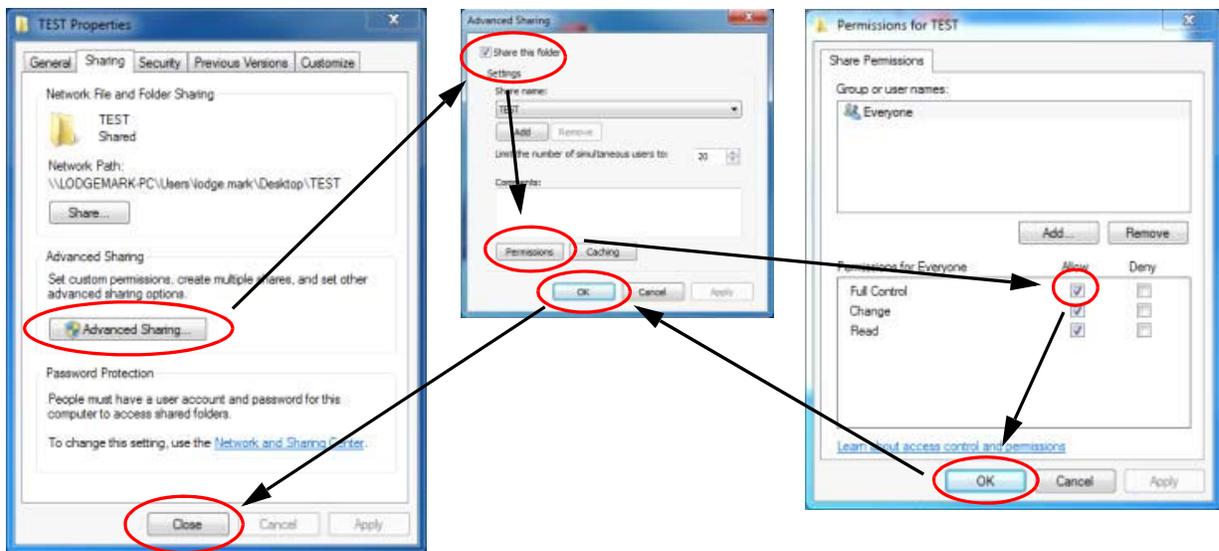
PPU and PC IP address should be kept in the same network segment.



Additional information: part 1

Step 3 On PC create a shared folder.

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



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

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

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



Additional information: part 1

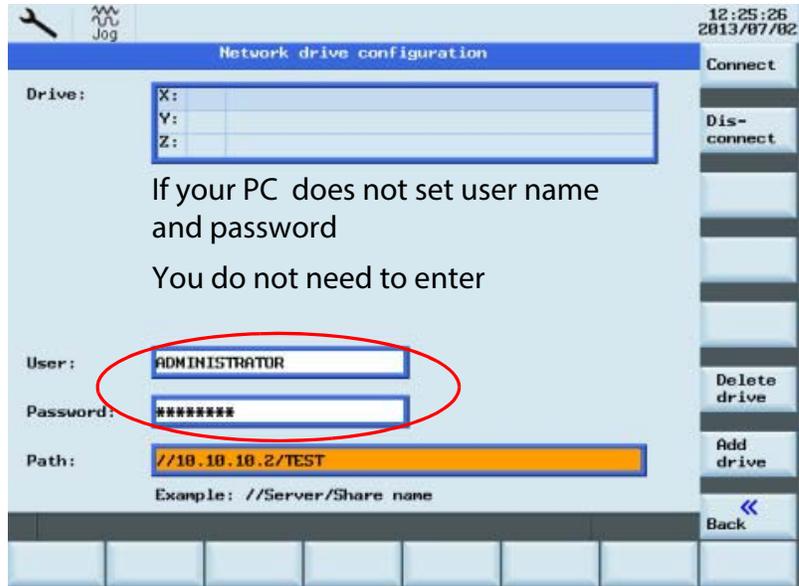
SEQUENCE

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

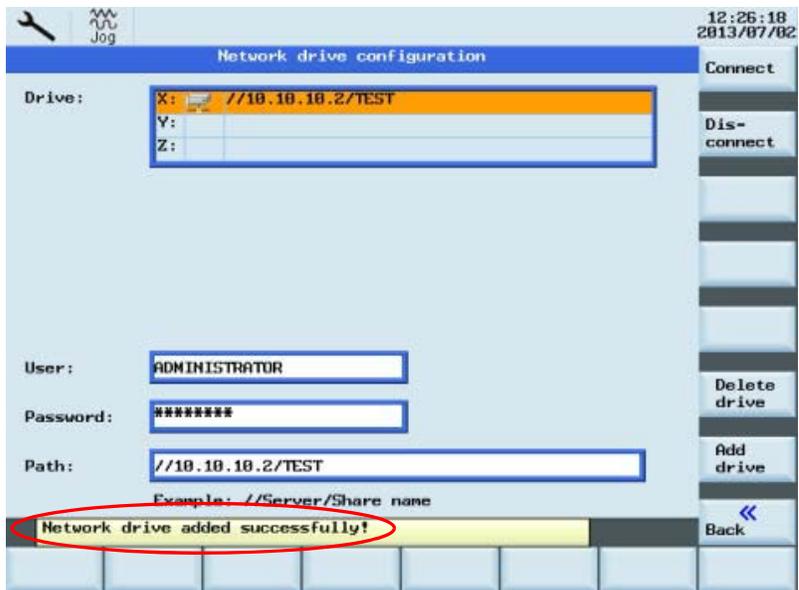
Note: Use "TAB" key to switching between different tasks boxes.



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.



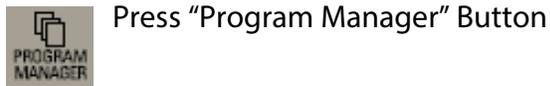
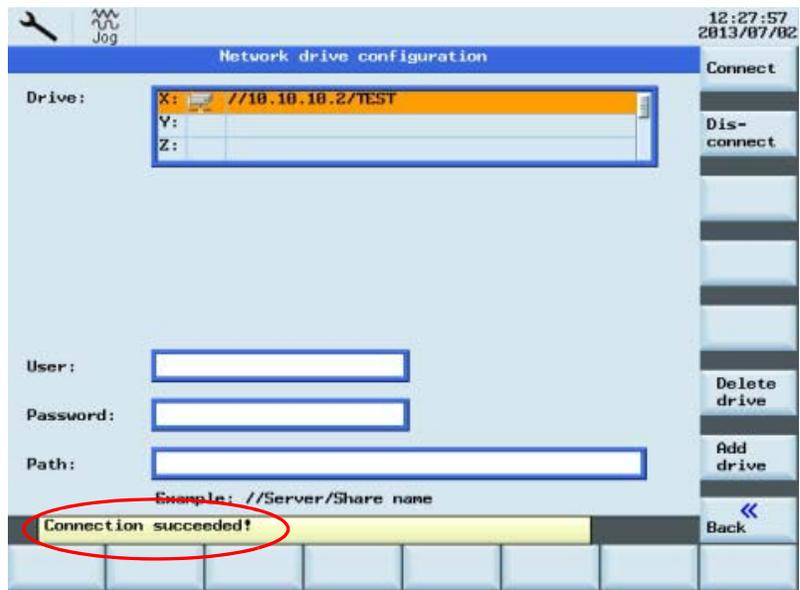
Additional information: part 1

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



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

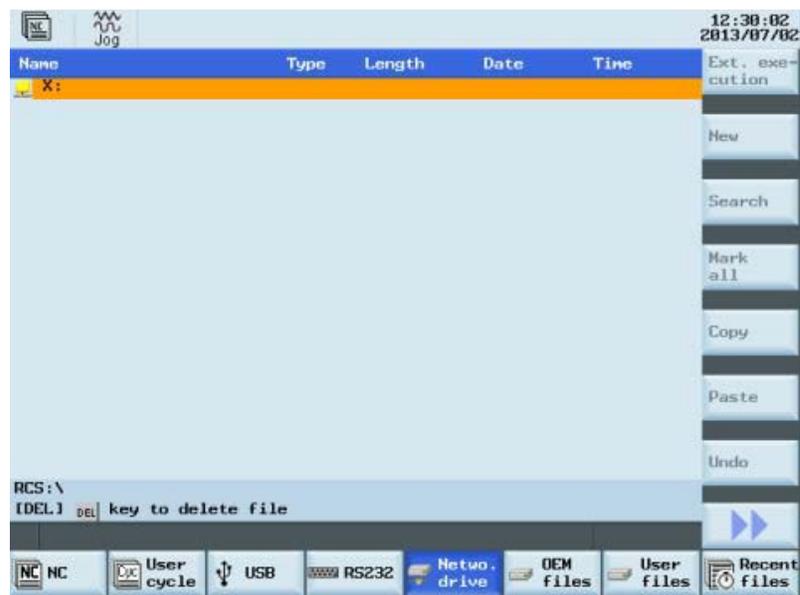
This will be shown with the text "Connection succeeded"



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



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



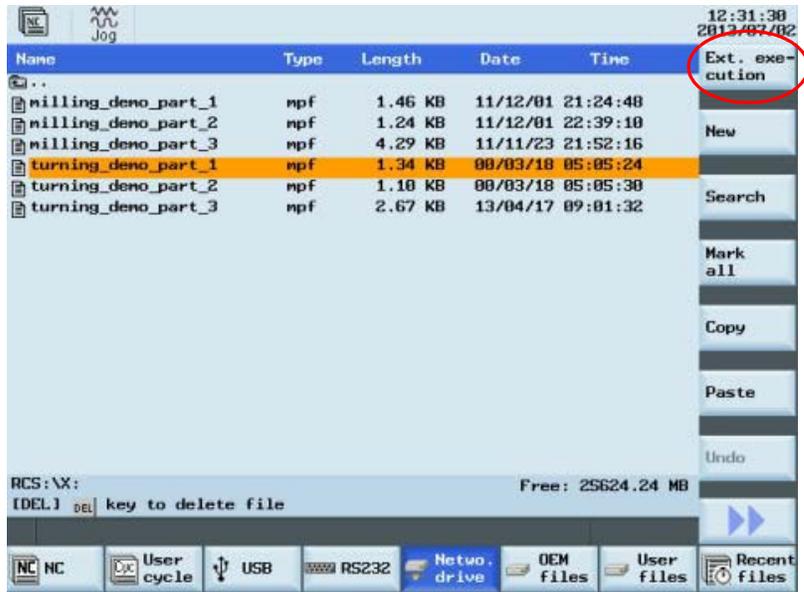
Additional information: part 1

SEQUENCE

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

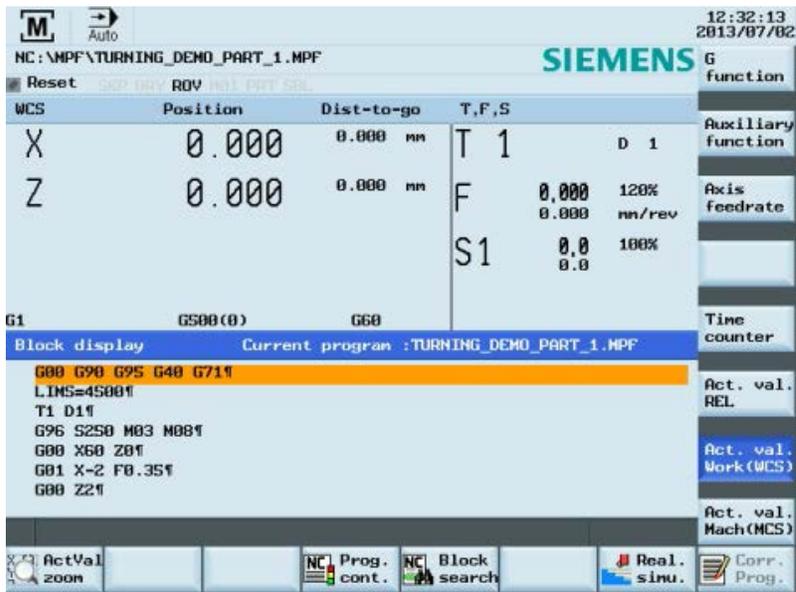
You can select the file you require to execute in AUTO mode, click "Ext. Execution".

Ext. execution



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

Press the "Cycle Start" button for machining operation.



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



Additional information: part 1



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.

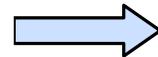


Press the "Help" key on the PPU.



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.

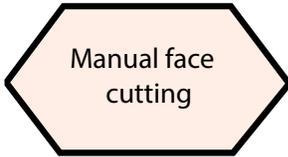
Press the "TOC" SK on the PPU.



The online help from the Siemens manual will be shown.



Additional information: part 1



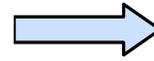
“Face cutting” is used to cut the oversized materials on the rough face before starting to machine.

Step 1

Press the “Machine” key on the PPU.



Press the “JOG” key on the MCP.

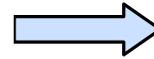


Press the “Sett.” SK on the PPU.



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

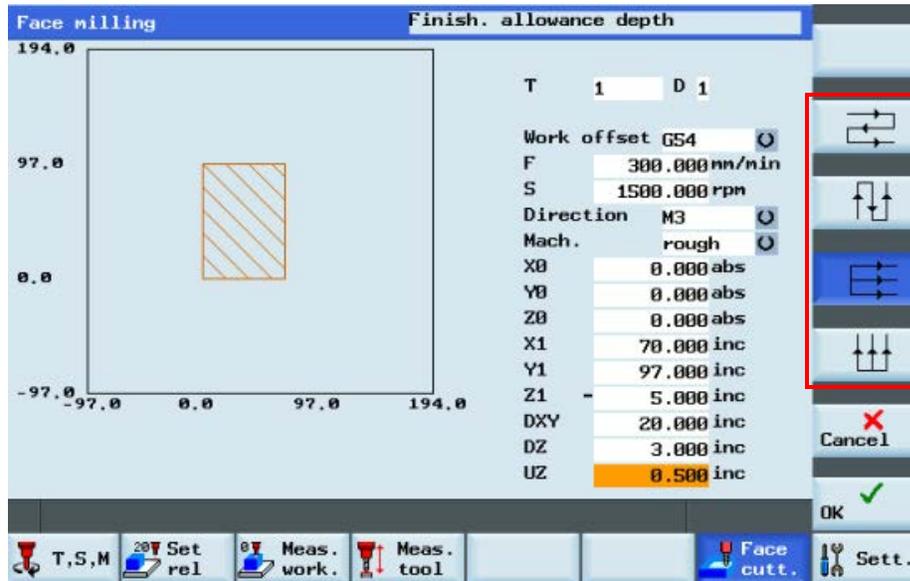
Press the “Input” key on the PPU to activate the settings.



Additional information: part 1

Step 2

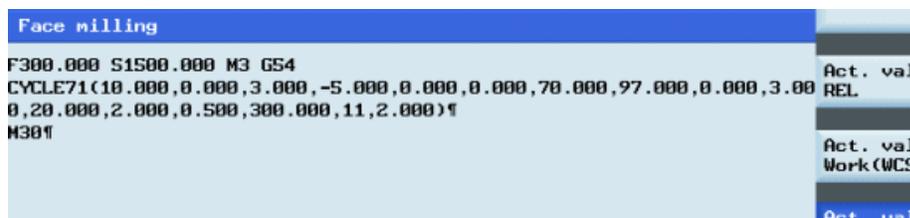
Press "Face cutt." SK on PPU.



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.

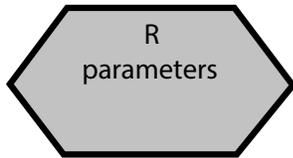


Adjust the override on the MCP gradually to the required values.



Additional information: part 1

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).



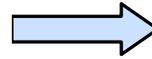
Additional information: part 1

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



Additional information: part 1

SEQUENCE



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

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

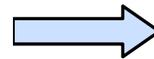


Make sure the password is set to the "CUSTOMER" access level.

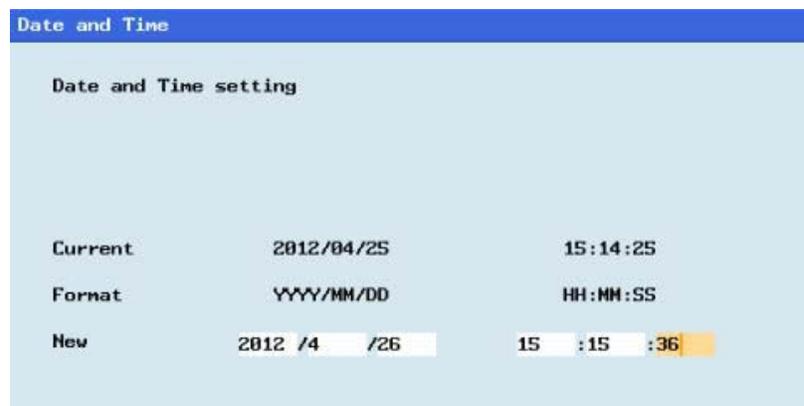
Press the "HMI" SK on the PPU.



Press the "Date time" SK on the PPU.



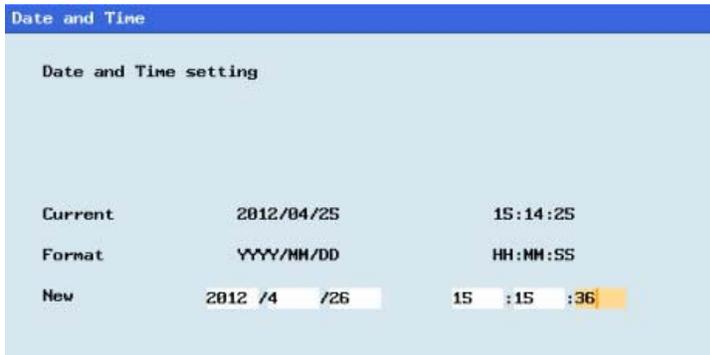
Enter a new "Date" and "Time".



