Grizzly Industrial, Inc. ®

MODEL G0876
8" X 27" ENCLOSED CNC MILL
w/AUTO TOOL CHANGER

OWNER'S MANUAL

(For models manufactured since 12/22)

V2.02.23

Keep for Future Reference

⚠ 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

Table of Contents (Cont.)

MODEL G0876 8" X 27" ATC ENCLOSED CNC MILL

Product Dimensions:

Weight	3080 lbs.
Width (side-to-side) x Depth (front-to-back) x Height	71 x 65 x 79 in.
Footprint (Length/Width)	110 x 86 in.

Shipping Dimensions:

Type	Wood Crate
Content	Machine
Weight	3350 lbs.
Length x Width x Height	80 x 72-1/2 x 85 in.
Must Ship Upright	Yes

Electrical:

Power Requirement	220V, 3-Phase, 60 Hz
Full-Load Current Rating	14.1A
Minimum Circuit Size	20A
Connection Type	Cord & Plug
Power Cord Length	6-1/2 ft.
Power Cord Gauge	12 AWG
Recommended Plug Type	15-20

Motors:

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 .... 60 Watts
Phase 3-Phase
Amps 0.24A
Speed ....1360 RPM
Type.. Servo
Power Transfer..Gearbox
Bearings ....Sealed & Permanently Lubricated

Coolant Pump

Horsepower .... 97 Watts
Phase 3-Phase
Amps 0.5A
Speed ....2800 RPM
Type.. Induction
Power Transfer..Direct Drive
Bearings ....Shielded & Permanently Lubricated

Main Specifications:

Operation Information

Max Distance Spindle to Column..11 in.
Max Distance Spindle to Table ....16-1/2 in.
Longitudinal Table Travel (X-Axis)...12-3/16 in.
Cross Table Travel (Y-Axis) ....7-3/4 in.
Vertical Head Travel (Z-Axis) ....11-3/4 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..8 Tools
Maximum Tool Weight... 13 lbs
Maximum Tool Diameter with Adjacent Tool ....2-3/4 in.
Tool Change Average Time ....6 Seconds

Table Info

Table Length ....27 in.
Table Width ....8-1/4 in.
Table Thickness ....2-3/8 in.
Table Height (from Floor/Base)...45 in.
Table Weight Capacity ....220 lbs.
Number of T-Slots 3
T-Slot Size...5/8 in.
T-Slots Centers ....2-1/2 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 .... 13 Gallons

Construction

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

Other Specifications:

Country of Origin... China
Warranty.. 1 Year
Approximate Assembly & Setup Time ....1-1/2 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
Powerful 3 HP Spindle Motor
50–10,000 RPM Spindle Speeds
Easy Access to Table from 3 Sides
Hands-Free Tool Changes with 8-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.

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

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

⚠CAUTION 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

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.

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.

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!

AWKWARD POSITIONS. Keep proper footing and balance at all times when operating machine. Do not overreach! Avoid awkward hand positions that make workpiece control difficult or increase the risk of accidental injury.

CHILDREN & BYSTANDERS. Keep children and bystanders at a safe distance from the work area. Stop using machine if they become a distraction.

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.

FORCING MACHINERY. Do not force machine. It will do the job safer and better at the rate for which it was designed.

NEVER STAND ON MACHINE. Serious injury may occur if machine is tipped or if the cutting tool is unintentionally contacted.

STABLE MACHINE. Unexpected movement during operation greatly increases risk of injury or loss of control. Before starting, verify machine is stable and mobile base (if used) is locked.

USE RECOMMENDED ACCESSORIES. Consult this owner's manual or the manufacturer for recommended accessories. Using improper accessories will increase the risk of serious injury.

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!

MAINTAIN POWER CORDS. When disconnecting cord-connected machines from power, grab and pull the plug—NOT the cord. Pulling the cord may damage the wires inside. Do not handle cord/plug with wet hands. Avoid cord damage by keeping it away from heated surfaces, high traffic areas, harsh chemicals, and wet/damp locations.

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

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

In case of specific dangers, we replace the pictogram with

general danger   by a warning of   injury to hands,   hazardous electrical voltage,   rotating parts.

1.3.2 Other pictograms

Activation forbidden!

Do not step into the machine!

Do not extinguish with water!

Access forbidden!

Wear protective boots!

Use ear protection!

Wear protective glasses!

Read the operating instruction before commissioning!

Warning: suspended loads!

Warning of oxidizing substances!

Caution, danger of explosive substances!

Warning: danger of slipping!

Protect the environment!

Contact address

1.4   Intended use

WARNING!

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

The CNC machine is designed and manufactured to be used for milling and drilling cold metals or other non-flammable materials or materials that do not constitute a health hazard by using commercial milling and drilling tools.
Using this machine it is possible to perform dry processing as well as processing by using cooling lubricants. ☞ "Cooling lubricants" on page 243

The limit values of the balances of the tools need to be observed. ☞ "Tools and tool holding fixtures" on page 21

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 or modified without the express approval of Grizzly Industrial, Inc., then the CNC machine is being used improperly.

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

We expressly point out that the guarantee or CE conformity will expire, if any constructive, technicolor procedural changes are not performed by Grizzly Industrial, Inc.

It is also part of intended use that you

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

WARNING!

Severe injuries due to non-intended use.

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

1.5 Reasonably foreseeable misuse

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

Any other use has to be discussed with the manufacturer.

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

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

Operators must be qualified.

1.5.1 Avoiding misuse

➡ Use of suitable cutting tools.

➡ Adapting the speed adjustment and feed to the material and workpiece.

➡ Clamp workpieces firmly and vibration-free.

ATTENTION!

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

WARNING!

Risk of injury caused by flying workpieces.

Clamp the workpiece in the machine vice. Make sure that the workpiece is firmly clamped in the machine vice and that the machine vice is firmly clamped onto 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 as a milling tool. 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 has been tested for operational safety. The construction and type are state of the art.

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

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

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

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

INFORMATION

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

❍ be duly qualified,
❍ and strictly follow these operating instructions.

In the event of improper use

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

Always switch off the CNC machine and disconnect it from the mains, when cleaning or maintenance work is carried out.

WARNING!

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

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

This is your responsibility being the operating company!

1.7 Qualification of personnel

1.7.1 Target group

This manual is addressed to

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

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

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

INFORMATION

Unclear responsibilities constitute a safety risk!

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

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

Operator

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

Qualified electrician

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

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

Qualified personnel

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

Instructed person

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

1.7.2 Authorized personnel

INFORMATION

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

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

SinuTrain made by Siemens is the perfect software-supplement for the G0876 CNC Mill.

This training software supports the rapid training for the operation of the control 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 to the personnel, objects and the environment.

Only authorized personnel may operate the CNC machine !

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

The operating company must

❍ train the personnel,
❍ instruct the personnel in regular intervals (at least once a year) on
  - all safety standards that apply to the CNC machine,
  - operation of the CNC machine,
  - generally accepted engineering standards.
  - possible emergency situations,

- ❍ check the personnel's knowledge level,
- ❍ document training/instruction in a operation book,
- ❍ require personnel to confirm participation in training/instructions by means of a signature,
- ❍ check whether the personnel is working safety and risk-conscious and observes the operating instructions.
- ❍ define and document the inspection deadlines for the machine.

The operator must

- ❍ be specially trained in handling and programming the CNC machine,
- ❍ know and understand the program sequence and the effects of the individual process parameters,
- ❍ keep an operator's log,
- ❍ before taking the machine in operation
  - have read and understood the operating manual,
  - be familiar with all safety devices and instructions.

Obligations of the operator

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

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

Additional requirements regarding the qualification

1.8 Operator positions

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

1.9 Safety devices

Use the CNC machine only with properly functioning safety devices.

Stop the CNC machine immediately, if a safety device fails or is faulty or becomes ineffective.

It is your responsibility!

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

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

WARNING!

If you bypass, remove or deactivate a safety device in any other way, you are endangering yourself and other personnel working with the CNC machine. The possible consequences are:
- ❍ **injuries due to tools, workpieces or fragments hereof which are flying off at high speed,**
- ❍ **contact with rotating or moving parts,**
- ❍ **fatal electrocution,**
- ❍ **pulling-in of clothes.**

The CNC machine features 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 "0" position, the lockable main switch can be secured against accidental or non-author-ised switching on by means of a padlock.

The power supply is interrupted by switchingoff the main plug.

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

Img. 1-1:   Main switch

WARNING!

Dangerous voltage even if the main switch is switched off.

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

1.9.2   Emergency stop button

ATTENTION!

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

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

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

Img. 1-2:   Emergency-stop push button

1.9.3   Control technical protection

WARNING!

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

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

If you temporarily bypass a controller in exceptional cases (e.g. during electrical repairs), you must continuously monitor the CNC machine.

1.9.4   Polycarbonate windows

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

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

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

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

☞ "Cleaning and replacing of the polycarbonate windows" on page 248

1.9.5   Prohibition, warning and mandatory labels

INFORMATION

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

1.10   Safety check

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

Check all safety devices

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

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

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

INFORMATION

Organise the checks according to the following table;

General check		
Equipment	**Check**	**OK**
Protective housing	Switching function, firmly bolted and not damaged	
Signs, Markers	Installed and legible	
Sight window	Check for mechanical damage (scratches, cracks). cracks, ☞ "Polycarbonate windows" on page 17	
Date:	**checked by (signature):**	

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

1.11 Personal protective equipment

For certain work personal protective equipment is required.

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

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

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

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

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

CAUTION!

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

1.12 Safety during operation

WARNING!

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

Avoid any unsafe work methods:

- ❍ The instructions mentioned in these operating instructions have to be strictly observed during assembly, operation, maintenance and repair.
- ❍ Do not work on the CNC machine, if your concentration is reduced, for example, because you are taking medication.
- ❍ Stay on the CNC machine until the program is terminated.
  The running program can be identified by means of the signal lamp.
  - Green light: Program run active
  - Yellow light: Malfunction
  - Red light: Actuated emergency stop push button

Img. 1-3:   Signal lamp

- ❍ Safely and firmly clamp the workpiece in place, before switching the CNC machine on.
- ❍ 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 (aluminum-/magnesium alloys), spontaneously inflammable or explosive particles (powder, dust, chips) might be generated, which might cause a fire and/or explosion.

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

Only the following extinguishing agents are permissible:

○ **solid extinguishing agent of fire class D (fires involving metals)**
○ **dry covering salts for magnesium**
○ **a mixture of sand and cast chips**
○ **argon (Ar) or nitrogen (N_2)**

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

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

1.13   Safety during maintenance

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

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

1.14   Disconnecting and securing the CNC machine

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

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

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

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

Place a warning sign on the CNC machine.

WARNING!

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

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

1.14.1 Using lifting equipment

WARNING!

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

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

Fasten the loads carefully.

Never walk under suspended loads!

1.14.2 Mechanical maintenance work

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

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

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

Check if they are working properly!

1.15 Unattended Operation

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

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

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

1.16 Accident report

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

There are many possible causes for "near misses".

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

INFORMATION

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

1.17 Electrical system

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

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

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

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

The deadlines must be set so that arising, foreseeable defects can be detected in a timely manner.

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

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

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

1.18 Inspection deadlines

Define and document the inspection deadlines for the machine.

1.19 Clamping devices for workpieces and tools

ATTENTION!

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

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

1.20 Tools and tool holding fixtures

CAUTION !

When using tools with larger diameters or at higher speeds!

The balancing of the tools has to amount to

❍ 0 - 6000 min $^{-1}$ - G 6,3
❍ from a speed of 6000 min $^{-1}$ - G 2,5

according to DIN / ISO 1940.

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.

⚠ WARNING

Electrocution, fire, shock, or equipment damage may occur if machine is not properly grounded and connected to power supply.
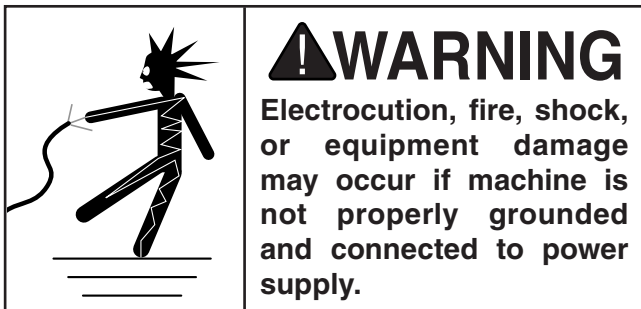
Full-Load Current Rating

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 ...14.1 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 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 208V, 220V, 230V, 240V
Cycle..60 Hz
Phase.. 3-Phase
Power Supply Circuit .... 20 Amps
Plug/Receptacle .... NEMA 15-20
Cord .... "S"-Type, 4-Wire, 12 AWG, 300 VAC

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.*

Grounding Instructions

This machine MUST be grounded. In the event of certain malfunctions or breakdowns, grounding reduces the risk of electric shock by providing a path of least resistance for electric current.

The power cord and plug specified under "Circuit Requirements for 220V" on the previous page has an equipment-grounding wire and a grounding prong. The plug must only be inserted into a matching receptacle (outlet) that is properly installed and grounded in accordance with all local codes and ordinances (see figure below).

GROUNDED 15-20 RECEPTACLE

15-20 PLUG

Current Carrying Blades

Grounding Pin
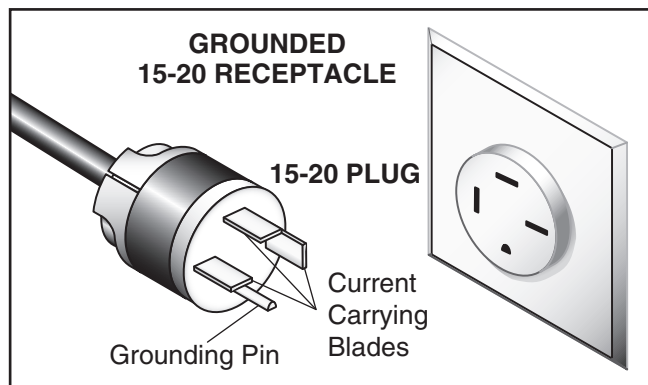
Typical 15-20 plug and receptacle.

⚠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.

⚠CAUTION
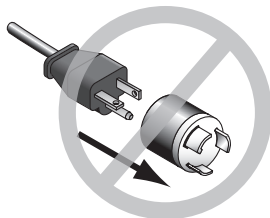
No adapter should be used with plug. If plug does not fit available receptacle, or if machine must be reconnected for use on a different type of circuit, reconnection must be performed by an electrician or qualified service personnel, and it must comply with all local codes and ordinances.

⚠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.

Improper connection of the equipment-grounding wire can result in a risk of electric shock. The wire with green insulation (with or without yellow stripes) is the equipment-grounding wire. If repair or replacement of the power cord or plug is necessary, do not connect the equipment-grounding wire to a live (current carrying) terminal.

Check with a qualified electrician or service personnel if you do not understand these grounding requirements, or if you are in doubt about whether the tool is properly grounded. If you ever notice that a cord or plug is damaged or worn, disconnect it from power, and immediately replace it with a new one.

Extension Cords

We do not recommend using an extension cord with this machine. If you must use an extension cord, only use it if absolutely necessary and only on a temporary basis.

Extension cords cause voltage drop, which can damage electrical components and shorten motor life. Voltage drop increases as the extension cord size gets longer and the gauge size gets smaller (higher gauge numbers indicate smaller sizes).

Any extension cord used with this machine must be in good condition and contain a ground wire and matching plug/receptacle. Additionally, it must meet the following size requirements:

Minimum Gauge Size ....10 AWG
Maximum Length (Shorter is Better).......50 ft.

3   Assembly and commissioning

INFORMATION

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

3.1   Scope of delivery

Compare the delivery volume with the attached packing list.

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

Check if all parts are firmly seated.

3.2   Transport

WARNING!

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

○ **Centers of gravity**

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

○ **Prescribed transportation position
(Marking of the top surface)**

○ **Means of transport to be used**

○ **Weights**
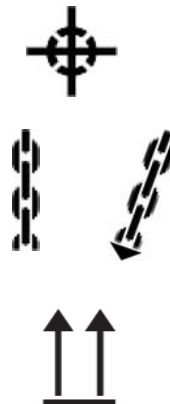
WARNING!

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

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

Observe the accident prevention regulations.

Fasten the loads carefully.

Never walk under suspended loads!

➜ Check the substructure. The substructure must bear the load.

➜ Disassemble the lateral parts of the wooden box.

➜ The CNC machine is lifted and transported with an appropriate handling device to the installation place by means of a fork-lift truck.

➜ Disassemble the clamping bolts which are used to fix the machine on the pallet.

➜ Lift the CNC machine carefully from the pallet of the transportation box by means of a crane or a fork-lift truck. ☞ "Total weight on page 3.

➜ Bring the CNC machine with an appropriate handling device, e.g. Electric pallet truck or fork-lift truck at their firm position.

➜ Make sure that no add-on-pieces are damaged or cause paintwork is damaged during transport.

WARNING!

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

3.3   Installation and assembly

3.3.1 Requirements regarding the installation site

Organize the working area around the CNC machine according to the local safety regulations. The working area for operating, maintenance and repair must not be restricted.

INFORMATION

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

3.3.2 Installation plan

Connection compressed air

Electrical supply connection
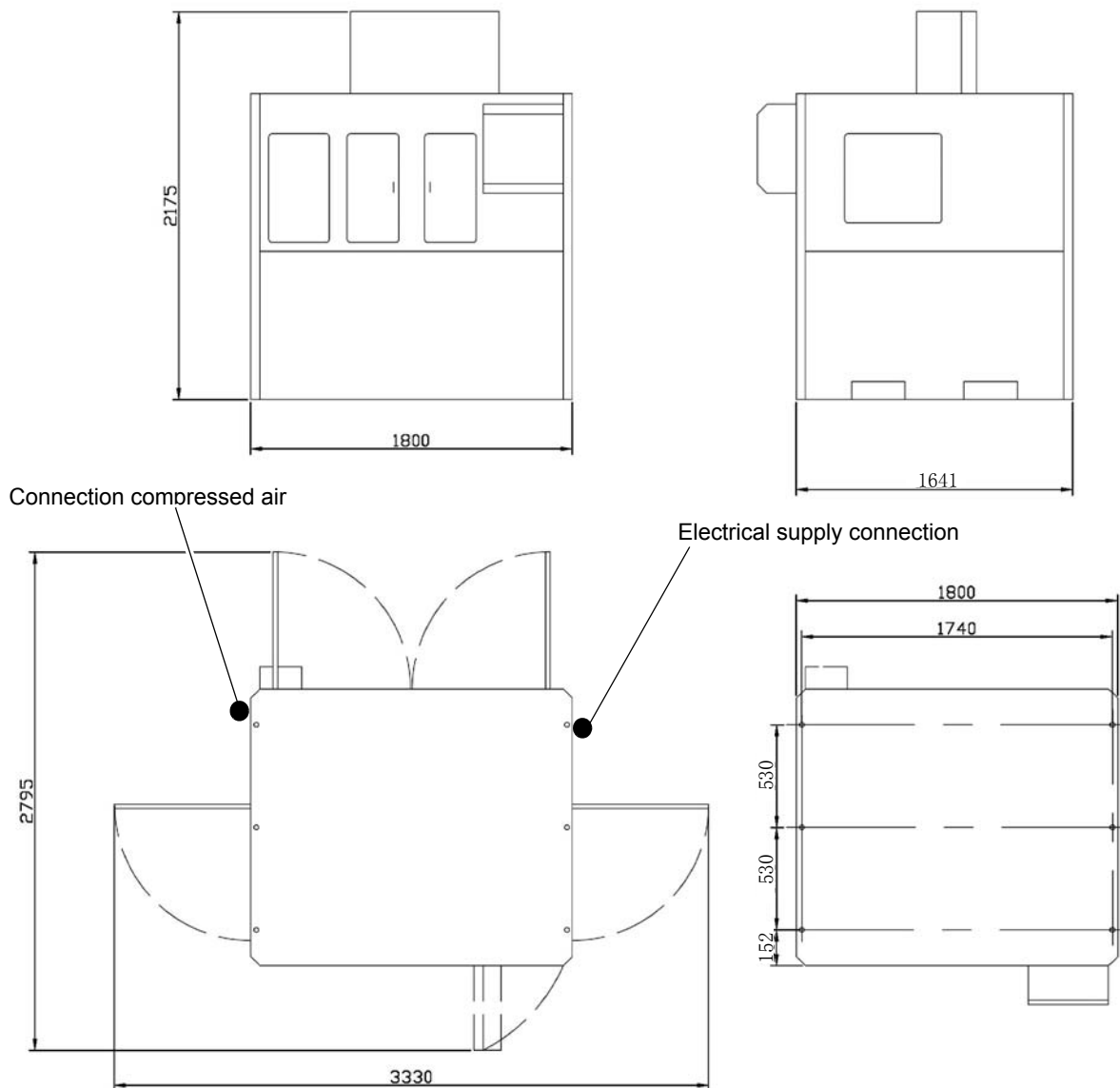
3.3.3 Machine mounting

Anchor-free assembly

➜ Align the CNC machine with a machine spirit level. The slope deviation of all levels must not exceed 0.04/1000mm.

➜ Adjust the height by screwing in or screwing out of the levelling screw. Screwing in the levelling screw causes that the rubber plate lifts itself as on the drawing from the same element.

➜ Fix the height adjustment by the the jackscrew with the help of the counternut.

➜ Check the alignment of the machine after a few days of usage.

Jackscrew

Counternut
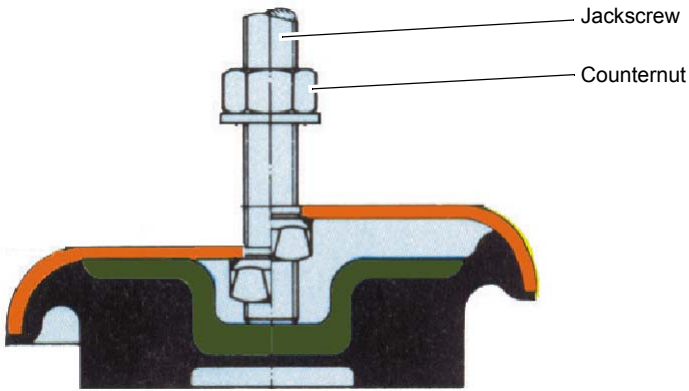
Img.3-1:   Oscillating element

Anchored assembly

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

ATTENTION!

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

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

3.3.4 Corrosion protection

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

3.3.5 Electrical connection

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

➜ Firmly connect the machine.

CAUTION!

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

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

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

ATTENTION!

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

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

ATTENTION!

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

ATTENTION!

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

ATTENTION!

Ensure that all 3 phases (L1, L2, L3) and the ground wire are connected correctly.

The neutral conductor (N) of its power supply is not connected.

3.3.6 Current in the Protective Earth Ground Wire

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

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

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

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

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

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

Type AC should never be used in converters.

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

Type B must be used for 3-phase converters.

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

Line systems

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

Prohibited operation

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

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

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

Permissible line supplies

Operation on TN and TT line systems

TN line system

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

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

TT system

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

3.3.7   Connection compressed air supply

➜ Open the cover and connect the compressed air supply with at least 6.5 bars to the quick-action coupling of the compressed air maintenance unit.

➜ Adjust a pressure of 6.3 bars using the set screw of the maintenance unit.

Manometer

Air pressure switch

Compressed air supply

Pneumatic oiler

Compressed air drainage

Img. 3-2:   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.

Throttle valve

1R3

Drip feed lubricator

1V4

Cylinder tool changer

Cylinder tool clamping   1M3

1M6

Throttle valve

1R5.1

Throttle valve

1Q3

1R5.2

Quick exhaust valve

1Y3
Solenoid valve   A | B   A 1

1Y5   A | B

Solenoid valve   B 1

R | P | S   A 2

A 2   R | P | S   B 2

Compressed air service unit

Manometer

1P3

1K4

1K1
Filter

1V3

1K.2

Pneumatic oiler

Air pressure switch

Pressure controller
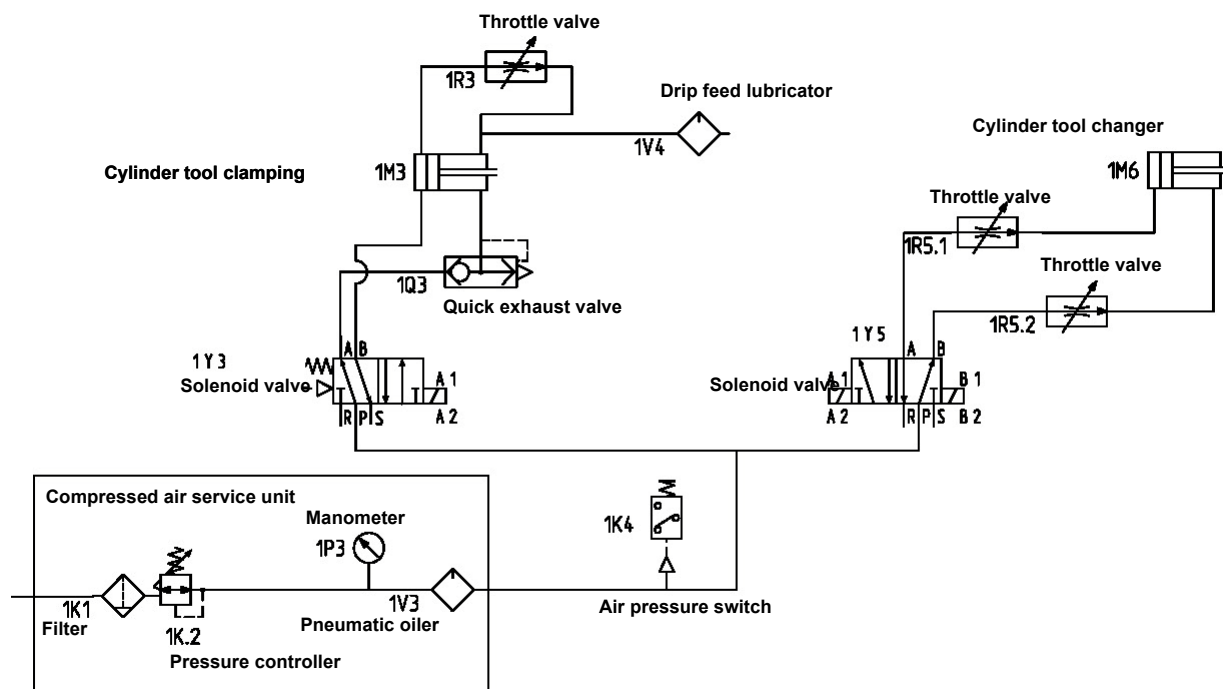
3.4 First commissioning

3.4.1 Refilling the cooling lubricant

INFORMATION

The CNC machine is delivered without cooling lubricant.

→ Fill the coolant / lubricant tank with an appropriate cooling lubricant via the machining room CNC machine. ☞ "Cooling lubricants" on page 243
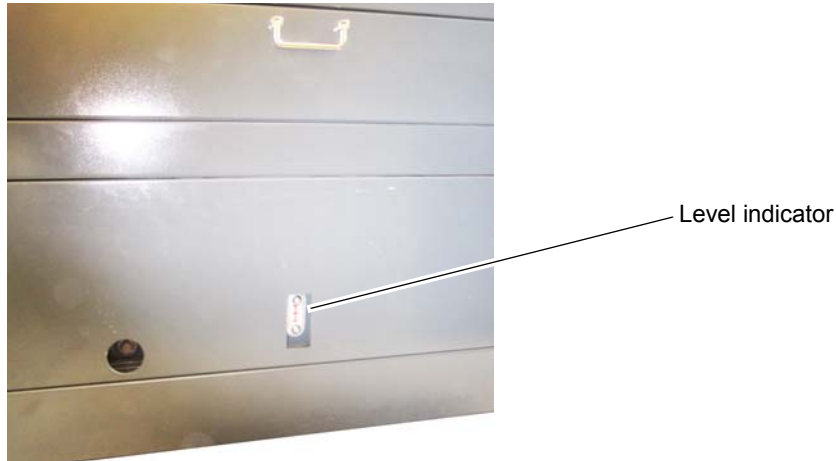
Img. 3-3:   Level indicator coolant / lubricant

ATTENTION!

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

INFORMATION

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

Make sure that the cooling lubricant is properly absorbed.

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

3.4.2 Cooling lubricants

INFORMATION

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

The company Grizzly Industrial, Inc. does not assume any guarantee for subsequent damages due to unsuitable cooling lubricants.

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

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

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

Therefore, Grizzly Industrial, Inc. cannot be held liable for machine damages caused by unsuitable coolants and lubricants as well as by inadequate maintenance and servic-ing of the coolant. In case of problems with the cooling lubricant and the slideway oil or grease, please contact your mineral oil supplier.

CAUTION!

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

☞ **"Cooling lubricants and tanks" on page 251**

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

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

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

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

3.5 Refill central lubrication system

The CNC machine is equipped with a central lubrication system.

INFORMATION

The CNC milling machine is delivered without lubricating oil. The central lubrication system is depending on the model in the working area on the column of the spindle or at other locations.

➜ Refill the reservoir with oil through the fill cap.
☞ "Cooling lubricants" on page 243

Img.3-4:   Old type of central lubricating system

Img.3-5:   Second type of central lubricating system

Type:	DRB 12210
Pressure	2.5 MPa
Power	10 Watt
Connection	230V ~ 50Hz / 115V ~ 60Hz
filling quantity	2 litres
discharge volume	96 ml / min

Setting the jumpers

Factory setting

30   75

Switch-on time (seconds)

5   10   20   30

Interval (minutes)

prohibited	5	10
15	20	25	30	35
40	45	50	55	60
65	70	75	80	85
90	95	100	105	110
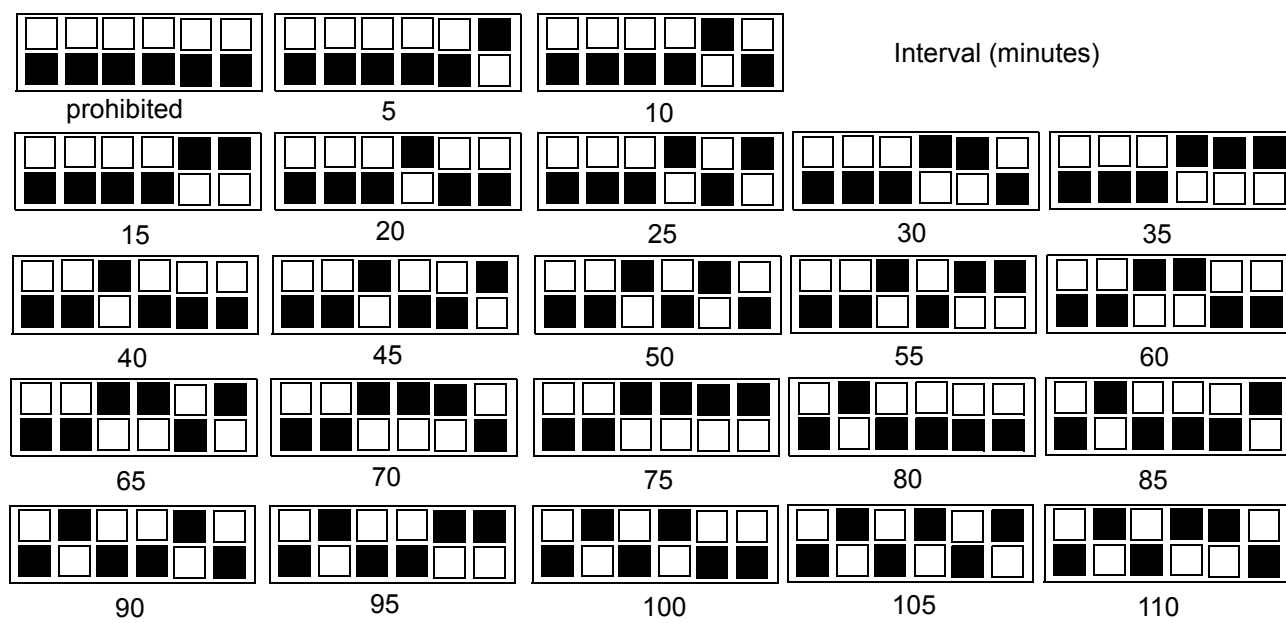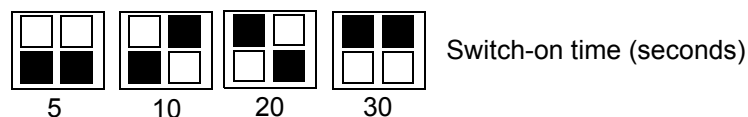
INFORMATION

The factory setting is 75 minutes as the interval between lubrication cycles and 30 seconds for the ON period of the lubrication oil pump.

Operating indicator lights DRB 12210

Status	Description	Note:
In Operation	The red LED lights	Oil is supplied to the lubrication points
Idling cycle	The red LED flashes	Pause between the lubrication cycles

The pressure relief valve of the central lubrication system is factory set to a pressure of 2.5 MPa.

3.6 Functional test and controls

Rotation coolant pump

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

INFORMATION

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

Proceeding to set the direction of rotation:

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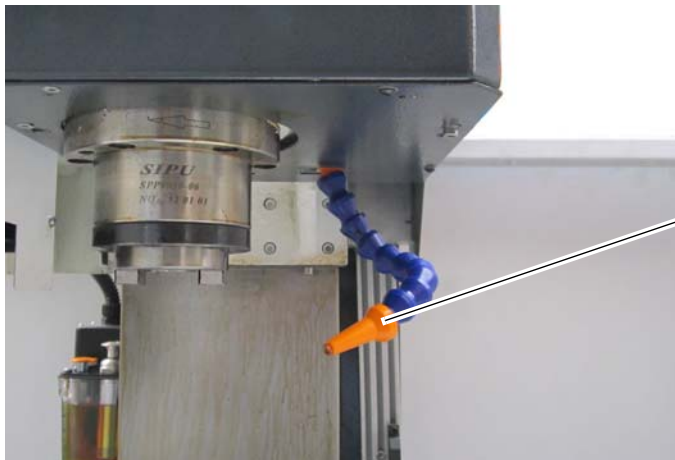
—————— Coolant supply

Img. 3-6:   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.

3.6.1 Warming up the machine

ATTENTION!

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

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

4 General information about CNC

4.1 Compensation of geometry

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

Required knowledge:

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

Basics:

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

4.2 Coordinate systems on CNC-machine tools

Types of coordinate systems

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

Generally they are divided into

❍ Cartesian coordinate system and
❍ the polar coordinate system

4.2.1 Cartesian coordinate system

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

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

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

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

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

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

4.2.2 Polar coordinate system

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

In the polar coordinate system a point is described by means of its distance (radius r) to the coordinate origin and its angle (a) to the defined axis. The angle (a) 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

.
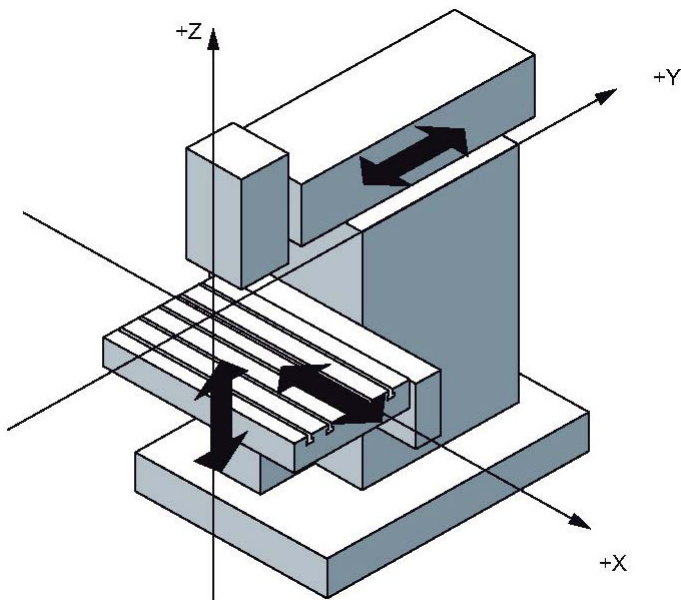
Img.4-1:   Vertical Z-axis

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°

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

Rectangular triangle

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

On a rectangular triangle the single sides are specially named.

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

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

In a rectangular triangle it applies:

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

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

The sum of the cathetus square is equal to the hypotenuse square and expressed as a equation:
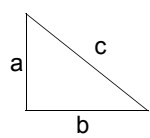
$$a^2 + b^2 = c^2$$

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 = opposite leg / adjacent leg

The results is:

opposite leg = adjacent leg x tan alpha

5 User interface, machine control panel

5.1 Screen arrangement

Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"
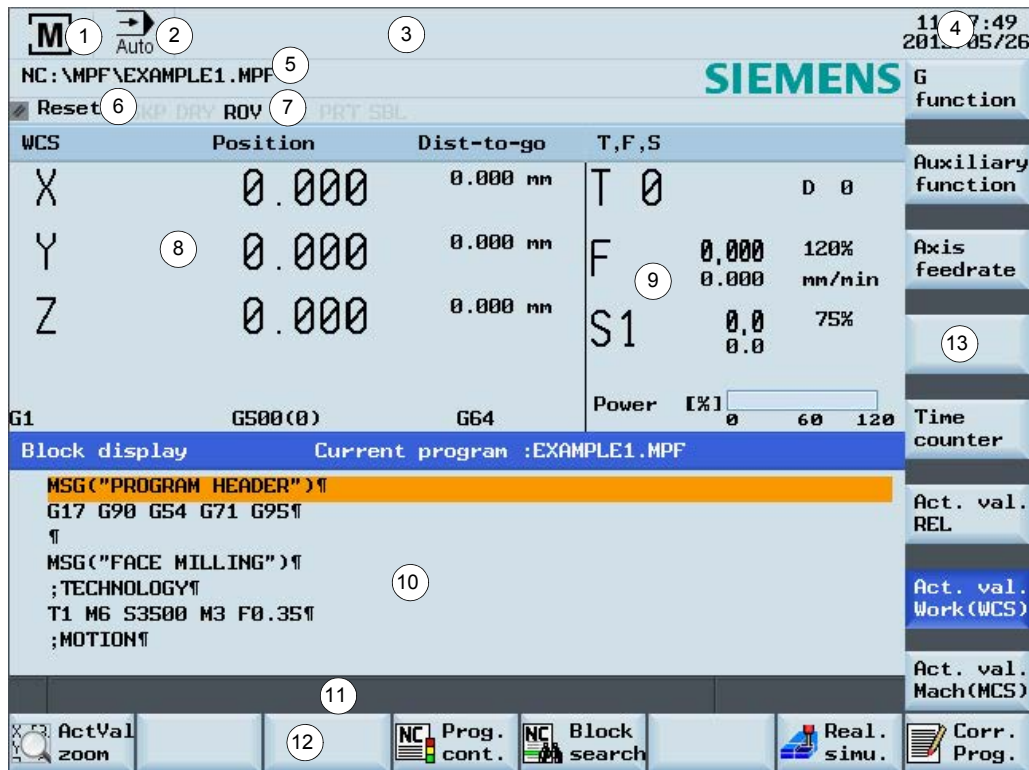
Img.5-1:   Screen arrangement

Status area

① Active operating area

② Active operating mode

③ Alarm and message prompt area

④ Current time and date

⑤ Program file name

⑥ Program status indication

⑦ Active program control modes

Application area

⑧ Actual value window

⑨ T, F, S window

⑩ Operating window with program block Mode Display / Data Display

Tip and softkey area
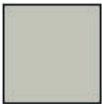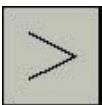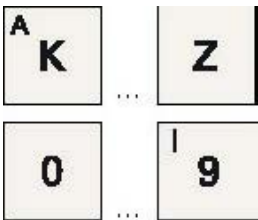
⑪ Information line

⑫ Horizontal softkey bar

⑬ Vertical softkey bar
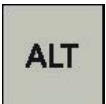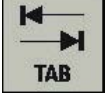
5.2   Elements on the PPU (Panel Processing Unit) front
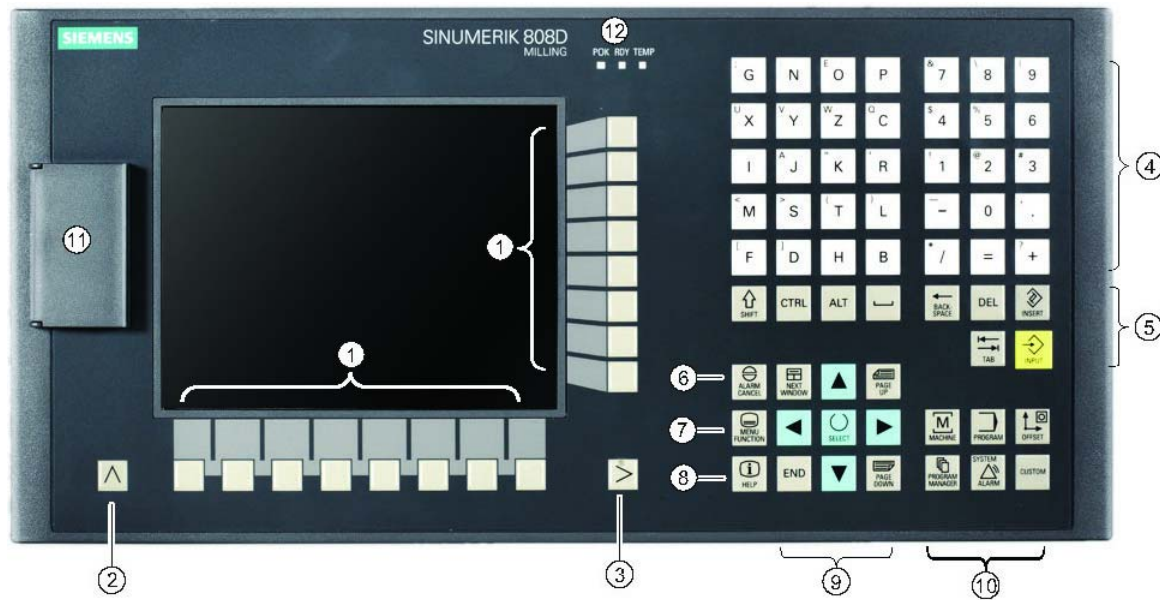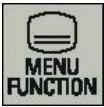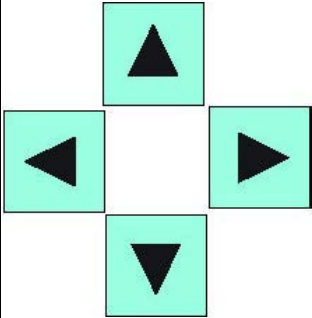
Img. 5-2:   Panel Processing Unit

Elements on the PPU		
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"		
		Description
①		**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	**Shift key**
	CTRL	**Control key**
	ALT	**Alternate key**
	⎵	**Space key**
	BACK-SPACE	**Backspace key** Deletes a character selected to the left of the cursor.
	DEL	**Delete key** Deletes the selected file or character.
	INSERT	**Insert key**
	TAB	**Tab key** • Indents the cursor by several characters. • Toggles between the input field and the selected program name.
	INPUT	Input key • Confirms your entry of a value. • Opens a directory or program.
⑥	ALARM CANCEL	**Alarm cancel key** Cancels alarms and messages that are marked with this symbol

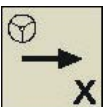| Elements on the PPU			
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"			
			Description
⑦		MENU FUNCTION	**Menu function key** Opens the wizard main screen.
⑧		ⓘ HELP	**Help key** Calls the context-sensitive help for the selected window, alarm, message, machine data, setting data, or end-user wizard.
⑨	Cursor keys		
		▲ ◄ ► ▼	**Cursor keys** up/down/left/right keys
		NEXT WINDOW	**Next window key** **No function is assigned to this key. Reserved for future use.**
		END	**End key** Moves the cursor to the end of a line.
		PAGE UP	**Page up key** Scrolls upwards on a menu screen
		PAGE DOWN	**Page down key** **Scrolls downwards on a menu screen**
		⟳ SELECT	**Selection key** • Toggles in selection lists and selection fields between several options. • 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	
	[M MACHINE icon]	Opens the "Machine" operating area
	[PROGRAM icon]	Opens the "Program" operating area
	[OFFSET icon]	Opens the "Offset" operating area
	[PROGRAM MANAGER icon]	Opens the "Program" operating area
	[SYSTEM ALARM icon]	Pressing this key opens the "Alarm" operating area. Holding down <SHIFT> while you press this key opens the "System" operating area.
	[CUSTOM icon]	Enables user's extension application, for example, to generate user dialogs with the EasyXLanguage function. For more information about this function, refer to the SINUMERIK 808D Function Manual.
⑪	**USB interface**	Connects to a USB device Examples: • Connects to an external USB memory sticker to transfer data between the USB sticker and the CNC. • Connects to an external USB keyboard for use as an external NC keyboard.
⑫	**Status LEDs** [POK RDY TEMP LED image]	**LED "POK"** Lights up green: The power supply for the CNC is switched on.
		LED "RDY" Lights up green: The CNC is ready for operation.
		LED "TEMP" Unlit: The CNC temperature is within the specified range. Lights up orange: The CNC temperature is out of range.

