Programming of CNC Machines

Machines

4 EDITION

Ken Evans

```
07777(PBWY)
N100 G20
N110 G0 G17 G40 G49
N120 T232 M6
N130 G0 G90 G54 X-.1
N140 G43 H232 Z.25 M
N150 Z.2
N160 G1 Z.1 F6.42
N170 G2 X-.0679 Y-7.6
N180 X-.1929 Y-7.794 Z
N190 X-.3179 Y-7.669 Z
N200 X-.1929 Y-7.544 Z
N210 X-.1705 Y-7.546 Z
N220 X-.0679 Y-7.669 Z
N230 X-.1929 Y-7.794 Z
N240 X-.3179 Y-7.669 Z
N250 X-.1929 Y-7.544 Z
N260 X-.1705 Y-7.546 Z
N270 X-.0679 Y-7.669 Z
N280 X-.1929 Y-7.794 Z
N290 X-.3179 Y-7.669 Z
N300 X-.1929 Y-7.544 Z
N310 X-.1705 Y-7.546 Z
N320 X-.0679 Y-7.669 Z
N330 X-.1929 Y-7.794 Z
N340 X-.3179 Y-7.669 Z
N350 X-.1929 Y-7.544 Z
N360 X-.1705 Y-7.546 Z
N370 X-.0679 Y-7.669 Z
N380 X-.0702 Y-7.6927
N390 G1 X0. Y-7.7062
N400 G2 X-.0223 Y-7.7
N410 X-.0003 Y-7.9343
N420 X-.0025 Y-7.9998
N430 G1 Y-8.
N440 G3 X0. Y-8.0025
N450 X.0025 Y-8. I0. J.
N460 G1 Y-7.9998
N470 G2 X.0003 Y-7.93
N480 X.0223 Y-7.7259
N490 X0. Y-7.7062 I.73
N500 G1 Y-7.6017
N510 G2 X-.1062 Y-7.6
N520 X-.0754 Y-7.9343
N530 X-.0773 Y-7.9949
N540 G3 X-.0775 Y-8. I.
N550 X0. Y-8.0775 I.07
N560 X.0775 Y-8. I0. J.
N570 X.0773 Y-7.9949
N580 G2 X.0753 Y-7.93
```

```
N220 X-.0679 Y-7.669 Z
N230 X-.1929 Y-7.794 Z
N240 X-.3179 Y-7.669 Z
N250 X-.1929 Y-7.544 Z
N260 X-.1705 Y-7.546 Z
N270 X-.0679 Y-7.669 Z
N280 X-.1929 Y-7.794 Z
N290 X-.3179 Y-7.669 Z
N300 X-.1929 Y-7.544 Z
N310 X-.1705 Y-7.546 Z
N320 X-.0679 Y-7.669 Z
N330 X-.1929 Y-7.794 Z
N340 X-.3179 Y-7.669 Z
N350 X-.1929 Y-7.544 Z
N360 X-.1705 Y-7.546 Z
N370 X-.0679 Y-7.669 Z
N380 X-.0702 Y-7.6927
N390 G1 X0. Y-7.7062
N400 G2 X-.0223 Y-7.72
N410 X-.0003 Y-7.9343
N420 X-.0025 Y-7.9998
N430 G1 Y-8.
N440 G3 X0. Y-8.0025 I
N450 X.0025 Y-8. I0. J.0
N460 G1 Y-7.9998
N470 G2 X.0003 Y-7.93
N480 X.0223 Y-7.7259 I
N490 X0. Y-7.7062 I.732
N500 G1 Y-7.6017
N510 G2 X-.1062 Y-7.69
N520 X-.0754 Y-7.9343
N530 X-.0773 Y-7.9949
N540 G3 X-.0775 Y-8. I.
N550 X0. Y-8.0775 I.077
N560 X.0775 Y-8. I0. J.0
N570 X.0773 Y-7.9949 I
N580 G2 X.0753 Y-7.93
N590 X.1062 Y-7.6978 I
N600 X0. Y-7.6017 I.648
N610 G1 Y-7.4887
N620 G2 X-.1929 Y-7.66
N630 X-.1503 Y-7.9343
N640 X-.1522 Y-7.99 I-.
N650 G3 X-.1525 Y-8. I.
N660 X0. Y-8.1525 I.152
N670 X.1525 Y-8. I0. J.1
N680 X.1522 Y-7.99 I-.1
N690 G2 X.1504 Y-7.93
N700 X.1929 Y-7.669 I.8
N710 X0. Y-7.4887 I.56
```

FOURTH EDITION

Programming of CNC Machines

Ken Evans

Notice to the reader: While every possible effort has been made to insure the accuracy of the information presented herein, the authors and publisher express no guarantee of the same. The authors and publisher do not offer any warrant or guarantee that omissions or errors have not occurred and may not be held liable for any damages resulting from the use of this text by the readers. The readers accept the full responsibility for their own safety and that of the equipment used in connection with the instructions in this text. There has been no attempt to cover all controllers or machine types used in the industry and the reader should consult the operation and programming manuals of the machines they are using before any operation or programming is attempted.

Dedication

In Memory of my Father, who taught me many things in life,
especially to love Jesus, to work hard, and to have fun.

To my loving wife, Marci, for her support, patience, and proofreading skills,
and who kept me on track, and offered countless valuable critiques of
my machinist English. To Dolores for being the best mother-in-law
I could ever have imagined.

Thanks to the many students I have taught over the years and the readers
of this text. I hope this text will help you have many years of success and an
exciting and prosperous CNC programming career.

Table of Contents

Acknowledgments

About the Author

Ken Evans has held diverse machining and related jobs throughout his career and is currently a CNC Programmer at a prominent aerospace company. He learned the machinist trade in 1976 at Cessna Aircraft in Wichita, Kansas. Evans began his formal teaching career in 1984 at the T.H. Pickens Technical Center in Colorado, while working full-time as a CNC machinist and quality control inspector.

From 1991 to 2010, he served as a Machine Tool Technology instructor at Davis Applied Technology College in Kaysville, Utah, teaching foundational through advanced-level courses in the machining curriculum, including Mastercam CAD/CAM classes for students, educators, and private industry. He also was designated a certified Project Lead the Way, Computer Integrated Manufacturing (CIM) instructor. From 1997 to 2003, Ken worked for a local machine tool distributor in Salt Lake City as a MAZAK certified Training and Applications Specialist and one of the nation's first Mazatrol Conversational Programming instructors.

Ken loves the outdoors; he enjoys gardening, mountain biking, and golf.

Preface

I am a full-time CNC Programmer at a prominent aerospace company and was formerly a Machine Tool Technology instructor at a local college with over 35 years of CNC operation, setup, and programming experience. My strong interest in the practical application of CNC is at the heart of this text; therefore, all theoretical explanations are kept to a minimum to facilitate an easy read and to promote a quick understanding of programming. Because of the wide range of information available in this text about the selection of tools, cutting speeds, and the technology of machining, I hope this book will reach a variety of readers. Included among these are: Pre-Engineering students, those already involved in programming or maintaining CNC machines, operators of conventional machines who may want to expand their knowledge beyond conventional machining, and, managers or other interested persons who may wish to purchase such machines in the near future. Finally, I hope anyone with an interest in learning about modern CNC machining methods will find the book to be beneficial.