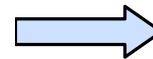
Press the "OK" SK on the PPU.



Additional information: part 1



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



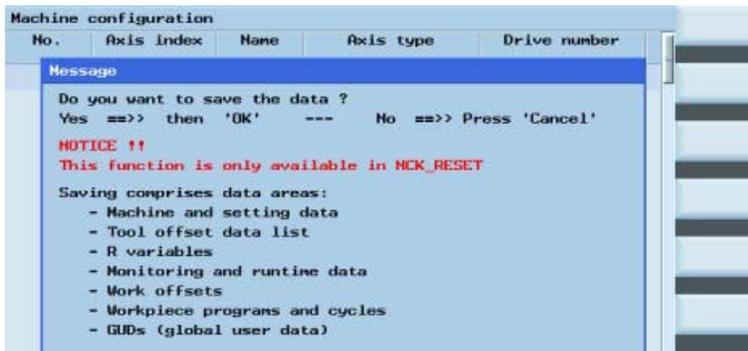
"Save data" enables the complete system to be backed up on the system CF card so that there is a system backup available to the operator.

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

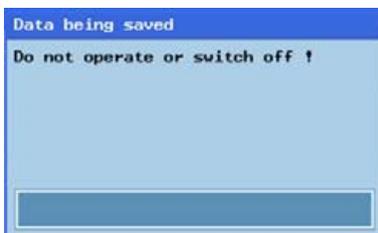
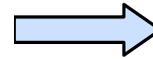


Make sure the password is set to the "CUSTOMER" access level.

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!



Additional information: part 1

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.



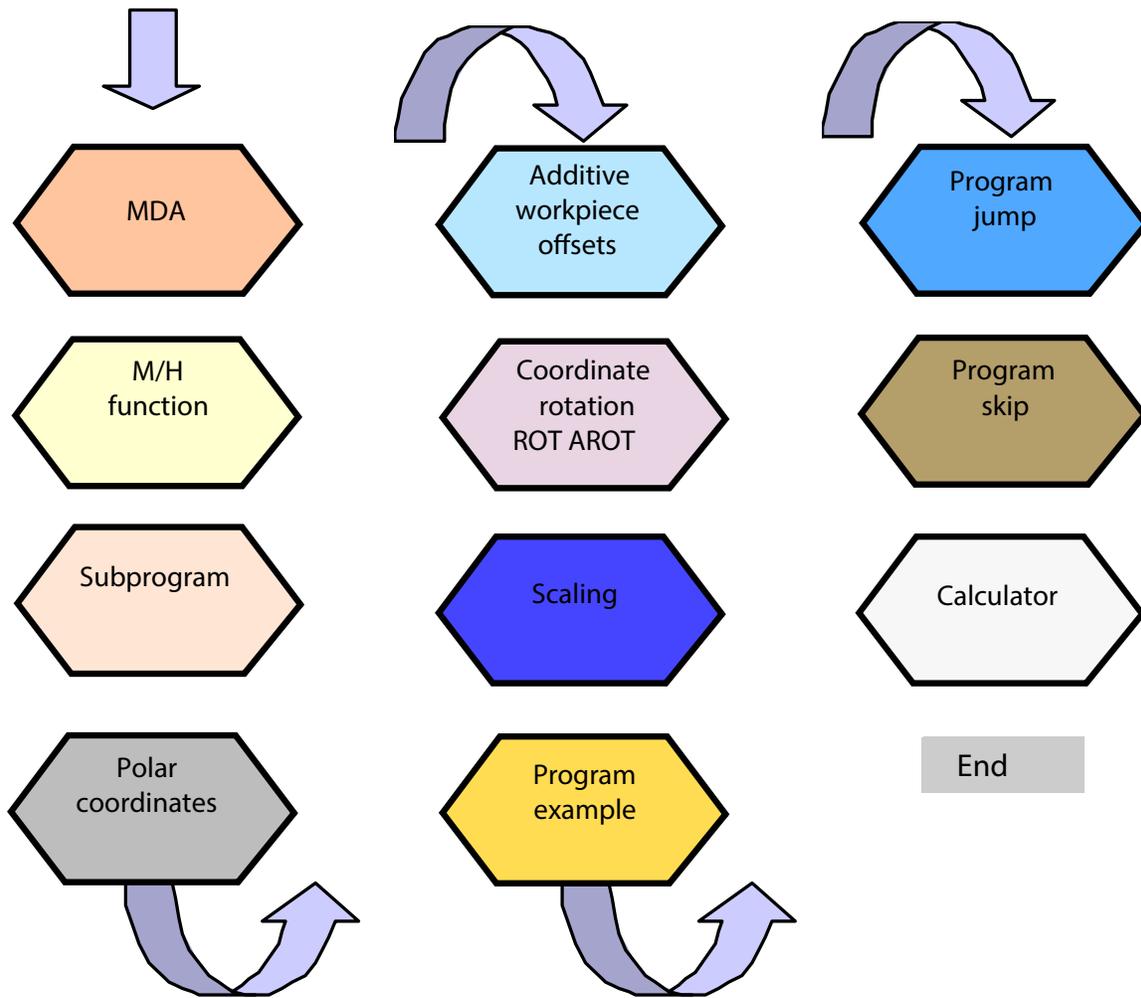
Additional information: part 2

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



Additional information: part 2

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.

The screenshot shows the Siemens CNC control interface in MDA mode. The top bar displays the 'M' key icon and 'MDA' mode. The main display area shows the following data:

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

Below the table, the program block is shown: G00 G54 G60. The status bar at the bottom indicates 'MDI - Block' and the current block: 'G54 X10 Y10 Z50'. The feedrate override is set to 0%.



Additional information: part 2

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.

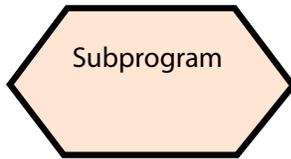
With H functions, the meaning of the values of a specific H function is defined by the machine tool manufacturer.

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		

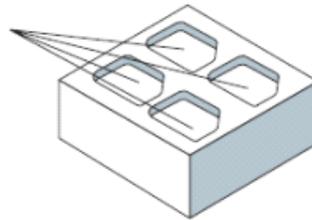


Additional information: part 2



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:

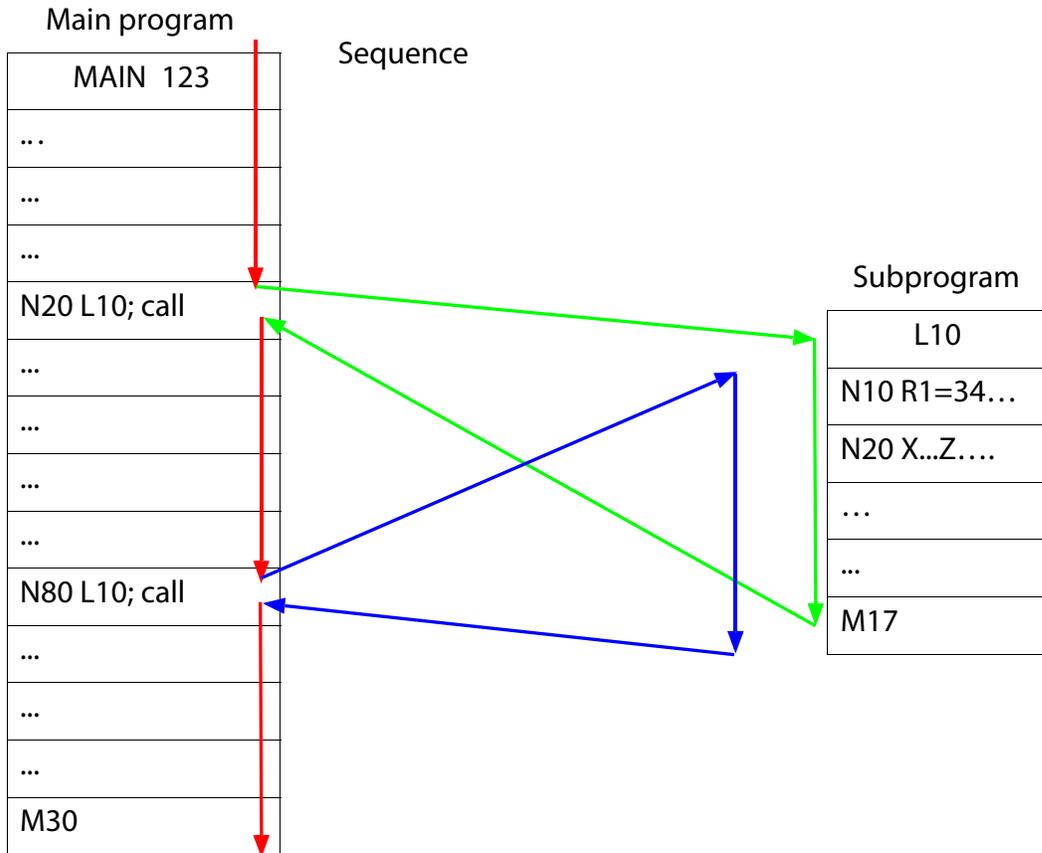
The name can contain letters, numbers and underscores and should be between 2 and 8 characters long.

Example: LRAHMEN7



Additional information: part 2

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.



Additional information: part 2



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.



Additional information: part 2

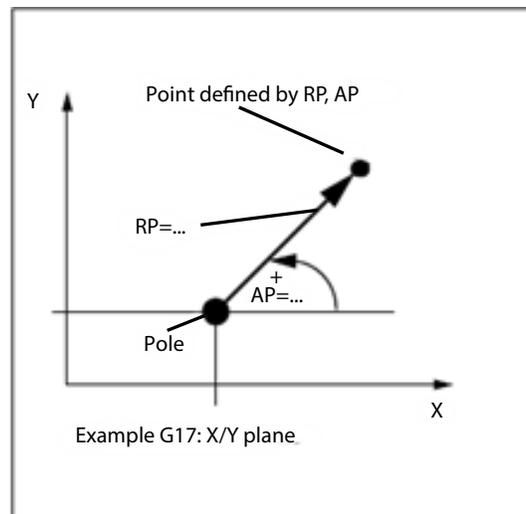
Basic Theory

- 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)
  
```



Additional information: part 2



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

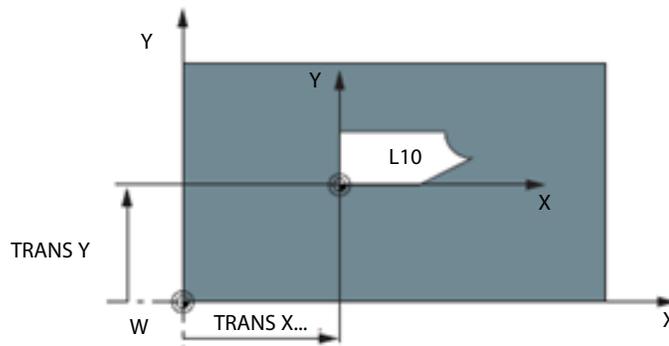
- For recurring shapes/arrangements in various positions on the workpiece.
- When selecting a new reference point for dimensioning.

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

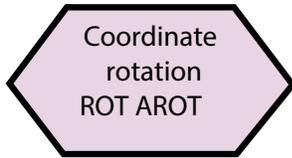
Programming example

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



Additional information: part 2

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).

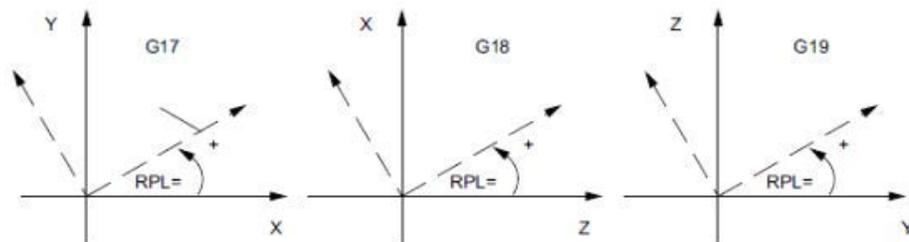
AROT RPL=... ; programmable offset, additive to existing offset (incremental)

ROT ; without values, clears old commands for offset

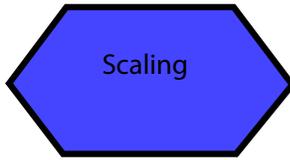
N10 G17

N20 AROT RPL=45 additive 45 degree rotation

L10 subprogram call



Additional information: part 2



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)

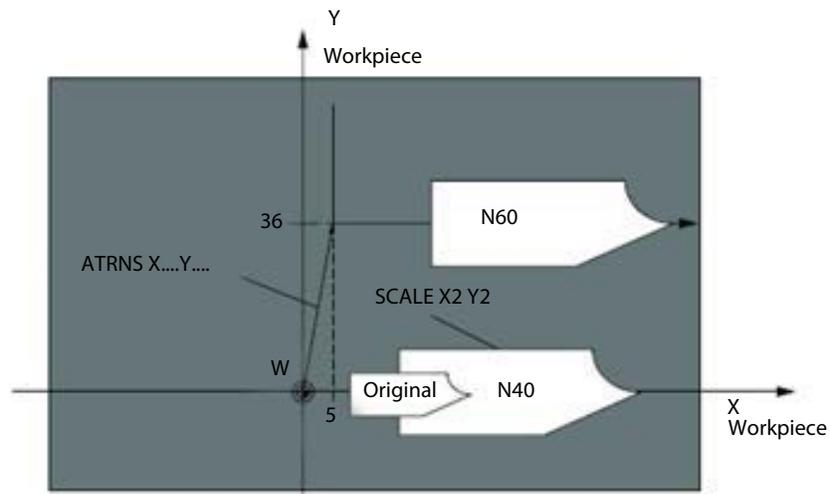
If a program contains SCALE or ASCALE, this must be programmed in a separate block.

Programming example

N10 G17

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

L10 ; subprogram call



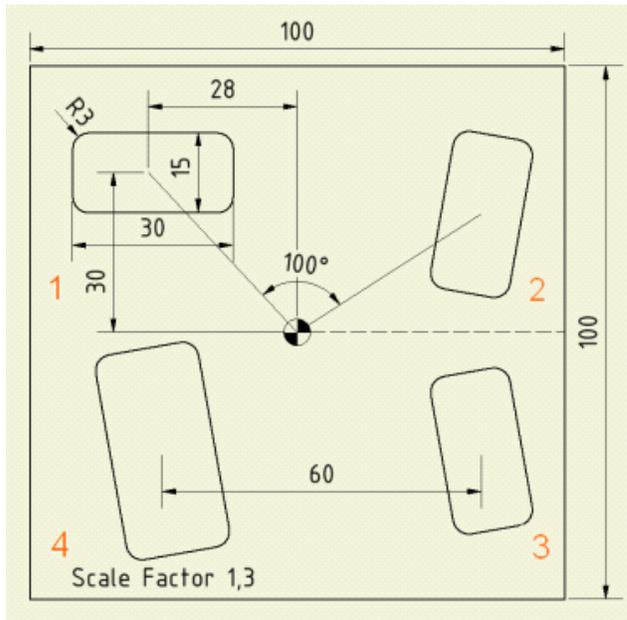
Additional information: part 2

SEQUENCE



This describes and analyzes the additive offsets, 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

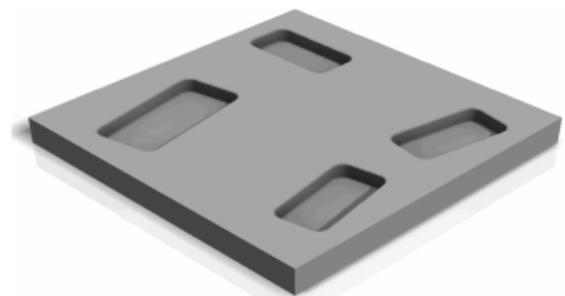
② Coordinate rotates 20°

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!



Additional information: part 2

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 5mm, 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



Additional information: part 2

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)

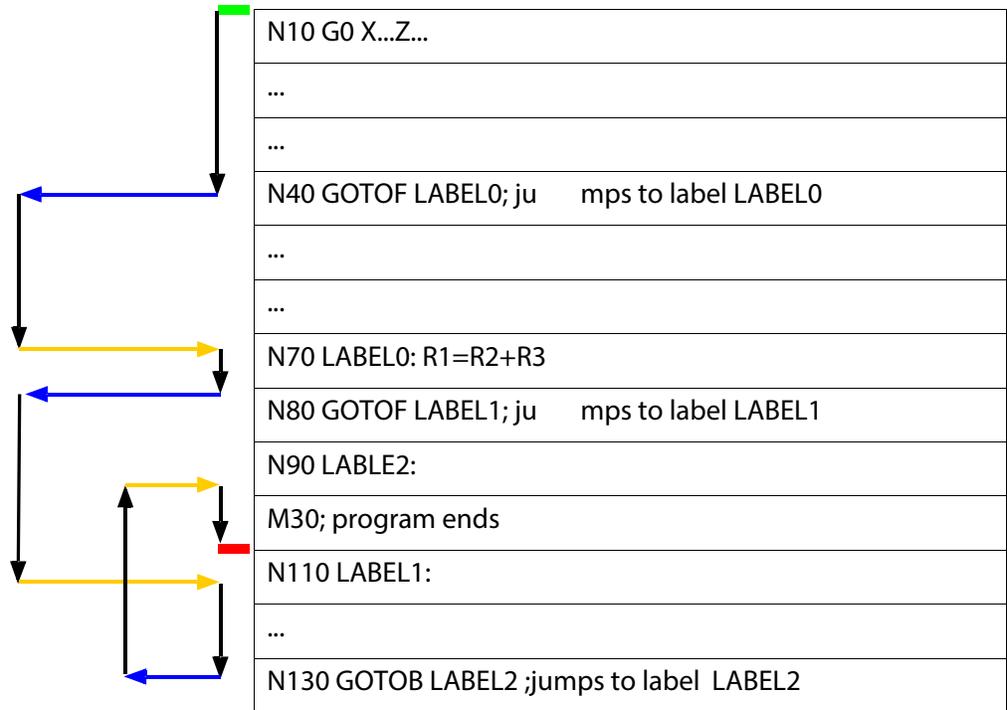
GOTOB+ label: Jump backward (in the direction of the start block of the program)

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



Additional information: part 2

Program execution



Unconditional jump example



Additional information: part 2

SEQUENCE



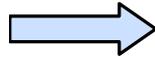
Method 1

“;” 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.

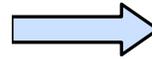
Using “;” code to add a remark to the N85 function, without any influence on the execution.



Additional information: part 2

Method 2

Press the "Machine" key on the PPU.



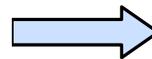
Press the "Auto" key on the MCP.



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



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



Skip

The screenshot shows the Siemens CNC control interface. At the top, the 'MACHINE' and 'AUTO' keys are visible. The 'SKP' key is highlighted with a red circle. The main display shows the current program 'TEST.MPF' and the following table:

MCS	Position	Dist-to-go	T,F,S	D	Feedrate	Condition
X	30.000	0.000 mm	T 1	1		Dry run
Y	156.000	0.000 mm	F	0.000	100%	Condit. stop
Z	0.000	0.000 mm	S1	0.0	100%	Skip

Below the table, the program string is displayed: 'G01 G500(0) G60 /Z30X100 Y156 M30'. The string '/Z30X100' is highlighted with a purple circle. The bottom of the screen shows various function keys like 'ActVal', 'NC Prog. cont.', 'Block search', 'Real. sinu.', and 'Corr. Prog.'.

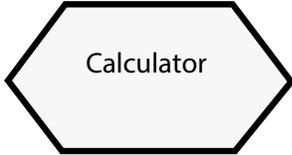
When "SKP" is displayed (red circle), the skip function has been activated.

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



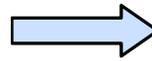
Additional information: part 2

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.



Additional information: part 2



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.



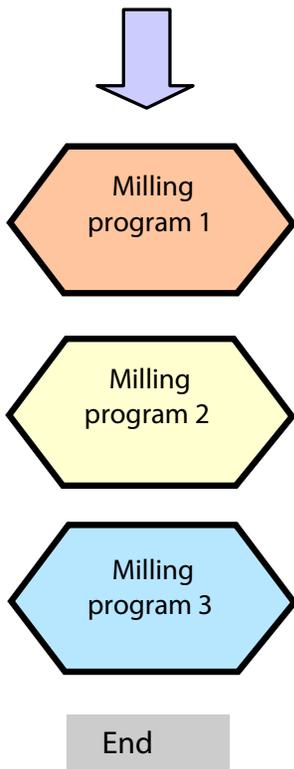
Sample program

Sample Program

Description

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

Content

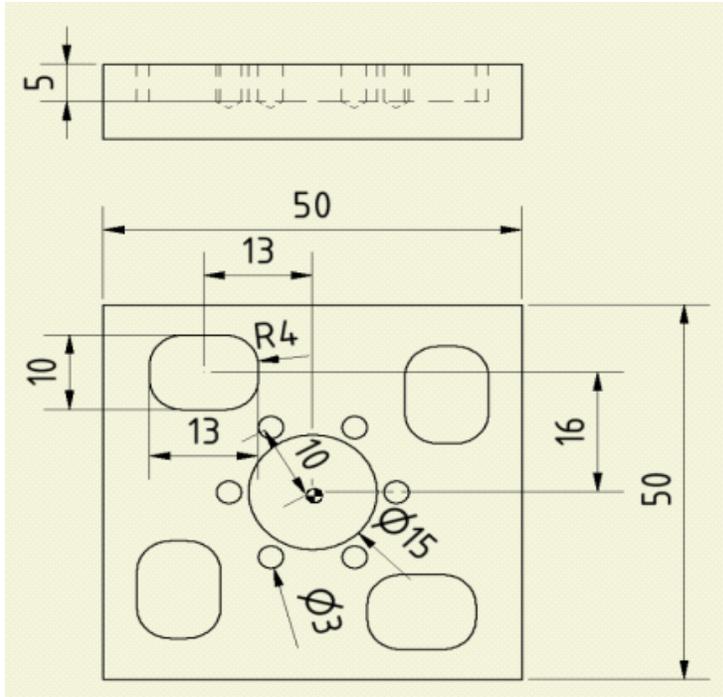


Sample program

Drawing



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

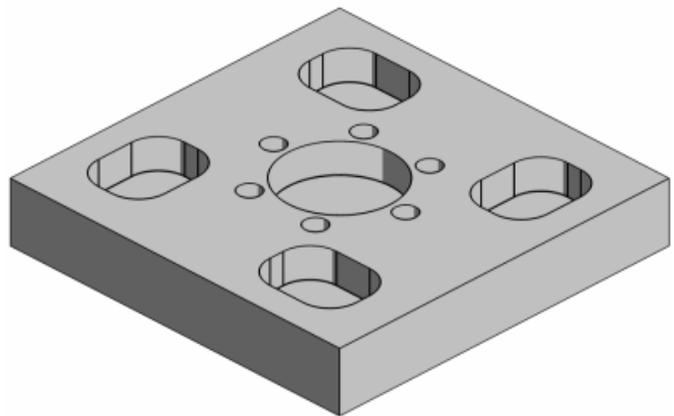


The zero point of the workpiece is located at the center point of the workpiece

Tool information:

T1 Milling tool D50

T2 Milling tool D8



Sample program

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,
length 13 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
```



