

Makino High-speed Vertical and Horizontal Machining Centers with Makino Professional 3 Control

Programming Guide

This guide is intended for use by Makino customer, employees and authorized service technicians for the safe programming and operation of Makino equipment.

This guide was developed through the combined efforts of:

Makino Customer Support

TM & Associates

1st Release Date: May 1, 2002

Revision(s) Date:

Revision 1 - October 25, 2002: Corrected Chapter 9 - Rigid Tap programming and typos in Chapter 6 and 7.

Copyright © 2002 Makino, Inc. All Rights Reserved

- No part of this manual may be reproduced or transmitted by any means or in any form to parties other than which it is intended, without the expressed written permission of Makino.
- All specifications and designs may change without notice.

Every attempt was made to ensure that the information in this publication was correct at the time of printing. As specifications and designs change, Makino is not responsible for information that becomes incorrect or inaccurate after publication.

The following icons are used in this document. Safety and precautionary statements are presented as shown below.



A **DANGER** icon precedes each danger statement. A **DANGER** icon indicates an existing hazardous condition with a high probability of death or severe injury. Precautions must be taken immediately to prevent personal injury or death.



A **WARNING** icon precedes each warning statement. A **WARNING** icon indicates a potentially hazardous situation with some probability of death or serious injury. Precautions must be taken immediately to prevent personal injury.



A **CAUTION** icon precedes each caution statement. A **CAUTION** icon indicates a potentially hazardous situation with some probability of personal injury or damage to the machine. Precautions must be taken immediately to prevent personal injury or machine damage.

- Precautionary statements must be followed to prevent personal injury or damage to the machine.
- In addition, all safety requirements and guidelines found in Makino manuals and the ANSI safety guidelines shipped with the machine and established company safety requirements and regulations must be followed. Taking personal responsibility for safety will prevent most accidents.

Notes are presented using the following icon format:



A **NOTE** icon indicates information that expands on information given or indicates where additional information can be found. Notes are presented immediately following the applicable content.

A copy of the ANSI Safety Manual, ANSI document number ANSI B11.8, was shipped with the machine. If this manual was lost or misplaced, another copy can be obtained from Makino at no charge.

Write us at:
Makino
P.O. Box 8003
7680 Innovation Way
Mason Ohio, 45040-8003
Attn: Information Services

Call or FAX us at:
Phone: (513) 573-7200
ask for Information Services
FAX: (513) 573-7360
Attn: Information Services

To contact Makino, for service after the sale:

Makino	
7680 Innovation Way	
Mason, Ohio 45040 USA	
Technical Support Services	(888) Makino4 (625-4664)
Mon.-Fri. 7:00 a.m. to 8:00 p.m.	Fax: (513) 459-1825
Sat. 8:00 a.m. to 2:00 p.m. (Eastern Standard Time)	
Parts Express	(888) Makino1 (625-4661)
24 hours, 7 days a week	Fax: (888)-881-9289
Training Services	(888) Makino1 (625-4661)

The following acronyms and abbreviations may appear in this document:

Term	Meaning
APC	Automatic Pallet Changer- work table (pallet) storage and exchange device
ATC	Automatic Tool Changer - tooling storage and exchange device
ccw	counterclockwise - direction of rotation
CNC	Computerized Numerical Control - control unit for machine tool system
cw	clockwise - direction of rotation
DEC	Deceleration - axis slow down for reference operations
EOB	End Of Block - program code at end of each block of data
E-Stop	Emergency Stop - machine condition halting all machine operation
FSE	Field Service Engineer - customer service technician
LS	Limit Switch - LS00 = limit switch 00
MDI	Manual Data Input - operating mode for direct input of NC data
MPG	Manual Pulse Generator - machine mounted or portable axis positioning device
MTC	Machine Tool Cabinet - Cabinet housing main machine electrics/electronics
NC	Numerical Control - input data (i.e. NC program)
PM	Preventive Maintenance - routine maintenance items and checks
PMC	Programmable Machine Controller - sub processor in machine tool system
PRO	Professional - (i.e. PRO 3) control model name and type
S/G	Splash Guard - machining area enclosure
SOL	Solenoid - SOL01 = Solenoid 01

AVISO!

Las personas que no pueden leer o entender en Ingles deben mandar traducir este manual y solicitar entrenamiento antes de operar o mantener la maquina. Todos los que trabajen en esta maquina deberan saber como operarla con toda seguridad y en forma correcta para evitar un posible dano.

ATTENTION!

Les personnes ne sachant pas lire l'anglais devront se faire lire et expliquer le manuel avant d'utiliser ou de faire l'entretien de la machine. Toute personne travaillant sur la machine doit savoir comment l'utiliser sans danger et correctement afin d'eviter tout accident.

HINWEIS

Alle Personen die nur Teile des Handbuches oder kein Englisch verstehen, müssen sich dieses Handbuch erklären lassen, bevor sie die Maschine in Betrieb nehmen oder instandhalten. Alle Personen, die an der Maschine arbeiten, müssen zur Vermeidung von Verletzungen zur korrekten Handhabung angeleitet werden.

WARNING!

Persons unable to read English or understand any part of this guide must have this guide read and explained to them before operating or maintaining the machine. Everyone working on the machine must know how to operate and maintain it safely and correctly to prevent possible serious injury.

Chapter 1	Safety Precautions
Chapter 2	Program Word Addresses
Chapter 3	Program Numbers - O Word
Chapter 4	Coordinate Systems
Chapter 5	Dimensional Input
Chapter 6	M, S, and T Functions
Chapter 7	Interpolation Functions
Chapter 8	Cutter Compensation C
Chapter 9	Canned Cycles
Chapter 10	Rigid Tapping
Chapter 11	Sub Programs
Chapter 12	Scaling
Chapter 13	Coordinate System Rotation
Chapter 14	Table Programming
Chapter 15	G and M Codes

Chapter 1 **Safety Precautions**

For a Safe Working Environment

Makino Vertical and Horizontal
Machining Centers



Chapter 1

Safety Precautions

Contents

1.1 Overview	1-1
1.2 General Safety	1-2
1.2.1 Danger, Warning, Caution Statements and Symbols	1-4
1.2.2 Access to Information	1-6
1.2.3 Point-of-operation	1-6
1.2.4 Lockout/Tagout Procedure	1-7
1.3 Personal and Professional Safety	1-8
1.4 Equipment and Operation Safety	1-10
1.5 Environmental Safety	1-13
1.6 Lifting Safety	1-14
1.6.1 Lifting Devices	1-14
1.6.1.1 Slings	1-14
1.6.1.2 Chains	1-14
1.6.1.3 Hooks	1-15
1.6.1.4 Eye Bolts	1-15
1.6.1.5 Rings	1-15
1.6.2 Lifting the Machine or Components	1-16
1.6.3 Lifting Workpieces and Fixtures	1-16
1.7 Mechanical Maintenance Safety	1-17
1.7.1 Mechanical Maintenance Equipment	1-17
1.8 Electrical Maintenance Safety	1-18
1.8.1 Electrical Maintenance Equipment	1-18
1.9 Machine Safety	1-19
1.9.1 Machine Safety Features	1-19
1.9.2 Location of Emergency Buttons	1-19
1.9.3 Location of Safety Labels	1-19
1.9.4 Pinch Points	1-20

- 1.10 Tooling Safety 1-21
 - 1.10.1 Standard Tooling Safety Precautions 1-21
 - 1.10.2 High-speed Machining Tooling Safety Precautions 1-22
- 1.11 Equipment and Operation Safety 1-23
 - 1.11.1 Working Inside the Machine 1-23
 - 1.11.2 Prior to Machining 1-23
 - 1.11.3 Workpiece Handling Safety 1-24
- 1.12 Coolant Safety 1-25
- 1.13 Installation Safety Instructions 1-26

1.1 Overview

This chapter provides proven general safety guidelines that, if followed, promote a safer working environment. It also contains safety information specific to this machine.

All machines have inherent hazards the manufacturer either designs out, guards against, or warns about. Even though the manufacturer has this responsibility, the machine user has ultimate responsibility to ensure that a safe working environment is maintained in and about the machine. This includes, but is not limited to, allowing only properly trained and qualified personnel to program, operate, and maintain the machine.

It is Makino's policy and responsibility to design, manufacture, and market machines that are reasonably safe for their intended use.

1.2 General Safety

Safety precautions or requirements discussed in Makino guides or manuals do not supersede any safety requirements imposed by local, state, or federal governments. All government-imposed safety guidelines (i.e. OSHA (Occupational Safety and Health Administration) and ANSI (American National Standard Institute)) shall take precedence.

To protect against injury or damage, operate the machine within the guidelines described in the provided documentation. Consider Makino's safety precautions supplemental to:

- Other Makino supplied guides or manuals related to programming, operation, and maintenance of this specific machine.
- Your company's safety rules and regulations.
- All local, state, and federal regulations.

Your particular application may require additional safety measures to ensure proper operator safety.

Additional copies of any guide or manual provided with your Makino machine may be purchased from Makino.

Prior To Installation, Operation, or Service

Do NOT attempt to install, operate, or service this machine until:

- You have read and understand the safety instructions in this chapter.
- You have read and understand all **Danger, Warning, and Caution** safety labels attached to the machine and its related equipment.
- You have read and understand all technical information provided.

Questions Regarding Safe Operation

Before you operate this machine - contact your Makino service group for answers to any questions about safety and your Makino machine.

1.2.1 Danger, Warning, Caution Statements and Symbols

It is important that all employees recognize each safety symbol on the machine and understand the meaning of **Danger**, **Warning**, and **Caution**.

These safety labels must never be removed, covered, or painted over, and must be replaced if they are damaged, defaced, or become unreadable.





Table 1-1 presents the meaning of each safety statement. It is important that all employees recognize and understand the meaning of the safety symbols shown in Table 1-2.

The location of machine safety labels is described in [section 1.9.3 \(pg 1-19\)](#).

TABLE 1-1 DEFINITION OF DANGER, WARNING, AND CAUTION STATEMENTS

Statement	Meaning
DANGER	DANGER is the most severe safety statement. This statement means that either severe personal injury or death may result if the instruction(s) is not followed.
WARNING	WARNING is the next level of safety statement. It indicates that if the instruction(s) is not followed severe personal injury could result.
CAUTION	A CAUTION is the least severe safety statement and indicates that if the instruction(s) is not followed damage to the equipment may result.

TABLE 1-2 SAFETY SYMBOL SHAPES AND THEIR MEANING

Symbol	Meaning
	<p>A triangle or diamond shaped safety symbol indicates a hazard exists in the area or behind the enclosure where the symbol is placed. The symbol at left indicates an electrical shock hazard exists.</p>
	<p>A solid colored circle shaped safety symbol with an icon inside indicates the action shown by the icon is mandatory. The symbol at left indicates that safety glasses are required.</p>
	<p>A square shaped safety symbol is used to convey information relative to the area where the symbol is placed. The symbol at left indicates the area is hot.</p>
	<p>A circle shaped safety symbol with a diagonal slash across the icon indicates the action is prohibited in or around that area. The symbol at left indicates that smoking is prohibited.</p>

1.2.2 Access to Information

We strongly recommend that a copy of these safety instructions and all guides, manuals, and technical information provided be kept near the machine. This information should be organized for quick access and use by all personnel with duties related to the machine.

Access to this information:

- May save a life.
- Will prevent or reduce injury, mis-operation, and machine damage.

In the event that parts of this information become damaged or unusable, replace them immediately.

1.2.3 Point-of-operation

“Point-of-operation” should be safeguarded to the greatest extent possible for all machining operations.

In addition to panel locks, interlocks, and guarding, certain areas of the machine are color coded for safety. It is important that the original color and vividness of these areas be maintained.

Different operations or applications may make the general purpose safeguards unsuitable and require additional safeguards. You and your company are required to use special safeguards in these situations.

To assist in designing point-of-operation safeguards for specific machining applications:

- OSHA has published a booklet called OSHA Publication No. 3067.
- Films regarding safety requirements for machine tools are available from ANSI and trade groups such as AMT (The Association for Manufacturing Technology).

1.2.4 Lockout/Tagout Procedure

The machine is powered by high voltage and other energy sources that represent potential hazards. To reduce the risk of injury or death establish, define, and practice a Lockout/Tagout procedure for the equipment in your facility.

Lockout/Tagout defines a minimum safe procedure to be followed by persons who might be confronted with unexpected energizing, start-up, or release of stored energy that could cause injury or death. At a minimum, all Lockout/Tagout procedures should include:



Electrical Shock Hazards Exist - Work Safely

Failure to turn the Main Power disconnect to the Off position, lock it out, and tag it could result in severe personal injury or death.

1. Set the machine's Main Power switch to the Off position.
2. Affix a **DO NOT START** tag with your name and department at, on, or near the machine's Main Power switch until all repairs are complete.
3. Turn the Main Power disconnect to the Off position.
4. Install an industry approved lockout device through the Main Power disconnect handle.
5. Install a tamper-proof padlock onto the lockout device. All persons performing maintenance on the machine at the same time shall install their own padlock onto the lockout device.
6. Disconnect and discharge the source of all energy sources (i.e. electrical, air, hydraulic, etc.).

Use the Lockout/Tagout policy and procedures defined by your company. If no Lockout/Tagout procedure has been communicated to you, please ask your supervisor to provide and explain one or obtain copies of standards to develop and implement Lockout/Tagout in your plant.

- ANSI has issued a standard to assist in the design and development of a Lockout/Tagout policy. It is written as a guide to follow when determining the specific procedures and training necessary and appropriate to your plant operations.
 - Copies of the ANSI standard are available from ANSI at 1430 Broadway, New York, NY 10018. Ask for: Standard ANSI Z244.1-1982, or as modified, for personal protection - Lockout/Tagout Of Energy Sources - Minimum Safety Requirements.

1.3 Personal and Professional Safety

General safety precautions should be practiced everyday, but never become common place. Safety is the responsibility of every person on the job site. In this regard, consider yourself responsible for safety in your workplace. No one is better positioned to eliminate or prevent unsafe conditions than you. Employees should practice the following, regardless of their position, title, or labor grade:

- Approach and intercede anytime you witness unsafe practices.
- Be prepared to report any condition which seems unsafe to your company's safety department or management staff. Such action could save lives. In particular, report such things as:
 - Missing or defective guards and protective devices.
 - Leaks and spills of cutting fluid, lubricants, or any liquids.
 - Improperly stored chemicals or flammable materials and any unusual fumes.
 - Loose, worn, or broken flooring.
 - Slippery, broken, or unstable platforms.
 - Missing, broken, or unstable handrails.
- Avoid making sudden movements or loud noises, engaging in horseplay, or other activities that may prove distracting and result in an unsafe or dangerous work environment. Remember that around machinery, it only takes a second's inattention to result in personal injury or death.
- Observe and obey all signs posted on the machine and its components and signs placed by your company, like:
 - NO SMOKING
 - SAFETY GLASSES MUST BE WORN WHILE OPERATING THIS MACHINE
 - DANGER - HIGH VOLTAGE, etc.
- Always wear approved eye protection and other safety equipment as defined by OSHA, the equipment manufacturer, your company safety policy, or MSDS (Material Safety Data Sheet) to meet the demands of current conditions while operating or working around the machine.
- Clean up any oil or other liquid spills immediately.

- Obtain an MSDS for each chemical (cutting fluids, lubricating oils, greases, etc.) used on or around the machine. Practice safe working habits and wear all protective equipment required. Know and understand the procedures to follow in the event:
 - Your skin is exposed to the chemical.
 - Your eyes are contacted by the chemical.
 - You ingest or inhale the chemical.

1.4 Equipment and Operation Safety

We urge that all machines be operated, equipped, and cared for in strict compliance with all local, state, and federal safety regulations. The following safety requirements apply to all machinery and should be practiced in addition to machine specific safety guidelines.

- Do not neglect your responsibility to ensure that all machines are operated, maintained, and equipped with complete regard for operator safety.

Only qualified personnel, trained in safety and all machine functions, should be allowed to operate, maintain, or repair the machine.

To be considered qualified, all personnel whose duties are related to the operation, maintenance, or repair of the machine should:

- Be familiar with the location and operation of all [**EMERGENCY**] (E-Stop) buttons.
- Study the related guides or manuals prior to attempting to operate, maintain, or repair the machine.
- Locate and understand all **Danger**, **Warning**, and **Caution** labels on the machine and accessories.

Machines are designed for operation by one person at a time.

- Persons other than the designated operator should stay out of the area during operation.
- Unauthorized personnel may increase the potential for injury or damage.

Be Sure to Always:

- Take steps to protect the machine tool and control cabinets from moisture and condensation. Moisture can damage electrical components, causing unintended machine motion, leading to injury or machine damage.
- Maintain the exact original color and vividness of any machining areas that are color coded as safety precautions.
- Ensure that all shields and guards are in place and all covers, cabinet doors, and terminal boxes are closed prior to operation.
- Avoid accidentally bumping or touching any machine controls. This may cause unintended machine motion leading to injury or machine damage.
- Be alert for loose, damaged, or worn parts on the machine and associated equipment. Immediately correct or report any loose items, noticeable change in operating noise, or unusual machine action.
- Be aware of the machine and auxiliary component's moving members and range of movement which may create pinch points. Always be cautious of these areas. Avoid pinch points entirely during machine operation.
- Keep the machine properly maintained. Perform maintenance at prescribed intervals, as described in the Maintenance Guide. Maintenance should be performed by authorized individuals.
- Check the fluid level of each reservoir regularly. Add only recommended fluids, as described in the Maintenance Guide.
- Keep the machine and work area clean and clear of chips, rags, etc. Cleaning should be performed, at least, daily.
- Keep away from the ATC (Automatic Tool Changer), APC (Automatic Pallet Changer), and other exchange devices while the machine is in automatic operation.
- Double check the set up, tooling, and workpiece before power up.
- Check all clamp bolts for tightness and verify the accuracy of all set ups before starting machining operation.
- Use caution when handling workpieces after machining. Machining can produce sharp edges and, at times, considerable heat.

You Should:

- Never wear loose-fitting clothing or accessories that may be entangled in cutting tools, revolving parts, conveyors, or other machine components. In particular, never wear any of the following items when working in a shop environment:
 - Jewelry (rings, bracelets, watches, or necklaces) and accessories (necktie, scarf, or loose-fitting belt)
 - Long hair (tie it back and contain it under a hair net or cap)
- Never operate any machinery after taking strong medication, using nonprescription drugs, or consuming alcoholic beverages.
- Not make modifications to the machine or associated equipment that may reduce the operational safety of the device. For example:
 - Never paint, alter, deface, or remove any **Danger**, **Warning**, or **Caution** label. Replacement labels are available from Makino.
 - Never change or by-pass the location of the stroke limit dogs, limit switches, interlock circuitry, etc.
 - Never operate the machine with any safety device disabled or removed or any cover open or removed.
- Not allow yourself to become trapped or caught in pinch points:
 - Always be aware of a safe exit when in or around any type of equipment. Know the machine travel limits and always keep all body parts clear of moving components.
 - Never enter an area restricted by a barrier without first properly shutting down the equipment and performing Lockout/Tagout.
 - Never perform any set up or set up check or reach into the work area or across moving parts while the machine is in automatic operation.
 - Never change a workpiece or perform a manual operation on a workpiece while the machine is in automatic operation.
- Never activate or press any operating control unless you are trained on the operation of that device, machine, or equipment.
- Never apply excessive pressure or force to tools or tool holders while they are positioned in the spindle.
- Never use compressed air to clean or blow chips or dirt off the table, workpiece, or machine.

1.5 Environmental Safety

Follow these specific practices when working at and around the machine:

- The machine area should be well lighted, dry, and free from obstructions. Keep the area around the machine clean and in good order at all times.
- Equipment should not be operated during severe thunderstorms or other electrical disturbances.
- In the event of a power failure, immediately turn the machine Off. Leave the machine Off until power is fully restored and normal operations can be safely resumed.
- Never step on machine covers as they can become slippery during normal operation and are not designed to support your weight.
- Never perform grinding operations in the vicinity of the machine. Grinding dust can contaminate components and cause premature wear resulting in inaccuracies or component failure.
- Never weld parts mounted on or in the machine. Electrical currents associated with welding can cause bearing damage or explosions resulting in serious injury or equipment damage.
- When a platform is placed around the machine, it should be extremely sturdy and have anti-slip surfaces.
- Keep all flammable liquids away from the work area.

1.6 Lifting Safety

These lifting safety precautions must be followed by all persons responsible for lifting. This includes, but is not limited to, lifting machinery, machine components, and workpieces:

- All lifting equipment must be properly rated for the weight being lifted.
- All lifting equipment must be inspected on a regularly scheduled basis.
- Any lifting device found to be defective must be repaired in accordance with industry standards.
- Any lifting device that cannot be repaired must be destroyed.

1.6.1 Lifting Devices

Follow these safety precautions when using lifting equipment such as slings, chains, hooks, eye bolts, rings, etc.

1.6.1.1 Slings

Use nylon slings when lifting irregularly shaped objects.

- Never exceed the lifting capacity of any sling.
- Protect slings with heavy padding when lifting objects with sharp edges.

Any sling that cannot be repaired must be destroyed.

1.6.1.2 Chains

Lifting chains must be stored properly to prevent damage when not in use.

When using chains:

- Never exceed the lifting capacity of any chain.
- Do not drag chains.
- Do not allow chains to become tangled or kinked.
- Never twist chains in an attempt to shorten their length. Use proper devices to shorten a chain's length.
- Use chains equipped with enclosed lifting hooks.

Any chain that cannot be repaired must be destroyed.

1.6.1.3 Hooks

When using hooks:

- Never exceed the lifting capacity of any lift hook.
- Do not repair lifting hooks.
- Destroy and discard bent lifting hooks.

1.6.1.4 Eye Bolts

To ensure a safe load carrying capacity:

- Never exceed the lifting capacity of any eye bolt.
- An eye bolt must be engaged to at least 90 percent of its threaded portion.
- Never weld or heat an eye bolt to a temperature in excess of 480°C (900°F). Heat exposure can change stock physical properties creating an unsafe eye bolt.
- Never use an inch eye bolt in a metric thread or vice versa:
 - Use inch series eye bolts in inch series threads.
 - Use metric series eye bolts in metric series threads.
- Never use a painted or coated eye bolt as this may cover up defects.
- Discard and destroy all defective eye bolts.

1.6.1.5 Rings

When possible, use a lifting ring or clevis instead of an eye bolt. Secure the lifting ring with the correct inch/metric series bolt.

- Never exceed the lifting capacity on any lifting ring or clevis.

Any ring that cannot be repaired must be destroyed.

1.6.2 Lifting the Machine or Components

Use only a lifting device and related equipment that is capacity rated to lift the total weight of the machine (and shipping skid, if the skid is attached during lifting). Always follow the lifting instructions provided in this Guide.

Before lifting, refer to shipping documents to obtain the weight of the machine or component being lifted. If the necessary weights cannot be determined, consult Makino. In addition:

- Only qualified riggers should perform machine lifting operations.
- Never place any portion of your body beneath a suspended object.
- Lift the machine only at the designated lifting points.
- Never lift the machine higher than necessary.
- Always maintain the center of gravity by properly adjusting the leg lengths of the lifting device.

1.6.3 Lifting Workpieces and Fixtures

Always use a lifting device and related lifting equipment that is capacity rated for the workpiece or fixture being lifted.

- Never place hands, arms, feet, or any other portion of your body beneath a suspended workpiece or fixture.
- Always maintain the center of gravity by properly adjusting the leg lengths of the lifting device.

1.7 Mechanical Maintenance Safety

Mechanical maintenance procedures for Makino machines should be performed by properly trained and qualified personnel. They should adhere to these specific practices when working with the machine:

- Always perform Lockout/Tagout before removing any safety guard, cover, barrier, or any basic component of the machine. See [section 1.2.4](#).
- Block vertical sliding members before disassembling any portion of their drive train.
- Be familiar with and adhere to safe lifting practices described in [section 1.6](#) before lifting the machine or components.
- Never return the machine to production until all safety guards, covers, barriers, or other safety devices are reinstalled and confirmed operational.

1.7.1 Mechanical Maintenance Equipment

Mechanical maintenance personnel should:

- Keep all hand tools in good repair.
- Use each hand tool in a manner for which it is intended.
- Never use any hand tool that is defective, broken, or worn out.
- Keep alert with regard to nearby hazards.

1.8 Electrical Maintenance Safety

Electrical maintenance procedures for Makino machines must be performed by properly trained and qualified personnel. They should adhere to these specific practices when working with the machine:

- Always perform Lockout/Tagout before working on electrical circuits. See [section 1.2.4](#).
- Use extreme care when troubleshooting with power On.
- With the power Off, identify and discharge any power retaining devices, such as capacitors, before starting any maintenance procedure within any electrical control cabinet.

1.8.1 Electrical Maintenance Equipment

Use only industry standard approved electrical testing equipment.

- Volt/Ohm Meters.
- Oscilloscopes.
- Static Ground Wrist Straps.

Machine Grounding

All Makino machines must be properly grounded at the time of installation.

- Proper grounding requires an isolated earth ground.
- Ground the machine in accordance with local, state, and federal regulations.

Machine Electrical Devices

Always replace defective electrical components (fuses, shunts, overloads, etc.) with components of the same capacity and rating as the original equipment. If you are unsure of the correct electrical component or part, refer to the machine's electrical drawings or consult Makino.

1.9 Machine Safety

In addition to safety precautions for all machining operations, follow these machine specific safety precautions.

- The energy generated by HSM (High-Speed Machining) requires special tooling safety considerations that must be fully understood by the operator. See [section 1.10](#).

1.9.1 Machine Safety Features

The machine provides the following safety features:

- **[EMERGENCY]** buttons, to immediately halt all machine functions.
- Manual interrupt buttons, located on auxiliary operation panels, to prevent automatic operation of that unit when active.
- S/G (Splash Guard), ATC, and MTC door interlocks, to halt machining operations when the doors are opened.

Descriptions, operational detail, and precautions related to these safety features are described in the Operation Guide.

1.9.2 Location of Emergency Buttons

[EMERGENCY] (E-stop) buttons are typically located on the following operation panels:

- Main
- ATC (Automatic Tool Changer)
- APC (Automatic Pallet Changer)
- Lift-up Chip Conveyor

1.9.3 Location of Safety Labels

Labels are attached to the machine at specific areas to identify safety risks and provide important instructions that must be read and followed. Typical locations include:

- Operation panels
- ATC magazine door
- Oil controller
- Refrigerated air dryer

1.9.4 Pinch Points

All machines have potential pinch point areas. Depending on the machine type and ancillary components, pinch points may occur in areas such as:

Pallet Changer

The Automatic Pallet Changer exchanges pallets between the machining area and the pallet stocker.

- The pallet changer arm moves up and down. Do not reach under or put anything under the arm during operation.
- Pallet exchange motion is clockwise (cw) and counterclockwise (ccw). Do not reach or put anything in the path of the pallet changer.
- An optional light curtain, which halts machine operation when interrupted, can be purchased to safeguard the stocker area.

Chip Conveyor

The chip conveyor removes chips from the coolant tank.

- Do not reach into a moving conveyor.

Rear Door Access

The rear access doors protect personnel while the machine tool is in operation.

- Never enter the machine or leave the guard doors open during machine operation.
- Only authorized maintenance personnel should enter this area.

Spindle and Tool

When manually loading a tool into the spindle:

- Do not place your hand between the tool holder and the spindle nose.

1.10 Tooling Safety

1.10.1 Standard Tooling Safety Precautions

You should:

- Always inspect all tool holders for nicks or other damage to ensure the accuracy of machining operations and to prevent damage to the equipment.
- Always keep the spindle and tool holder tapers clean.
- Always use sharp tools and keep tool lengths as short as possible.
- Always use speeds and feeds appropriate for cutting conditions and tooling limitations.
- Always use a glove or shop cloth and avoid the cutting edges when handling or changing tools.
- Never reach into the machining area until the spindle is at a complete stop.
- Never operate the machine while wearing gloves or using a shop cloth.

1.10.2 High-speed Machining Tooling Safety Precautions

HSM (High-Speed Machining) requires additional tooling safety precautions:

- Operators should receive training to ensure a solid understanding of the potential energy generated in HSM applications and to observe proper safety practices.
- All tools placed in the spindle should be rated for the maximum spindle rpm. Using tooling rated for maximum rpm prevents unsafe conditions due to improper spindle speed selection.
- Balance all tooling to Makino's recommendation.
- Not all tools are designed for HSM and balancing alone will not make them safe for this application.
 - Confirm suitability for high-speed application with the tool manufacturer.
 - Balance the entire tool assembly to prevent stacking component tolerances.
- When machining at high-speed with inserted tools:
 - Tooling must be rated for the maximum spindle speed available.
 - All inserts should be screwed directly to the cutter body (screw down). Clamps are insufficient at high rpm.
 - Fasten inserts at the manufacturer's recommended torque with the recommended anaerobic sealing compound to prevent loosening.
 - Routinely replace mounting screws to avoid material fatigue.

1.11 Equipment and Operation Safety

In addition to the general safety precautions in [section 1.2](#), adhere to these specific practices when working with the machine.

- Never use any device not specifically designed for the intended use.
- Never replace tooling or tighten tool holders in the spindle.
- Never lay tools on the machine where they may interfere with machine movement or become entangled with the workpiece.
- Never exceed the specified maximum workpiece weight.
- Never perform any set up work while the machine is in operation.

1.11.1 Working Inside the Machine

When working inside the machine, be aware that axes, spindle, ATC or APC, and chip conveyor operation and flying chips or coolant are hazards that could result in serious injury or death. Therefore:

- Enter the machining area only when all machine motion is completely stopped and the control is in the STOP or RESET state.
- Be careful not to slip on the tilted telescopic covers.
- Remember injury can occur if the body is caught between the S/G door and the frame.
- Ensure that coolant on the ceiling does not drip into the eyes.

1.11.2 Prior to Machining

- Check with the manufacturer of any accessory not designed, built, or supplied by Makino to ensure the device will operate properly and safely under the proposed operating conditions.
- Load or unload workpieces only when the spindle and machine are at a complete stop and the control is in STOP or RESET state.
- Prior to unattended operation, prove-out part programs to ensure clearance between machine parts, fixturing, and the workpiece.
- Before moving any axis, make sure there are no obstructions within the range of movement.
- Confirm the set up prior to automatic operation. Check the tightness of all clamping devices, fixture mounting, tool holders, etc.