In this fourth edition, you will notice many features that will improve your reading experience. Chapter objectives are listed at the beginning of each chapter, specific terminology is presented, and study questions are added at the end of each chapter to confirm understanding. Throughout the text, figure captions are added to aid clarity.

In the first chapter, the foundation is laid with CNC Basics that set the tone for successful programming. The second chapter on CNC Machine Operation gives the reader perspective about CNC operation and setup procedures, because the first exposure a machinist has to CNC is usually as an operator. Operators will not be concerned right away with programming. However, after some time, practice, and the confidence of the owner, operators are given greater responsibility, e.g. changing wear offsets, performing setups, and minor program editing. The first and second chapters emphasize the development of machine setup and program editing skills. Students, machinists, supervisors, and design and manufacturing engineers will benefit from these chapters by learning foundational skills associated with setup and operation of CNC machine tools, prior to programming.

Chapters 3 and 4 focus on the components and development of program code for CNC turning and machining centers, with over 50 programming examples.

Today CNC programming is completed mostly by use of Computer-Aided Design and Computer-Aided Manufacturing (CAD/CAM); the popularity of Conversational Programming at the machine controller is increasing. Two chapters are devoted to the subject in this edition. Chapter 5, the CAD/CAM chapter, has been updated to include the current version of Mastercam X8

software. Chapter 6 is a new chapter that has been added to introduce Feature-Based Machining using Siemens NX9.0 CAM.

A new Chapter 7 has been added to feature FANUC NC Guide i programming for the popular Oi control. Step-by-step examples are given for Turning and Machining Center programs.

Chapter 8, the Mazatrol™ Conversational Programming chapter, has been expanded to include programming examples and study questions. An example program is created using MazaCAM™ off-line programming software by SolutionWare. Many new machine tool controllers come standard with some form of Conversational Programming, two of which are presented in this text.

The appendix contains many useful charts, techniques, and math formulas used for line-by-line programming for user reference. The glossary of terms has been expanded to include more definitions.

The Student Workbook for Programming of CNC Machines, Fourth Edition, is also available from Industrial Press as a companion to the text to provide a complete CNC training curriculum.

The purpose of this book is to expand the reader's current knowledge of CNC programming by providing full descriptions of all program functions and their practical applications. The book contains information on how to program turning and milling machines; this information is applicable to almost all control systems. The FANUC controller model is referenced here in order to provide clear explanations about one unified system. It is one of the most widely accepted, popular numerical control systems used worldwide.

Ken Evans, January 2016

PART 1

CNC Basics

OBJECTIVES:

1. Recognize the importance of safety when working with CNC machines.
2. Become familiar with tool and work holding methods for CNC machining.
3. Learn how to calculate proper feeds and speeds for CNC machining.
4. Learn how to plan for CNC programming by using process planning documents.
5. Become familiar with coordinate systems and their use in CNC programming.
6. Learn terminology associated with the basics of CNC.
7. Learn the ABCs of CNC program format.

SAFETY

As you begin to learn about CNC programming, it is important to become aware of and learn how to practice safe working habits. You should not operate any machine without first understanding the basic safety procedures necessary to protect yourself and others from injury and the equipment from damage. Most CNC machines are provided with a number of safety devices (door interlocks, etc.) that protect personnel and equipment from injury or damage. However, operators should not rely solely on these safety devices, but should operate the machine only after reading and fully understanding the safety precautions and basic operating practices outlined in the maintenance and operation manuals provided with the equipment. The following are some Do's and Don'ts that should be practiced when working with CNC machines.

Safety Rules for NC and CNC Machines

Do's:
- Wear safety glasses and safety shoes at all times.
- Know how to stop the machine under emergency conditions.
- Keep the surrounding area well lit, dry, and free from obstructions.
- Keep hands out of the path of moving parts during machining operations.
- Perform all setup procedures and loading or unloading of workpieces with the spindle stopped.
- Follow recommended safety policies and procedures when operating machinery, handling parts or tooling, and lifting.
- Make sure machine guards are in position during operation.
- Keep wrenches, tools, and parts away from the machine's moving parts.
- Make sure fixtures and workpieces are securely clamped before starting the machine.
- Inspect cutting tools for wear or damage prior to use.

Don'ts:
- Never operate a machine until properly instructed in its use.
- Never wear neckties, long sleeves, wristwatches, rings, gloves, or loose long hair when operating any machine.

- Never attempt to remove metal chips with hands or fingers.
- Never direct compressed air at yourself or others.
- Never operate an NC/CNC machine without first consulting the specific operator manual for the machine.
- Never place hands near a revolving spindle.
- Electrical cabinet doors are to be opened only by qualified personnel for maintenance purposes.

MAINTENANCE

A large investment has been made to purchase CNC equipment. It is very important to recognize the need for proper maintenance and a general upkeep of these machines. At the beginning of each opportunity to work on any turning or machining center, verify that all lubrication reservoirs are properly filled with the correct oils. The recommended oils are listed in the operation or maintenance manuals typically provided with the equipment. Sometimes there is a placard (plate) with a diagram of the machine and numbered locations for lubrication and the oil type is found on the machine. Most modern CNC machines have sensors that will not allow operation of the machine when the way or spindle oil levels are too low. Pneumatic (air) pressures need to be at a specified level and regulated properly. If the pressure is too low, some machine functions will not operate until the pressure is restored to normal. The standard air pressure setting is listed in pounds per square inch (PSI) and a pressure regulator is commonly located at the rear of the machine. Keeping machines maintained and in their optimum health is required to avoid costly failures and ensure maximum productivity. Check with your company and follow their Total Productive Maintenance (TPM) program. Refer to the operator or maintenance manuals for recommended maintenance activities.

Coolant Reservoir

The coolant tank level should be checked and adjusted as needed prior to use. A site glass is normally mounted on the tank for easy viewing. Use an acceptable water-soluble coolant mix, synthetic coolant, or cutting oil. Periodically the coolant tank should be cleaned and refilled. The coolant PH level should be checked routinely with a refractometer; the mixture should be adjusted in order to prevent bacterial growth. When synthetic coolants are used, the coolant system may stay clean longer.

Because machine slide-ways need constant lubrication during operation, automatic oiler systems inject an appropriate amount of oil at intervals determined by the builder. There is almost always excess oil that finds its way into the coolant system. Because of this condition, companies employ the use of add-on oil skimmers designed to clean the coolant. The "tramp oil" must be removed and discarded properly. Some new machine tools are incorporating sealed lubrication systems to help alleviate this problem.

Daily Maintenance Activities

Do's:

- Verify that all lubrication reservoirs are filled.
- Verify air pressure level by examining the regulator on the machine.

Part 1 CNC Basics

- Check that the chip pan, coolant level, and mixture are correct; clean or fill, as needed.
- Make sure that automatic chip removal equipment is operational when the machine is cutting metal.
- Be sure that the worktable and all mating surfaces are clean and free from nicks or burrs.
- Check to see that the chuck pressure setting is adequate for clamping the work to be machined.
- Clean up the machine at the end of use with a wet/dry vacuum or wash machine guards with coolant to remove chips from the working envelope.