Sample program

```

;=====Repeat rectangular pocket milling 3
times=====
N200 R EPEAT _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=====

```



Sample program

Machining Process

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

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

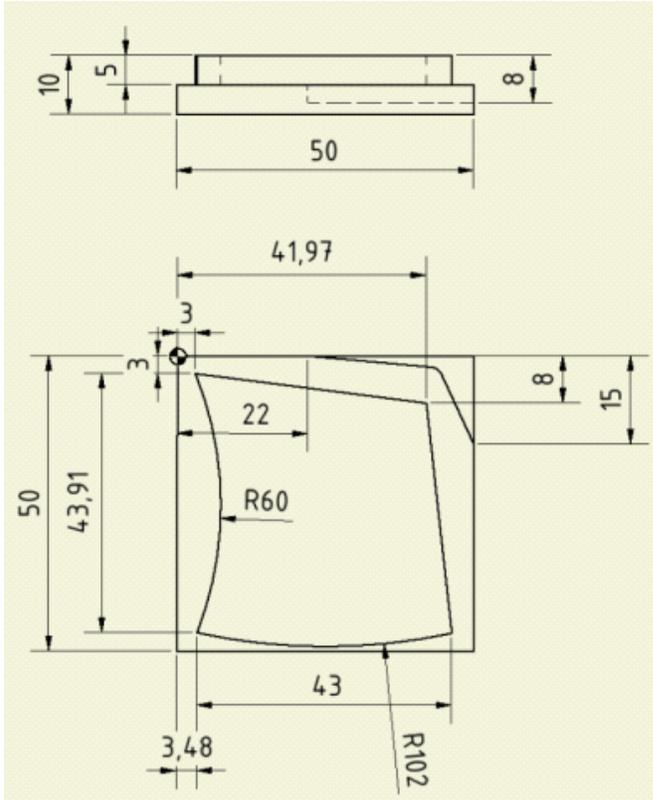


Sample program

Drawing



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



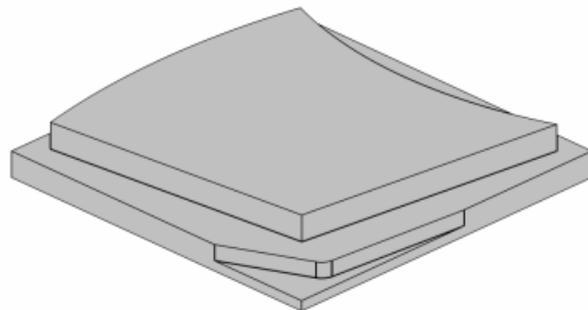
Workpiece zero point is located in the top left corner.

Tool information:

T1 Milling tool D50

T2 Milling tool D12

T4 Milling tool D10



Sample program

Machining Process

```
N10 G17 G90 G60 G54
N20 T1 D1 ;FACEMILL D50
N30 M6
N40 S3500 M3
N50 G0 X0 Y0
N60 G0 Z2
;=====Start face milling=====
N70 CYCLE71( 50, 1, 2, 0, 0, 0, 50, -50, , 1,
40, , 0.1, 300, 11, )
N80 S4000 M3
N90 CYCLE71( 50, 0.1, 2, 0, 0, 0, 50, -50, , 1,
40, , 0, 250, 32, )
;=====Start contour milling=====
N100 T2 D2 ;ENDMILL
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 too I radius compensation
N240
```



Sample program

SUB_PART_2.SPF

CONTOUR

```
G1 7 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 */
```

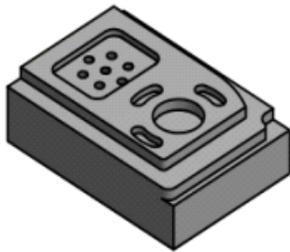
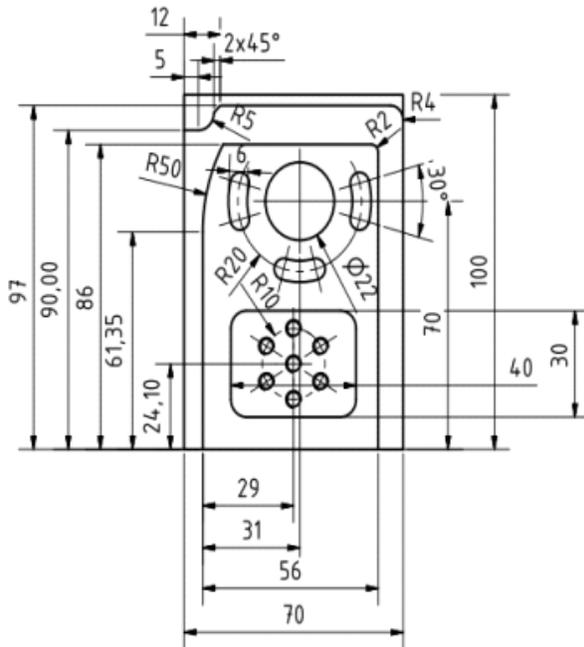
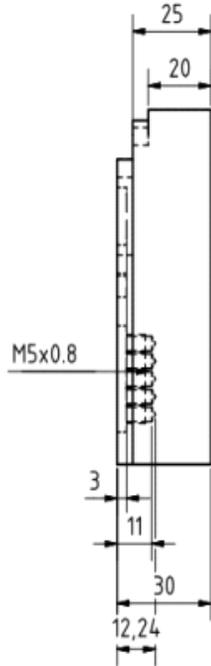


Sample program

Drawing



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



Tool information

T1 Milling tool D50

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



Sample program

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=====
```



Sample program

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====
```



Sample program

```

N430 SU PA 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 po sitive direction, rough
machining, use G1 vertical groove center to insert.
N570 repeat N600 milling process, the difference
is the machining allowance.
;==Rectangular pocket milling end==

```



Sample program

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,
depression 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=====
```



Sample program

```
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, 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
```



Sample program

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 */



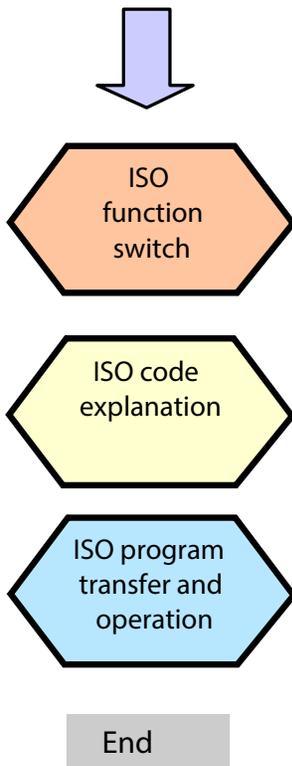
ISO Mode

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 the ISO mode chapter can be run in 808D ISO mode.

Content



ISO Mode

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

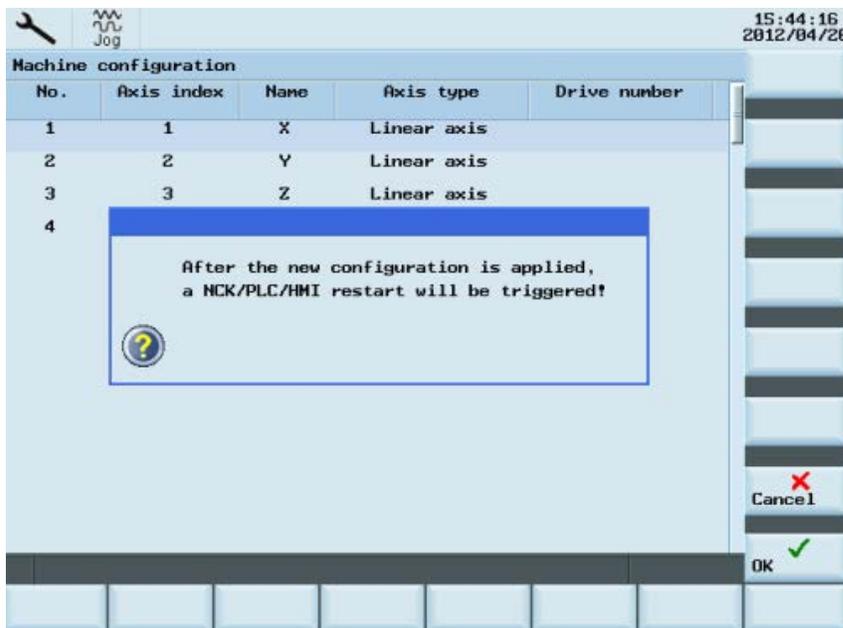
Press the "Shift" + "System - Alarm" keys on the PPU.
Input the manufacturer' s password ("SUNRISE")



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.

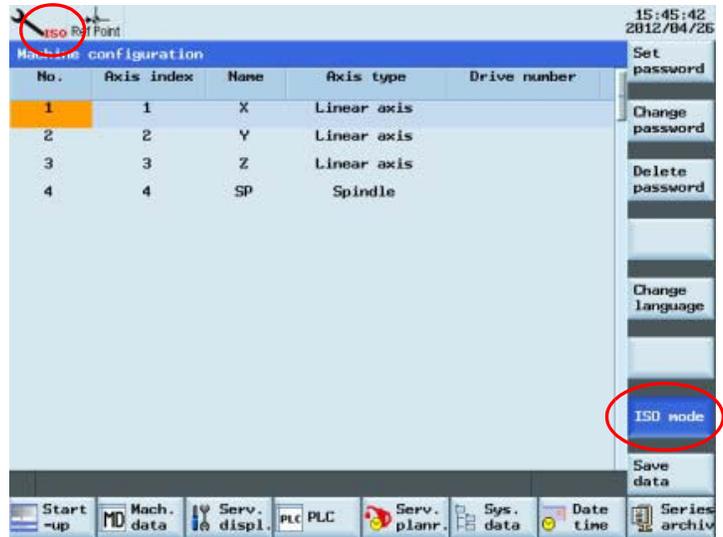


ISO Mode

Basic Theory

After pressing "OK", the system restarts automatically.

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



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

Method 2



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

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

```

N0 G291
N5 G17 G90 G54 G71 F1
N20 T1 H1
N25 MSG("Tool No 1 in use")
N35 S4000 M3
N40 CYCLE71( 50.00000, 2.00000, 2.00000, 0.00000, 0.00000, 0.0000
N45 S4500 M3
    
```

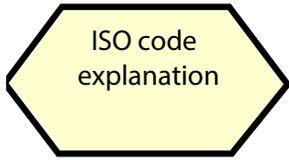


G291/G290 commands must be set separately in a line!

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



ISO Mode



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



ISO Mode

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



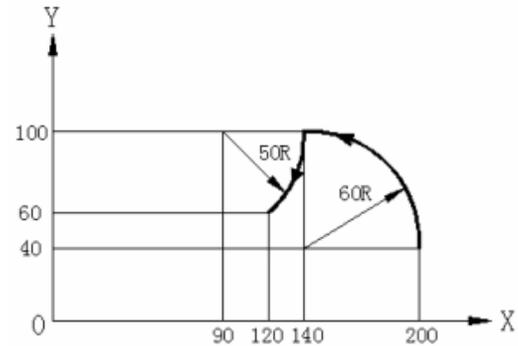
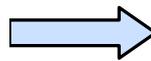
ISO Mode

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

G02 circular interpolation in positive direction

G03 circular interpolation in negative direction

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



Method 1 (use incremental to describe circular radius)

```
G92 X200.0 Y40.0 Z0
G90 G03 X140.0 Y100.0 I-60.0 F300.0
G02 X120.0 Y60.0 I-50.0
```

Method 2 (use parameter R to describe circular radius)

```
G92 X200.0 Y40.0 Z0
G90 G03 X140.0 Y100.0 R60.0 F300
G02 X120.0 Y60.0 R50.0
```

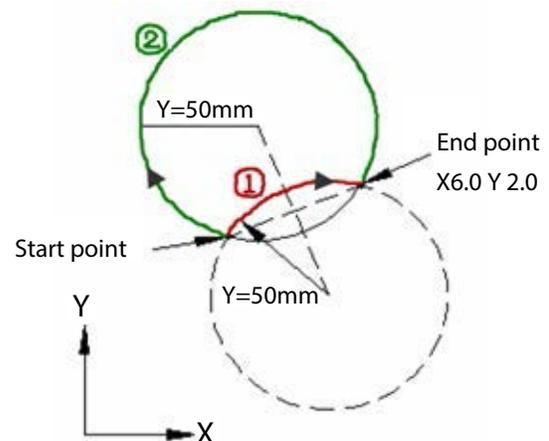
When specifying circle radii with parameter **R**

Circles less than 180° is assigned positive values

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**



ISO Mode

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 in 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 in 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



ISO Mode

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
```