5.2.1 Key combination

Element	Description
Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"	
Element	**Description**
<ALT> + <X>	Opens the "Machine" operating area
<ALT> + <V>	Opens the "Program" operating area
<ALT> + <C>	Opens the "Offset" operating area
<ALT> + <B>	Opens the "Program" operating area
<ALT> + <M>	**Opens the "Alarm" operating area**
<ALT> + <N> <SHIFT> + [SYSTEM ALARM]	**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> + <B>	Selects text in program blocks.
<CTRL> + <C>	Copies the selected text.
<CTRL> + <D>	Shows pre-defined slides on the screen.
<CTRL> + <P>	Captures screens
<CTRL> + <R>	Restarts the HMI
<CTRL> + <S>	Saves start-up archives

5.3 Controls on the machine control panel
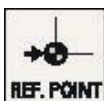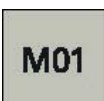
Img. 5-3: Machine control panel

Controls on the machine control panel	
Element	**Function**
	Drive voltage Push button with indicator light • Indicator light On, drive voltage activated • Indicator light Off, drive voltage deactivated
K11	**Manual tool change** Enables or disables the manual tool change. The manual tool change is only possible with the sliding door open.

	Element	**Description**
	Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"	
①		<Emergency stop button> Activate the button in situations where • life is at risk. • there is the danger of a machine or workpiece being damaged. All drives will be stopped with the greatest possible braking torque.
②		**Handwheel key (with an LED status indicator)** Controls the axis movement with external handwheels.
③		**Tool number display** Displays the current tool number
④	Operating mode keys (all with LED status indicators)	
		Operating mode "JOG"
		Operating mode "REF. POINT" (reference point approach)
		Operating mode "AUTO" (automatic mode)
		Operating mode "MDA" Manual program input, automatic execution
⑤	Program control keys (all with LED status indicators)	
		Program test key Disables the output of setpoints to axes and the spindle. The control system only "simulates" the traverse movements in order to verify the correctness of the program.
		Conditional stop key Stops the program at every block in which miscellaneous function M01 is programmed.
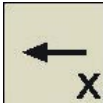
	Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"	
	Element	**Description**
	ROV	**Rapid override key** Adjusts axis feedrate override
	SINGLE BLOCK	**Single block key** Activates single block execution mode
⑥	User-defined keys (all with LED status indicators)	
	LAMP	**Machine luminaire control key** Pressing this in any operating mode switches on/off the machine luminaire. LED on: The machine luminaire is switched on. LED off: The machine luminaire is switched off.
	COOLANT	**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.
	DOOR	**Safety door control key** When all axes and the spindle stop operation, pressing this key unlocks the safety door. LED on: The safety door is unlocked. LED on: The safety door is locked.
	MAG CW	**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.
	MAG REF	**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.
	MAG CCW	**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.
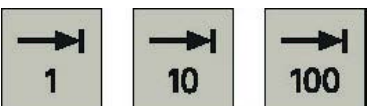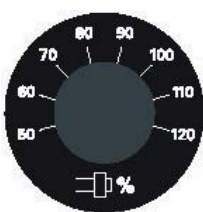
	Element	Description
	Excerpt from the manual of the operating instructions for "SINUMERIK 808D OPM Operation"	

	Element	Description
	CHIP FWD	**Forward rotation of the chip conveyor** (active only in JOG mode) Pressing this key in any operating mode starts the forward rotation of the chip conveyor. LED on: The chip conveyor starts forward rotation. LED off: The chip conveyor stops rotation. ℹ **INFORMATION** The G0876 does not have a chip conveyor. Therefore, the key has no function.
	CHIP REV	**Reverse rotation of the chip conveyor** (active only in JOG mode) Keeping pressing this key in any operating mode rotates the chip conveyor in reverse order. Releasing the key changes the chip conveyor to the previous forward rotation or stop state. LED on: The chip conveyor starts reverse rotation. LED off: The chip conveyor stops reverse rotation. ℹ **INFORMATION** The G0876 does not have a chip conveyor. Therefore, the key has no function.
	K9 … K12	**User-defined keys** ℹ **INFORMATION** The <K11> button is associated with the manual tool change.
⑦	Axis traversing keys	
	X	**X axis key** Traverses the X axis in the positive direction.
	X	**X axis key** Traverses the X axis in the negative direction.
	Z	**Z axis key** Traverses the Z axis in the negative direction.
	Z	**Z axis key** Traverses the Z axis in the positive direction.

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 feed rate 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 havebacked 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 Shift+ System Alarm ⚠ ALARM

Step 2

Set password ➜ Enter customer's or manufacturer's password.

Change password ➜ Change customer's or manufacturer's password.

Delete password ➜ Delete customer's or manufacturer's password.

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.

Eliminate or have all malfunctions rectified promptly. 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 authorisation. Notify the person responsible immediately of any modification.
☞ "Safety during operation" on page 18

6.2 Control and indicating elements
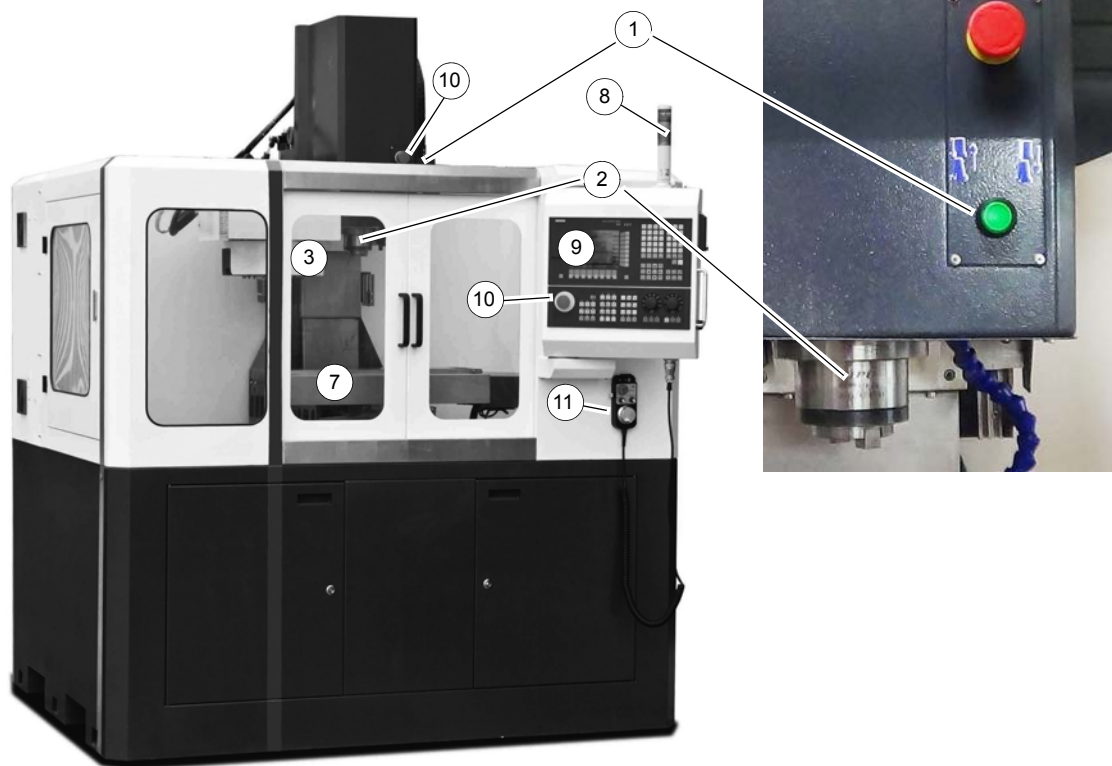
Img.6-1:

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 indicator lamp lights up green --> CNC- program is running)
9	Machine control panel
10	Emergency-stop button
11	Handwheel for manual travelling with Emergency Stop push button and acknowledgement button

Signal lamp

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

6.3 Operational modes

Manually controlled operation

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

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

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

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

Automatic mode

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

6.4 Programming

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

Manual resp. part programming:

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

Automatic programming:

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

6.5 Operation of the machine

6.5.1 Turning on the CNC- machine

➜ Switch on the main switch. ☞ "Lockable main switch" on page 15

➜ Wait until the control is completely started.

➜ Press the push button "Drive control ON" (1) .

➜ Unlock the "Emergency Stop push buttons" (2) .
  - on the spindle head (not visible in the picture),
  - on the MCP,
  - on the electronic handwheel.

➜ Close - if not yet closed - the sliding door.

➜ Press the push button "Reset"   (3) .

Img.6-2:   Operating area

6.5.2 Reference point approach after turning on

INFORMATION

After turning on the G0876 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 G0876 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.

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

Make sure that the "feed override switch" is not set to "zero".
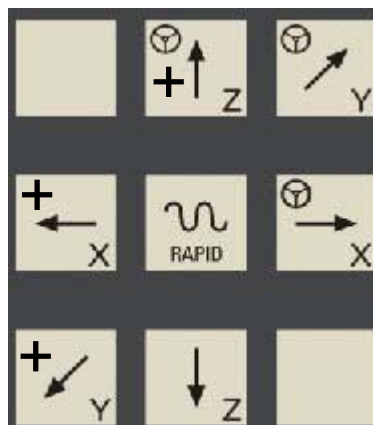
➜ Traverse each axis to the machine zero point until the referenced symbol is shown on the respective axis.

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

Model G0876 (Mfd. Since 12/22)

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.
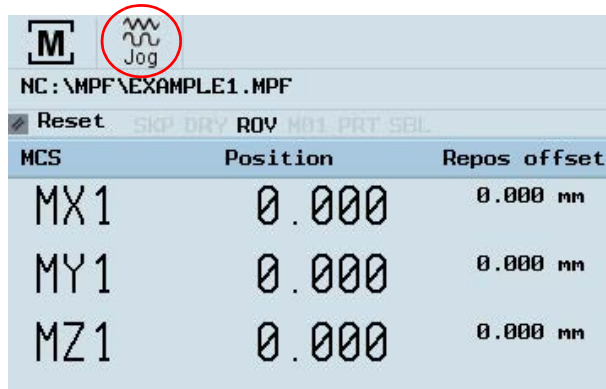
→ For the following working steps, please proceed as described in „Operation and programming" of the Siemens SINUMERIK 808D Operation instructions.
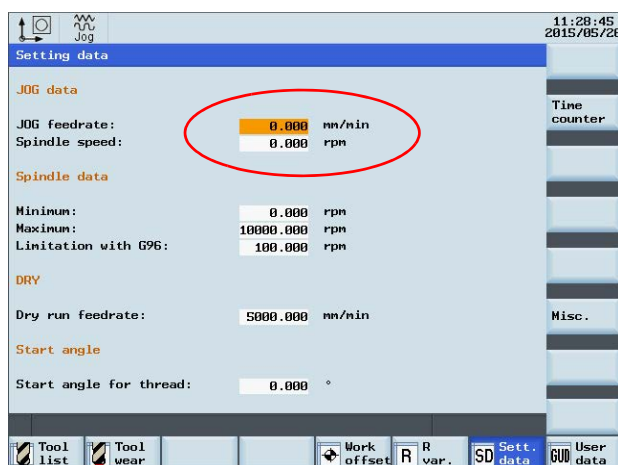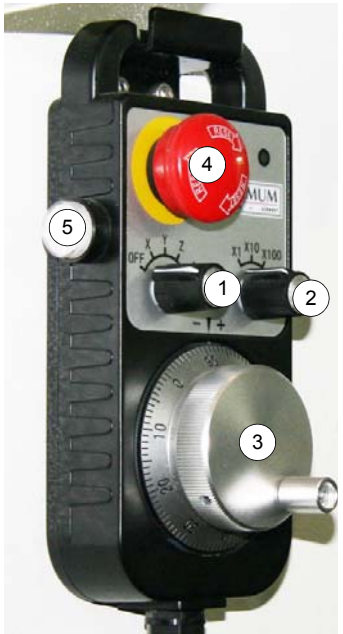
6.5.3 Using the electronic handwheel

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 hand-wheel.

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

WARNING!

A manual movement of axes with open door is not possible. The G0876 has no acknowledgment 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   Insert tool

INFORMATION

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

➜ Change over to the mode <JOG>.

➜ Press the "Offset" key on the machine control panel.

➜ Press the "Tool list" softkey

➜ For the following working steps, please proceed as described in „Operation and programming" of the Siemens SINUMERIK 808D Operation instructions.
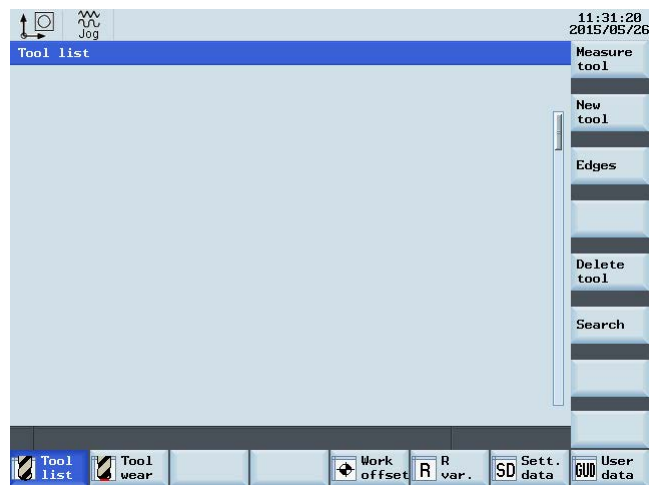
6.5.5   Manual tool change

ATTENTION!

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

➜ Press the push button "Open/Close door" and open the sliding door.

➜ Check that the LED on the push button <K11>lights up.
Press - if necessary - the push button <K11>. The manual tool change is thereby enabled again.

➜ Press the push button "manual tool change" to loosen or to clamp the tool.
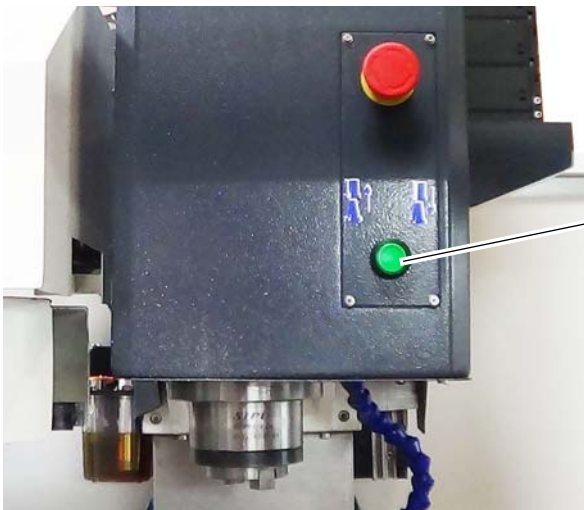
Push button manual
Changing the tool

Img.6-3:   Spindle head

6.5.6 Clamping the workpiece

ATTENTION!

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

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

WARNING!

Risk of injury caused by flying workpieces.

➡ Clamp the workpiece in the machine vice.

➡ Make sure that the workpiece is firmly clamped in the vise.

Workpiece

Machine vice
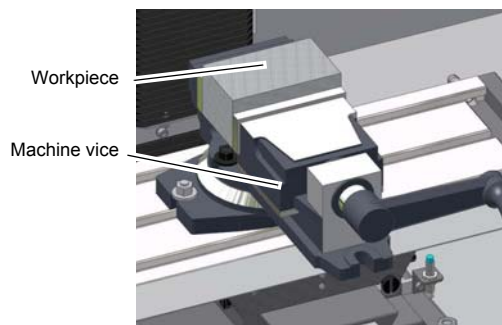
Img.6-4:   Clamping the workpiece

6.5.7 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 push button <K11>lights up.
Press - if necessary - the push button <K11>. The manual tool change is thereby enabled again.

➡ Press the push button "manual tool change" to loosen or to clamp the tool.
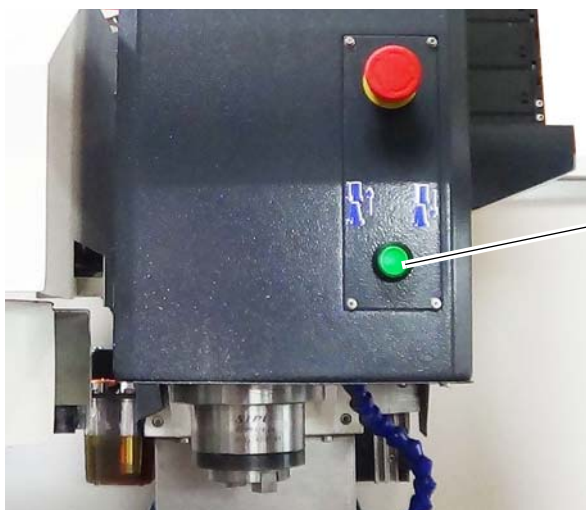
Push button manual Changing the tool

Img.6-5:   Spindle head

6.5.8 Turning off the CNC- machine

➜ Push the EMERGENCY stop push button.

For a long-term standstill of the CNC machine, switch it off at the main switch.
☞ "Disconnecting and securing the CNC machine" on page 19

6.6 Operational modes

Manually controlled operation

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

Automatic mode

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

6.7 Programming

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

Manual resp. part programming:

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

Automatic programming:

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

DIN Code and ISO Code:

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

6.8 Starting the program

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

WARNING!

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

CAUTION!

Before starting the program, the sliding door of the separating protective equipment must be closed.
- ❍ **Completely close the separating protective equipment.**
- ❍ **Change to "AUTO/MDA" mode.**

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

6.9   Central lubrication system

The CNC machine is equipped with a central lubrication system.

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

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

6.10   Data interfaces and current collection

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

The control is equipped with the following data interfaces.

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

6.11   Selecting the speed

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

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

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

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

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

Tool diameter [mm] shell end mill and plain milling cutter	Steel 10 - 25 m/min	Grey cast iron 10 - 22 m/min	Al alloy age-hardened 150 - 350 m/min
	Speed [min^{-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

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

INFORMATION

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

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

INFORMATION

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

7 Brief instruction 808D Milling

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

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

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

7.1 Preparation

Description

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

Content

Basic Theory

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.
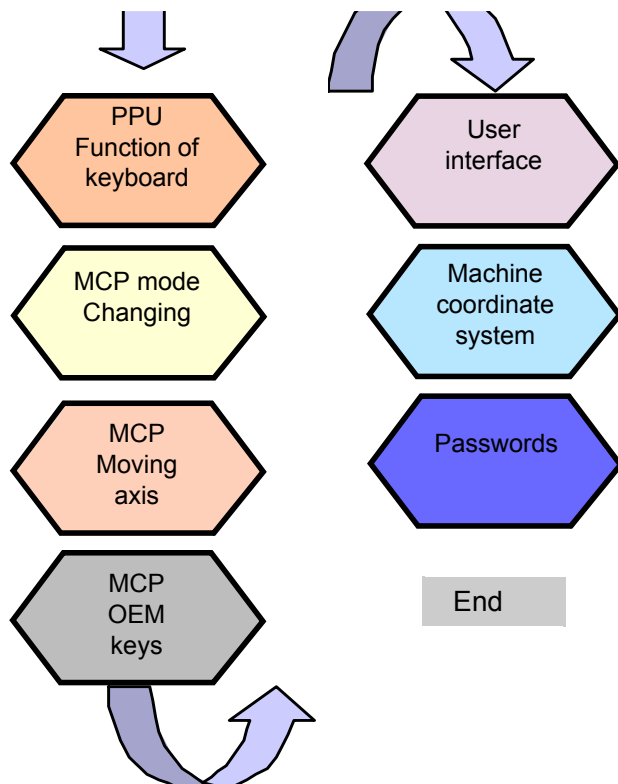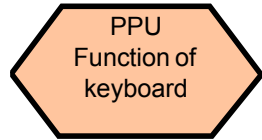
Menu navigation

Operating area navigation

MCP mode
Changing
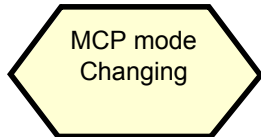
Mode navigation

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

JOG - MDA - AUTO

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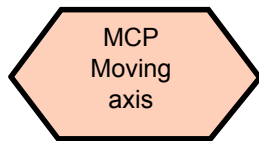
Axis movement
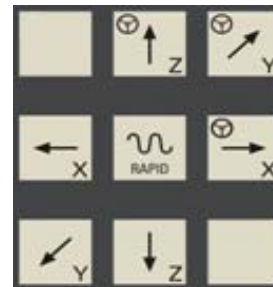
The 808D machine control panel (MCP) is used to control manual operation of the axis.

The machine can be moved with the appropriate keys.

MCP
OEM
keys
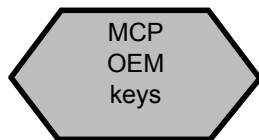
OEM keys

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

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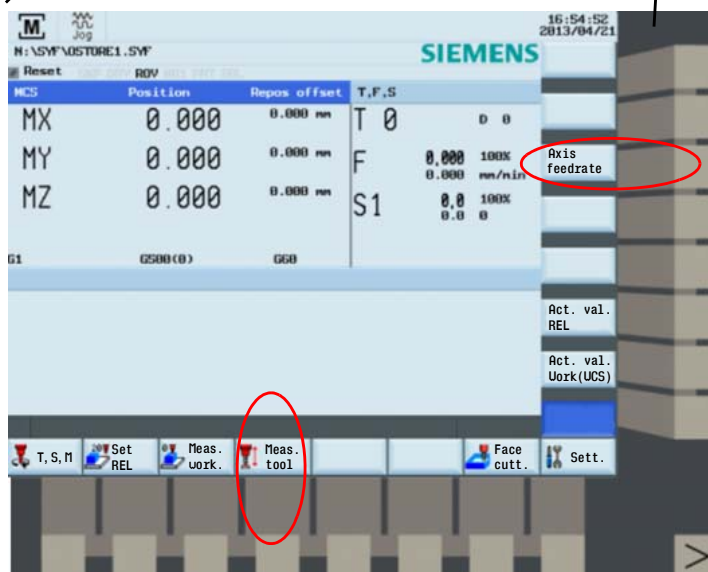
808D (PPU) has eight
vertical softkeys (abbr.
SKs) on the right of the
screen. These SKs can
be activated with the
corresponding button
(located on the right).

808D (PPU) has eight horizontal SKs on the bottom of the screen. These SKs can
be activated with the corresponding button (located below).
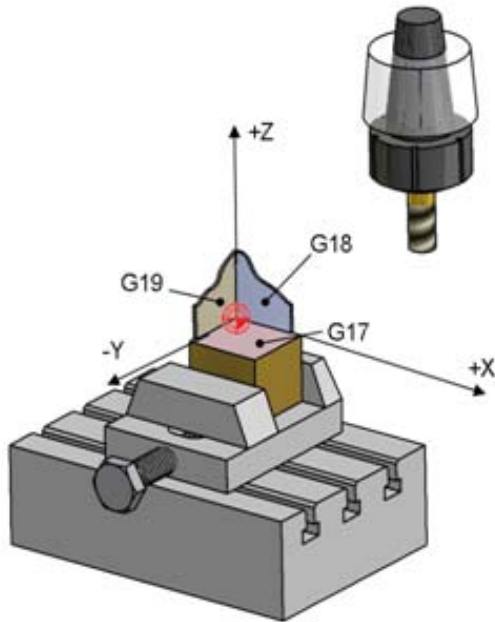
The Sinumerik 808D uses a coordinate system which is derived from the DIN 66217 standard.

The system is an international standard and ensures compatibility between machines and coordinate programming.

The primary function of the coordinate system is to ensure that the tool length and tool radius are calculated correctly in the respective axis.

SEQUENCE

Passwords

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

No password   Machine operator

Customer's password   Advanced operator

Manufacturer's password   OEM engineer

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.

SHIFT + SYSTEM ALARM →
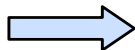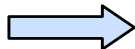
Set password

Change password

Delete password

Step 2

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

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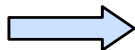
Delete password → Delete customer's or manufacturer's password

End

7.2 Switch On and Referencing

Description

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

Content

```
        ⬇

   ╱‾‾‾‾‾‾‾‾╲
  ╱ Switch on ╲
  ╲   the     ╱
   ╲ machine ╱
    ╲‾‾‾‾‾‾╱

   ╱‾‾‾‾‾‾‾‾╲
  ╱ Reference ╲
  ╲   the     ╱
   ╲ machine ╱
    ╲‾‾‾‾‾‾╱

   ┌──────────┐
   │   End    │
   └──────────┘
```

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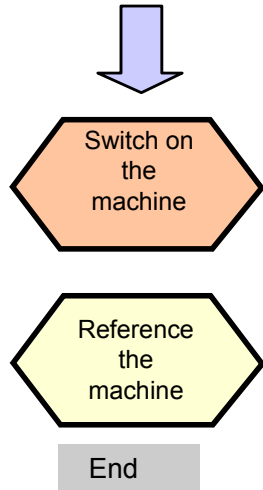
Switch on
the
machine
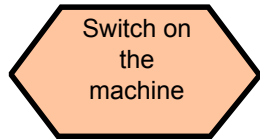
Please note the explicit switching on rules as specified by the machine manufacturer.

Step 1

Turn on the main switch of the machine.
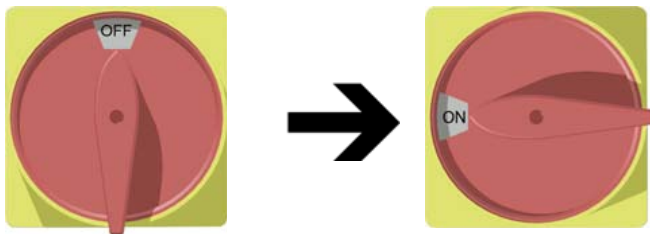
Step 2

Make sure you perform the following operation!

Release all the EMERGENCY STOP buttons on the machine!

End

SEQUENCE

Reference the machine
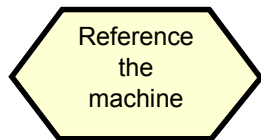
If your machine is configure with ABS encoder, you do not need to reference the axis of the machine.

If your machine is fitted with INC encoder, After power on, the machine must first be referenced!

Step 1
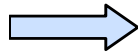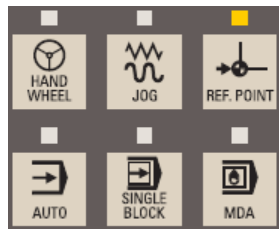
After power on, the machine will be in the reference point approach mode (default).

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

Step 2
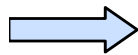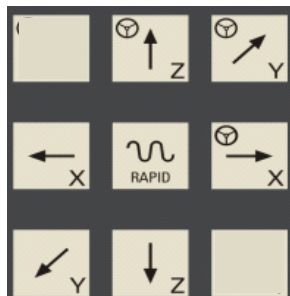
The axes are referenced with the corresponding axis traversing keys.

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

Model G0876 (Mfd. Since 12/22)
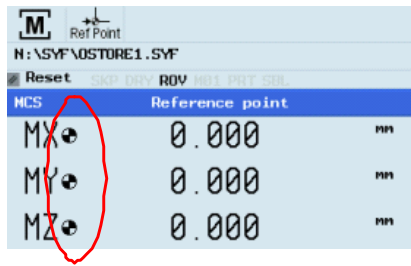
After completing the referencing procedure for all axes, the referenced symbol is displayed next to the axis identifier.

Step 3
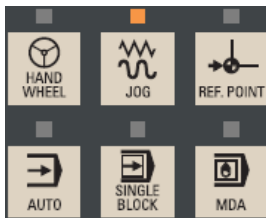
After returning to JOG mode, use the axis traversing keys to move the machine manually.
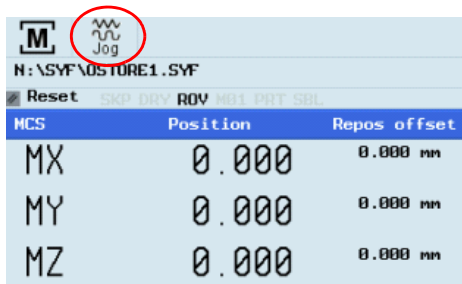
The machine can now be operated in JOG mode.

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

End

Model G0876 (Mfd. Since 12/22)

7.3 Tool Setup

Description

This unit describes how to create and set up tools.

Content

```
        │                    ╭──────╮
        ▼                    ╰───►

   ⬡ Create          ⬡ Start
     tool               spindle

   ⬡ Create tool      ⬡ Measure
     edge               tool

   ⬡ Load tool into   ⬡ JOG
     active             spindle
     position

   ⬡ Move             ⬡ Execute M
     machine with       function
     handwheel
        ╰────►           End
```
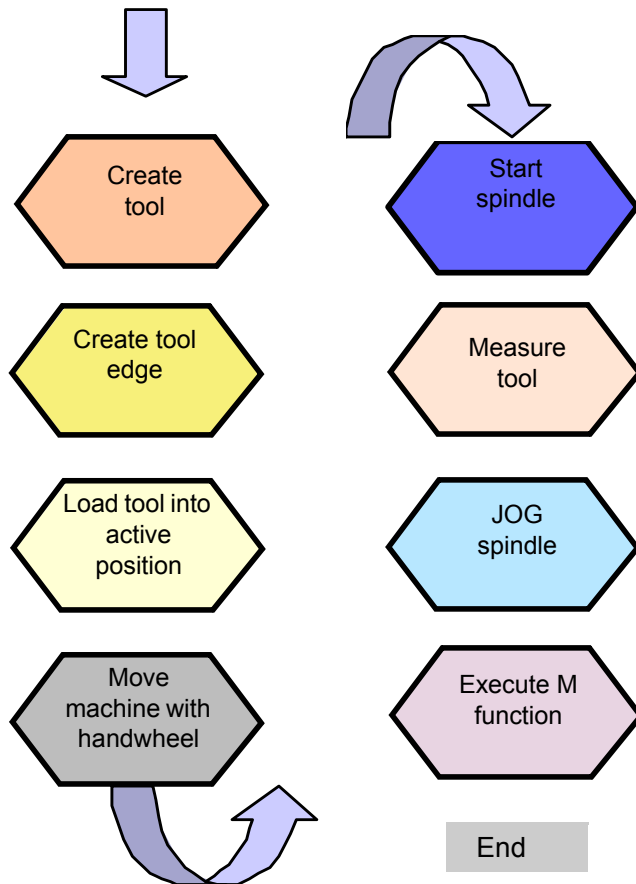
SEQUENCE
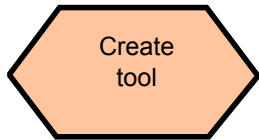
Create
tool

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

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

Press "Offset" on the PPU.   OFFSET

Press the "Tool list" SK on the PPU.   Tool
list

SEQUENCE

Step 2

Press the "New tool" SK on the PPU. ⟹ New tool

Select the type of tool required. ⟹ Milling tool

Enter "1" at "Tool No." Drilling tool

Tapping tool

Ball end mil. tool

Tool list

New milling tool

Tool No.: 1

Identifier
milling tool
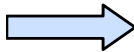
Press the "OK" SK on the PPU. ⟹ OK ✓

Enter the "Radius" of the milling tool.

Jog 11:19:04 2012/04/25

Tool list

Type	T	D	Geometry	
			Length1	Radius
	1	1	0.000	4

Milling tool

Drilling tool

Press the "Input" button on the PPU. ⟹ INPUT
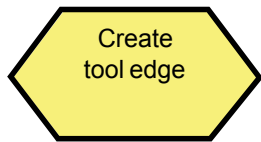
Create tool edge

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

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

Press the "Offset" key on the PPU.

Press the "Tool list" SK on the PPU.
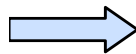
Use direction keys to select the tool which needs to add a tool edge.   Or
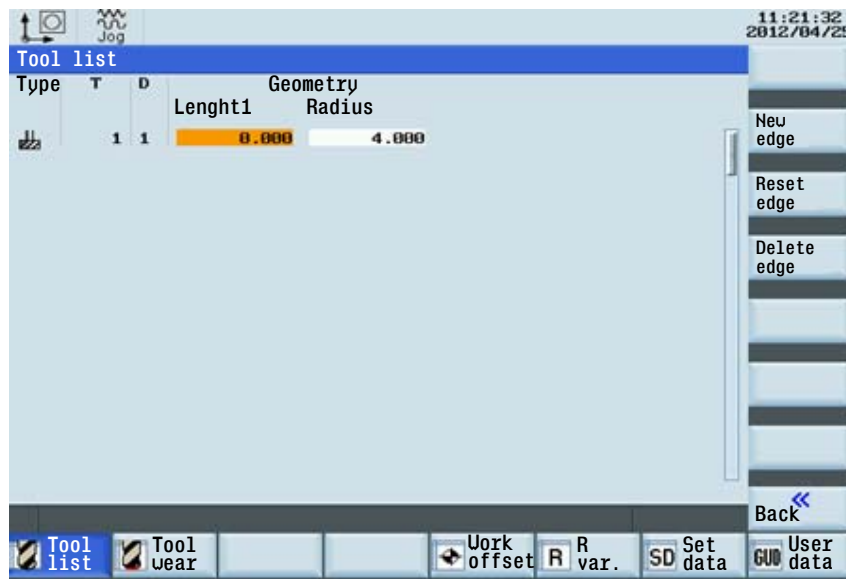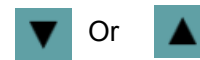
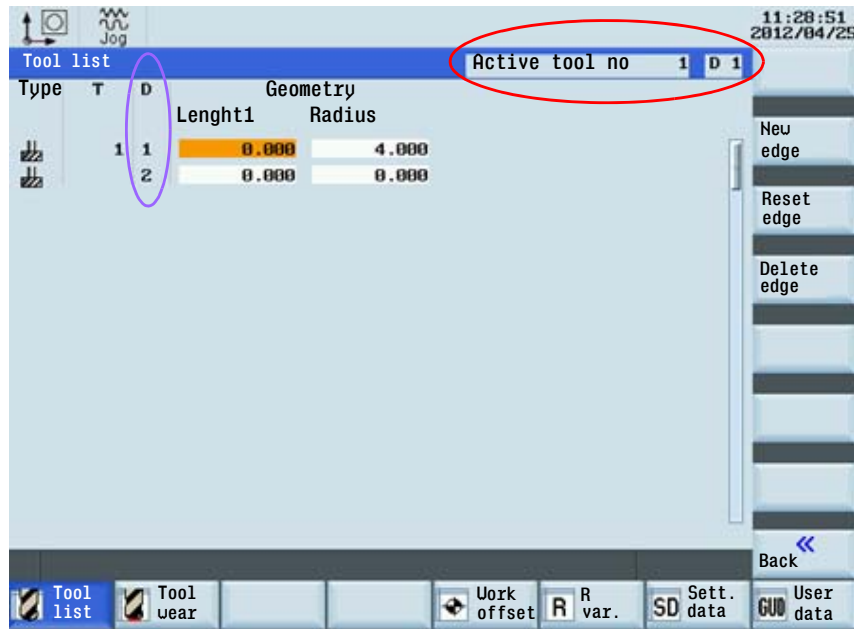Press the "Edges" SK on the PPU.
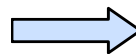
Press the "New edge" SK on the PPU.

SEQUENCE

Step 2

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

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

A maximum of nine tool edges can be created for each tool!

Different tool lengths and radii can be saved in different tool edges as required.

Please select the right tool edge for machining according to requirement!
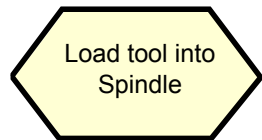
A tool must have been created in the system before it can be loaded into the active position.

Press the "Machine" key on the PPU ⟹ [M MACHINE]

Press the "JOG" key on the MCP ⟹ [JOG]

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

Enter tool number "1" in "T" ⟹ [INPUT]

```
T,S,M
Tool change          1      D1
Spindle speed               rpm
Spindle direction        ↻
Activate zerooffset      ↻
Other M function
    Press ´CYCLE START´ ⊙ to activate above functions.      Back «
  T,S,M   Set    Meas.   Meas.                   Face    Sett.
          rel    work.   tool                    cutt.
```

Press "CYCLE START" on the MCP ⟹ [CYCLE START]

```
 M    Jog                                          11:30:07
                                                   2012/04/25
                                         SIEMENS
 Reset   SKP DRY ROV M01 PRT SBL
 MCS       Position     Repos offset   T,F,S
  X        0.000        0.000 mm    T 1           D  1
  Y        0.000        0.000 mm    F   0.000  100%
                                        0.000  mm/min
  Z        0.000        0.000 mm    S1    0.0  100%
                                          0.0  0
```

Press the "Back" SK on the PPU ⟹ back «

SEQUENCE

The tool are usually loaded manually into the spindle.

The tool will be automatically loaded into the spindle with an automatic tool changer.
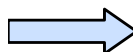
Move machine with handwheel
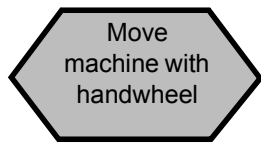
Make sure there is no obstruction when moving the tool to avoid a crash.

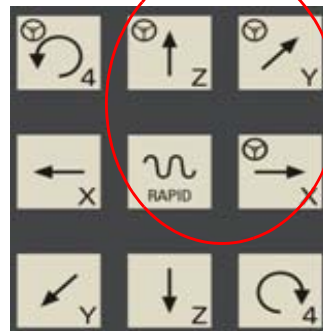A handwheel can control the axis motion instead of the "JOG" button.