Most new CNC machines are equipped with guards that envelop the worktable. The guards protect the ways and sensitive micro-switches installed as limit switches for table movement. Guards also help keep the surrounding floor space clean, but there is still the task of chip disposal. Some larger production machines incorporate a chip conveyor, which carries the chips to a drum on the floor on either side of the machine for easy removal. Even with these features, there is still a need for chip cleanup inside the working envelope at least once a day. If chips are allowed to gather within the guards, they will eventually find their way around the guards that protect the machine ways. Over time, some of the chips might become embedded into the ways and cause irreparable damage.

Another problem that may occur as the chips collect is that they bunch up and are pushed into contact with the micro-switches. This contact stops the machine from working because the switches send a signal to the control that indicates table travel limit has been exceeded. This message prevents the machine from operating until the chips are removed. If chips get within the guards around the micro-switches, it is necessary to remove those guards and clean. If this extent of cleaning becomes necessary, the machine should be turned off and a Lock-Out/Tag-Out should be incorporated to prevent injury. Remember: it is essential to replace the guards after cleanup.

It is very important to thoroughly clean the machine when many chips are present. The exterior of the machine usually will need only wiping down with a clean rag. You can clean the ways and the working envelope without damaging the machine by using coolant to wash the machine table and the guards free of chips. Another effective cleaning method is to use a wet/dry vacuum to pick up the chips. Along with the chip conveyor system, these two methods have proven hard to beat.

It is NOT recommended that you use compressed air to blow away the chips from the ways. It is, however, appropriate to use compressed air to remove chips and coolant from the workpiece itself or work holding fixtures such as a vise. The problem with using compressed air to clean up around the ways is that when chips are blasted away from the table, many are forced behind the guards, further worsening the micro-switch problem described above.

Last but not the least important is the cleanliness of the worktable, tools, and area. Be sure to clean off any metal chips and remove any nicks or burrs on the clamping or mating surfaces. *Always* clean the machine after use.

Tool Clamping Methods

Proper selection of cutting tools and work holding methods are paramount to the success of any machining operation. The scope of this text is not intended to teach all of the necessary information regarding tooling. You must consult the appropriate tooling catalogs, websites, and online resources for selection of tool holders and cutting tools that are relevant to the required operation. Least expensive is not necessarily best.

Sound machining principals require that the most rigid set-up possible be used that does not allow large overhangs of tools or workpieces. Ignoring these basic principles can cause tool and workpiece deflection and vibration that will contribute to poor surface finish and, eventually, tool damage, which also makes it difficult to maintain dimensional accuracy.

Just as with the rest of the machine tool, there are components used with the actual cutting tool that make it what it is. Obviously, the tool cutting edge is where the metal removal takes place. Without proper tool clamping, the cutting action may not produce the desired results. Therefore, it is very important to carefully select the most effective tool clamping method.

In the case of a simple operation of milling a contour on a part, we may select a collet or a positive locking (posi-lock) end mill holder for the end mill. The correct choice would depend on the actual features of the part to be machined and its dimensional tolerance. If the amount of metal to be removed is minimal and the tolerance allows, then a collet would probably suffice. But if a considerable amount of metal is to be removed (more than two-thirds of the tool diameter on a single depth of cut pass), then the posi-lock end mill holder selection is important. The reason for selecting the posi-lock holder is that under heavy cuts, a collet may not be able to grip the tool tightly enough. This situation could allow the tool to spin within the collet while cutting is in progress, with the result of ruining the collet and possibly damaging the part being machined. There is a tendency for the tool to dive into the workpiece when the tool spins within the collet and so damage to the part may occur.

Note: Most high speed steel (HSS) end mills have a flat ground on them to facilitate the use of the posi-lock holder. This flat area allows for a set-screw to lock into it, creating a rigid and stable tool clamping method. The clamping method for drills could be either a collet or a drill chuck. A keyed drill chuck usually is used for heavier metal removal or larger holes, whereas the keyless-type drill chuck is suitable for small holes. Generally, in the case of larger drills, a collet will be necessary to hold the tool. When holes are to be drilled, remember to center-drill or spot-drill first, so that the tool does not have a tendency to wander off location. The center-drill may be held in the same manner as a drill. For high volume/accuracy applications, hydraulic shrink-fit tool holders may perform best; when high rev/min are required, tool balancing is imperative for best accuracy.

For turning, the selection of the type of tool holder is determined by the finished part geometry and the part material. There are a variety of tool holder styles as well as indexable insert shapes available to accomplish the desired part shape and size.

For more information on the proper selection of inserts and tool holders, refer to the Machinery's Handbook section titled "Indexable Inserts".

Another valuable resource for technical data regarding the selection of inserts and tool holding are the ordering catalogs, online advice, and optimization applications from the tool and insert manufacturers.

Part 1 CNC Basics

CUTTING TOOL SELECTION

Cutting tools are a very important aspect of machining. If the improper tool and/or tool clamping method is used, the result will most likely be a poorly machined part. Always research and use the best tool and clamping method for a given operation. With the high speed and high performance of CNC machines, the proper selection process becomes increasingly important. The entire CNC machining process can be compromised by a lack of good tool planning and improper use.

There are many different types of machining operations performed on either turning or machining centers. The tool is where the action is, so if improper selection takes place here, the whole machining sequence will be affected. Years of study have been dedicated to this subject and are documented within reference manuals, buyer's guides, and online applications. Using these references will be helpful for correctly choosing a tool for a given operation.

Remember that in your selection process you are searching for the optimum metal-cutting conditions. The best way to understand how to choose the proper conditions is by studying the available data such as: the machine capabilities; the specific type of operation; the proper cutting tool(s) and tool clamping method(s); the geometry of the part to be made; the workpiece and cutter material; and the method of clamping the part.

It is important to utilize the most technologically advanced methods of metal removal available. Do not hesitate to research this new technology. For example, in recent years, there have been numerous cutting tool innovations that include indexable insert coatings and materials such as: Titanium Nitride (TiN); Titanium Carbon Nitride (TiCN) applied through Chemical Vapor Deposition (CVD), or Physical Vapor Deposition (PVD); Ceramic; Cubic Boron Nitride (CBN); and Polycrystalline Diamond (PCD). These advances have enabled increased cutting speeds and decreased tool wear, providing for higher production throughput. Another tool clamping innovation is modular tooling. This is a standardization of tool holders to facilitate the quick change of tools, decreasing setup time. *Refer to the tool and insert ordering catalogs and online applications from the tool and insert manufacturers for more information on modular tooling.*

TOOL COMPENSATION FACTORS

Important information about the tool must be given to the machine control unit (MCU) for the machine to be able to use the tool effectively. In other words, the MCU needs the tool identification number, the tool length offset (TLO), and the specific diameter of each tool. A TLO is a measurement given to the control unit to compensate for the tool length when movements are commanded. The cutter diameter compensation (CDC) offset is used by the control to compensate for the diameter of the tool during commanded movements.

The tool number identifies where the tool is located within the storage magazine or turret and often is the order sequence in which it is used. Each is assigned a tool length offset number. This number correlates with the pocket or turret position number and, in the case of a milling machine, is where the measured offset distance from the cutting tip to the spindle face is stored. For example, Tool No. 1 will have TLO No. 1. Finally, when milling, the diameter of the tool is compensated. In most cases, the programmer has taken the diameter of the tool into account. In other words, the programmed tool

path is written with a specific tool size in mind. However, more commonly the part geometry is programmed in order to facilitate the use of different tool diameters for a specified operation. When using the part geometry rather than the toolpath centerline for a specific tool diameter, an additional offset is called from within the program called cutter diameter compensation (CDC).