ISO Mode

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

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

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 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
```



ISO Mode

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

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

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 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
```



ISO Mode

Basic Theory

G84 Tapping cycle

Common programming structures:

G84 X—Y—Z—R—P—F—K

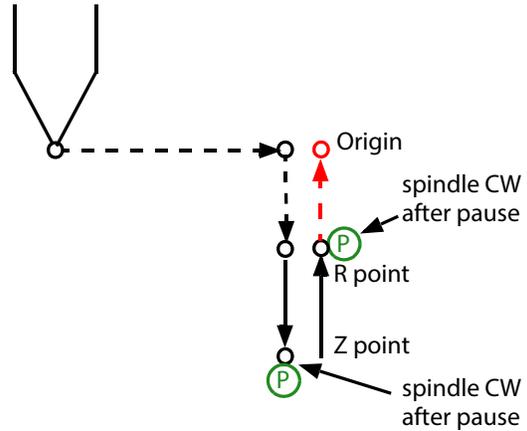
Motion process:

- 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:

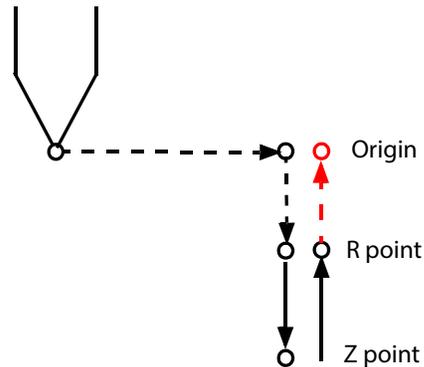
- ① Drilling motion (-Z) -> cutting feed
- ② Motion at the bottom of the hole -> none
- ③ Retraction motion (+Z) -> cutting feed

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**



ISO Mode

G86 boring cycle

Common programming structures:

G86 X—Y—Z—R—F—K

Motion process:

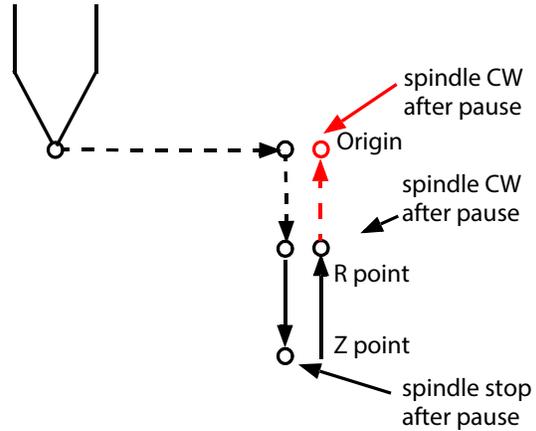
- ① Drilling motion (-Z) -> cutting feed
- ② Motion at the bottom of the hole -> spindle stop
- ③ Retraction motion (+Z) -> fast feed

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:

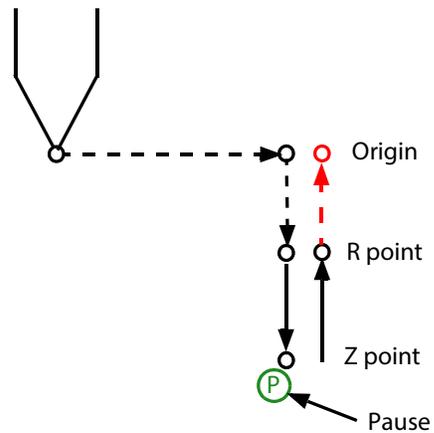
- ① Drilling motion (-Z) -> cutting feed
- ② Motion at the bottom of the hole -> pause
- ③ Retraction motion (+Z) -> cutting feed

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



ISO Mode

Basic Theory

G87 Boring cycle I / reverse boring cycle II

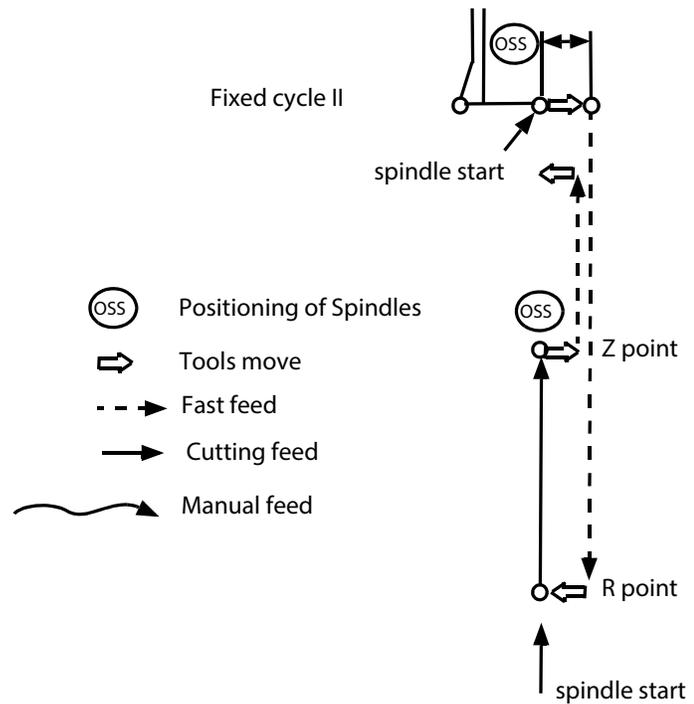
Common programming structures:

G87 X—Y—Z—R—Q—P—F—L

Motion process:

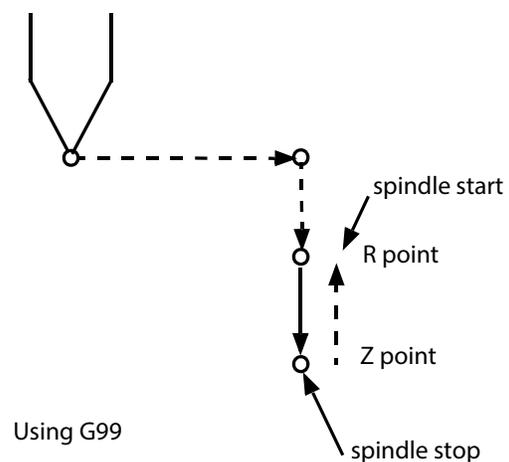
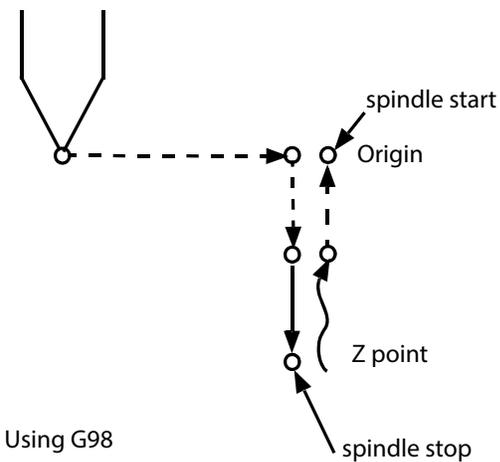
- ① Drilling motion (-Z) -> cutting feed
- ② Motion at the bottom of the hole -> spindle stops
- ③ Retraction motion (+Z) -> manual operation or fast feed

G87 execution operation graphic:



G87 execution operation graphic:

Fixed cycle I



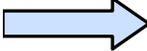
ISO Mode

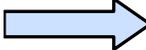


The ISO mode function provided by the 808D can easily operate the existing ISO program!

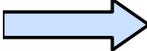
Step 1 Transfer ISO files in USB device to 808D.

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!



ISO Mode

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.



ISO Mode

Step 3 Program execution



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

Operate as described above.

Tool and workpiece setup → simulation → test → machining.

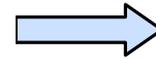
Step 4 Transfer the ISO files in the 808D to the USB device.

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

Press the "NC" SK on the PPU.



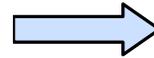
Use the "Cursor + Select" keys to select the required program which is then highlighted.



Press the "Copy" SK on the PPU.



Press the "USB" SK on the PPU.



Press the "Paste" SK on the PPU.



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



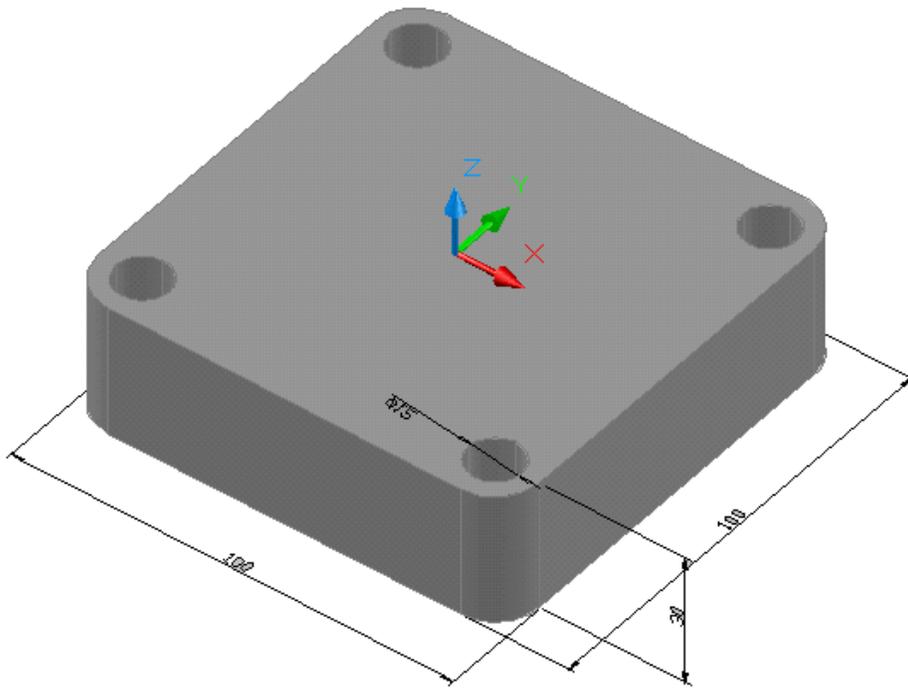
ISO Mode

Basic Theory

Step 5 Sample program



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



ISO Mode

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)



ISO Mode

Basic Theory

Standard Siemens programming.

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

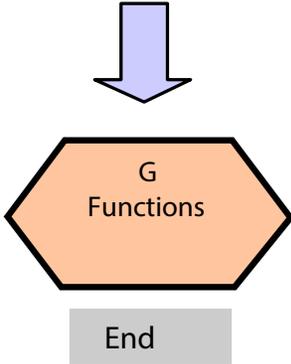
```
N10 T1D1M6 ; contour milling tool
N20 G54G90G40G17
N30 M3S2000M8
N40 G0Z25
N50 X0Y-70
N55 CYCLE72("SUB_PART_4", 50, 0, 2, -5, 2.5,
0.1, 0.1, 200, 200, 111, 41, 2, 20, 200, 2, 20)
N60 T2D1M6 ; quill, drill center hole
N70 M3S2500M8
N80 MCALL CYCLE82( 50, 0, 2, 0, 2, 0)
N90 CYCLE802( 111111111, 111111111, 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 */
```



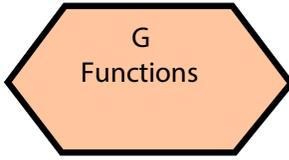
Appendix

Appendix

Content



Appendix



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



Appendix

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



Appendix

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



Terminology/Glossary

Terminology/Glossary

Term	Explanation
Cross table, milling table	Bearing surface, clamping surface for the workpiece with X- and Y-axis travel
CAD	Computer Aided Design
CAM	Computer Aided Manufacturing
CAP	Computer Aided Planning
CNC	Computerized Numerical Control
NC	Numerical Control
ATC	Automatic Tool Changer
MCP	Machine Control Panel
PPU	Poly Power Unit
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 rotary 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.



WIRING

These pages are current at the time of printing. However, in the spirit of improvement, we may make changes to the electrical systems of future machines. Compare the manufacture date of your machine to the one stated in this manual, and study this section carefully.

If there are differences between your machine and what is shown in this section, call Technical Support at (570) 546-9663 for assistance BEFORE making any changes to the wiring on your machine. An updated wiring diagram may be available. **Note:** *Please gather the serial number and manufacture date of your machine before calling. This information can be found on the main machine label.*

WARNING

Wiring Safety Instructions

SHOCK HAZARD. Working on wiring that is connected to a power source is extremely dangerous. Touching electrified parts will result in personal injury including but not limited to severe burns, electrocution, or death. Disconnect the power from the machine before servicing electrical components!

MODIFICATIONS. Modifying the wiring beyond what is shown in the diagram may lead to unpredictable results, including serious injury or fire. This includes the installation of unapproved after-market parts.

WIRE CONNECTIONS. All connections must be tight to prevent wires from loosening during machine operation. Double-check all wires disconnected or connected during any wiring task to ensure tight connections.

CIRCUIT REQUIREMENTS. You MUST follow the requirements at the beginning of this manual when connecting your machine to a power source.

WIRE/COMPONENT DAMAGE. Damaged wires or components increase the risk of serious personal injury, fire, or machine damage. If you notice that any wires or components are damaged while performing a wiring task, replace those wires or components.

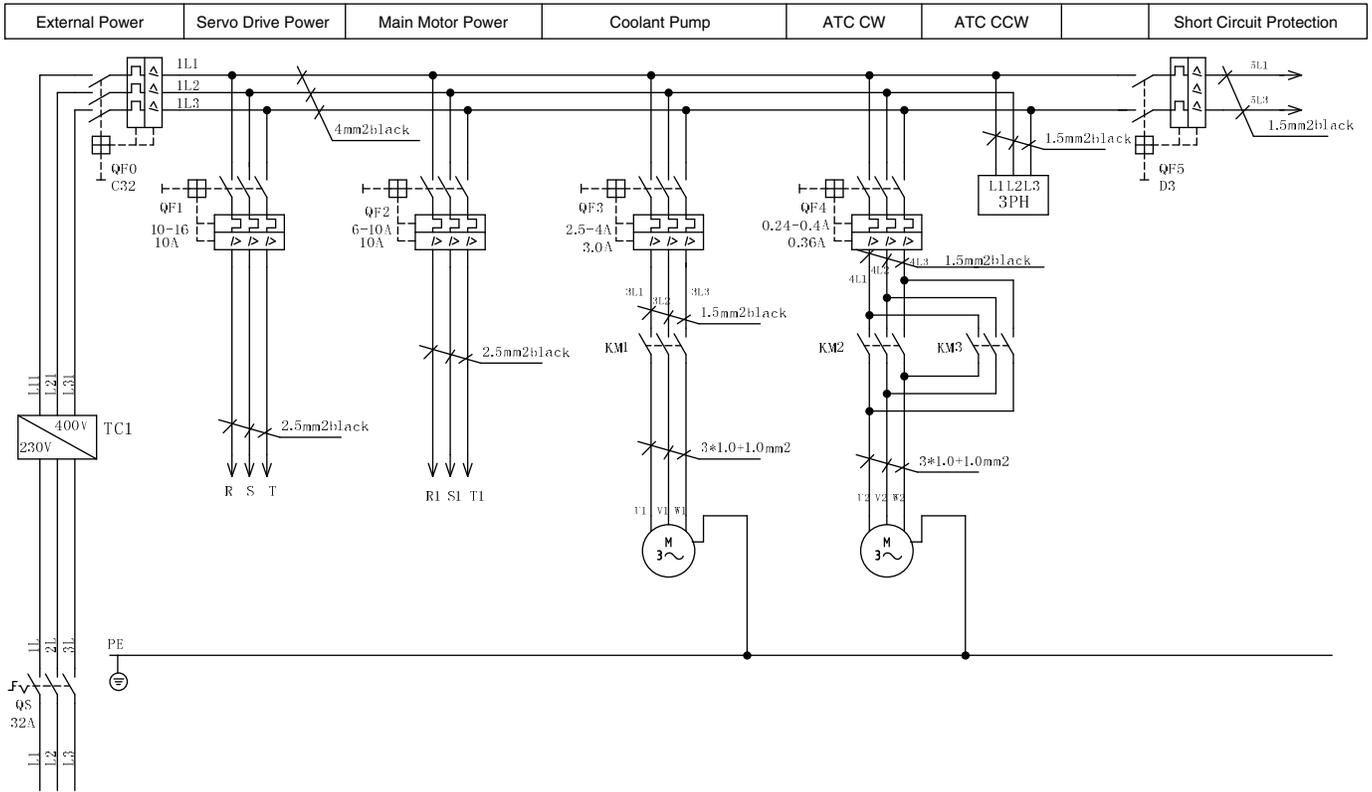
MOTOR WIRING. The motor wiring shown in these diagrams is current at the time of printing but may not match your machine. If you find this to be the case, use the wiring diagram inside the motor junction box.

CAPACITORS/INVERTERS. Some capacitors and power inverters store an electrical charge for up to 10 minutes after being disconnected from the power source. To reduce the risk of being shocked, wait at least this long before working on capacitors.