Press the "Machine" key on the PPU ➡️ 🅼 MACHINE

Press the "Handwheel" key on the MCP ➡️ HAND WHEEL

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

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

MCS	Position	Repos offset
⊚X	0.000	0.000 mm
Y	0.000	0.000 mm
Z	0.000	0.000 mm

Select the required override increment according to the buttons on the right(this selection fits all axes) ⟹ [→ 1] [→ 10] [→ 100]
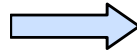
The handwheel increment is "0.001 mm" ⟹ [→ 1]
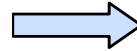
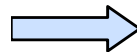The handwheel increment is "0.010 mm" ⟹ [→ 10]

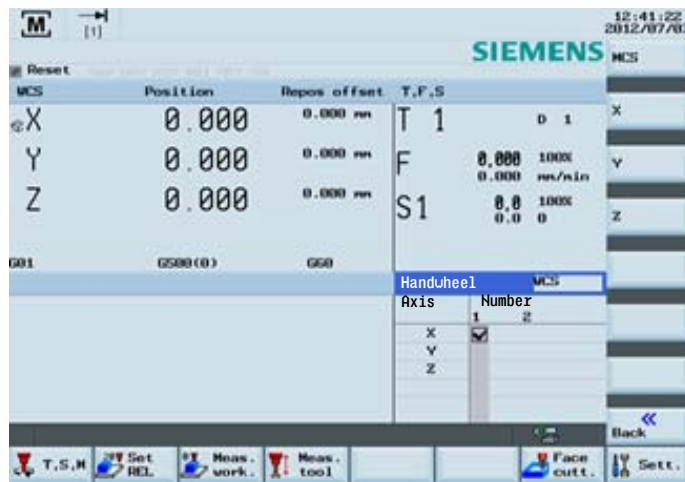The handwheel increment is "0.100 mm" ⟹ [→ 100]
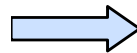
The selected axis can now be moved with the handwheel.

Press "JOG" on MCP to end the function of "Handwheel". ⟹ [JOG]

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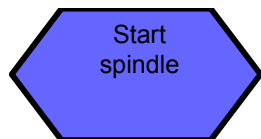

Handwheel

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

SEQUENCE

Start
spindle

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

Start the spindle before adjusting tools as follows:
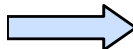
Press the "Machine" key on the PPU →
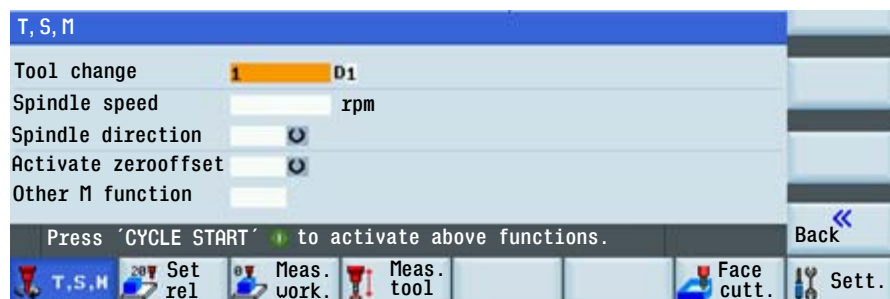
Press the "JOG" key on the MCP →

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

Enter "500" at "Spindle speed" → ▼

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

T,S,M		
Tool change	1	D1
Spindle speed		rpm
Spindle direction	↻	
Activate zerooffset	↻	
Other M function		

Press ´CYCLE START´ 🔵 to activate above functions.   Back ≪

| T,S,M | Set rel | Meas. work. | Meas. tool | | | Face cutt. | Sett. |

Press the "CYCLE START" key on the MCP →

OPTIMUM

Reset

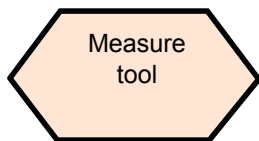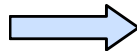| MCS	Position	Repos offset	T,F,S		
X	0.000	0.000 mm	T 1		D 1
Y	0.000	0.000 mm	F	0.000	100%
				0.000	mm/min
Z	0.000	0.000 mm	S1	500.0	100%
				500.0	0

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

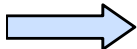Press the "Back" SK on the PPU

Measure tool

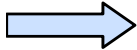A tool must have been created and loaded before it can be measured!

Step 1   Measure length
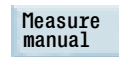
Press the "Machine" key on the PPU

Press the "JOG" key on the MCP

Press the "Meas. tool" SK on the PPU

Press the "Measure manual" SK on the PPU
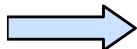
SEQUENCE

Press the axis keys on the MCP to move the tool to the set position above the workpiece.
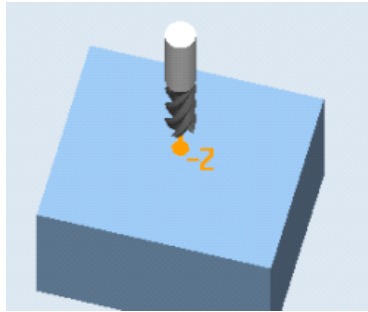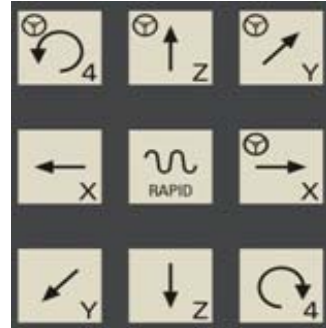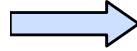
Note: The following text describes the required settings in the workpiece coordinate system

"X / Y / Z" zero points as:"X0" / "Y0" / "Z0"

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

or
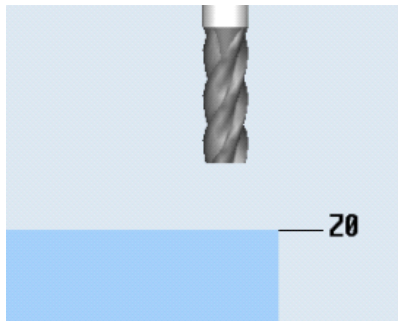
Move directly to zero point
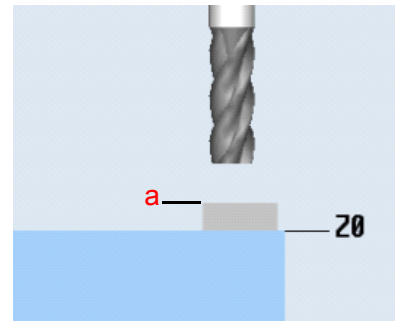
Use a setting block.

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

⟹

SELECT

Enter "0" for "Z0"

(If the setting block is used, then the value would be thickness a)

⟹

INPUT
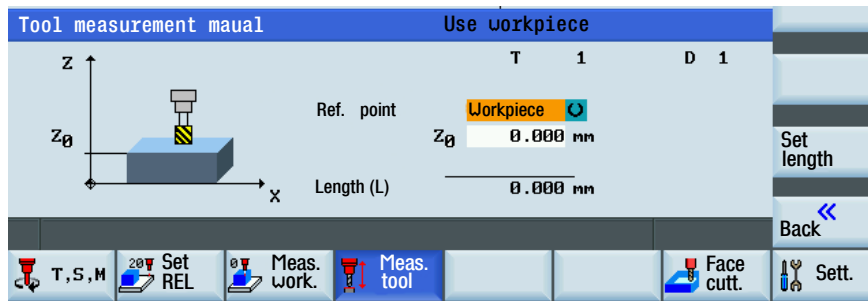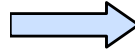
Press the "Set length" SK on the PPU

⟹

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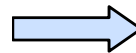
The measured tool length is now shown in "Length (L)". This value is also saved in the length value column of the corresponding tool list at the same time.

Step 2	Measure diameter

Press the "Diameter" SK on the PPU

⟹

Diameter
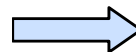
SEQUENCE

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

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

                    or

Move directly to zero point   Use a setting block.

Enter "0" at "X0"

Enter "0" at "Y0"

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

Tool measurement manual — Workpiece edge

Press the "Set diameter" SK on the PPU

Press the "Back" SK on the PPU

Model G0876 (Mfd. Since 12/22)
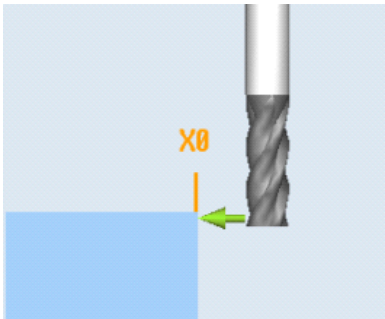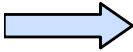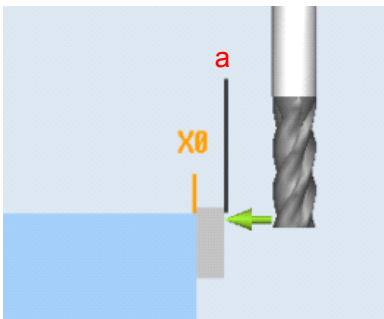
SEQUENCE

JOG spindle
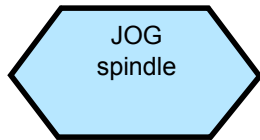
A tool must be loaded to the spindle.

Press the "Machine" key on the PPU → [M MACHINE]

Press the "JOG" key on the MCP → [HAND WHEEL]
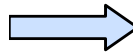
Press the spindle direction key on the MCP to start/stop the spindle. → [SPINDLE LEFT] [SPINDLE STOP] [SPINDLE RIGHT]
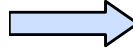
Press "Spindle left" on the MCP to start the spindle in the counter-clockwise direction. → [SPINDLE LEFT]
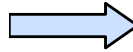
Press "Spindle stop" on the MCP to stop the spindle. → [SPINDLE STOP]
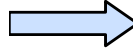
Press "Spindle right" on the MCP to start the spindle in the clockwise direction. → [SPINDLE RIGHT]
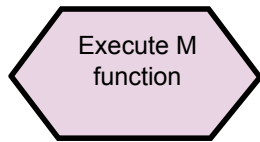
Execute M function

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

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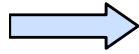
Press the "T.S.M" SK on the PPU.

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

T,S,M	
Tool change	D1
Spindle speed	rpm
Spindle direction	↻
Activate zerooffset	↻
Other M function	8

Press ´CYCLE START´ ⏻ to activate above functions.   Back «

| T,S,M | Set rel | Meas. work. | Meas. tool | | | Face cutt. | Sett. |
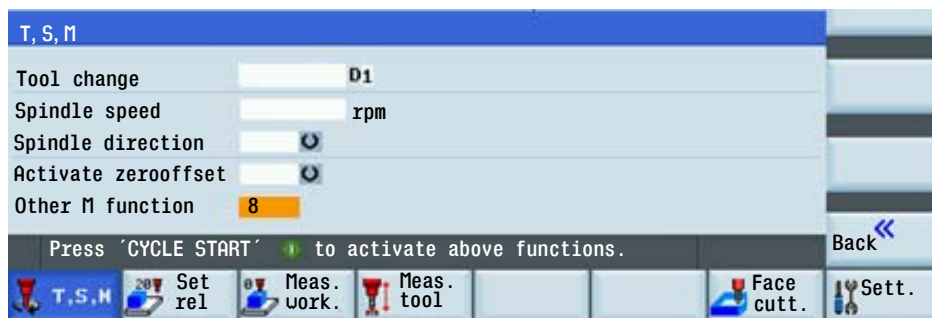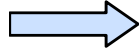
Press "CYCLE START" on the MCP.

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.
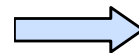
7.4 Workpiece Setup

Description

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

Content

```
        ↓

  ╱─────────╲
 ╱  Manual   ╲
│   start     │
 ╲  spindle  ╱
  ╲─────────╱

  ╱─────────╲
 ╱  Create   ╲
│ workpiece   │
 ╲  offset   ╱
  ╲─────────╱

  ╱─────────╲
 ╱ Test tool ╲
│offset results│
 ╲           ╱
  ╲─────────╱

  ┌─────────┐
  │   End   │
  └─────────┘
```

SEQUENCE

Manual start spindle
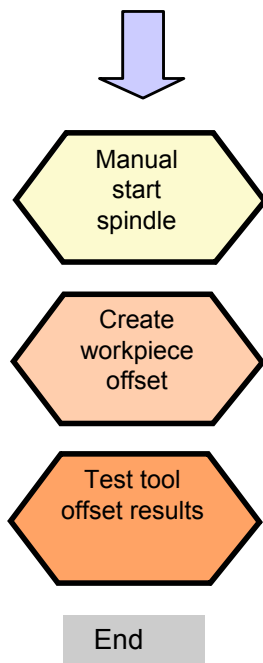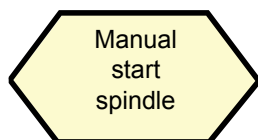
A tool must have been loaded into the spindle.

Before measuring, the spindle can be started as follows:

Press the "Machine" key on the PPU

Press the "JOG" key on the MCP

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

Enter "500" at "Spindle speed"

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

T,S,M	
Tool change	D1
Spindle speed	500 rpm
Spindle direction	M3
Activate zerooffset	
Other M function	

Press ´CYCLE START´ to activate above functions.

T.S.M | Set rel | Meas. work. | Meas. tool | | | Face cutt. | Sett.

Back
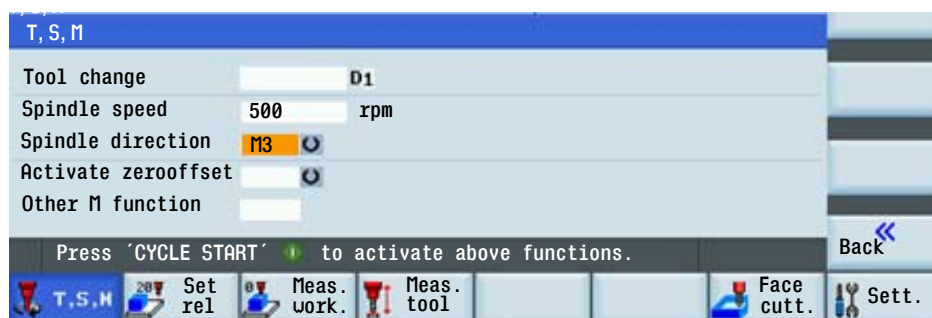
Press the "CYCLE START" key on the MCP

SEQUENCE

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

Press the "Back" SK on the PPU

Create workpiece offset

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

Make sure the active tool is the measured tool!

Press the "Machine" key on the PPU.
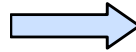
Press the "JOG" key on the MCP.

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

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

Model G0876 (Mfd. Since 12/22)
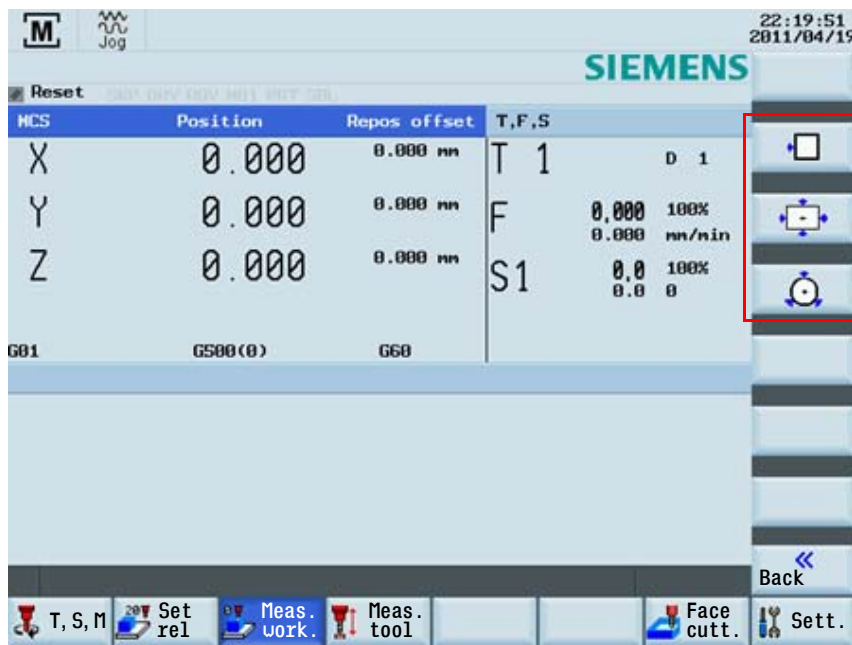
SEQUENCE

| Method 1 | This method is normally for setting the zero point of the workpiece at the edge of the workpiece. |

Using a tool that has a measured "Tool length & radius", move the tool to a known position on the workpiece. Using either JOG or Handwheel, scratch an edge and then calculate the zero point of the workpiece.

The process of setting the "X" zero point ("X0") is described below.

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

Press the appropriate SK to select the feed axis which needs to be set up.
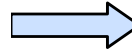
Press the axis traverse keys to move the tool to the required setting position in the X axis.
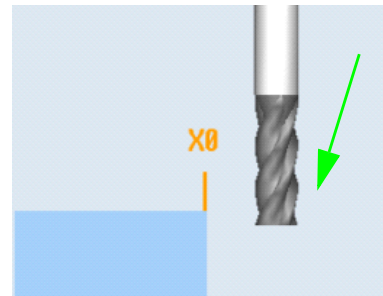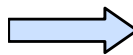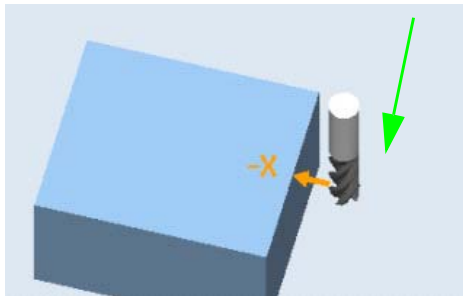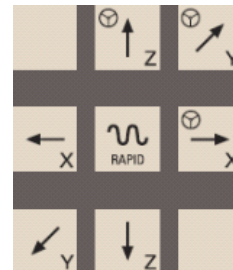
Press the "Handwheel" key on the MCP to position the tool at the X0 edge of the workpiece.
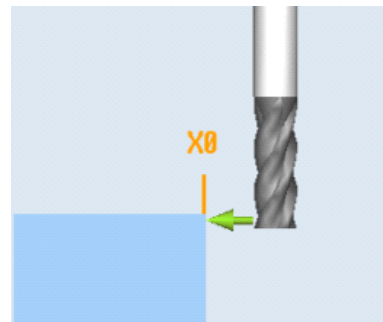
Select "Save in" Offset "G54" (or other offset).

Select "Measuring direction" as "-".

(This value should be chosen according to realities)

Set "Distance" as "0".

Press the "Set WO" SK on the PPU.

"Step 2" must be repeated for the setting of Y and Z zero points.

If you change the tool because of wear/damage during the machining process, you must remeasure the length of the tool.

SEQUENCE

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

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

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

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

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

Save P1

Save P1

Save P2

Repeat the process for positions 2, 3 and 4.

(When the setting is complete, the buttons will be shown in blue.)

Save P3

Save P4

Press the "Set WO" SK on the PPU.

Set UO

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

Workpiece measurement, center of rectangle

Work offset   G54

Offset   X₈   33.400 mm
                Y₈   136.300 mm

Compensation data have been activated!

Save P4

Set UO

Back

T,S,M   Set rel   Meas. work.   Meas. tool   Face cutt.   Sett.
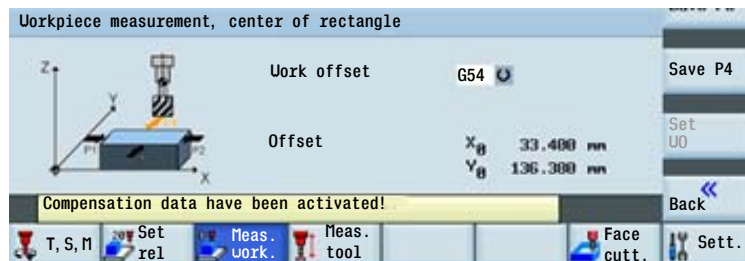
Method 3 This method is normally used for setting the zero points at the center point of a circular workpiece.

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.)
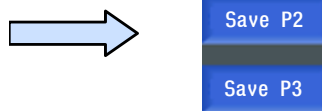
Save P1

Save P2

Save P3

Press the "Set WO" SK on the PPU.

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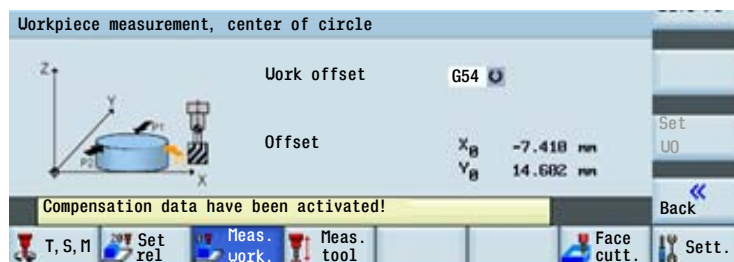
You have then finished setting the zero point of the workpiece as the center point of the circular workpiece.

Workpiece measurement, center of circle

Work offset   G54

Offset X_B -7.418 mm
                   Y_B 14.682 mm

Compensation data have been activated!

T,S,M | Set rel | Meas. work. | Meas. tool | | Face cutt. | Sett.

Set UO

« Back

SEQUENCE

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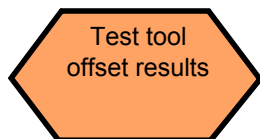
The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

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

Press the "Machine" key on the PPU ➡ **M** MACHINE
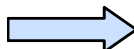
Press the "MDA" key on the MCP. ➡ MDA

Press the "Delete file" SK on the PPU. ➡ Delete file
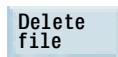
Enter the test program recommended on the right. (can also be customized) ➡ **G54 (select offset panel as required)**
T1 D1
G00 X0 Y0 Z5
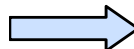
Press the "ROV" key to ensure the "ROV" function is active (lit up). ➡ ROV
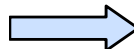
Note: The ROV function activates the feedrate override switch under the G00 function.

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

Press "CYCLE START" on the MCP. ➡ CYCLE START

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

7.5   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

Program structure	Definition of target position	Milling circles and arcs
Create program	Rapid motion	Moving to a fixed position
Edit program	Tools and motion	Controlling the spindle
Imperial and Metric system	Behaviors at corners	Setting a delay in the program
		End
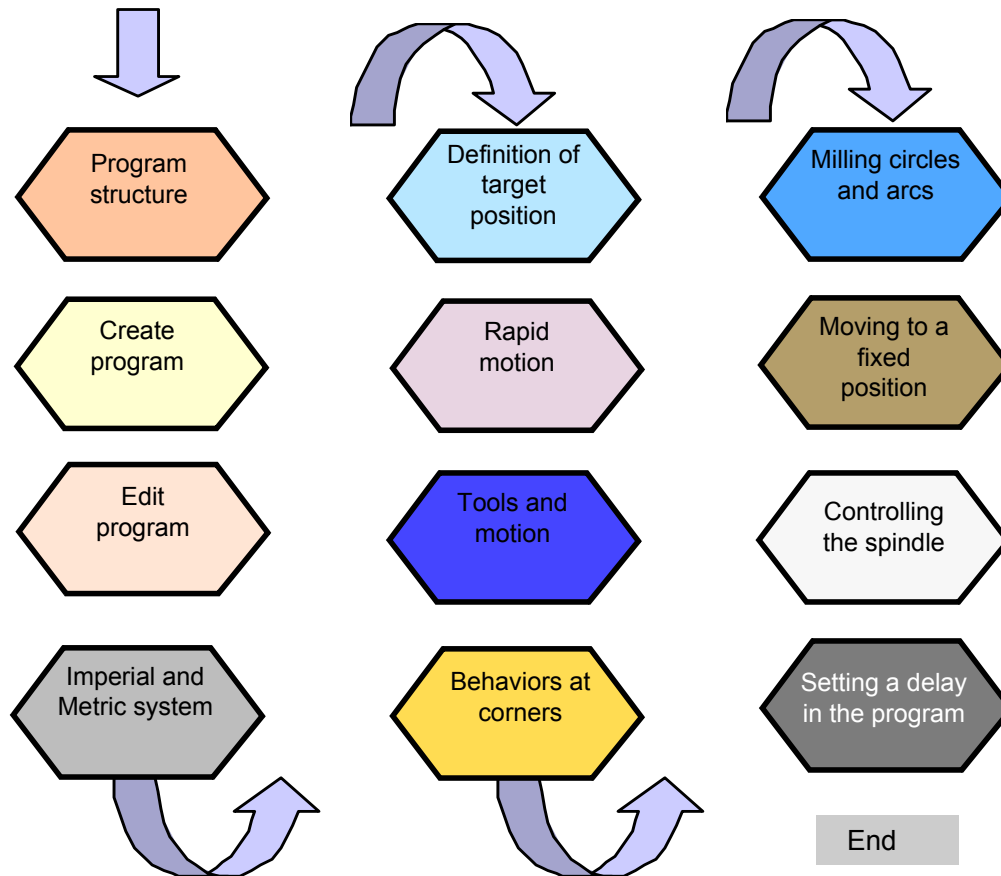
Basic Theory
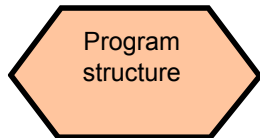
Program
structure

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

Header	**N5 G17 G90 G54 G71**
T, F, S function	**N10 T1 D1 M6**
	N15 S5000 M3 G94 F300
Geometry data / motion	**N20 G00 X100 Y100 Z5**
	N25 G01 Z-5
	N30 Z5
Return to change tool	**N35 G00 Z500 D0**
T, F, S function	**N40 T2 D1 M6**
	N45 S3000 M3 G94 F100
Geometry data / motion	**N50 G00 X50 Y50 Z5**
	N55 G01 Z-5
	N60 Z5
Return to change tool	**N65 G00 Z500 D0**
T, F, S function	**N70 T3 D1 M6**
	N75 S3000 M3 G94 F100
Geometry data / motion	**N80 G00 X50 Y50 Z5**
	N85 G01 Z-5
	N90 Z5
Return to change tool	**N95 G00 Z500 D0**
End/stop motion	**N100 G00 G40 G53 X0 Y0 Z500 D0**
	M30

SEQUENCE

Create program

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

Step 1

Programs can be created with the "program manager".

You can select the "program manager" using the key located on the PPU.

Step 2

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

Step 3

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

Name	Type	Length	Date	Time	
					Execute
					New

Model G0876 (Mfd. Since 12/22)
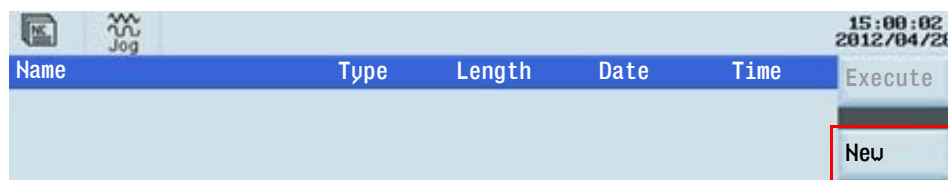
Step 4

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

Choose "New" to create a program.

Choose "New directory" to create a file.
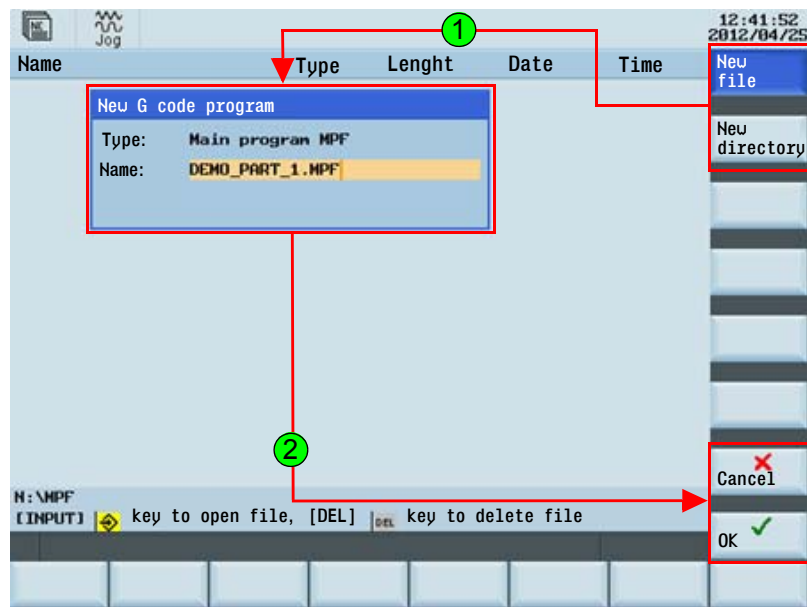
Step 5

Now the program is opened and can be edited.
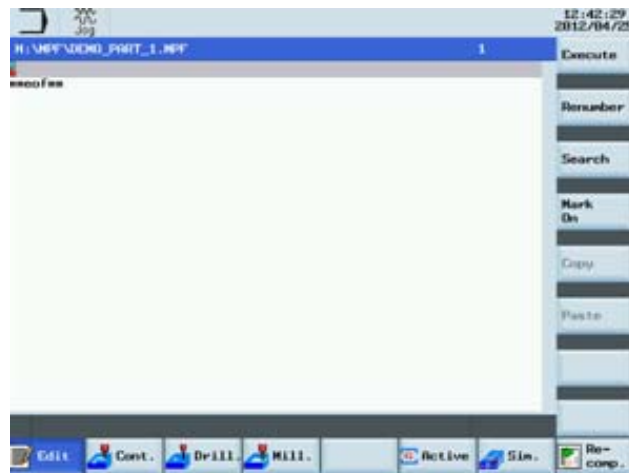
The system will save it automatically after editing.

End

Basic Theory
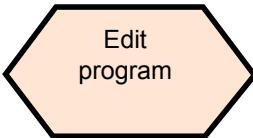
Edit
program

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

```
N:\MPF\DEMO_PART_1.MPF                                    1
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 ¶
;====================Facemilling start ¶
N80 CYCLE71( 50.00000, 2.00000, 2.00000, 0.00000, 0.00000, 0.00000,
70.00000, 100.00000, 0.00000, 2.00000, 40.00000, 2.00000, 0.20000, 5
00.00000, 41, 5.00000) ¶
N90 S4500 M3 ¶
N100 CYCLE71(50,2,2,0,0,0,70,100,0,2,40,2,0.2,300,22,5) ¶
;====================Facemilling end ¶
N110 SUPA G00 Z300 D0 ¶
N120 SUPA G00 X300 Y300 ¶
;====================Pathmilling start ¶
N130 T3 D1 ¶
N140 MSG("Please change to Tool No 3") ¶
N150 M05 M09 M00 ¶
N160 S5000 M3 G94 F300 ¶
```

12:50:53
2012/04/25
Auto

Execute

Renumber

Search

Mark
On

Copy

Paste

Edit | Cont. | Drill. | Mill. | Activate | Sin. | Re-comp.

Model G0876 (Mfd. Since 12/22)

Inches and mm

G71

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

Header

N5 G17 G90 G54 G71

T, F, S function

N10 T1 D1 M6

N15 S5000 M3 G94 F300

Geometry data / motion

N20 G00 X100 Y100 Z5

N25 G01 Z-5

N30 Z5

Return to change tool

N35 G00 Z500 D0

G70

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

Header

N5 G17 G90 G54 G70

T, F, S function

N10 T1 D1 M6

N15 S5000 M3 G94 F300

Geometry data / motion

N20 G00 X3.93 Y3.93 Z5

N25 G01 Z-0.787

N30 Z0.196

Return to change tool

N35 G00 Z19.68 D0

Definition of target position

G500

All absolute path data will be relative to this position. The position is written in the G500 (basic) zero offset.
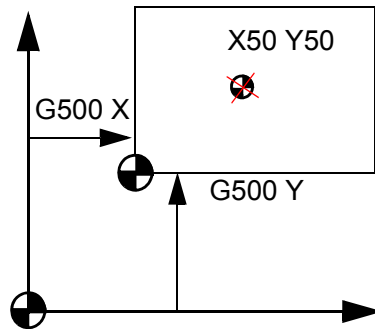
X50 Y50

G500 X

G500 Y

N5 G17 G90 G500 G71

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

Or

G54 G55 G56 G57 G58 G59

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

X50 Y50

G54 X

G54 Y
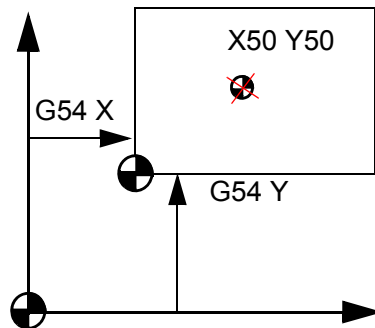
N5 G17 G90 G54 G71

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

Or

G500 + G54

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

G54 X0
G54 Y0

G500 X

G500 Y
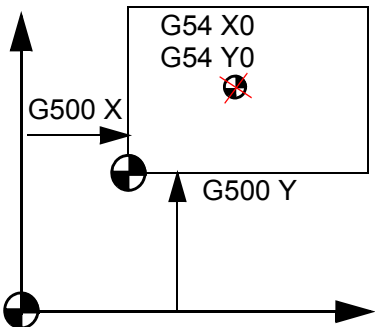
N5 G17 G90 G500 G71

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

G90

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

N5 G17 **G90** G54 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 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 Z**19.68** D0

Rapid motion
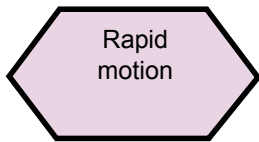
G00

When G00 is activated in the program, the axis will traverse at the maximum axis speed in a straight line.
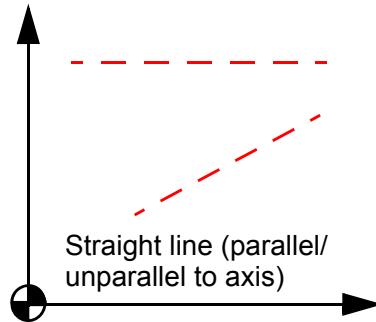
Straight line (parallel/unparallel to axis)

N5 G17 G90 G54 G71

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

Tools and motion
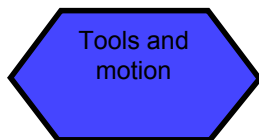
T1 D1 M06

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

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

N5 G17 G90 G54 G71

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

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

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

1. Feed per minute → G94

2. Feed per revolution of the spindle → G95

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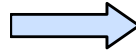
N10 T1 D1 M6
N15 S5000 M3 G95 F0.3
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z500 D0

Straight line (parallel/ unparallel to axis)
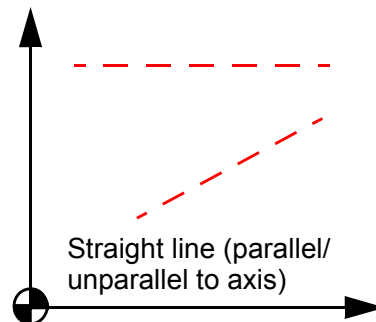
Basic Theory
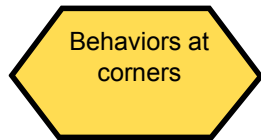
Activation/deactivation of the tool radius compensation when working on the part contour.

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.

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

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

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

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

Contour feedrate with
CFC

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

Feedrate calculated when using tool center, inside or outside of the contour
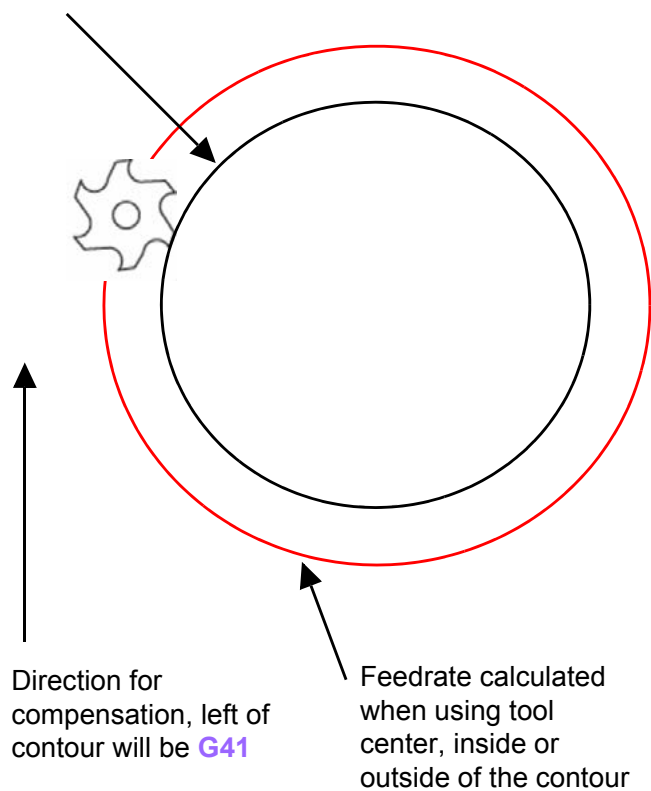
The result of the two commands will be such that the cutter goes very fast around a corner or slow on the contour.

Basic Theory

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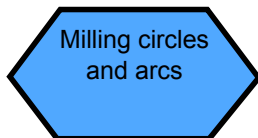
The circle radius shown in the example on the right can be produced with the specified part program code.