In many cases today, tool presetting and tool management systems are used to accurately input the TLO and diameter data values via network connection to the machine tool. This prevents the incorrect data from being entered via the keyboard and does not use machine time for measuring. Tools are also often fitted with Radio Frequency Identification (RFID) chips that carry this information to the machine tool.

Figure 1-1
Milling Tool Holder
Courtesy Kennametal
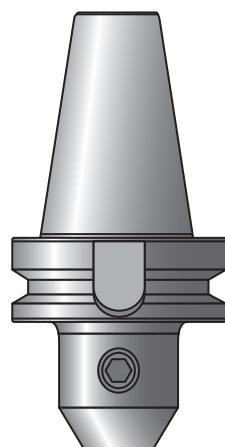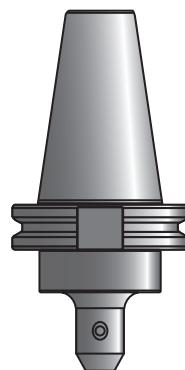
Figure 1-2
BT Tool Holder
Courtesy Kennametal

TOOL CHANGING

CNC equipment enables more efficient machining by allowing the combination of several operations into a single setup. This combination of operations requires the use of multiple cutting tools. An automatic tool changer (ATC) is a standard feature on most CNC machining centers, while many CNC knee-mills still require manual installation of the tool. The illustrations in this section show types of tool holders used on CNC machines; they have distinct physical differences and all of the holders are tapered. In Figure 1-1, the tool holder has only one ring and is designed for machines that require manual tool changes. The tools in Figures 1-2 (BT), 1-3 (CAT), and 1-4 (HSK) are designed for machines that have automatic tool changers. The rings act as a gripping surface for a tool changer. The tapered portion of the holder is the actual surface that is in contact with the mating taper of the spindle. These tapers are standardized by the industry and are numbered according to size. Common sizes for CAT tooling are: No. 30, 40, and 50. Common sizes for HSK tooling are: 40, 50, 63, 80, and 100.

Figures 1-3
CAT Tool Holder
Courtesy Kennametal

One benefit of these tapers over the standard R-8 Bridgeport style of tool holder is the increased surface area in contact with the mating taper of the spindle. The increased surface area makes the tool setup more rigid and stable. Big-Plus® style tool holders mate the taper and the flange to further increase the contacted surface area.
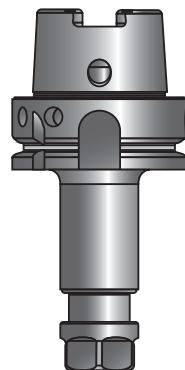
Figure 1-4
HSK Tool Holder
Courtesy Kennametal

Another feature on the tool holder is the notch or cutout on the centerline of the tool (there is an identical cutout on the opposite side). These cutouts enable axial orientation within the spindle and tool changer. As the holder is inserted into the spindle, the cutouts enable it to be locked into place in exactly the same orientation each and every time it is used. This orientation makes a real difference when trying to perform very precise operations such as boring a diameter. These notches also aid the spindle driving mechanism.

On CNC machines with a manual tool change, the holder is inserted into the machine and rotated until the holder pops into place (axial orientation is done by hand). The draw bar is then tightened to clamp the tool holder in place. Finally, another component of the CNC automatic tool changing system is the retention knob or pull-stud (Figures 1-5 and 1-6). Machining centers need the retention knob or pull-stud to pull the tool into the spindle and clamp the holder. This knob is threaded into the small end of the taper, as shown. Note: There are several styles of knobs available. The operator should consult the appropriate manufacturer manual for specifications required in their situation.

Figure 1-5
CAT Tool Holder With
Retention Knob
Courtesy Kennametal

METAL CUTTING FACTORS

Many tool and work holding methods used on manual machines are also used on CNC machines. The machines themselves differ in their method of control, but otherwise they are very similar. The major objective of CNC is to increase productivity and improve quality by consistently controlling the machining operation. Knowledge of the exact capabilities of the machine and its components, as well as the tooling involved, is imperative when working with CNC. It is

Figure 1-6
Retention Knob
Courtesy Kennametal

necessary for CNC programmers to have a thorough knowledge of the CNC machines they are responsible for programming. This may involve an ongoing process of research and training updates with the ultimate goal of obtaining a near optimum metal-cutting process. From this research and training come a decrease in the cycle time necessary to produce each part, thus lowering the per piece cost to the consumer. Fine tuning of the machining process for high-speed production gives more control over the quality of the product on a consistent basis. The following are some of the most important factors that affect the metal cutting process.

The Machine Tool

The machine used must have the physical ability to perform the machining. If the planned machining cut requires 10 horsepower from the spindle motor, a machine with

only 5 horsepower will not be an efficient one to use. It is important to work within the capabilities of the machine tool. The stability, rigidity, and repeatability of the machine are of paramount importance as well. Always take these things into consideration when planning for machining.

The Cutting Fluid or Coolant

The metal cutting process is one that creates friction between the cutting tool and the workpiece. A cutting fluid or coolant is necessary to lubricate and remove heat and chips from the tool and workpiece during cutting. Water alone is not sufficient because it only cools and does not lubricate; it will also cause rust to develop on the machine ways and table. Also, because of the heat produced, water vaporizes and thus compromises the cooling effect. A mixture of lard-based soluble oil and water creates a good coolant for most light metal-cutting operations. Harder materials, like stainless steel and high alloy composition steels require the use of a cutting-oil for the optimum results. Advancements have been made with synthetic coolants as well. Finally, the flow of coolant should be as strong as possible and be directed at the cutting edge to accomplish its purpose. Some machine tools are equipped thru-spindle/tool and high pressure coolant that really aid in cutting zone cooling and removal of chips. Programmers and machine operators should research available resources like the *Machinery's Handbook* and coolant manufacturer data, for information about the proper selection and use of cutting fluids for specific types of materials. The manufacturer data will include the coolant mixing ratio requirements and PH-level checking parameters.

The Workpiece and the Work Holding Method

The material to be machined has a definite effect on decisions about what tools will be used, the type of coolant necessary, and the selection of proper speeds and feeds for the metal-cutting operation.

The shape or geometry of the workpiece affects the metal-cutting operation and determines the type of work holding method that will be used. This clamping method is important for CNC work because of the high performance expected. It must hold the workpiece securely, be rigid, and minimize the possibility of any flex or movement of the part.

The Cutting Speed

Cutting speed is the rate at which the circumference of the tool moves past the workpiece in surface feet (sf/min) or meters per minute (m/min) to obtain satisfactory metal removal.

The cutting speed factor is most closely related to the tool life. Many years of research have been dedicated to this aspect of metal-cutting operations. The workpiece and the cutting tool material determine the recommended cutting speed. The *Machinery's Handbook* is an excellent source for information pertaining to determining proper cutting speed. If incorrect cutting speeds, spindle speeds, or feedrates are used, the results will be poor tool life, poor surface finishes, and even the possibility of damage to the tool and/or part.