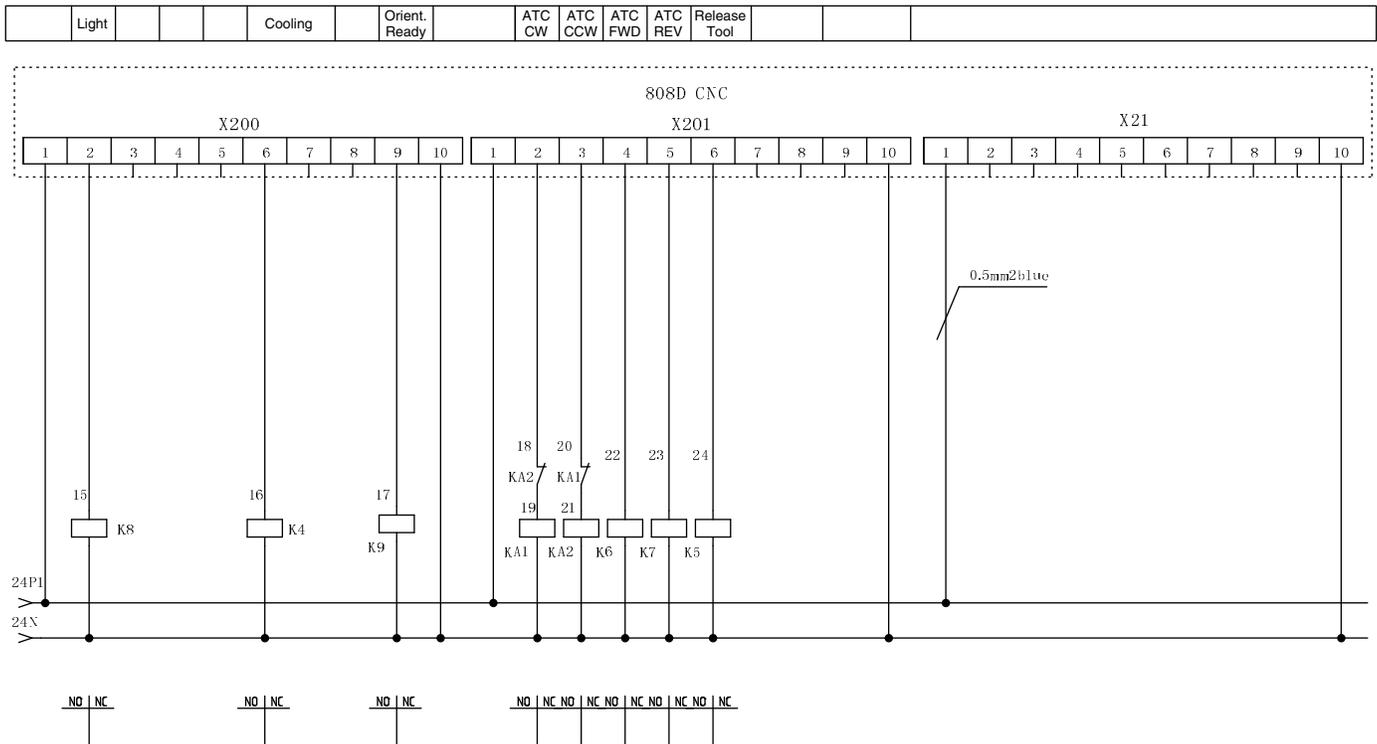
EXPERIENCING DIFFICULTIES. If you are experiencing difficulties understanding the information included in this section, contact our Technical Support at (570) 546-9663.



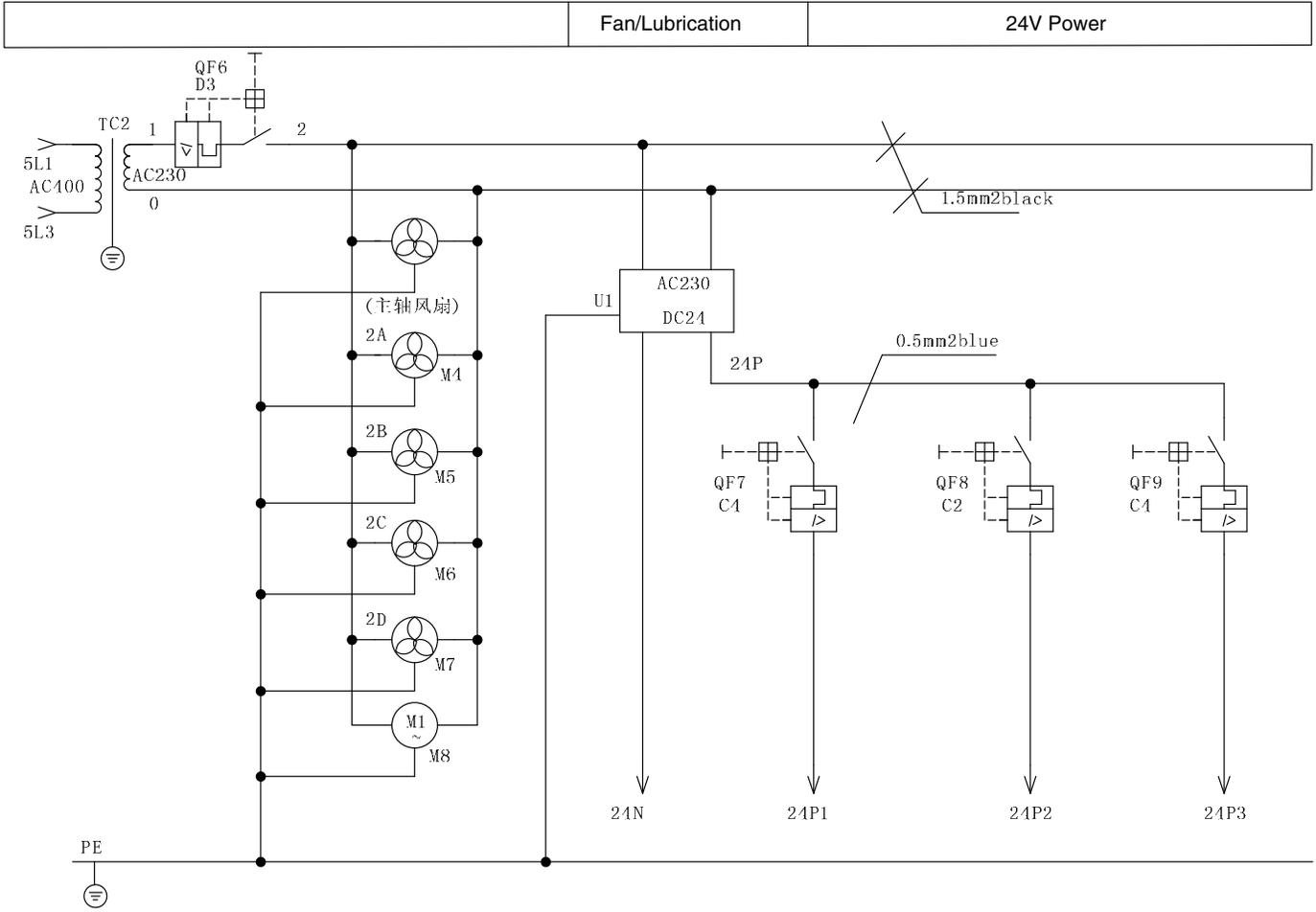
System Wiring Diagram (1)



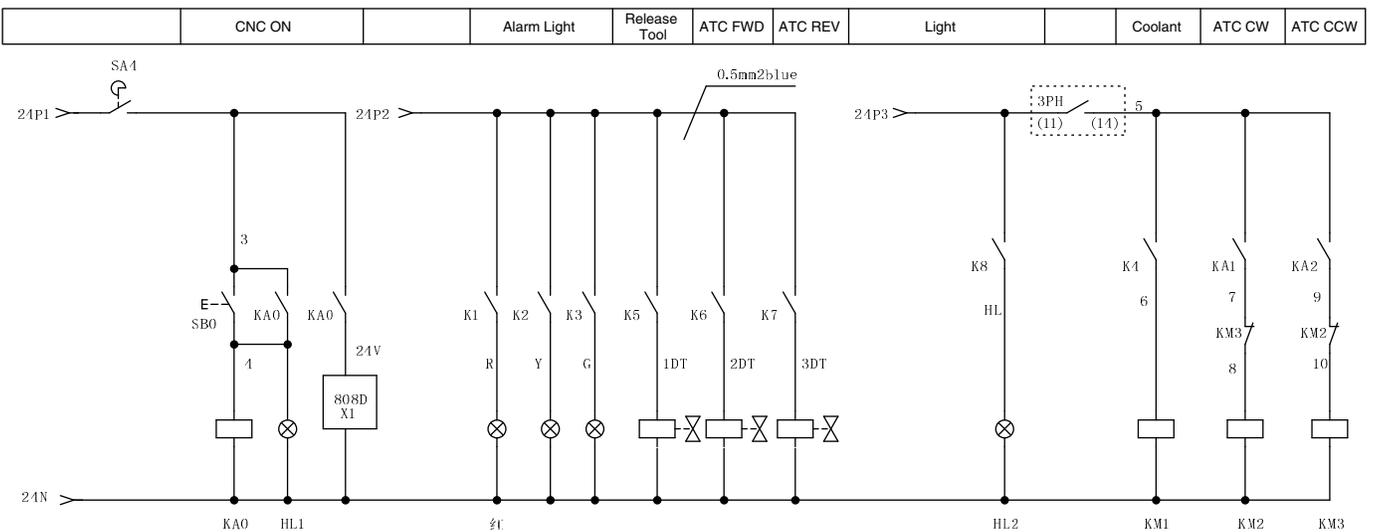
System Wiring (2) Diagram



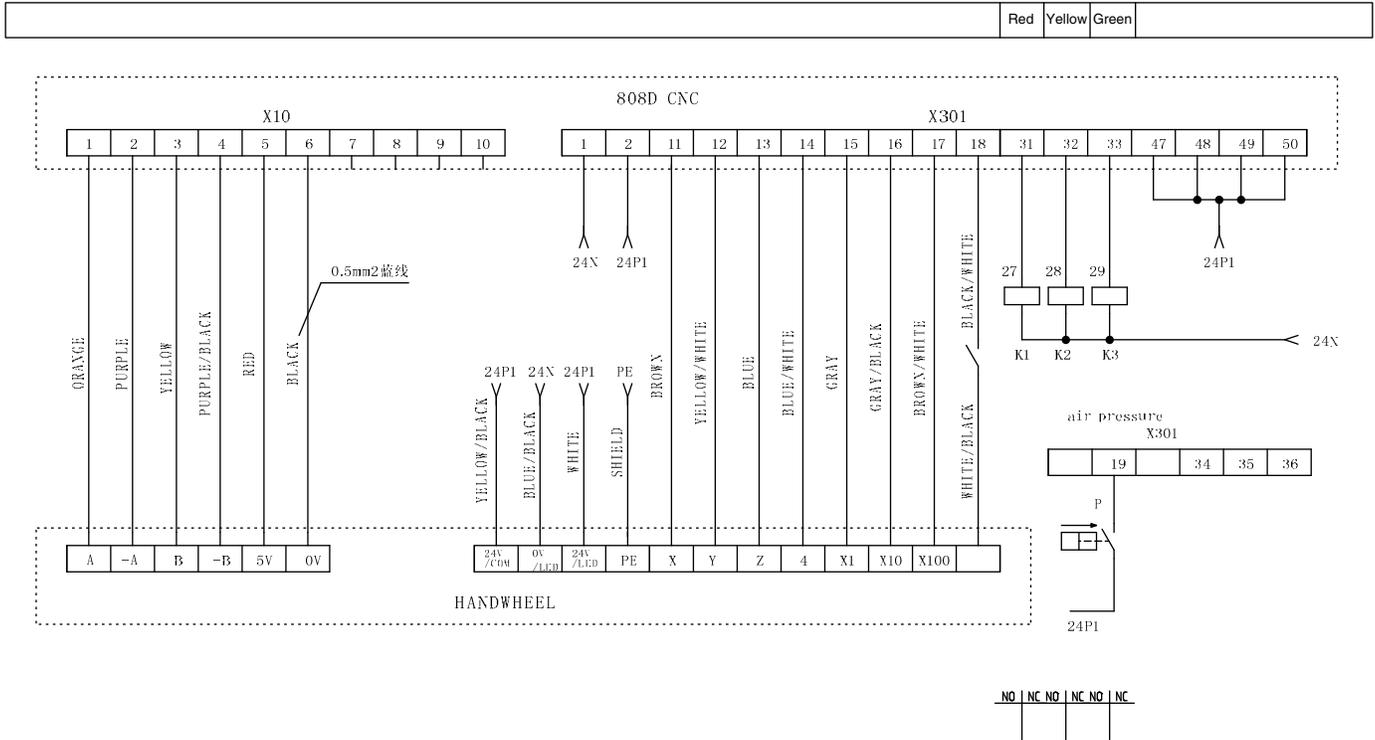
System Wiring Diagram (3)