1.11.3 Workpiece Handling Safety

These specific lifting safety precautions must be followed by all personnel responsible for lifting workpieces. In addition to the general lifting precautions in [section 1.6](#), adhere to these specific practices:

- The machine has a maximum allowable table capacity. Never exceed its weight limit.
- Load and unload workpieces or fixturing only when the spindle and machine are at a complete stop and the control is in STOP or RESET state.
- Use lifting equipment for heavy workpieces and other materials. Never attempt to lift anything beyond a reasonable weight without proper devices.
- Use extreme care not to place fingers or hands where they may be pinched while loading or unloading a workpiece or fixture.

1.12 Coolant Safety

The machine is equipped with several coolant options. Follow these safety guidelines with all possible coolant configurations:

Always:

- Read and apply the proper handling methods described in the coolant's MSDS.
- Adjust coolant lines and flow only when the spindle and machine are at a complete stop and the control is in STOP or RESET state.
- Select a water soluble coolant, whenever possible. Oil based coolants or cutting oils may, under certain conditions, create a fire hazard.
- Be careful when handling coolant fluids. In some instances, these fluids have caused irritation and harmful effects to skin. Persons with any type skin allergy or sensitive skin should not handle coolants without proper protection.
- Avoid breathing coolant.

1.13 Installation Safety Instructions

The following information is related to machine installation. Adhere strictly to the installation instructions provided in this Guide:

- Review the general lifting safety rules in [section 1.6](#) and the specific procedures prior to moving the machine.
- Review the general electrical safety rules in [section 1.8](#) and the specific procedures prior to connecting power to the machine.

Chapter 2 Program Word Addresses

Word Address Formats

Makino Vertical and Horizontal
Machining Centers



Chapter 2 Program Word Addresses

Contents

2.1 Overview	2- 1
2.2 Program Word Addresses.	2- 2
2.3 Miscellaneous Program Codes.	2- 4
2.3.1 Tape Start Code	2- 4
2.3.2 Leader	2- 4
2.3.3 Program Start Code	2- 5
2.3.4 Comment Codes	2- 5
2.3.5 / - Slash Code	2- 6
2.3.5.1 Multiple Block Skip	2- 6

2.1 Overview

All commands in an NC program require the use of a word address plus a numeric value. This chapter describes the word address allowed by the Pro 3 control and their numeric formats. Also included are certain other program commands, such as the: Control In and Out codes, Block Skip slash code, etc.



Information Subject to Change

Every effort was made to ensure the accuracy of the data in this chapter at the time of publication. Machine and control specifications are subject to change without notice.

2.2 Program Word Addresses

Table 2-1 provides a description of the basic word addresses used on the Pro 3 control. Both inch and metric formats and a brief description of the word usage are included.

EXAMPLE: The word formats are shown as $X_{\pm} \& 4.4$ digits and $X_{\pm} \& 5.3$ digits; meaning there are 4 digits to the left and 4 to the right of the decimal point for the inch format and 5 digits to the left and 3 to the right of the decimal point for the metric format.

These formats are for conventional NC programs, not macro type programming statements.

TABLE 2-1 WORD ADDRESS FORMAT CHART

	Description	Inch Format	Metric Format	Remarks
A	Rotary Axis	$A_{\pm} \& 5.3$ digits	$A_{\pm} \& 5.3$ digits	4 th Axis on VMC's, 5 th Axis on HMC's
B	Rotary Axis	$B_{\pm} \& 5.3$ digits	$B_{\pm} \& 5.3$ digits	4 th Axis on HMC's
C	Chamfer Word	C & 4.4 digits	C & 5.3 digits	Used with Auto. Corner Chamfer (Option)
D	Tool Radius Compensation Number	D & 2 digits	D & 2 digits	Range D0 - D99 (Only D2 is used with Custom Side Tool Management)
E	Not Used			
F	Feed Rate Word	F & 4.2 digits	F & 5.1 digits	Range varies by machine model
G	Preparatory Function	G & 2 digits	G & 2 digits	Defines mode of operation (Special case Ggg.g)
H	Tool Length Offset Number	H & 2 digits	H & 2 digits	Range H0 - H99 (Only H1 is used with Custom Side Tool Management)
I	X Axis Arc Center Point	$I_{\pm} \& 4.4$ digits	$I_{\pm} \& 5.3$ digits	Used with Circular Interpolation - G2/G3
J	Y Axis Arc Center Point	$J_{\pm} \& 4.4$ digits	$J_{\pm} \& 5.3$ digits	Used with Circular Interpolation - G2/G3
K	Z Axis Arc Center Point	$K_{\pm} \& 4.4$ digits	$K_{\pm} \& 5.3$ digits	Used with Circular Interpolation - G2/G3
L	Data Set	L & 2 digits	L & 2 digits	Table identifier in G10 block
	Sub Program	L & 4 digits	L & 4 digits	Sub Program repeat word. Used instead of P Word for repeating Sub Programs
	Canned Cycle	L & 4 digits	L & 4 digits	Canned Cycle Repeat Word - L0 to L9999
M	Miscellaneous Function	M & 3 digits	M & 3 digits	Controls program execution and machine functions
N	Sequence Number	N & 5 digits	N & 5 digits	Range N1 to N99999
O	Program Number	O & 4 digits	O & 4 digits	Range O1 to O9999

TABLE 2-1 WORD ADDRESS FORMAT CHART (CONTINUED)

	Description	Inch Format	Metric Format	Remarks
P	Dwell	P & 8 digits	P & 8 digits	Used in G4 block, unit = .001 second
	Subprogram Program Number	P & 8 digits	P & 8 digits	Used in M98 block, format Pxxxxpppp Where: xxxx = subprogram repeat and pppp = subprogram number
				Used in M98 block, format Pppppxxxx Where: pppp = subprogram repeat and xxxx = subprogram number
	Canned Cycle	P & 8 digits	P & 8 digits	Dwell in Canned Cycles G76, G82, G87 and G89, unit = .001 second
Q	Shift Amount	Q & 4.4 digits	Q & 5.3 digits	Used with Canned Cycles G73, G76, G83, and G87
R	Arc Radius	R± & 4.4 digits	R± & 5.3 digits	Used with Circular Interpolation - G2/G3
	R Point	R± & 4.4 digits	R± & 5.3 digits	Reference point in Canned Cycles
	Angle	R± & 6 digits	R± & 6 digits	Angle of Rotation in G68 block Range = -360000 to +360000
S	Spindle Speed	S & 5 digits	S & 5 digits	Programmable in 1 RPM Increments (range varies by machine model)
T	Tool Number	T & 4 digits	T & 4 digits	Range T0 to T9999 Optional Range T0 to T99999999
U	Not Used			
V	Not Used			
W	Not Used			
X	X Axis Address	X± & 4.4 digits	X± & 5.3 digits	Minimum input = 0.0001" and 0.001mm
	Dwell	X & 5.3 digits	X & 5.3 digits	Used in G4 block Where: unit = .001 second and range = X.001 to X99999.999
Y	Y Axis Address	Y± & 4.4 digits	Y± & 5.3 digits	Minimum input = 0.0001" and 0.001mm
Z	Z Axis Address	Z± & 4.4 digits	Z± & 5.3 digits	Minimum input = 0.0001" and 0.001mm

2.3 Miscellaneous Program Codes

NC programs may require the use of certain other program codes. This section describes these codes and their ISO and EIA formats.

Use of the ISO character set is the preferred over the EIA character set because of the limited characters of EIA. Macro programming requires use of some characters that are not available in the EIA character set.

2.3.1 Tape Start Code

% for ISO coding; ER for EIA coding.

- The very first code of an NC program.
- If input media is tape, leader section can follow the Tape Start code and can contain man-readable coding to identify the program.
- Not required for programs entered by System P or from a PC.
- May be required by certain portable mini-file devices.

2.3.2 Leader

Used when the input media is from tape and used for man-readable coding.

- A Program Start code must immediately follow the leader section to signify the start of NC program coding.

2.3.3 Program Start Code

LF for ISO coding; ER for EIA coding.

- Also called the End Of Block (EOB).
- This code is input by the Enter key when generating programs with System P or on a PC.
- Must be the first code after the leader section and prior to the O program number.
- The code used (LF or ER) sets up the parity check for the remainder of the program.
 - If LF is used, all other codes in the program must be even parity (ISO).
 - If ER is used, all other codes of the program must be odd parity (EIA).
- The last code after all information is entered for a block of information (EOB).

2.3.4 Comment Codes

Used to signal the beginning and end of comments in a block of information.

For ISO coding:

"(" Control Out signals the start of a comment; ")" Control In signals the end of a comment. The use of "(" and ")" is required for the Pro 3 control.

For EIA coding:

Channel configuration 2-4-5 represents the Control Out code, and 2-4-7 represents the Control In code.

- Information programmed between the Control Out and Control In codes are not executed by the control.
- An LF, or ER, (EOB code) must not be programmed in a comment since it will be treated as the Control In code and the control will try to execute the data from that point to the next EOB code.
- Comments can be programmed throughout the program.

2.3.5 / - Slash Code

Used with the Block Skip feature to ignore data programmed in a block of information.

- Use in conjunction with the [BLOCK SKIP] button (or soft key).
 - When [BLOCK SKIP] is turned On, all blocks of information preceded by a / code are ignored during program execution.
- The / code should be placed at the beginning of the block to be ignored during program execution.
 - Information coded from the / code to the next EOB code is ignored when [BLOCK SKIP] is On. This applies even if the / code is placed at a position in the block other than at the start of the block.

2.3.5.1 Multiple Block Skip

Used in conjunction with the [BLOCK SKIP] buttons (or soft keys).

The slash (/) code can be programmed with a number 1 through 9 (/1 through /9). / and /1 are the same Block Skip function and corresponds to the setting of the [BLOCK SKIP] button. /2 through /9 corresponds to the settings of the Multiple Block Skip buttons or soft keys.

The number 1 can be omitted except when the number 1 and another [BLOCK SKIP] soft key setting are used to delete the same block.

EXAMPLE: The number 1 and 3 Block Skip functions are to be used for block N500.

Correct way to program:

/1/3 N500 ;

Incorrect way to program:

//3 N500 ;

Chapter 3 **Program Numbers - O Word**

Program Numbers, Names, and
Program Protection

Makino Vertical and Horizontal
Machining Centers



Chapter 3 Program Numbers - O Word

Contents

3.1 Overview	3- 1
3.2 Program Numbers and Names	3- 2
3.2.1 Program Numbers	3- 2
3.2.1.1 O Word	3- 2
3.2.1.2 (:) Colon Code	3- 2
3.2.2 Program Names	3- 2
3.3 Program Protection	3- 3
3.3.1 Registering Programs	3- 3
3.3.2 Program Directory Back-up	3- 3

3.1 Overview

All programs stored in the Pro 3 program directory must have an O program number of up to four numeric characters. This chapter describes the use of the O word for program numbers, how to name programs and how to protect programs by parameter settings.



Information Subject to Change

Every effort was made to ensure the accuracy of the data in this chapter at the time of publication. Machine and control specifications are subject to change without notice.

3.2 Program Numbers and Names

Part programs, sub-programs and macros are stored in the Pro 3 program directory under a program number.

Standard program storage provides the capability to store a total of 64 programs. This includes part programs, sub-programs, and macros.

3.2.1 Program Numbers

3.2.1.1 O Word

The O letter address and up to 4 digits are used to define the program number.

The O word is programmed in the range of:

O1 through O9999

3.2.1.2 (:) Colon Code

A (:) colon code can be used instead of the letter O if ISO coding is used.

The (:) code is programmed in the range of:

:1 through :9999

3.2.2 Program Names

The Program Number block can also contain a name to provide a means to identify the program stored in the directory.

- The Program Name follows the O word (or : code), and is programmed within the Control Out and Control In codes, and may contain up to 30 characters.
 - Characters in excess of 30 are ignored and not displayed.
- The Program Name is displayed during program execution.

3.3 Program Protection

All programs stored in the directory are protected (from editing or accidental deletion) by a Memory Lock key switch. The Pro 3 provides additional program protection through the use of parameter settings.

Two ranges of part program numbers can be inhibited from editing by setting Pro 3 control parameter 3202. Parameter 3202 is an eight bit parameter with bits 0 and 4 used to inhibit program editing.

- When parameter 3202, Bit 0 is set to 1:
 - Program numbers in the range of O8000 through O8999 cannot be edited or deleted.
- When parameter 3202, Bit 4 is set to 1:
 - Program numbers in the range of O9000 through O9999 cannot be edited or deleted.

3.3.1 Registering Programs

When registering (storing) programs, the Memory Lock key switch must be set to the unlocked position.

- If registering 8000 or 9000 series programs, the appropriate parameter bit must be set to 0.

3.3.2 Program Directory Back-up

When performing a program directory back-up, parameter 3203, bits 0 and 4 must be set to 0 to include all the programs stored in the directory.

Chapter 4 **Coordinate Systems**

Absolute, Machine, and Relative
Coordinates

Makino Vertical and Horizontal
Machining Centers



Chapter 4 Coordinate Systems

Contents

4.1 Overview	4- 1
4.2 Machine Coordinates	4- 2
4.2.1 Machine Coordinate Programming	4- 2
4.2.2 Programming Example	4- 3
4.3 Absolute Coordinates	4- 4
4.3.1 Absolute Coordinate Programming	4- 4
4.3.2 Programming Example	4- 5
4.4 Machine Zero and Program Zero	4- 6
4.4.1 Machine Zero	4- 6
4.4.1.1 Location of Machine Zero - HMC's	4- 6
4.4.1.2 Location of Machine Zero - VMC's	4- 6
4.4.2 Program Zero	4- 7
4.5 WCS's (Work Coordinate Systems)	4- 8
4.5.1 Standard WCS's - G54 to G59	4- 8
4.5.2 Setting Standard WCS's with G10	4- 9
4.5.2.1 Programming Example	4- 10
4.6 EWCS's (Expanded Work Coordinate Systems) - G54.1	4- 11
4.6.1 Setting EWCS's with G10	4- 11
4.6.1.1 Programming Example	4- 12
4.7 Local Coordinate Systems - G52	4- 14
4.7.1 Programming Example	4- 15
4.8 Reference Point Positioning	4- 16
4.8.1 Reference Point Positioning with G28 and G30	4- 17
4.8.1.1 G28 and G30 Intermediate Points	4- 17
4.8.1.2 Programming Example	4- 18
4.8.1.3 Avoiding Intermediate Point Positioning	4- 19
4.9 Positioning from the Reference Point - G29	4- 20

- 4.9.1 Change of Work Coordinate System Offsets 4- 20
- 4.10 Reference Point Return Check - G27 4- 21
- 4.11 Floating Reference Point Return - G30.1 4- 22
 - 4.11.1 Avoiding Intermediate Point Positioning 4- 22

4.1 Overview

This chapter describes the differences between the Machine, Absolute, and Relative Coordinate systems, Other topics discussed in this chapter include:

- Machine Zero
- Program Zero
- Work Coordinate Systems
- Reference Point Positioning



Information Subject to Change

Every effort was made to ensure the accuracy of the data in this chapter at the time of publication. Machine and control specifications are subject to change without notice.

4.2 Machine Coordinates

Machine Coordinates represent the position of the machine axes relative to Machine Zero. The position of each axis is shown in the Machine Coordinate display which is updated by all axis moves. This includes moves made by program commands in the Auto and MDI modes and by power feed/jog moves in a Manual mode.

4.2.1 Machine Coordinate Programming

All machine axes (X, Y, Z, B, etc.) can be programmed in Machine Coordinates. Machine Coordinate programming is performed using the G53 command:

G53 - Machine Coordinate System Selection

G53 is a non-modal command and must be programmed in every block requiring a Machine Coordinate move.

- G53 does not cancel the active interpolation modes of G1, G2, or G3.
- Axis commands programmed in the G53 block are relative to Machine Zero.
- All offsets (work coordinate, tool length, and tool radius) are ignored in a Machine Coordinate block.
- Axis traverse rates are: rapid if G0 is active, or at the programmed feedrate if G1, G2, or G3 is active.

4.2.2 Programming Example

Figure 4-1 illustrates two points, P1 and P2, dimensioned relative to Machine Zero.

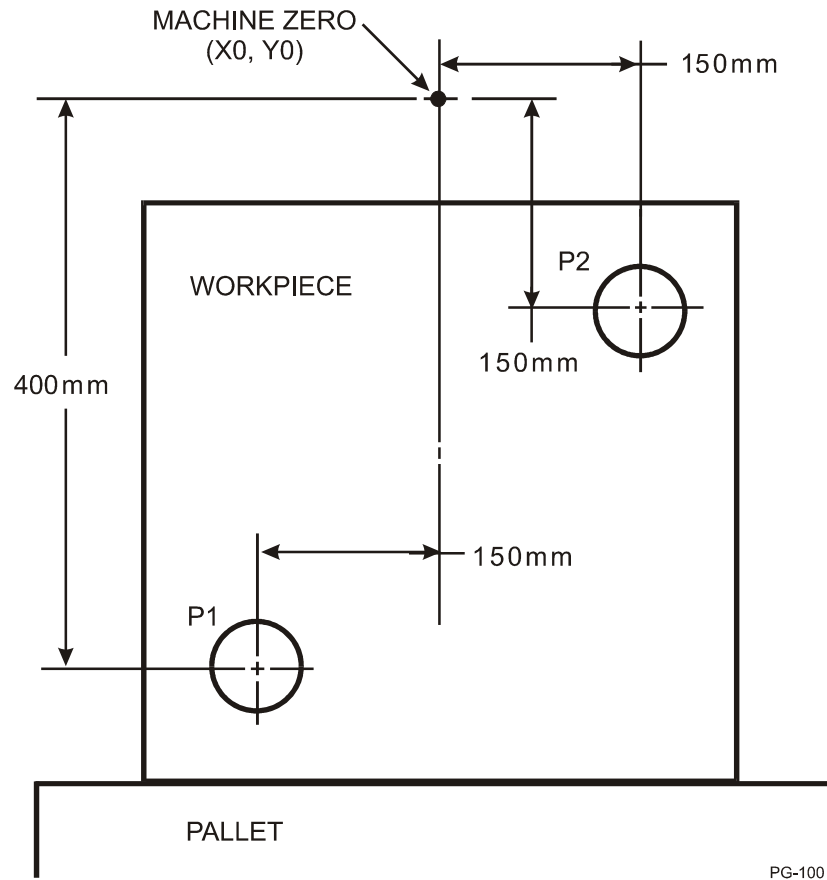


FIGURE 4-1 MACHINE COORDINATE PROGRAMMING EXAMPLE

In this example the program blocks are:

P1 = Nnnnn G53 X-150.0 Y-400.0 ;

P2 = Nnnnn G53 X150.0 Y-150.0 ;

4.3 Absolute Coordinates

Absolute Coordinates represent the position of the machine axes relative to Program Zero. The position of each axis is shown in the Absolute Coordinate display which is updated by all axis moves. This includes moves made by program commands in the Auto and MDI modes and by power feed/jog moves in a Manual mode.

- Axis moves made in the G91 Incremental Input mode also update the Absolute Coordinate display.

Program Zero is the position on the workpiece from where all Absolute Coordinates are calculated. Program Zero is established using work coordinate system offsets to define its position relative to Machine Zero.

4.3.1 Absolute Coordinate Programming

All machine axes (X, Y, Z, B, etc.) can be programmed in Absolute Coordinates. Absolute Coordinate programming is performed using the G90 command.

G90 - Absolute Input

- G90 is a modal command cancelled only by the G91 Incremental Input command.
- G90 is the initialized state at machine power on, after a RESET, and at end of program (by M2 or M30).
 - The default state of G90 is established by a CNC parameter 3402, bit 3.
- Axis commands programmed in the G90 mode are relative to Program Zero.
- Absolute Coordinates are offset by active offsets (work coordinate, tool length, and tool radius).
- Axis traverse rates are: rapid if G0 or any Canned Cycle is active, or at the programmed feedrate if G1, G2, or G3 is active.

4.3.2 Programming Example

Figure 4-2 illustrates two points, P1 and P2, defined as Absolute Coordinates relative to Program Zero established using the G54 work coordinate system.

- For this example, the G54 work coordinate system must have the X and Y data fields set to:

X-150.000 and Y-400.000

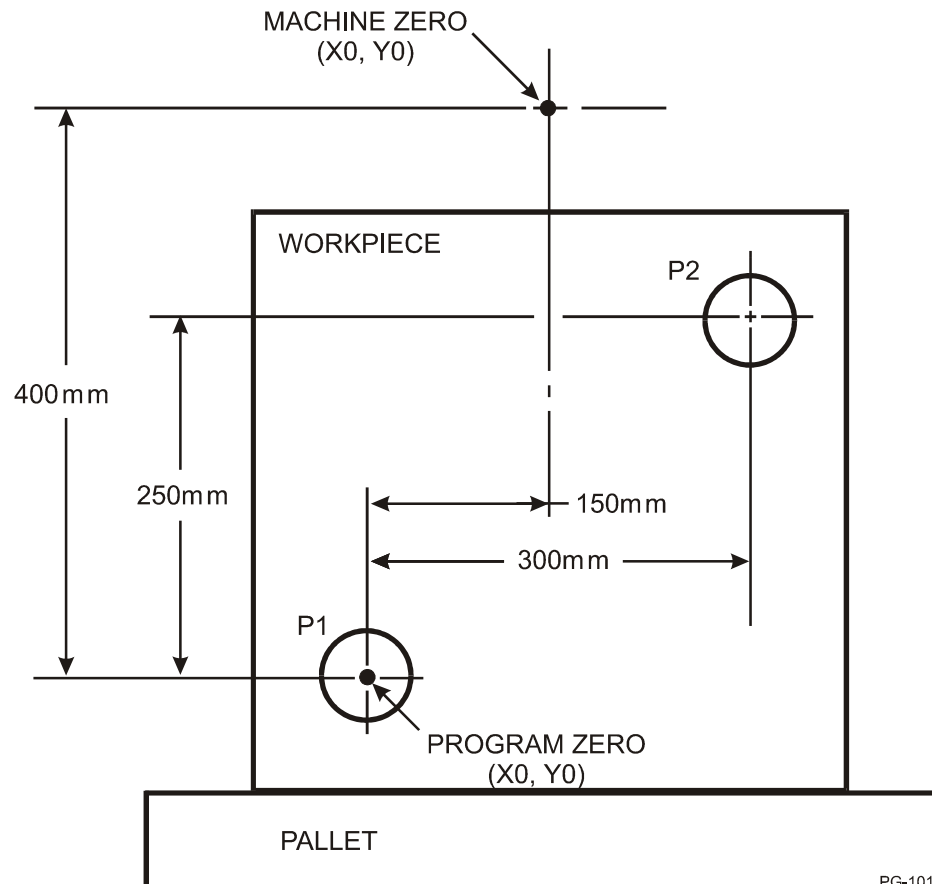


FIGURE 4-2 ABSOLUTE COORDINATE PROGRAMMING EXAMPLE

In this example the program blocks are:

P1 = Nnnnn G54 G0 X0.0 Y0.0 ;

P2 = Nnnnn G54 G0 X300.0 Y250.0 ;

4.4 Machine Zero and Program Zero

4.4.1 Machine Zero

Machine Zero is the position of the machine axes after the machine has been turned on and alignment performed.

4.4.1.1 Location of Machine Zero - HMC's

On HMC's (Horizontal Machining Centers), Machine Zero is typically located as follows:

- X axis - The center of the pallet (pallet centered in X axis).
- Y axis - The full plus (+) end of its travel (full up position).
- Z axis - The full plus (+) end of its travel (the tool change position).
- B axis - The long locator edge towards the spindle and parallel to the X axis.

From Machine Zero, X+ and X- would be from the pallet centerline. Y- and Z- would be toward the work zone.

4.4.1.2 Location of Machine Zero - VMC's

On VMC's (Vertical Machining Centers), Machine Zero is typically located as follows:

- X axis - The full plus (+) end of its travel (table all the way to the left).
- Y axis - The full plus (+) end of its travel (table towards the operator).
- Z axis - The full plus (+) end of its travel (the tool change position).

From Machine Zero, X-, Y- and Z- would be toward the work zone.

4.4.2 Program Zero

Program Zero is the position on the workpiece from where all Absolute Coordinates are calculated.

- Until a Program Zero position is established, Machine Zero and Program Zero are located at the same position.
- The position of Program Zero is generally selected based on how the part print is dimensioned or the position that is easiest to calculate the coordinates for the program.

Program Zero is established by one of several methods:

- Work Coordinate System (G54 through G59).
- With G10 statements to set values into the Work Coordinate System.
- G92 - Work Coordinate System Setting. G92 is not often used because of the Work Coordinate System (G54 through G59) feature.

4.5 WCS's (Work Coordinate Systems)

4.5.1 Standard WCS's - G54 to G59

WCS's (Work Coordinate Systems) are used to establish Program Zero when it is located at a position other than Machine Zero. Six sets of WCS's are provided as standard equipment with the capability to offset the X, Y, Z, and rotary axes (A or B or C).

A seventh WCS (EXT #0) is also supplied as standard equipment. WCS (EXT #0) is always active with its values automatically applied to the active WCS G54 to G59.

- A WCS is made active by a G code in the program:
 - G54 = #1 Work Coordinate System
 - G55 = #2 Work Coordinate System
 - G56 = #3 Work Coordinate System
 - G57 = #4 Work Coordinate System
 - G58 = #5 Work Coordinate System
 - G59 = #6 Work Coordinate System
- G54 to G59 are modal codes that remain active until canceled by each other.
- G54 is active at machine power On, after a RESET, and at end of program (by M2 or M30).
 - If the Pro 3 control parameter 3407 Bit 6 is set to 1, the active WCS G54 to G59 is not canceled by RESET or at end of program.
- The G54 to G59 should be programmed either before or with the axis commands that require the WCS to be active.
- Axis commands programmed in the WCS block move to the programmed position relative to the active G54 to G59 code.
- WCS values are entered by:
 - the operator at the CRT,
 - G10 commands in the MDI mode, or,
 - G10 commands in the NC program.
- An NC program can use any one or all six of the WCS's within the same program. Only one WCS can be active at a time.

- Typical uses for WCS's are:
 - For HMC's, a different WCS could be used for each index position. Example: position B0 would use G54, B90 would use G55, B180 would use G56, etc.
 - For VMC's, a different WCS could be used for each part setup when multiple parts are set up on the table.
 - Different WCS can be used on the same part surface to simplify calculating the coordinates for a special part feature.
 - A G92 work coordinate shift can be used with an active WCS to establish a different Program Zero relative to the current Program Zero established by the active WCS.



Important G92 Considerations

Any work coordinate shift made with a G92 command will remain active (unless canceled) when changing from one WCS to another.

For this reason, we recommend using G10 statements to set new values rather than the G92 code because G10 affects only the active work coordinate system.

G92 Work Coordinate shifts are cancelled by a Manual Reference Return operation.

4.5.2 Setting Standard WCS's with G10

WCS values can be set with MDI or NC program commands using a G10 block. In the G10 block:

- The L2 code signifies that the standard (G54 to G59) WCS's are to be set.
 - L2 must be programmed in each G10 block.
- The WCS number (G54 to G59) to be set is specified by a P code.
 - P1 = G54, P2 = G55, P3 = G56, P4 = G57, P5 = G58, and P6 = G59.
 - The P code must be programmed in each G10 block.
 - The common EXT #0 offset fields can also be set by specifying P0.
- The WCS axes fields to be set are specified using the axes word addresses (X, Y, Z, B, etc.) along with the setting value.
 - Only the axes word addresses specified in the G10 block are set. Word addresses not specified in the G10 block remain unchanged.

- WCS setting values are affected by the active G90 Absolute or G91 Incremental Input mode.
 - If G90 is active, the programmed value replaces the existing value.
 - If G91 is active, the control algebraically sums the programmed value to the existing value.

4.5.2.1 Programming Example

Setting Example in the G90 Absolute Input mode.

In this example the X and Y fields of WCS #2 (G55) are set.

- The G90 Absolute Input mode is used so that the existing value will be replaced with the programmed value.

N100 G90 G10 L2 P2 X-100.000 Y-200.000 ;

When N100 is executed, the existing values in the X and Y fields of WCS #2 (G55) are set to X-100.000 and Y-200.000.

Setting Example in the G91 Incremental Input mode.

In this example the X and Y fields of WCS #3 (G56) are set.

- The G91 Incremental Input mode is used so that the existing value will be updated by the programmed value.

N200 G91 G10 L2 P3 X0.100 Y-0.200 ;

When N200 is executed, the existing values in the X and Y fields of WCS #3 (G56) are updated by the values programmed for X and Y.

Existing Value	Programmed Value	Resulting Value
X150.000	X0.100	X150.100
Y200.000	Y-0.200	Y199.800

4.6 EWCS's (Expanded Work Coordinate Systems) - G54.1

As an option, the number of WCS's can be expanded to include an additional 300 offsets. The number of available EWCS's is dependant upon the supplied option.

When an EWCS option is supplied, these WCS's are made active by a G54.1 code and a P word in the range of P1 up to P300.

- If G54.1 is programmed without a P word, P1 WCS is assumed.
- If a P word is programmed out of the supplied offset range an error is generated.
- All previous programming rules stated for the standard G54 to G59 WCS's also apply to the EWCS's.

4.6.1 Setting EWCS's with G10

EWCS values can be set with MDI or NC program commands using a G10 block. In the G10 block:

- The L20 code signifies that an optional EWCS is to be set.
 - L20 must be programmed in each G10 block.
- The EWCS number to be set is specified by a P code.
 - P1 = G54.1 #1 through P300 = G54.1 #300.
 - The P code must be programmed in each G10 block.
- The EWCS axes fields to be set are specified using the axes word addresses (X, Y, Z, B, etc.) along with the setting value.
 - Only the axes word addresses specified in the G10 block are set. Word addresses not specified in the G10 block remain unchanged.
- WCS setting values are affected by the active G90 Absolute/Incremental mode.
 - If G90 is active, the programmed value replaces the exiting value.
 - If G91 is active, the control algebraically sums the programmed value to the existing value.

4.6.1.1 Programming Example

Setting Example in the G90 Absolute Input mode.

In this example the X and Y fields of EWCS #2 (G54.1 - #2) are set.