Part 1 CNC Basics

The Spindle Speed

When referring to a milling or a turning operation, the spindle speed of the cutting tool or chuck must be accurately calculated relating to the conditions present. This speed is measured in revolutions per minute, r/min (formerly known as RPM), and is dependent upon the type and condition of material being machined. This factor, coupled with a depth of cut, gives the information necessary to find the horsepower required to perform a given operation. In order to create a highly productive machining operation, all these factors should be given careful consideration. Refer to the formulas below needed to calculate r/min.

For Inch Units:

$$r/\text{min} = \frac{12 \times CS}{\pi \times D}$$

For Metric Units:

$$r/\text{min} = \frac{1000 \times CS}{\pi \times D}$$

where:

CS = Cutting Speed from the charts in *Machinery's Handbook*

π = 3.1417

D = Diameter of the workpiece or the cutter

Many modern machine controllers have a feature that allows automatic calculation of feeds and speeds that is based on appropriate operator input of the cutting conditions. Often when using Computer Aided Manufacturing (CAM), feeds and speeds data can be extracted from available machining libraries.

The Feedrate

Feedrate is defined as the distance the tool travels along a given axis in a set amount of time, generally measured in inches per minute in/min (formerly known as IPM) for milling or inches per revolution in/rev (formerly known as IPR) for turning. This factor is dependent upon the selected tool type, the calculated spindle speed, and the depth of cut. Refer to the *Machinery's Handbook* and cutting tool manufacturer data for the chip load recommendations and review the formula below that is necessary to calculate this aspect of the metal-cutting operation.

$$F = R \times N \times f$$

where:

F = Feed in in/min or mm/min

R = r/min calculated from the preceding formula

N = the number of cutting edges

f = the chip load per tooth recommended from the *Machinery's Handbook*

The Depth of Cut

The depth of cut is determined by the amount of material to be removed from the workpiece, cutting tool flute length or insert size, and the power available from the

machine spindle. Always use the largest depth of cut possible to ensure the least effect on the tool life.

Cutting speed, spindle speed, feedrate, and depth of cut are all important factors in the metal-cutting process. When properly calculated, the optimum metal-cutting conditions will result. *Refer to the Machinery's Handbook, tool and insert ordering catalogs, and online applications from the tool and insert manufacturers for more information on recommended depths of cut for particular tooling.*

PROCESS PLANNING FOR CNC

Certain steps must be followed in order to produce a machined part that meets specifications given in an engineering drawing or blueprint. These steps need to be organized in a logical sequence to produce the finished part in the most efficient manner. Before machining begins, it is essential to go through the procedure called process planning. The following are the steps in the process:

1. Study the engineering drawing or blueprint.
2. Select the proper raw material or rough stock as described in the engineering drawing or blueprint.
3. Study the engineering drawing or blueprint and determine the best sequence of individual operations needed to machine the required geometry.
4. Transfer the information onto planning charts.
5. While the part is still mounted on the machine, use in-process inspection to check dimensional values as they are completed.
6. Make necessary corrections and deburr.
7. Perform a 100% dimensional inspection when the part is finished and log the results of the first article inspection on the quality control check sheet.
8. Take corrective action if any problems are identified.
9. Begin production.

Planning Documents

An engineering drawing or blueprint may be thought of as a map that defines the destination. This destination is the end product. The roads available to get to this destination may be numerous. We do not start the trip without first determining what the destination is and how we are going to get there.

Planning sheets resemble the required path to the destination. They are written descriptions of how to get there (to the end product). The following are descriptions of sample planning documents.

The Engineering Drawing or Blueprint

The information given on the engineering drawing or blueprint will include the material, overall shape and the dimensions for part features (Figure 1-7). The geometry determines the type of machine (mill or lathe) to be used to produce the part. By studying the engineering drawing or blueprint, material and operations (drilling, milling, boring, etc.) can be identified. The tools and work holding method can also be determined.

Part 1 CNC Basics
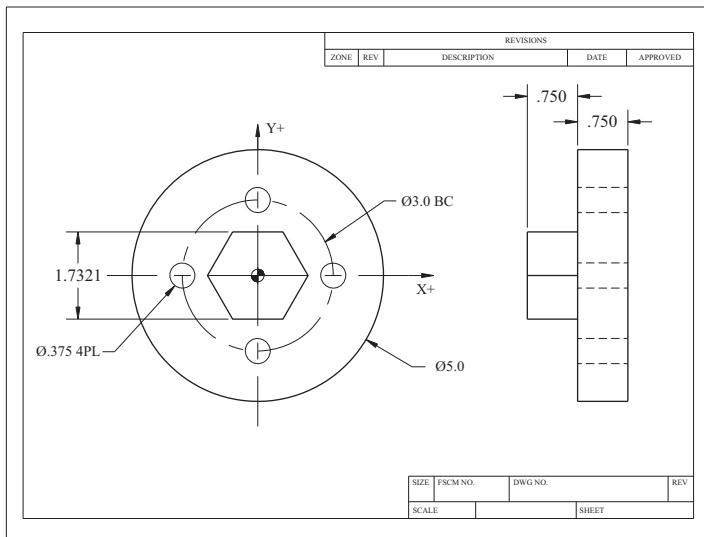
Figure 1-7
Machined Part
Engineering Drawing

Chart 1-1
Process Planning Operation Sheet

Date				Prepared By	
Part Name				**Part Number**	
Quantity				**Sheet ___ of ___**	
Material					
Raw Stock Size					
Operation Number	**Machine Used**	**Description of Operation**		**Time**	

Occasionally, the geometry will require multiple machines to manufacture the part, and thus additional operations will be necessary.

Operation Sheet

The purpose of this planning document is to identify the correct order for operations to be performed and the machine to be used. For example, suppose you are required

to produce the part shown in Figure 1-7. You would first saw cut the rough stock into blanks and then turn the part on a lathe to create the five-inch diameter and rough turn the diameter for the hexagon. Next, you would use a milling machine to cut the hexagon and drill the bolt-hole circle. Before any inspecting the part for accuracy, you would deburr the part.

The operation sheet is particularly useful when many identical parts are machined (production run). The operation sheet is similar to directions or a how-to approach. The process needed to manufacture the finished part has been decided in advance and is documented for future use.

When small batches of parts are to be made, there may not be an operation sheet. It is the machinist's responsibility to study the engineering drawing or blueprint and decide the necessary steps to machine the part. The operation sheet can aid in this decision making process (Chart 1-1).

With CNC machining, multiple part geometry features can be performed in one setup. In some cases, when using a CNC mill turn center, a part might be machined to its completed status without ever using another machine. This is very efficient and another advantage of the use of CNC equipment.

To complete an operation sheet, study the engineering drawing or blueprint; then decide on the steps necessary to machine the part. Document the machining process and refine any problems the process has. Then list the operations in the correct sequence in which they will be performed.

The top section of the operation sheet is for reference information and includes:

- The date the document is prepared or revised

- The name of the person preparing it

- The part name and the part number (from the engineering drawing or blueprint)

- The quantity of parts to be manufactured

Because some parts require a large number of operations, it is possible that more than one operation sheet will be needed to document the whole process. The top section also includes a sheet numbering system (Sheet _____ of _____). This information must be included. Other information included on the operation sheet header is the material, the raw stock size for the part, and the operations list.