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

When working in the XY coordinate system, the interpolation parameters I and J are available.
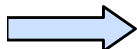
N5 G17 G90 G500 G71

N10 T1 D1 M6

N15 S5000 M3 G94 F300

N20 G00 X-20 Y-20 Z5

N25 G01 Z-5

N30 G41 X0 Y0

N35 Y50

N40 X100

N45 G02 X125 Y15 I-12 J-35

N50 G01 Y0

N55 X0

N60 G40 X-20 Y-20

N35 G00 Z500 D0

Note:

N45 can also be written as follows

N45 G02 X125 Y15 CR=37

Two common types of defining circles and arcs:

① :**G02/G03 X_Y_I_J_;**

② :**G02/G03 X_Y_CR=_;**

Arcs ≤180º,CR is a positive number

Arcs >180º,CR is negative number

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

Determine tool radius of T1 D1
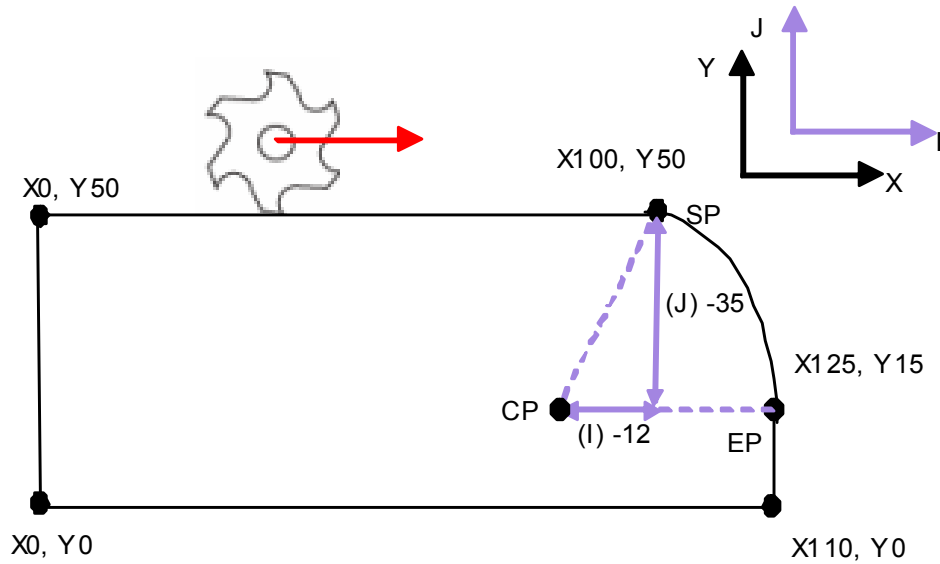
SP = start point of circle

CP = center point of circle

EP = end point of circle

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

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

G2 = define circle direction in traversing direction = G2 clockwise

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

Basic Theory

Moving to a
fixed position
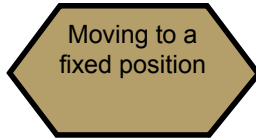
Using the code **G74**, the machine can move to the reference point automatically.
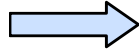
N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 **G74 Z=0** ;reference point

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

N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-5
N30 Z5
N35 **G74 Z=0** ;reference point
N40 **G75 X=0** ;fixed point

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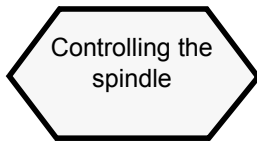
The following functions can be used to influence the operation of the spindle:

M3 accelerate to programmed speed clockwise

M4 accelerate to programmed speed counter-clockwise

M5 spindle decelerate to stop

M19 orient the spindle to a specific angular position.

```
N5 G17 G90 G500 G71

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

Setting a delay in the program
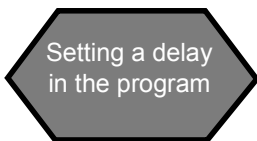
G04 can be used to pause the tools' movements during operation

G04 F5: Program pause of 5 s

This makes the surface of the workpiece much smoother

```
N5 G17 G90 G500 G71

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

7.6 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

```
    ↓                           ↷

Radius and               Hole
chamfers                 positioning

Hole                     Contour milling
centering                with cycle

Drilling                 Milling slots
holes                    and spigots

Tapping                  End
    ↻
```

Model G0876 (Mfd. Since 12/22)
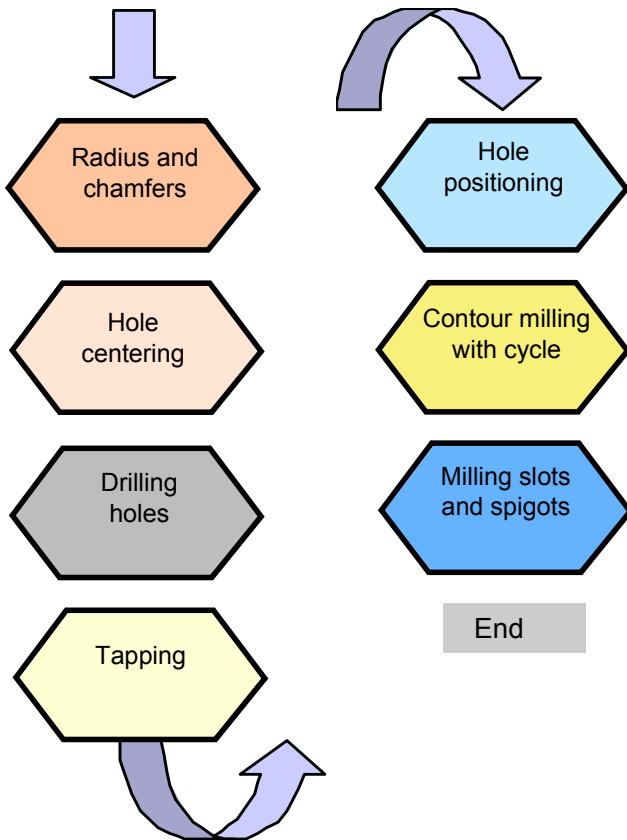
Basic Theory

Radius and chamfers

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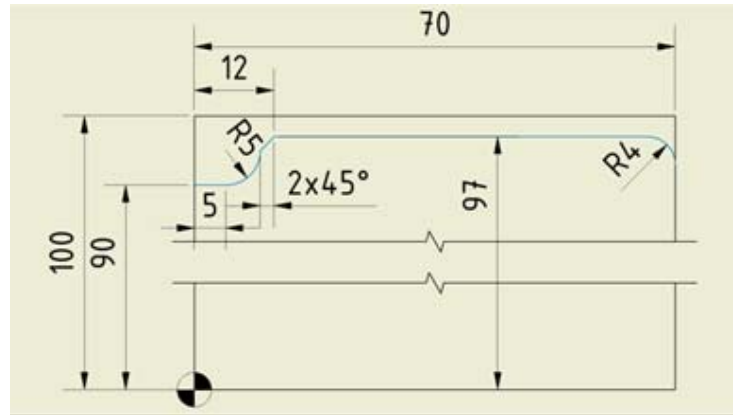
RND = Radii

CHR = Chamfer

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

CHF=Chamfer

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

actual effect

N55 SUPA G00 Z300 D0

N60 SUPA G00 X300 Y300

N65 T3 D1

N70 MSG("Please change to Tool No 3")

N75 M05 M09 M00

N80 S5000 M3 G94 F300

N85 G00 X-6 Y92

N90 G00 Z2

N95 G01 F300 Z-10

N100 G41 Y 90

N102 G01 X 5

N105 G01 X12 RND=5

N110 G01 Y97 CHR=2

N115 G01 X70 RND=4

N120 G01 Y90

N125 G01 G40 X80

N130 G00 Z50
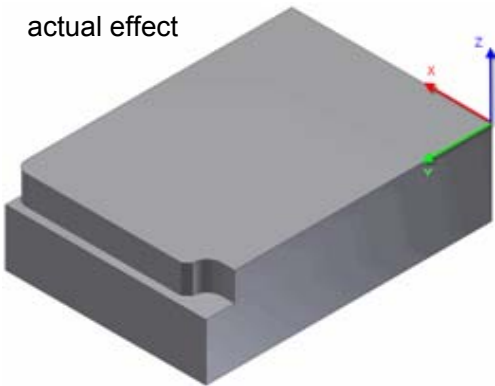
Basic Theory
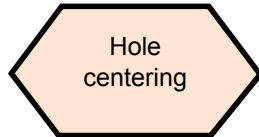
Hole
centering

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

CYCLE81: Without delay at current hole depth

CYCLE82: With delay at current hole depth

Drill.

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

Drilling centering → Center drilling
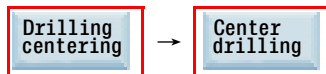
Select "Drilling centering" using the vertical SKs , or select "Center drilling" ,and parameterize the cycle according to requirements.
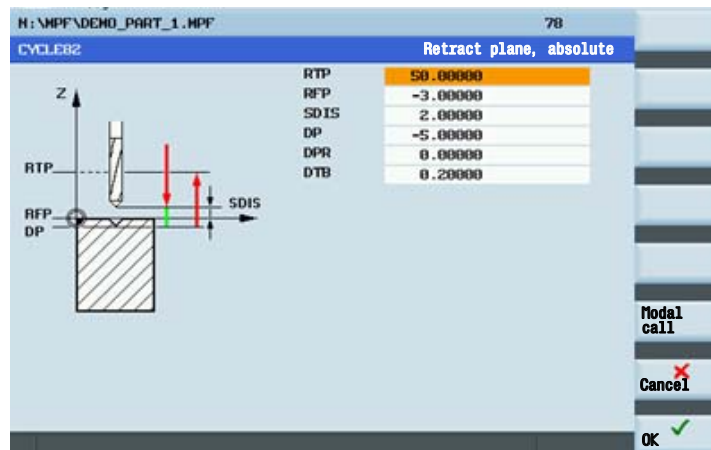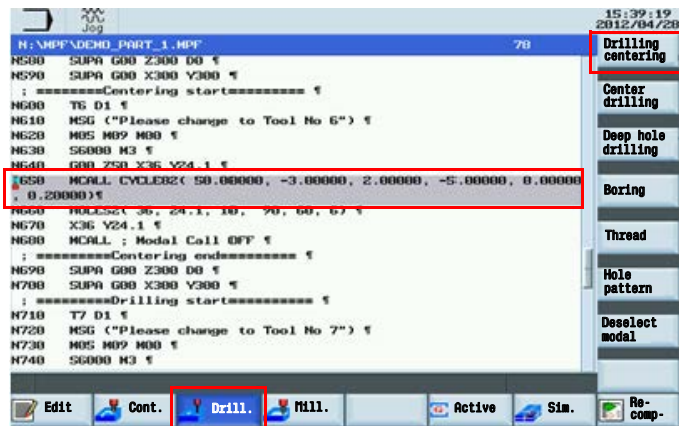
Model G0876 (Mfd. Since 12/22)

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.

| Modal call |
| Cancel |
| OK ✓ |

actual effect

Parameters	Meanings
RTP=50	Coordinate value of turning position is 50 (absolute)
RFP=-3	Coordinate value of hole edge starting position under workpiece zero point surface is 3 (absolute)
SDID=2 (frequently used values 2~5)	Safety distance, feed path changes from quick feed to machine feed 2 mm away from RFP face
DP=-5	Coordinate position of final drilling depth is -5 (absolute)
DTB=0.2	Delay of 0.2 s at final drilling depth
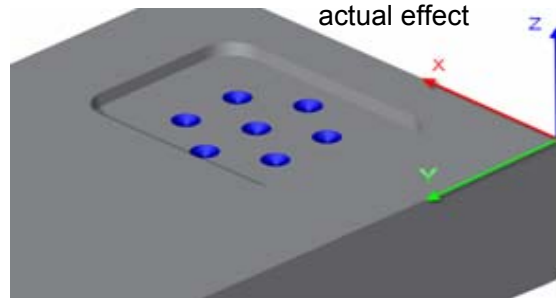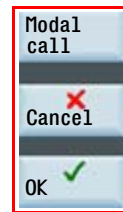
RTP	50.00000
RFP	-3.00000
SDIS	2.00000
DP	-5.00000
DPR	0.00000
DTB	0.20000

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

Basic Theory

Drilling holes

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

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

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

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

Select "Deep hole drilling" using the vertical SKs and parameterize the cycle according to requirements.
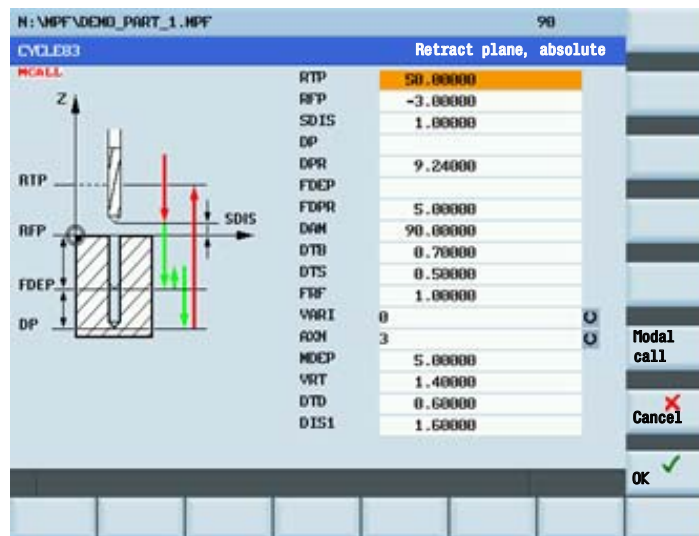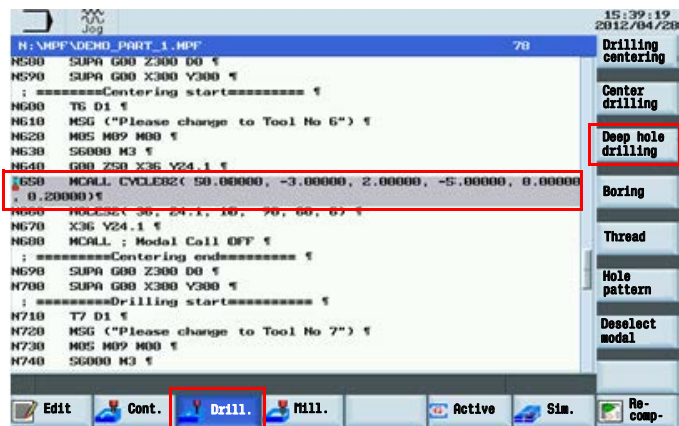
Model G0876 (Mfd. Since 12/22)

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

This will drill a hole at the current position.

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

The information is transferred as shown below.

actual effect

For specific parameter commands, see the next page
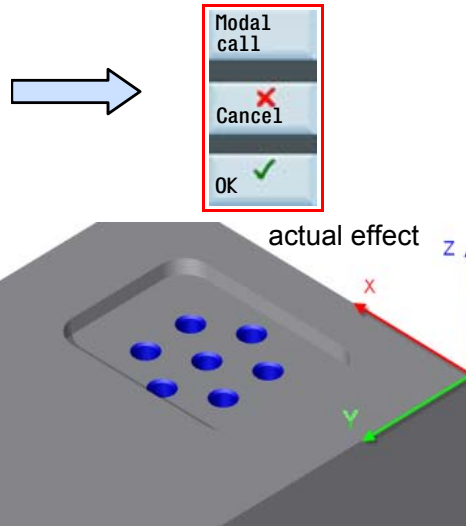
RTP	50.00000
RFP	-3.00000
SDIS	1.00000
DP	
DPR	9.24000
FDEP	
FDPR	5.00000
DAM	90.00000
DTB	0.70000
DTS	0.50000
FRF	1.00000
VARI	0
AXN	3
MDEP	5.00000
VRT	1.40000
DTD	0.60000
DIS1	1.60000

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

N330 X20 Y20 ; Hole will be drilled

N335 X40 Y40 ; Hole will be drilled

N340 MCALL

N345 X60 Y60 ; Hole will not be drilled

Basic Theory

	For descriptions of RTP, RFP, SDIS and DP, please see ☞ Page 119	
FDEP=5	Reach first drilling hole depth. Z axis coordinate is -5 (absolute coordinate value)	
FDPR=5	From the reference plane, drill downwards 5mm	
DAM=90	Decrement is 90	
DTB=0.7	Pause 0.7 s during final tapping of thread depth (discontinuous cutting)	DTB <0: Unit is r
DTS=0.5	Stops at the start position for 0.5 s (for VARI=1,removal active)	DTS <0: Unit is r
FRF=1 (range:0.001~1)	Original effective feed rate remains unchanged	Feed rate modulus
VARI=0	Interruption in drilling is active	VARI=1 retraction of active quill back to reference plane
AXN=3	AXN is tool axis,under appointed G17 use Z axis	The value of AXN decides which axis to use
MDEP=5	Minimal drilling depth 5 mm	This parameter activates only when DAM <0
VRT=1.4	Interruption in drilling, the retraction value of the quill is 1.4 mm	VRT=0 → retraction value is 1mm VRT>0 → retraction value is appointed value
DTD=0.6	Pauses at the position of final drilling depth for 0.6 s	DTD <0:unit is r, DTD =0:same as DTB
DIS1=1.6	When reinserting a quill, you can program a distance limit of 1.6 mm	For specific explanations please refer to the standard handbook

DAM parameter

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

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

In case the residual depth <2xFDPR,the last 2 cutting depth are half of the residual depth.

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

Basic Theory

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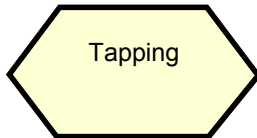
The easiest way to tap a hole is to use
CYCLE84: Solid tap holder
CYCLE840: With floating tap holder.

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

Drill.

The relevant cycle can now be found using the vertical SKs on the right.
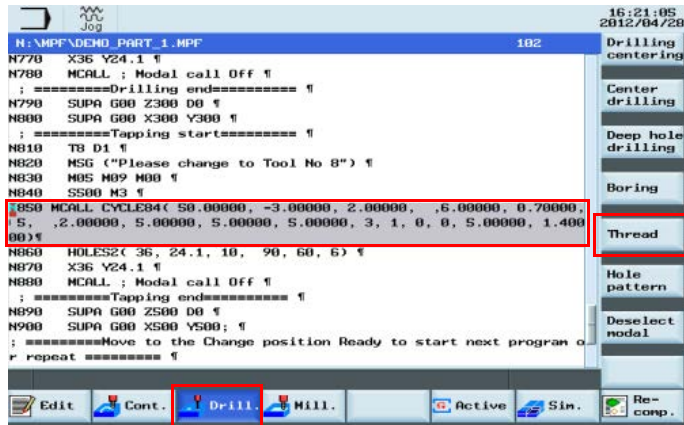
Thread → Rigid tapping

Select "Thread" using the vertical SKs ,and then select "Rigid tapping", and parameterize the cycle according to requirement.
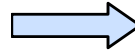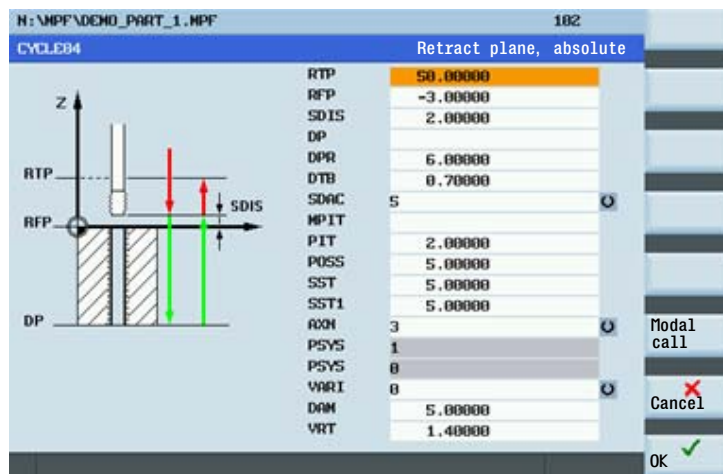
Model G0876 (Mfd. Since 12/22)

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

This will drill a hole at the current position.

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

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

Examples are shown on the next page .
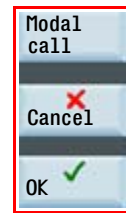
Basic Theory

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

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

N330 X20 Y20 ; Hole will be tapped

N335 X40 Y40 ; Hole will be tapped

N340 MCALL

N345 X60 Y60 ; Hole will not be tapped

actual effect
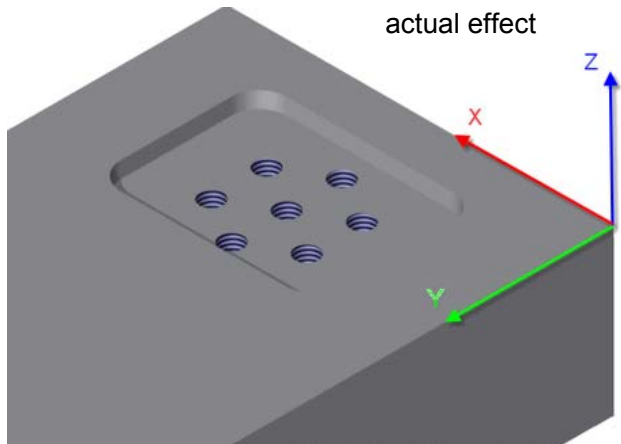
	For descriptions of RTP, RFP, SDIS and DP, please see ☞ Page 119
	For descriptions of AXH, VARI, DAM and VRT, please see ☞ Page 122

Parameters	Meanings	Remarks
DTB=0.7	Pause 0.7 s during final tapping to thread depth (discontinuous cutting)	
SDAC=5	Spindle state after cycle is M5	Enter values 3/4→M3/M4
PIT=2(Range of values:0.001~2 000 mm)	Right hand thread with 2mm pitch	Evaluate value→left hand thread
POSS=5	Spindle stops at 5º (unit: º)	
SST=5	Tapping thread spindle speed is 5 r/min	
SST1=5	Retraction spindle speed is 5 r/ min	Direction is opposite to SST SST1=0 →speed is same as SST
	SST and SST1 control the spindle speed and the Z axis feed position synchronously. During execution of CYCLE 84, the switches of the feed rate override and the cycle stop (feed hold) are deactivated.	

Basic Theory

Hole positioning

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

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

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

Hole pattern → Hole circle

Select "Hole pattern" using the vertical SKs ,and then select "Hole circle", and parameterize the cycle according to requirement.
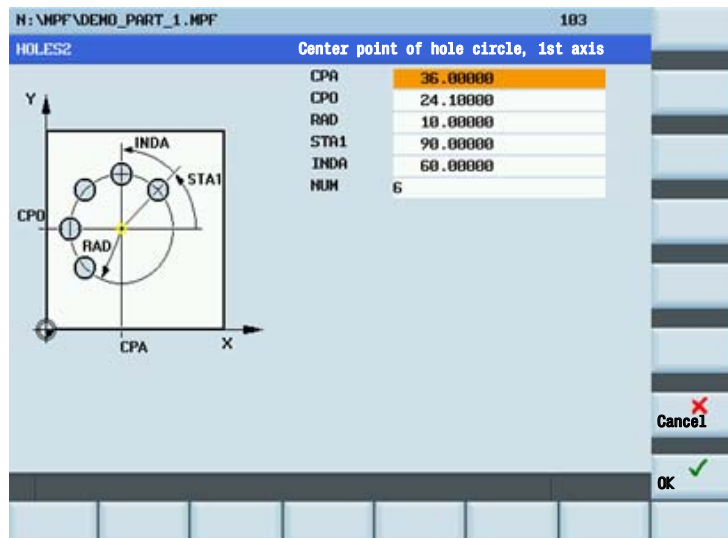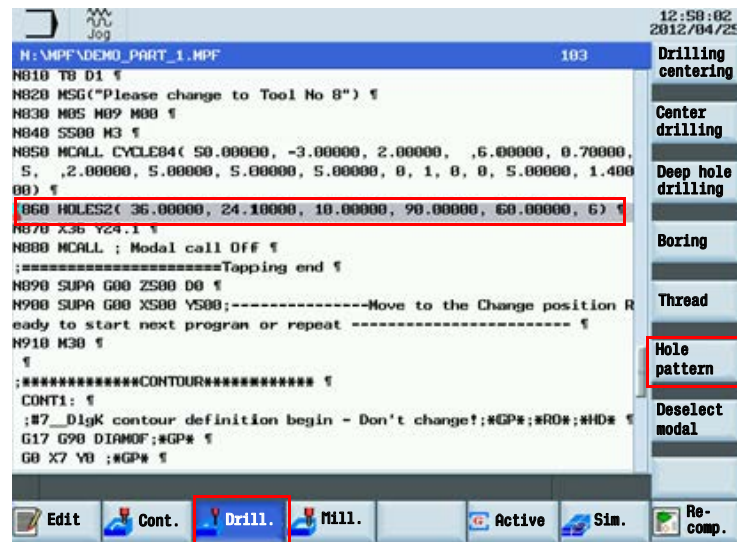
Model G0876 (Mfd. Since 12/22)

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.

CPA	36.00000
CPO	24.10000
RAD	10.00000
STA1	90.00000
INDA	60.00000
NUM	6
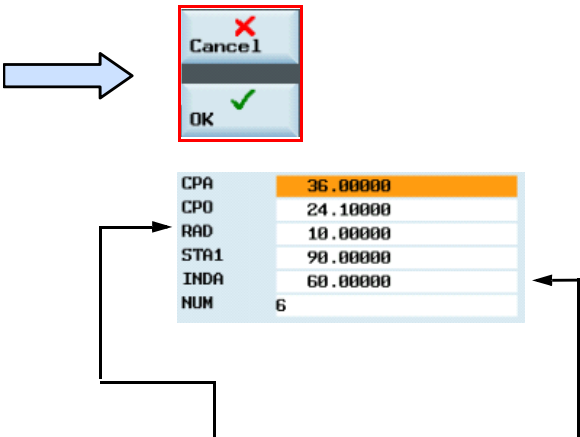
N325 MCALL CYCLE82(50.00000, -3.00000, 2.00000, -5.00000, 0.00000, 0.20000)

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

N335 X36 Y24.1

N340 MCALL ; Modal Call OFF

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

Basic Theory

Contour milling with cycle

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

The cycle can be found and parameterized via the "Mill." SK.
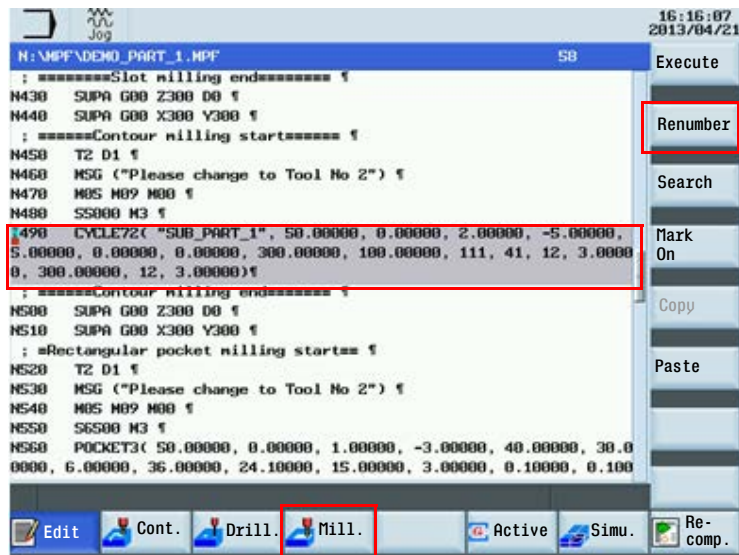
The "Contour milling" SK can be found in the vertical SKs on the right.

The parameterization is performed as in this figure.

Model G0876 (Mfd. Since 12/22)
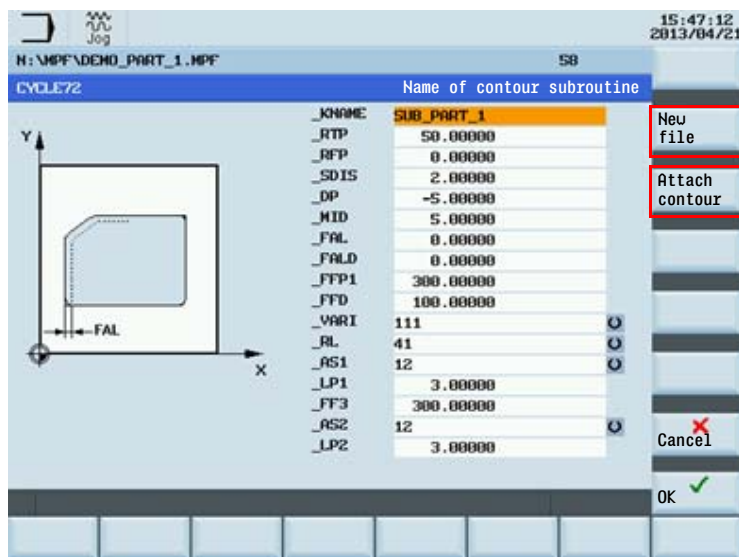
New file

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

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

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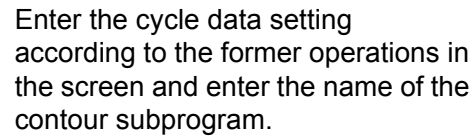
Press "New file " on the PPU to create contour information in the .SPF file. The cursor moves to the contour editing position automatically.

New file

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

Basic Theory

After opening the contour data setting window, please make the following settings:
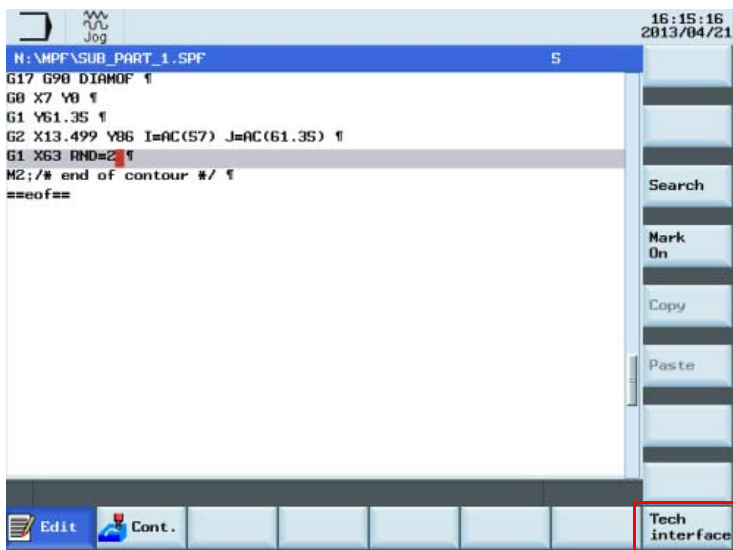
Enter appropriate coordinates based on the data from the technical drawing.

After completing the steps, the system will return to the edit interface. Press "Technical interface" on the PPU to return to the interface for setting the cycle data.
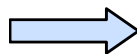
After finishing the parameter settings of CYCLE72, press the "OK" SK on the PPU to insert the corresponding cycles in the main program.

Model G0876 (Mfd. Since 12/22)
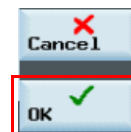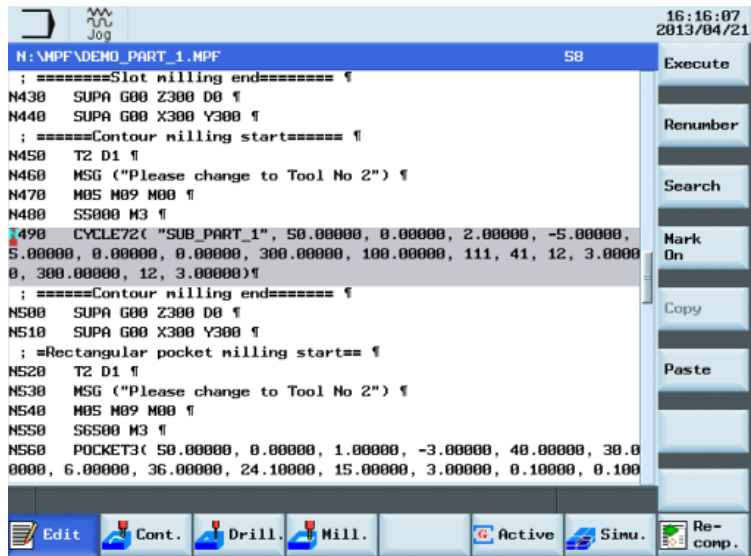
After all the settings take effect, the selected cycle and set data will be transferred to corresponding part program automatically

(for further information, see next page).

Basic Theory

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

actual effect

_KNAME	SUB_PART_1	
_RTP	50.00000	
_RFP	0.00000	
_SDIS	2.00000	
_DP	-5.00000	
_MID	5.00000	
_FAL	0.00000	
_FALD	0.00000	
_FFP1	300.00000	
_FFD	100.00000	
_VARI	111	⟳
_RL	41	⟳
_AS1	12	⟳
_LP1	3.00000	
_FF3	300.00000	
_AS2	12	⟳
_LP2	3.00000	

N245 CYCLE72("SUB_PART_1", 50.00000, 0.00000, 2.00000, -5.00000, 5.00000, 0.00000, 0.00000, 300.00000, 100.00000, 111, 41, 12, 3.00000, 300.00000, 12, 3.00000)
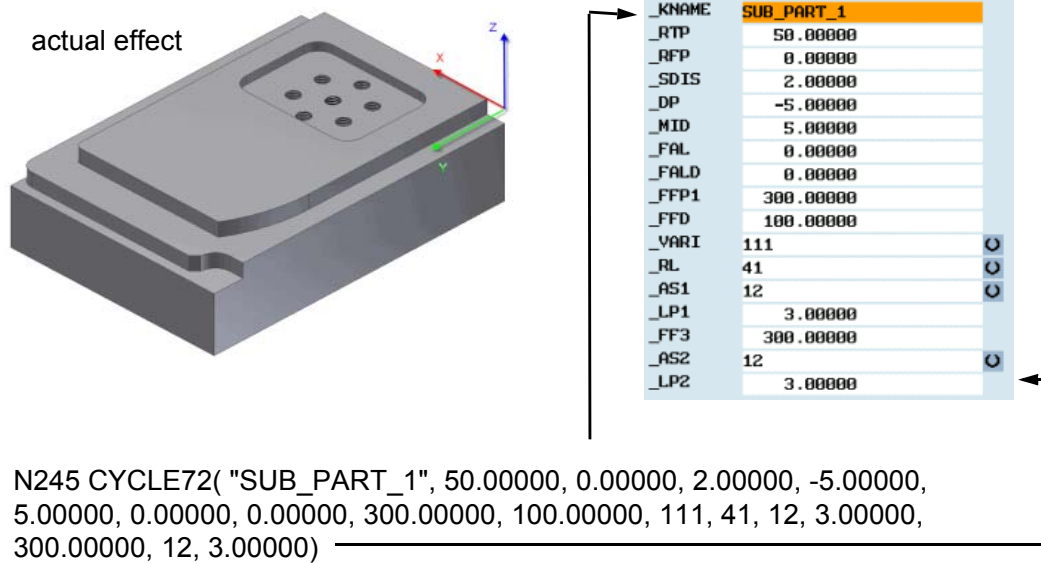
For descriptions of RTP, RFP, SDIS and DP, please see ☞ Page 119		
Parameters	Meanings	Remarks
KNAME= CONT1:CONT1_E	Set the name of the contour subprogram as "CONT1" (":CONT1_E" is automatically created)	The first two positions of the program name must be letters
MID=5	The maximal feed depth is 5 mm	
FAL=0	Finishing allowance at the contour side is 0 mm	
FALD=0	Finishing allowance at the bottom plane is 0 mm	
FFP1=300	Tool feed rate on plane is 300 mm/min	
FFD=100	Feed rate after inserting the tool in the material is 100 mm/min	
VARI=111	Use G1 to perform rough machining, and back to the depth defined by the RTP+SDIS at the completion of the contour	For other parameters, please refer to the standard manual
RL=41(absolute value)	PL=41→use G41 to make tool compensation on the left side of the contour	PL=40→G40, PL=42→G42
AS1=12	Approach the contour along the 1/4 circle on the path in space	For other parameters, please refer to the standard manual
LP1=3	The radius of the approaching circle is 20 mm	The length of the approaching path is along the line to approach
FF3=300	The feed rate during retraction of the path is 300 mm/min	
AS2=12	Return along the 1/4 circle on the path in space	Parameter explanations are the same as for AS1
LP2=3	The radius of the return circle is 20 mm	The length of the returning path is along the line to approach

Basic Theory

Milling slots and spigots

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

The cycle can be found and parameterized via the "Mill." SK.
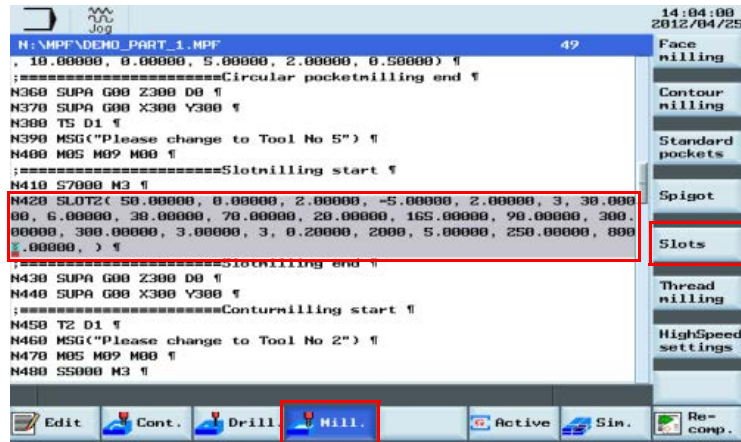
The relevant cycle can be found using the vertical SKs on the right.

Slots → Circumf. slots

Select "slot" using the vertical SKs and parameterize the cycle according to requirement.
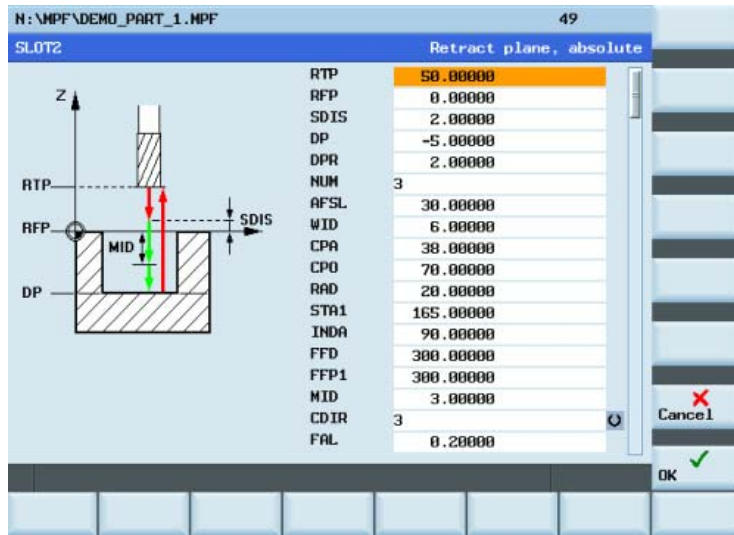
Model G0876 (Mfd. Since 12/22)

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

This will perform milling at the position defined in the cycle.
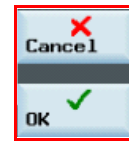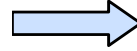
actual effect
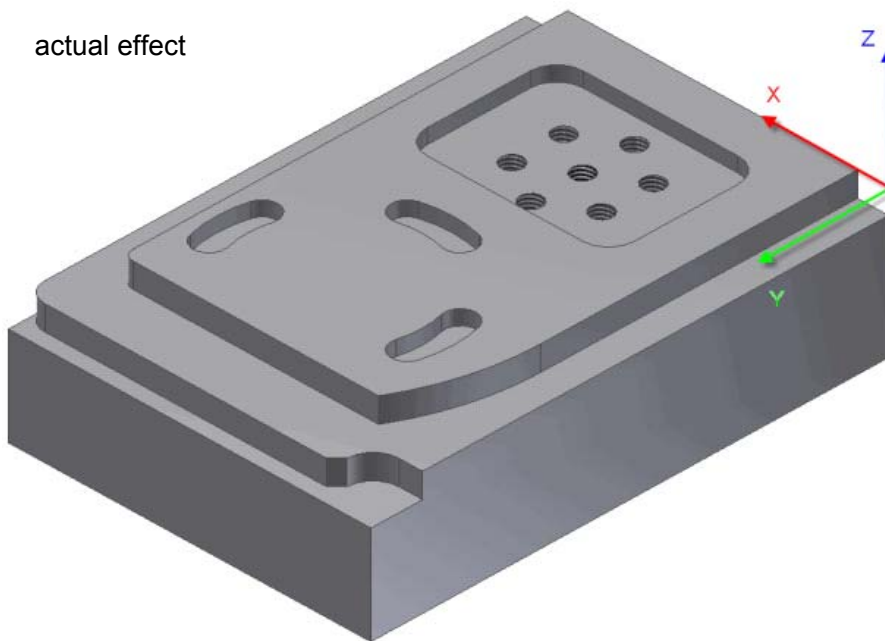
Basic Theory

RTP	50.00000
RFP	0.00000
SDIS	2.00000
DP	
DPR	3.00000
NUM	3
AFSL	30.00000
WID	6.00000
CPA	38.00000
CPO	70.00000
RAD	20.00000
STA1	165.00000
INDA	90.00000
FFD	300.00000
FFP1	300.00000
MID	3.00000
CDIR	3
FAL	0.20000
VARI	0
MIDF	5.00000
FFP2	250.00000
SSF	8000.00000
FFCP	

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

For descriptions of RTP, RFP, SDIS, DP and DPR, please see ☞ Page 119

For descriptions of CPA, CPO and RAD, please see ☞ Page 129

For descriptions of FFD and FFP1, please see ☞ Page 135

Model G0876 (Mfd. Since 12/22)

Parameters	Meanings		Remarks
NUM=3	Three slots on the circle		
AFSL=30	Angle slot length is 30º		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 165º		
INDA=90	Incremental angle, angle between the slots is 90º		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.		

25

20

M5x0.8

97   90,00

3

11

30

12,24

Status	Änderungen	Datum	Name

Model G0876 (Mfd. Since 12/22)

DEMO PART MILLING

	Datum	Name			1
Gezeichnet					A4
Kontrolliert					
Norm					

7.7   Simulate Program

Description

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

Content

```
      ↓

  ⬡ Simulate
    program (Axis
    do not move) ⬡

    [ End ]
```

Basic Theory

Simulate program (Axis do not move)
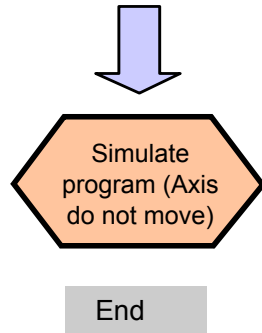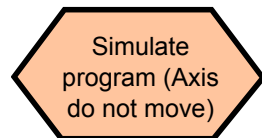
A part program must have been created before it can be tested using "Simulation".

Step 1

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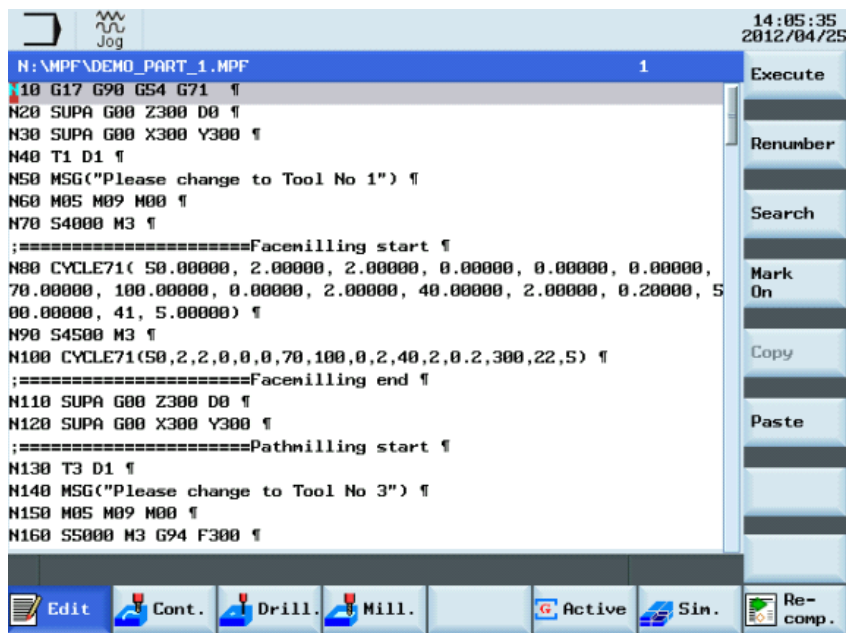
SEQUENCE

Step 2

Press the "Simu." SK on the PPU.
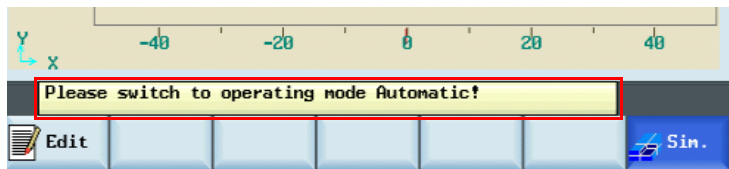
If the control is not in the correct mode, a message will be displayed at the bottom of the screen.