- The G90 Absolute Input mode is used so that the existing value will be replaced with the programmed value.

N100 G90 G10 L20 P2 X-100.000 Y-200.000 ;

When N100 is executed, the existing values in the X and Y fields of EWCS #2 are set to X-100.000 and Y-200.000.

Setting Example in the G91 Incremental Input mode.

In this example the X and Y fields of EWCS #3 (G54.1 - #3) are set.

- The G91 Incremental Input mode is used so that the existing value will be updated by the programmed value.

N200 G91 G10 L20 P3 X0.100 Y-0.200 ;

When N200 is executed, the existing values in the X and Y fields of EWCS #3 are updated by the values programmed for X and Y.

Existing Value	Programmed Value	Resulting Value
X150.000	X0.100	X150.100
Y200.000	Y-0.200	Y199.800

4.7 Local Coordinate Systems - G52

A local coordinate system is a coordinate system that is established relative to an active work coordinate system. A local coordinate system is used to temporarily establish a different Program Zero then call a subprogram to run a repetitive operation for a specific part feature.

- A local coordinate system is established by programming a G52 and the appropriate axis coordinates to define Program Zero of the local coordinate system.

G52 Xxxxxx.xxx Yyyyyy.yyy ;

- The values of X and Y are Absolute Coordinates from Program Zero of the active work coordinate system to Program Zero of the local coordinate system.
- A local coordinate system can be used with any of the standard WCS's and EWCS's
- The axis (or axes) move following the G52 block must be made in the G90 mode.
- An active local coordinate system is canceled by programming a G52 and axis coordinate values of zero.

G52 X0 Y0 ;

- Manual reference return and RESET operations also cancel the active local coordinate system.
- If a G92 position set operation is performed while a local coordinate system is active, any axis not programmed in the G92 block is not changed.

4.7.1 Programming Example

Figure 4-3 shows Program Zero established using the standard G54 WCS. Program Zero of the local coordinate system is set by the G52 block.

- Program Zero is established by setting X-300.000 and Y-150.000 values into the standard G54 - #1 WCS.

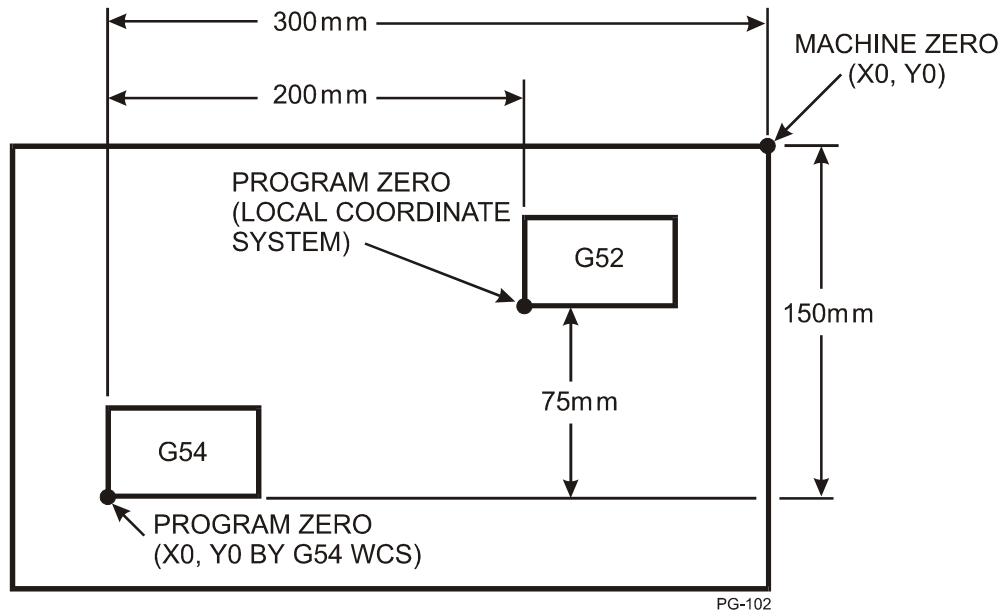


FIGURE 4-3 G52 - LOCAL COORDINATE SYSTEM

To set the local coordinate system

To set the local coordinate system, program the following blocks:

```
Nnnnn G54 ;  
Nnnnn G52 X200.0 Y75.0 ;
```

To cancel the local coordinate system

To cancel the local coordinate system, program the following block:

```
Nnnnn G52 X0.0 Y0.0 ;
```

4.8 Reference Point Positioning

The primary reference point on the machine is the position where the machine axes (X, Y, Z, B, etc.) are located after machine power is turned on and alignment is performed.

- In addition to the primary reference point, a 2nd, 3rd, and 4th reference point can be established.
- All reference points are established relative to Machine Zero by setting values into the X, Y, and Z components of the following control parameters:
 - 1240 = Primary Reference Point
 - 1241 = 2nd Reference Point
 - 1242 = 3rd Reference Point
 - 1243 = 4th Reference Point
- The machine axes can be moved TO any of the four reference points by program command:
 - G27 - Reference Point Return Check (Primary Reference Point)
 - G28 - Reference Point Return (Primary Reference Point)
 - G30 P2 - 2nd Reference Point Return (P2 can be omitted)
 - G30 P3 - 3rd Reference Point Return
 - G30 P4 - 4th Reference Point Return
- In a G30 block, P2, P3, and P4 define the 2nd, 3rd and 4th reference point respectively. It is not necessary to program P2 when positioning to the 2nd reference point.
 - G30, P2 is standard on all A-Series HMC's since the 2nd Reference Point is used as the tool change position.
- The machine axes can be moved away FROM a reference point by a G29 command:
 - G29 - Return From Reference Point
- When moving TO a reference point by G28 or G30, the move is made through a position called an Intermediate Point. Also, when moving away FROM the reference point with G29, the move is made through the Intermediate Point programmed in the G28 or G30 block.

4.8.1 Reference Point Positioning with G28 and G30

Reference Point positioning is performed using G28 and G30.

- G28 moves the axes to Machine Zero
- G30 moves the axes to a 2nd, 3rd, or 4th Reference Point.
- X, Y, and Z commands represent an Intermediate Point. See [section 4.8.1.1](#).
- To avoid overtravel alarm conditions:
 - Tool length compensation MUST be canceled by programming G49 after the G28 or G30 block.
 - Cutter Compensation C MUST be canceled by programming G40 after the G28 or G30 block.
- All Reference Point positioning moves are made non-linearly at the rapid traverse rate.

4.8.1.1 G28 and G30 Intermediate Points

The Intermediate Point is the position (X, Y, and Z axes) that the tool moves through when using G28, G29, or G30.

- The Intermediate Point is programmed with X, Y, and Z coordinates in the G28 or any of the G30 blocks.
- Intermediate Point coordinates can be programmed with absolute (G90) or incremental (G91) values.
 - Incremental values of zero result in no Intermediate Point move, see [section 4.8.1.3](#).



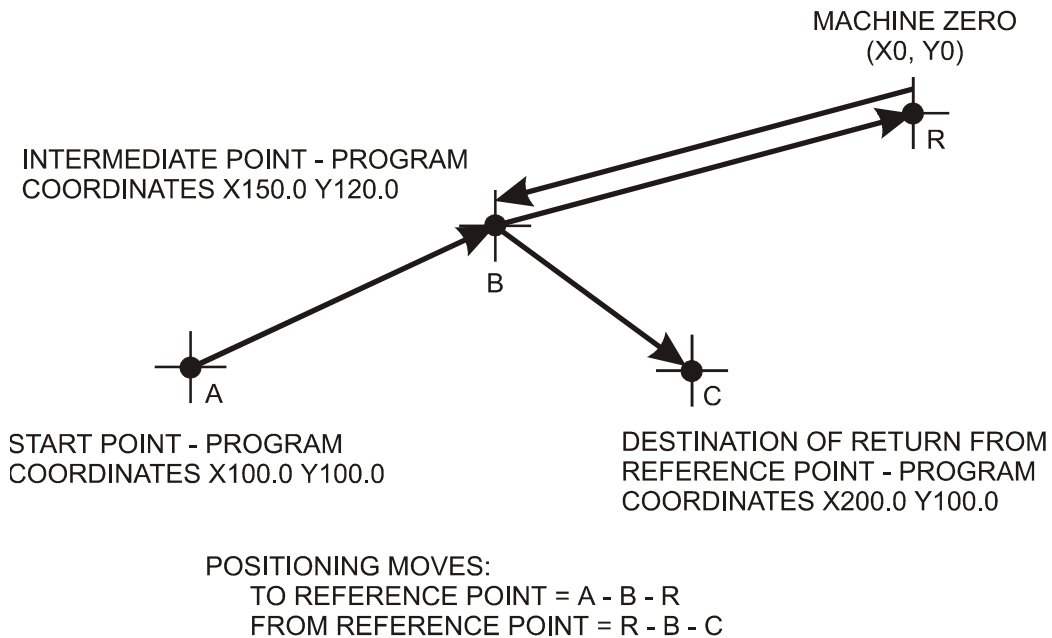
Intermediate Points Are Retained

The Intermediate Point coordinates are retained by the control until new coordinate values are commanded. This could cause a wreck if positioning to a retained Intermediate Point coordinate is not interference free.

4.8.1.2 Programming Example

Figure 4-4 shows the X and Y axes moves of a G28 block when moving to the Reference Point (R). The moves start at Point A, go through Point B, then on to the Reference Point R. The Intermediate Point move is the result of programming absolute coordinates in the G28 block.

Nnnnn G90 G28 X150.0 Y120.0 ;



PG-103

FIGURE 4-4 REFERENCE POINT POSITIONING

G29 Return From Reference Point

When the G29 function is used following a move to the Reference Point by a G28 or G30 block, the first move is from Point R to Point B. The axes then move to Destination Point C specified by the axis commands in the G29 block.

Move to Reference Point (R)

Nnnnn G90 G28 X150.0 Y120.0 ;

Nnnnn ;

Move to Destination Point (C)

Nnnnn G29 X200.0 Y100.0 ;

4.8.1.3 Avoiding Intermediate Point Positioning

In most cases it is not desirable to move through an Intermediate Point when positioning to a reference point.

- To avoid Intermediate Point positioning, program the X, Y, and Z coordinates of the G28 and G30 blocks in the Incremental (G91) mode as follows:

```
N200 G91 G28 X0 Y0 Z0 ;
```

```
N210 G49 ;
```

```
N220 G40 ;
```

- With G91 and coordinate values of zero, the Intermediate Point is programmed but no axis motion is made because the programmed values are zero.

4.9 Positioning from the Reference Point - G29

Positioning FROM the reference point back to the work zone can be done by a block of information containing a G29 and axis coordinates.

- The move FROM the reference point is made through the Intermediate Point that was established in the G28 or G30 block used to move TO the reference point.
- The axis coordinates programmed in the G29 block represents the final position, in the work zone, and are not Intermediate Point coordinates.
- The axis coordinates of the G29 block can be programmed as absolute (G90) or incremental (G91) values.

Movements through the Intermediate Point and to the axis coordinates programmed in the G29 block are made non-linearly at the rapid traverse rate.

4.9.1 Change of Work Coordinate System Offsets

When the machine axes are at a reference point and a different work coordinate system offset is activated:

- The location of the Intermediate Point coordinates will be shifted by the new work coordinate shift offset.
- The destination of the axis coordinates programmed in the G29 block will be shifted by the new work coordinate system offsets.

4.10 Reference Point Return Check - G27

The G27 is used when it is necessary to move to the primary reference point and verify that the machine axes did reach the reference point.

- In the G27 block, axis coordinates are programmed that define the position of the primary reference point. These coordinates do not represent an Intermediate Point as with G28 and G30.
- When the G27 block is executed, the machine axes move to the primary reference point non-linearly at the rapid traverse rate.
 - If all axes reached their primary reference point, the Reference Position lights turn On and the next program block is executed.
 - If an axis does not reach its primary reference point, the respective Reference Position light does not turn On and an alarm is generated.
- The axes coordinates of the G27 block can be programmed in the absolute (G90) or incremental (G91) mode.
- To avoid overtravel alarms:
 - Tool length compensation must be canceled by programming G49 after in the G27 block.
 - Cutter Compensation C must be canceled by programming G40 after the G27 block.

4.11 Floating Reference Point Return - G30.1

The floating reference point return (G30.1) is an optional control feature that provides the capability for the operator to set a position for the X, Y, and Z axes that is positioned to by the G30.1 command. This position can be used to make setup adjustments, tool adjustments, manual tool changes, etc.

- The floating reference point position is set by the operator and retained through machine power off.
- The floating reference point position is in Machine Coordinates. In other words, the position selected is relative to Machine Zero.
- The move to the floating reference point is made non-linearly at the rapid traverse rate through an Intermediate Point programmed by axis commands in the G30.1 block.
- The Intermediate Point coordinates can be programmed with absolute (G90) or incremental (G91) values.
- The machine axes can be moved away FROM the floating reference point by a G29 command:
 - G29 - Return From Reference Point
- To avoid overtravel alarms:
 - Tool length compensation MUST be canceled by programming G49 after the G30.1 block.
 - Cutter Compensation C MUST be canceled by programming G40 after the G30.1 block.

4.11.1 Avoiding Intermediate Point Positioning

In most cases it is not desirable to move through and Intermediate Point when positioning to a floating reference point.

- To avoid Intermediate Point positioning, program the X, Y, and Z coordinates of the G30.1 block in the Incremental (G91) mode as follows:
N200 G91 G30.1 X0 Y0 Z0 ;
N210 G49 ;
N220 G40 ;
- With G91 and coordinate values of zero, the Intermediate Point is programmed but no axis motion is made since the values are zero.

Chapter 5 Dimensional Input

Inch/Metric - G20/G21, Absolute - G90,
and Incremental - G91 Input

Makino Vertical and Horizontal
Machining Centers



Chapter 5 Dimensional Input

Contents

5.1 Overview	5- 1
5.2 Inch/Metric Input Units - G20/G21	5- 2
5.3 Decimal Point Programming	5- 3
5.4 Absolute/Incremental Input - G90/G91	5- 4
5.4.1 Absolute Input - G90	5- 4
5.4.2 Incremental Input - G91	5- 5
5.4.3 Programming Example	5- 5

5.1 Overview

This chapter discusses the use of G20 and G21 for switching dimensional input modes, decimal point programming, and the difference between absolute and incremental input commands.



Information Subject to Change

Every effort was made to ensure the accuracy of the data in this chapter at the time of publication. Machine and control specifications are subject to change without notice.

5.2 Inch/Metric Input Units - G20/G21

The dimensional input unit, inch or millimeters, is established by two G codes:

G20 - Inch Units

G21 - Metric Units (Millimeters)

- The established mode, G20 or G21 remains active through machine power Off. A Data Reset or end of program command (M2 or M30) does not affect the currently active mode.
- The inch/metric mode **MUST NOT** be switched within a program. The control does not convert inches to millimeters (or millimeters to inches). Instead, when the mode is switched, for example, from inch input to metric input, the decimal point is simply moved one place to the right.
 - If the active mode is inch and the X axis is at a current position of X10.0000" and the mode is changed by executing a G21, the current position of X becomes X100.000 mm instead of X254.000 mm which is the metric equivalent of X10.0000".
- The numeric values contained in the offset tables are not converted. Instead, the decimal point of these values is also moved one place to the right or left depending on how the mode is switched (G20 to G21 or G21 to G20).
- Switching from G20 to G21, or G21 to G20 should be done in the MDI mode then all offset values must be converted and input for the current mode.
- If tools are set automatically on the machine, the G20/G21 mode **MUST** be established before the tools are set.
- At machine power On, an AUTO ZERO cannot be performed in the Metric mode until after performing a Manual Reference Return operation.
- After switching modes always "un-align" the machine then perform a Manual Reference Return.

5.3 Decimal Point Programming

The numeric values of certain word addresses can be input with a decimal point. The word addresses that a decimal point can be used with include:

A, B, C, F, I, J, K, Q, R, X, Y, and Z

- The control is setup to use decimal point programming.
- If a value for a word address that uses a decimal point is programmed without a decimal point, the least input increment is assumed.

Inputting an X.1 is processed by the control to be:

X0000.1000 with G20 Inch Input, and
X00000.100 with G21 Metric Input.

Inputting an X1 is processed by the control to be:

X0000.0001 with G20 Inch Input, and
X00000.001 with G21 Metric Input.

- A maximum of eight digits can be programmed for certain word addresses. If the word contains more than eight digits an alarm is generated.
- The control will truncate and round numeric values when a decimal point is used.

Inputting X1.23456 would be truncated and rounded to:

X1.2346 with G20 Inch Input, and
X1.235 with G21 Metric Input.

- The control converts a numeric value input with a decimal point to an integer then checks integer value against the eight digit maximum rule.

Inputting X123456.7 will cause an alarm, even though only seven digits are programmed, since the converted integer value is processed as:

X1234567000 with G20 Inch Input, and
X123456700 with G21 Metric Input.

In both cases above, the eight digit maximum rule is exceeded.

5.4 Absolute/Incremental Input - G90/G91

Programmed coordinates can be input as Absolute or Incremental values.

5.4.1 Absolute Input - G90

G90 Absolute Input is a modal code, established at machine power ON and canceled only by G91 Incremental Input. G90 is also established when:

- RESET is pressed, or an
- M2 or M30 end of program command is executed.
- G90 and G91 can be switched within the programmed.
- G90 Absolute Input means that all programmed coordinates are calculated from a single point (Program Zero).
- The sign, plus (+) or minus (-), of an absolute coordinate does not indicate a direction of axis travel. It indicates the position of the coordinate relative to Program Zero (the Cartesian Coordinate System).
 - X+ and Y+ is in the 1st Quadrant
 - X- and Y+ is in the 2nd Quadrant
 - X- and Y- is in the 3rd Quadrant
 - X+ and Y- is in the 4th Quadrant



G90/G91 Default Parameter Settings

Pro 3 control parameter 3402, Bit 3 establishes G90 or G91 at machine power on. Setting Bit 3 = 0 selects the G90 Absolute Mode and setting Bit 3 =1 selects G91 Incremental Mode.

5.4.2 Incremental Input - G91

G91 Incremental Input is a modal code canceled only by G90 Absolute Input. G91 is also reset to G90 at machine power On and when:

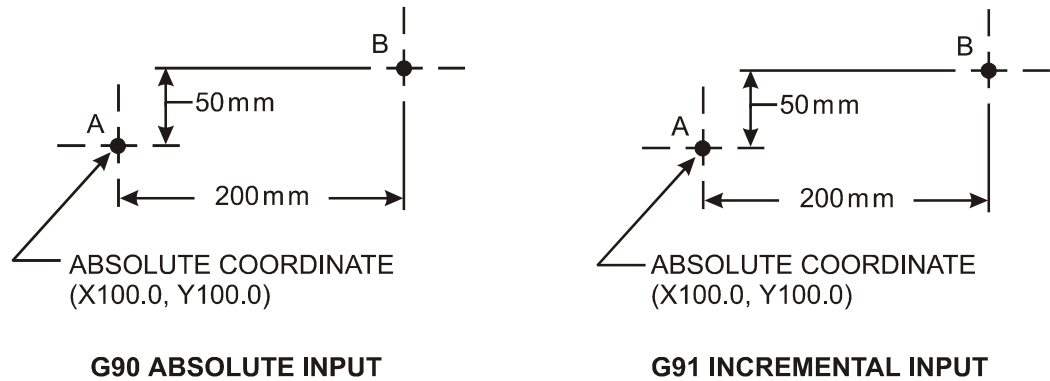
- RESET is pressed, or an
- M2 or M30 end of program command is executed.
- G91 and G90 can be switched within the programmed.
- G91 Incremental Input means that the programmed coordinate(s) is calculated from the current position to the target position.
- The signs, plus (+) and minus (-), denote the direction of axis travel to get to the target position.
- G90 and G91 can both be programmed in the same block, the code that is programmed last will be modal in succeeding blocks.

G00 G90 Xx.xxx Yy.yyy G91 Zz.zzz ;

X and Y are absolute and Z is incremental.

5.4.3 Programming Example

Figure 5-1 shows X and Y move commands using G90 Absolute and G91 Incremental Input modes.



MOVE COMMANDS FROM POINT A to POINT B

WITH G90 ABSOLUTE INPUT ACTIVE
G0 G90 X300.0 Y150.0 ;

WITH G91 INCREMENTAL INPUT ACTIVE
G0 G91 X200.0 Y50.0 ;

PG-104

FIGURE 5-1 G90/G91 PROGRAMMING EXAMPLE

Chapter 6 M, S, and T Functions

Miscellaneous (M) Codes, Spindle
Commands, and Tool Control

Makino Vertical and Horizontal
Machining Centers



Chapter 6

M, S, and T Functions

Contents

6.1 Overview	6-1
6.2 Program Control M Codes	6-2
6.2.1 M0 - Program Stop	6-2
6.2.2 M1 - Optional Program Stop	6-2
6.2.3 M2 - End Of Program	6-2
6.2.4 M30 - End Of Program - Reset/Rewind	6-3
6.2.5 M98 - Sub Program Call	6-3
6.2.6 M99 - End of Sub Program	6-3
6.3 Spindle and Coolant Control - S and M Codes	6-4
6.3.1 Spindle Speed Control - S Code	6-4
6.3.2 Spindle Start and Stop - M Codes	6-4
6.3.3 Calculating Spindle Speeds	6-5
6.3.3.1 Calculation for G20 Inch Input	6-5
6.3.3.2 Calculation for G21 Metric Input	6-5
6.3.4 Spindle Speed Override - M Codes	6-6
6.3.5 Coolant Control - M Codes	6-6
6.4 Tool Management	6-7
6.4.1 Tool Storage and Changing	6-7
6.4.2 Tool Search, Change, and Storage Sequence	6-8
6.4.3 Tool Length Offset - H and Tool Radius Offset - D	6-9
6.4.3.1 Transferring H and D-Values from the Main Program	6-11
6.4.4 T Code, M6 Code, and Tool Change Programs	6-12
6.4.4.1 T Code	6-12
6.4.4.2 M6 Code	6-12
6.4.4.3 Tool Change Programs	6-13
6.4.5 Activating and Cancelling Offsets - G43, G44, G49	6-16
6.4.5.1 Programming Example	6-16
6.4.6 Tool Change Programming Examples	6-18

6.4.6.1 Tool Changes for Like Parts 6-18
6.4.6.2 Tool Changes for Unlike Parts 6-19

6.1 Overview

This chapter discusses the following functions relative to M, S, and T codes:

- The use of M codes for program control, coolant control, and tool changes.
- Spindle speed commands.
- Custom Side Tool Management of magazine pot numbers, T codes, and tool length (H codes) and tool radius (D codes) offsets.



Information Subject to Change

Every effort was made to ensure the accuracy of the data in this chapter at the time of publication. Machine and control specifications are subject to change without notice.

6.2 Program Control M Codes

Certain M codes are used to control program execution. These codes are:

M0 - Program Stop

M1 - Optional Program Stop

M2 - End Of Program

M30 - End Of Program - Reset/Rewind

M98 - Sub Program Call

M99 - End of Sub Program

6.2.1 M0 - Program Stop

- Stops spindle rotation and coolant flow.
- Effective at the end of the block in which it is programmed.
- Cycle is resumed by pressing CYCLE START.

6.2.2 M1 - Optional Program Stop

- Effective only when the OPTIONAL STOP button is On.
- Stops spindle rotation and coolant flow.
- Effective at the end of the block in which it is programmed.
- Cycle is resumed by CYCLE START.

6.2.3 M2 - End Of Program

- Stops spindle rotation and coolant flow.
- Effective at the end of the block in which it is programmed.
- Signals the end of the program.
- Does not rewind the tape if using a tape reader. However, it does reset the cursor to the beginning of the program.

6.2.4 **M30 - End Of Program - Reset/Rewind**

- Stops spindle rotation and coolant flow.
- Effective at the end of the block in which it is programmed.
- Signals the end of the program.
- Realigns the program to its beginning.
- Initiates a pallet change if Random Pallet mode is active.
- Rewinds the tape if a tape reader is being used.

6.2.5 **M98 - Sub Program Call**

- Stops execution of the main program (or Sub Program if M98 is calling another Sub Program) and starts execution of the Sub Program specified by the P code programmed in the M98 block.

6.2.6 **M99 - End of Sub Program**

- Stops execution of the current Sub Program and returns program execution back to the calling program.
- If the current Sub Program was called from another Sub Program, program execution returns to the calling Sub Program.
- Program execution can be restarted in the calling program at a specific sequence number (Nxxxxx) specific by a P code (Pxxxxx) in the M99 block.

6.3 Spindle and Coolant Control - S and M Codes

The machine spindle is controlled by a combination of the programmed S and M codes. Coolant for the cutting tool is also controlled by an M code.

6.3.1 Spindle Speed Control - S Code

Spindle speeds are programmed with a five digit S code, in one rpm increments.

Ssssss - spindle speed in rpm

- The S code is programmed without a decimal point and leading zeros can be omitted.
- The spindle speed ranges vary by machine model and available spindle options.
- Actual spindle starting and stopping is controlled by an M code.
- Once the spindle is started, a new rpm can be programmed, to change the spindle speed, without programming another spindle start M code.

6.3.2 Spindle Start and Stop - M Codes

Spindle start, direction of rotation and stop is controlled by M codes.

M3 - Spindle On Clockwise - used for right-hand tools.

M4 - Spindle On Counterclockwise - used for left-hand tools.

M5 - Spindle Off - this code is seldom used since most program control M codes (such as M0, M1, M2, M30, etc.) automatically stops spindle rotation.

- A spindle On command must be programmed following any M code that automatically stops spindle rotation, such as M6 Tool Change, since the spindle does not automatically restart.
- M3 and M4 are effective at the beginning of the block in which they are programmed.
- M5 is effective at the end of the block in which it is programmed.

6.3.3 Calculating Spindle Speeds

The speed at which a given tool should be run is based on several things; the tool diameter, the tool material, the type of machining (drilling, tapping, reaming, milling, etc.) and the surface cutting speed of the material that is being machined.

- The tool manufacturer generally supplies recommended surface cutting speeds for the tool.
- Material surface cutting speeds can be found in machining handbooks and data sheets provided by the material supplier.
- Other things to consider include; the rigidity of the tool and part setup, the surface finish that must be maintained and tool life.

6.3.3.1 Calculation for G20 Inch Input

$$\text{RPM} = (3.82 \times \text{SFM}) \div \text{DIAM}$$

Where:

3.82 = a constant ($12 \div 3.14159$)

SFM = material cutting speed in surface feet per minute

DIAM = tool diameter (or largest diameter being machined with the tool)

6.3.3.2 Calculation for G21 Metric Input

$$\text{RPM} = (318.3 \times \text{SMM}) \div \text{DIAM}$$

Where:

318.3 = a constant ($1000 \div 3.14159$)

SMM = material cutting speed in surface meters per minute

DIAM = tool diameter (or largest diameter being machined with the tool)

6.3.4 Spindle Speed Override - M Codes

Two M codes are used to control whether or not the Spindle Speed Override switch is functional.

M58 - M59 Mode Cancel (Override Switch Functional)

M59 - Spindle Speed Override Cancel (Override Switch Not Functional)

- M58 and M59 are effective at the beginning of the block in which they are programmed.
- M59 is canceled, and M58 becomes active, when:
 - Machine power is turned On
 - RESET is pressed
 - An end of program code (M2 or M30) is executed
- The override range is 50% to 120% of the programmed spindle speed.

6.3.5 Coolant Control - M Codes

Coolant start and stop is controlled by M codes. In the following list of M codes, an * indicates the coolant feature is optional equipment.

M7* - Mist Coolant or Air Blow On

M8 - Nozzle Coolant On.

M9 - Coolant Off. This code is seldom used since most program control M codes (such as M0, M1, M2, M30, etc.) will automatically stop coolant flow.

M26* - Through Spindle Coolant On

M27* - Tap Oil (mist, jet) On

M96* - Through Tool Coolant On

M97* - High Power Coolant On

- The appropriate M code to start coolant flow must be programmed following any M code that automatically turns coolant off, such as M0.
- Coolant On codes are effective at the beginning of the block in which they are programmed.
- M9 is effective at the end of the block in which it is programmed.

6.4 Tool Management

This Tool Management description is based on the Pro 3 Custom Side Tool Management feature and includes:

- Tool Storage and Changing
- Sequence of Tool Search, Change, and Storage
- Tool Length Offset - H code
- Tool Radius Offset -D code
- T code, M6 code, and Tool Change Programs
- Activating and Canceling Offsets - G43, G44, G49
- Tool Change Programming Examples

During NC program execution, the control always operates from the NC Side. However, all tool data is stored on the Custom Side. By a tool change program, tool data is transferred from the Custom Side to the NC Side when a tool change is executed. This simplifies programming since unique H and D codes are not required in the program. Operation is also simplified since all data relative to a programmed tool number is displayed on the TOOL DETAIL screen.

6.4.1 Tool Storage and Changing

Tools used by the NC program are stored in the ATC Magazine until called by executing a T code followed by an M6 command.

The number of tools that a magazine will hold varies by machine model and available options.

- On horizontal machines, the tools are always returned to the same pocket number in the magazine. This is called sequential tool storage. Programmed tool numbers are assigned to magazine pocket numbers on the Custom Side - TOOL DETAIL screen.
- On vertical machines, when a new tool is removed from the magazine the old tool is returned to same magazine pocket from which the new tool was removed. This is called random tool storage. Programmed tool numbers are assigned to magazine pocket numbers on the Custom Side - TOOL DETAIL screen.

6.4.2 Tool Search, Change, and Storage Sequence

The following is the sequence of events that occur when the T code and M6 code are executed. The sequence of events varies depending on whether the machine is a horizontal or vertical.