CNC Setup Sheet

The CNC setup sheet is the document that tells the machinist what tools are to be used and any specific information related to tools. For instance, it may be necessary to have a certain amount of tool projection/extension for a drill to be able to completely machine through a part. This document is where the operator/CNC machinist finds this information. In Part 5 of this text, you will be introduced to CAD/CAM and how you can develop CNC setup sheets within the CAD/CAM programs. Many companies today are going to "paperless" factories, wherein these documents will be on the intranet in

electronic form. The CNC setup sheet has two sections (Chart 1-2). The top section is for reference information and includes:

- The date the document is prepared or revised

- The name of the person preparing it

- The part name and part number (from the engineering drawing or blueprint)

- The machine being used

  Note: If more than one machine is to be used to manufacture a single part, separate setup sheets are completed for each machine.

- The CNC part program used in the manufacturing process

- Workpiece zero reference points for the part (program zero)

- Work holding devices

  Note: If more than one device is needed, the operation number(s) and process are also included.

The lower half of this form lists the tool(s) by number, description, and offset. There is a column for comments, remarks, or explanations, if needed. Specific tool requirements, like minimum tool length projection/extension, can be entered in the comments section.

Chart 1-2
Process Planning CNC Setup Sheet

Date	Prepared By
Part Name	Part Number
Machine	Program Number

Workpiece Zero: X _____   Y _____   Z _____

Setup Description:

Tool Number	Offset Number	Tool Description	Comments

Part 1 CNC Basics

Quality Control Check Sheet

This planning document is used for the final inspection stage of the machining process. Once the part is completed, it is necessary to check all of the dimensions listed from the engineering drawing or blueprint to verify they are within the specified tolerance. The Quality Control Check Sheet is an excellent method to document the results of this inspection and a valuable tracking tool.

Reference information is similar to the other planning documents. Included are:

- The date the document is prepared or revised
- The name of the person checking the part
- The part name and part number (from the engineering drawing or blueprint).

On the check sheet, 100% of the engineering drawing or blueprint dimensions and their tolerances are written down in list form. Using this method, sequentially go through each of the dimensions and log the results. This assures that the machined part meets the specifications given on the engineering drawing or blueprint. As the part is checked and verified, some dimensions may not meet specifications. It is important to identify these incorrect values, emphasizing them for correction whether with red ink or a highlighter pen (or by changing font color or highlight if in electronic form). You could also include details in the comments section of the QC Check Sheet (Chart 1-3). If dimensions are found that do not meet specifications, corrective action must be taken.

Chart 1-3
Process Planning Quality Control Check Sheet

Date			Checked By	
Part Name			Part Number	
			Sheet ___ of ___	
Blueprint Dimension	**Tolerance**	**Actual Dimension**	**Comments**	

Part 1 CNC Basics

TYPES OF NUMERICALLY CONTROLLED MACHINES

There are two basic groups of numerically controlled machines: Numerical Control (NC) and Computer Numerical Control (CNC).

In an NC system, the program is run from a punched tape where it is impossible to store such a program in memory. For a punched tape to be used again to machine another part, it must be rewound and read from the beginning. This routine is repeated every time the program is executed. If there are errors in the program and changes are necessary, the tape will need to be discarded and a new one punched. The process is costly and error prone; although this type is still in use, it is becoming obsolete.

Machines with a CNC system are equipped with a computer, consisting of one or more microprocessors and memory storage facilities. Some CNC machines have hard drives and are network configurable. Program data is entered through Manual Data Input (MDI) at the control panel keyboard, via an RS232 communications interface port or via Ethernet from a remote source like a personal computer (PC) network or from a USB drive. The control panel enables the operator to make corrections (edits) to the program stored in memory, thereby eliminating the need for new punched tape.

Types of CNC machines have expanded vastly over the last decade. Turning and machining centers are the focus of this book, but there are many other types of machines using Computerized Numerical Control. For example, there are: multi-task mill turn centers, electrical discharge machines (EDM), grinders, lasers, turret punches, and many more. Also, there are many different designs of machining and turning centers. Some of the machining centers have rotary axes and some turning centers have live tooling and secondary spindles. For this text, the focus will be limited to vertical machining centers with three axes and turning centers with two axes. These types of machines are considered the foundation of all CNC learning. All operations on these machines can be carried out automatically. Human involvement is limited to setting up, loading and unloading the workpiece, and entering the amounts of dimensional offsets into registers on the control.

WHAT IS CNC PROGRAMMING?

CNC programming is a method of defining machine tool movements through the application of numbers and corresponding coded letter symbols. As shown in the list below, all phases of production are considered in programming, beginning with the engineering drawing or blueprint and ending with the final product:

- Engineering drawing or blueprint
- Work holding considerations
- Tool selection
- Preparation of the part program
- Part program tool path verification
- Measuring of tool and work offsets
- Program test by dry run
- Automatic operation or CNC machining

Part 1 CNC Basics

Begin all programming by closely evaluating the engineering drawing or blueprint; emphasizing assigned tolerances for particular operations, tool selection, and the choice of a machine. Next, select the machining process. The machining process refers to the selection of fixtures and determination of the operation sequence. Following that, select the appropriate tools and determine the sequence for their application. Before writing a program, calculate the spindle speeds and feed rates.

When program writing begins, give special attention to the specific tool movements necessary to complete the finished part geometry, including non-cutting movements. Identify individual tools and note them in the program manuscript. Also note miscellaneous functions for each tool such as: flood coolant, spindle direction, r/min and feedrates (these items will be covered in greater detail in the following chapters). Then, once the program is written, transfer it to the machine through an input medium like one of the following: punched tape, floppy disk, USB, RS-232 interface, or Ethernet.

Initiate the machining by preparing the machine for use, commonly called setup. For example, measure and input workpiece zero and tool length offsets into CNC memory registers. Many modern controllers have a function for graphical simulation of the programmed tool path on the cathode ray tube (CRT). This enables the machinist or set-up person to verify that the program has no errors, and to visually inspect the tool path movements. If all looks well, machine the first part with increased confidence. After completion, a thorough dimensional inspection will compare dimensions of the final product to those on the engineering drawing or blueprint. Correct any differences between the actual dimensions and the dimensions on the drawing by inserted values into the offset register of the machine. In this manner, you can obtain the correct dimensions of consecutively machined parts.

INTRODUCTION TO THE COORDINATE SYSTEM

All machines are equipped with the basic traveling components, which move in relation to one another as well as in perpendicular directions. CNC turning centers are equipped with a turret and tool carrier, which travels along two axes (Figures 1-8 and 1-9).

Note that in the following drawings of lathes, the cutting tool and turret is located on the positive side of spindle centerline. This is a common design of modern CNC turning centers. For visualization purposes, in this book the cutting tool will be shown upright. In reality, it is mounted with the insert facing down and the spindle is rotated clockwise for cutting.

Note: the direction of spindle rotation in turning—clockwise (CW) or counterclockwise (CCW)—is determined by looking from the headstock towards the tailstock and tool orientation.

Machining centers are milling machines equipped with a traversing worktable or column, which travels along two axes, and a spindle with a driven tool that travels along a third axis (Figure 1-10).