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

Model G0876 (Mfd. Since 12/22)
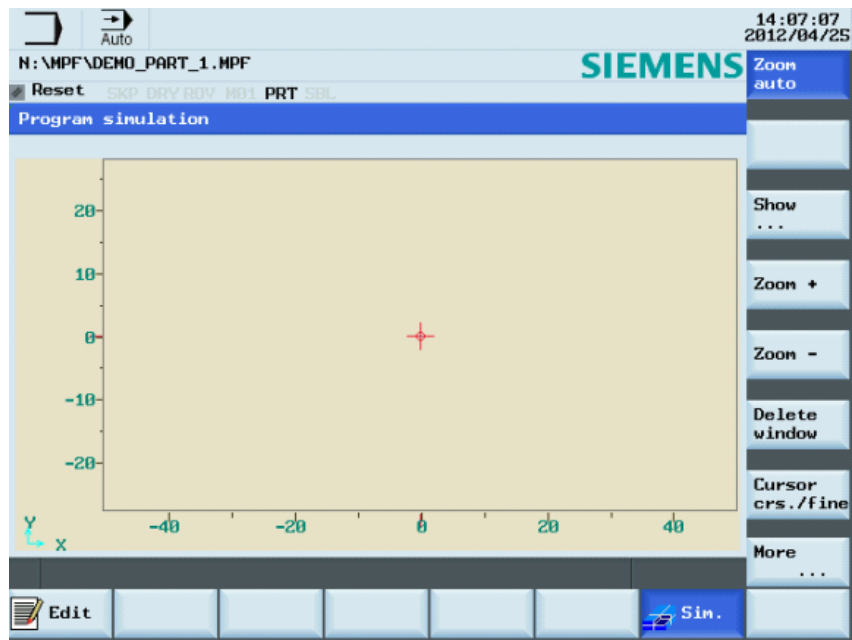
Press the "CYCLE START" key on the MCP.

If the control is not in the correct mode, a message will be displayed at the bottom of the screen.
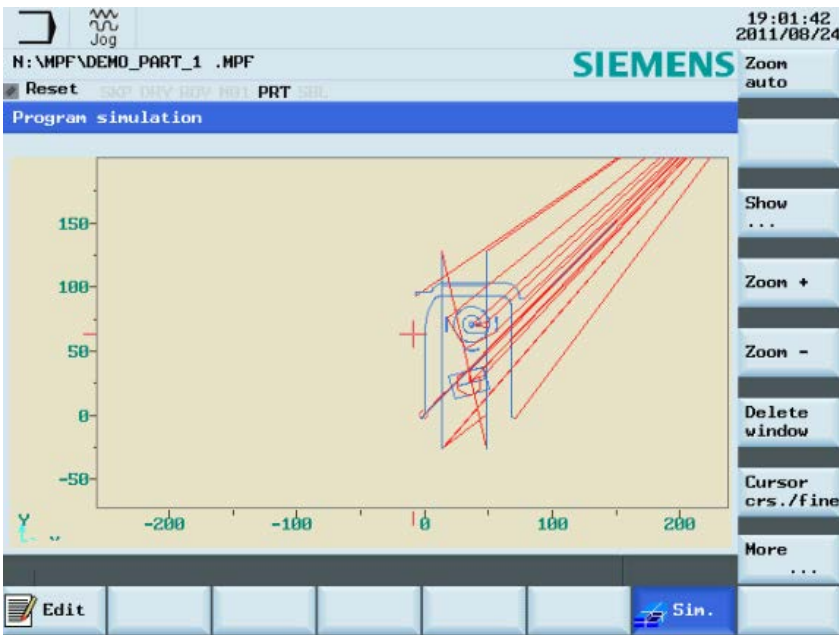
Press the "Edit" SK on the PPU to return to the program.
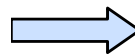
End

7.8　Test Program

Description

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

Content

```
          ⬇

    ⬡ Program
      Execution ⬡

    ⬡ Dry Run ⬡

    ▭ End ▭
```

Model G0876 (Mfd. Since 12/22)
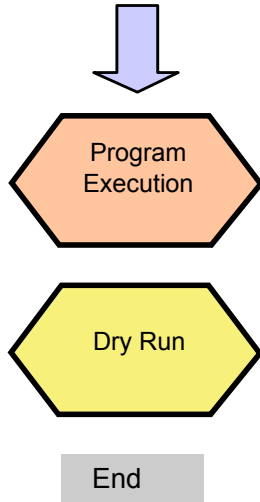
Basic Theory

Program
Execution

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

		14:05:35 2012/04/25
N:\MPF\DEMO_PART_1.MPF	1	Execute
X10 G17 G90 G54 G71 ¶		
N20 SUPA G00 Z300 D0 ¶		
N30 SUPA G00 X300 Y300 ¶		Renumber
N40 T1 D1 ¶		

Press the "Execute" SK on the PPU.

Execute

		14:11:53 2012/04/25
M Auto		
N:\MPF\DEMO_PART_1.MPF	SIEMENS	G function
Reset	SKP DRY ROV M01 PRT SBL	
MCS	Position	Dist-to-go	T,F,S
X	0.000	0.000 mm	T 1	D 1	Auxiliary function
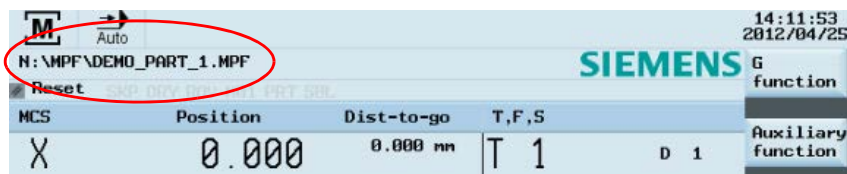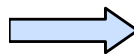
The control is now in AUTO mode with the current opened program storage path being displayed and the AUTO lamp on the MCP is on.

HAND WHEEL | JOG | REF. POINT
AUTO | SINGLE BLOCK | MDA
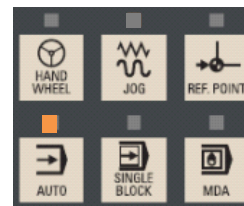
Now the program is ready to start and the actual operation will be described in the next section!

SEQUENCE

Dry Run

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

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

Step 1

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

Press the "Offset" key on the PPU.

| Tool list | Tool wear | | | Work offset | R R var. | SD Set data | GUD User data |

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

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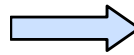
Use the traversing key to move to the required position. The position is now highlighted.

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

DRY

Dry run feedrate: 2000   mm/min   Misc.

Start angle

Start angle for thread: 0.000   °

| Tool list | Tool wear | | | Work offset | R R var. | SD Set data | GUD User data |
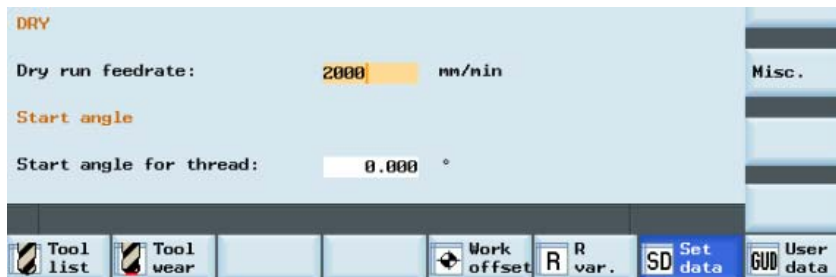
Press the "Input" key of the PPU.

INPUT

Model G0876 (Mfd. Since 12/22)

Press the "Machine" key on the PPU. ➡ M MACHINE
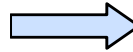
Press the "Prog. cont." SK on the PPU. ➡ NC Prog. cont.

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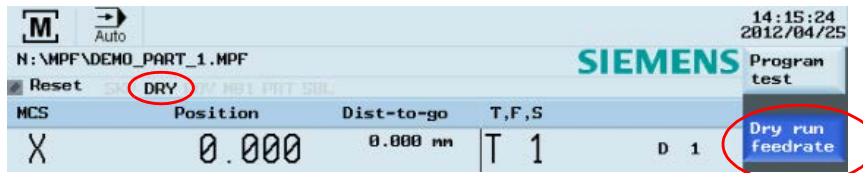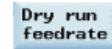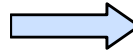
Note: The "DRY" symbol is shown and the "Dry run feedrate" SK is highlighted in blue.

Press the "Back" SK on the PPU. ➡ Back «
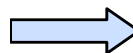
Step 2 Make sure the feedrate override on the MCP is 0%.

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.) ➡ DOOR
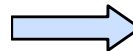
Press "CYCLE START" on the MCP to execute the program. ➡ CYCLE START
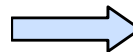
Turn the feedrate override gradually to the required value.

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

7.9   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

Time Counter

Piece-Machining

Tool Wear

End
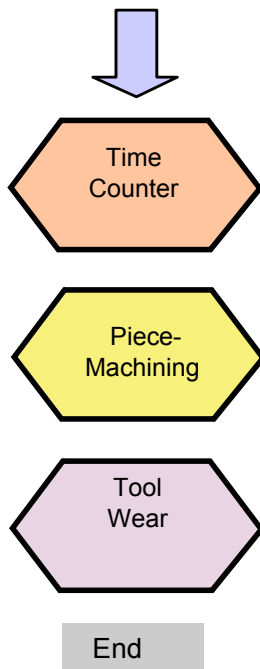
Basic Theory
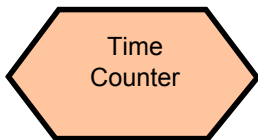
Time
Counter

Make sure the machine has been referenced before machining workpieces!

Step 1

Press the "Machine" key on the PPU.
➡️

| M |
| MACHINE |

Press the "Auto" key on the MCP.
➡️

| ⇥ |
| AUTO |
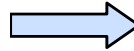
Press the "Time counter" SK on the PPU.
➡️

| Time counter |

```
Block display          DEMO_PART_1.MPF      Time, counter

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

SEQUENCE

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

`Cycle time    0000:00:06h`

"Time left" shows how much time remains before the program ends.
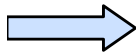
`Time left    0000:00:00h`

Step 2

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

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

SELECT
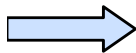
Enter the number of workpieces you require to be machined in "Required".
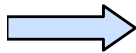
`Required        45`

"Actual" shows the number of workpieces that have been machined.
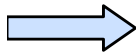
`Actual          8`

```
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 ti0000:00:00h
 N30 SUPA G00 X300 Y300¶
 N40 T1 D1¶                             Counter    Yes        ○
 N50 MSG("Please change to Tool No 1")¶ Required       45
 N60 M05 M09 M00¶                       Actual          8
 N70 S4000 M3¶

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

Model G0876 (Mfd. Since 12/22)

Piece-Machining

Make sure the program is correct before machining pieces!
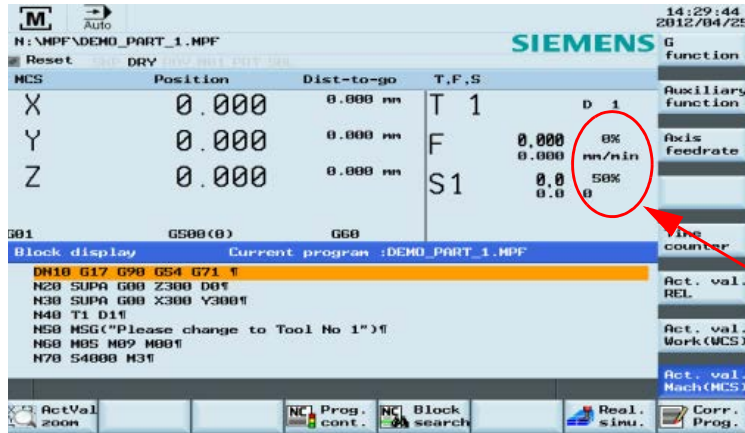
Set the program in the ready-to-start status as shown on the left in accordance with the "Program execution" sequences.

Perform the relevant safety precautions!

Make sure that only "AUTO" mode and "ROV" mode are activated

(or select the M01 function if required).
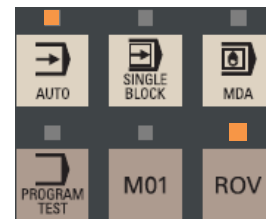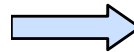
Notes:M01 function → program will stop at the position where there is M01 code.

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

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

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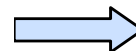
SEQUENCE

Tool
Wear

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.
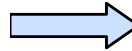
Tool
wear

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

Or
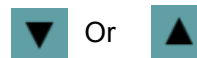
		14:32:25 2012/04/25
Tool wear	Active tool no	1 D 1

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

« Back

| Tool list | Tool wear | | | Work offset | R var. | Set data | User data |

SD Sett. data

Model G0876 (Mfd. Since 12/22)

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

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

Positive value: The tool moves away from the workpiece

Negative value: The tool moves closer to the workpiece

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

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

Positive value: tool is away from workpiece (set radius bigger than real one)

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

Press "Input" on the PPU to activate the compensation.
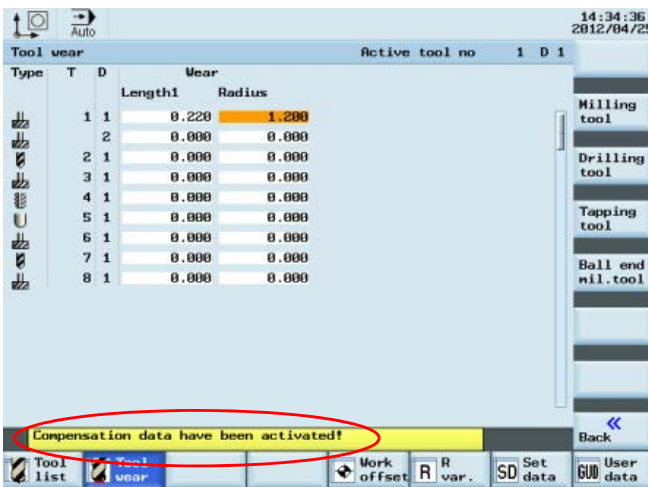
7.10   Program Restart

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

Model G0876 (Mfd. Since 12/22)
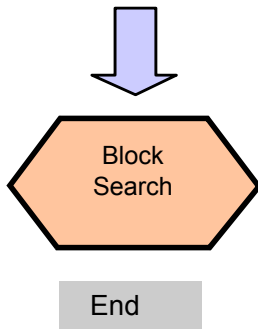
Basic Theory
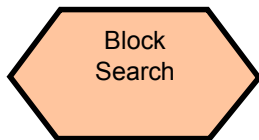
Block
Search

Press the "Machine" key on the PPU.
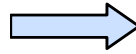
Press the "Auto" key on the MCP.

Press the "Block search" SK on the PPU.

Press the "Interr. point" SK on the PPU and the cursor will move to the last interrupted program line.
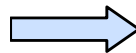
Note: The cursor can be moved to the required program block with the traversing keys.
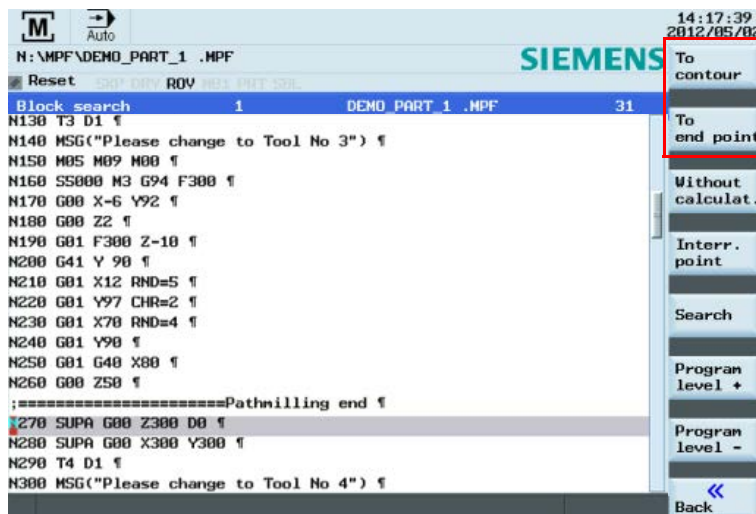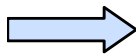
Note: The "To contour" and "To end point" functions.

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

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

Press the "To end point" SK on the PPU.

(can also press "To contour" if required)

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.

Model G0876 (Mfd. Since 12/22)

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

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

7.11 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

```
        ↓                         ↪

   RS232 + USB              R parameters

 Shared Network              Time change
 Drive (only
 ADVANCED)

     Help                    Save data

 Manual face                Gear change
 cutting
        ↪                      End
```

SEQUENCE

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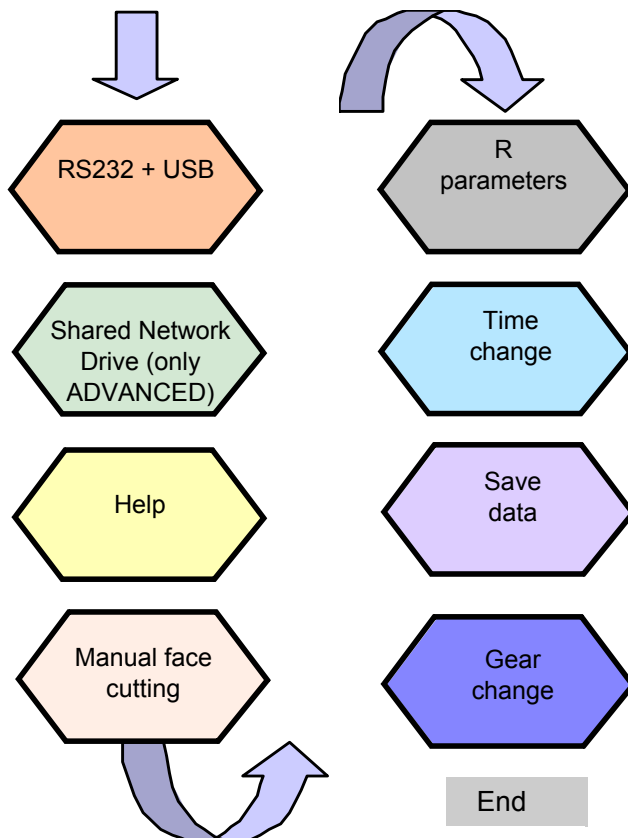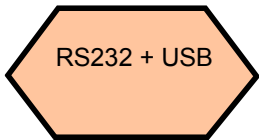
RS232 is used to transfer the programs to and from the NC.

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

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

Press "Program Manager" on the PPU. ⟹ PROGRAM MANAGER
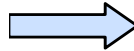
Press the "RS232" SK on the PPU. ⟹ RS232

Press the "Settings" SK on the PPU. ⟹ Settings

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

Communications settings	
Device	RTS CTS
Baud rate	19200
Stop bits	1
Parity	None
Data bits	8
End of transmis.	1a
Confirm overwrite	NO
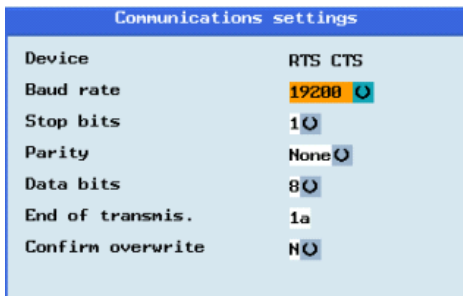
Press the "Save" SK on the PPU. ⟹ Save

Press the "Back" SK on the PPU. ⟹ « Back

SEQUENCE

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

Press the "NC" SK on the PPU. ⟹ `NC NC`

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

Press the "Copy" SK on the PPU. ⟹ `Copy`
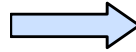
Press the "RS232" SK on the PPU. ⟹ `RS232`

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

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

Press the "Send" SK on the PPU. ⟹ `Send`

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

```
Sending of data                19200,8,1,NONE
File:     _N_DEMO_PART_1 _MPF
From      N:
To        RS232
Bytes:    4095
```
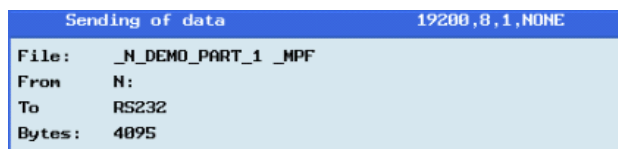
If there is a problem during transfer of the part program, a window will be displayed.

```
RS232 communications are interrupted!
Please check the cable and the receive
program.
Continue transmission?
(!)
```
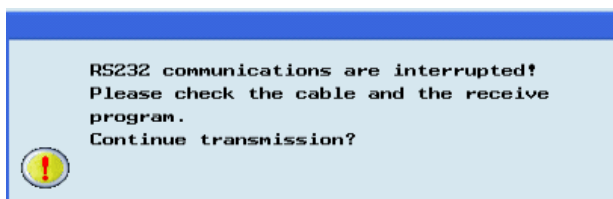
You can continue sending the part program.

Press the "OK" SK on the PPU. ⟹ **OK** ✓

Or you can abort the sending of the part program.

Press the "Cancel" SK on the PPU. ⟹ **Cancel** ✗
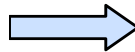
Step 3   Transfer a part program to the PPU from a PC.

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

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

Press the "Accept" SK on the PPU. ⟹ Accept ✓

```
Receiving of data          19200,8,1,NONE
File:
From    RS232
To
Bytes:
```
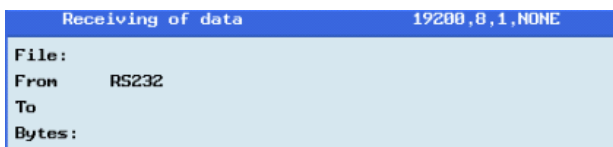
Check the interface setting and start the communication software to send the program from PC.

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

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

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

SEQUENCE

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

Step 4   Use the "Copy" and "Paste" SKs to transfer the part program from NC to USB.

Connect a USB device with sufficient memory to the USB interface on the PPU.
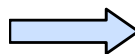
Press the "NC" SK on the PPU. ➡ `NC NC`

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

Press the "Copy" SK on the PPU. ➡ `Copy`

Press the "USB" SK on the PPU. ➡ `USB`
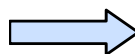
Press the "Paste" SK on the PPU. ➡ `Paste`
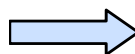
Step 5   Use the "Copy" and "Paste" SKs to transfer the part program from NC to USB.

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

Press the "USB" SK on the PPU. ➡ `USB`

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

Press the "Copy" SK on the PPU. ➡ `Copy`

Press the "NC" SK on the PPU. ➡ `NC NC`

Press the "Paste" SK on the PPU. ➡ `Paste`

Model G0876 (Mfd. Since 12/22)
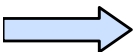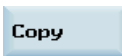
SEQUENCE

Shared Network
Drive (only
ADVANCED)
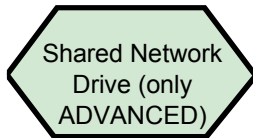
A shared network drive can be made using an ethernet connection between the PC and the PPU so the transferring and backup of NC programs can be performed easier.

Step 1   Set PPU IP address.

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

Press key: SHIFT + SYSTEM ALARM   Press key: >

Press "Serv. Displ." SK   ⟹   Serv. displ.

Press "Serv. Displ." SK   ⟹   Service control
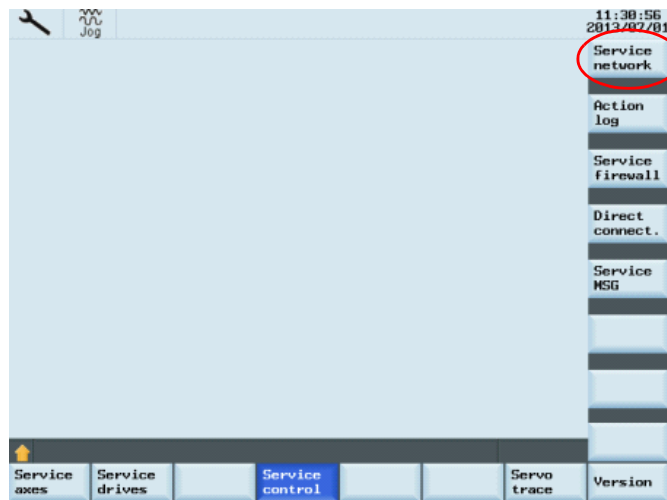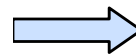
Press "Network Info" button to enter the "Local Configuration Data"   ⟹   Service network

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

① DHCP is set to "No"

② IP address and subnet mask can be arbitrarily set according to need. (Right given only as an example)

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

Save

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

Model G0876 (Mfd. Since 12/22)

SEQUENCE

| Step 2 | Set the PC's static IP address. |

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

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

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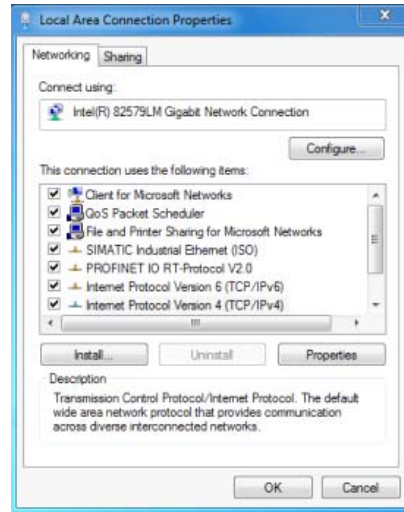
In the dialog box, select "Use the following IP address" and fill in the required IP   address.

(Shown right only given as an example)

Select "OK" to complete the setup.
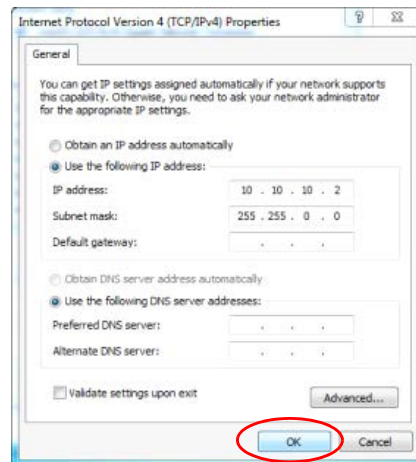
Note: The address "10.10.10.2" setting is based on the first step in the IP address of the PPU.

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

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

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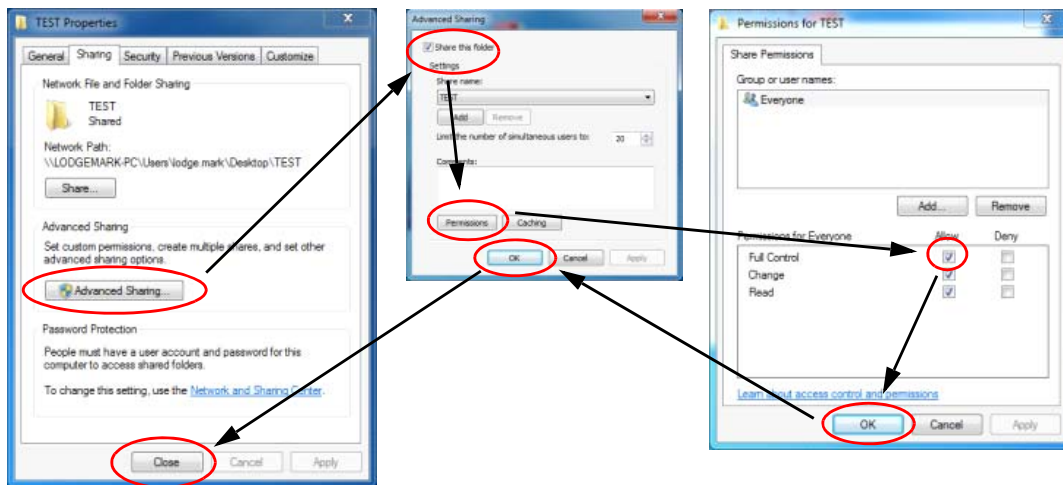
In the dialog window, select "Advanced Sharing"

Then check "Share this folder"

Then select "Permissions" and check "Full control"

Select "OK" - "Ok" - "Close" to activate the settings.

In this folder you can put some machining program.

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

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

SEQUENCE

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

Server: IP address

Share Name: the name of the shared folder

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

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

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

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

Connect

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

This will be shown with the text "Connection succeeded"
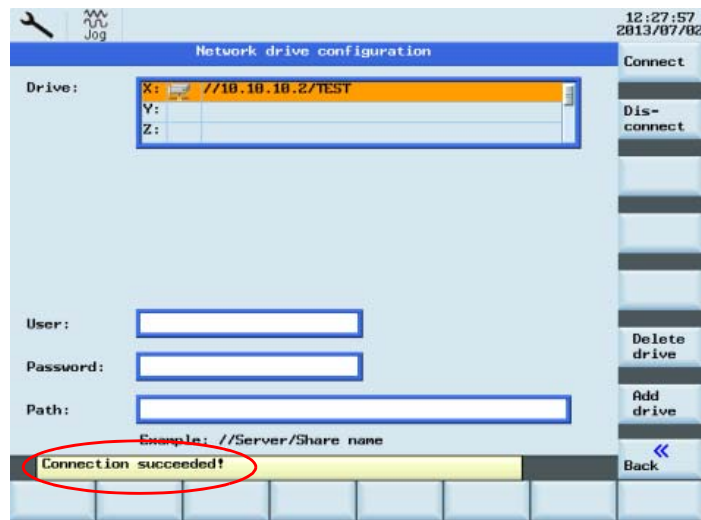
Press "Program Manager" Button

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

Netwo. drive

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

INPUT

Model G0876 (Mfd. Since 12/22)
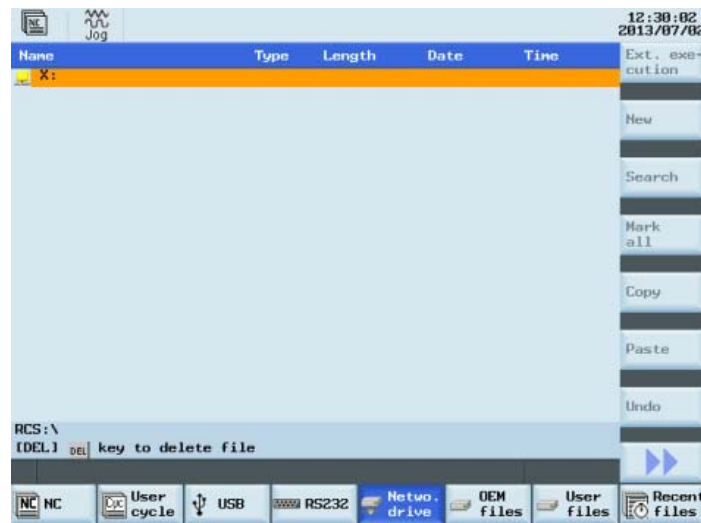
SEQUENCE

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

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

Ext. exe-cution
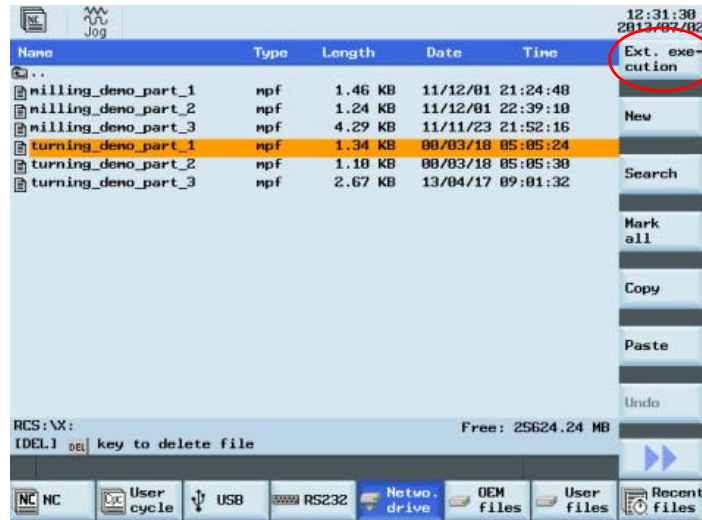
The system will automatically jump to AUTO mode, select the appropriate NC program.

Press the "Cycle Start" button for machining operation.
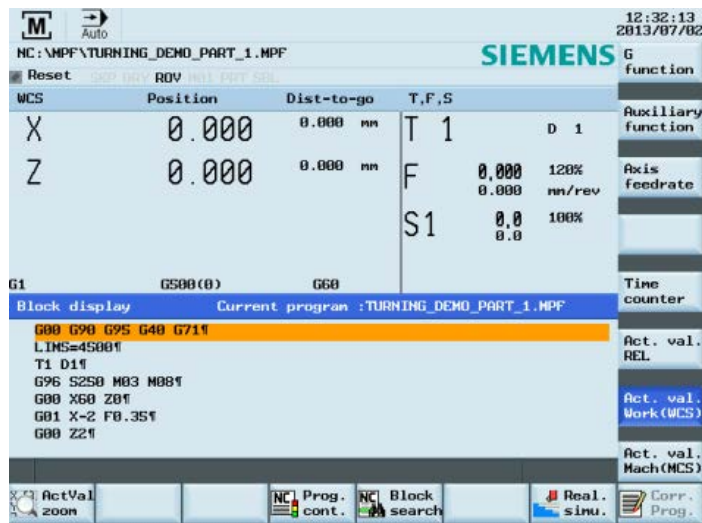
Note: You can also use the "Copy", "Paste" key to achieve "NC", "USB" and "Network Drive" moving files.
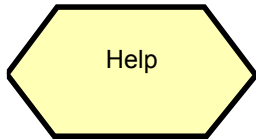
A shared network drive can be made using an ethernet connection between the PC and the PPU so the transferring and backup of NC programs can be performed easier.

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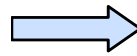
The help information related to the current topic will be shown on screen.

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.
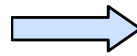
The online help from the Siemens manual will be shown.

"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. $\implies$ [M MACHINE icon]
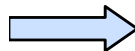
Press the "JOG" key on the MCP. $\implies$ [JOG icon]
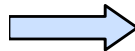
Press the "Sett." SK on the PPU. $\implies$ [Sett. icon]

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

Press the "Input" key on the PPU to activate the settings. $\implies$ [INPUT icon]
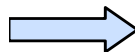
Settings		mm > inch
Retraction plane	10.000 mm	
Safety distance	3.000 mm	
JOG - Feedrate	0.000 mm/min	
Variable increment	0 inc	
Dir. of rotation	M3	

T,S,M	Set rel	Meas. work.	Meas. tool			Face cutt.	Sett.

Press "Face cutt." SK on PPU.

⟹
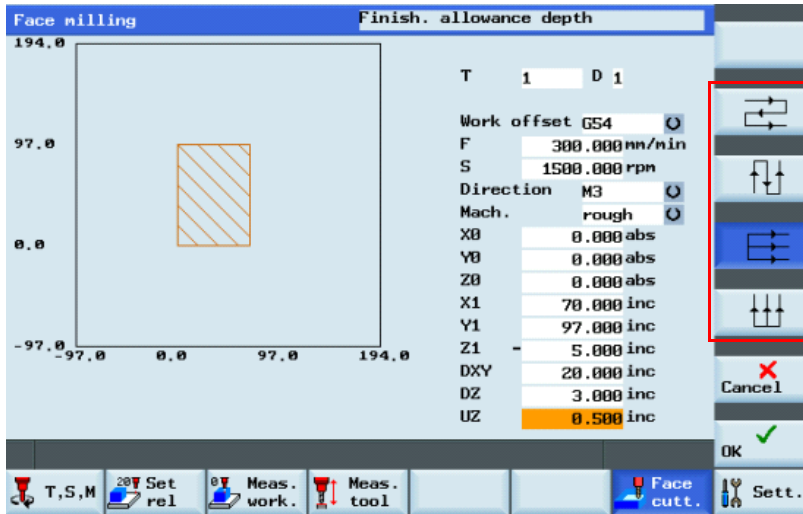
Face cutt.

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

⟹

OK ✓
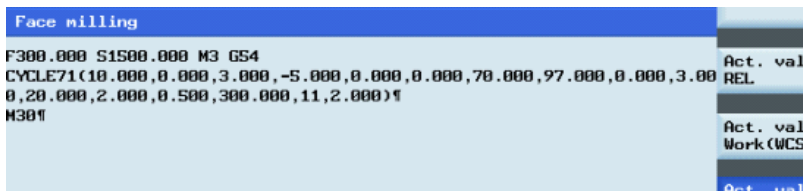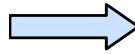
The system now automatically creates the programs.

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

Press the "Cycle Start" key on the MCP.

⟹

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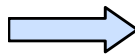
Adjust the override on the MCP gradually to the required values.

SEQUENCE

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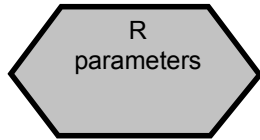
The arithmetic parameters are used in a part program for value assignment, and also for some necessary value calculations. The required values can be set or calculated by the control system during program execution. Some of the common arithmetic functions are shown below:

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

Note:

Reprocessing stop

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

Below shows the connection between program and "R variables" status windows

Press the "Offset" key on the PPU.

Press the "R var." SK on the PPU.
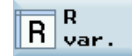
N10 G17 G90 G54

N20 T1 D1

N30 S2500 M03 M08

N40 G00 X-10.0 Y0 Z10

N50 R1=0 R2=0 R3=0

N60 STOPRE

N70 M00

N80 R1=1

N90 STOPRE

N100 M00

N110 R2=2

N120 STOPRE

N130 M00

N140 R3=R1+R2

N150 STOPRE

N160 G00 X=R3

N170 M30

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

R variables

R0	0.000000
R1	0.000000
R2	0.000000
R3	0.000000
R4	0.000000
R5	0.000000

R variables

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

R variables

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

R variables

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

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

SEQUENCE

Time change

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

Press "Shift" and "Alarm" on the PPU simultaneously.
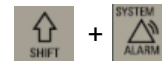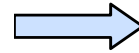
Make sure the password is set to the "CUSTOMER" access level.

Press the "HMI" SK on the PPU.
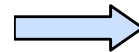
Press the "Date time" SK on the PPU.
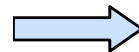
Date and Time

Date and Time setting

Current	2012/04/25	15:12:26
Format	YYYY/MM/DD	HH:MM:SS
New	0000 /00 /00	00 :00 :00

Enter a new "Date" and "Time".

Date and Time

Date and Time setting

Current	2012/04/25	15:14:25
Format	YYYY/MM/DD	HH:MM:SS
New	2012 /4 /26	15 :15 :36

Press the "OK" SK on the PPU.

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

Save data

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

Press "Shift" and "Alarm" on the PPU simultaneously.
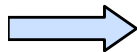
Make sure the password is set to the "CUSTOMER" access level.

Press the "Save data" SK on the PPU.

Machine configuration

No.	Axis index	Name	Axis type	Drive number

Message

Do you want to save the data ?
Yes ==>> then 'OK'   ---   No ==>> Press 'Cancel'

NOTICE !!
This function is only available in NCK_RESET

Saving comprises data areas:
- Machine and setting data
- Tool offset data list
- R variables
- Monitoring and runtime data
- Work offsets
- Workpiece programs and cycles
- GUDs (global user data)

Press the "OK" SK on the PPU.
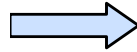
Data being saved

Do not operate or switch off !