- **Sequence of events for Horizontal machines:**

- When the T code is executed:

The magazine is searched for the new tool

The ATC Arm removes the new tool from the magazine and moves it to the Ready Position

The new tool is held at the Ready Position until the M6 is executed

- When the M6 is executed:

The tool exchange occurs between the new tool and the old tool in the spindle

The next program block is executed

The magazine is searched for the pocket number for the old tool

The old tool is returned to the magazine

- **Sequence of events for Vertical Machines:**

- When the T code is executed:

The magazine is searched for the new tool (this is the ready position on vertical machines)

- When the M6 is executed:

The tool exchange occurs between the new tool and the old tool in the spindle

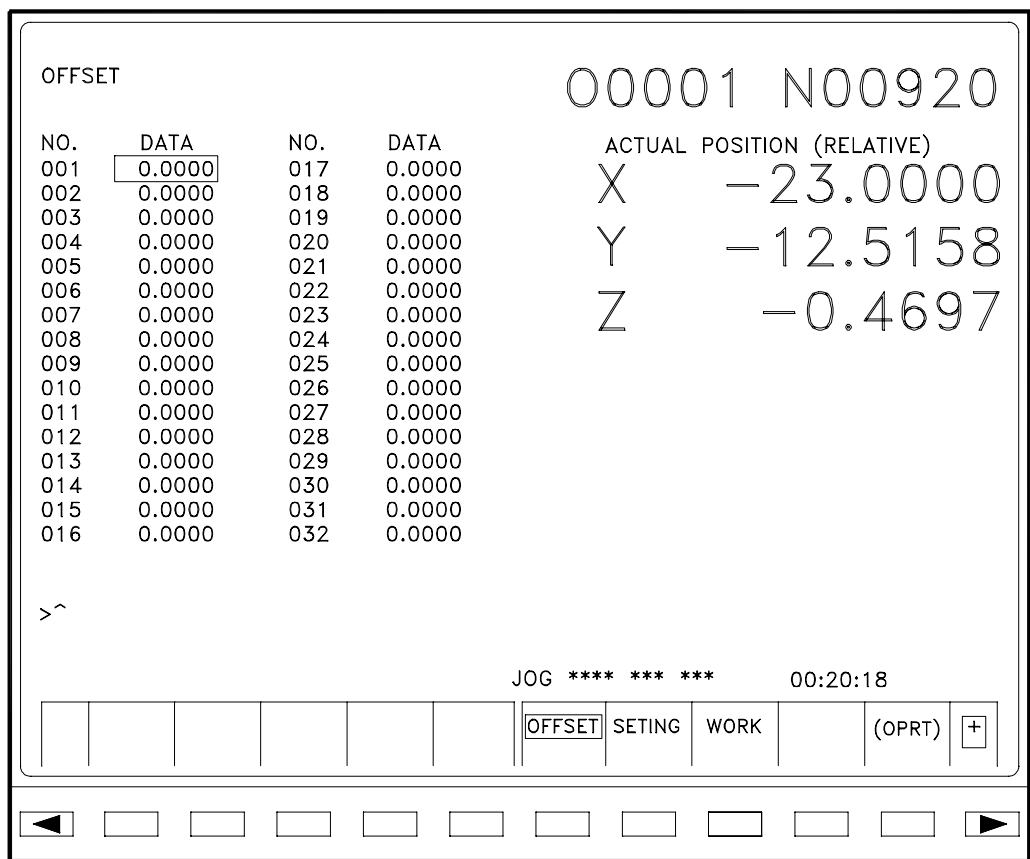
The old tool is returned to the magazine in the same pocket vacated by the new tool

The next program block is executed

6.4.3 Tool Length Offset - H and Tool Radius Offset - D

Without the Custom Side Tool Management feature, unique H and D codes are required for each tool used in the NC program. These H and D codes represent NC Side tool offset numbers and are programmed in the range of 01 to 99. See Figure 6-1.

- The H code represents a tool length offset
- The D code represents a tool radius offset
- The offset values for the H and D codes are entered on the NC Side - OFFSET screen.



3CNCI2003A

FIGURE 6-1 NC SIDE - OFFSET SCREEN

With the Custom Side Tool Management feature and a tool change program, unique H and D codes are not required in the part program.

- Tool length and radius offset values are stored on the Custom Side - TOOL DETAIL screen.
- On the TOOL DETAIL screen, PTN's (programmed tool numbers) are assigned to the magazine pocket numbers in the PTN column. Adjacent to the PTN column are the H-Value and D-Value columns. See Figure 6-2.
- Tool length offset values are entered in the H-Value column for the PTN (the T code in the NC program).
- Tool radius values are entered in the D-Value column for the PTN.
- The values entered in the H and D columns for a PTN are automatically transferred to the NC Side when the tool change program is executed by the M6 code.

TOOL DETAIL									1/2
POT	PIN	KIND	FTN	LIFE	REMAIN	H-VALUE	D-VALUE	STATUS	
SP 1	<input type="text" value="1"/>	1	1	1	1	0.	0.		
HX									
1	1		1	1	1	0.	0.		
2	2		0	1	1	0.	0.		
3	3		0	0	0	0.	0.		
4	4		0	0	0	0.	0.		
5	5		0	0	0	0.	0.		
6	6		0	0	0	0.	0.		
7	7		0	0	0	0.	0.		
8	8		0	0	0	0.	0.		
9	9		0	0	0	0.	0.		
10	10		0	0	0	0.	0.		
11	11		0	0	0	0.	0.		
12	12		0	0	0	0.	0.		
13	13		0	0	0	0.	0.		
14	14		0	0	0	0.	0.		
15	15		0	0	0	0.	0.		
16	0		0	0	0	0.	0.		
17	0		0	0	0	0.	0.		
INPUT=									
P	<input type="text" value="INPUT"/>	DISPLY						POT NO	
O	DISPLY	SHIFT						SEARCH	
								PIN NO	
								SEARCH	

3CUSI3001A

FIGURE 6-2 CUSTOM SIDE - TOOL DETAIL SCREEN

6.4.3.1 Transferring H and D-Values from the Main Program

Normally, the M56 is not used in the main program except in a special case when the tool is already in the spindle at the start of the program. In this case, the T code is not activated by the M6. Therefore, to make the necessary tool length and/or tool radius offsets active, the H and D-Values stored on the Custom Side must be transferred to the NC Side by the following sequence of commands:

Program Block	Block Description
N100 T1 ;	Tool number of tool in the spindle
N110 G90 H1 D2 ;	Preps NC Side tool offset numbers 1 and 2 for transfer of values from Custom Side
N120 M56 ;	Transfers H and D-Values of T1 from Custom Side to NC Side
N130 T2 ;	Readies next tool
N140 G0 X__ Y__ S__ M03 ;	Move to machining start position
N150 G0 G43 H1 Z50.0 ;	Activate tool length offset and move Z to safe start position

Note:

T1 is an example representing the first tool in the program. This T code can be any number for the first tool. H1 and D2 are the same H and D codes used in the tool change program.

6.4.4 T Code, M6 Code, and Tool Change Programs

6.4.4.1 T Code

The T code is the PTN (Programmed Tool Number) and is independent of magazine pot numbers. PTN's are assigned to magazine pot numbers (POT #) on the Custom Side - TOOL DETAIL screen.

The T code is programmed in the range of:

T1 through T9999 (or T1 through T99999999 optional)

- The T code should be programmed in a block by itself
- The T code readies the tool for the tool change.

6.4.4.2 M6 Code

M6 in the main program calls the tool change program that performs several functions besides the actual tool exchange. The M6 in the tool change program causes the actual tool change.

- The M6 should be programmed in a block by itself with the exceptions:
 - A sequence number (Nxxxxx)
 - A comment statement to identify the tool going into the spindle

Program Example:

```
N100 T10 ;
```

```
N110 M6 (0.50 DIAM. DRILL) ;
```

6.4.4.3 Tool Change Programs

The Tool Change Program is a protected Sub Program stored in memory and called by the M6 command in the main program. The tool change program varies slightly depending on the type of machine.

Tool Change Program for A Series Horizontal Machining Centers.

Program Block	Block Description
O9001 M9 ;	O9001 is a protected Sub Program. M9 turns all coolant off.
G00 G28 G80 G91 Z0 M19 ; G40 G49 G30 X0 Y0 M6 ;	Reference Z axis. Orient stop the spindle. Cancel cutter and tool length compensation. Rapid X and Y to 2 nd reference position. M6 causes the tool exchange to occur.
G90 H1 D2 ;	Preps NC Side tool offset numbers 1 and 2 to accept absolute offset values from the Custom Side.
M56 ;	Transfers the values set in H and D on the Custom Side to the NC side, see Note below.
M99 ;	Ends the Tool Change Sub Program

Note:

The tool length offset (H-Values) and tool radius (D-Values) for a tool are stored on the Custom Side. When the Tool Change Program is executed, M56 transfers the values of H and D from the Custom Side to the NC Side. Any two numbers in the range of 1 to 99 can be used for H and D, H1 and D2 are typically used.

Tool Change Program for all other Horizontal and Vertical Machining Centers.

Program Block	Block Description
O9001 M9 ;	O9001 is a protected Sub Program. M9 turns all coolant off.
G00 G28 G80 G91 Z0 M19 ; G40 G49 G28 Y0 M6 ;	Reference Z axis. Orient stop the spindle. Cancel cutter and tool length compensation. Reference Y axis. M6 causes the tool exchange to occur.
G90 H1 D2 ;	Preps NC Side tool offset numbers 1 and 2 to accept absolute offset values from the Custom Side.
M56 ;	Transfers the values set in H and D on the Custom Side to the NC side, see Note below.
M99 ;	Ends the Tool Change Sub Program

Note:

The tool length offset (H-Values) and tool radius (D-Values) for a tool are stored on the Custom Side. When the Tool Change Program is executed, M56 transfers the values of H and D from the Custom Side to the NC Side. Any two numbers in the range of 1 to 99 can be used for H and D, H1 and D2 are typically used.

6.4.5 Activating and Cancelling Offsets - G43, G44, G49

Tool length offsets are activated and cancelled by G codes, as follows:

G43 - Tool Length Compensation + Direction

G44 - Tool Length Compensation - Direction

G49 - Tool Length Compensation Cancel

- The + and - direction of G43 and G44 are relative to the + and - direction of the Z axis.
- G43, or G44, is programmed in the block with the first Z axis move to the part following a tool change.
- G49 is used whenever it is necessary to cancel tool length offsets.

6.4.5.1 Programming Example

Makino machining centers are programmed using G43 to activate tool length offset in the plus (+) direction. This requires that the Z Work Coordinate Offset be entered as a minus (-) value.

Figure 6-3 shows two examples; one with Z0 (Program Zero) established at the Pallet centerline, and the other with Z0 (Program Zero) established at the part face.

Using Figure 6-3 as an example, the program commands to move Z to the Safe Start Position are as follows:

Program Zero at Pallet centerline

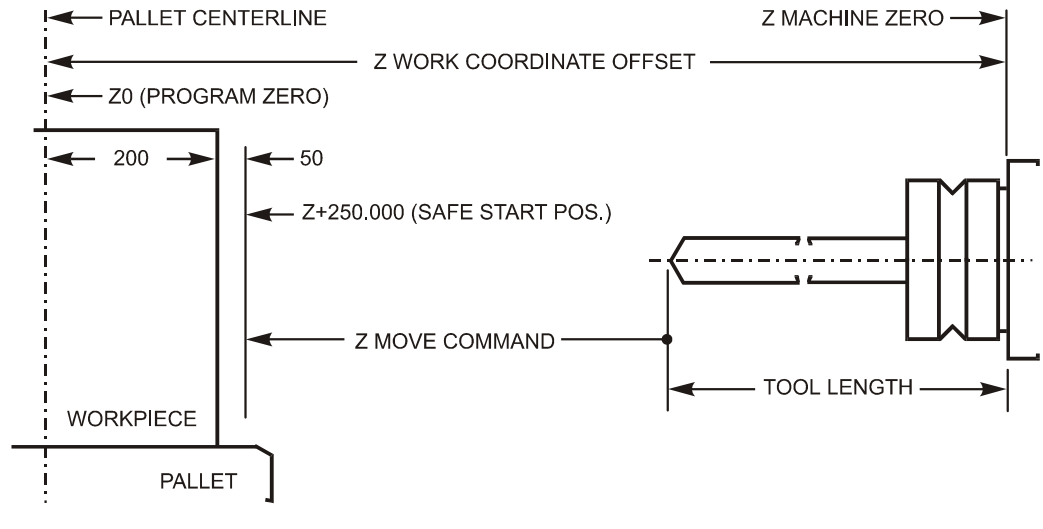
Nnnnn G54 G43 G0 Z250.0 H1 ;

Program Zero at part face

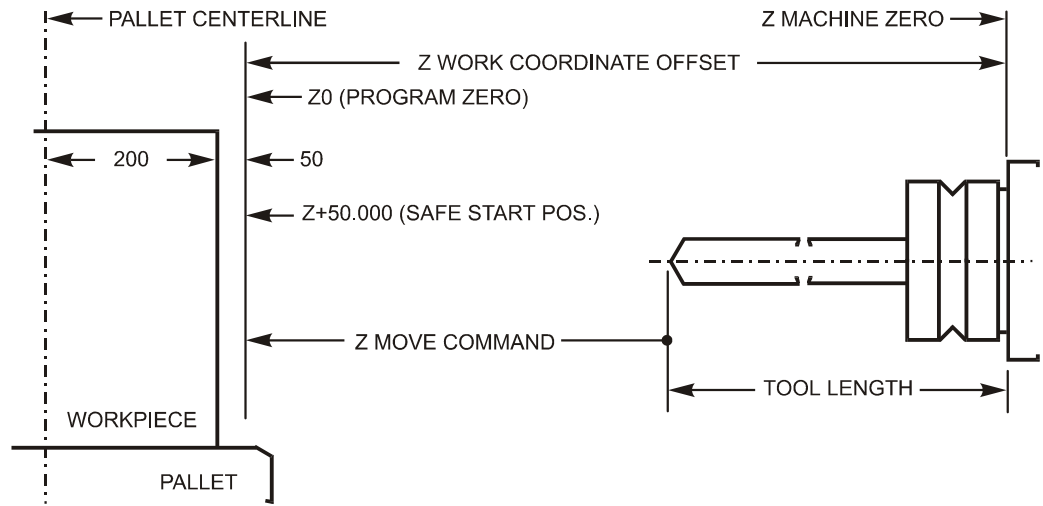
Nnnnn G54 G43 G0 Z50.0 H1 ;

In both examples, G54 activates the Work Coordinate Offset and G43 activates the Tool Length Offset. H1 is required to specify the offset number where the Tool Length Offset is stored. In both cases, the tool stops at the same position.

Z0 AT PALLET CENTERLINE



Z0 AT PART FACE



PG-111

FIGURE 6-3 ACTIVATING TOOL LENGTH OFFSETS

6.4.6 Tool Change Programming Examples

The following tool change programming examples are based on two conditions:

- Tool changes for like parts
- Tool changes for unlike parts

6.4.6.1 Tool Changes for Like Parts

When the part program is to be repeated (Like Parts) the first tool used by the program will always be the same tool. In this case, the very first time the program is run, the first tool used by the program must be manually loaded into the spindle. The part program automatically loads the first tool into the spindle at the end of the program.

Program Block	Block Description
N... ;	
N100 T1 (T1 - 4" F. MILL) ;	Activates T code
N110 G90 H1 D2 ;	Required since M6 does not load first tool
N120 M56 ;	Transfers tool data from Custom Side to NC Side
N130 T2 ;	T2 to ready position
N... G0 X__ Y__ S__ M__ ;	Move to machining position and start spindle
N140 G0 G43 H1 Z50.0 ;	Activate tool length offset, move Z to safe start position
N... ;	Machining with T1
N200 M6 (T2 - C. DRILL) ;	Loads T2 in spindle
N210 T3 ;	T3 to ready position
N... G0 X__ Y__ S__ M__ ;	Move to machining position and start spindle
N220 G0 G43 H1 Z50.0 ;	Activate tool length offset, move Z to safe start position
N... ;	Machining with T2
N... ;	
N300 M6 (T3 - .5 DRILL) ;	Loads T3 in spindle
N310 T1 ;	T1 to ready position
N... G0 X__ Y__ S__ M__ ;	Move to machining position and start spindle
N320 G0 G43 H1 Z50.0 ;	Activate tool length offset, move Z to safe start position
N... ;	Machining with T3
N400 M6 (T1 - 4" F. MILL) ;	Exchanges T3 with T1
N410 M30 ;	End of program

6.4.6.2 Tool Changes for Unlike Parts

When the next part program to be run is a different program (Unlike Parts), the first tools used by the two programs will most likely be different tools. In this case, the first tool used by a program should be loaded into the spindle by the program. This means that a tool must not be in the spindle at the start of the program.

Program Block	Block Description
N... T0 ;	T0 must be assigned a pocket number
N100 T1 (T1 - 4" F. MILL) ;	T1 to ready position
N110 M6 ;	Loads T1 in spindle
N120 T2 ;	T2 to ready position
N... G0 X___ Y___ S___ M___ ;	Move to machining position and start spindle
N130 G0 G43 H1 Z50.0 ;	Activate tool length offset, move Z to safe start position
N... ;	Machining with T1
N... ;	
N200 M6 (T2 - C. DRILL) ;	Loads T2 in spindle
N210 T3 ;	T3 to ready position
N... G0 X___ Y___ S___ M___ ;	Move to machining position and start spindle
N220 G0 G43 H1 Z50.0 ;	Activate tool length offset, move Z to safe start position
N... ;	Machining with T2
N... ;	
N300 M6 (T3 - .5 DRILL) ;	Loads T3 in spindle
N310 T0 ;	T0 to ready position
N... G0 X___ Y___ S___ M___ ;	Move to machining position and start spindle
N320 G0 G43 H1 Z50.0 ;	Activate tool length offset, move Z to safe start position
N... ;	Machining with T3
N... ;	
N400 M6 (T0 - Dummy Tool) ;	Loads T0 in spindle
N410 M30 ;	End of program

NOTES:

SKETCHES:

Chapter 7 Interpolation Functions

Positioning, Linear and Circular Interpolation, Corner Chamfering and Rounding, Polar Coordinate Input

Makino Vertical and Horizontal Machining Centers



Chapter 7 Interpolation Functions

Contents

7.1 Overview	7-1
7.2 Rapid Positioning - G0	7-2
7.3 Linear Interpolation - G1	7-3
7.4 Circular Interpolation - G2 and G3	7-5
7.4.1 Direction of G2 and G3	7-6
7.4.2 Arc Center Point Programming with I, J, K	7-8
7.4.3 Arc Center Point Programming with R	7-9
7.4.4 Circular Interpolation Programming Example	7-10
7.5 Angle Chamfering and Corner Rounding	7-11
7.6 Polar Coordinate Input - G15 and G16 (Option)	7-13
7.7 Dwell - G4	7-18

7.1 Overview

Interpolation functions are the methods of moving the tool about the work piece, generally for milling operations. Included in the discussion of interpolation are the following modes of operation:

- Rapid Positioning - G0
- Linear Interpolation - G1
- Circular Interpolation - G2 and G3
- Angle Chamfering and Corner Rounding
- Polar Coordinate Input - G15 and G16
- Dwell - G4

7.2 Rapid Positioning - G0

When G0 is executed, the programmed axes move at their own rapid traverse rate independently of each other. The tool path, therefore, becomes non-linear rather a straight line.

Pro 3 control parameter - 1401 - Bit 1 = 0 establishes G0 non-linear positioning.

- G0 is the initialized state of Group 01 G codes.
- G0 is a modal code that cancels all other modes of interpolation.
- Rapid traverse rates vary by machine model.
- Any one axis, or all axes, can be commanded in the G0 block.
- Axis commands can be Absolute - G90, or Incremental - G91 values.
- Rapid traverse is overridden by the [RAPID OVERRIDE] switch on the Main Operation Panel.

7.3 Linear Interpolation - G1

When G1 is executed, the programmed axes move at the feed commanded by the F word. When more than one axis is programmed in the G1 block, the tool path is linearly interpolated which means that the tool path will be a straight line.

- G1 is a Group 01 G code.
- G1 is a modal code that cancels all other modes of interpolation.
- The maximum programmable feedrate varies by machine model.
- Any one axis, or all axes, can be commanded in the G1 block.
- Axis commands can be Absolute - G90, or Incremental - G91 values.
- Feedrates are overridden by a [FEED RATE OVERRIDE] switch on the Main Operation Panel.

7.4 Circular Interpolation - G2 and G3

When a G2 or G3 is executed, the programmed axes move in a circular motion at the feedrate commanded by the F word. G2 directs tool path movement in the clockwise direction and G3 directs tool path movement in the counterclockwise direction.

Circular motion is about a center point defined by a combination of two words (the words being I, J, and K depending on the active Plane Select mode) or by an R word that defines the radius being machined.

- G2 and G3 are Group 01 G codes.
- G2 and G3 are modal codes that cancel each other and all other interpolation modes.
- The maximum programmed feedrate varies by machine model.
- Axis commands can be Absolute - G90, or Incremental - G91 values.
- Feedrates are overridden by a [FEED RATE OVERRIDE] switch on the Main Operation Panel.
- Only two axes can be circular interpolated at one time. The two axes that can be interpolated together is controlled by the active Plane Select mode (G17, G18, or G19).

When G17 - X Y Plane Select is active:

The X and Y axes are interpolated together.

- The arc center is defined by I and J words. I represents the X axis and J represents the Y axis.
- R can be used instead of I and J.

When G18 - Z X Plane Select is active:

The Z and X axes can be circular interpolated together.

- The arc center is defined by K and I words. K represents the Z axis and I represents the X axis.
- R can be used instead of K and I.

When G19 - Y Z Plane Select is active:

The Y and Z axes can be circular interpolated together.

- The arc center is defined by J and K words. J represents the Y axis and K represents the Z axis.
- R can be used instead of J and K.

7.4.1 Direction of G2 and G3

G2 directs circular interpolation in the clockwise direction and G3 directs circular interpolation in the counterclockwise direction.

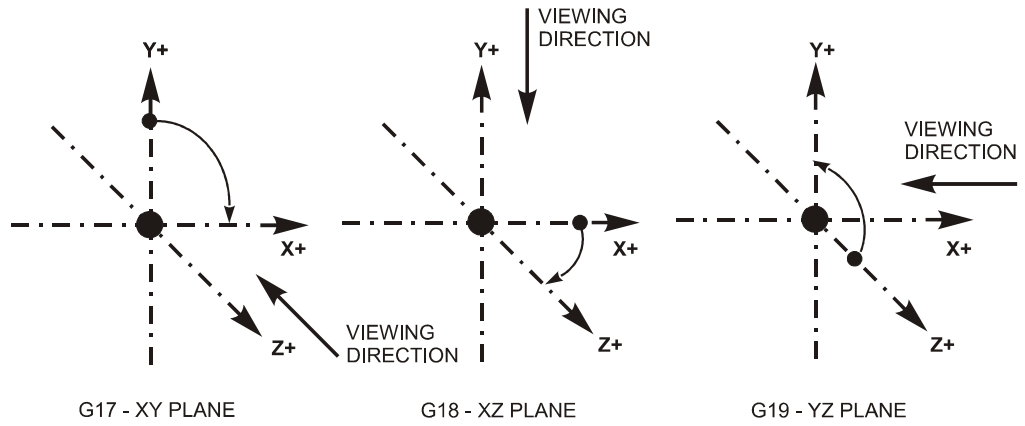
- The direction of G2 and G3 is determined by viewing the two axes that are being circular interpolated from the other axis in a positive to negative direction. See Figure 7-1.

For G17 - XY Plane, the X and Y axes are viewed from Z+ toward Z-.

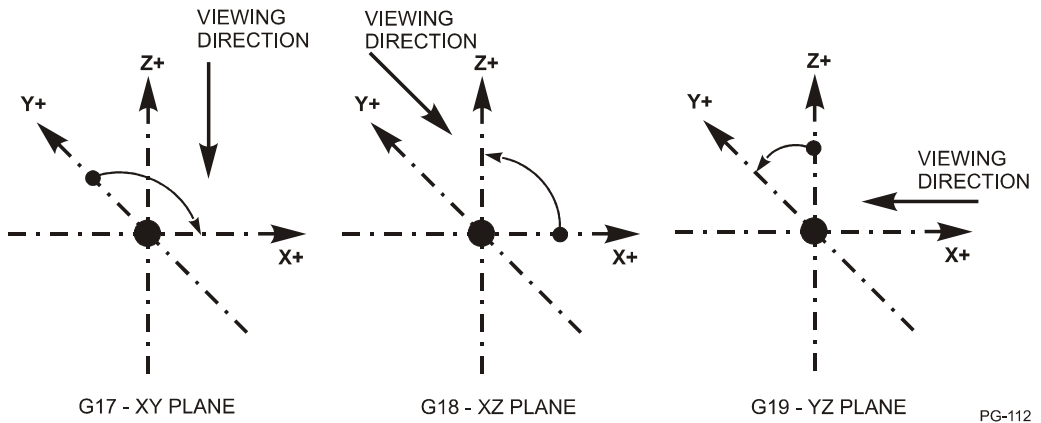
For G18 - ZX Plane, the Z and X axes are viewed from Y+ toward Y-.

For G19 - YZ Plane, the Y and Z axes are viewed from X+ toward X-.

DIRECTION OF G2 FOR HORIZONTAL MACHINES



DIRECTION OF G2 FOR VERTICAL MACHINES



PG-112

FIGURE 7-1 DIRECTION OF TRAVEL FOR CIRCULAR INTERPOLATION

7.4.2 Arc Center Point Programming with I, J, K

The center point of the arc being machined is defined by I, J, K words for the axes X, Y, Z, respectively.

- The I, J, and K words that are used is dependent upon the active G17, G18, or G19 Plane Select mode.
 - For G17 - XY Plane, I represents X and J represents Y.
 - For G18 - ZX Plane, K represents Z and I represents X.
 - For G19 - YZ Plane, J represents Y and K represents Z.
- The arc center point coordinates are incremental values measured from the arc start point to its center point. If the direction from the arc start point to the center point is negative, the minus (-) sign must be programmed.
- An arc center point coordinate that is equal to zero (I0, J0, or K0) can be omitted.
- When the arc end point is at the same coordinate location as its start point (a 360° circle), the end point coordinates are omitted and only the center point coordinates are programmed with G2 or G3.

Program Block

N100 G17 ;

N110 G2 I-xx.xxx ;

Block Description

Sets the XY Plane Select mode. Not required unless G18 or G19 is active.

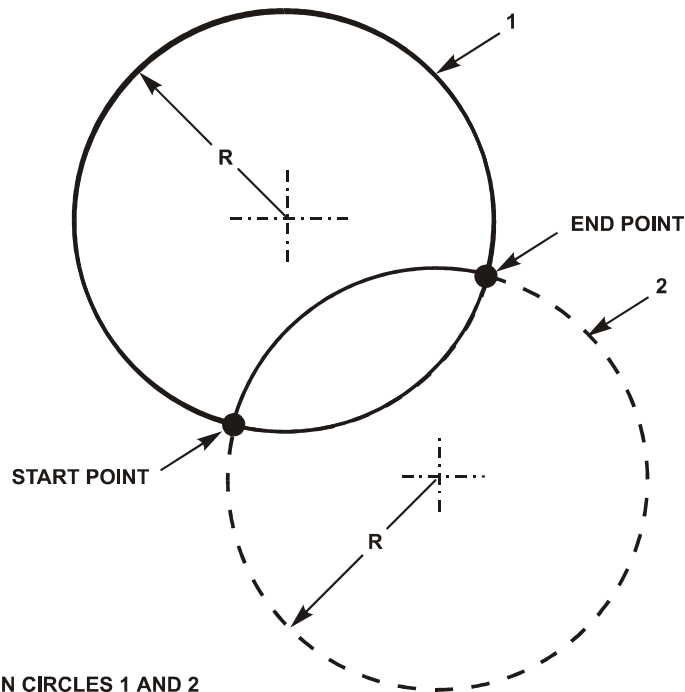
G2 sets the circular clockwise direction. I-xx.xxxx is an incremental distance from the arc start point to its center point.

The J center point coordinate for the Y axis is omitted since the arc start and center points are located at the same Y coordinate.

7.4.3 Arc Center Point Programming with R

The center point of the arc being machined can be defined by an R word that represents the radius of the arc being machined.

- The R word is a positive (+) value when the arc is less than 180°.
- The R word is a negative (-) value when the arc is greater than 180°.
- If the I, J, K and R words are programmed in the same block, the R word is used and the other commands are ignored.
- If an axis word X, Y, or Z is commanded in the G2 or G3 block that does not comply with the active Plane Select mode an alarm is generated.
- Full circle arcs (360°) must be programmed with the appropriate combination of I, J, and K words.



R = THE SAME IN CIRCLES 1 AND 2

PG-113

FIGURE 7-2 ARC CENTER POINT - R WORD

7.4.4 Circular Interpolation Programming Example

Figure 7-3 shows an example of circular interpolation using Cutter Compensation. In this example:

- The cutter moves to the Arc Start Point, through the Intermediate Point, then to the Arc End Point.
- The Arc Center Point is defined using I and J and is measured from the Arc Start Point to the Arc Center Point.
 - I and J are both incremental, values.
- With Cutter Compensation, X, Y, I, and J are calculated on the part surface versus the tool path centerline.

G2 - CIRCULAR INTERPOLATION

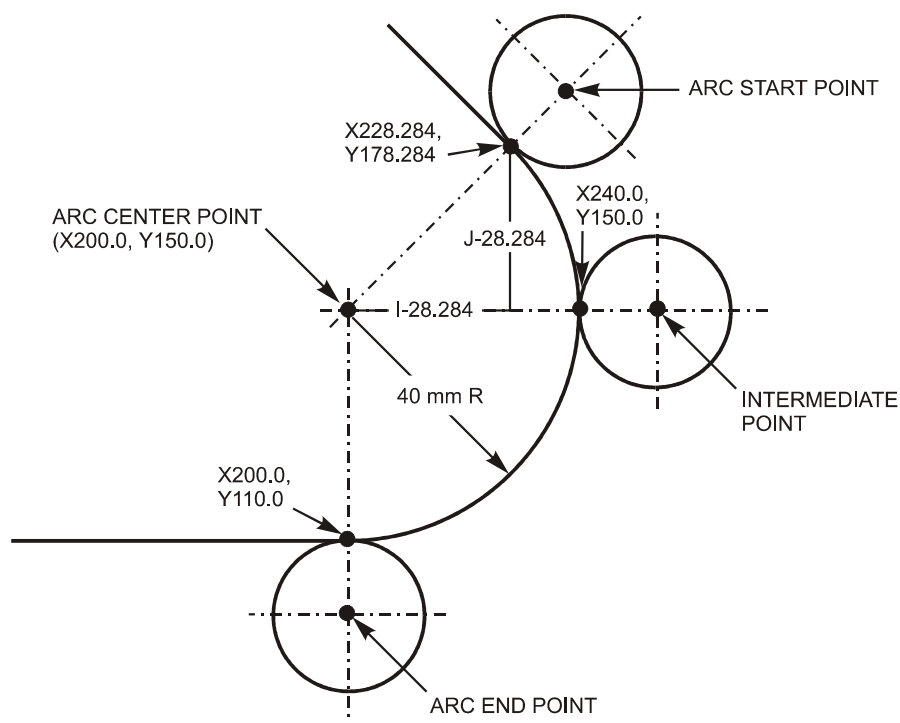


FIGURE 7-3 CIRCULAR INTERPOLATION PROGRAMMING EXAMPLE

Program Block

N100 G41 G1 X228.284 Y178.284 D2 F250.0 ;

N110 G2 X200.0 Y110.0 I-28.284 J-28.284 ;

Block Description

Linear move to Arc Start Point

Circular move to Arc End Point

7.5 Angle Chamfering and Corner Rounding

Angle Chamfering and Corner Rounding provides the capability to program a 45 degree chamfer or a corner radius by including a C or R word in the G1, G2, or G3 block.