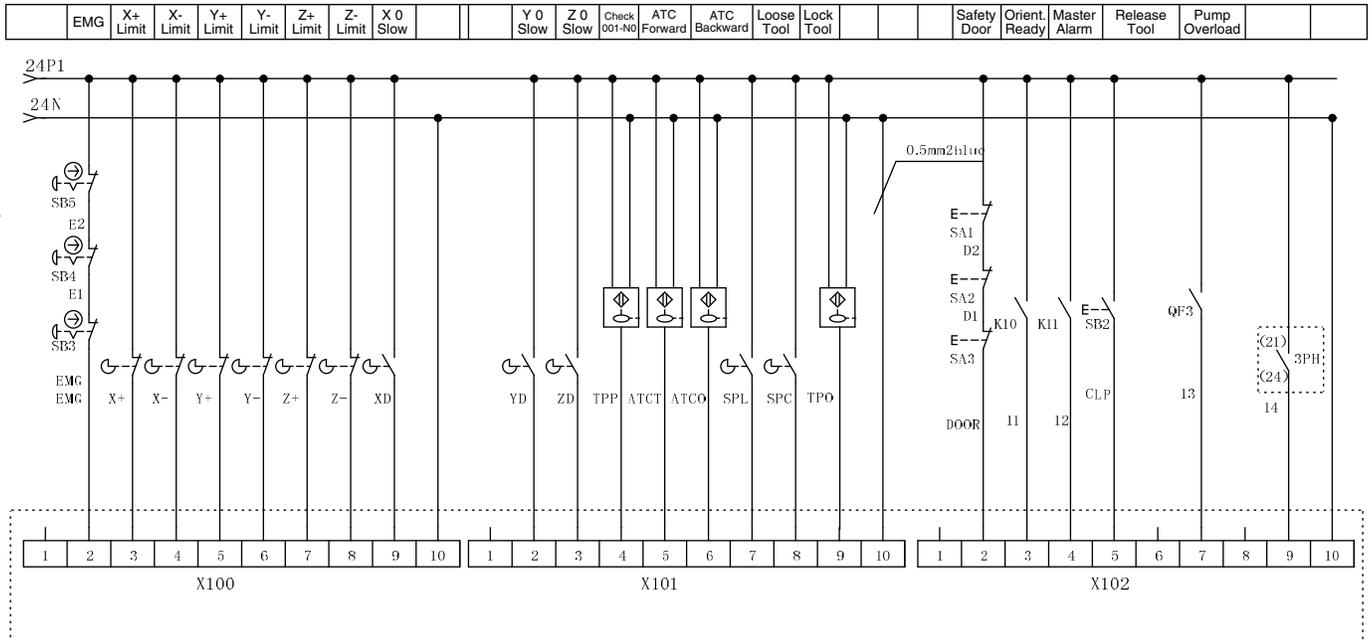
System Wiring (4) Diagram



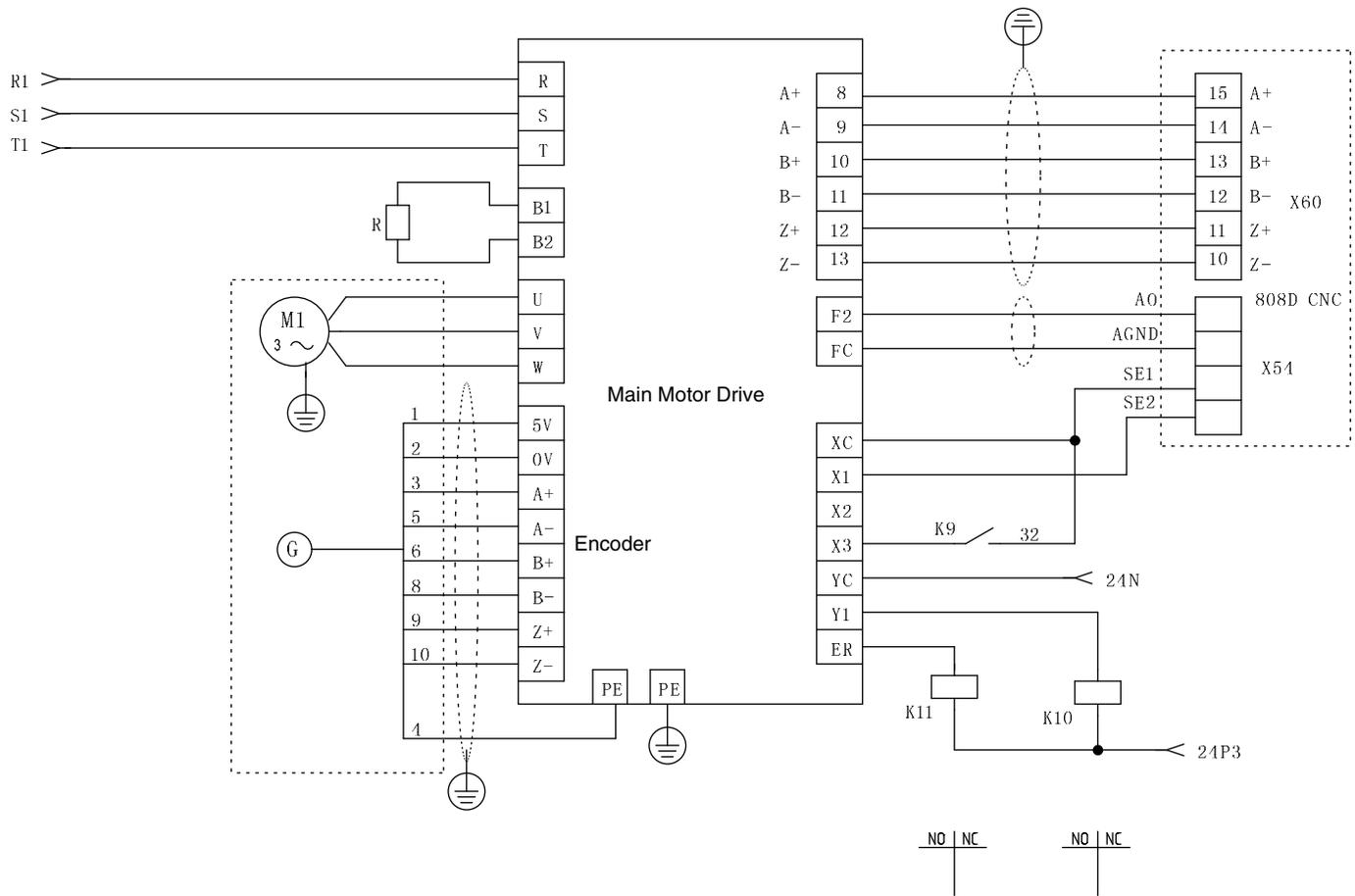
System Wiring (5) Diagram



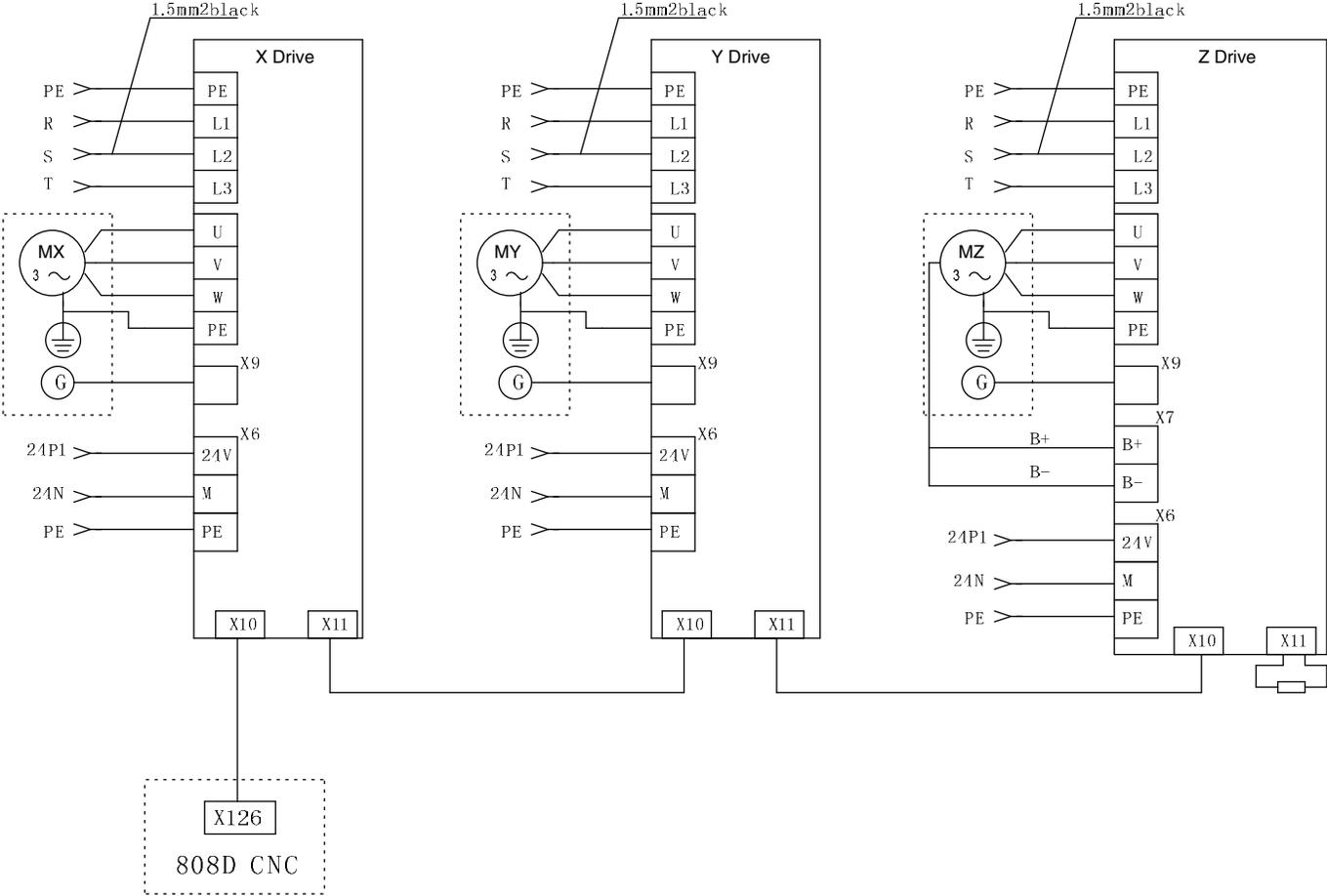
System Wiring (6) Diagram



Main Motor Wiring Diagram



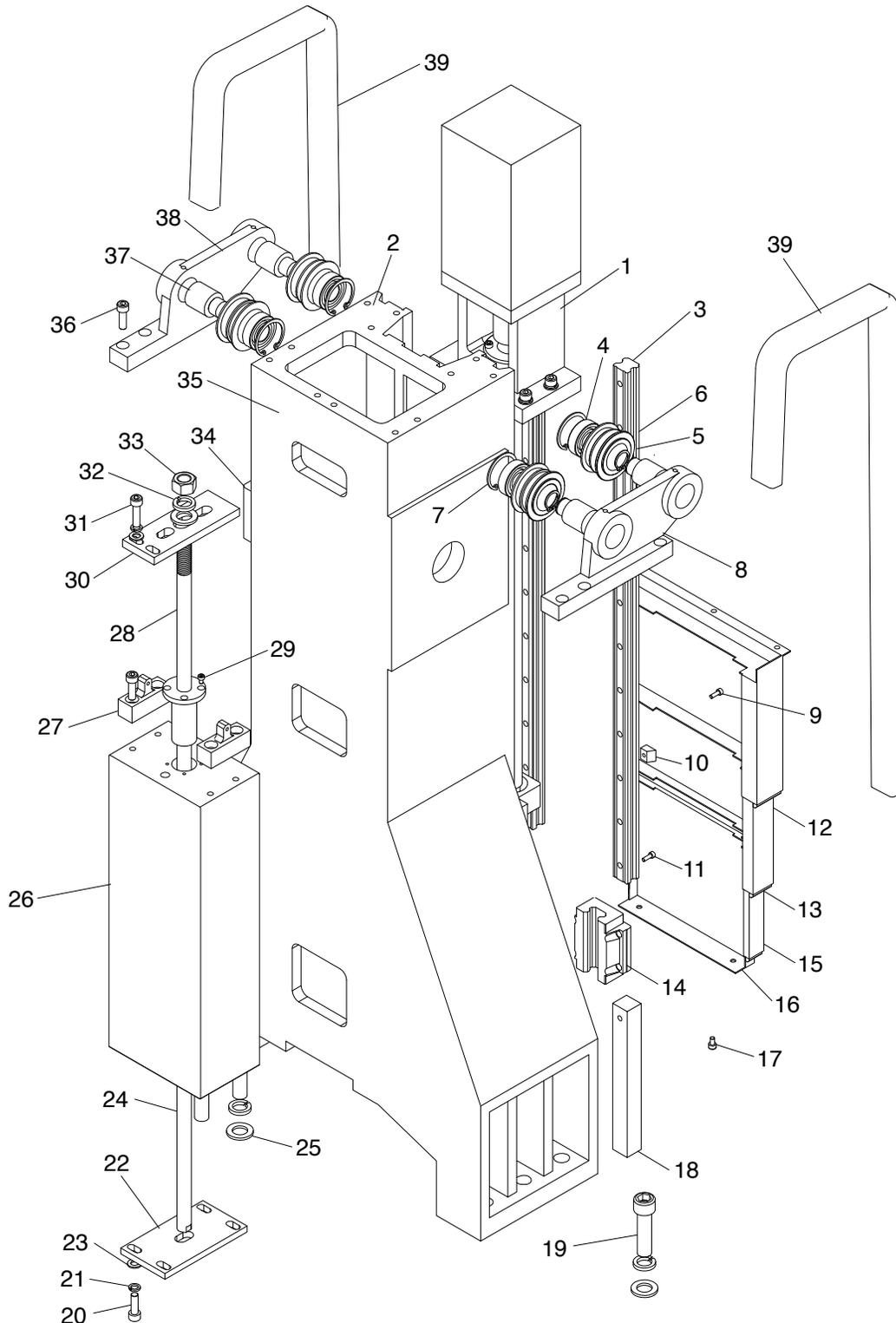
X/Y/Z Axes Wiring Diagram



PARTS

We do our best to stock replacement parts when possible, but we cannot guarantee that all parts shown are available for purchase. Call (800) 523-4777 or visit www.grizzly.com/parts to check for availability.

Column



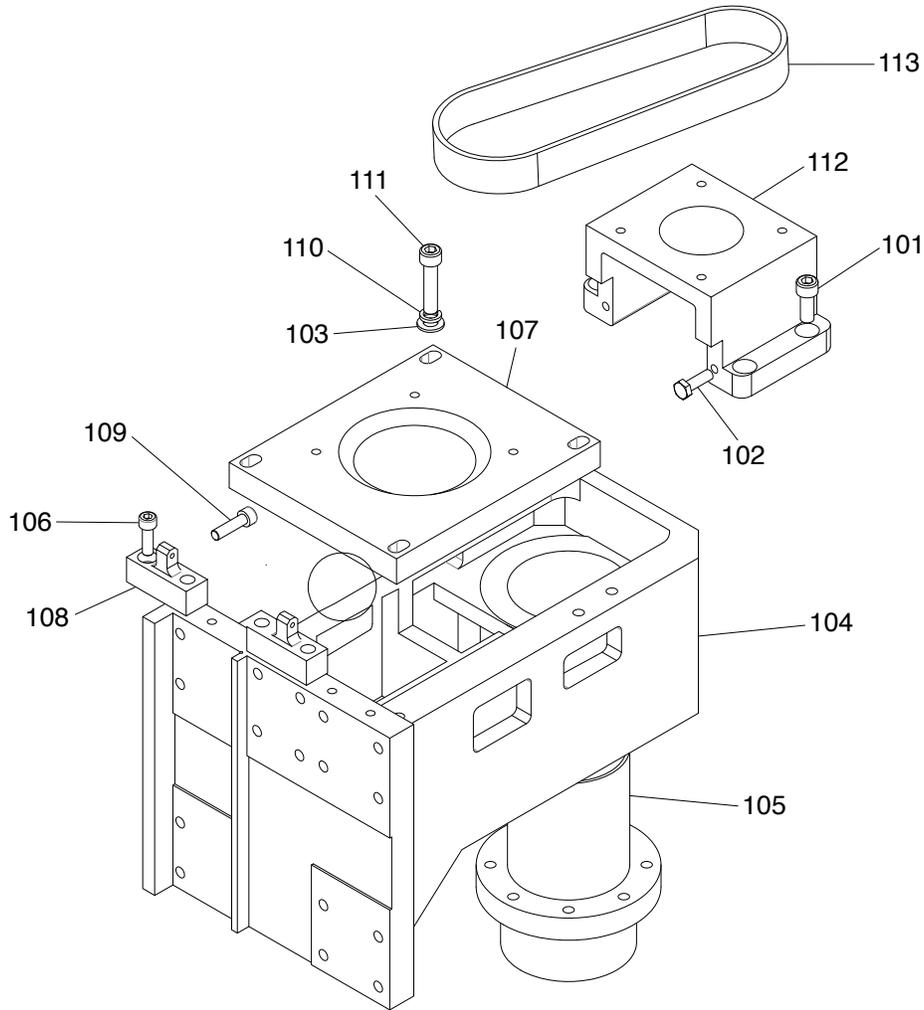
Column Parts List

REF	PART #	DESCRIPTION
1	P0877001	BEARING BLOCK
2	P0877002	COLUMN
3	P0877003	LINEAR GUIDE
4	P0877004	BALL BEARING 6004
5	P0877005	RETAINING RING
6	P0877006	GUIDE ROLL
7	P0877007	RETAINING RING 42
8	P0877008	HOLDER
9	P0877009	SOCKET HEAD SCREW M4-.7 X 12
10	P0877010	PLATE
11	P0877011	SOCKET HEAD SCREW M6-1 X 25
12	P0877012	COVER A
13	P0877013	COVER B
14	P0877014	LINEAR GUIDE SLIDE
15	P0877015	COVER C
16	P0877016	PLATE
17	P0877017	SOCKET HEAD SCREW M5-.8 X 8
18	P0877018	PLATE
19	P0877019	SOCKET HEAD SCREW M16-2 X 60
20	P0877020	SOCKET HEAD SCREW M8-1.25 X 12

REF	PART #	DESCRIPTION
21	P0877021	SPRING RING 8
22	P0877022	PLATE
23	P0877023	WASHER 4
24	P0877024	GUIDE
25	P0877025	WASHER 16
26	P0877026	BALANCE WEIGHT
27	P0877027	BLOCK
28	P0877028	SHAFT
29	P0877029	SOCKET HEAD SCREW M4-.7 X 6
30	P0877030	PLATE
31	P0877031	SOCKET HEAD SCREW M8-1.25 X 30
32	P0877032	SPRING RING 16
33	P0877033	HEX NUT M16-2
34	P0877034	BLOCK
35	P0877035	WASHER 6
36	P0877036	SOCKET HEAD SCREW M8-1.25 X 25
37	P0877037	SHAFT
38	P0877038	HOLDER
39	P0877039	BALANCE CHAIN



Milling Head

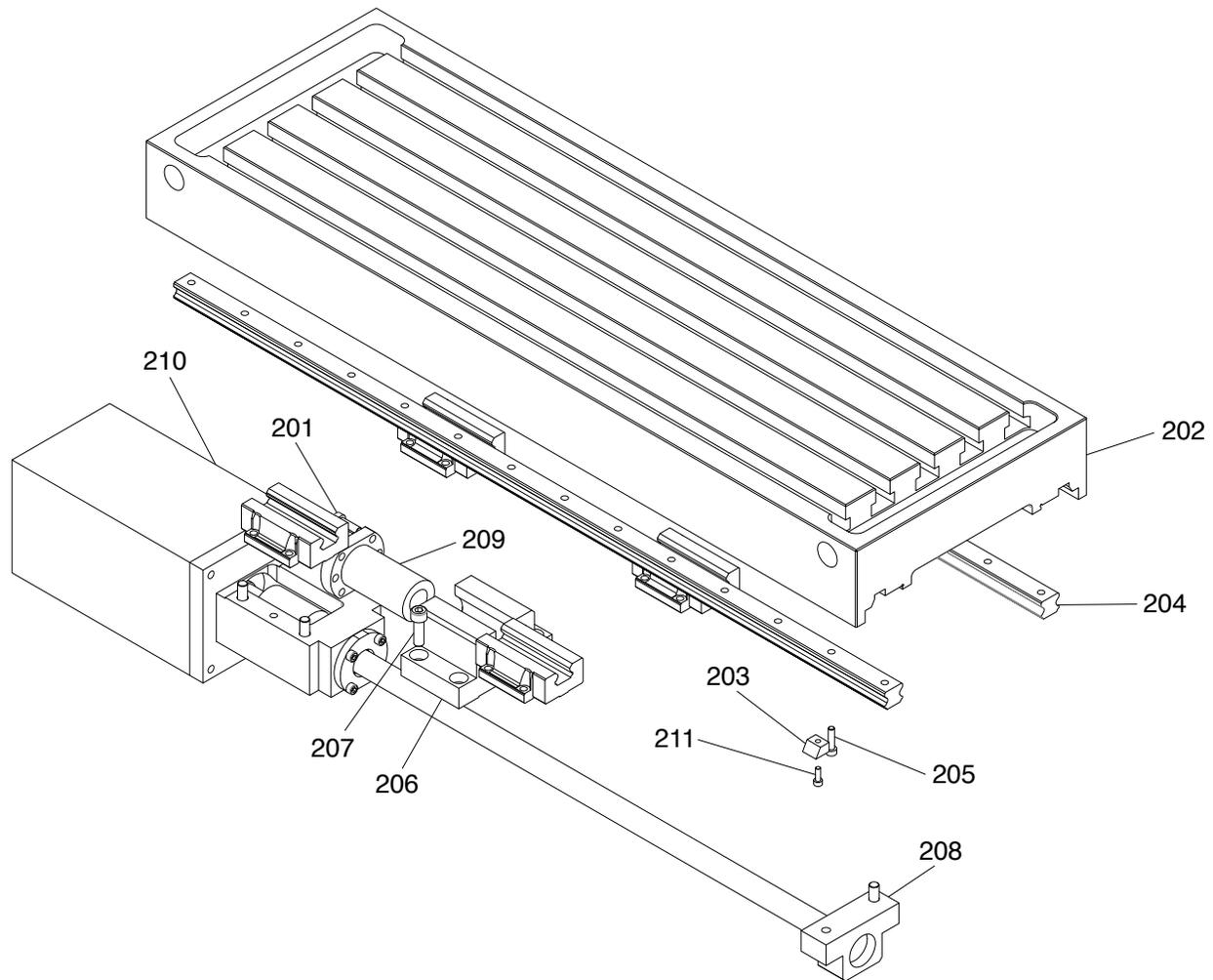


REF	PART #	DESCRIPTION
101	P0877101	SOCKET HEAD SCREW M10-1.5 X 25
102	P0877102	HEXAGON SCREW M8-1.25 X 25
103	P0877103	WASHER 10
104	P0877104	HOUSING
105	P0877105	MILLING SPINDLE
106	P0877106	SOCKET HEAD SCREW M8-1.25 X 25
107	P0877107	PLATE