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

SEQUENCE

Gear
change

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

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

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

M40   Automatic gear selection

M41   Gear stage 1

M42   Gear stage 2

M43   Gear stage 3

M44   Gear stage 4

M45   Gear stage 5

Example:

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

S0...500   Gear stage 1   → M41

S400..1200   Gear stage 2   → M42

S1000..2000   Gear stage 3   → M43

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

7.12   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

MDA

M/H function

Subprogram

Polar coordinates

Additive workpiece offsets

Coordinate rotation ROT AROT

Scaling

Program example

Program jump

Program skip

Calculator

End

SEQUENCE

MDA

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

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

Press the "Machine" key on the PPU. ⟹ 🅼 MACHINE

Press the "MDA" key on the PPU. ⟹ ▣ MDA

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. ⟹ ◈ CYCLE START

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

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 10176.225	100% mm/min	
Z	50.000	0.000 mm	S1	0.0 0.0	100% 0	

🅼 ▣ MDA

⊘ Stop   SKP DRY **ROV** M01 PRT SBL

10:44:36
2012/04/28

SIEMENS

G function

Auxiliary function

Axis feedrate

Save file

Delete file

Act. val. REL

G00   G54   G60

MDI - Block

█0 G54 X10 Y10 Z50¶
==eof==

SEQUENCE

M/H
function

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

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

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

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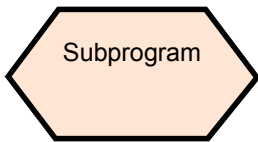
Subprogram

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

Subprogram for positions of the four pockets.
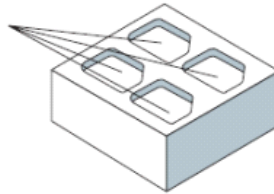
Example

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

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

However, the following rule should be observed:

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

Example: LRAHMEN7

SEQUENCE

Main program

MAIN 123
...
...
...
N20 L10; call
...
...
...
...
N80 L10; call
...
...
...
M30

Sequence

Subprogram

L10
N10 R1=34…
N20 X...Z….
…
...
M17
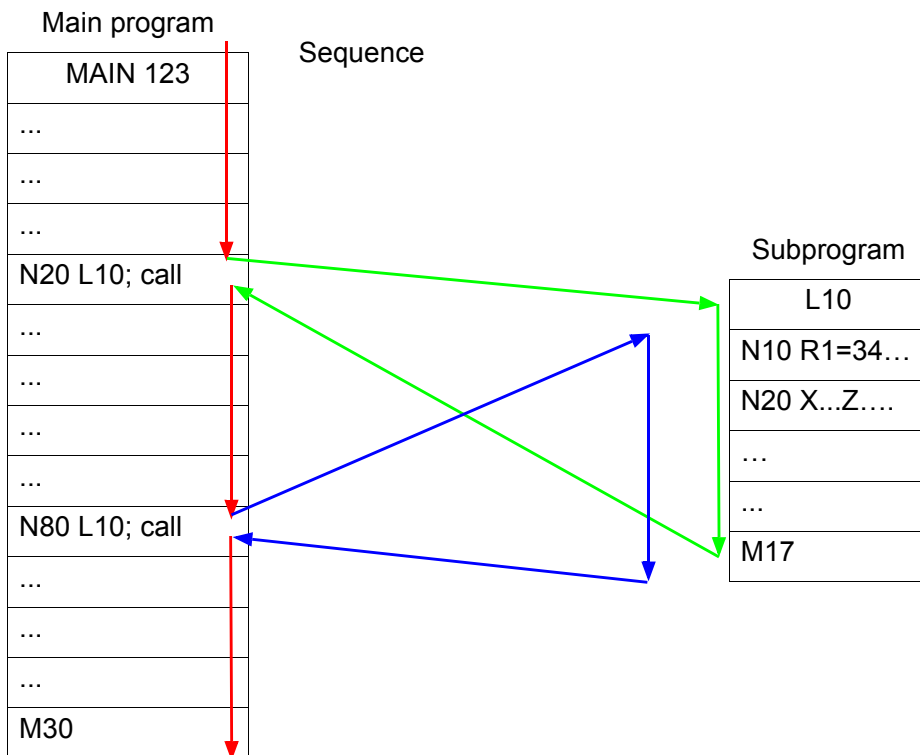
Subprograms can be called from a main program, and also from another subprogram. In total, up to eight program levels, including the main program, are available for this type of nested call.
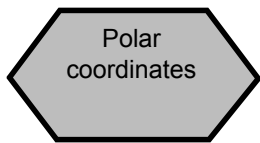
In addition to the common specification in Cartesian coordinates (X, Y, Z), the points of a workpiece can also be specified using polar coordinates.

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

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

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

The polar angle AP= is always referred to the horizontal axis (abscissa) of the plane (for example, with G17: X axis). Positive or negative angle specifications are possible. The positive angle is defined as follows:

Starting from the plus direction of X axis and rotates CCW.

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

Basic Theory

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

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

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

Programming example

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

Example G17: X/Y plane
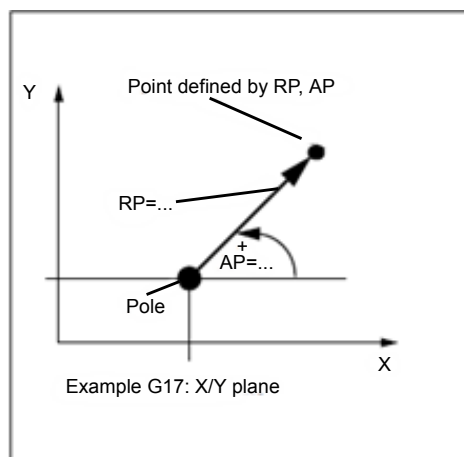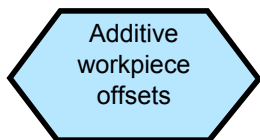
The programmable workpiece offsets TRANS and ATRANS can be used in the following cases:

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

This results in the current workpiece coordinate system.

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

Programming example

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

SEQUENCE

Coordinate
rotation
ROT AROT
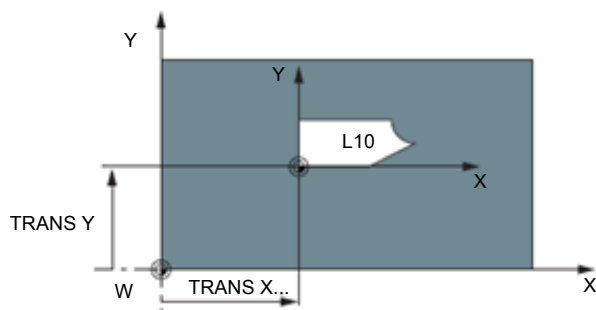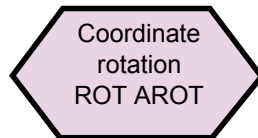
The programmable rotation ROT, AROT can be used:

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

ROT RPL=...	; programmable rotation offset (absolute).
AROT RPL=…	; programmable offset, additive to existing offset (incremental)
ROT	; without values, clears old commands for offset
N10 G17	
N20 AROT RPL=45	additive 45 degree rotation
L10	subprogram call

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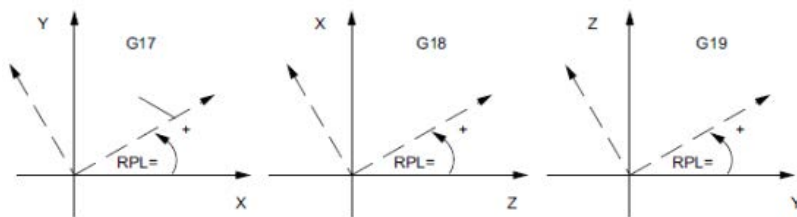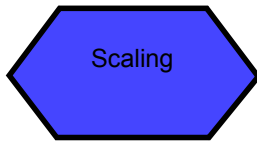
A scale factor can be programmed for all axes with SCALE, ASCALE. The path is enlarged or reduced by this factor in the specified axis. The currently set coordinate system is used as the reference for the scale change.

SCALE X...Y... Z...   ; programmable rotation offset (absolute)

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

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

Programming example

N10 G17

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

L10   subprogram call
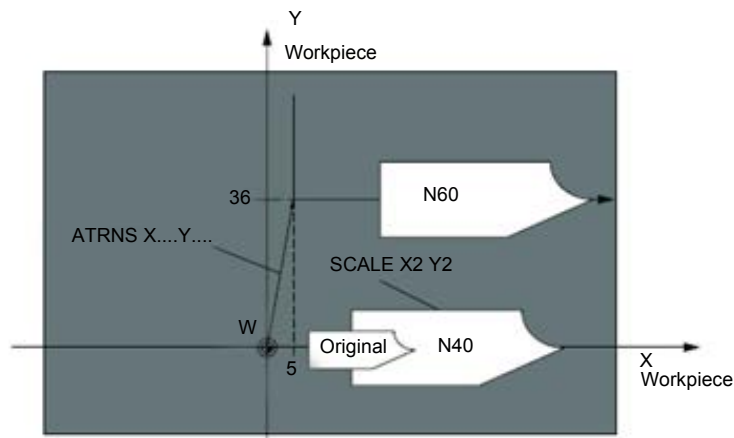
SEQUENCE

Program
example

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

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

Scale Factor 1,3
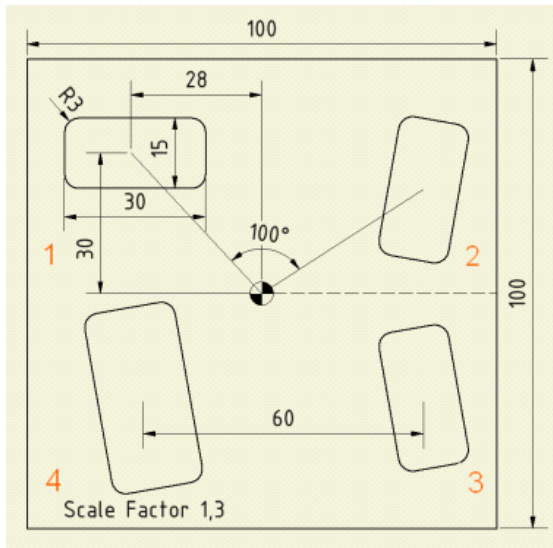
Drawing 1 — original workpiece machining

Drawing 2 — coordinate rotates 100º

Drawing 3 — ① Drawing 2 along X axis mirror image

　　　　② Coordinate rotates 20º

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

　　　　② enlarge 1.3 times in X and Y

　　　　direction

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

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

SEQUENCE
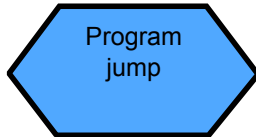
Program
jump

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

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

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

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

Program execution

N10 G0 X...Z...
...
...
N40 GOTOF LABEL0; jumps to label LABEL0
...
...
N70 LABEL0: R1=R2+R3
N80 GOTOF LABEL1; jumps to label LABEL1
N90 LABLE2:
M30; program ends
N110 LABEL1:
...
N130 GOTOB LABEL2 ;jumps to label LABEL2
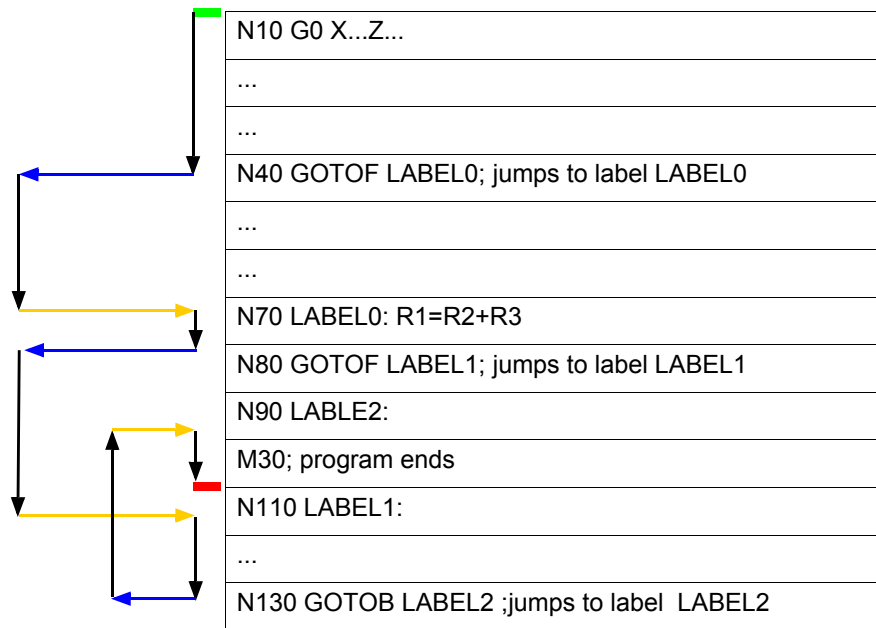
Unconditional jump example

SEQUENCE

Program
skip

Method 1

";" code

Using ";" code at the beginning of the block can skip this string.

";" can also be used to add remarks to the block.

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

N5 G17 G90 G500 G71

N10 T1 D1 M6
N15 S5000 M3 G94 F300
N20 G00 X50 Y50 Z5
N25 G01 Z-20
N30 Z5
...
N85 T2 D1 M6 ; Tool change
N90 S5000 M3 G94 F300
; N95 G00 X60 Y55 Z10

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

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

Model G0876 (Mfd. Since 12/22)

Press the "Machine" key on the PPU. ⟹
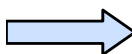
Press the "Auto" key on the MCP. ⟹

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

Press the "Prog cont." SK on the PPU. ⟹ Skip
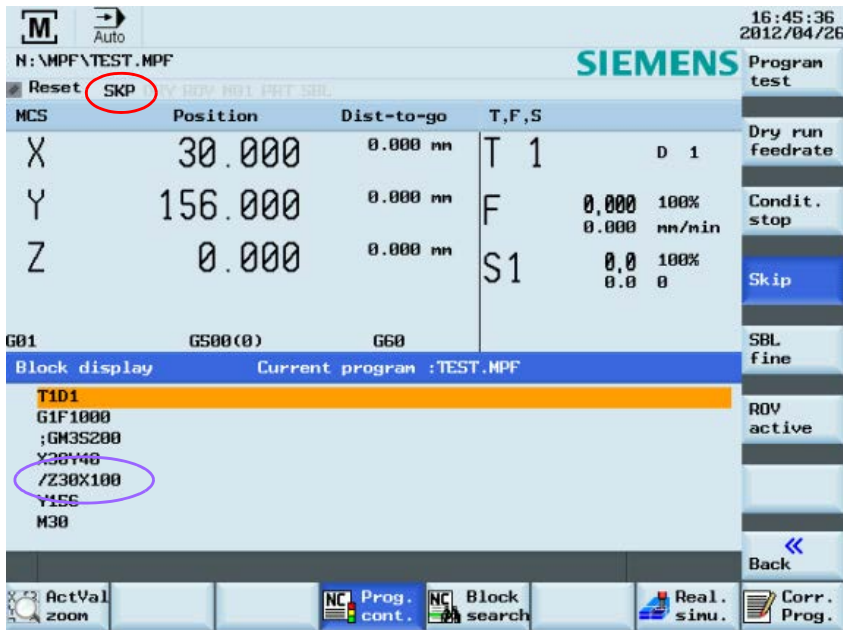
When "SKP" is displayed (red circle), the skip function has been activated.

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

SEQUENCE
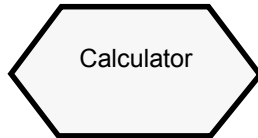
Calculator

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

Press the "=" SK on the PPU.

Model G0876 (Mfd. Since 12/22)
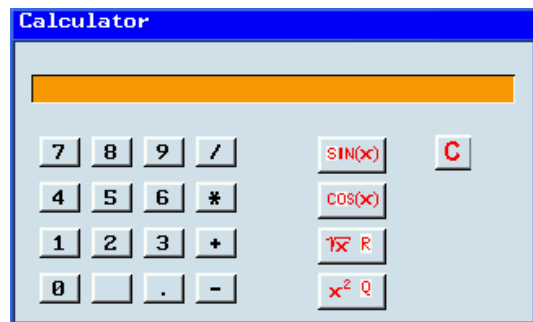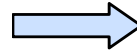
C Delete　Press this SK to delete the contents in the calculator.

« Back　Press this SK to exit the calculator screen.

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

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

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

7.13 Sample Program

Description

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

Content
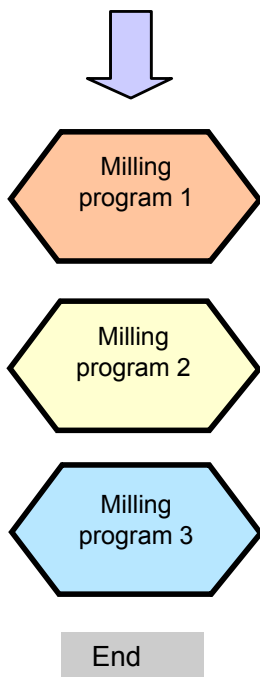
Milling
program 1

Milling
program 2

Milling
program 3

End

Drawing

Milling program 1
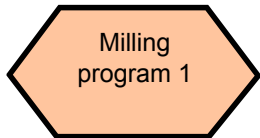
Make sure all the preparations and safety measures have been performed before machining!
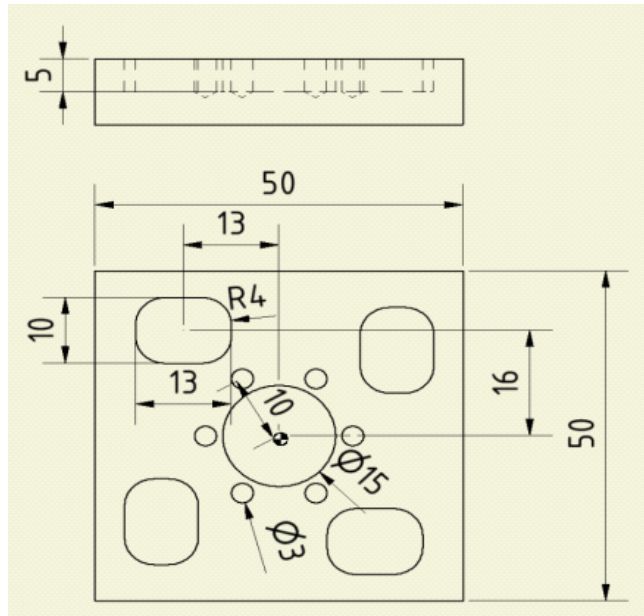
The zero point of the workpiece is located at the center point of the workpiece

Tool information:

T1 Milling tool D50
T2 Milling tool D8
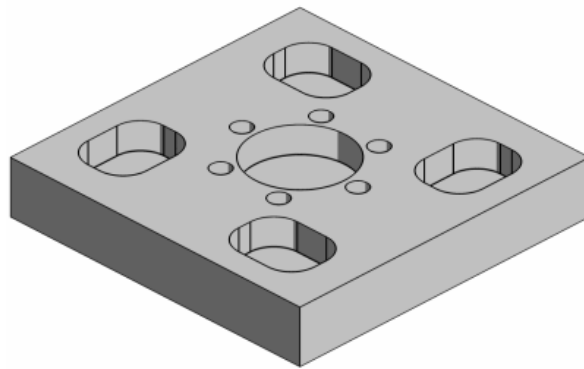
Machining Process

N10 G17 G90 G54 G60 ROT	**N10**
N20 T1 D1; FACEMILL	**N20** tool 1 is plane milling tool
N30 M6	**N30**
N40 S4000 M3 M8	**N40**
N50 G0 X-40 Y0	**N50**
N60 G0 Z2	**N60**
; =======Start face milling========	; =========Start face milling========
N70 CYCLE71(50, 1, 2, 0, -25, -25, 50, 50, 0, 1, , , 0, 400, 11,)	**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 S4500	**N80**
N90 CYCLE71(50, 1, 2, 0, -25, -25, 50, 50, 0, 1, , , 0, 400, 32,)	**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=========	; =========End face milling=========
N100 G0 Z100	**N100**
N110 T2 D1 ; ENDMILL D8	**N110** tool 2 is face milling tool, diameter 8 mm
N120 M6	**N120**
N130 S4000 M3	**N130**
N140 M8 G0 X-13 Y16	**N140**
N150 G0 Z2	**N150**
; =======Start rectangular pocket roughing==========	; ===Start ① rectangular pocket roughing===
N160 _ANF:	**N160** _ANF: Milling start sign
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)	**N170** milling rectangular groove (depth 5 mm, length13 mm, width 10 mm, corner radius 4 mm, groove base point coordinate (X-13,Y16), angle between groove vertical axis and plane X axis is 0º), feedrate 300 mm/min, milling direction G2, rough machining, use G1 vertical groove center to insert.
; ==Adaptive rotation around Z axis==	; ==Adaptive rotation around Z axis==
N180 AROT Z90	**N180** rotation in positive direction 90º
N190 _END:	**N190** _END: Milling end sign

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

; ====Repeat ② ③ ④   rectangular pocket milling 3 times=====
N200   Repeat N160 ~ N190 operation three times
; =======Cancel rotation=========
N210   cancel all the coordinate rotation commands
N220

; ===Start ①   rectangular pocket finishing===
N230   _ANF1: Milling start sign
N240   milling rectangular groove (depth \ length \ width \ corner radius \ base point \ corner angles are the same as the above parameters), plane feedrate300 mm/min, depth direction feedrate200 mm/min, milling direction G2, finish machining.
; ==Adaptive rotation around Z axis===
N250   rotation in positive direction 90º
N260   _END1: Milling end sign

; ====Finishing ② ③ ④   rectangular pocket milling =====
N270   repeat N230~N260 operation three times
N280   cancel all the coordinate rotation commands
; =====Cancel rotation==========

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

Drawing
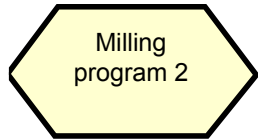
Make sure all the preparations and safety measures have been performed before machining!
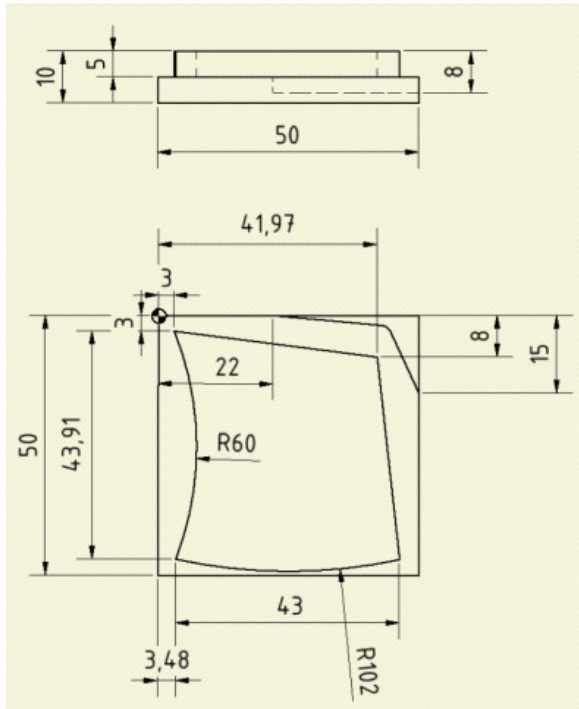
Workpiece zero point is located in the top left corner.

Tool information:

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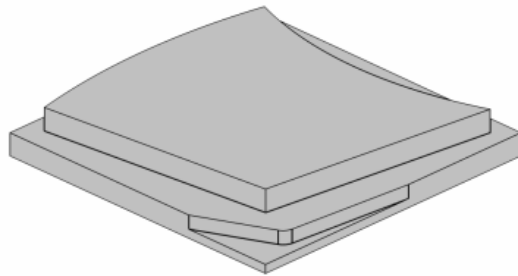
Machining Process

N10   G17 G90 G60 G54
N20   T1 D1 ;FACEMILL D50
N30   M6
N40   S3500 M3
N50   G0 X0 Y0
N60   G0 Z2
  ; ======Start face milling========
N70   CYCLE71(50, 1, 2, 0, 0, 0, 50, -50 , , 1, 40, , 0.1, 300, 11,)
N80   S4000 M3
N90   CYCLE71(50, 0.1, 2, 0, 0, 0, 50, -50 , , 1, 40, , 0, 250, 32,)
  ; =====Start contour milling=====
N100   T2 D2 ;END MILL
N110   M6
N120   S3500 M6
N130   CYCLE72("SUB_PART_2", 50, 0, 2, -5, 2, 0.1, 0.1, 300, 300, 11, 42, 1, 4, 300, 1, 4)
  ; ======Start path milling with radius compensation ========
N140   T4 D1 ;ENDMILL D10
N150   M6
N160   S4000 M3
N170   G0 X55 Y-15
N180   G0 Z2
N190   G1 F300 Z-8
N200   G42 G1 Y-15 X50
N210   G1 X44 Y-2 RND=2
N220   G1 Y0 X 22
N230   G40 Y30
N240   M30

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

SUB_PART_2.SPF   ***CONTOUR****

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

Drawing
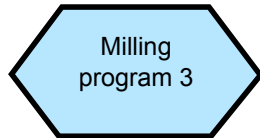
Milling program 3

Part of the cycles in the program are taken as examples in Section 5, "Create Part Program Part 2"!
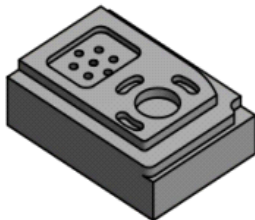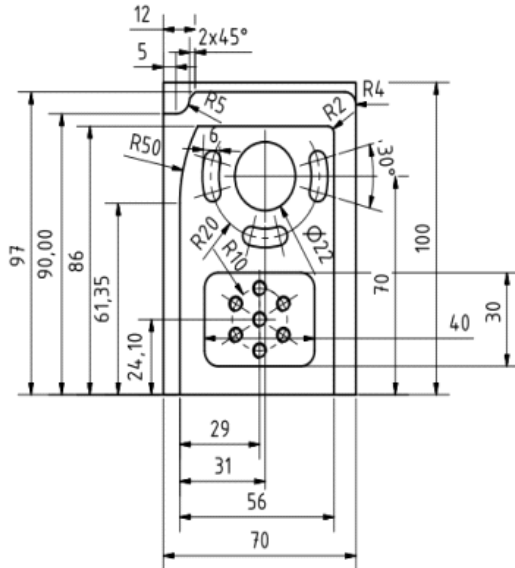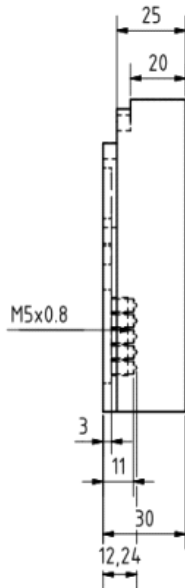
M5x0.8

25
20
3
11
30
12,24

12
5
2x45°
R5
R50
R4
R2
30°
Ø22
R20
R10
97
90,00
86
61,35
24,10
100
70
40
30
29
31
56
70

Tool information

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

Machining Process

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

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

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

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

Machining Process

N270 SUPA G00 Z300 D0	**N270**
N280 SUPA G00 X300 Y300	**N280**
N290 T4 D1	**N290**
N300 MSG ("Please change to Tool No 4")	**N300** hint:change to tool 4
N310 M05 M09 M00	**N310**
; ===Circular pocket milling start====	; ====Circular pocket milling start===
N320 S5000 M3	**N320**
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)	**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 S5500 M3	**N340**
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)	**N350** repeat N370 milling process, the difference is the machining allowance.
; ===Circular pocket milling end====	; ====Circular pocket milling end===
N360 SUPA G00 Z300 D0	**N360**
N370 SUPA G00 X300 Y300	**N370**
N380 T5 D1	**N380**
N390 MSG ("Please change to Tool No 5")	**N390** hint:change to tool 5
N400 M05 M09 M00	**N400**
; =======Slot milling start=======	; =======Slot milling start======
N410 M3 S7000	**N410**
N420 SLOT2(50, 0, 2, , 3, 3, 30, 6, 38, 70, 20, 165, 90, 300, 300, 3, 3, 0.2, 0, 5, 250, 3000,)	**N420** milling slot(depth 3 mm, machining 3 slots, slot angle 30º, slot width 6 mm, basic circle center point coordinate(X38,Y70), basic circle radius 20 mm, start angle 165º, slot incremental angle 90º, 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=======	; =======Slot milling end======

N430   SUPA G00 Z300 D0
N440   SUPA G00 X300 Y300
; ======Contour milling start======
N450   T2 D1
N460   MSG ("Please change to Tool No 2")
N470   M05 M09 M00
N480   S5000 M3
N490   CYCLE72("SUB_PART_3", 50, 0, 2, -5, 5, 0, 0, 300, 100, 111, 41, 12, 3, 300, 12, 3)
; ======Contour milling end=======
N500   SUPA G00 Z300 D0
N510   SUPA G00 X300 Y300
; =Rectangular pocket milling start==
N520   T2 D1
N530   MSG ("Please change to Tool No 2")
N540   M05 M09 M00
N550   S6500 M3
N560   POCKET3(50, 0, 1, -3, 40, 30, 6, 36, 24.1, 15, 3, 0.1, 0.1, 300, 300, 0, 11, 12, 8, 3, 15, 0, 2)
N570   POCKET3(50, 0, 1, -3, 40, 30, 6, 36, 24.1, 15, 3, 0.1, 0.1, 300, 300, 0, 12, 12, 8, 3, 15, 0, 2)
; ==Rectangular pocket milling end==

; ======Contour milling start======

N450

N460   hint:change to tool 2

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=======

; =Rectangular pocket milling start==

N530   hint:change to tool 2

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 15º), finishing allowance 0.1 mm, feedrate surface machining and cutting direction machining is 300 mm/min, milling in positive direction, rough machining, use G1 vertical groove center to insert.
N570   repeat N600 milling process, the difference is the machining allowance.
; ==Rectangular pocket milling end==

Machining Process

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

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

N790	SUPA G00 Z300 D0	**N790**
N800	SUPA G00 X300 Y300	**N800**
	; ========Tapping start========	; ==========Tapping start========
N810	T8 D1	**N810**
N820	MSG ("Please change to Tool No 8")	**N820** hint:change to tool 8
N830	M05 M09 M00	**N830**
N840	S500 M3	**N840**
N850	MCALL CYCLE84(50, -3, 2, , 6, 0.7, 5, , 2, 5, 5, 5, 3, 0, 0, 0, 5, 1.4)	**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 5º , 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	HOLES2(36, 24.1, 10, 90, 60, 6)	**N860** hole arrangement circular center coordinate (X36,Y24.1), circular radius 10 mm, start angle 90º, angle between the holes is 60º, circular hole number 6
N870	X36 Y24.1	**N870** continue drilling with X36,Y24.1) as the center tapping
N880	MCALL ; Modal call Off	**N880** cancel mode recall instruction
	; ========Tapping end=========	; ==========Tapping end=========
N890	SUPA G00 Z500 D0	**N890**
N900	SUPA G00 X500 Y500;	**N900**
	; ========Move to the change position Ready to start next program or repeat =========	; ======Move to the change position Ready to start next program or repeat ======
N910	M30	**N910**

Machining Process

SUB_PART_3.SPF

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 */

CONTOUR*

7.14   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 is ISO mode chapter can be run in 808D ISO mode.

Content

ISO function switch

ISO code explanation

ISO program transfer and operation

End

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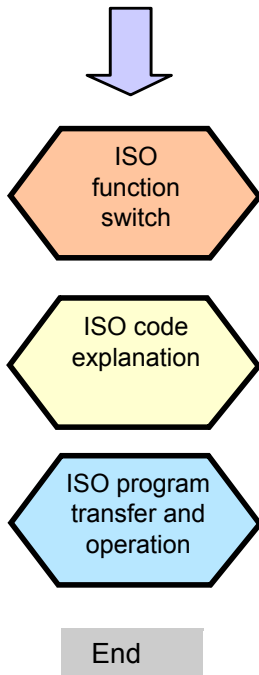
ISO function switch
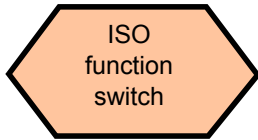
Siemens standard machining codes are implemented in DIN mode. The 808D also provides appropriate functions for implementing the ISO commands, but the ISO mode must be activated during operation.

ISO function switch

Method 1

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

SHIFT + ALARM (SYSTEM)

Press the "ISO mode" SK on the right.

ISO mode

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

OK

Machine configuration

No.	Axis index	Name	Axis type	Drive number
1	1	X	Linear axis	
2	2	Y	Linear axis	
3	3	Z	Linear axis	
4				

After the new configuration is applied, a NCK/PLC/HMI restart will be triggered!

Cancel

OK

15:44:16
2012/04/26
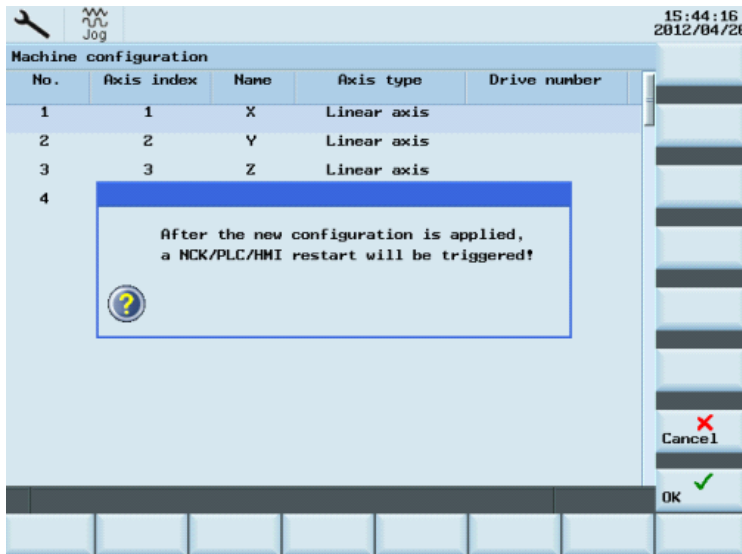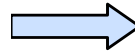
Basic Theory

After pressing "OK", the system restarts automatically.

After restarting, press "Shift" + "System - Alarm" again and if the symbol in the red circle appears, ISO mode is already activated.
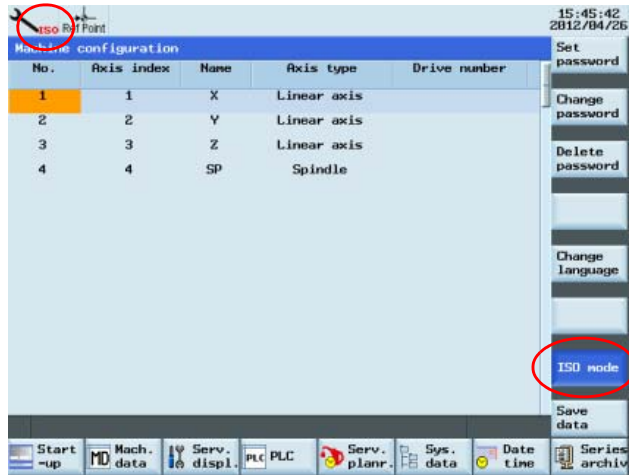
A red ISO appears at the top of the screen and the ISO mode button on the right is highlighted in blue.

Method 2

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

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

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

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

If ISO is displayed at the top of the screen, it is activated.
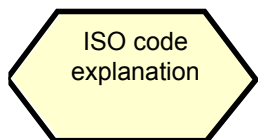
Brief description of typical, frequently used ISO codes

ISO code	Description	Compare with DIN
G00	Orientation (rapid traverse)	As DIN
G1	Linear difference	As DIN
G17/G18/G19	XY plane / ZX plane / YZ plane	As DIN
G20/G21	Input in inch/mm	G70/G71
G41/G42/G40	Left tool tip radius compensation / right tool tip radius compensation / cancel tool radius compensation	As DIN
G54 ~ G59	Select workpiece coordinate system	As DIN
G80	Cancel fixed cycle	
G90/G91	Absolute/incremental programming	
G94/G95	Feedrate F in mm/min / mm/r	As DIN
S	Spindle speed	As DIN
，R	Reverse circle (note the form there must be "，" before R parameter)	RND
M3/M4/M5	Spindle right / spindle left / spindle stop	As DIN
M98 P _L_	Subprogram call (P+ subprogram name/ L+ times)	Program name + L
M99	End of Subroutine	M17

Basic Theory

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

G43/G44 and **G49**

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

G43: Tool length compensation in positive direction
G44: Tool length compensation in negative direction
G49: Cancel tool length compensation
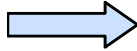
H01→Offset value 20.0
H02→Offset value -30.0
H03→Offset value 30.0
H04→Offset value -20.0

G90 **G43** Z100.0 **H01**;Z will reach 120.0
G90 **G43** Z100.0 **H02**;Z will reach 70.0
G90 **G44** Z100.0 **H03**;Z will reach 70.0
G90 **G44** Z100.0 **H04**;Z will reach 120.0

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

G98 :Fixed cycle back to the original point
G99 :Fixed cycle back to R point
G80 :Cancel the fixed cycle
Pausing function G04
G04 X5.0 >delay 5 s
G04 P5 >delay 5 ms

N5   G90 T1 M06
N10 M3 S2000; spindle rotation
N20 G99 G81 X300 Y-250 Z-150
      R-10 F120;**after orientation drilling, back to R point**
N30 X1000.   ;after orientation drilling, back to R point
N40 G04 X2.0 ;delay 2 s
N50 G98 Y-550 ;after orientation drilling, back to start point
N60 G80   ;cancel the fixed cycle
N70 M5   ;spindle rotation stop
N80 M30

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

G02 circular interpolation in positive direction

G03 circular interpolation in negative direction
You can specify the circle end point in the following X/Z address for both. You can also describe circle radii with I, J, K incremental or use parameter R to specify radii directly.
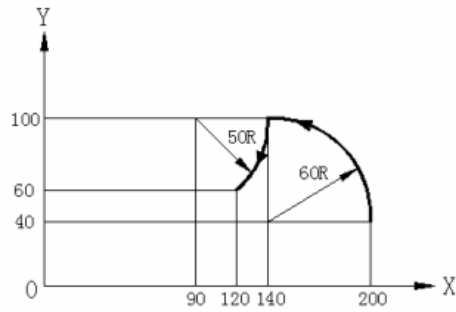
Method 1 (use incremental to describe circular radius)
G92 X200.0 Y40.0 Z0
G90 G03 X140.0 Y100.0 I-60.0 F300.0
     G02 X120.0 Y60.0 I-50.0

Method 2 (use parameter R to describe circular radius)
G92 X200.0 Y40.0 Z0
G90 **G03 X140.0 Y100.0 R60.0 F300**
     G02 X120.0 Y60.0 **R50.0**

When specifying circle radii with parameter **R**

Circles less than 180° 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**

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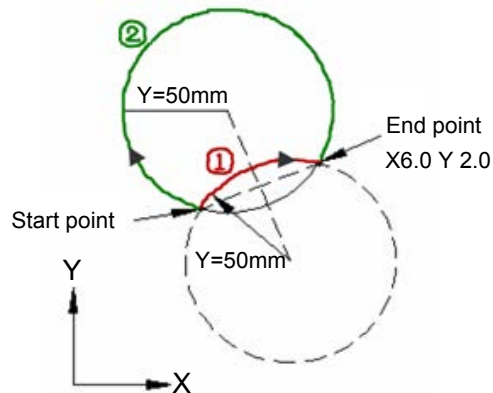
P.	Descriptions	Unit	Applied range and note
Frequently used letter meanings of typical fixed cycle codes in ISO mode.			
X/Y	Cutting end point X/Z absolute coordinate values		G73 / G74 / G76 G81 ~ G87 / G89
Z	The distance incremental value between R point and the bottom of the hole, or the absolute coordinate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

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

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

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

G73 fast-speed deep hole drilling
Common programming structures:
G73 X—Y—Z—R—Q—F—K
Motion process:

① Drilling motion (-Z) → intermediate feed

② Motion at the bottom of the hole → none

③ Retraction motion (+Z) → fast feed

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

G74 reverse tapping cycle
Common programming structures:
G74 X—Y—Z—R—P—F—K
Motion process:

① Drilling motion(-Z) → cutting feed

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

③ Retraction motion(+Z) → cutting feed

G74 application example program:

M4 S100 ;spindle rotation
G90 G99
G74 X300 Y-250 Z-150 R-120 P300 F120
;after orientation drill 1st hole, back to R point
Y-550　　　 ;after orientation drill 2nd hole, back to R point
Y-750　　　 ;after orientation drill 3rd hole, back to R point
X1000　　　;after orientation drill 4th hole, back to R point **Y-550**　　　 ;after orientation drill 5th hole, back to R point **G98 Y750** ;after orientation drill 6th hole, back to R point **G80** ;cancel fixed cycle
G28 G91 X0 Y0 Z0　　　;back to reference point
M5　　　　　 ;spindle rotation stop
M30

Basic Theory

G76 Boring cycle
Common programming structures:
G76 X—Y—Z—R—Q—P—F—K
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → spindle stop directional

③ Retraction motion (+Z) → fast feed

G76 application example program:

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

G81 Drilling cycle (fixed point drilling)
Common programming structures:
G81 X—Y—Z—R—F—K
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → none

③ Retraction motion (+Z) → fast feed

G81 application example program:

M3 S2000   ;spindle rotation
G90 G99 G81 X300 Y-250 Z-150 R-10 F120
     ;after orientation drill 1st hole, back to R point
Y-550 ;after orientation drill 2nd hole, back to R point
Y-750 ;after orientation drill 3rd hole, back to R point
X1000 ;after orientation drill 4th hole, back to R point
Y-550 ;after orientation drill 5th hole, back to R point
G98 Y-750;after orientation drill 6th hole, back to start plane
G80   ;cancel fixed cycle
G28 G91 X0 Y0 Z0   ;back to reference point
M5   ;spindle rotation stop
M30

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

G82 application example program:

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

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

G83 application example program:

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

Basic Theory

G84 Tapping cycle
Common programming structures:
G84 X—Y—Z—R—P—F—K
Motion process:

① Drilling motion (-Z) → cutting feed

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

③ Retraction motion (+Z) → cutting feed

G84 execution operation graphic:
With command G99 without operation in red line
With command G98 with operation in red line
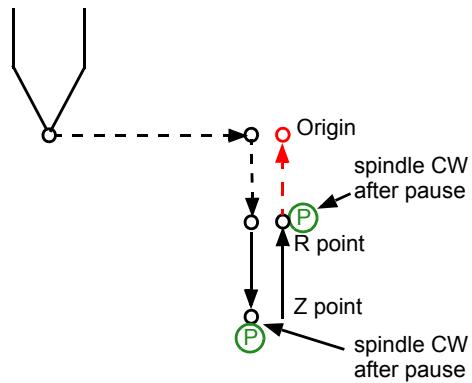
G85 boring cycle
Common programming structures:
G85 X—Y—Z—R—F—K
Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole →
none

③ Retraction motion (+Z) → cutting feed

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

Model G0876 (Mfd. Since 12/22)
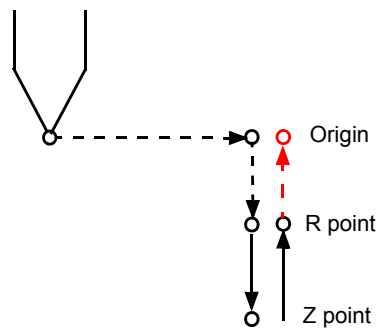
G86 boring cycle
Common programming structures:
G86 X—Y—Z—R—F—K
Motion process:
① Drilling motion (-Z) → cutting feed
② Motion at the bottom of the hole → spindle stop
③ Retraction motion (+Z) → fast feed

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**

spindle CW after pause

Origin

spindle CW after pause

R point

Z point

spindle stop after pause

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**

Origin

R point

Z point

P Pause

Basic Theory

G87 Boring cycle Ⅰ / reverse boring cycle Ⅱ

Common programming structures:

G87 X—Y—Z—R—Q—P—F—L

Motion process:

① Drilling motion (-Z) → cutting feed

② Motion at the bottom of the hole → spindle stops

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

G87 execution operation graphic:

Fixed cycle II→

spindle start

(OSS) Positioning of Spindles

⇨ Tools move

- - ▶ Fast feed

──▶ Cutting feed

⌇▶ Manual feed

Z point

R point

spindle start

G87 execution operation graphic:

Fixed cycle Ⅰ

spindle start
Origin

Z point

spindle stop

Using G98

spindle start
R point

Z point

spindle stop

Using G99

Model G0876 (Mfd. Since 12/22)
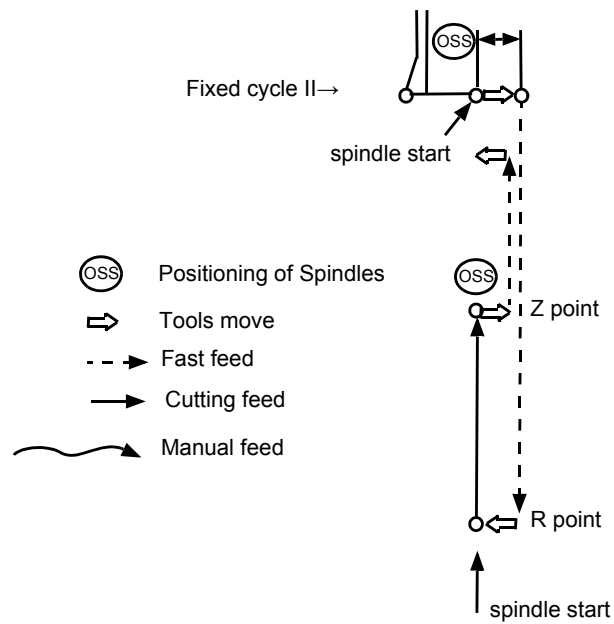
ISO program transfer and operation
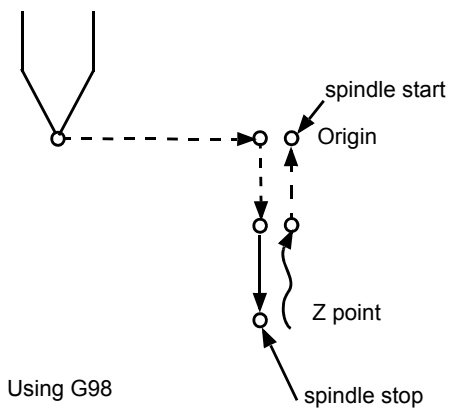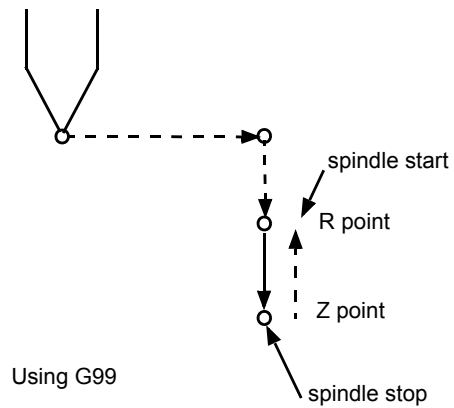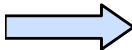
The ISO mode function provided by the 808D can easily operate the existing ISO program!