Corner chamfers and radii can be inserted between the following types of blocks:

G1 and G1 blocks

G1 and G2 (or G3) blocks

G2 (or G3) and G1 blocks

G2 and G3 blocks

G3 and G2 blocks

- The C word is used for chamfering and defines the size of the chamfer measured from the virtual corner (sharp corner).
- The R word is used for corner rounding and defines the size of the radius for the corner.
- The C and R words are included in the blocks that move the tool to the corner.
- Consecutive blocks can be programmed with C or R words.
- The C and R word formats are:
 - For Inch Input - G20
Ccccc.cccc
Rrrrr.rrrr
 - For Metric Input - G21
Cccccccc.ccc
Rrrrrr.rrr
- The block immediately following a corner chamfering or rounding block must be a G1, G2, or G3 block.

- Corner chamfering or rounding can be performed only in the axes of the active Plane Select mode:
 - X and Y for G17
 - Z and X for G18
 - Y and Z for G19

These functions cannot be performed for parallel axes.

- Corner chamfering or rounding cannot be specified in a block that immediately follows a coordinate system change command:
 - G92 - Position Set
 - G54 through G59 - Work Coordinate Systems
- Corner chamfering or rounding cannot be specified in a block that immediately follows a reference position command:
 - G28 - Reference Point Return
 - G29 - Return From Reference Point
 - G30 - 2nd, 3rd, 4th Reference Point Return
 - G30.1 - Floating Reference point Return
- The following G codes cannot be programmed in a block that specifies corner chamfering or rounding:
 - G codes of Group 00, except G4 - Dwell
 - G68 - Coordinate System Rotation
- If the C or R word causes the axes to move further than the block would have moved without C or R, an alarm is generated.
- A travel distance of 0 (zero) is regarded when chamfering or rounding is specified and:
 - two consecutive linear interpolation blocks are executed and the angle between the two straight lines are within +1 degree,
 - when a linear interpolation block intersects with a circular interpolation block and the angle between the straight line and arc tangent line is within +1 degree, or,
 - when two circular blocks are executed and the angle between the tangents to the arcs at the intersection is within +1 degree.

7.6 Polar Coordinate Input - G15 and G16 (Option)

Polar Coordinate Input is a way to program an end point by using a radius and angle. Polar Coordinate Input is controlled by two G codes:

G15 - Polar Coordinate Command Cancel

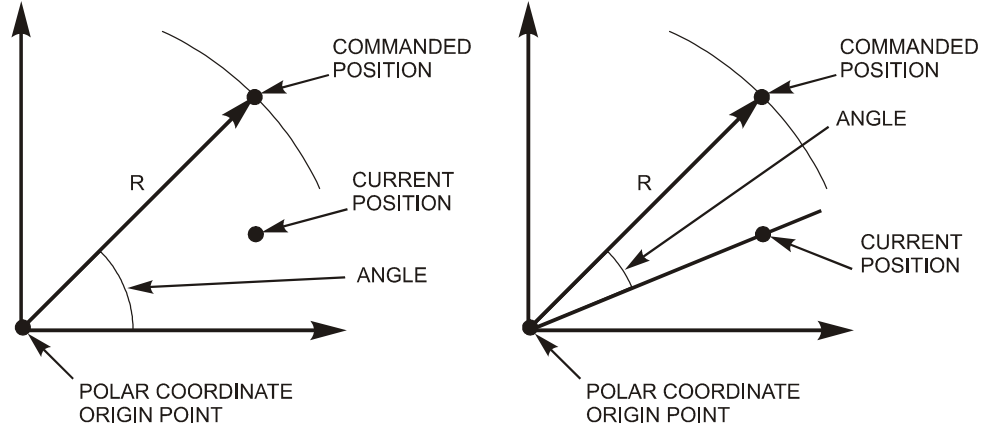
G16 - Polar Coordinate Command

- G15 and G16 are modal G codes that remain in effect until canceled by the opposing code. G15 is active at machine power on, or when an end of program command (M2 or M30) is executed.
- The radius and angle of the end point are programmed using X, Y, and Z word addresses. The two word addresses that are used is dependent upon the active Plane Select mode (G17, G18, or G19) as follows:
 - With G17 - XY Plane Select active:
X is used for the radius, and
Y is used for the angle.
 - With G18 - ZX Plane Select active:
Z is used for the radius, and
X is used for the angle.
 - With G19 - YZ Plane Select active:
Y is used for the radius, and
Z is used for the angle.
- The plus (+) direction of the angle is in the counterclockwise direction. The minus (-) direction is in the clockwise direction.
- The radius and angle can be programmed with either Absolute (G90) or Incremental (G91) coordinates.
- The Polar Coordinate Input origin (zero) point, from which the radius is measured, is established by programming either a G90 or G91 code in the same block with G16.
 - When G90 is used, the Polar Coordinate Input origin point is established at the zero point of the active work coordinate system or local coordinate system.
 - When G91 is used, the current position of the axes becomes the Polar Coordinate Input origin point.

- Polar Coordinate Input can be used with the interpolation codes G1, G2, and G3 and with the canned cycle G codes.
 - When programming circular interpolation with G2 or G3, the center point of the arc must be programmed with R.
- The following commands cannot be executed while G16 is active:
 - G4 - Dwell
 - G10 - Data Setting
 - G22 - Stored Stroke Check
 - G51 - Scaling
 - G52 - Local Coordinate System Setting
 - G53 - Machine Coordinates
 - G68 - Coordinate System Rotation
 - G92 - Work Coordinate System Setting

LOCAL COORDINATE ZERO AND
POLAR COORDINATE ORIGIN POINT AT SAME POINT

RADIUS SPECIFIED BY ABSOLUTE COMMAND



POLAR COORDINATE ORIGIN POINT SET AT CURRENT POSITION

RADIUS SPECIFIED BY INCREMENTAL COMMAND

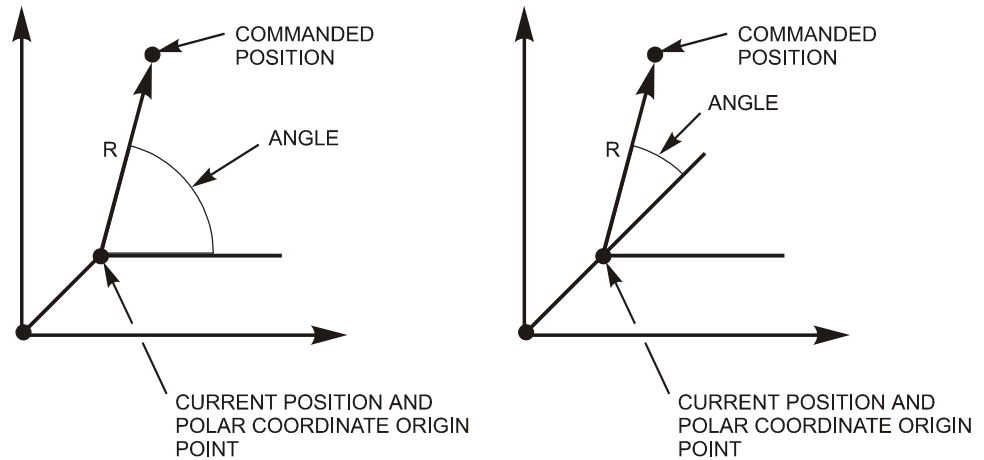
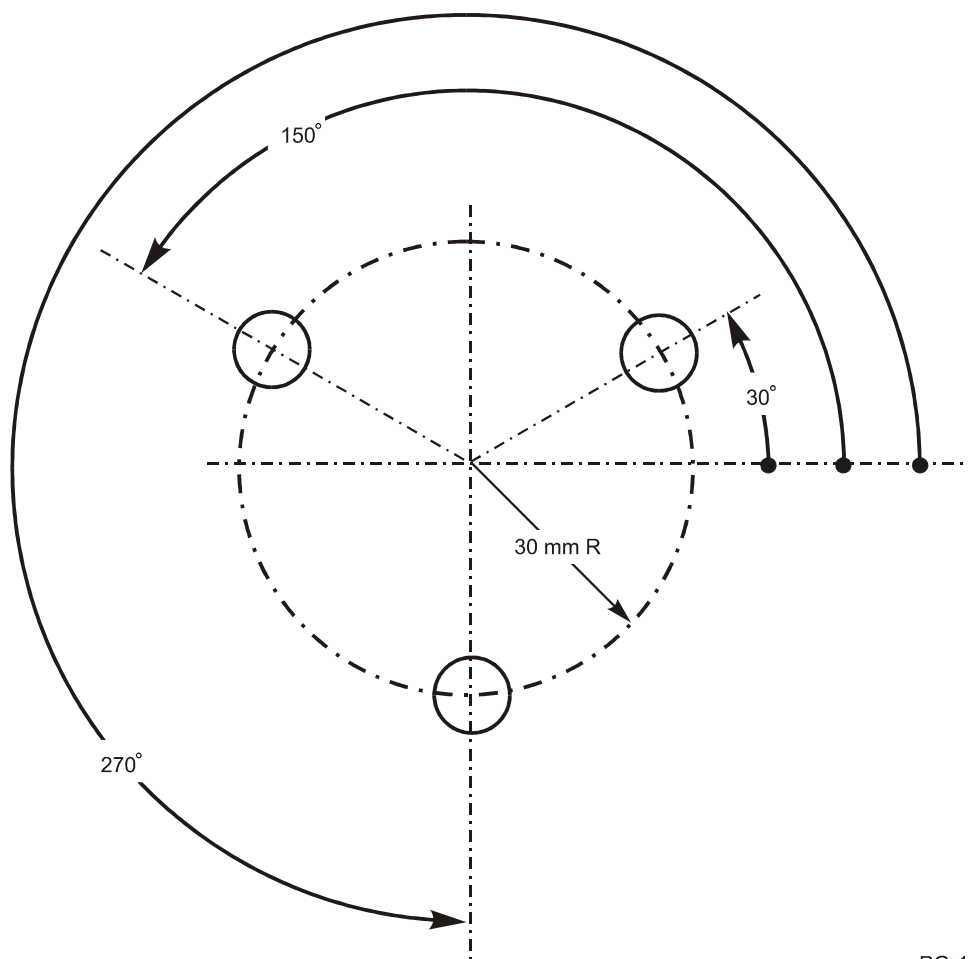


FIGURE 7-4 POLAR COORDINATE INPUT

PG-114

BOLT CIRCLE - PROGRAMMED WITH POLAR COORDINATES



PG-115

FIGURE 7-5 BOLT CIRCLE - POLAR COORDINATE PROGRAMMING EXAMPLE

Program Block	Block Description
N100 G54 ;	Activates Work Coordinate System
N110 T1 ;	Ready tool 1
N120 M6 ;	Tool change command
N130 G43 Z50.0 H1 S1000 M3 ;	Move Z to Safe Start position, start spindle
N140 G52 X200.0 Y150.0 ;	Establish Polar Coordinate Origin Point
N150 G17 G90 G16 ;	Polar Coordinate Input On
N160 G81 X30.0 Y30.0 Z-75.0 R2.5 F250.0 ;	Drill Cycle on 30° Angle and 30 mm Radius
N170 Y150.0 ;	150° Angle on 30 mm Radius
N180 Y270.0 ;	270° Angle on 30 mm Radius
N190 G80 G15 ;	Cancel Drill Cycle and Polar Coordinate Input
N200 G52 X0.0 Y0.0 ;	Cancel Local Coordinate System
N210 G91 G28 Z0 ;	Refer Z axis
N220 G90 ;	Set Absolute Input mode

7.7 Dwell - G4

G4 Dwell provides the capability to hold the machine axes motionless, for a specified amount of time, by inhibiting execution of the next block of information.

- The G4 code is a non-modal code.
- G4 does not cancel the active interpolation mode (G1, G2, or G3).
- The amount of dwell time is specified in seconds and is programmed with either an X or P word address.
 - When X is used a decimal point is programmed and the dwell time is programmed in the range:

X.001 to X99999.999

- When P is used a decimal point is not programmed and the dwell time is programmed in the range:

P1 to P99999999

- The unit of P is .001 second. P1000 is equivalent to 1 second.
- When neither an X nor P is programmed in the G4 block, the control does an Exact Stop which means the axes are decelerated to a stop at the end of the block before the next block is executed.

Chapter 8 **Cutter Compensation C**

Cutter Compensation On and Off
G41, G42, G40

Makino Vertical and Horizontal
Machining Centers



Chapter 8

Cutter Compensation C

Contents

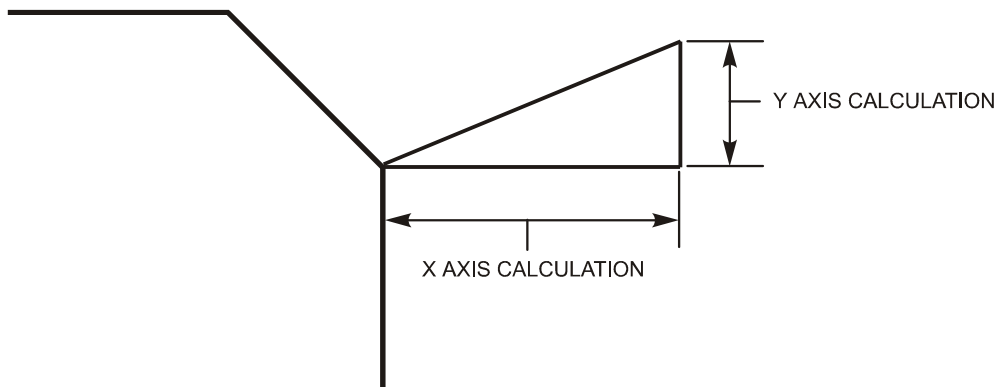
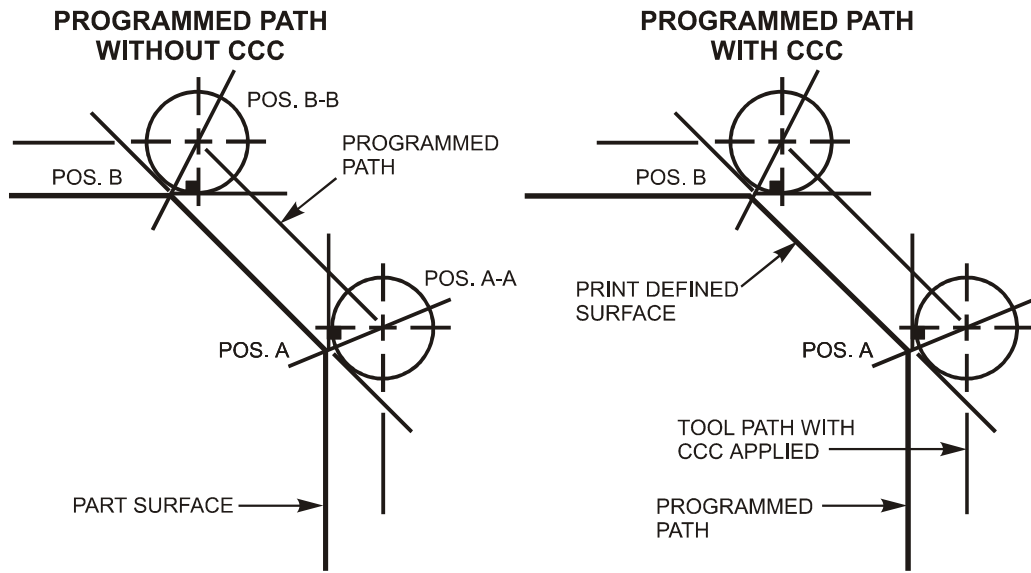
8.1 Cutter Compensation C - G40, G41, G42 and D Word.	8-1
8.1.1 Cutter Compensation C - General	8-1
8.2 Cutter Compensation C - G Codes and D Word.	8-3
8.2.1 D Word	8-3
8.2.2 G Codes - G40, G41, G42	8-4
8.3 Considerations for Cutter Selection	8-6

8.1 Cutter Compensation C - G40, G41, G42 and D Word

8.1.1 Cutter Compensation C - General

The Cutter Compensation C (CCC) feature provides the capability to program part surface features without regard to the TNR (Tool Nose Radius). In other words the center of the tool is programmed for milling cuts, like normal hole making operations, and the control offsets the programmed path by the TNR value stored in the control. See Figure 8-1.

- With CCC, the coordinates of the part surface being programmed can generally be copied from the part print.
- Since the TNR is stored in the control, any size cutter that fits the part geometry can be used to machine the part.
 - Certain criteria must be considered when selecting the cutter size. See “Considerations for Cutter Selection” on page 8 - 6 for additional information.
- Without CCC, the programmed tool path must be offset from the part surface by an amount equal to the radius of the tool being used to machine the part.
 - Additionally, special calculations are required when machining angular cuts in order to maintain TNR tangency at the start and end of the cut.
 - Also, in order to machine the part to the correct size, the cutter used to machine the part must be exactly the same size that the programmer used when calculating the programmed coordinates.
- No more than two non-motion blocks can be programmed while CCC (G41 or G42) is active.
 - Non-motion blocks include any blocks that are programmed with zero tool movement or without axis commands.
 - The control buffers two blocks of information when CCC is active in order to calculate the correct offset end point. The offset end point calculation is based on the end points of the current and next blocks and the active TNR value.



NOTE: WITHOUT CCC
 POS. A-A IS CALCULATED RELATIVE TO POS.A
 POS. B-B IS CALCULATED RELATIVE TO POS.B

NOTE:
 IN BOTH CASES ABOVE, THE PART SURFACE WILL BE THE SAME.
 LESS CALCULATIONS ARE REQUIRED WHEN CCC IS USED.

PG-105

FIGURE 8-1 TOOL PATH CALCULATION W/VO CUTTER COMPENSATION

8.2 Cutter Compensation C - G Codes and D Word

- CCC is controlled by three G codes and a D word:
 - G40 - CCC Cancel
 - G41 - CCC On - Tool Left
 - G42 - CCC On - Tool Right
 - D - Tool Offset Number, where the TNR value is stored.

8.2.1 D Word

The D word represents the Offset Number, in the CNC Side Offset Table, where the TNR value is stored.

- The D word is either a two or three digit number (Ddd or Dddd) that represents an offset number in the Tool Offset Table. The value entered into the tool offset number, defined by the programmed D word, is the size of the TNR being used to machine the part.
 - The two digit D word is programmable in the range of D1 to D99.
 - The two digit D word is used when the control has only the standard tool offset feature of 99 offsets.
 - The three digit D word is programmable in the range of D1 to D264.
 - The three digit D word is used when the control has the extended tool offset feature.
- The D word is programmed in the G41 (or G42) start-up block.
- Programming a D0 cancels the active D word number and will cause the TNR value to be interpolated out thus returning the tool path back to the center of the tool.
- The D word number is chosen by the programmer and can be any number in the programmable range. However, as a suggestion, the chosen D word number should be a number that can be easily related to the tool number that will be using the D word.
 - For example, if the tool number is T01, a D word of D11, D21, D31, etc. could be used. D01 should not be used since offset number 01 will most likely be used for the tool length offset value.

D Words when the Custom Side Tool Management Feature is Used

When using the Custom Side Tool Management feature, TNR values are stored in the "D - Value" column of the TOOL DETAIL screen. In this case, the TNR value is automatically transferred to the CNC Side Offset Table when a tool change is executed.

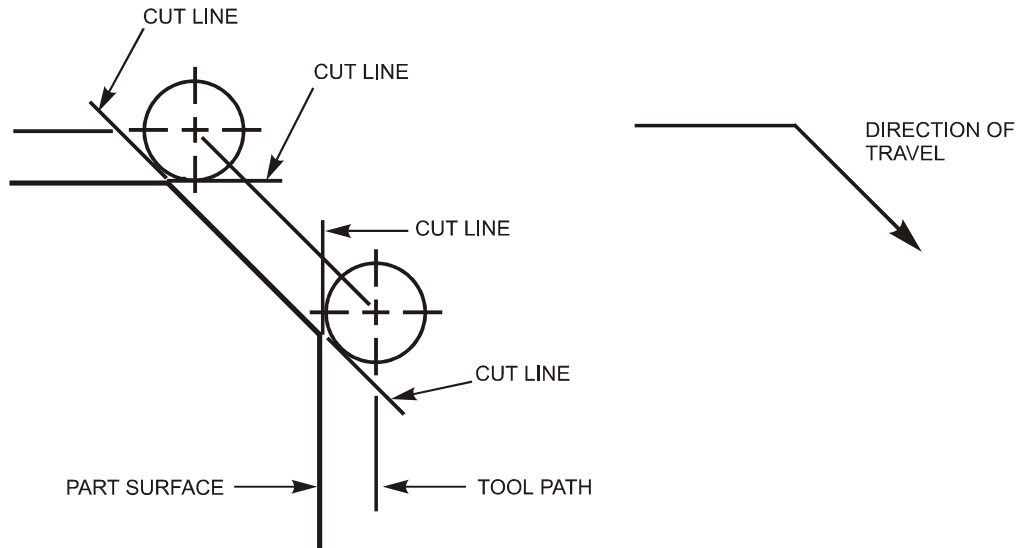
The Tool Change Programmed is typically written to transfer TNR values to offset number 02 in the Offset Table. In this case, the D word will be D02 for all tools using cutter compensation.

8.2.2 G Codes - G40, G41, G42

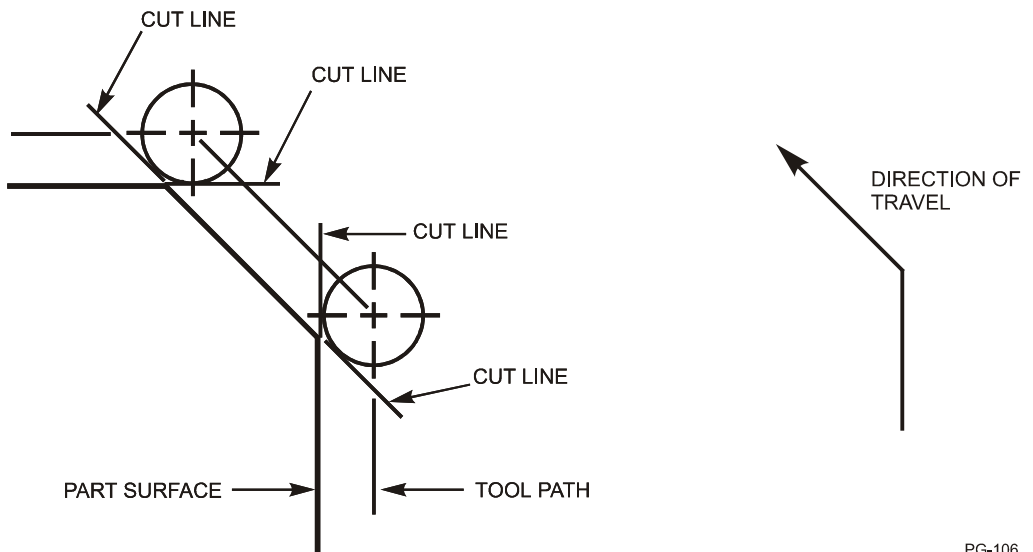
G41 and G42 are used to activate CCC and are programmed in a block with an axis move that is called a start-up block.

- G41 and G42 are modal codes that cancel each other. G40 cancels G41 and G42.
- G41 - CCC On - Tool Left is used when the cutter path is on the left side of the cut line. The cut line being the part surface. See Figure 8-2.
 - G41 always results in a climb milling operation.
- G42 - CCC On - Tool Right is used when the cutter path is on the right side of the cut line. The cut line being the part surface. See Figure 8-2.
 - G42 always results in a conventional milling operation.
- The start-up block is the block of information that moves the tool to the part surface to be machined. The start-up block can be programmed in the G0 or G1 mode. If the start-up block is programmed with G2 or G3 an alarm is generated.
- G40 CCC Cancel cancels G41 and G42 and returns the tool path back to the center of the tool.
 - G40 is the initialized state at machine power On. G40 is also active after a RESET operation and when an end of program (M2 or M30) is executed.
 - It is not necessary to program D0 with G40 to cancel CCC.
- G40 should be programmed in a block with axes coordinates that move the tool away from the part surface.
 - G40 can be programmed in the G0 or G1 mode. If G40 is executed while G2 or G3 is active an alarm is generated.

G41 - CCC ON - TOOL LEFT (CLIMB MILLING)



G42 - CCC ON - TOOL RIGHT (CONVENTIONAL MILLING)



PG-106

FIGURE 8-2 G42 TOOL RIGHT/G41 TOOL LEFT EXAMPLE

8.3 Considerations for Cutter Selection

When Cutter Compensation C is programmed, the programmer may chose to program the tool path by one of two ways:

- The tool path can be programmed with a zero TNR value meaning that part print dimensions are programmed.
 - This method of programming requires that the whole TNR (radius of the cutter) value be entered into the appropriate tool offset number.
- The tool path can be programmed with a TNR value of the cutter expected to be used to machine the part.
 - This method of programming requires that only the difference between the intended and actual cutter size used is entered into the appropriate tool offset number

Depending on how the tool path is programmed, the following should be considered when selecting the cutter for machining.

- The selected cutter size should coincide with the RPM and feed rates that are programmed for the operation.
 - For example, if the programmed RPM and feed rates are based on using a 1.0" diameter cutter and a .25" diameter cutter is used, the programmed RPM and feed rates will not be acceptable of the .25" cutter.
- The part geometry must able to be machined by the selected cutter size.
 - For example, if the smallest radius of the operation is 1.0", a cutter having a 1.0" or less radius must be used to machine the part.

Chapter 9 Canned Cycles

Canned Cycle Operations - G73, G74,
G76, and G81 through G89

Makino Vertical and Horizontal
Machining Centers



Chapter 9 Canned Cycles

Contents

9.1 Canned Cycles	9-1
9.1.1 Canned Cycles - General	9-2
9.1.2 Z Retract Positions - G98 and G99	9-4
9.1.3 Canned Cycle Repeat Function - L Word	9-4
9.1.4 Canned Cycle - Word Address Format Table	9-5
9.2 Canned Cycle Descriptions.	9-8
9.2.1 G73 - High Speed Peck Drilling Cycle	9-8
9.2.2 G74 - Left Hand Tap Cycle	9-8
9.2.3 G76 - Fine Boring Cycle	9-9
9.2.4 G81 - Drilling Cycle	9-10
9.2.5 G82 - Drilling/Counter Boring Cycle	9-10
9.2.6 G83 - Peck Drilling Cycle	9-10
9.2.7 G84 - Right Hand Tap Cycle	9-11
9.2.8 G85 - Boring Cycle	9-11
9.2.9 G86 - Boring Cycle	9-11
9.2.10 G87 - Back Boring Cycle	9-12
9.2.11 G88 - Boring Cycle	9-13
9.2.12 G89 - Boring Cycle	9-14

9.1 Canned Cycles

Canned cycles are machining functions that perform a predetermined sequence of operations by a single G code. Programming is simplified since after the canned cycle operation is started, only the information that changes has to be programmed.

Twelve canned cycles are provided that include:

G73 - High Speed Peck Drilling Cycle

G74 - Left Hand Tapping Cycle

G76 - Fine Boring Cycle

G81 - Drilling Cycle

G82 - Drilling/Counter-Boring Cycle

G83 - Peck Drilling Cycle

G84 - Right Hand Tapping Cycle

G85 - Boring Cycle - Feed in, Feed Out

G86 - Boring Cycle - Feed in, Spindle Stop, Rapid Out

G87 - Boring Cycle - Back Boring

G88 - Boring Cycle - Feed in, Dwell, Spindle Stop, Manual Retract

G89 - Boring Cycle - Feed in, Dwell, Feed Out

Canned cycle G codes are modal codes that are canceled by each other or by the G80 Cancel Cycle. Canned cycles are also canceled by G0, G1, G2, and G3.

9.1.1 Canned Cycles - General

A canned cycle consists of six basic operations as follows:

Operation 1 - Rapid X and Y to position. Z will also position to its Initial Level if it is programmed in the first X and Y move.

Operation 2 - Rapid to R Plane.

Operation 3 - Hole Machining.

Operation 4 - Operation at bottom of hole. This will vary depending on the canned cycle.

Operation 5 - Retraction to R Plane. The method of retraction varies depending on the canned cycle.

Operation 6 - Rapid Z to Initial Level.

- The positioning axes are determined by the active Plane Select mode, G17, G18, or G19. Unless special tooling is being used, the G17 XY Plane Select mode is used and the positioning axes are X and Y.
 - The Z axis is the machining axis.
- X, Y, Z, and R coordinates can be programmed as G90 - Absolute or G91 - Incremental values.
- Z axis positions of a canned cycle include the Initial Level, R Plane, and Hole Depth.

Initial Level

- The Initial Level should be programmed in a block by itself before the first canned cycle block and after the block that moves the X and Y axes to the first hole making position. The Z axis rapids to the Initial Level.
- Typically the Initial Level is programmed to a position that is 50 mm (2.0") above the part.

It can be a position that is clear of clamping devices or clear of the highest surface feature on the part.

- Tool length offsets should be applied (G43 or G44) when positioning to the Initial Level.