All axes of machines are oriented in an orthogonal coordinate system (each axis is perpendicular to the other), for example, the Cartesian coordinate system or right-hand rule system (Figure 1-11).

X+
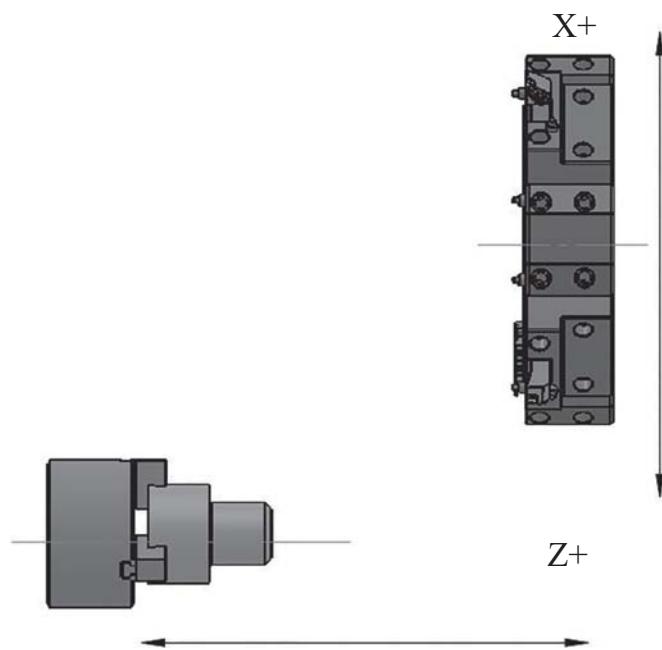
Figure 1-8
Turning Center Axes
Courtesy Kennametal

Z+
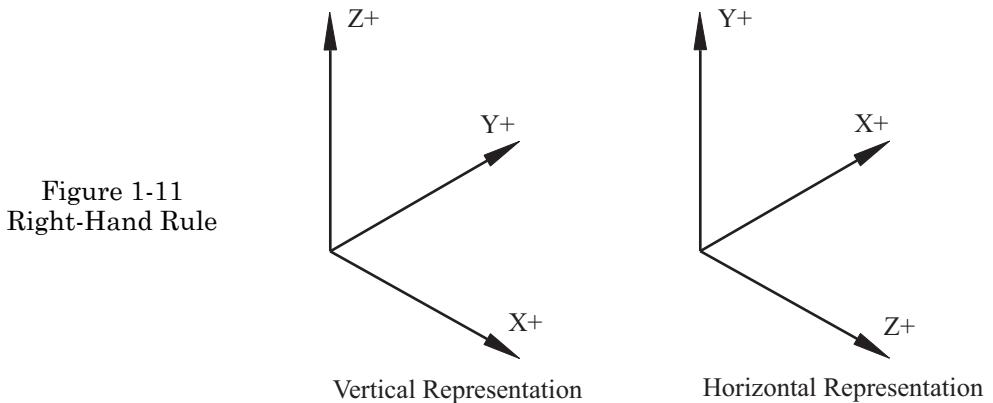

Figure 1-9
Two-Axis Turning Center
Courtesy MAZAK Corporation

Figure 1-10
Three-Axis Machining Center
Courtesy MAZAK Corporation

Figure 1-11
Right-Hand Rule

Z+

Y+

X+

Vertical Representation

Y+

X+

Z+

Horizontal Representation

The Right-Hand Rule System

In discussing the X, Y, and Z axes, the right-hand rule establishes the orientation and the description of tool motions along a positive or negative direction for each axis. This rule is recognized worldwide and is the standard for which axis identification was established.

Use Figure 1-11 to help you visualize this concept. For the vertical representation, the palm of your right hand is laid out flat in front, face up, the thumb will point in the positive X direction. The forefinger will be pointing the positive Y direction. Now fold over the little finger and the ring finger and allow the middle finger to point up. This forms the third axis, Z, and points in the Z positive direction. The point where all three of these axes intersect is called the origin or zero point. When looking at any vertical milling machine, you can apply this rule. For the horizontal mill, the same steps described above could be applied if you were lying on your back.

COORDINATE SYSTEMS

Visualize a grid on a sheet of graph paper with each segment of the grid having a specific value. Now place two solid lines through the exact center of the grid and perpendicular to each other. By doing this, you have constructed a simple, two-dimensional coordinate system. Carry the thought a little further and add a third imaginary line. This line passes through the same center point as the first two lines but is vertical; that is, it rises above and below the sheet on which the grid is placed. This additional line, which is called the Z-axis, represents the third axis in the three-dimensional coordinate system.

Two-Dimensional Coordinate System

A two-dimensional coordinate system, such as the one used on a lathe, uses the X and Z axes for measurement. The X-axis runs perpendicular to the workpiece and the Z-axis is parallel with the spindle centerline. When working on the lathe, we are working with a workpiece that has only two dimensions, the diameter and the length. On engineering drawings or blueprints, the front view generally shows the features that define the finished shape of the part for turning. In order to see how to apply this type of coordinate system, study Figures 1-12, 1-13, and 1-14.
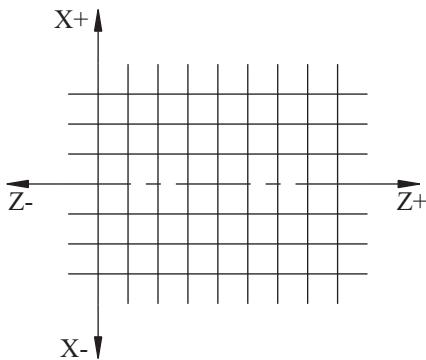
Figure 1-12
Two-Dimensional Coordinate
System
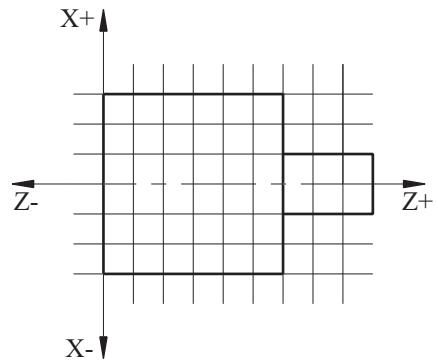
Figure 1-13
Part Drawing Overlaid on 2D
Coordinate System
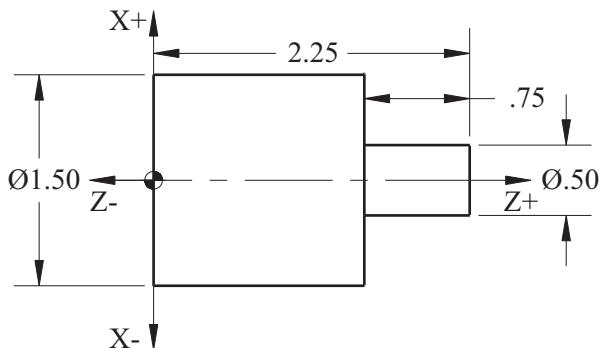
Figure 1-14
Two-Dimensional Turned
Part Drawing

Think of the cylindrical work piece as if it were flat or as shown in the top view of the part blueprint. Next, visualize the coordinate system superimposed over the engineering drawing or blueprint of the workpiece, aligning the X-axis with the centerline of the diameter shown. Then align the Z-axis with the end of the part, which will be used as an origin or zero-point. In most cases, the finished part surface nearest the spindle face will represent this Z-axis datum and the centerline will represent the X-axis. Where the two axes intersect is the origin or zero point. By laying out this "grid," we now can apply the coordinate system and define where the points are located to enable programmed creation of the geometry from the blueprint. Another point to consider on a lathe is that the cutting takes place on only one side of the part or the radius because the part rotates and is symmetrical about the centerline. In order to apply the coordinate system in this case, all we need is the basic contour features of one-half of the part (on one side of the diameter); the other half is a mirror image. When given this program coordinate information, the lathe will automatically produce the mirror image.