Step 1   Transfer ISO files in USB device to 808D.
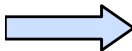
Connect the USB device with the stored target programs to the USB interface on the PPU.

Press the "USB" SK on the PPU. → USB

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

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

Press the "NC" SK on the PPU. → NC NC

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

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

Step 2   Make the necessary changes to the ISO programs.

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

Basic Theory

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.

Model G0876 (Mfd. Since 12/22)

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.   ⟹   NC NC

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

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

Press the "USB" SK on the PPU.   ⟹   USB

Press the "Paste" SK on the PPU.   ⟹   Paste
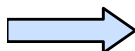
A specified ISO program is then stored in the USB and can be executed as required.

Basic Theory

Sample program

Make sure the current system is in ISO mode!

Make sure all preparations and safety measures have been performed!

Model G0876 (Mfd. Since 12/22)
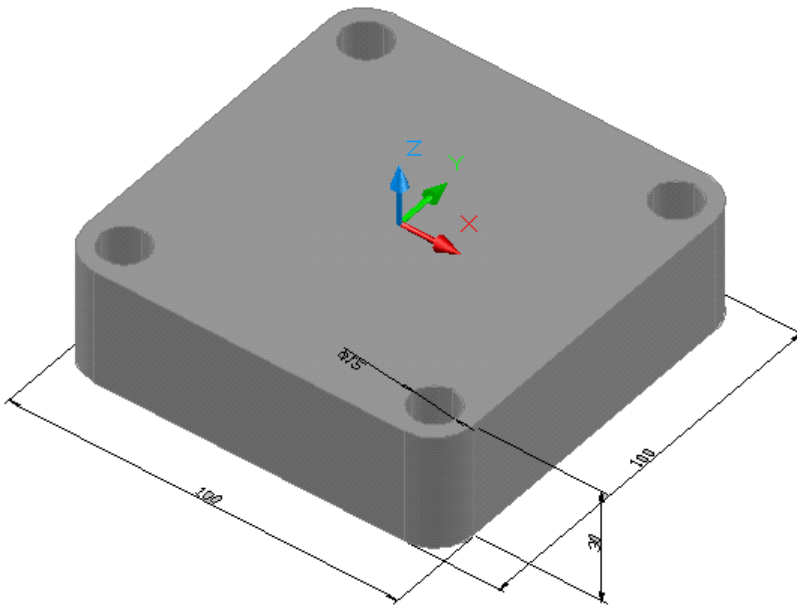
ISO programs can be executed in the 808D as follows:

```
N10   G291
N20   T1M6
N30   G0G54G90G40
N40   M3S1200F200
N50   G43H1Z50
N60   G0X0Y-70
N70   Z5M8
N80   G1Z-5
N90   G01G41X20D1
N100  G03X0Y-50R20
N120  G1X-50,R10
N130  Y50,R10
N140  X50,R10
N150  Y-50,R10
N160  X40
N170  X0
N180  G03X-20Y-70R20
N190  G1G40X0
N200  G0Z50
```

```
N210  T2M6
N220  M3S3000F100
N230  G43H2Z50
N240  G0X40Y-40
N250  Z20
N260  G81Z-2R10
N270  Y40
N290  X-40
N300  Y-40
N310  G80
N320  G0Z50

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

Note: This program opens/exits ISO mode with the G291/G290 command. It is recommended to use the first method to open ISO mode — using the ISO mode active button on the PPU (described above)

Basic Theory

Standard Siemens programming.

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

```
N10   T1D1M6 ; contour milling tool
N20   G54G90G40G17
N30   M3S2000M8
N40   G0Z25
N50   X0Y-70
N55   CYCLE72( "SUB_PART_4", 50, 0, 2, -5, 2.5,
0.1, 0.1, 200, 200, 111, 41, 2, 20, 200, 2, 20)
N60   T2D1M6 ; quill, drill center hole
N70   M3S2500M8
N80   MCALL CYCLE82( 50, 0, 2, 0, 2, 0)
N90   CYCLE802( 111111111, 111111111, 40, -
40, 40, 40, -40, 40,
-40, -40,  ,)
N100  MCALL
N110  T3D1M6 ; quill; deep hole drilling
N120  M3S2500M8
N130  MCALL CYCLE83( 50, 0, 2,
-20,  ,-5,  ,3, 0.5, 1, 1, 1, 3, 3, 0,  ,0)
N140  CYCLE802( 111111111, 111111111, 40, -
40, 40, 40, -40, 40,
-40, -40,  ,)
N150  MCALL
```

```
N160  G0G40G90Z60
N170  M09M05
N180  M30

; SUB_PART_4.SPF

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

7.15   Appendix

Content

G Functions

End

Model G0876 (Mfd. Since 12/22)
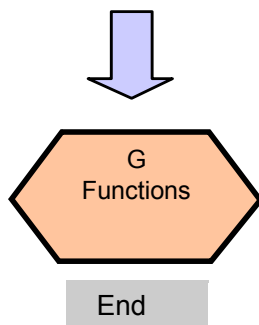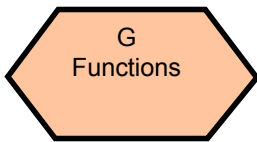
Group 1: Modally valid motion commands

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

Group 6: Plane selection	
Name	Meaning
G17 *	X/Y plane
G18	Z/X plane
G19	Y/Z plane

Group 7: Tool radius compensation	
Name	Meaning
G40 *	Tool radius compensation OFF
G41	Tool radius compensation left of contour
G42	Tool radius compensation right of contour

Group 8: Settable zero offset	
Name	Meaning
G500 *	Settable work offset OFF
G54	1st settable zero offset
G55	2nd settable zero offset
G56	3rd settable zero offset
G57	4th settable zero offset
G58	5th settable zero offset
G59	6th settable zero offset

Group 9: Frame suppression	
Name	Meaning
G53	Non-modal skipping of the settable work offset
G153	Non-modal skipping of the settable work offset including base frame

Group 10: Exact stop — continuous – path mode	
Name	Meaning
G60 *	Exact positioning
G64	Continuous — path mode

Group 11: Exact stop, non-modal	
Name	Meaning
G09	Non-modal exact stop

Group 12: Exact stop window modally effective	
Name	Meaning
G601 *	Exact stop window
G602	Exact stop window, course, with G60, G9

Group 13: Workpiece measuring inch/metric	
Name	Meaning
G70	Inch dimension data input
G71 *	Metric dimension data input
G700	Inch dimension data input; also for feedrate F
G710	Metric dimension data input; also for feedrate F

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

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

Group 16: Feedrate override modally effective	
Name	Meaning
CFC *	Feedrate override with circle ON
CFTCP	Feedrate override OFF

Group 18: Behavior at corner when working with tool radius compensation	
Name	Meaning
G450 *	Transition circle
G451	Point intersection

Group 44: Path segmentation with SAR modally effective	
Name	Meaning
G340 *	Approach and retraction in space (SAR)
G341	Approach and retraction in the plane (SAR)

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

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

8 M - Code list, M functions

8.1 M-function for milling machines according to PAL

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

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

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

No.	Function
M99	

8.2   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

9 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

10   SINUMERIK 808 D

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

Manuals are strictly necessary for the operator, CNC programmer and also for the maintenance and repair personnel of the F4.

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

Manuals that are required for the maintenance and repair personnel.

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

Additional useful manuals and information

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

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

- ❍ Mechanical installation manual
- ❍ Electrical installation
- ❍ Parameter Manual

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

http://support.automation.siemens.com

For any questions regarding the CNC control, please contact:

Siemens AG, A&D techsupport

Phone (+49) 0180 50 50 222

mailto: techsupport@ad.siemens.de

Siemens AG Hotline, Helpline

Phone (+49) 0180 50 50 111

11   Maintenance

In this chapter you will find important information about

❍ Inspection
❍ Maintenance
❍ Repair

of the CNC machine.

ATTENTION!

Properly performed regular maintenance is an essential prerequisite for
❍ **operational safety,**
❍ **failure-free operation,**
❍ **long service life of the CNC machine and**
❍ **the quality of the products which you manufacture.**

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

ENVIRONMENTAL PROTECTION

During work on the cooling lubricant equipment please make sure that
❍ **collecting containers with sufficient capacity for the amount of liquid to be collected are used.**
❍ **Liquids and oils should not be split on the ground.**

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

Collect leakages

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

Disposal

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

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)

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 one-component paint. Take this criterion into account when selecting your cooling lubricant.

Grizzly Industrial, Inc. does not assume any guarantee for subsequent damages due to unsuitable cooling lubricants.

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

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

ATTENTION!

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

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

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

ATTENTION!

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

ATTENTION!

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

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

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

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

❍ The manufacturer should be able to provide a certificate concerning skin tolerance.
❍ The mineral oil 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 non-adhesive according to VKIS sheet 9: Adhesive and residue behavior.
❍ It should not attack the varnish of the machine.
❍ It should not attack any machine elements (metals, elastomers).
❍ Low foaming behavior of the emulsion.
❍ It should be as finely dispersed as possible in order to avoid clogging in 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 personnel should carry out maintenance and repair work on the CNC machine.

Validation

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

Documentation

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

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. Any indicated intervals therefore are only valid for the corresponding approved con-ditions.

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 polycarbonate sight windows using a suitable cleaning agent.	
	Compressed air supply	Draining	☞ "Drain the compressed air service unit" on page 250	
Start of work, Daily after every maintenance or repair work	**Lubricant tank Slideway oil**	Fill level control	Manual Check the quantity and refill the lubricant tank of the central lubrication system. See also ☞ "Cool-ing lubricants" on page 243 3.5 "Refill central lubrication system" on page 33	
	Oil collecting tank	Empty	Check the oil level in the oil collecting gutter. Empty the oil collecting tank.	

Interval	Where?	What?	How?	Check ✔
Every week	CNC milling machine	☞ "If the machine is cold, e.g. directly after having transported the machine, it should be warmed up at a spindle speed of only 500 1/min for the first 30 minutes." on page 35		
Every week		Oiling Lubricate	Oil all bare steel surfaces. Use an acid-free oil, e.g. weapon oil or motor oil. Press the push-button of the central lubricating system.	
Every week	Cooling lubricant tank	Fill level status control	Check for liquid level, concentration, pH value, bacteria and fungal decay.	
Weekly	Cooling lubricant lubricant	Measure pH value	Check the pH value. If required, replace the cooling lubricant.	
Every week	Drip feed lubricator Compressed air cylinder	Fill level control	Check the liquid level and refill if required.	
Every month	Gear belt Spindle head	Check Readjusting Replacing	Check the drive belt on wear and excessive clearance. If required, readjust the gear belt and/or replace it.	
Annually	Clutch Coupling Drive Z-, Y-, X-axis	Check Replace	Check if coupling is worn and check coupling slack. check If necessary, replace the coupling.	

Interval	Where?	What?	How?	Check ✔
Every 1000 operating hours	**Wipers on the slides**	Check Replace	Check the wipers on the slides. Replace if damaged immediately.	
	Electrical cabinet	Cleaning	☞ "Cleaning the electrical cabinet" on page 249	
	Servo motors	Inspection	Connections on the servo motors.	
Every six months	**Tool changer**	Check the tool change function. Positioning	By manual changing	
As required,	**Chip collection tray**	Cleaning	Clean the chip collection tray of the cooling lubricant equipment.	
60 months	**Protective covers**	Replacing the sight window	☞ "Cleaning and replacing of the polycarbonate windows" on page 248	

11.3.1 Cleaning and replacing of the polycarbonate windows

WARNING!

The polycarbonate sight windows are part of the safety device on your CNC machine. Damaged, scratched or even broken polycarbonate sight windows must be replaced immediately.

A soft cloth should be used to clean the machine safety glass.

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

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

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

11.3.2 Cleaning the electrical cabinet

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

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

Clean the inside of the electrical cabinet regularly.

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

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

We recommend that the electrical cabinet is cleaned every 1000 operating hours .
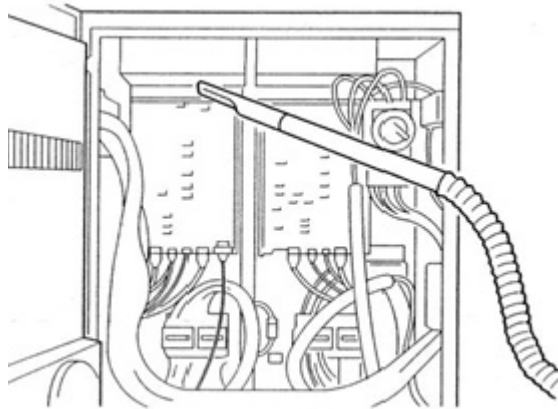
11.4 Interlock switch sliding door

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

Unlocking
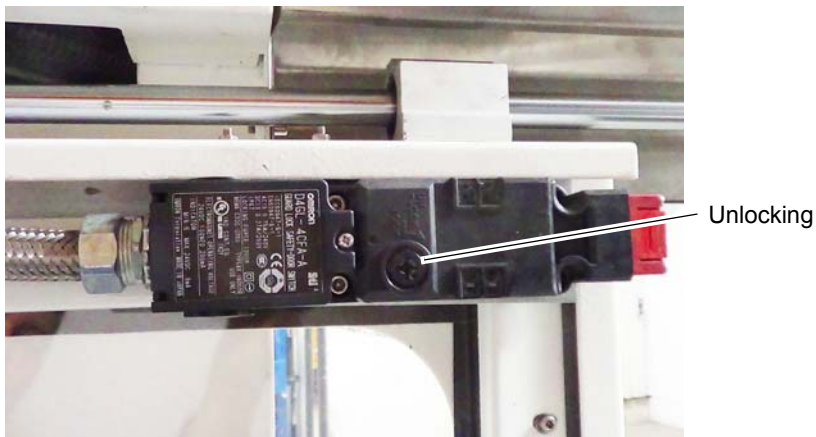
Img.11-1:   Interlock switch sliding door

11.4.1 Drain the compressed air service unit

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

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

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

Drainage

Img.11-2: Compressed air service unit

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 filing 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/1) or nitrate content to more than 50 ppm (mg/1)
❍ 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 (DIN 51369) Local measurement method: with pH paper (Special indicators with suitable measuring range)	weekly [1]	if pH value decreases > 0.5 based on initial filing: Measures in accordance manufacturer's recommendations > 1.0 based on initial filing: Replace cooling lubricant, clean cooling lubricant circulation system
Usage concentration	Manual refractometer	weekly [1]	Method results in incorrect values with tramp oil content
Base reserve	Acid titration in accordance with Manufacturer's recommendation	as required	Method is independent of tramp oil content
Nitrite content	Test sticks method or laboratory method	weekly [1]	> 20 mg/L nitrite: Replace cooling lubricant or part or inhibiting additives; otherwise NDELA (N-nitrosodiethanolamine) in the cooling lubricant system and in the air must be determined > 5 mg/L NDELA in the cooling lubricant system: Replacement, clean and disinfect cooling lubricant circulation system, find nitrite source and, if possible, rectify.
Nitrate/nitrite content of the preparation water, if this is not removed from the public grid	Test sticks method or laboratory method	as required	Use water from the public grid if there is water from the pubic grid has > 50 mg/l nitrate: Inform the waterworks

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 aftermarket 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)

External Power	Servo Drive Power	Main Motor Power	Coolant Pump	ATC CW	ATC CCW	Short Circuit Protection

1L1
1L2
1L3

QF0
5SJ6 3207

QF1
5SJ6 3108

4mm2black

2.5mm2black

R S T

1L11 2L11 3L11

400V
230V TC1

1L 2L 3L

PE

QS
20A

1L 2L 3L

QF2
6-10A
10A

2.5mm2black

R1 S1 T1

QF3
1.6-2.4A
2.4A

3L1 3L2 3L3

1.5mm2black

KM1

3*1.0+1.0mm2

U1 V1 W1

M
3~

QF4
0.24-0.4A
0.36A

4L1 4L2 4L3

1.5mm2black

KM2   KM3

3*1.0+1.0mm2

U2 V2 W2

M
3~

L1L2L3
3PH

1.5mm2black

QF5
5SJ6 2038

5L1
5L3

1.5mm2black

System Wiring Diagram (2)

	Fan/Lubricate	24V Power	

QF6
5SJ6 1038

TC2

1
AC230
0

5L1
AC400
5L3

2

(SP FAN)

2A
M4

2B
M5

2C
M6

2D
M7

M
~
M8

PE

AC220
DC24

U1

1.5mm2black

24P

0.5mm2blue

QF7
5SJ6 1047

QF8
5SJ6 1027

QF9
5SJ6 1047

24N   24P1   24P2   24P3

Model G0876 (Mfd. Since 12/22)

System Wiring Diagram (3)

		CNC ON			Alarm Light	Release Tool	ATC FWD	ATC REV			Lamp		Coolant	ATC CW	ATC CCW

24P1 ⟩— SA4 3 24P2 ⟩ 0.5mm2blue 24P3 ⟩ 3PH (11) (14) 5

E-- SB0 KA0 KA0 24V
4
808D X1

K1 R K2 Y K3 G K5 1DT K6 2DT K7 3DT

K8 HL K4 6 KA1 7 KM3 8 KA2 9 KM2 10

24N ⟩
KA0 HL1 red yellow green HL2 KM1 KM2 KM3

N0 | NC

System Wiring Diagram (4)

	EMG	X+ Limit	X- Limit	Y+ Limit	Y- Limit	Z+ Limit	Z- Limit	X 0 Slow		Y 0 Slow	Z 0 Slow	Check Tool-No	ATC Forward	Loose Tool	Lock Tool	TC REF			Safety Door	Master Alarm	Release Tool	Pump Overload	
													ATC Backward						Orientation Ready				

24P1 ⟩
24N ⟩

0.5mm2blue

SB5
E2 SB4
E1 SB3

EMG X+ X− Y+ Y− Z+ Z− XD YD ZD TPP ATCT ATC0 SPL SPC TP0

E-- SA1 D2
E-- SA2 D1 K10 K11 E-- SB2 QF3 (21) (24) 3PH
E-- SA3
DOOR 11 12 CLP 13 14

1	2	3	4	5	6	7	8	9	10

X100

1	2	3	4	5	6	7	8	9	10

X101

1	2	3	4	5	6	7	8	9	10

X102

STOP *READ ELECTRICAL SAFETY ON PAGE 253!*

System Wiring Diagram (5)

	Light			Cooling	Orientation Ready	ATC CW	ATC CCW	ATC FWD	ATC REV	Release Tool			

808D CNC

X200

1	2	3	4	5	6	7	8	9	10

X201

1	2	3	4	5	6	7	8	9	10

X21

1	2	3	4	5	6	7	8	9	10

0.5mm2blue

18 20 22 23 24
KA2 KA1

15 16 17 19 21
K8 K4 K9 KA1 KA2 K6 K7 K5

24P1

24N

N0 NC   N0 NC   N0 NC   N0 NC N0 NC N0 NC N0 NC N0 NC

System Wiring Diagram (6)

CN1

R1   R
S1   S
T1   T

+
R   PB

M1   U
3~   V
     W

Main Motor Drive

CN2
5V 12
0V 13
A+ 8
A- 3
B+ 9
B- 4
Z+ 15
Z- 14

G

ENCODER

PE PE

22   15 A+
7   14 A-
21   13 B+
6   12 B-   X60
36   11 Z+
35   10 Z-

808D CNC

16   AI1   A0
1   AGND   AGND

                SE1   X54
24P3   SE2

27 XC   24N
11 X1
41 X3   X3 K9
10 X4
15 TA1   24N   K10
44 TC1
14 TA2   24N
43 TC2   TC2   K11   24P3

Handwheel Wiring Diagram

			Red	Yellow	Green	

X10

1	2	3	4	5	6	7	8	9	10

808D CNC

X301

1	2	11	12	13	14	15	16	17	18	31	32	33	47	48	49	50

24N 24P1

27 28 29 24P1

K1 K2 K3 24N

Orange Purple Yellow Purple / Black Red Black

0.5mm2blue

24P1 24N 24P1 PE

Yellow / Black Blue / Black White Shield Brown Yellow / White Blue Blue / White Gray Gray / Black Brown / White White / Black Black / White

Air Pressure

X301

	19		34	35	36

P

24P1

A	– A	B	– B	5V	0V

24V /COM	0V /LED	24V /LED	PE	X	Y	Z	4	X1	X10	X100	

HANDWHEEL

NO	NC	NO	NC	NO	NC

X/Y/Z Motors Wiring Diagram

X Drive

PE	PE
R	L1
S	L2
T	L3
	U
	V
	W
	PE

MX 3～ G

X9

X6

24P1	24V
24N	M
PE	PE

X10	X11

Y Drive

PE	PE
R	L1
S	L2
T	L3
	U
	V
	W
	PE

MY 3～ G

X9

X6

24P1	24V
24N	M
PE	PE

X10	X11

Z Drive

PE	PE
R	L1
S	L2
T	L3
	U
	V
	W
	PE

MZ 3～ G

X9

X7

B+	B+
B-	B-

X6

24P1	24V
24N	M
PE	PE

X10	X11

X126

808D CNC

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.

Housing

Model G0876 (Mfd. Since 12/22)
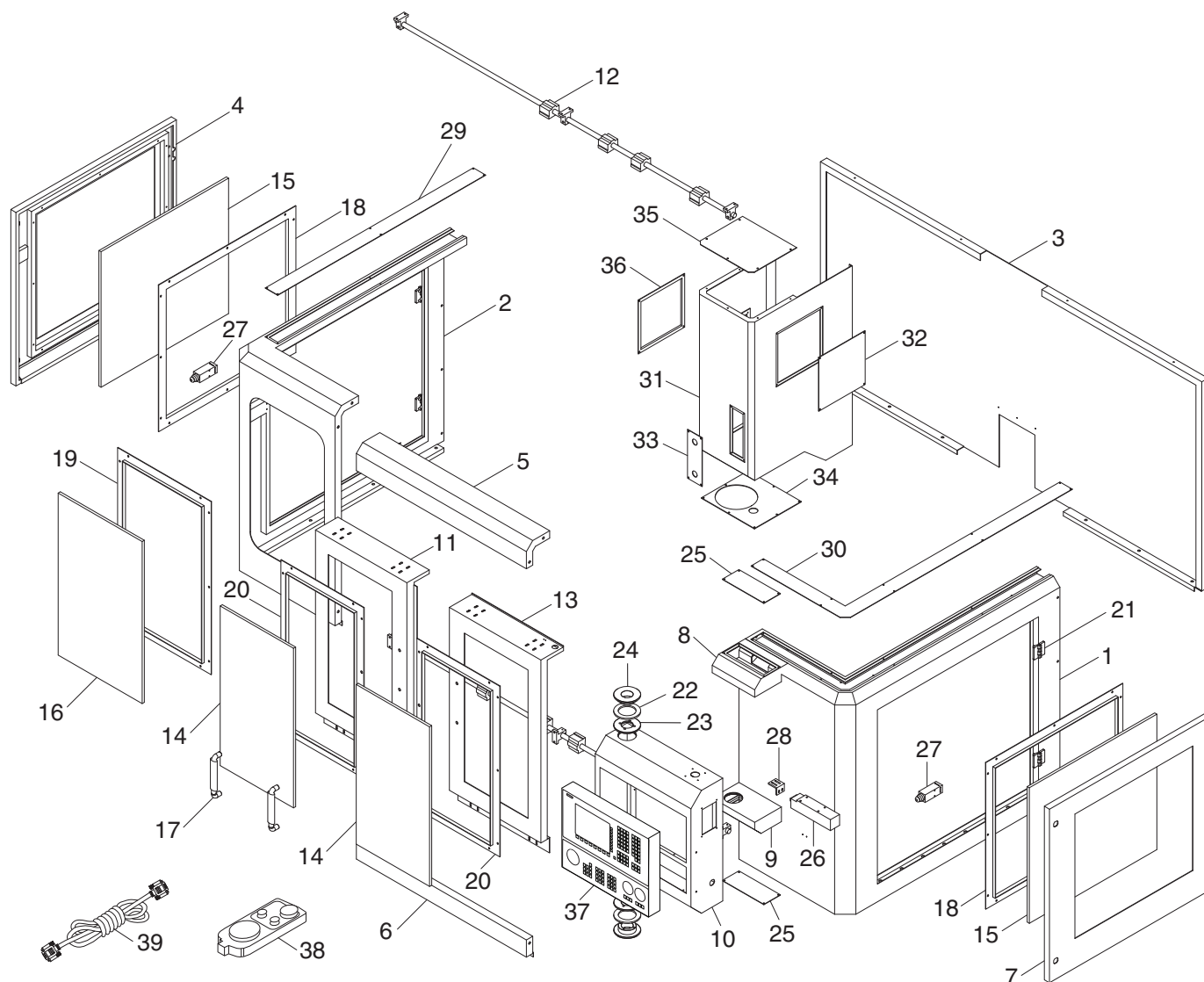
Housing Parts List

REF	PART #	DESCRIPTION
1	P08760001	FRAME CORNER (RIGHT FRONT)
2	P08760002	FRAME CORNER (LEFT FRONT)
3	P08760003	FRAME (REAR)
4	P08760004	SIDE DOOR (LEFT)
5	P08760005	UPPER DOOR FRAME (FRONT)
6	P08760006	LOWER DOOR FRAME (FRONT)
7	P08760007	SIDE DOOR (RIGHT)
8	P08760008	UPPER SUPPORT BRACKET
9	P08760009	LOWER SUPPORT BRACKET
10	P08760010	CONTROL PANEL CABINET
11	P08760011	SLIDING DOOR (LEFT)
12	P08760012	SLIDING TRACK
13	P08760013	SLIDING DOOR (RIGHT)
14	P08760014	SLIDING DOOR WINDOW
15	P08760015	DOOR WINDOW (SIDE)
16	P08760016	WINDOW (FRONT LEFT)
17	P08760017	DOOR HANDLE
18	P08760018	DOOR WINDOW SUB-FRAME (SIDE)
19	P08760019	WINDOW SUB-FRAME (FRONT LEFT)
20	P08760020	SLIDING DOOR SUB-FRAME

REF	PART #	DESCRIPTION
21	P08760021	DOOR HINGE HL009
22	P08760022	FLANGE #7
23	P08760023	FLANGE #6
24	P08760024	FLANGE #4
25	P08760025	SUPPORT COVER
26	P08760026	DOOR SWITCH
27	P08760027	LIMIT SWITCH QKS8
28	P08760028	LATCH SUPPORT ZL103
29	P08760029	FRAME COVER (LEFT)
30	P08760030	FRAME COVER (RIGHT)
31	P08760031	SPINDLE MOTOR HOUSING
32	P08760032	HOUSING COVER (RIGHT)
33	P08760033	HOUSING COVER (FRONT)
34	P08760034	HOUSING COVER (BOTTOM)
35	P08760035	HOUSING COVER (TOP)
36	P08760036	HOUSING COVER (LEFT)
37	P08760037	CONTROLLER PANEL SIEMENS SINUMERIK
38	P08760038	HANDHELD CONTROLLER PENDANT RICHAUTO
39	P08760039	CONTROLLER PANEL CORD

Model G0876 (Mfd. Since 12/22)

BUY PARTS ONLINE AT GRIZZLY.COM!
Scan QR code to visit our Parts Store.

-259-
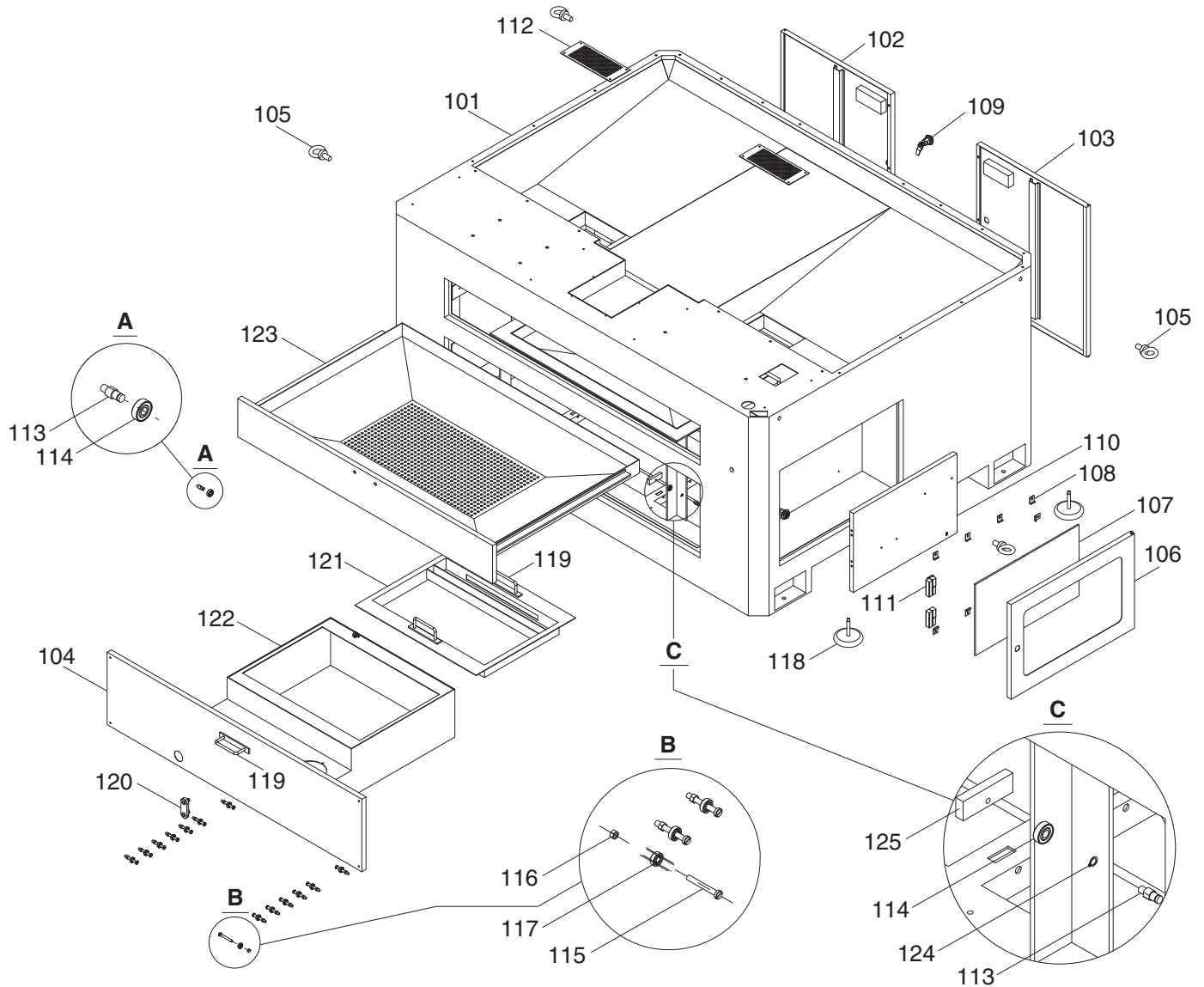
Base

REF	PART #	DESCRIPTION
101	P08760101	BASE
102	P08760102	FRONT DOOR (RIGHT)
103	P08760103	FRONT DOOR (LEFT)
104	P08760104	COOLANT TANK COVER
105	P08760105	EYE BOLT M16-2 X 28
106	P08760106	SIDE DOOR
107	P08760107	WINDOW
108	P08760108	WINDOW CLIPS
109	P08760109	DOOR LOCK MS718
110	P08760110	LUBRICANT PUMP MOUNTING PLATE
111	P08760111	HINGE HL019
112	P08760112	SCREEN
113	P08760113	SHAFT

REF	PART #	DESCRIPTION
114	P08760114	BALL BEARING 607ZZ
115	P08760115	DRAWER GUIDE SHAFT
116	P08760116	HEX NUT
117	P08760117	BALL BEARING 607ZZ
118	P08760118	ADJUSTABLE FOOT M12-1.75 X 100
119	P08760119	DRAWER HANDLE LS506
120	P08760120	OIL LEVEL SCALE
121	P08760121	COOLANT TANK FILTER
122	P08760122	COOLANT TANK
123	P08760123	CHIP DRAWER
124	P08760124	EXT RETAINING RING 7MM
125	P08760125	BEARING SUPPORT

Model G0876 (Mfd. Since 12/22)
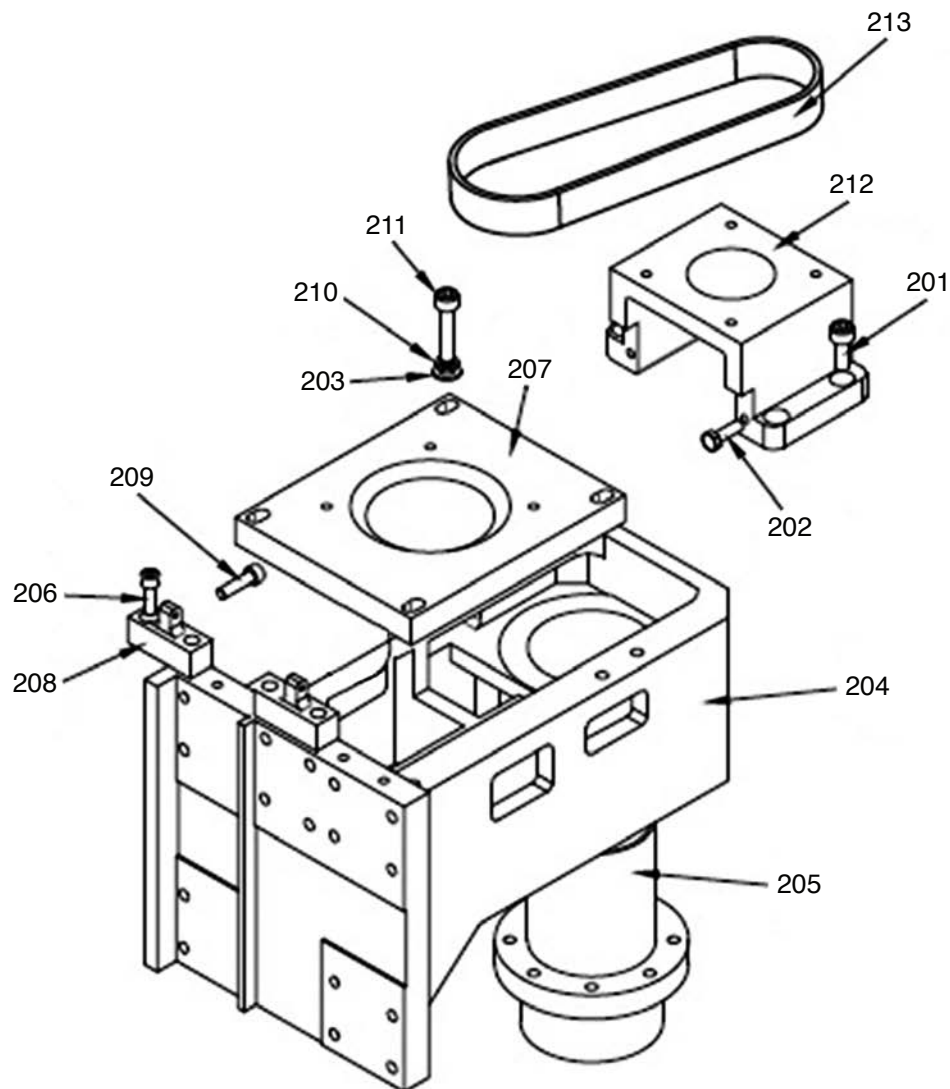
Milling Head

REF	PART #	DESCRIPTION
201	P08760201	CAP SCREW M10-1.5 X 25
202	P08760202	HEX BOLT M8-1.25 X 25
203	P08760203	FLAT WASHER 10MM
204	P08760204	MILL HEAD
205	P08760205	MILLING SPINDLE
206	P08760206	CAP SCREW M8-1.25 X 25
207	P08760207	MOUNTING PLATE

REF	PART #	DESCRIPTION
208	P08760208	MOUNTING BLOCK
209	P08760209	CAP SCREW M8-1.25 X 30
210	P08760210	LOCK WASHER 10MM
211	P08760211	CAP SCREW M10-1.5 X 45
212	P08760212	MOTOR MOUNT
213	P08760213	BELT

Model G0876 (Mfd. Since 12/22)

BUY PARTS ONLINE AT GRIZZLY.COM!
Scan QR code to visit our Parts Store.