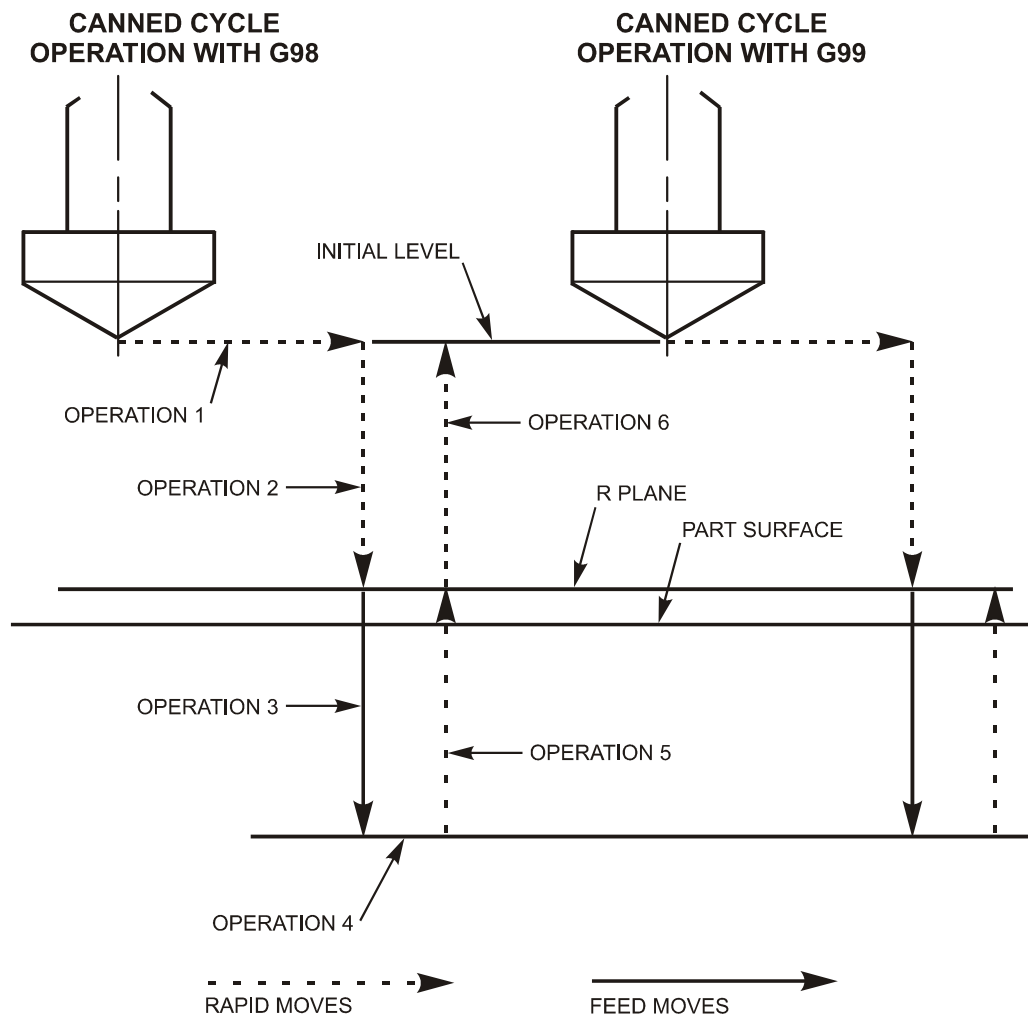
If G43 or G44 is programmed in the canned cycle block, the tool length offsets are applied when the Z axis positions to the R Plane.

R Plane

- The R Plane is programmed in the block with the canned cycle G code and is the Z axis position, chosen by the programmer, where the tool stops rapid motion of Operation 2 and starts feed motion of Operation 3.

Hole Depth

- The hole bottom (depth) is programmed in the block with the canned cycle G code and is the Z axis position where Operation 3 stops.



NOTES:
 WITH G98 ACTIVE, THE TOOL RETRACTS FROM OPERATION 4 TO THE INITIAL LEVEL.
 WITH G99 ACTIVE, THE TOOL RETRACTS FROM OPERATION 4 TO THE R PLANE.

PG-107

FIGURE 9-1 CANNED CYCLE OPERATION STEPS

9.1.2 Z Retract Positions - G98 and G99

The Z retract position after completion of Operation 3 is controlled by two G codes:

G98 - Return to Initial Level

G99 - Return to R Plane

G98 is the initialized state at machine power on, after a RESET, and after an end of program code (M2 or M30) is executed.

Normally G99 is used since this minimizes Z axis travel while operating in a canned cycle mode.

G98 is used when it is necessary to retract to a position that is clear of all obstructions, such as clamping devices or other part surface features.

9.1.3 Canned Cycle Repeat Function - L Word

When a hole pattern is equally spaced, the cycle can be repeated by programming an L word.

The L word specifies the number of times the cycle is to be repeated and is programmed in the range of:

L0 to L9999

When using the L word, the first hole of the repeatable pattern must be specified in the Incremental (G91) mode. The cycle is repeated at the incremental distance and in the direction of the signed increment by the number of times specified by L. Both X and Y coordinates can be specified when the repeat pattern is on an angle.

9.1.4 Canned Cycle - Word Address Format Table

TABLE 9-1 CANNED CYCLE - WORD ADDRESS FORMATS

	Inch Format	Metric Format	Remarks
G73 High Speed Peck Drill Cycle - Partial Retract During In-Feed to Break Chips			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z±zzzz.zzzz	Z±zzzzz.zzz	Hole Depth
R	R±rrrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0.
Q	Qqqqqq.qqqq	Qqqqqq.qqq	Unsigned Cut Depth Incremental, Equal Distance
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G73 is to Repeat, L1 to L9999
G74 Left-Hand Tap - Feed In, Reverse Spindle, Feed Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z±zzzz.zzzz	Z±zzzzz.zzz	Hole Depth
R	R±rrrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G74 is to Repeat, L1 to L9999
G76 Fine Boring Cycle - Feed In, Dwell, Orient Stop, Shift, Rapid Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z±zzzz.zzzz	Z±zzzzz.zzz	Hole Depth
R	R±rrrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0
Q	Qqqqqq.qqqq	Qqqqqq.qqq	Shift Amount at Depth - (+) Increment
P	Ppppppppp	Ppppppppp	Dwell at Depth, Unit = .001 Second
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Lll	Lll	Number of Times G76 is to Repeat, L1 to L99
G81 Drill Cycle - Feed In, Rapid Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z±zzzz.zzzz	Z±zzzzz.zzz	Hole Depth
R	R±rrrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G81 is to Repeat, L1 to L9999

TABLE 9-1 CANNED CYCLE - WORD ADDRESS FORMATS (CONTINUED)

	Inch Format	Metric Format	Remarks
G82 Drill/Counter Bore - Feed In, Dwell, Rapid Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0
P	Ppppppppp	Ppppppppp	Dwell at Depth, Unit = .001 Second
F	Fffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G82 is to Repeat, L1 to L9999
G83 Peck Drill Cycle - Full Retract During In-Feed to Clear Chips			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0.
Q	Qqqqq.qqqq	Qqqqq.qqq	Unsigned Cut Depth Incremental, Equal Distance
F	Fffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G83 is to Repeat, L1 to L9999
G84 Right Hand Tap - Feed In, Reverse Spindle, Feed Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0.
F	Fffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G84 is to Repeat, L1 to L9999
G85 Bore Cycle - Feed In, Feed Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0.
F	Fffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G85 is to Repeat, L1 to L9999

TABLE 9-1 CANNED CYCLE - WORD ADDRESS FORMATS (CONTINUED)

	Inch Format	Metric Format	Remarks
G86 Bore Cycle - Feed In, Stop Spindle, Rapid Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0.
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G86 is to Repeat, L1 to L9999
G87 Back Bore Cycle			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0
Q	Qqqqq.qqqq	Qqqqqq.qqq	Shift Amount - (+) Increment
P	Ppppppppp	Ppppppppp	Dwell at Depth, Unit = .001 Second
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Lll	Lll	Number of Times G87 is to Repeat, L1 to L99
G88 Boring Cycle - Feed In, Dwell, Stop Spindle, Manual Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0.
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G88 is to Repeat, L1 to L9999
G89 Bore Cycle - Feed In, Dwell, Feed Out			
X	X±xxxx.xxxx	X±xxxxx.xxx	X Axis Coordinate
Y	Y±yyyy.yyyy	Y±yyyyy.yyy	Y Axis Coordinate
Z	Z+zzzz.zzzz	Z+zzzzz.zzz	Hole Depth
R	R±rrrr.rrrr	R±rrrrr.rrr	R Plane, Absolute Coordinate Relative to Z0
P	Ppppppppp	Ppppppppp	Dwell at Depth, Unit = .001 Second
F	Ffff.ff	Fffff.f	Cutting Feed Rate
L	Llll	Llll	Number of Times G89 is to Repeat, L1 to L9999

9.2 Canned Cycle Descriptions

9.2.1 G73 - High Speed Peck Drilling Cycle

G73 is used for drilling operations where it is desirable to interrupt the in-feed motion in order to break chips. This cycle does not fully retract the Z axis until after the programmed depth has been reached.

The basic word addresses for G73 are:

G73 X___ Y___ Z___ R___ Q___ F___ L___

- The unique word address for G73 is Q which defines the depth of cut for each in-feed motion.
- Q must be programmed in the G73 block and is an unsigned incremental value. All in-feed motions are of equal distances.
- The retract amount at the end of each in-feed motion is defined by control parameter 5115. The factory setting is 100. The retract motion is at the rapid rate.

9.2.2 G74 - Left Hand Tap Cycle

G74 is used to machine left hand threads. The spindle must be started in the counterclockwise direction before executing the G74 cycle. The spindle reverses to the clockwise direction when hole depth is reached.

The basic word addresses for G74 are:

G74 X___ Y___ Z___ R___ F___ L___

- The tapping feed rate is programmed with the F word and represents the tap lead in millimeters (or inches) per minute.
- Feedrate overrides are ignored during the in-feed and out-feed motion of the cycle.
- Feedhold is ignored during the in-feed and out-feed motion of the cycle.
- A compensating tap driver should be used for this cycle.

9.2.3 G76 - Fine Boring Cycle

G76 is used for boring operations where it is desirable to feed to depth, dwell, stop and orient the spindle, then shift the axes to prevent dragging the tool along the finished bore while retracting the Z axis.

The basic word addresses for G76 are:

G76 X____ Y____ Z____ R____ Q____ P____ F____ L____

- The unique word addresses for G76 are Q and P.
 - Q defines the shift amount of the axes.
 - P defines a dwell at the bottom of the hole before the spindle is stopped.
- Q must be programmed in the G76 block and is an unsigned incremental value.
- The shift direction is set in control parameter 5101, bits 4 (RD1) and 5 (RD2) on the Pro 3 control.

The standard factory setting shift direction for Horizontal Machines is negative in the X axis. The standard factory setting shift direction for Vertical Machines is negative in the Y axis.

NOTE:

By setting parameter 5103, Bit 0 = 1, in the Pro 3 control, I and/or J can be used instead of Q. This allows programming a value with I and/or J to shift the tool in the desired direction. The format of I and J is:

I±iiii.iiii J±jjjj.jjjj - with G20 Inch Mode

I±iiii.iii J±jjjj.jjj - with G21 Metric Mode

IMPORTANT:

The Q word is a modal word that is retained until its value is changed. The Q word is also used with the G73 and G83 cycles. The value programmed for the G76 cycle will be used for the G73 and G83 cycles unless a new value is programmed.

- P must be programmed in the G76 block and is a dwell programmed in seconds. The P word range is:

P0 to P99999999 for both Inch and Metric input.

9.2.4 G81 - Drilling Cycle

G81 is used for drilling operations where a dwell at hole depth is not necessary.

The basic word addresses for G81 are:

G81 X___ Y___ Z___ R___ F___ L___

9.2.5 G82 - Drilling/Counter Boring Cycle

G82 is used for drilling or counter-boring operations where it is desirable to feed to depth and dwell for a specified amount of time before retracting the tool.

The basic word addresses for G82 are:

G82 X___ Y___ Z___ R___ P___ F___ L___

- The unique word address for G82 is P. P defines a dwell at the bottom of the hole before retracting the tool.
- P must be programmed in the G82 block and is a dwell programmed in seconds. The P word range is:

P0 to P99999999 for both Inch and Metric input.

9.2.6 G83 - Peck Drilling Cycle

G83 is used for drilling operations where it is desirable to withdraw the tool from the hole in order to clear chips. During execution of this cycle, the tool feeds to a depth defined by the Q word, retracts to the R Plane, rapids back into the hole to a position just short of the previous drilled depth then feeds another Q increment. Cycle is continued in this manner until hole depth is reached.

The basic word addresses for G83 are:

G83 X___ Y___ Z___ R___ Q___ F___ L___

- The unique word address for G83 is Q which defines the depth of cut for each in-feed motion.
- Q must be programmed in the G83 block and is an unsigned incremental value. All in-feed motions are of equal distances.

9.2.7 G84 - Right Hand Tap Cycle

G84 is used to machine right hand threads. The spindle must be started in the clockwise direction before executing the G84 cycle. The spindle reverses to the counterclockwise direction when hole depth is reached.

The basic word addresses for G84 are:

G84 X_____ Y_____ Z_____ R_____ F_____ L_____

- The tapping feed rate is programmed with the F word and represents the tap lead in millimeters (or inches) per minute.
- Feedrate overrides are ignored during the in-feed and out-feed motion of the cycle.
- Feedhold is ignored during the in-feed and out-feed motion of the cycle.
- A compensating tap driver should be used for this cycle.

9.2.8 G85 - Boring Cycle

G85 is used for boring operations where it is desirable is feed in and out of the bore at the same feed rate.

The basic word addresses for G85 are:

G85 X_____ Y_____ Z_____ R_____ F_____ L_____

9.2.9 G86 - Boring Cycle

G86 is used for boring operations where it is desirable is feed in, stop the spindle (not oriented) then rapid out of the bore.

The basic word addresses for G86 are:

G86 X_____ Y_____ Z_____ R_____ F_____ L_____

9.2.10 G87 - Back Boring Cycle

G87 is used for back boring operations. When G87 is executed, cycle operates in the following manner:

X and Y axes move to hole position,
the spindle stops and orients,
X (or Y) axis shifts opposite the direction of spindle orientation,
the tool is rapid traversed into the hole,
the axis shifts back to the hole position,
the spindle restarts,
the Z axis fed in a positive direction to the programmed depth,
the spindle stops and orients,
X (or Y) axis shifts opposite the direction of spindle orientation,
the tool is rapid traversed out of the hole,
the axis shifts back to the hole position,
the spindle restarts,

The basic word addresses for G87 are:

G87 X___ Y___ Z___ R___ Q___ P___ F___ L___

- The unique word address for G87 are Q and P. Q defines the shift amount of the axes and P defines a dwell at the bottom of the hole before the spindle is stopped.
- Q must be programmed in the G87 block and is an unsigned incremental value.
- The shift direction is set in control parameter 5101, bits 4 (RD1) and 5 (RD2) in the Pro 3 control.
 - The standard factory setting shift direction for Horizontal Machines is negative in the X axis. The standard factory setting shift direction for Vertical Machines is negative in the Y axis.

NOTE:

By setting parameter 5103, Bit 0 = 1, on the Pro 3 control, I and/or J can be used instead of Q. This allows programming a value with I and/or J to shift the tool in the desired direction. The format of I and J is:

I±iiii.iiii J±jjjj.jjjj - with G20 Inch Mode

I±iiii.iii J±jjjj.jjj - with G21 Metric Mode

IMPORTANT:

The Q word is a modal word that is retained until its value is changed. The Q word is also used with the G73 and G83 cycles. The value programmed for the G76 cycle will be used for the G73 and G83 cycles unless a new value is programmed.

- P must be programmed in the G87 block and is a dwell programmed in seconds. The P word range is:

P0 to P99999999 for both Inch and Metric input.

9.2.11 G88 - Boring Cycle

G88 is used for boring operations where it is desirable to feed in, dwell for a specified time, stop the spindle (not oriented) then manually retract out of the bore.

The basic word addresses for G88 are:

G88 X____ Y____ Z____ R____ P____ F____ L____

- The unique word address for G88 is P. P defines a dwell at the bottom of the hole before the spindle is stopped.
- P must be programmed in the G88 block and is a dwell programmed in seconds. The P word range is:

P0 to P99999999 for both Inch and Metric input.

9.2.12 G89 - Boring Cycle

G89 is used for boring operations where it is desirable to feed in, dwell for a specified time, then feed back out of the hole

The basic word addresses for G89 are:

G89 X___ Y___ Z___ R___ P___ F___ L___

- The unique word address for G89 is P. P defines a dwell at the bottom of the hole before the spindle is stopped.
- P must be programmed in the G89 block and is a dwell programmed in seconds. The P word range is:

P0 to P99999999 for both Inch and Metric input.

Chapter 10 Rigid Tapping

Rigid Tap Mode M135 with Canned
Cycles - G84 and G74

Makino Vertical and Horizontal
Machining Centers



Chapter 10 Rigid Tapping

Contents

10.1 Rigid Tapping	10-1
10.1.1 Rigid Tap Mode - M135	10-1
10.1.2 Programming Example	10-2
10.2 Peck Rigid Tapping - G84 and G74	10-3

10.1 Rigid Tapping

Rigid tapping provides the capability of performing tapping operations without using a compensating tap driver.

- Rigid tapping is accomplished by synchronizing the programmed feedrate and spindle speed to maintain the tap lead.
- Rigid peck tapping can also be performed.

10.1.1 Rigid Tap Mode - M135

Rigid tap cycles (G84 and G74) follow the same six basic Operations as described for non-rigid tap cycles.

The following commands set up the standard G84 (or G74) cycle to be a rigid tap cycle. See the “Programming Example” on page 10 - 2.

- M135 and S word must be programmed in a block prior to the G84 (or G74) canned cycle block.
 - It is not necessary to start the spindle (with M3 or M4) since the cycle automatically starts the spindle when the Z axis starts its feed motion.
- For 11M, 12M, and 15M controls:
 - The M135 block must contain an S word to define the spindle speed for the tapping operation.
 - For G84 right hand rigid tapping, the spindle must be started in the clockwise direction.
 - For G74 left hand rigid tapping, the spindle must be started in the counterclockwise direction.

10.1.2 Programming Example

Rigid tapping cycles are programmed in the following sequence:

Program Block	Block Description
N800 M6 ;	Loads 1/2-13 tap
N810 T8 ;	Readies next tool
N820 G0 X___ Y___ ;	Position X and Y to first hole location
N830 G43 Z___ H1 M8 ;	Position Z to Initial Level
N840 M135 S500 ;	Turns on Rigid Tap mode, establishes a spindle speed of 500 rpm
N850 G84 Z___ R___ F38.46 ;	Tap Cycle block, F38.46 sets the feedrate for the tap based on 500 rpm
N860 X___ Y___ ;	Position X and Y to next hole location
N870 X___ Y___ ;	Position X and Y to next hole location
N880 M6 ;	Tool Change for next machining operation

10.2 Peck Rigid Tapping - G84 and G74

Peck rigid tapping can be performed by adding a Q word to the G84 (or G74) block.

- Q is an unsigned incremental value used to define the increment of each in-feed motion of the peck tap cycle.
- All in-feed motions of the cycle are equal distances.
- All other data previously described for rigid tapping also applies to the peck tap cycles.

One of two peck rigid tapping cycles can be performed depending on the setting of parameter 5200, Bit 5. Both peck cycles are programmed in the same manner.

- When Pro 3 control parameter 5200 - Bit 5 is set to 0, The high speed peck rigid tapping cycle is selected.
 - The high speed peck rigid tapping cycle does not fully retract the tap after the in-feed motion. The retract distance is specified in parameter 5213.
- When Pro 3 control parameter 5200 - Bit 5 is set to 1, the standard peck rigid tapping cycle is selected.
 - The standard peck rigid tapping cycle retracts to the R Plane after each in-feed motion.

Chapter 11 Sub Programs

Sub Program Calls, Repeating, and
Nesting

Makino Vertical and Horizontal
Machining Centers



Chapter 11 Sub Programs

Contents

11.1 Sub Programs	11-1
11.2 Sub Program Numbers	11-1
11.3 Sub Program M Codes	11-2
11.3.1 Sub Program - P Word	11-3
11.3.1.1 Sub Program Call - Four Digit P Word	11-3
11.3.1.2 Sub Program Call - Eight Digit P Word	11-3
11.4 Sub Program Nesting	11-4
11.5 Main Program Looping with M99	11-5

11.1 Sub Programs

Sub Programs are used for a fixed sequence of operations, or frequently repeated patterns, to simplify programming.

- Sub Programs are stored in memory separate from the Main Program.
- Sub Programs are called from the Main Program.
- Sub Programs can call other Sub Programs. This is referred to as Sub Program Nesting, see “Sub Program Nesting” on page 11 - 4.

11.2 Sub Program Numbers

Sub Programs are stored in memory by a Program Number the same as a Main Programs.

- Sub Program numbers are addressed with a letter O (or by a colon (:)) code when ISO coding is used) followed by four digits.

O1234 ; or :1234 ;

11.3 Sub Program M Codes

Sub Programs are controlled by two M codes:

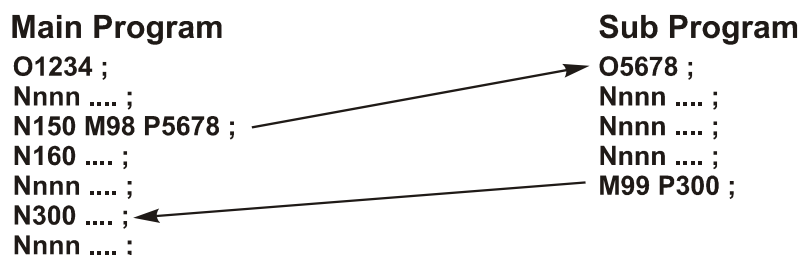
M98 - Sub Program Call

- Sub Programs are called by a block containing the M98 and a P word to specify the Sub Program number.
- M98 in a Main Program calls a Sub Program.
- M98 in a Sub Program calls another Sub Program (referred to as nesting).

M99 - End of Sub Program

- M99 is used to end a Sub Program and return program execution to the calling program. Program execution resumes immediately after the block of information containing the M98 call statement.
- When a P word is programmed in the M99 block, program execution resumes in the calling program at the Sequence Number specified by the P word.

Example:



PG-108

Example Description

- Block N150 of Main Program O1234 calls Sub Program O5678.
- When M99 is executed in the Sub Program, execution of the Main Program will resume at block N300.
- Without a P word in the M99 block, execution of the Main Program would resume at block N160.
- The P word in the M99 block can contain up to five digits (the same as the N word); leading zeros can be omitted.

11.3.1 Sub Program - P Word

Sub Programs can be called by either a four digit (Pxxxx) or eight digit (Pxxxxxxxx) word.

11.3.1.1 Sub Program Call - Four Digit P Word

Use a four digit P word to call a Sub Program, execute it one time, then return to the calling program.

Nxxxx M98 Pxxxx ;

- The P word is programmed in the M98 Sub Program Call block and is used to identify the Sub Program to be called.

11.3.1.2 Sub Program Call - Eight Digit P Word

Use an eight digit P word to call a Sub Program, repeat its execution, then return to the calling program.

Nxxxx M98 Pxxxxxxxx ;

- The first four digits following the P address specifies the number of times the Sub Program is to repeat.
- The last four digits specifies the Sub Program number being called.

Example:

Nxxxx M98 P51234 ;

This block calls Sub Program number 1234 and repeats 5 times.

When using the repeat feature, the four digits of the program number must be programmed even if any of these digits are zero.

Example:

Nxxxx M98 P50001 ;

This block calls Sub Program number 1 and repeats 5 times.

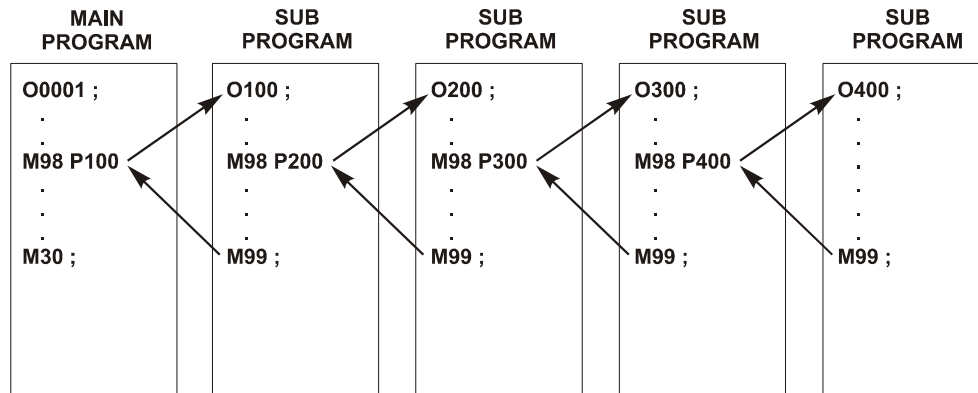
- A Sub Program can be repeated up to 9999 times with a single call in the M98 block. Leading zeros of the P word can be omitted.

11.4 Sub Program Nesting

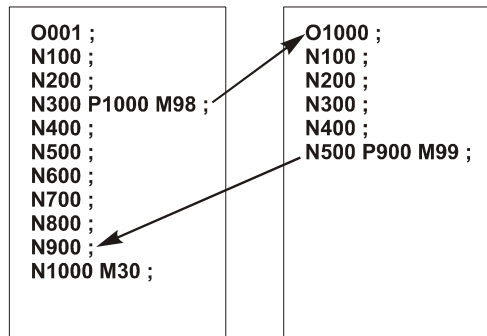
A Sub Program can call another Sub Program. This is called Sub Program nesting. Up to four Sub Programs can be nested.

Figure 11-1 shows an example of four level nesting, how the P word is used to resume operation at a specific Sequence Number in the calling program, and an example of the eight digit P word.

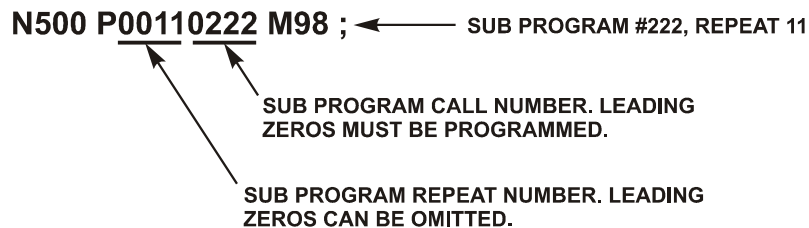
SUB PROGRAM NESTING - 4 LEVELS



USING P WORD FOR RETURN



8 DIGIT P WORD FOR SUB PROGRAM CALLING AND REPEATING



PG-109

FIGURE 11-1 SUB PROGRAMMING EXAMPLES

11.5 Main Program Looping with M99

The M99 code can be used in the Main Program to cause looping of the program.

- An M99 in the Main Program will cause the Main Program to loop back to its beginning and restart.
- An M99 and a P word in the Main Program will cause the Main Program to loop back to the sequence number defined by the P word and restart from that point.
 - If the Block Skip slash (/) code is programmed ahead of the M99 code, /Nnnnnn M99 Pppppp, the M99 block is not executed if Block Skip is turned On.

Chapter 12 **Scaling**

Automatic Scaling of Programmed
Coordinates

Makino Vertical and Horizontal
Machining Centers



Chapter 12 Scaling

Contents

12.1	Scaling - G50 and G51	12-1
12.2	Scaling All Axes at the Same Rate	12-2
12.3	Scaling Each Axis Individually at Different Rates	12-3
12.4	Scaling with Circular Interpolation.	12-5

12.1 Scaling - G50 and G51

Scaling provides a means to either magnify or reduce the final size of the workpiece from the original programmed coordinates. Depending on the setting of certain control parameters, scaling is performed on all axes at the same rate or on each axis separately at different rates.

NOTE:

Scaling is a standard feature on all Vertical Machines and an option on Horizontal Machines.

- Scaling is controlled by two G codes:

G50 - Scaling Off

G51 - Scaling On

- G50 scaling off is the initialized state at machine power on, after a RESET, and after an End of Program (M2 or M30) is executed.
- G50 and G51 are modal codes that remain in effect until canceled by each other.
- G50 must be programmed in a block by itself.
- Scaling is not applied to the values of cutter compensation and tool length offsets.
- G50 - Scaling Off must be active before executing:
 - G27 - Reference Position Return Check
 - G28 - Return to Reference Position
 - G29 - Return from Reference Position
 - G30 - 2nd, 3rd, 4th Reference Return Position
 - G92 - Work Coordinate System Setting
- Scaling is not applied to the movements of:
 - The Q word and retract value of the G73 and G83 Peck Drilling cycles or the Q word shift amount of the Fine Boring G76 and Back Boring G87.

The CRT displays the position after scaling is applied.

If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the movement may become zero. This could affect tool movement when Cutter Compensation C is active.

12.2 Scaling All Axes at the Same Rate

Scaling of all axes (X, Y, and Z) is performed when control parameter 5401, Bit 0 is set to 0.

- The scaling rate is either programmed by a P word, or, if P is not programmed the rate is taken from the setting of parameter 5411.
- Scaling is turned on by the following program block:

```
G51 X_____ Y_____ Z_____ P_____
```

The X, Y, and Z coordinates in the G51 block are used to define the center point for scaling. Omitting any or all axis coordinates in the G51 block causes the current position of the axis that was omitted to be defined as the center of scaling for that axis.

- The P word defines the rate of scaling.
 - The value of P is positive and programmed without a decimal point in units of .00001 or .001 depending on the setting of the Pro 3 control parameter 5400, Bit 6.
 - When parameter 5400, Bit 7 = 0, the input unit of P is .00001. The P word range is from:
P1 to P999999 (actually P.00001 to P9.99999)
 - A P word less than P100000 (actually P1.00000) results in reduced scaling.
 - A P word greater than P100000 (actually P1.00000) results in magnified scaling.
 - When parameter 5400, Bit 7 = 1, the input unit of P is .001. The P word range is from:
P1 to P999999 (actually P.001 to P999.999)
 - A P word less than P1000 (actually P1.000) results in reduced scaling.
 - A P word greater than P1000 (actually P1.000) results in magnified scaling.
 - When P is not programmed, the scaling rate of all axes is the value set in parameter 5411. The units of this parameter setting is either .00001 or .001 depending on the setting of parameter 5400, Bit 6.