REF	PART #	DESCRIPTION
108	P0877108	PLATE
109	P0877109	SOCKET HEAD SCREW M8-1.25 X 30
110	P0877110	SPRING RING 10
111	P0877111	SOCKET HEAD SCREW M10-1.5 X 45
112	P0877112	MOTOR HOLDER
113	P0877113	DRIVE BELT



Milling Table

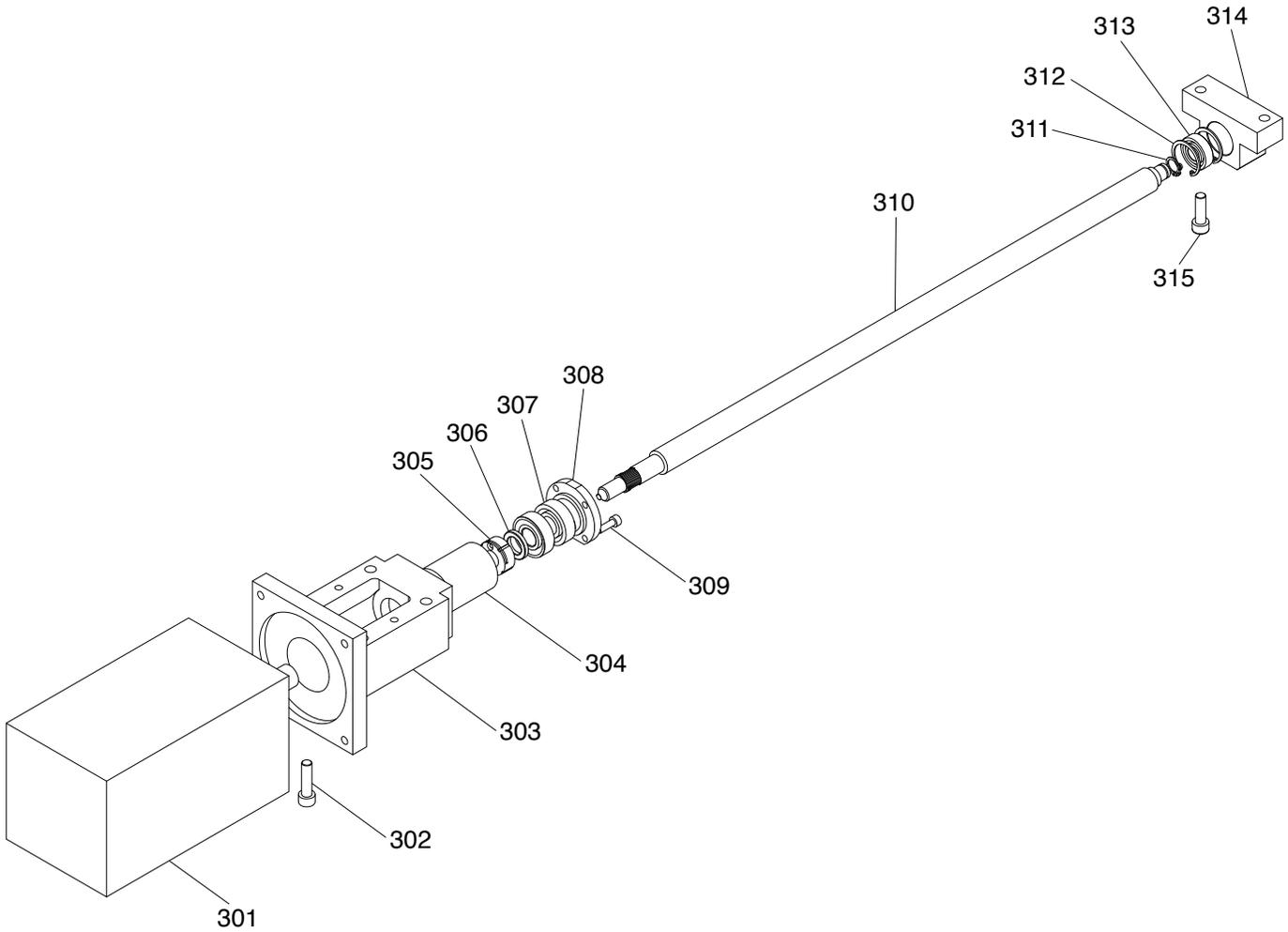


REF	PART #	DESCRIPTION
201	P0877201	SOCKET HEAD SCREW M6-1 X 20
202	P0877202	MILLING TABLE
203	P0877203	PLATE
204	P0877204	LINEAR GUIDE
205	P0877205	SOCKET HEAD SCREW M5-.8 X 20
206	P0877206	HOLDER

REF	PART #	DESCRIPTION
207	P0877207	SOCKET HEAD SCREW M8-1.25 X 25
208	P0877208	HOLDER
209	P0877209	SPINDLE NUT
210	P0877210	LINEAR GUIDE SLIDE
211	P0877211	SOCKET HEAD SCREW M4-.7 X 12



X-Axis

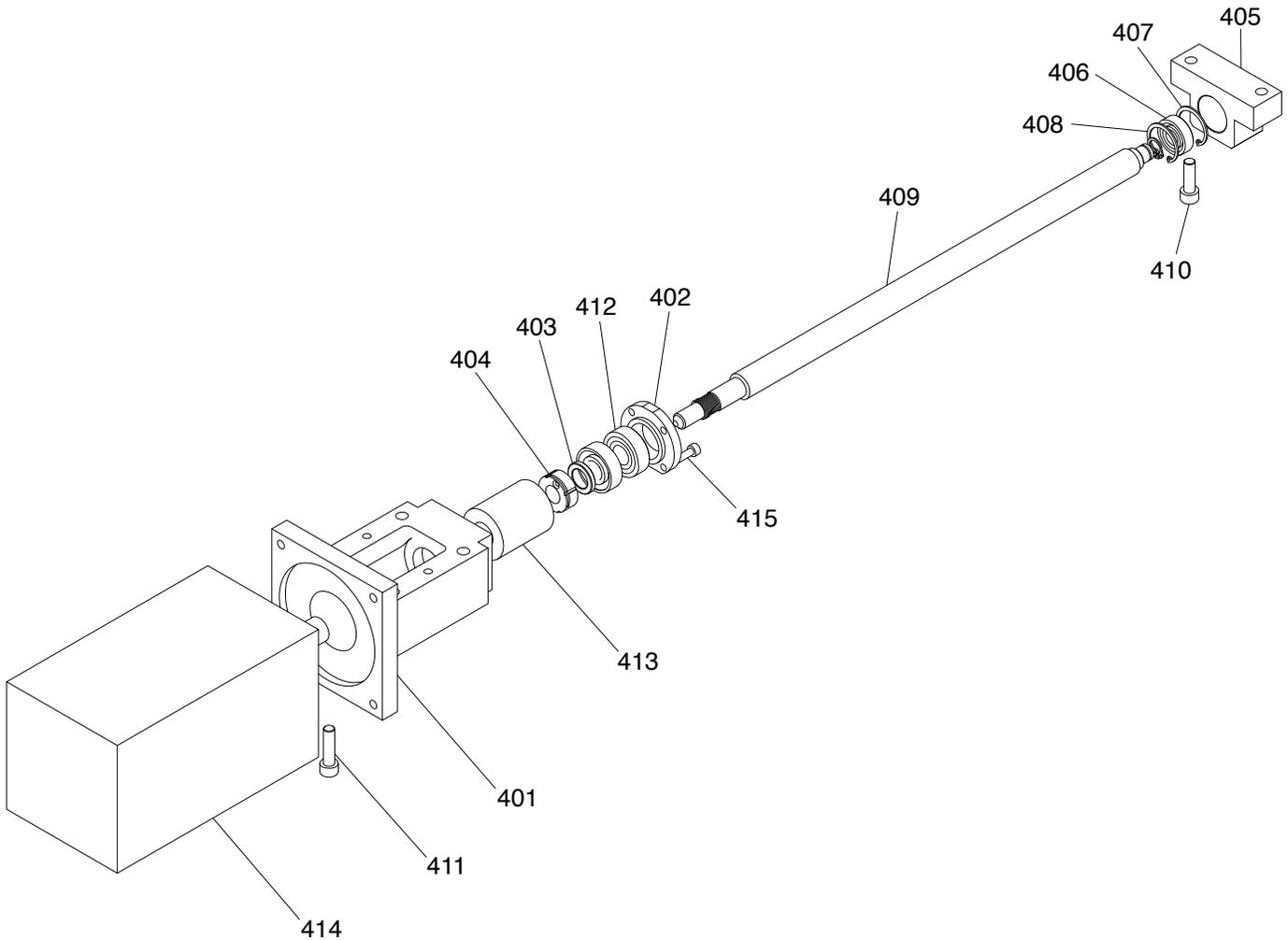


REF	PART #	DESCRIPTION
301	P0877301	MOTOR
302	P0877302	SOCKET HEAD SCREW M8-1.25 X 30
303	P0877303	BEARING BLOCK
304	P0877304	CLUTCH
305	P0877305	CLAMPING NUT
306	P0877306	RING
307	P0877307	BALL BEARING 7202
308	P0877308	FLANGE

REF	PART #	DESCRIPTION
309	P0877309	SOCKET HEAD SCREW M5-.8 X 16
310	P0877310	SPINDLE
311	P0877311	RETAINING RING 12
312	P0877312	RETAINING RING 28
313	P0877313	BALL BEARING 6001
314	P0877314	BEARING BLOCK
315	P0877315	SOCKET HEAD SCREW M8-1.25 X 25



Y-Axis

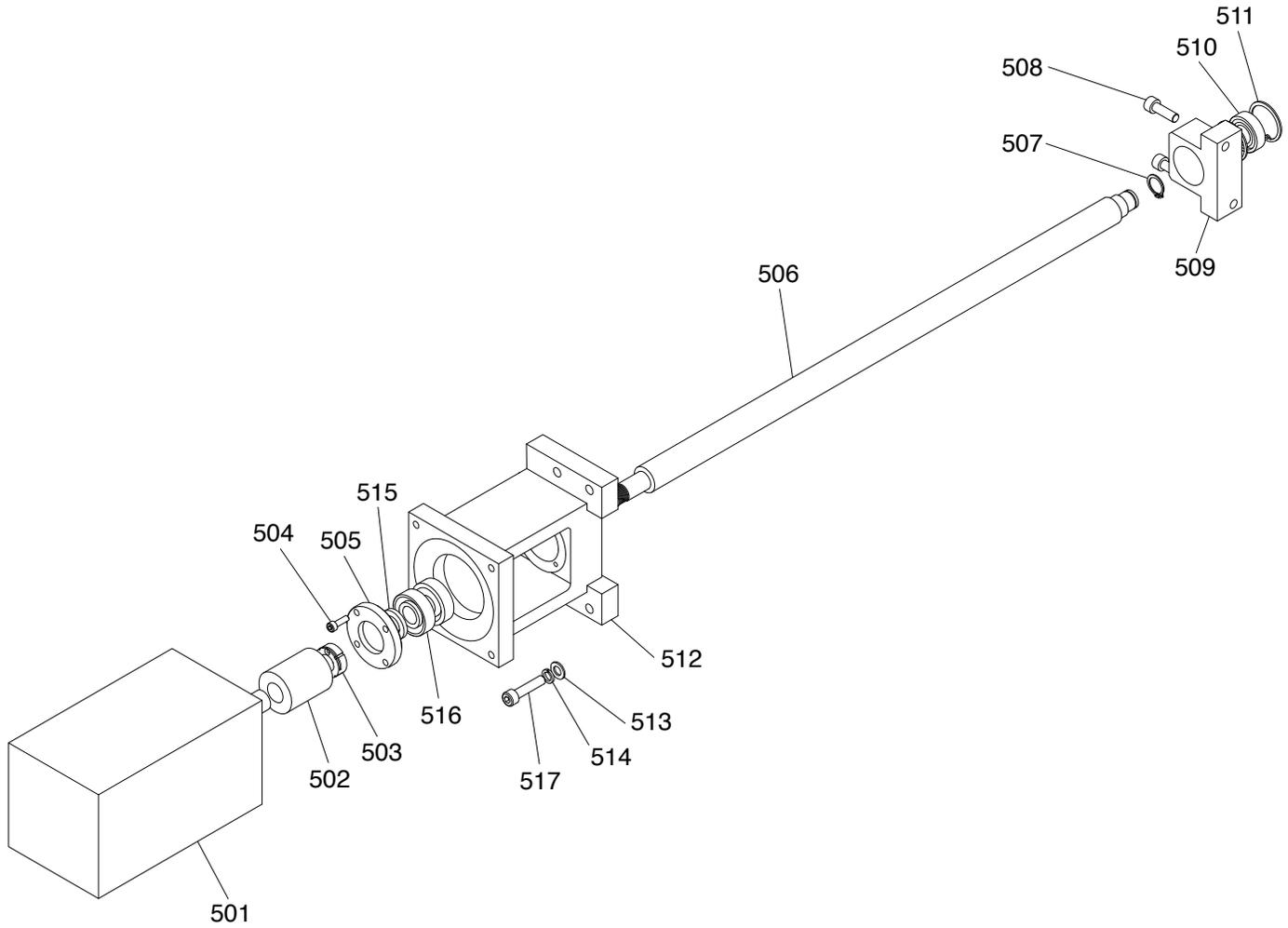


REF	PART #	DESCRIPTION
401	P0877401	BEARING BLOCK
402	P0877402	FLANGE
403	P0877403	RING
404	P0877404	CLAMPING NUT
405	P0877405	BEARING BLOCK
406	P0877406	BALL BEARING 6001
407	P0877407	RETAINING RING 28
408	P0877408	RETAINING RING 12

REF	PART #	DESCRIPTION
409	P0877409	SPINDLE
410	P0877410	SOCKET HEAD SCREW M8-1.25 X 25
411	P0877411	SOCKET HEAD SCREW M8-1.25 X 30
412	P0877412	BALL BEARING 7202
413	P0877413	CLUTCH
414	P0877414	MOTOR
415	P0877415	SOCKET HEAD SCREW M5-.8 X 16



Z-Axis

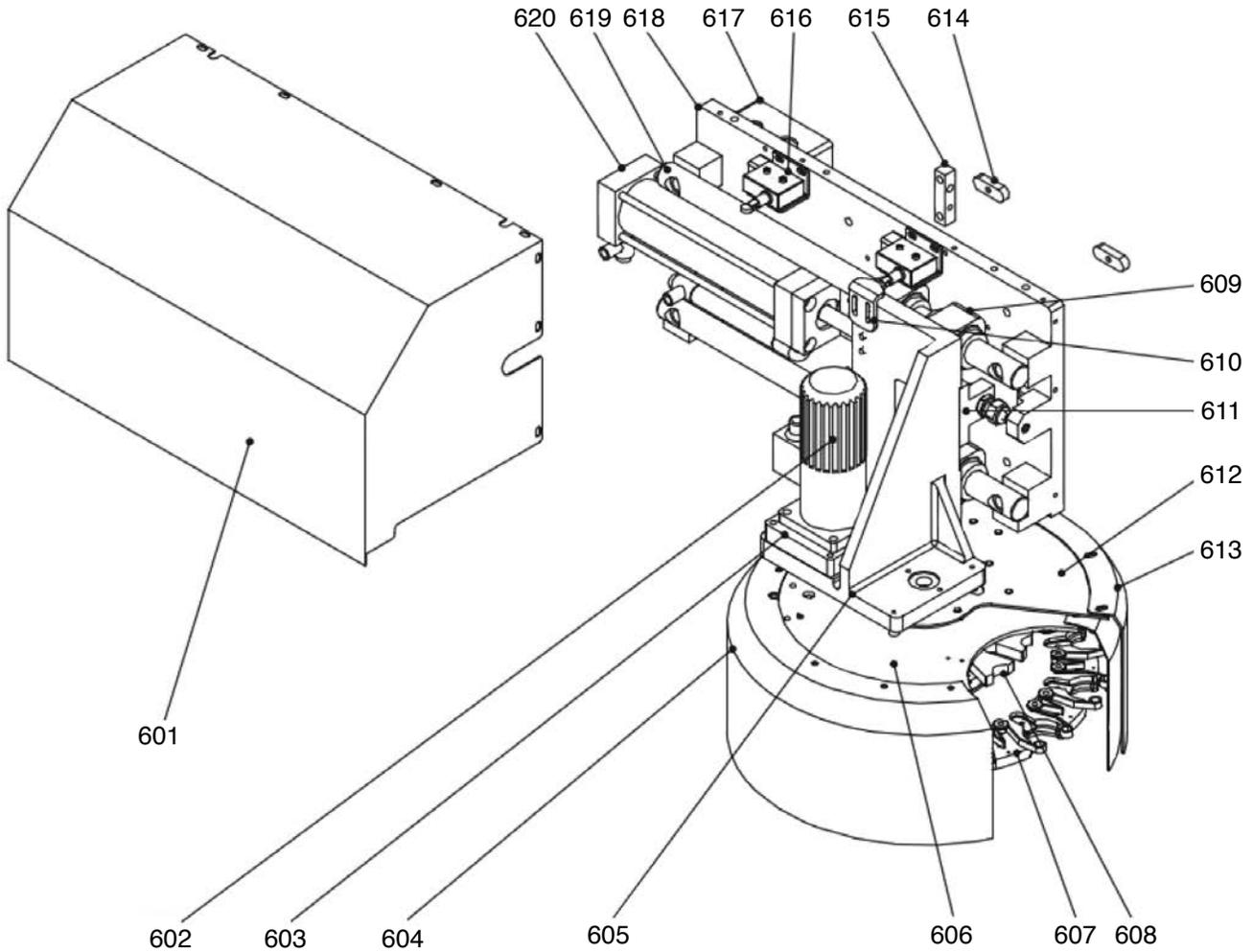


REF	PART #	DESCRIPTION
501	P0877501	MOTOR
502	P0877502	CLUTCH
503	P0877503	CLAMPING NUT
504	P0877504	SOCKET HEAD SCREW M5-.8 X 16
505	P0877505	FLANGE
506	P0877506	SPINDLE
507	P0877507	RETAINING RING 17
508	P0877508	SOCKET HEAD SCREW M8-1.25 X 25
509	P0877509	BEARING BLOCK