-261-
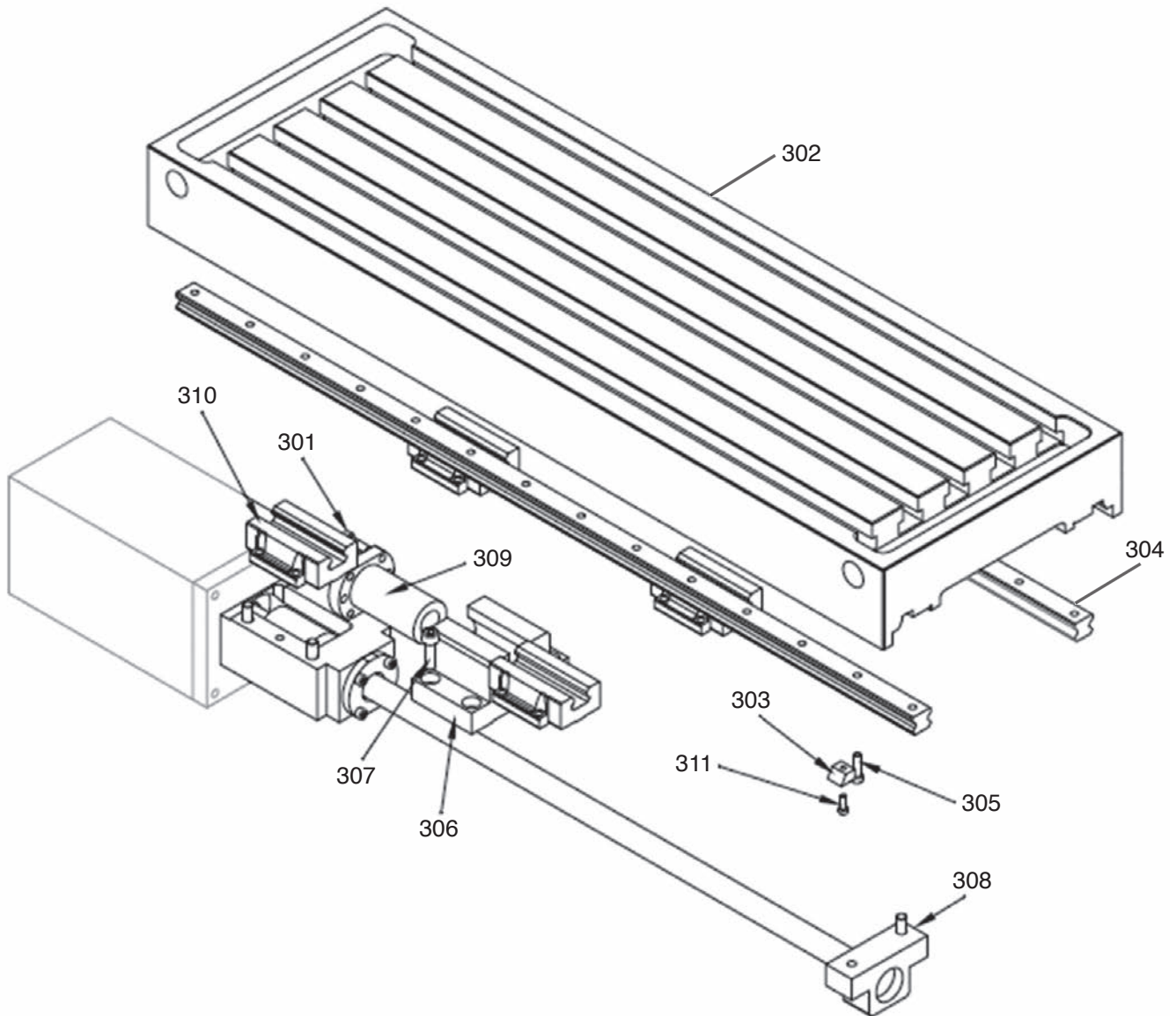
Milling Table

REF	PART #	DESCRIPTION
301	P08760301	CAP SCREW M6-1 X 20
302	P08760302	MILLING TABLE
303	P08760303	PLATE
304	P08760304	LINEAR GUIDE
305	P08760305	CAP SCREW M5-.8 X 20
306	P08760306	MOUNTING PLATE

REF	PART #	DESCRIPTION
307	P08760307	CAP SCREW M8-1.25 X 25
308	P08760308	LEADSCREW BLOCK
309	P08760309	SPINDLE NUT
310	P08760310	LINEAR GUIDE SLIDE
311	P08760311	CAP SCREW M4-.7 X 12

X-Axis
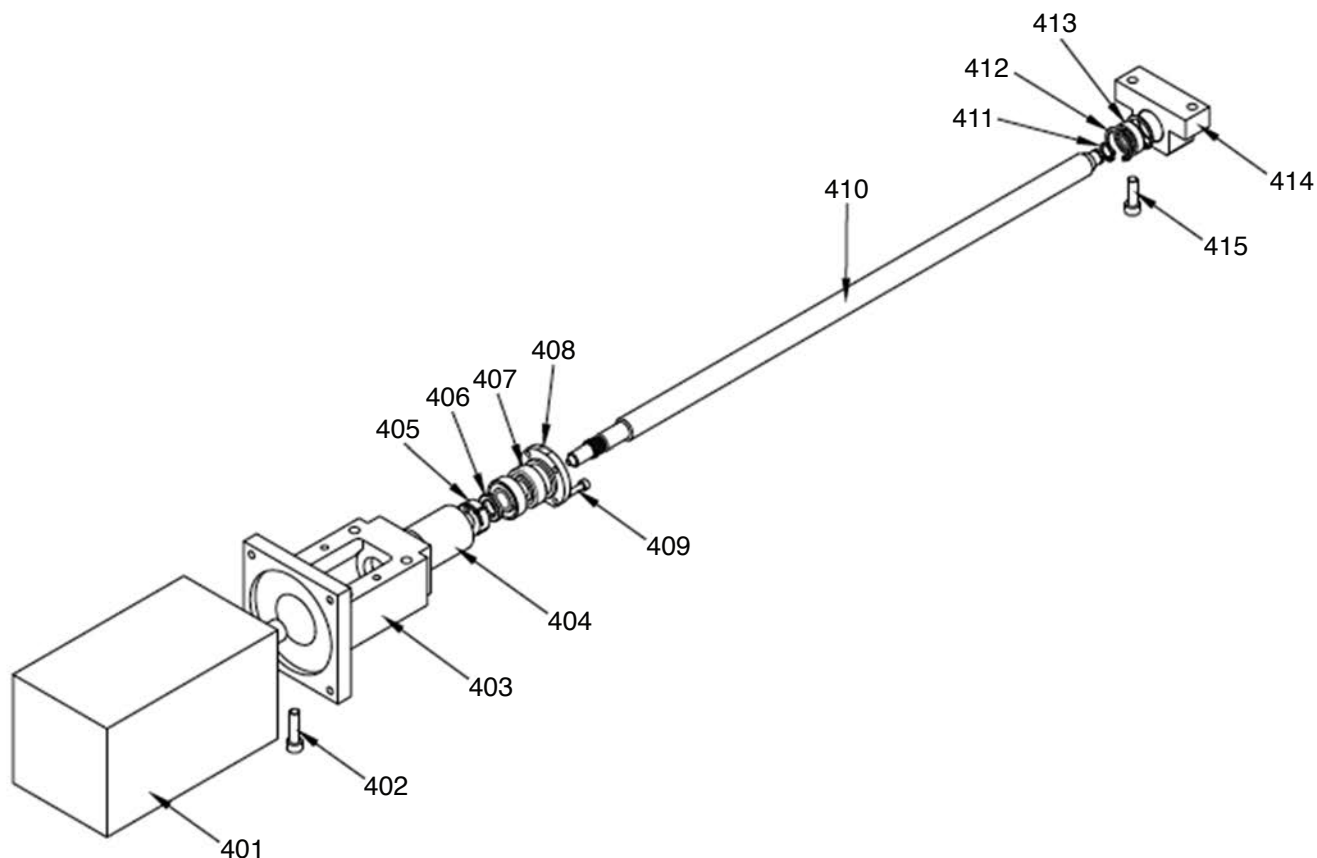
REF	PART #	DESCRIPTION
401	P08760401	X-AXIS SERVO MOTOR
402	P08760402	CAP SCREW M8-1.25 X 30
403	P08760403	BEARING BLOCK
404	P08760404	CLUTCH
405	P08760405	CLAMPING NUT
406	P08760406	EXT RETAINING RING
407	P08760407	ANGULAR CONTACT BEARING 7002
408	P08760408	MOUNTING FLANGE

REF	PART #	DESCRIPTION
409	P08760409	CAP SCREW M5-.8 X 16
410	P08760410	X-AXIS LEADSCREW
411	P08760411	EXT RETAINING RING 12MM
412	P08760412	EXT RETAINING RING 28MM
413	P08760413	BALL BEARING 6001-2RS
414	P08760414	BEARING BLOCK
415	P08760415	CAP SCREW M8-1.25 X 25

Y-Axis
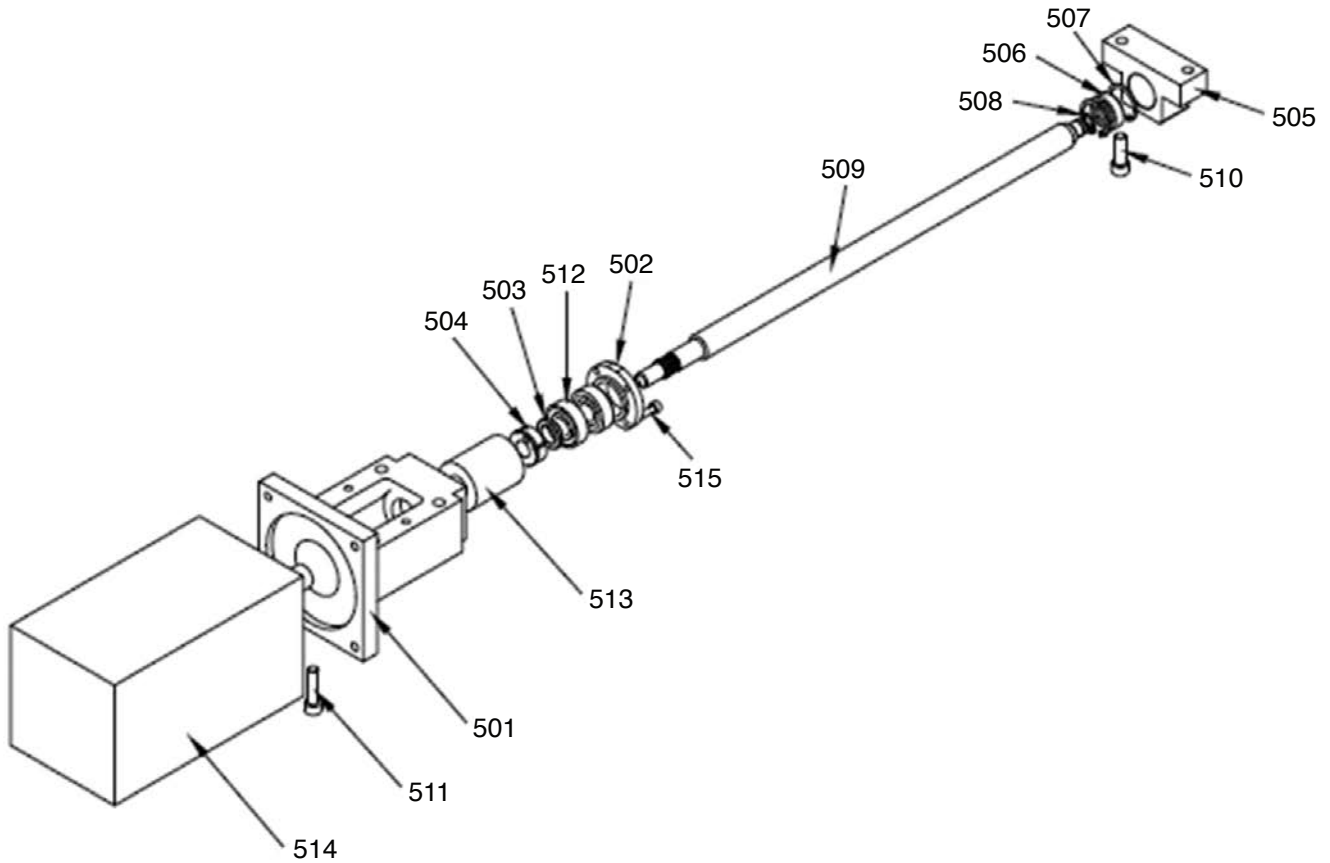
REF	PART #	DESCRIPTION
501	P08760501	BEARING BLOCK
502	P08760502	MOUNTING FLANGE
503	P08760503	EXT RETAINING RING
504	P08760504	CLAMPING NUT
505	P08760505	BEARING BLOCK
506	P08760506	BALL BEARING 6001-2RS
507	P08760507	EXT RETAINING RING 28MM
508	P08760508	EXT RETAINING RING 12MM

REF	PART #	DESCRIPTION
509	P08760509	Y-AXIS LEADSCREW
510	P08760510	CAP SCREW M8-1.25 X 25
511	P08760511	CAP SCREW M8-1.25 X 30
512	P08760512	ANGULAR CONTACT BEARING 7202
513	P08760513	CLUTCH
514	P08760514	Y-AXIS SERVO MOTOR
515	P08760515	CAP SCREW M5-.8 X 16

Z-Axis
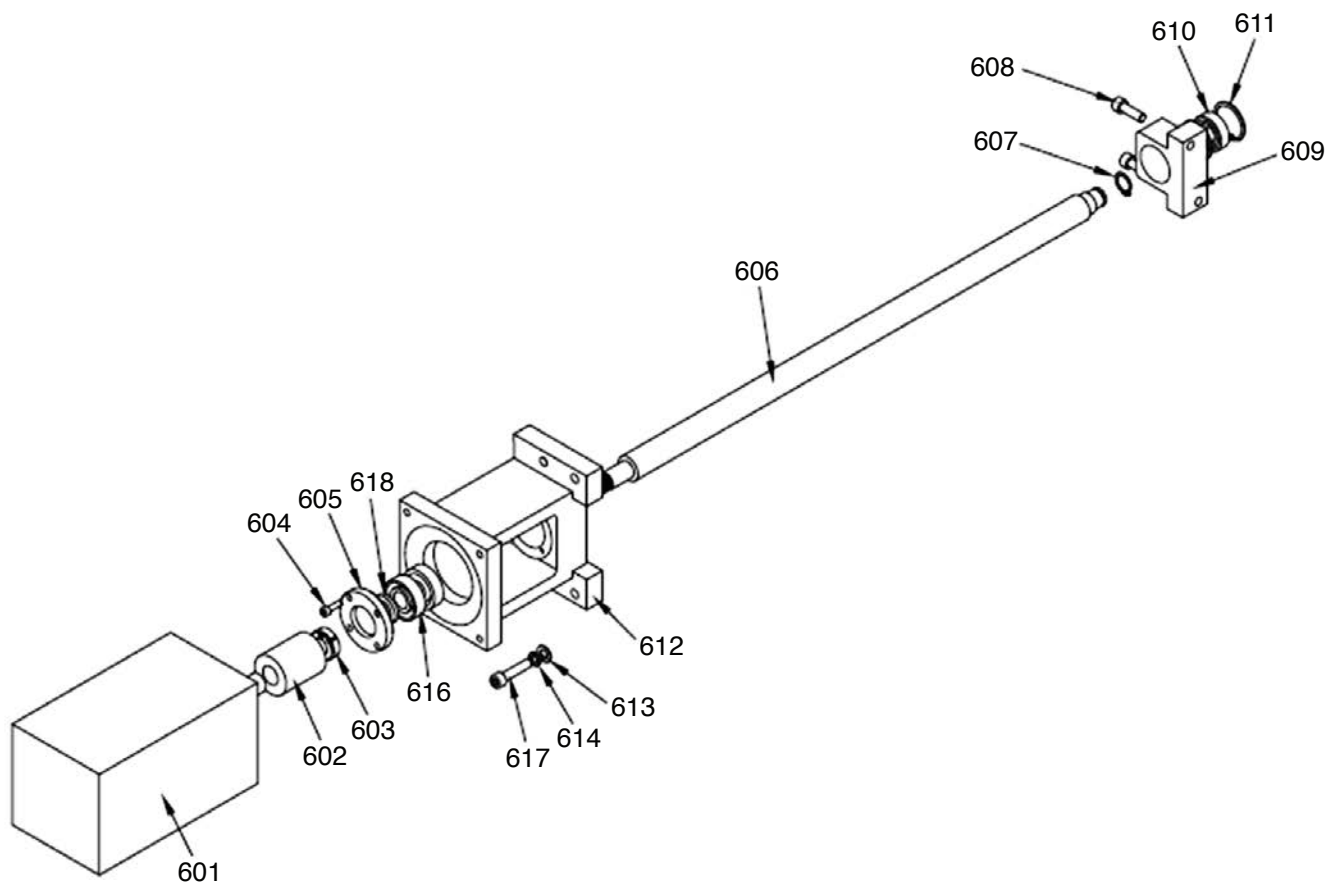
REF	PART #	DESCRIPTION
601	P08760601	Z-AXIS SERVO MOTOR
602	P08760602	CLUTCH
603	P08760603	CLAMPING NUT
604	P08760604	CAP SCREW M5-.8 X 16
605	P08760605	MOUNTING FLANGE
606	P08760606	Z-AXIS LEADSCREW
607	P08760607	EXT RETAINING RING 17MM
608	P08760608	CAP SCREW M8-1.25 X 25
609	P08760609	BEARING BLOCK

REF	PART #	DESCRIPTION
610	P08760610	BALL BEARING 6003-2RS
611	P08760611	EXT RETAINING RING 35MM
612	P08760612	BEARING BLOCK
613	P08760613	FLAT WASHER 8MM
614	P08760614	LOCK WASHER 8MM
616	P08760616	ANGULAR CONTACT BEARING 7203
617	P08760617	CAP SCREW M8-1.25 X 35
618	P08760618	SPACER

Model G0876 (Mfd. Since 12/22)

BUY PARTS ONLINE AT GRIZZLY.COM!
Scan QR code to visit our Parts Store.

-265-

Column

Model G0876 (Mfd. Since 12/22)
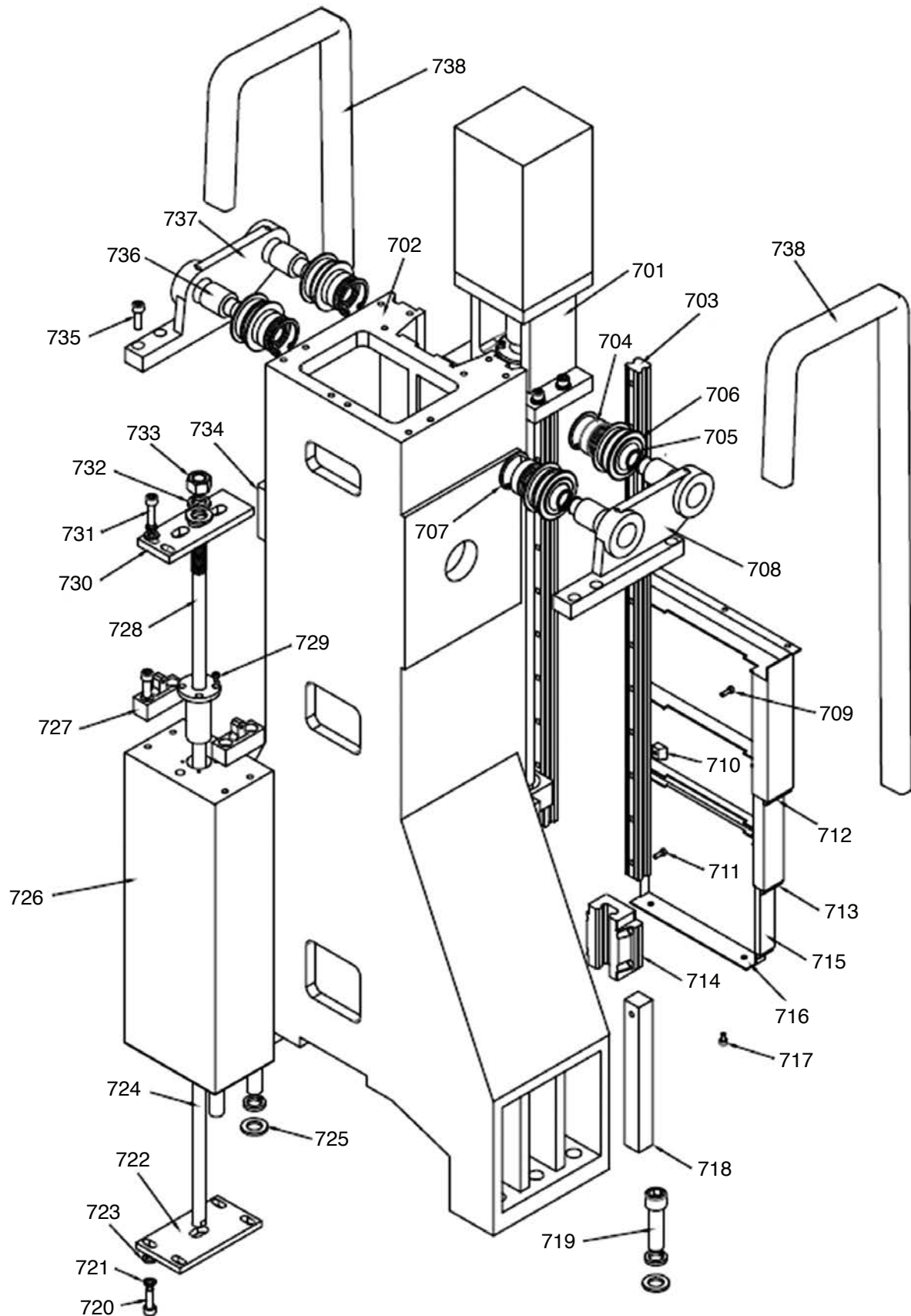
Column Parts List

REF	PART #	DESCRIPTION	REF	PART #	DESCRIPTION
701	P08760701	BEARING BLOCK	720	P08760720	CAP SCREW M8-1.25 X 12
702	P08760702	COLUMN	721	P08760721	LOCK WASHER 8MM
703	P08760703	LINEAR GUIDE	722	P08760722	PLATE
704	P08760704	BALL BEARING 6004-2RS	723	P08760723	FLAT WASHER 8MM
705	P08760705	RETAINING RING	724	P08760724	GUIDE
706	P08760706	GUIDE ROLLER	725	P08760725	FLAT WASHER 16MM
707	P08760707	EXT RETAINING RING 42MM	726	P08760726	BALANCE WEIGHT
708	P08760708	ROLLER MOUNT	727	P08760727	BLOCK
709	P08760709	CAP SCREW M4-.7 X 12	728	P08760728	SHAFT
710	P08760710	PLATE	729	P08760729	CAP SCREW M4-.7 X 6
711	P08760711	CAP SCREW M6-1 X 25	730	P08760730	PLATE
712	P08760712	COVER A	731	P08760731	CAP SCREW M8-1.25 X 30
713	P08760713	COVER B	732	P08760732	LOCK WASHER 16MM
714	P08760714	LINEAR GUIDE SLIDE	733	P08760733	HEX NUT M16-2
715	P08760715	COVER C	734	P08760734	BLOCK
716	P08760716	PLATE	735	P08760735	CAP SCREW M8-1.25 X 25
717	P08760717	CAP SCREW M5-.8 X 8	736	P08760736	SHAFT
718	P08760718	PLATE	737	P08760737	HOLDER
719	P08760719	CAP SCREW M16-2 X 60	738	P08760738	CABLE CARRIER

Model G0876 (Mfd. Since 12/22)

BUY PARTS ONLINE AT GRIZZLY.COM!
Scan QR code to visit our Parts Store.

-267-

Auto Tool Changer
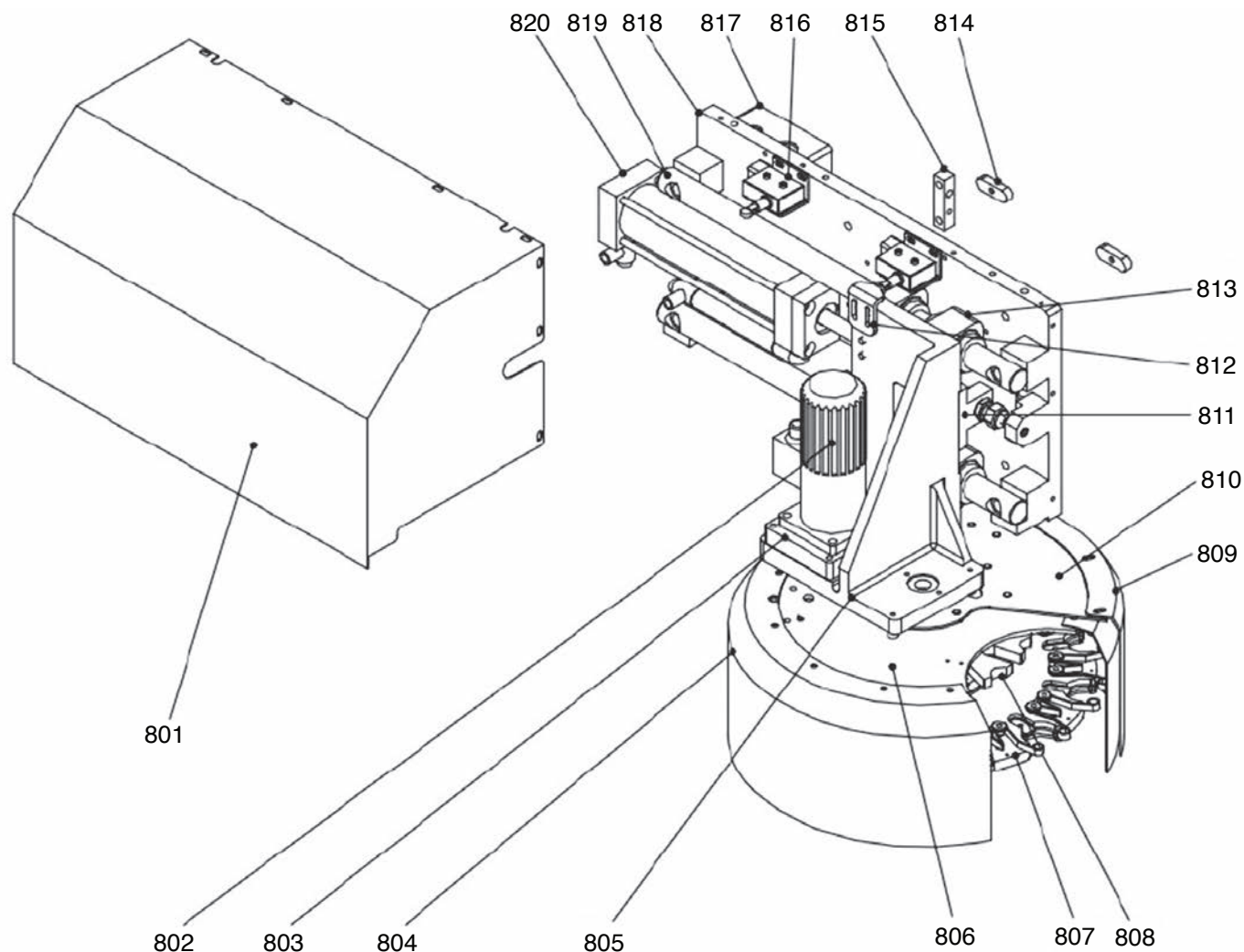
REF	PART #	DESCRIPTION
801	P08760801	ATC COVER
802	P08760802	ATC MOTOR
803	P08760803	ATC MOTOR MOUNTING PLATE
804	P08760804	ATC COVER
805	P08760805	ATC MOUNTING BLOCK
806	P08760806	ATC PLATE
807	P08760807	ATC TOOL HOLDER
808	P08760808	PLATE
809	P08760809	COVER
810	P08760810	PLATE

REF	PART #	DESCRIPTION
811	P08760811	GUIDE
812	P08760812	LIMIT STOP
813	P08760813	GUIDE
814	P08760814	PLATE
815	P08760815	BLOCK
816	P08760816	LIMIT SWITCH
817	P08760817	MANIFOLD
818	P08760818	BASE PLATE
819	P08760819	GUIDE ROD
820	P08760820	PNEUMATIC CYLINDER

Model G0876 (Mfd. Since 12/22)

Electrical Cabinet
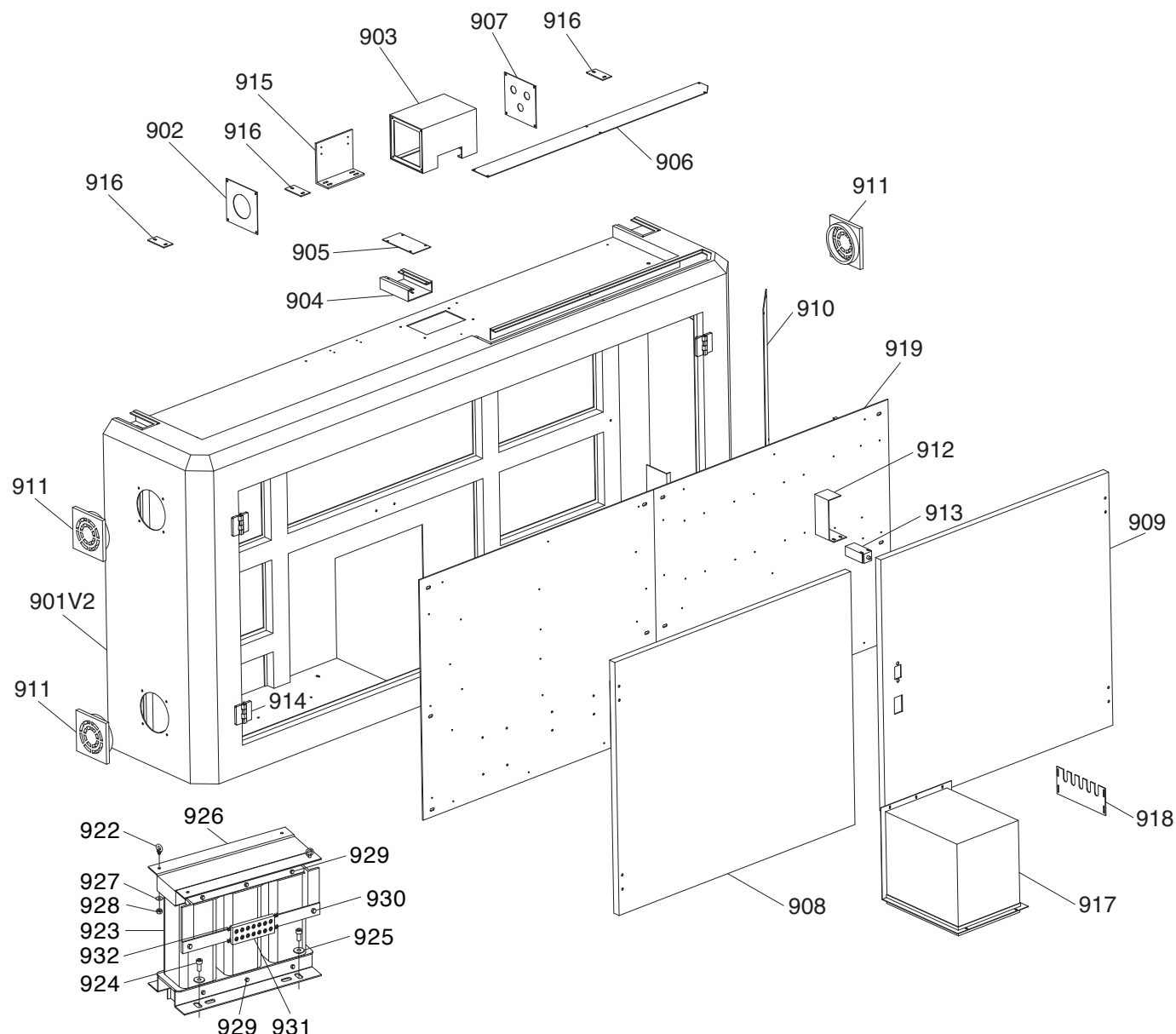
REF	PART #	DESCRIPTION
901V2	P08760901V2	ELECTRICAL CABINET V2.12.22
902	P08760902	FACEPLATE (LEFT)
903	P08760903	ROUTING BOX
904	P08760904	MOUNTING PLATE
905	P08760905	COVER
906	P08760906	CABINET COVER
907	P08760907	FACEPLATE (RIGHT)
908	P08760908	DOOR (LEFT)
909	P08760909	DOOR (RIGHT)
910	P08760910	SIDE COVER
911	P08760911	FAN
912	P08760912	SWITCH SUPPORT
913	P08760913	DOOR LOCK QKS7
914	P08760914	DOOR HINGE HL009
915	P08760915	CABLE CARRIER MOUNT

REF	PART #	DESCRIPTION
916	P08760916	DEAD PLATE
917	P08760917	Y-AXIS MOTOR HOUSING
918	P08760918	COVER
919	P08760919	ELECTRICAL BOARD ASSEMBLY
922	P08760922	EYE BOLT M10-1.5 X 20
923	P08760923	TRANSFORMER LUHO JB/T8750 230V-400V
924	P08760924	CAP SCREW M8-1.25 X 20
925	P08760925	FLAT WASHER 8MM
926	P08760926	STEEL FRAME
927	P08760927	FLAT WASHER 10MM
928	P08760928	HEX NUT M10-1.5
929	P08760929	FLANGE BOLT M8-1.25 X 105
930	P08760930	HEX BOLT M6-1 X 15
931	P08760931	TERMINAL BAR 14P
932	P08760932	CLEAT W/SCREW

Model G0876 (Mfd. Since 12/22)

BUY PARTS ONLINE AT GRIZZLY.COM!
Scan QR code to visit our Parts Store.

-269-

Pneumatic System
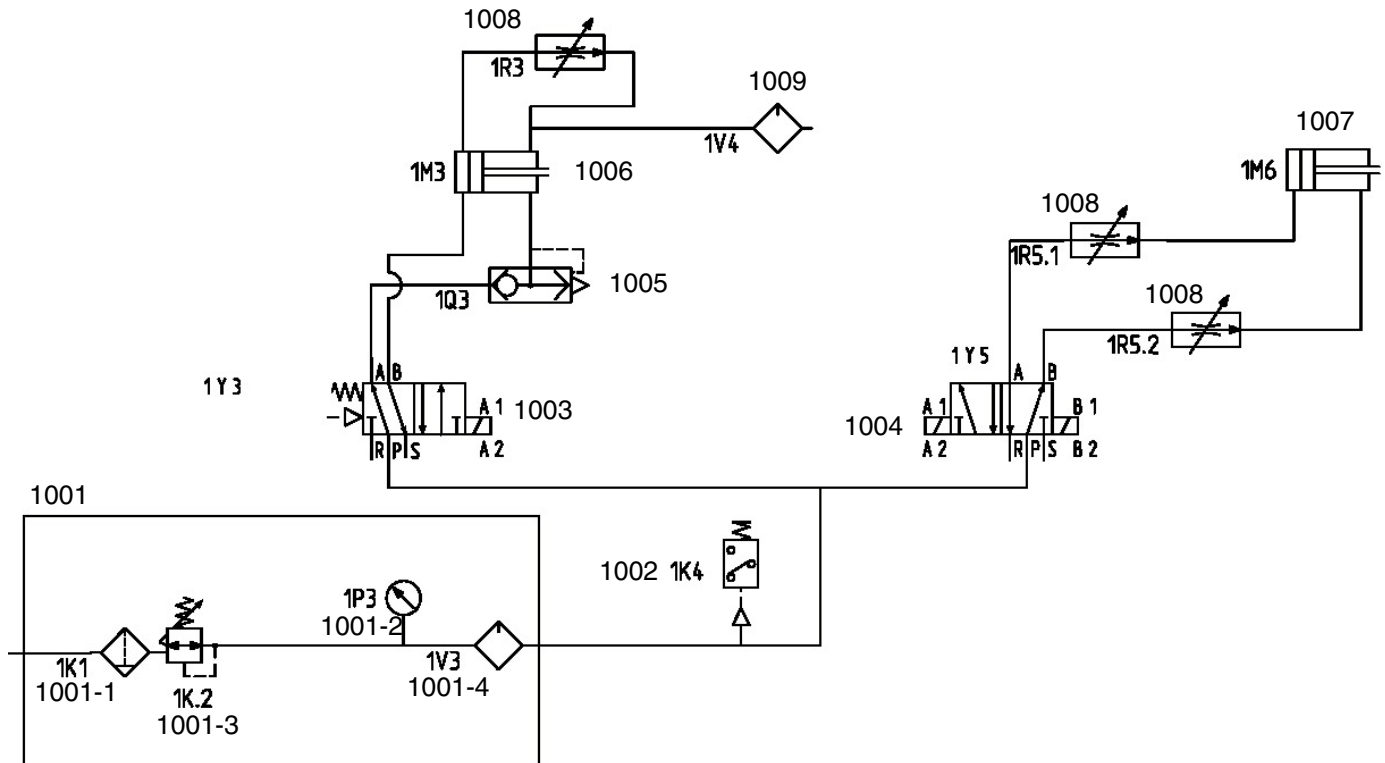
REF	PART #	DESCRIPTION
1001	P08761001	COMPRESSED AIR ASSEMBLY
1001-1	P08761001-1	SINTERED METAL FILTER
1001-2	P08761001-2	MANOMETER
1001-3	P08761001-3	PRESSURE REGULATOR
1001-4	P08761001-4	LUBRICATOR
1002	P08761002	PRESSURE SWITCH
1003	P08761003	WAY VALVE

REF	PART #	DESCRIPTION
1004	P08761004	WAY VALVE
1005	P08761005	QUICK EXHAUST VALVE
1006	P08761006	CYLINDER TOOL SPANNER
1007	P08761007	CYLINDER TOOL CHANGER
1008	P08761008	THROTTLE VALVE
1009	P08761009	DRIP LUBRICATOR

Labels & Cosmetics

1116
1115
1114
1113
1112

1101

1102
240V

1101

1103

1101

1104

1101

1105

1106
Required
| PSI | MPa |
| 90 | 0.6 |

1101

1101

1101

1101

1101

1101

G0876

grizzly.com

1110

1111

Grizzly Industrial
MODEL G0876
8" x 27" ENCLOSED CNC MILL

Specifications

Spindle Motor: 3 HP, 220V, 3-Phase, 60 Hz
X-Axis Motor: 1 HP, 220V, 3-Phase, 60 Hz
Y-Axis Motor: 1 HP, 220V, 3-Phase, 60 Hz
Z-Axis Motor: 1.2 HP, 220V, 3-Phase, 60 Hz
Full-Load Amp Draw: 13.3A
Longitudinal Table Travel (X): 12-3/16"
Cross Table Travel (Y): 7-3/4"
Vertical Head Travel (Z): 11-3/4"
Max. Distance Spindle to Column: 11"
Max. Distance Spindle to Table: 16-1/2"
End Milling Capacity: 1"
Face Milling Capacity: 2-1/2"
Rapid Feed Rate (X,Y,Z): 400 IPM
Spindle Taper: BT30
Spindle Speeds: Variable, 50–10,000 RPM
Weight: 3080 lbs.

WARNING!
To reduce the risk of serious personal injury while using this machine:
1. Read and understand owner's manual before operating.
2. Always wear ANSI-approved safety glasses.
3. Properly ground machine—connect to permanently grounded metal wiring system or an equipment-grounding conductor.
4. Disconnect power before setting up, adjusting, or servicing.
5. DO NOT remove any electronic safety switches.
6. Avoid getting entangled in moving parts—tie back long hair, roll up long sleeves, and DO NOT wear loose clothing, gloves, or jewelry.
7. Keep all doors closed and covers in place during operation.
8. Always remove adjustment tools before starting.
9. Alway use a clamp or vise to secure workpiece to table during operation.
10. Keep hands away from spindle area when engaging pull stud/changing tools.
11. Never attempt to slow or stop spindle with hands or tools.
12. Always use proper feeds and speeds for operation and workplace.
13. Never leave machine running unattended.
14. Always use a brush or vacuum to remove metal chips; DO NOT use your hands, rags, or compressed air.
15. DO NOT operate when tired or under influence of drugs or alcohol.
16. DO NOT expose to rain or use in wet locations.
17. Prevent unauthorized use by children or untrained users; restrict access or disable machine when unattended.

1109

grizzly.com

1108

G0876

1107
Fluid Capacities
Component	Capacity	Type
Bearings	1/2 gal.	Grizzly #T23963 or ISO 32 equivalent
Coolant	13 gal.	Grizzly #H0245 or water-soluble equivalent

1101

1101

REF	PART #	DESCRIPTION
1101	P08761101	ELECTRICITY LABEL
1102	P08761102	240V ELECTRICITY LABEL
1103	P08761103	TOUCH-UP PAINT, GRIZZLY PUTTY
1104	P08761104	TOUCH-UP PAINT, GRIZZLY GREEN
1105	P08761105	DISCONNECT HARDWIRED LABEL
1106	P08761106	AIR REQUIREMENTS LABEL
1107	P08761107	FLUID CAPACITIES LABEL
1108	P08761108	MODEL NUMBER LABEL

REF	PART #	DESCRIPTION
1109	P08761109	GRIZZLY.COM LABEL
1110	P08761110	GRIZZLY NAMEPLATE-SMALL
1111	P08761111	MACHINE ID LABEL
1112	P08761112	PINCH/AMPUTATION HAZARD LABEL
1113	P08761113	BIOHAZARD LABEL
1114	P08761114	DO NOT OPEN DOOR LABEL
1115	P08761115	EYE INJURY HAZARD LABEL
1116	P08761116	READ MANUAL LABEL

⚠WARNING

Safety labels help reduce the risk of serious injury caused by machine hazards. If any label comes off or becomes unreadable, the owner of this machine MUST replace it in the original location before resuming operations. For replacements, contact (800) 523-4777 or www.grizzly.com.

Model G0876 (Mfd. Since 12/22)

BUY PARTS ONLINE AT GRIZZLY.COM!
Scan QR code to visit our Parts Store.

-271-

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/forms/warranty**, or you can scan the QR code below to be automatically directed to our warranty registration page. Enter all applicable information for the product.

WARRANTY

grizzly.com

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

VISA MasterCard DISCOVER NOVUS® AMERICAN EXPRESS®