12.3 Scaling Each Axis Individually at Different Rates

Scaling of each axis individually (X, Y or Z) is performed when the Pro 3 control parameter 5401, Bit 0 is set to 1.

- The scaling rate of each axis is programmed with I, J, and K words that represent X, Y, and Z respectively.
- Scaling is turned on by the following program block:

```
G51 X___ Y___ Z___ I___ J___ K___
```

- The X, Y, and Z coordinates in the G51 block are used to define the center point for scaling. Omitting any or all axis coordinates in the G51 block causes the current position of the axis that was omitted to be defined as the center of scaling for that axis.
- The I, J, and K words define the rate of scaling for the respective axis. Omitting I, J, or K words result in that axis (or axes) not to be scaled.
- The value of I, J, and K words is positive or negative (the - sign must be programmed) and is programmed without a decimal point in units of .00001 or .001 depending on the setting of parameter 5400, Bit 6.
- When the value of I, J, or K is negative, a mirror image of that axis is generated.
 - When parameter 5400, Bit 7 = 0, the input unit of I, J, and K is .00001. The I, J, and K word range is from:
+/-1 to +/-999999 (actually +/-0.00001 to +/-9.99999)
 - A word value in the range of +100000 to +1 (actually +1.00000 to +0.00001) results in reduced scaling of a non-mirrored image.
 - A word value in the range of -100000 to -1 (actually -1.00000 to -0.00001) results in reduced scaling of a mirrored image.
 - A word value greater than +100000 (actually +1.00000) results in magnified scaling of a non-mirrored image.
 - A word value greater than -100000 (actually -1.00000) results in magnified scaling of a mirrored image.
 - When parameter 5400, Bit 7 = 1, the input unit of I, J, and K is .001. The I, J, and K word range is from:
+/-1 to +/-9999 (actually +/-0.001 to +/-9.999)
 - A word value in the range of +1000 to +1 (actually +1.000 to +0.001) results in reduced scaling of a non-mirrored image.
 - A word value in the range of -1000 to -1 (actually -1.000 to -0.001) results in reduced scaling of a mirrored image.

- A word value greater than +1000 (actually +1.000) results in magnified scaling of a non-mirrored image.
- A word value greater than -1000 (actually -1.000) results in magnified scaling of a mirrored image.
- When I, J, and K are not programmed, the scaling rate of all axes is the value set in the Pro 3 control parameter 5421. The units of this parameter setting is either .00001 or .001 depending on the setting of parameter 5400, Bit 6.
- When a mirror image is applied to one axis of the specified plane, the following applies:
 - Direction of G2 and G3 are reversed.
 - Offset direction for Cutter Compensation is reversed.
 - Rotation angle for Coordinate System Rotation is reversed.
 - Offset direction for Cutter Compensation B (including G39) is reversed.

12.4 Scaling with Circular Interpolation

Scaling of circular interpolation (G2 and G3 blocks) can be performed when different rates of magnification are applied to each axis. However, the tool path generated for the circular blocks will be different depending on whether the arc center point is defined with an R word or with I and J words. In either case, the tool path that is generated will not be elliptical.

- Correct way to program circular blocks.
 - When the circle center point is defined by an R word, the circular arc is magnified by the larger value of the I and J words programmed in the G51 block.
- Incorrect way to program circular blocks.
 - When the circle center point is defined by I and J words, the circular arc is not magnified. Instead, an arc is generated as defined by the G2 (or G3) block, then a linear move is made in one axis to satisfy the end point defined the magnification values of the I and J words programmed in the G51 block. The axis that makes the linear move is the axis that has the larger magnification value.

Chapter 13 **Coordinate System Rotation**

Rotating X and Y Coordinate System

Makino Vertical and Horizontal
Machining Centers



Chapter 13

Coordinate System Rotation

Contents

13.1	Coordinate System Rotation - G68 and G69	13-1
13.2	CSR - G Codes and R Word.	13-1
13.3	Angle of Rotation - R Word.	13-2
13.3.1	Coordinates for Center of Rotation	13-2
13.4	Using CSR with Other Programming Functions	13-4
13.4.1	CSR and Cutter Compensation	13-4
13.4.2	CSR, Scaling, and Cutter Compensation	13-4

13.1 Coordinate System Rotation - G68 and G69

The CSR (Coordinate System Rotation) option provides a means to rotate the coordinate system of the NC program to compensate for setup errors or simplify programming for patterns that are rotated at various positions on the workpiece.

- If the feature is used to compensate for setup errors, the NC program must be modified to include the necessary CSR commands.

When used to program patterns, a Sub Program can be used to program the pattern one time then called to repeat at the various angular positions of the pattern.

13.2 CSR - G Codes and R Word

CSR is controlled by two G codes and an R word:

G68 - Coordinate System Rotation On

G69 - Coordinate System Rotation Off

R - Angle of Rotation

- G69 CSR Off - is the initialized state at machine power on, after a RESET, and after an End of Program (M2 or M30) is executed.
- G68 and G69 are modal codes that remain in effect until canceled by each other.
- G68 CSR On - must be active prior to executing a Cutter Compensate C command, G41 - CCC On - Tool Left or G42 - CCC On - Tool Right.
- G69 CSR Off - can be programmed in a block with other data.
- G69 CSR Off - must be active before executing:
 - G17 - XY Plane Select
 - G18 - ZX Plane Select
 - G19 - YZ Plane Select

13.3 Angle of Rotation - R Word

The R word is used to define the angle of rotation.

- The R word is programmed in units of $.001^\circ$ without a decimal point. Example R1000 is 1° rotation in the counterclockwise direction.
- The R word is programmable in the range of:
-360000 to +360000
- A plus (+) R value is counterclockwise rotation. The (+) sign may be omitted.
- A minus (-) R value is clockwise rotation. The (-) sign must be programmed.
- When the R word is omitted from the G68 block, the angle and direction of rotation defaults to the value set in parameter 5410.

13.3.1 Coordinates for Center of Rotation

The center point of rotation is programmed with absolute coordinates in the G68 block.

- When one or both coordinates are omitted in the G68 block, the current position of the omitted axis, or axes, is considered to be the center point.
- The first move to the part following the G68 block must be made with absolute coordinates. See Figure 13-1.

EFFECT OF INCREMENTAL MOVE FOLLOWING G68 BLOCK

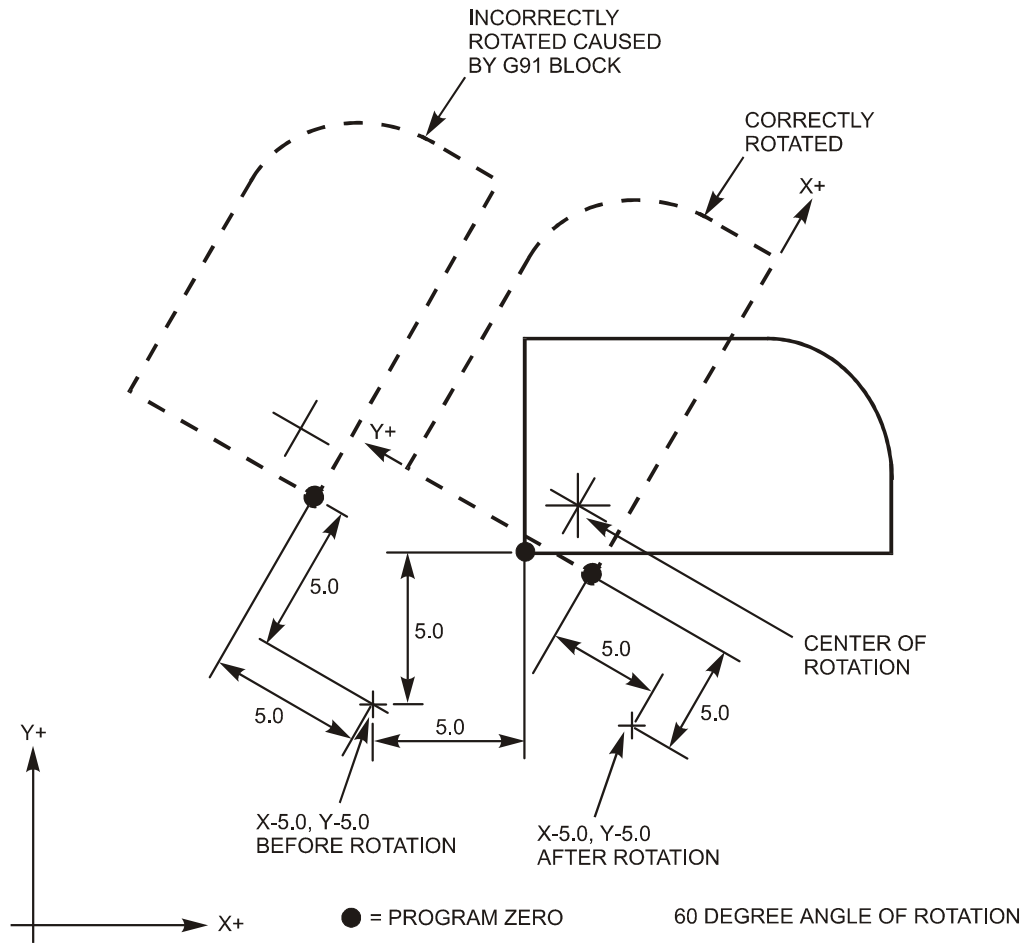


FIGURE 13-1 COORDINATE SYSTEM ROTATION EXAMPLE

PG-110

13.4 Using CSR with Other Programming Functions

13.4.1 CSR and Cutter Compensation

CSR can be programmed with Cutter Compensation C. Use the following guidelines when programming G40, G41, and G42 with G68 - CSR.

- G41 or G42 should be executed after executing the G68 code.
- G40 - CCC Cancel and G69 - Coordinate System Rotation Off can be programmed together in the same block.

13.4.2 CSR, Scaling, and Cutter Compensation

The coordinate system can be rotated and scaled at the same time.

- If G68 is executed while scaling is active (G51), the coordinate system rotation center point will also be scaled.
- The angle of rotation will not be scaled.
- When a move command is issued, scaling is applied first then the coordinates are rotated.
- When Cutter Compensation is not used, specify Scaling and Coordinate System Rotation in the following order:

Program Block	Block Description
N1000 G51 ;	Scaling On
N1010 G68 X___ Y___ R___ ;	Coordinate System Rotation On
N1020 ;	Machining Blocks
N1030 ;	
N1040 ;	
N1050 ;	
N1060 ;	
N1070 G69 ;	Coordinate System Rotation Off
N1080 G50 ;	Scaling Off
N1090 ;	Next machining operation

- When Cutter Compensation is used, specify Scaling, Coordinate System Rotation and Cutter Compensation in the following order:

Program Block	Block Description
N1000 G51 ;	Scaling On
N1010 G68 X___ Y___ R___ ;	Coordinate System Rotation On
N1020 G1 G41 X___ Y___ F___ D2 ;	Cutter Compensation On
N1030 ;	Machining Blocks
N1040 ;	
N1050 ;	
N1060 G1 G40 X___ Y___ ;	Cutter Compensation Off
N1070 G69 ;	Coordinate System Rotation Off
N1080 G50 ;	Scaling Off
N1090 ;	Next machining operation

Chapter 14 Table Programming

Index and NC Rotary Table Operation
and Programming

Makino Vertical and Horizontal
Machining Centers



Chapter 14

Table Programming

Contents

14.1 Overview	14-1
14.2 1° and 5° Index Tables	14-2
14.2.1 B Axis Coordinate System	14-2
14.2.2 B Axis Commands	14-3
14.2.3 MDI Operation	14-3
14.2.4 B Position for Pallet Change	14-4
14.2.4.1 Pallet Change Program	14-4
14.3 NC Rotary Tables	14-5
14.3.1 B Axis Coordinate System	14-5
14.3.2 B Axis Commands	14-6
14.3.3 MDI Operation	14-7
14.3.4 B Position for Pallet Change	14-8
14.3.4.1 Pallet Change Program	14-8

14.1 Overview

VMC/HMC machining centers may be equipped with one of three table types: 1° or 5° indexer or a 360,000 position NC rotary table. The table, regardless of type, is considered by the control to be the B (or 4th) axis.

All table types position bi-directionally and are indexed through MDI or program commands. The NC rotary table can also be indexed using manual controls.

This chapter presents operation, MDI input, and programming procedures for all table types.

If the machine is equipped with 1° or 5° index table, see [section 14.2 \(pg -2\)](#) for details. If an NC rotary table is supplied, see [section 14.3 \(pg -5\)](#) for details.

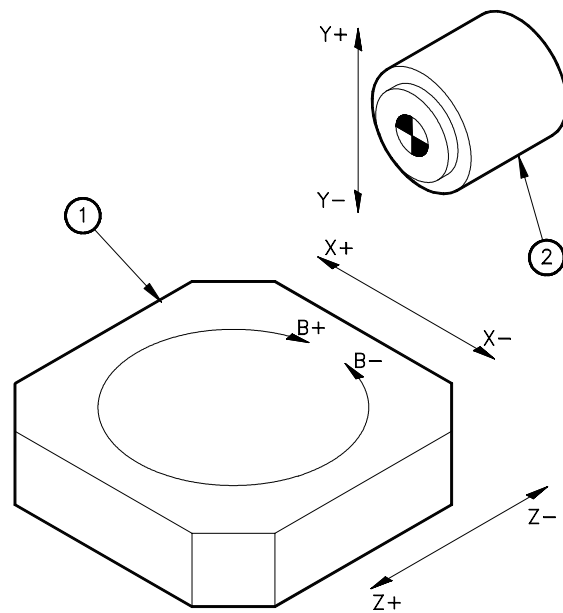
14.2 1° and 5° Index Tables

Operation, MDI input, and programming is the same for both the 1° and 5° index tables. B is the 4th axis in the machine coordinate system, (Figure 14-1).

The table is indexable in 1° or 5° increments, depending the table supplied.

A positive (or more positive) B command indexes the table clockwise.

A negative (or more negative) command indexes the table counterclockwise.



16IA03A001A

FIGURE 14-1 B AXIS CONFIGURATION

14.2.1 B Axis Coordinate System

Like the three primary axes (X, Y, and Z), the B axis coordinate system consists of absolute, relative, and machine coordinates.

B absolute coordinates – represents the B axis position relative to program zero. Work coordinate system offsets can be used to offset the B axis from Machine Zero like any of the primary axes. Offset values must be in 1° or 5° degree increments, depending on the table supplied.

B relative coordinates – represents where the B axis is relative to the position where its relative coordinate system was last originated (set to zero).

B machine coordinates – represents where the B axis is relative to B Machine Zero.

14.2.2 B Axis Commands

1° and 5° index tables are indexed by MDI input or program commands. The commands are the same in either case. Index tables cannot be positioned manually.

B word

Input range is **B0 to B±360** in 1° or 5° increments, depending on the table supplied.

The B word is programmable in the G90 (absolute) or G91 (incremental) input mode.

M15 code

A B positive (or more positive) command indexes the table in the clockwise direction. An M15 input in the same block with a B word forces a counterclockwise index.

EXAMPLE: From B0, a B270 command by itself causes a 270° index in the clockwise direction. From B0, a B270 M15 command causes a 90° index in the counterclockwise direction. In both cases, the table is indexed to the B270 position.

14.2.3 MDI Operation

Table indexes for setup or workpiece inspection must be commanded in the MDI mode. Manual controls are not provided.

To perform an MDI index:

1. Reference the primary axes; Z first.
 - Use the G28 reference return command, or the Auto Zero - Routine Function.
2. Select MDI.
3. Key-in Bxxx (xxx being the positive or negative index position.)
4. Press [EOB], [INSERT], then cycle [START].

14.2.4 B Position for Pallet Change

Prior to a pallet change, the B axis must be at its Machine Zero position. The MPC software interrogates the B machine coordinate system to determine if the B axis is actually at B0; since B absolute coordinates are influenced by work coordinate offsets and G92 register preset functions.

To index B to Machine Zero:

Reference the primary axes first before using either of the following procedures:

By G28 reference return:

G91 G28 B0 ;	Reference B axis.
G90 ;	Absolute input.

By machine coordinate programming:

G0 G53 B0 ;	B axis to Machine Zero.
--------------------	-------------------------

14.2.4.1 Pallet Change Program

The pallet change program, stored in CNC memory and called by M60, may be setup to automatically position the B axis to Machine Zero before executing the pallet change. In this case, it is not necessary to position B to Machine Zero before executing a pallet change M60 command.

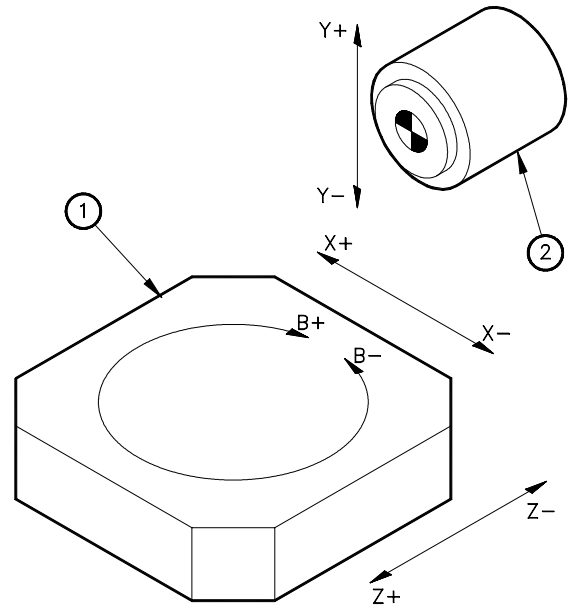
14.3 NC Rotary Tables

This section presents operation, MDI input, and programming for the NC rotary table. B is the 4th axis in the machine coordinate system, (Figure 14-2).

The NC rotary table is indexable in 0.001° increments.

A positive (or more positive) B command indexes the table clockwise.

A negative (or more negative) B command indexes the table counterclockwise.



16IA03A001A

FIGURE 14-2 B AXIS CONFIGURATION

14.3.1 B Axis Coordinate System

Like the three primary axes (X, Y, and Z), the B axis coordinate system consists of absolute, relative, and machine coordinates.

B absolute coordinates – represents the B axis position relative to program zero. Work coordinate system offsets can be used to offset the B axis from Machine Zero like any of the primary axes. Offset values can be in 0.001° degree increments.

B relative coordinates – represents where the B axis is relative to the position where its relative coordinate system was last originated (set to zero).

B machine coordinates – represents where the B axis is relative to B Machine Zero.

14.3.2 B Axis Commands

NC rotary tables are indexed by MDI input or program commands. The commands are the same in either case. Index tables can also be positioned manually.

B word

Input range is **B±99999.999** in 0.001° increments.

The B word is programmable in the G90 (absolute) or G91 (incremental) input mode.

M10/M11 - 4th Axis Clamp/Unclamp

NC rotary tables are equipped with a mechanical clamp to stabilize the axis during machining operations. The axis must be unclamped prior to an index then clamped before machining.

Both M10 and M11 should be programmed in blocks without other data.

M11 ;	4 th axis unclamp.
G0 G90 B90.0 ;	Index table to absolute 90° position at rapid.
M10 ;	4 th axis clamp.

G codes

Several G codes affect B axis programming.

G0 - Rapid Positioning	Positions the B axis at its rapid traverse rate. Any of the primary axes can be programmed in the same block.
G1 - Linear Interpolation	Positions the B axis at a feed rate specified by an F code. Any of the primary axes can be programmed in the same block.
G94 - Feed/Minute	For a B axis move only, the feed rate is in degrees per minute. When a B and any primary axis are programmed in the same block, the feed rate specified by the F code is a Feed Rate Number.
G93 - Inverse Time (option)	With this option, the F code is programmed as a feed distance per minute value. With G93, the F code is non-modal and is required in all motion blocks.

F code

The F code format varies depending on the active modes of G20, G21, G93, and G94.

F code in G94 mode.

With G94, the F code format is:

G20 - Inch Mode	Fxxxx.xx
G21 - Metric Mode	Fxxxxx.x

Calculating the FRN (Feed Rate Number)

When a simultaneous B and any other primary axis are programmed in the same block, the F code represents an FRN. To calculate the FRN:

$$\text{FRN} = \frac{\sqrt{\text{B axis move}^2 + \text{Linear axis move}^2}}{\text{Time to make linear move}}$$

F code in G93 mode.

With G93, the F code format is Fxxx.xxx for both Inch and Metric modes. With G93, the F code is non-modal and must be programmed in every motion block.

14.3.3 MDI Operation

Table indexes for setup or workpiece inspection must be commanded in the MDI mode. The example below is shown using the G0 Rapid Positioning and G90 Absolute modes.

To perform an MDI index:

1. Reference the primary axes; Z first.
 - Use the G28 reference return command, or the Auto Zero - Routine Function.
2. Select MDI.
3. Key-in the following blocks of information:
 - A. M11 then press [EOB] and [INSERT].
 - B. G0 G90 Bxxxxx.xxx then press [EOB] and [INSERT].
(xxxxx.xxx being the positive or negative index position.)
 - C. M10 then press [EOB], [INSERT], and cycle [START].

14.3.4 B Position for Pallet Change

Prior to a pallet change, the B axis must be at its Machine Zero position. The MPC software interrogates the B machine coordinate system to determine if the B axis is actually at B0; since B absolute coordinates are influenced by work coordinate offsets and G92 register preset functions.

The Pallet Change program, called by M60 (M30 in the Random mode), may be setup to automatically position B to its Machine Zero position. If it is not, position B to Machine Zero by one of two examples shown below.

To index B to Machine Zero:

Reference the primary axes first before using either of the following procedures:

By G28 reference return:

M11 ;	4 th axis unclamp.
G91 G28 B0 ;	Reference B axis.
M10 ;	4 th axis clamp.
G90 ;	Absolute input.

By machine coordinate programming:

M11 ;	4 th axis unclamp.
G0 G53 B0 ;	B axis to Machine Zero.
M10 ;	4 th axis clamp.

14.3.4.1 Pallet Change Program

The pallet change program, stored in CNC memory and called by M60, may be setup to automatically position the B axis to Machine Zero before executing the pallet change. In this case, it is not necessary to position B to Machine Zero before executing a pallet change M60 command.

Chapter 15 G and M Codes

Preparatory Functions - G Codes and
Miscellaneous Functions - M Codes

Makino Vertical and Horizontal
Machining Centers



Chapter 15

G and M Codes

Contents

15.1 Preparatory Functions - G Codes	15-1
15.1.1 G Code Groups	15-1
15.1.2 Modal and Non-Modal G Codes	15-2
15.1.3 G Code Table	15-3
15.2 Miscellaneous Functions - M Codes	15-7
15.2.1 M Code Types and Classes	15-7
15.2.2 M Codes Listed by Function	15-8
15.2.2.1 Standard M Codes	15-8
15.2.2.2 4 th Axis (Full Rotary)/5 th Axis	15-9
15.2.2.3 Block Skip	15-9
15.2.2.4 Coolant and Chip Control	15-10
15.2.2.5 Pallet Changer	15-11
15.2.2.6 Spindle Control	15-11
15.2.2.7 Measuring Systems	15-12
15.2.2.8 Tool Data Management M Codes	15-13
15.2.2.9 Tool Monitoring (AC, SL, and TL)	15-13
15.2.2.10 Spare Tool	15-14
15.2.2.11 Acoustic Emission	15-14
15.2.2.12 GI/SGI M Codes	15-14
15.2.2.13 User I/O M Codes	15-14
15.2.2.14 Remote Control Nozzle	15-15
15.2.2.15 Maintenance M Codes	15-17
15.2.2.16 Miscellaneous (3 Digit) M Codes	15-18
15.2.3 M Codes Listed in Numerical Order	15-20

15.1 Preparatory Functions - G Codes

Preparatory Functions (G codes) establish the conditions under which the CNC operates during the execution of a part program or MDI commands.

15.1.1 G Code Groups

G codes are divided into several groups by function. It is normal for G codes of different groups to reside in the same block of data. However, since the function of G codes within the same group are related, use only one G code from the same group per block.

- Only one G code from a group can be used in a block of information.
- If two or more G codes of the same group are placed in the same, the CNC acts upon the last appearing g code from that group.

Example:

```
N230 G0 G1 X10.5 Y5.43 ;
```

The CNC acts upon the function "G01". If a feedrate is not active, a PS alarm is generated by the CNC. In a case where a feedrate had been called prior to the execution of this block, the X and Y axes will feed to the coordinate positions at that feedrate.

NOTE:

Group 03 (G90/G91) is an exception to the above example. G90 (Absolute Input) and G91 (Incremental Input) are acted upon by the CNC from the point at which they appear in the block. To ensure proper positioning, place G90 or G91 at the beginning of the block in which the positioning system is to be changed.

Example:

```
N230 G90 X8.75 G91 Y13.4 ;
```

X8.75 is an Absolute coordinate and Y13.4 is an incremental move.

15.1.2 Modal and Non-Modal G Codes

G codes are of two basic types, Modal and Non-Modal.

Modal G Codes

Modal G codes are effective from the time they are initiated until they are canceled or replaced by another G code from the same group. Several G codes are set to become active when the CNC is turned "ON".

Non-Modal G Codes

Non-Modal G codes are effective only for the block in which they are specified. Most of these "one shot" G codes reside in Group 00.

15.1.3 G Code Table

Table 15-1 lists the G codes available on the Pro 3 control. This table does not differentiate between standard and optional G codes. Programming an optional G code that has not been supplied will result in a PS programming alarm.

- G codes preceded by an asterisk (*) are parameter settable to be the initialized state when the control is turned On, after a RESET, and at end of program (M2 or M30).
- Function descriptions preceded by a sharp sign (#) are considered basic on MAKINO machine tools.

TABLE 15-1 PREPARATORY FUNCTIONS - G CODES

G Code	Group	Function	
* G00	01	# Rapid Positioning	
* G01		# Linear Interpolation (Feed Rate)	
G02		# Circular/Helical Interpolation CW	
G03		# Circular/Helical Interpolation CCW	
G04	00	# Dwell	
G05		High Speed Remote Buffer Operation	
G07.1		Cylindrical Interpolation	
G09		# Exact Stop Check	
G10		# Data Setting	
G11		# Data Setting Mode Cancel	
G12.1	25	Polar Coordinate Interpolation Mode	
G13.1		Polar Coordinate Interpolation Mode Cancel	
* G15	18	Polar Coordinate Command Cancel	
G16		Polar Coordinate Command	
* G17	02	# XpYp Plane	Xp = X or Parallel Axis
* G18		# ZpXp Plane	Yp = Y or Parallel Axis
* G19		# YpZp Plane	Zp = Z or Parallel Axis
G20	06	# Inch Input	
G21		# Metric Input	
* G22	04	Stored Stroke Check Function On	
G23		Stored Stroke Check Function Off	
G25	24	Spindle Speed Fluctuation Off	
G26		Spindle Speed Fluctuation On	

TABLE 15-1 PREPARATORY FUNCTIONS - G CODES

G Code	Group	Function
G27	00	# Reference Point Return Check
G28		# Automatic Reference Point Return
G29		# Return From Reference Point
G30		Return To 2nd, 3rd, 4th Reference Point
G30.1		Floating Reference Return
G31		# Skip Function
G33	01	Thread Cutting
G37	00	Tool Length Automatic Measurement
G39		Cutter Radius Compensation Corner Rounding
* G40	07	#/ Cutter Radius Compensation Cancel / 3 Dimensional Tool Comp. Cancel
G41		#/ Cutter Radius Compensation Left / 3 Dimensional Tool Comp. Left
G42		#/ Cutter Radius Compensation Right / 3 Dimensional Tool Comp. Right
G40.1	19	Normal Direction Control Cancel Mode
G41.1		Normal Direction Control Left Side On
G42.1		Normal Direction Control Right Side On
G43	08	# Tool Length Compensation + Direction
G44		# Tool Length Compensation - Direction
G45	00	# Tool Offset Increase
G46		# Tool Offset Decrease
G47		# Tool Offset Double Increase
G48		# Tool Offset Double Decrease
* G49	08	# Tool Length Compensation Cancel
* G50	11	Scaling Cancel
G51		Scaling
* G50.1	18	# Programmable Mirror Image Cancel
G51.1		# Programmable Mirror Image On
G52	00	Local Coordinate System Setting
G53		Machine Coordinate System Selection

TABLE 15-1 PREPARATORY FUNCTIONS - G CODES

G Code	Group	Function	
* G54	14	# Work Coordinate System 1 Selection	
G54.1		# Additional Work Coordinate System Selection	
G55		# Work Coordinate System 2 Selection	
G56		# Work Coordinate System 3 Selection	
G57		# Work Coordinate System 4 Selection	
G58		# Work Coordinate System 5 Selection	
G59		# Work Coordinate System 6 Selection	
G60	00	# Single Direction Positioning	
G61	15	# Exact Stop Mode	
G53		Machine Coordinate System Selection	
* G54	14	# Work Coordinate System 1 Selection	
G62		# Automatic Corner Override Mode	
G63		# Tapping Mode	
G64		# Cutting Mode	
G65	00	# Macro Call	
G66		# Macro Call A	
* G67		# Macro Call A/b Cancel	
G68	16	Coordinate System Rotation	
* G69		Coordinate System Rotation Cancel	
G73	09	# Peck Drilling Cycle	
G74		# Counter Tapping Cycle (Left Hand Tapping)	
G76		# Fine Boring Cycle	
* G80		# Canned Cycle Cancel / Ext. Operation Function Cancel	
G81		# Drilling Cycle, Spot Boring / Ext. Operation	
G82		# Drilling Cycle, Counter Boring	
G83		# Peck Drilling Cycle	
G84		# Tapping (Right Hand Tapping)	
G85		# Boring Cycle	
G86		# Boring Cycle	
G87		# Back Boring Cycle	
G88		# Boring Cycle	
G89		# Boring Cycle	
* G90		03	# Absolute Command (Position)
* G91			# Incremental Command (Positioning)

TABLE 15-1 PREPARATORY FUNCTIONS - G CODES

G Code	Group	Function
G92	00	#/ Absolute Zero Preset
* G94		# Feed Per Minute
G95		Feed Per Revolution
G96	17	Constant Surface Feed Control
* G97		Constant Surface Feed Control Cancel
* G98	10	# Canned Cycle Initial Level Return
G99		# Canned Cycle R Point Level Return

15.2 Miscellaneous Functions - M Codes

M codes perform specific functions defined by the machine tool builder. The majority of the M codes provided in this section are for machine options which may not be supplied on your machine.