REF	PART #	DESCRIPTION
510	P0877510	BALL BEARING 6003
511	P0877511	RETAINING RING 35
512	P0877512	BEARING BLOCK
513	P0877513	WASHER 8
514	P0877514	SPRING RING 8
515	P0877515	SPACER
516	P0877516	BALL BEARING 7203
517	P0877517	SOCKET HEAD SCREW M8-1.25 X 35



Tool Changer

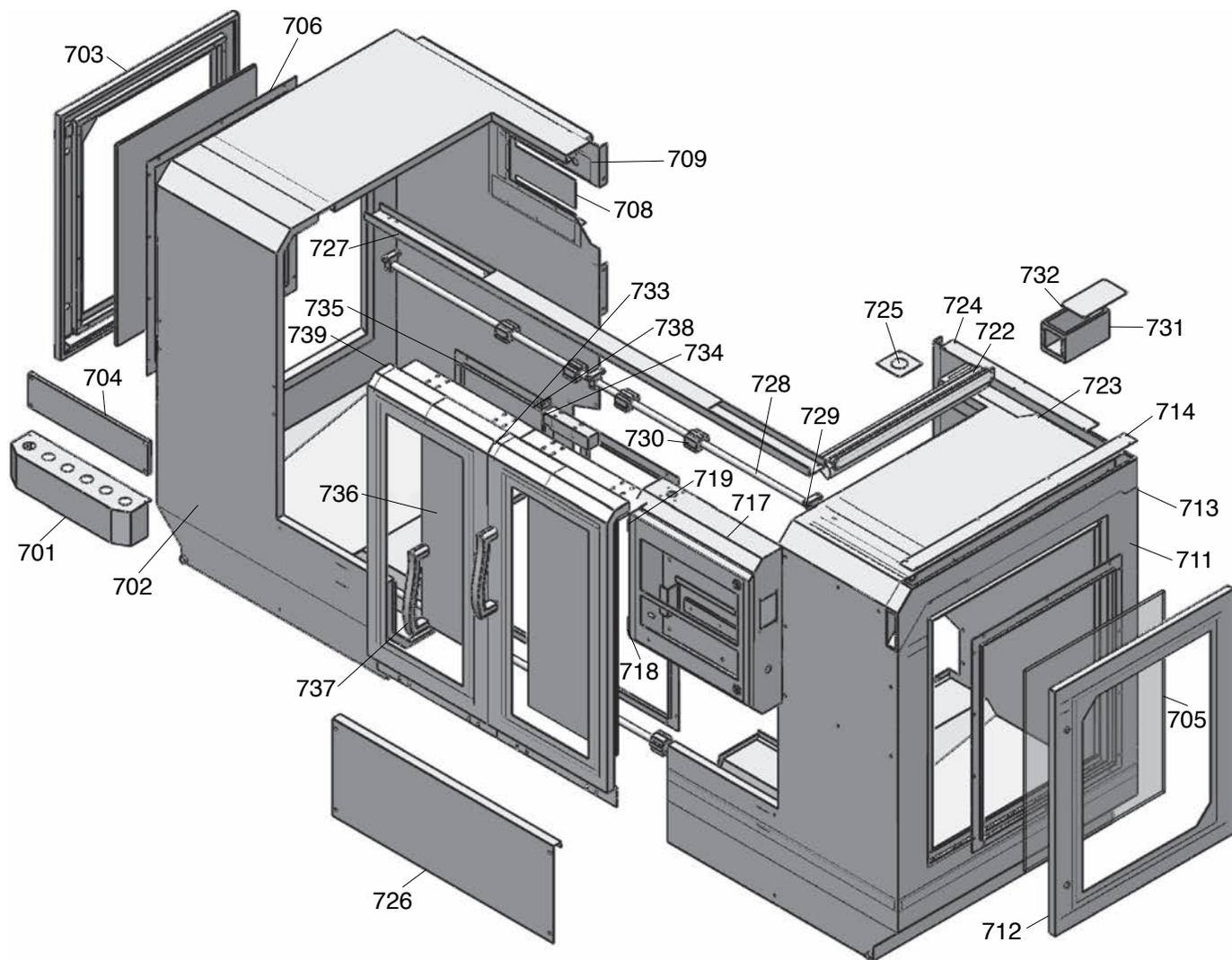


REF	PART #	DESCRIPTION
601	P0877601	COVER
602	P0877602	MOTOR
603	P0877603	MOTOR PLATE
604	P0877604	COVER
605	P0877605	HOLDER
606	P0877606	PLATE
607	P0877607	TOOL HOLDER
608	P0877608	PLATE
609	P0877609	COVER
610	P0877610	PLATE

REF	PART #	DESCRIPTION
611	P0877611	GUIDE
612	P0877612	LIMIT STOP
613	P0877613	GUIDE
614	P0877614	PLATE
615	P0877615	BLOCK
616	P0877616	END SWITCH
617	P0877617	MANIFOLD
618	P0877618	BASE PLATE
619	P0877619	GUIDE ROD
620	P0877620	PNEUMATIC CYLINDER



Housing

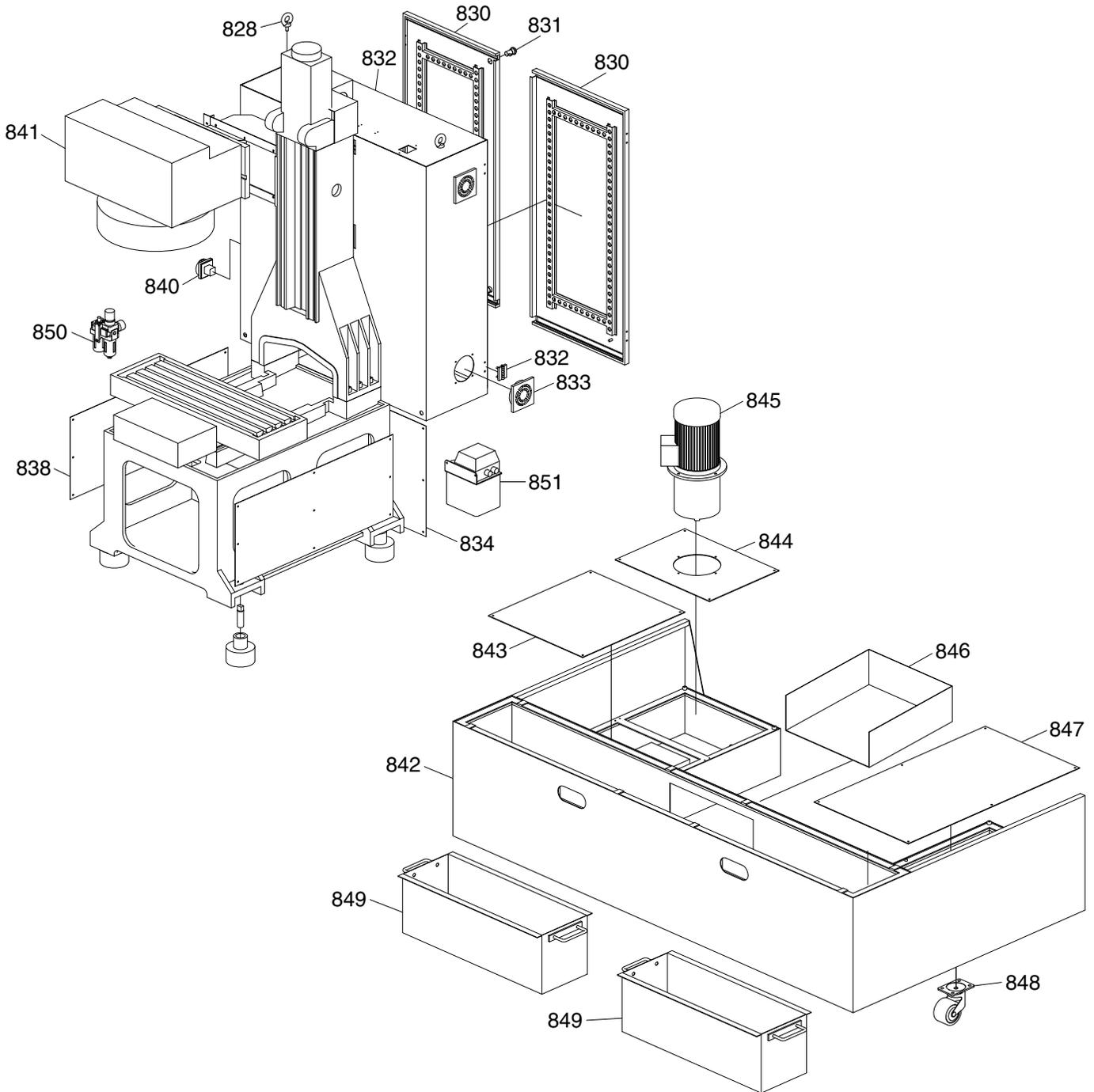


REF	PART #	DESCRIPTION
701	P0877701	HOLDER STORAGE
702	P0877702	LEFT COVER
703	P0877703	LEFT SIDE WINDOW
704	P0877704	HOLDER STORAGE PLATE
705	P0877705	WINDOWPANE L/R
706	P0877706	WINDOWPANE PLATE
707	P0877707	DOOR GUIDE SUPPORT
708	P0877708	MAGAZINE COVER
709	P0877709	MAGAZINE A BREAKWATER
710	P0877710	MAGAZINE B BREAKWATER
711	P0877711	RIGHT COVER
712	P0877712	RIGHT SIDE WINDOW
713	P0877713	WIRE TRUNKING
714	P0877714	TRUNKING COVER
715	P0877715	SWITCH COVER
716	P0877716	CONTROL CABINET COVER
717	P0877717	CONTROL CABINET FRAME
718	P0877718	HINGE A
719	P0877719	HINGE B
720	P0877720	LOCK MS718

REF	PART #	DESCRIPTION
721	P0877721	FUNNEL
722	P0877722	LIGHT
723	P0877723	4TH AXIS TRUNKING
724	P0877724	4TH AXIS TRUNKING COVER
725	P0877725	4TH AXIS TRUNKING COVER PLATE
726	P0877726	LOWER COVER
727	P0877727	UPPER RAIL BRACKET
728	P0877728	SLIDE BAR
729	P0877729	SLIDE BAR SUPPORT
730	P0877730	SLIDING BLOCK
731	P0877731	TRUNKING
732	P0877732	TRUNKING COVER
733	P0877733	RIGHT DOOR
734	P0877734	DOOR SWITCH COVER
735	P0877735	FLAT
736	P0877736	WINDOW PANE
737	P0877737	DOOR HANDLE
738	P0877738	BLOT SUPPORT
739	P0877739	LEFT DOOR



Stand & Coolant Tank

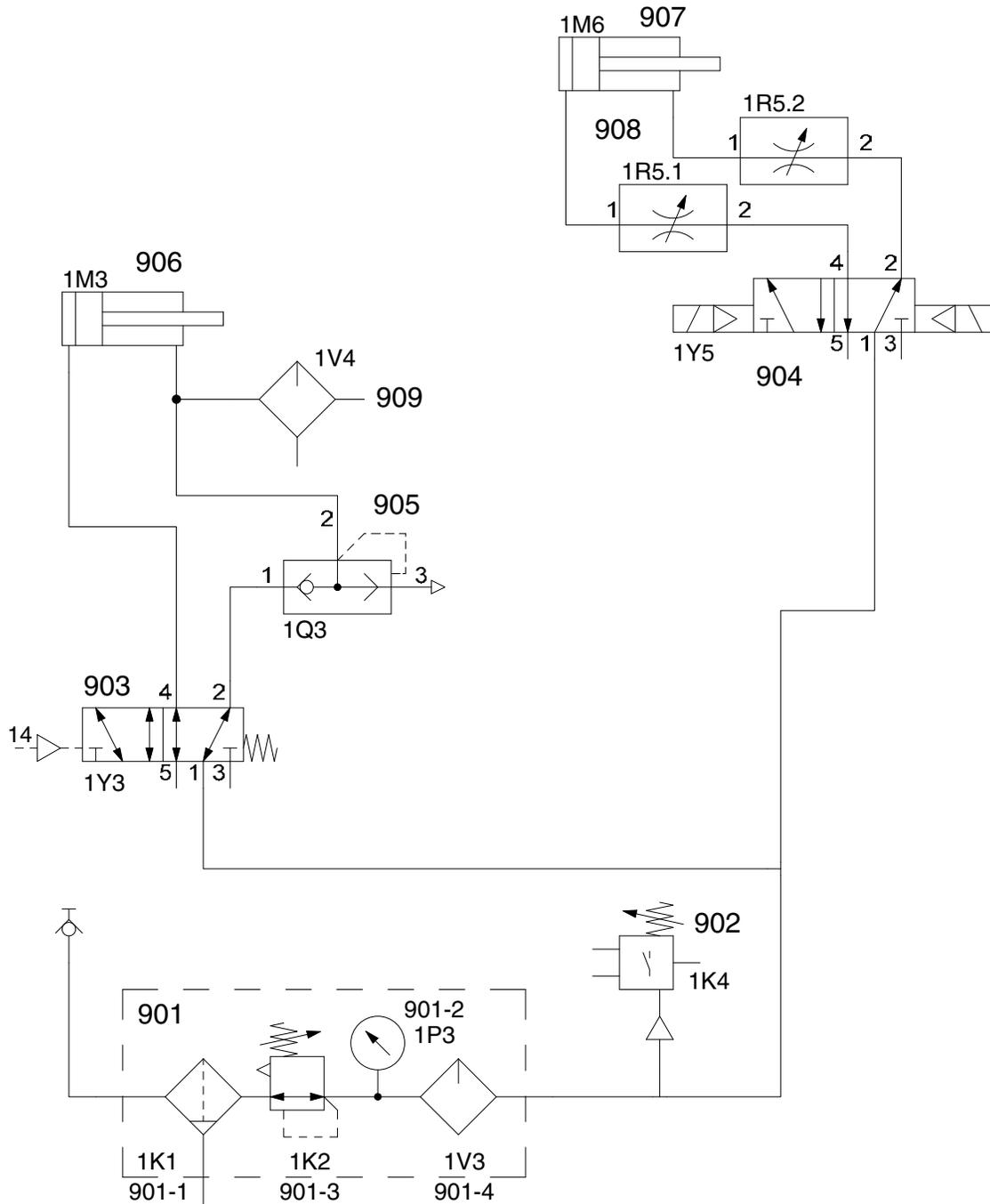


REF	PART #	DESCRIPTION
828	P0877828	O-SCREW
830	P0877830	ELECTRICAL PANEL DOOR
831	P0877831	LOCK
832	P0877832	HINGE
833	P0877833	FAN
834	P0877834	COVER
838	P0877838	COVER
840	P0877840	MAIN SWITCH
841	P0877841	TOOL CHANGER
842	P0877842	SUBSTRUCTURE

REF	PART #	DESCRIPTION
843	P0877843	COVER
844	P0877844	COVER
845	P0877845	COOLANT PUMP
846	P0877846	COVER
847	P0877847	COVER
848	P0877848	ROLL
849	P0877849	FILTER
850	P0877850	SERVICE UNIT
851	P0877851	OIL PUMP



Pneumatic System



REF	PART #	DESCRIPTION
901	P0877901	COMPRESSED AIR ASSEMBLY
901-1	P0877901-1	SINTERED METAL FILTER
901-2	P0877901-2	MANOMETER
901-3	P0877901-3	PRESSURE REGULATOR
901-4	P0877901-4	LUBRICATOR
902	P0877902	PRESSURE SWITCH
903	P0877903	WAY VALVE

REF	PART #	DESCRIPTION
904	P0877904	WAY VALVE
905	P0877905	QUICK EXHAUST VALVE
906	P0877906	CYLINDER TOOL SPANNER
907	P0877907	CYLINDER TOOL CHANGER
908	P0877908	THROTTLE VALVE
909	P0877909	DRIP LUBRICATOR



WARRANTY & RETURNS

Grizzly Industrial, Inc. warrants every product it sells for a period of **1 year** to the original purchaser from the date of purchase. This warranty does not apply to defects due directly or indirectly to misuse, abuse, negligence, accidents, repairs or alterations or lack of maintenance. This is Grizzly's sole written warranty and any and all warranties that may be implied by law, including any merchantability or fitness, for any particular purpose, are hereby limited to the duration of this written warranty. We do not warrant or represent that the merchandise complies with the provisions of any law or acts unless the manufacturer so warrants. In no event shall Grizzly's liability under this warranty exceed the purchase price paid for the product and any legal actions brought against Grizzly shall be tried in the State of Washington, County of Whatcom.

We shall in no event be liable for death, injuries to persons or property or for incidental, contingent, special, or consequential damages arising from the use of our products.

The manufacturers reserve the right to change specifications at any time because they constantly strive to achieve better quality equipment. We make every effort to ensure that our products meet high quality and durability standards and we hope you never need to use this warranty.

In the event you need to use this warranty, contact us by mail or phone and give us all the details. We will then issue you a "Return Number," which must be clearly posted on the outside as well as the inside of the carton. We will not accept any item back without this number. Proof of purchase must accompany the merchandise.

Please feel free to write or call us if you have any questions about the machine or the manual.

Thank you again for your business and continued support. We hope to serve you again soon.

To take advantage of this warranty, you must register it at <https://www.grizzly.com/secureforms/warranty-card>, or you can scan the QR code below to be automatically directed to our warranty registration page. Enter all applicable information for the product.



grizzly.com[®]

TOOL WEBSITE

Buy Direct and Save with Grizzly[®] – Trusted, Proven and a Great Value!
~Since 1983~

*Visit Our Website Today For
Current Specials!*

**ORDER
24 HOURS A DAY!
1-800-523-4777**