The M codes listed in [section 15.2.2](#) are divided into groups by specific functions. The M codes listed in [section 15.2.3](#) are listed in numeric order.

15.2.1 M Code Types and Classes

M Code Types

M codes are of two basic types:

- Modal
 - Modal M codes are effective from the time they are executed until they are canceled or replaced by another related M code.
- Non-Modal
 - Non-Modal M codes are effective only for the block in which they are specified. The Non-Modal M codes once executed are inactive until executed again.

M Code Classes

M codes are of three classes. In the following M code tables, the **Effect.** (Effectivity) column contains an A, W, or S. This terminology means:

A

M codes classified with an A are acted upon after axis commands programmed in the same block.

W

M codes classified with an W are acted upon before axis commands programmed in the same block.

S

M codes classified with an S must be programmed in a block without other data, except a sequence number.

15.2.2 M Codes Listed by Function

15.2.2.1 Standard M Codes

M Code	Effect.	Description
M000	A	Program Stop
M001	A	Optional Stop
M002	A	End Of Program
M003	W	Spindle Start In Cw Direction
M004	W	Spindle Start In Ccw Direction
M005	A	Spindle Stop
M006	W	Automatic Tool Change
M007	W	Mist Coolant Or Air Blow On
M008	W	Nozzle Coolant On
M009	A	Coolant Off
M015	W	Index Table Ccw Rotation
M019	A	Spindle Orientation
M021	S	X Axis Mirror Image On
M022	S	Y Axis Mirror Image On
M023	S	Mirror Image Off
M029	S	Unconditional Power Out
M030	A	End Of Program & Rewind
M048	W	M49 Mode Cancel
M049	A	Feed Rate Override Cancel
M057	S	Tool Number Registering Mode On
M058	S	M59 Mode Cancel
M059	S	Spindle Speed Override Cancel
M068	S	No. 2 M-code Designation
M098	A	Sub-program Access
M099	S	Return To Main Program

15.2.2.2 4th Axis (Full Rotary)/5th Axis

M Code	Effect.	Description
M010	A	NC Rotary Table Clamp
M011	S	NC Rotary Table Unclamp
M012	A	5th Axis Clamp
M013	S	5th Axis Unclamp
M016	S	4th Axis Mirror Image On

15.2.2.3 Block Skip

M Code	Effect.	Description
M140	S	Automatic Block Skip On
M141	S	Automatic Block Skip Off
M142	S	Block Skip Return
M146	S	0° Surface Ready Control
M147	S	90° Surface Ready Control
M148	S	180° Surface Ready Control
M149	S	270° Surface Ready Control
M150	S	Block Skip (2 To 9) Off
M152	S	Block Skip 2 On
M153	S	Block Skip 3 On
M154	S	Block Skip 4 On
M155	S	Block Skip 5 On
M156	S	Block Skip 6 On
M157	S	Block Skip 7 On
M158	S	Block Skip 8 On
M159	S	Block Skip 9 On

15.2.2.4 Coolant and Chip Control

M Code	Effect.	Description
M007	W	Mist Coolant Or Air Blow On
M008	W	Nozzle Coolant On
M009	A	Coolant Off
M026	W	Through Spindle Coolant On
M027	W	Tap Oil.(Mist, Jet) On
M028	W	Through Tool Air On
M077	S	Through Spindle Air On
M096	W	Through Tool Coolant On
M097	W	High Power Coolant On
M301	S	Coolant And Spdl Pause And Store
M308	S	Coolant And Spindle, Recover
M312	S	Coolant Stop And Store
M313	S	Coolant Recover
M740	S	Spiral Chip Conveyor Off
M741	W	Spiral Chip Conveyor On
M742	S	Oil Skimmer Off
M743	S	Oil Skimmer On
M752	S	Mist Coolant On
M753	S	Work Clean Cool Or Clean Air On
M754	S	Base Coolant On
M755	S	TELSCO Coolant On
M756	S	Air Blow On
M757	A	TELSCO Coolant Off
M758	S	Work Clean Cool Or Clean Air Off

15.2.2.5 Pallet Changer

M Code	Effect.	Description
M051	S	M52 Mode Cancel
M052	S	Auto. Unload Inhibit Mode At ALM
M060	A	Pallet Change
M061	A	Pallet Change
M062	A	Pallet Change
M063	A	Pallet Change
M064	A	Pallet Change
M065	A	Pallet Change
M066	A	Pallet Change
M067	A	Pallet Change
M078	S	Revolv Light1 On No Ready Pallet
M079	S	Revolv Light2 On No Ready Pallet
M080	A	Pallet Change
M084	S	Pallet 1 Load Check
M095	S	Pallet 2 Load Check
M600	A	Pallet Change

15.2.2.6 Spindle Control

M Code	Effect.	Description
M003	W	Spindle Start In CW Direction
M004	W	Spindle Start In CCW Direction
M005	A	Spindle Stop
M019	A	Spindle Orientation
M058	S	M59 Mode Cancel
M059	S	Spindle Speed Override Cancel
M135	W	Rigid Tap Mode On
M301	S	Coolant And Spdl Pause And Store
M308	S	Coolant And Spindle, Recover
M310	S	Spindle Stop And Store
M311	S	Spindle Recover
M319	W	Pre-orientation

15.2.2.7 Measuring Systems

M Code	Effect.	Description
BROKEN TOOL SENSOR		
M014	A	BTS Arm Advance
M018	A	BTS Arm Retract
M020	A	BTS Check
M433	S	BTS LS300 Off Check
AUTO TOOL LENGTH MEASUREMENT		
M032	S	Tool Length Auto Measuring Mode
M033	S	Tool Length Circuit Check
M036	S	Prior Tool Check Mode
M037	S	Tool Data Register Mode Off
M920	S	Tool Length Measuring Registration
M921	S	Tool Diameter Measuring Registration
M922	S	M920, M921 Addition Mode
WORK AUTO MEASUREMENT SYSTEM (AMS)		
M031	S	Work Automatic Measuring Mode
M034	S	Auto Measure & Call Of Spare Tool
M035	S	Auto Measure & Pallet Unloading
M085	S	Auto Measuring Approach Mode
M086	S	Auto Measuring Fast Feed Mode
M747	S	Measuring Head Cleaning Air On
M748	W	IMM Cleaning Air Unconditional On
M749	A	IMM Cleaning Air Unconditional Off
M750	W	IMM Cleaning Air On
M751	A	IMM Cleaning Air Off
M910	S	Work Auto. Measuring M31 Mode Cancel
M911	S	MP7 Measuring Head Alarm Check
M941	S	MP7 Measuring Head Check
M949	S	Work Measure System Is Abnormal (Machining Restart)
M957	S	Work Measure System Is Abnormal (Machining Abnormal)
M961	S	External Skip Mode Off
M962	S	External Skip Mode On

15.2.2.8 Tool Data Management M Codes

M Code	Effect.	Description
M233	S	Tool Data Dump
M234	S	Print Out Tool Data
M932	S	INT Data Register Mode On
M933	S	Tool Type Data Register Mode On
M934	S	Tool Alarm Data Register Mode On
M935	S	Tool Status Data Register Mode On
M936	S	Tool Size Data Register Mode On
M975	S	Tool Kind Data Register Mode On
M979	S	Air Blow Time Data Register Mode On

15.2.2.9 Tool Monitoring (AC, SL, and TL)

M Code	Effect.	Description
M038	S	AC, SL Value Temporary Alter Mode On
M039	S	AC, SL Value Temporary Alteration
M040	S	TL Remain. Data Register Mode On
M041	S	SL Coefficient Setting 125%
M042	S	SL Coefficient Setting 150%
M043	S	SL Coefficient Setting 200%
M044	S	SL, AC Coefficient 0%
M046	S	SL Data Register Mode On
M047	S	AC Data Register Mode On
M050	S	AC Coefficient 100%, SI Coefficient 200%
M094	S	Tool Data Delete Mode On
M469	S	TL Alarm Suspension Mode On
M930	S	Tool Life Data Register Mode On
M931	S	Tool Remain Data Register Mode On
M934	S	Tool Alarm Data Register Mode On

15.2.2.10 Spare Tool

M Code	Effect.	Description
M053	S	Tool Offset Data Register Mode
M054	S	Tool Offset Minus Regist. Mode
M056	S	Tool Offset Data Transfer Command
M093	S	Spare Tool Register Mode On
M094	S	Tool Data Delete Mode On
M105	S	Direct Spare Tool Change
M106	S	Direct Spindle Tool Change Inhibit Mode
M107	S	Direct Spindle Tool Change Permit Mode

15.2.2.11 Acoustic Emission

M Code	Effect.	Description
M090	S	AE Function Off
M091	S	AE Mode
M092	S	AE Small Diameter Tool Mode (Less Than 5 mm)

15.2.2.12 GI/SGI M Codes

M Code	Effect.	Description
M250	S	GI/SGI Standard Mode
M251	S	GI/SGI High Speed Mode
M252	S	GI/SGI High Accuracy Mode
M253	S	GI/SGI Special Mode

15.2.2.13 User I/O M Codes

M Code	Effect.	Description
M580	S	User I/O M Code
M581	S	User I/O M Code
M582	S	User I/O M Code
M583	S	User I/O M Code
M584	S	User I/O M Code
M585	S	User I/O M Code
M586	S	User I/O M Code
M587	S	User I/O M Code

15.2.2.14 Remote Control Nozzle

M Code	Effect.	Description
M801	W	Remote Control Nozzle Position 1
M802	W	Remote Control Nozzle Position 2
M803	W	Remote Control Nozzle Position 3
M804	W	Remote Control Nozzle Position 4
M805	W	Remote Control Nozzle Position 5
M806	W	Remote Control Nozzle Position 6
M807	W	Remote Control Nozzle Position 7
M808	W	Remote Control Nozzle Position 8
M809	W	Remote Control Nozzle Position 9
M810	W	Remote Control Nozzle Position 10
M811	W	Remote Control Nozzle Position 11
M812	W	Remote Control Nozzle Position 12
M813	W	Remote Control Nozzle Position 13
M814	W	Remote Control Nozzle Position 14
M815	W	Remote Control Nozzle Position 15
M816	W	Remote Control Nozzle Position 16
M817	W	Remote Control Nozzle Position 17
M818	W	Remote Control Nozzle Position 18
M819	W	Remote Control Nozzle Position 19
M820	W	Remote Control Nozzle Position 20
M821	W	Remote Control Nozzle Position 21
M822	W	Remote Control Nozzle Position 22
M823	W	Remote Control Nozzle Position 23
M824	W	Remote Control Nozzle Position 24
M825	W	Remote Control Nozzle Position 25
M826	W	Remote Control Nozzle Position 26
M827	W	Remote Control Nozzle Position 27
M828	W	Remote Control Nozzle Position 28
M829	W	Remote Control Nozzle Position 29
M830	W	Remote Control Nozzle Position 30
M831	W	Remote Control Nozzle Position 31
M832	W	Remote Control Nozzle Position 32
M833	W	Remote Control Nozzle Position 33
M834	W	Remote Control Nozzle Position 34
M835	W	Remote Control Nozzle Position 35

M Code	Effect.	Description
M836	W	Remote Control Nozzle Position 36
M837	W	Remote Control Nozzle Position 37
M838	W	Remote Control Nozzle Position 38
M839	W	Remote Control Nozzle Position 39
M840	W	Remote Control Nozzle Position 40
M841	W	Remote Control Nozzle Position 41
M842	W	Remote Control Nozzle Position 42
M843	W	Remote Control Nozzle Position 43
M844	W	Remote Control Nozzle Position 44
M845	W	Remote Control Nozzle Position 45
M846	W	Remote Control Nozzle Position 46
M847	W	Remote Control Nozzle Position 47
M848	W	Remote Control Nozzle Position 48
M849	W	Remote Control Nozzle Position 49
M850	W	Remote Control Nozzle Position 50
M851	W	Remote Control Nozzle Position 51
M852	W	Remote Control Nozzle Position 52
M853	W	Remote Control Nozzle Position 53
M854	W	Remote Control Nozzle Position 54
M855	W	Remote Control Nozzle Position 55
M856	W	Remote Control Nozzle Position 56
M857	W	Remote Control Nozzle Position 57
M858	W	Remote Control Nozzle Position 58
M859	W	Remote Control Nozzle Position 59
M860	W	Remote Control Nozzle Position 60
M861	W	Remote Control Nozzle Position 61
M862	W	Remote Control Nozzle, 1 Back & Forth Movement
M863	W	Remote Control Nozzle, Contin. Back & Forth Movement
M870	W	Remote Control Nozzle User Set Position 1
M871	W	Remote Control Nozzle User Set Position 2
M872	W	Remote Control Nozzle User Set Position 3
M874	W	Remote Control Nozzle User Set Position 5
M875	W	Remote Control Nozzle User Set Position 6
M876	W	Remote Control Nozzle User Set Position 7
M877	W	Remote Control Nozzle User Set Position 8
M878	W	Remote Control Nozzle User Set Position 9
M879	W	Remote Control Nozzle User Set Position 10

15.2.2.15 Maintenance M Codes

MAINTENANCE CODES

The potential for injury or damage exists when using Maintenance M codes. Try using the PMC Maintenance screen functions first, as they are interlocked to prevent damage.

M Code	Description
M664	Maintenance M Function (Pot Tool Clamp)
M665	Maintenance M Function (Pot Tool Unclamp)
M666	Maintenance M Function (ATC Arm 180° CW)
M667	Maintenance M Function (ATC Arm 180° CCW)
M680	Maintenance M Function (ATC Arm 60° CW)
M681	Maintenance M Function (ATC Arm 60° CCW)
M682	Maintenance M Function (ATC Arm In)
M683	Maintenance M Function (ATC Arm Out)
M684	Maintenance M Function (Spindle Tool Clamp)
M685	Maintenance M Function (Spindle Tool Unclamp)
M686	Maintenance M Function (Pot Vertical)
M687	Maintenance M Function (Pot Horizontal)
M698	Maintenance Mode Off
M699	Maintenance Mode On

15.2.2.16 Miscellaneous (3 Digit) M Codes

M Code	Effect.	Description
M146	S	0° Surface Ready Control
M147	S	90° Surface Ready Control
M148	S	180° Surface Ready Control
M149	S	270° Surface Ready Control
M192	S	EFIN Cancel Mode Off
M193	S	EFIN Cancel Mode On
M230	S	Program Registration Mode Of Machine Parameters
M231	S	Program Output Of Machine Parameters
M232	S	Print Out Of Machine Parameters
M235	S	Send Machining End To Host
M302	S	Confirmation of M303, M304 Operation Finish
M303	S	Spindle Forward Command Return Already Complete
M304	S	Spindle Reverse Command Return Already Complete
M307	S	Coolant And Spindle Stop And Store
M315	S	Oilmatic Off
M316	S	Oilmatic On
M318	S	Orientation Setting Stop Position External
M325	S	Contouring Mode Off
M326	S	Contouring Mode On
M390	S	Spindle Tool Clamp
M391	S	Spindle Tool Unclamp
M392	S	Preparation Fro Spindle Tool Unclamp
M417	S	Confirm Pallet No. 1 On Machine Table
M418	S	Confirm Pallet No. 2 On Machine Table
M419	S	FIN A Confirm Of Pallet On Machine Table
M430	S	Pallet Stick Check
M462	S	Send Out Pallet No
M478	S	Record Machining Finish Time
M479	S	Record Machining Start Time
M601	S	B Axis Index Program End
M602	S	DNC Program End (One Touch Function)
M603	S	Tool Shunt End (One Touch Function)
M645	S	Spindle Tool Alarm Set And Spindle Tool Change
M710	S	Mist Colletor Off
M711	S	Mist Colletor On

M Code	Effect.	Description
M712	S	Turn Off Through Spindle Air
M736	S	Machine Light On
M737	S	Machine Light Off
M925	S	M926 Mode Cancel
M926	S	LC Ring Sensor Observation Mode

15.2.3 M Codes Listed in Numerical Order

M Code	Effect.	Description
M000	A	Program Stop
M001	A	Optional Stop
M002	A	End Of Program
M003	W	Spindle Start In CW Direction
M004	W	Spindle Start In CCW Direction
M005	A	Spindle Stop
M006	W	Automatic Tool Change
M007	W	Mist Coolant Or Air Blow On
M008	W	Nozzle Coolant On
M009	A	Coolant Off
M010	A	NC Rotary Table Clamp
M011	S	NC Rotary Table Unclamp
M012	A	5th Axis Clamp
M013	S	5th Axis Unclamp
M014	A	BTS Arm Advance
M015	W	Index Table CCW Rotation
M016	S	4th Axis Mirror Image On
M018	A	BTS Arm Retract
M019	A	Spindle Orientation
M020	A	Bts (Broken Tool Detection) Check
M021	S	X Axis Mirror Image On
M022	S	Y Axis Mirror Image On
M023	S	Mirror Image Off
M026	W	Through Spindle Coolant On
M027	W	Tap Oil (Mist, Jet) On
M028	W	Through Tool Air On
M029	S	Unconditional Power Cut
M030	A	End Of Program And Rewind
M031	S	Work Automatic Measuring Mode
M032	S	Tool Length Auto Measuring Mode
M033	S	Tool Length Circuit Check
M034	S	Auto Measure & Call Of Spare Tool
M035	S	Auto Measure & Pallet Unloading
M036	S	Prior Tool Check Mode
M037	S	Tool Data Register Mode Off

M Code	Effect.	Description
M038	S	AC, SL Value Temporary Alter Mode On
M039	S	AC, SL Value Temporary Alteration
M040	S	TL Remain Data Register Mode On
M041	S	SL Coefficient Setting 125%
M042	S	SL Coefficient Setting 150%
M043	S	SL Coefficient Setting 200%
M044	S	SL/ AC Coefficient 0%
M046	S	SL Data Register Mode On
M047	S	AC Data Register Mode On
M048	W	M49 Mode Cancel
M049	A	Feed Rate Override Cancel
M050	S	AC Coefficient 100%, SL Coefficient 200%
M051	S	M52 Mode Cancel
M052	S	Auto. Unload Inhibit Mode At ALM
M053	S	Tool Offset Data Register Mode
M054	S	Tool Offset Minus Register Mode
M056	S	Tool Offset Data Transfer Command
M057	S	Tool Number Registering Mode On
M058	S	M59 Mode Cancel
M059	S	Spindle Speed Override Cancel
M060	A	Pallet Change
M061	A	Pallet Change
M062	A	Pallet Change
M063	A	Pallet Change
M064	A	Pallet Change
M065	A	Pallet Change
M066	A	Pallet Change
M067	A	Pallet Change
M068	S	No. 2 M-Code Designation
M077	S	Through Spindle Air On
M078	S	Revolving Light1 On No Ready Pallet
M079	S	Revolving Light2 On No Ready Pallet
M080	A	Pallet Change
M081	S	Local Tracer Mode (For Copy Machines Only)
M082	S	Local Cutter Mode (For Copy Machines Only)
M083	S	Tracer/Cutter Mode (For Copy Machines Only)
M084	S	Pallet 1 Load Check

M Code	Effect.	Description
M085	S	Auto. Measuring Approach Mode
M086	S	Auto. Measuring Fast Feed Mode
M090	S	AE Function Off
M091	S	AE Mode
M092	S	AE Small-Diameter Tool Mode
M093	S	Spare Tool Register Mode On
M094	S	Tool Data Delete Mode On
M095	S	Pallet 2 Load Check
M096	W	Through Tool Coolant On
M097	W	High Power Coolant On
M098	A	Sub-Program Access
M099	A	Return To Main Program
M105	S	Direct Spare Tool Change
M106	S	Direct SP Tool Chg Inhibit Mode
M107	S	Direct SP Tool Chg Permit Mode
M135	W	Rigid Tap Mode On
M140	S	Automatic Block Skip On
M141	S	Automatic Block Skip Off
M142	S	Block Skip Return
M146	S	0° Surface Ready Control
M147	S	90° Surface Ready Control
M148	S	180° Surface Ready Control
M149	S	270° Surface Ready Control
M150	S	Block Skip (2 To 9) Off
M152	S	Block Skip 2 On
M153	S	Block Skip 3 On
M154	S	Block Skip 4 On
M155	S	Block Skip 5 On
M156	S	Block Skip 6 On
M157	S	Block Skip 7 On
M158	S	Block Skip 8 On
M159	S	Block Skip 9 On
M192	S	EFIN Cancel Mode Off
M193	S	EFIN Cancel Mode On
M230	S	Program Registration Mode Of Machine Parameters
M231	S	Program Output Of Machine Parameters
M232	S	Print Out Of Machine Parameters

M Code	Effect.	Description
M233	S	Tool Data Dump
M234	S	Printout Tool Data
M235	S	Send Machining End To Host
M250	S	GI/SGI Standard Mode
M251	S	GI/SGI High Speed Mode
M252	S	GI/SGI High Accuracy Mode
M253	S	GI/SGI Special Mode
M301	S	Coolant And Spdl Pause And Store
M302	S	Confirmation of M303, M304 Operation Finish
M303	S	Spindle Forward Command Return Already Complete
M304	S	Spindle Reverse Command Return Already Complete
M307	S	Coolant And Spindle Stop And Store
M308	S	Coolant And Spindle Recover
M310	S	Spindle Stop And Store
M311	S	Spindle Recover
M312	S	Coolant Stop And Store
M313	S	Coolant Recover
M315	S	Oilmatic Off
M316	S	Oilmatic On
M318	S	Orientation Setting Stop Position External
M319	W	Pre-orientation
M325	S	Contouring Mode Off
M326	S	Contouring Mode On
M390	S	Spindle Tool Clamp
M391	S	Spindle Tool Unclamp
M392	S	Preparation Fro Spindle Tool Unclamp
M417	S	Confirm Pallet No. 1 On Machine Table
M418	S	Confirm Pallet No. 2 On Machine Table
M419	S	FIN A Confirm Of Pallet On Machine Table
M430	S	Pallet Stick Check
M433	S	BTS LS300 Off Check
M462	S	Send Out Pallet No
M463	S	Printout Mode Output Required
M464	S	Command Spindle T-No. Output Request
M465	S	Spindle Number "0" Output
M466	S	Unload Pallet No. Appointed
M469	S	TL Alarm Suspension Mode On

M Code	Effect.	Description
M478	S	Record Machining Finish Time
M479	S	Record Machining Start Time
M480	S	Macro Data (#100 - #149) Upload
M580	S	User I/O M-Code
M581	S	User I/O M-Code
M582	S	User I/O M-Code
M583	S	User I/O M-Code
M584	S	User I/O M-Code
M585	S	User I/O M-Code
M586	S	User I/O M-Code
M587	S	User I/O M-Code
M600	A	Pallet Change
M601	S	B Axis Index Program End
M602	S	DNC Prog End (One-touch Function)
M603	S	Tool Shunt End (One-touch Func)
M608	S	Independent Carried In Function
M609	S	Independent Carried Out Function
M645	S	Spindle Tool Alarm Set And Spindle Tool Change
M661	S	ATC & B Axis Simultaneous Movement
M662	S	ATC & B Axis Simultaneous Movement
M664	S	Mainte M Func (Pot Tool Clamp)
M665	S	Mainte M Func (Pot Tool Unclamp)
M666	S	Mainte M Func (Atc Arm 180dec Cw)
M667	S	Mainte M Func (Atc Arm 180dec Ccw)
M680	S	Mainte M Func (Atc Arm 60dec Cw)
M681	S	Mainte M Func (Atc Arm 60dec Ccw)
M682	S	Mainte M Func (Atc Arm In)
M683	S	Mainte M Func (Atc Arm Out)
M684	S	Mainte M Func (Spindle Tool Clamp)
M685	S	Mainte M Func (Spindle Tool Unclamp)
M686	S	Mainte M Func (Pot Vertical)
M687	S	Mainte M Func (Pot Horizontal)
M698	S	Maintenance Mode Off
M699	S	Maintenance Mode On
M700	S	Coolant Oilmatic +0.5 C
M701	S	Coolant Oilmatic 0 C
M702	S	Coolant Oilmatic -0.5 C

M Code	Effect.	Description
M703	S	Coolant Oilmatic -1.0 C
M704	S	Coolant Oilmatic -1.5 C
M705	S	Coolant Oilmatic -2.0 C
M709	S	Don't Wait DEN On Coolant Stop
M710	S	Mist Colletor Off
M711	S	Mist Colletor On
M712	S	Turn Off Through Spindle Air
M736	S	Machine Light On
M737	S	Machine Light Off
M740	S	Spiral Chip Conveyor Off
M741	W	Spiral Chip Conveyor On
M742	S	Oil Skimmer Off
M743	S	Oil Skimmer On
M747	S	Measuring Head Cleaning Air On
M748	W	IMM Cleaning Air Unconditional On
M749	A	IMM Cleaning Air Unconditional Off
M750	W	IMM Cleaning Air On
M751	A	IMM Cleaning Air Off
M752	S	Mist Coolant On
M753	S	Work Clean Cool Or Clean Air On
M754	S	Base Coolant On
M755	S	TELSCO Coolant On
M756	S	Air Blow On
M757	A	TELSCO Coolant Off
M758	S	Work Clean Cool Or Clean Air Off
M801	W	Remote Control Nozzle Position 1
M802	W	Remote Control Nozzle Position 2
M803	W	Remote Control Nozzle Position 3
M804	W	Remote Control Nozzle Position 4
M805	W	Remote Control Nozzle Position 5
M806	W	Remote Control Nozzle Position 6
M807	W	Remote Control Nozzle Position 7
M808	W	Remote Control Nozzle Position 8
M809	W	Remote Control Nozzle Position 9
M810	W	Remote Control Nozzle Position 10
M811	W	Remote Control Nozzle Position 11
M812	W	Remote Control Nozzle Position 12

M Code	Effect.	Description
M813	W	Remote Control Nozzle Position 13
M814	W	Remote Control Nozzle Position 14
M815	W	Remote Control Nozzle Position 15
M816	W	Remote Control Nozzle Position 16
M817	W	Remote Control Nozzle Position 17
M818	W	Remote Control Nozzle Position 18
M819	W	Remote Control Nozzle Position 19
M820	W	Remote Control Nozzle Position 20
M821	W	Remote Control Nozzle Position 21
M822	W	Remote Control Nozzle Position 22
M823	W	Remote Control Nozzle Position 23
M824	W	Remote Control Nozzle Position 24
M825	W	Remote Control Nozzle Position 25
M826	W	Remote Control Nozzle Position 26
M827	W	Remote Control Nozzle Position 21
M828	W	Remote Control Nozzle Position 28
M829	W	Remote Control Nozzle Position 29
M830	W	Remote Control Nozzle Position 30
M831	W	Remote Control Nozzle Position 31
M832	W	Remote Control Nozzle Position 32
M833	W	Remote Control Nozzle Position 33
M834	W	Remote Control Nozzle Position 34
M835	W	Remote Control Nozzle Position 35
M836	W	Remote Control Nozzle Position 36
M837	W	Remote Control Nozzle Position 31
M838	W	Remote Control Nozzle Position 38
M839	W	Remote Control Nozzle Position 39
M840	W	Remote Control Nozzle Position 40
M841	W	Remote Control Nozzle Position 41
M842	W	Remote Control Nozzle Position 42
M843	W	Remote Control Nozzle Position 43
M844	W	Remote Control Nozzle Position 44
M845	W	Remote Control Nozzle Position 45
M846	W	Remote Control Nozzle Position 46
M847	W	Remote Control Nozzle Position 47
M848	W	Remote Control Nozzle Position 48
M849	W	Remote Control Nozzle Position 49

M Code	Effect.	Description
M850	W	Remote Control Nozzle Position 50
M851	W	Remote Control Nozzle Position 51
M852	W	Remote Control Nozzle Position 52
M853	W	Remote Control Nozzle Position 53
M854	W	Remote Control Nozzle Position 54
M855	W	Remote Control Nozzle Position 55
M856	W	Remote Control Nozzle Position 56
M857	W	Remote Control Nozzle Position 57
M858	W	Remote Control Nozzle Position 58
M859	W	Remote Control Nozzle Position 59
M860	W	Remote Control Nozzle Position 60
M861	W	Remote Control Nozzle Position 61
M862	W	R C Nozzle 1 Back & Forth Movement
M863	W	R C Nozzle Continuous Back & Forth Movement
M870	W	R C Nozzle User Set Position 1
M871	W	R C Nozzle User Set Position 2
M872	W	R C Nozzle User Set Position 3
M873	W	R C Nozzle User Set Position 4
M874	W	R C Nozzle User Set Position 5
M875	W	R C Nozzle User Set Position 6
M876	W	R C Nozzle User Set Position 7
M877	W	R C Nozzle User Set Position 8
M878	W	R C Nozzle User Set Position 9
M879	W	R C Nozzle User Set Position 10
M910	S	Work Auto. Measuring M31 Mode Cancel
M911	S	MP7 Measuring Head Alarm Check
M920	S	Tool Length Measuring Registration
M921	S	Tool Diameter Measuring Registration
M922	S	M920, M921 Addition Mode
M925	S	M926 Mode Cancel
M926	S	LC Ring Sensor Observation Mode
M927	S	Set First Used Tool Flag
M928	S	First Used Tool Flag Off
M929	S	Bad Result Of First Used Tool Flag
M930	S	Tool Life Data Register Mode On
M931	S	Tool Remain Data Register Mode On
M932	S	ITN Data Register Mode On

M Code	Effect.	Description
M933	S	Tool Type Data Register Mode On
M934	S	Tool Alarm Data Register Mode On
M935	S	Tool Status Data Register Mode On
M936	S	Tool Size Data Register Mode On
M938	S	RAKURAKU Checker: Change To Jog Mode
M939	S	RAKURAKU Checker: Finish
M940	S	Auto Finish At Work Result Abnormal
M941	S	MP7 Measuring Head Check
M949	S	Work Measuring System Is Abnormal (Machining Restart)
M957	S	Work Measuring System Is Abnormal (Machining Abnormal)
M961	S	External Skip Mode Off
M962	S	External Skip Mode On
M975	S	Tool Kind Data Register Mode On
M979	S	Air Blow Time Data Register Mode On