Three-Dimensional Coordinate System

Although the mill uses a three-dimensional coordinate system, the same concept (using the top view of the engineering drawing or blueprint) can be used with rectangular workpieces. As with the lathe, the Z-axis is related to the spindle. However, in the case of the three-dimensional rectangular workpiece, the origin or zero-point must be
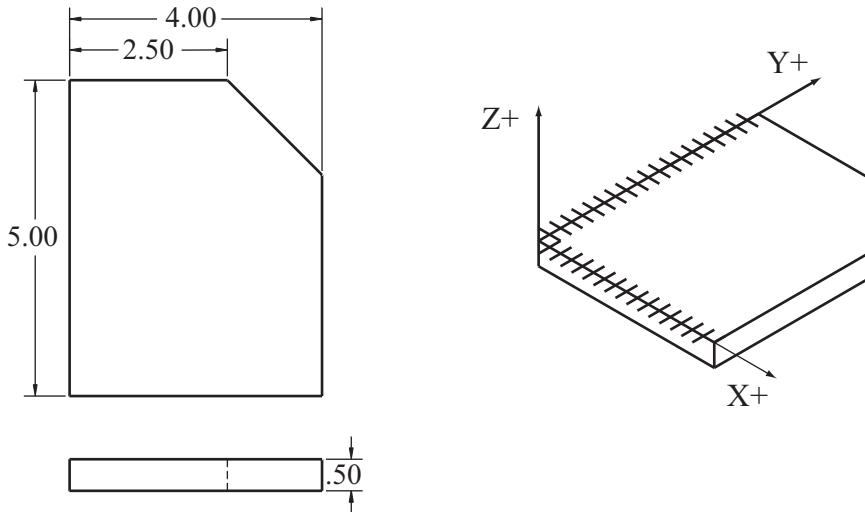
Figure 1-15
Three-Dimensional Coordinate System

defined differently. In the example shown in Figure 1-15, the lower left-hand corner of the workpiece is chosen as the zero-point for defining movements using the coordinate system. The thickness of the part is the third dimension or Z-axis. When selecting a zero-point for the Z-axis of a particular part, it is common to use the top surface.

The Polar Coordinate System

If a circle is drawn on a piece of graph paper so that the center of the circle is at the intersection of two lines and the edges of the circle are tangent to any line on the paper. This will help in visualizing the following statements. Let's consider the circle center as the origin or zero-point of the coordinate system. This means that some of the points defined within this grid will be negative numbers. Now draw a horizontal line through the center and passing through each side of the circle. Then draw a vertical line through the center also passing through each side of the circle. Basically, we've made a pie with four pieces. Each of the four pieces or segments of the circle is known as a quadrant. The quadrants are numbered and progress counter-clockwise. In Quadrant No. 1, both the X- and Y-axis point values are positive. In Quadrant No. 2, the X-axis point values are negative and the Y-axis point values are positive. In Quadrant No. 3, both the X- and Y-axis point values are negative. Finally, in Quadrant No. 4, the X-axis point values are positive while the Y-axis values are negative. This quadrant system is applied in all cases, regardless of the axis of rotation. The drawings in Figure 1-16 illustrate the values (negative or positive) of the coordinates, depending on the quarter circle (quadrant) in which they appear.

Although the rectangular coordinate system can be used to define points on the circle, a method using angular values may also be specified. We still use the same origin or zero-point for the X- and Y-axes. However, the two values that are being considered are an angular value for the position of a point on the circle and the length of the radius joining that point with the center of the circle. To understand the polar coordinate system, imagine that the radius is a line circling around the center origin or zero-point. Thinking in terms of hand movements on a clock, the three-o'clock position
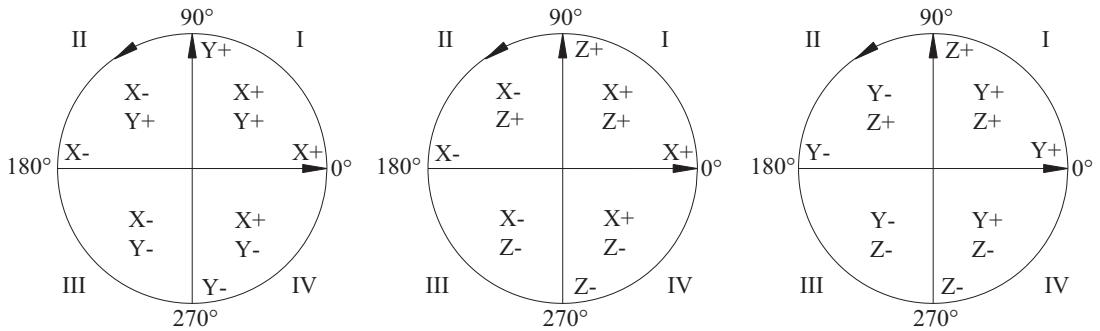
Figure 1-16
Polar Coordinate System Quadrants

has an angular value of 0° counted as the "starting point" for the radius line. The twelve-o'clock position is referred to as the 90° position, nine-o'clock is 180°, and the six-o'clock position is 270°. When the radius line lies on the X-axis in the three-o'clock position, we have at least two possible angular measurements. If the radius line has not moved from its starting point, the angular measurement is known as 0°. On the other hand, if the radius line has circled once around the zero point, the angular measurement is known as 360°. Therefore, the movement of the radius determines the angular measurement. If the direction in which the radius rotates is counter-clockwise, angular values will be positive. A negative angular value (such as –90°) indicates that the radius has rotated in a clockwise direction. Note: A 90° angle (clockwise rotation) places the radius at the same position on the grid as a +270° (counter-clockwise) rotation.

Sometimes the engineering drawing or blueprint will not specify a rectangular coordinate, but will give a polar system in the form of an angle for the location of a feature. With some basic trigonometric calculations, this information can be converted to the rectangular coordinate system.

The same polar coordinates system applies regardless of the axis of rotation, as is shown once again in Figure 1-16. When rotation is around the X-axis, the rotational axis is designated as A; the Y-axis, the rotational axis is designated as B; and the Z-axis, the rotational axis is designated as C. These are considered additional axes and are known as the fourth axis.

All operations of CNC machines are based on three axes: X, Y, and Z.

1. *(X0, Y0, Z0)*
2. *(X0, Y0, Z+)*
3. *(X0, Y–, Z+)*
4. *(X0, Y–, Z0)*
5. *(X–, Y–, Z0)*
6. *(X–, Y0, Z0)*
7. *(X–, Y0, Z+)*
8. *(X–, Y–, Z+)*
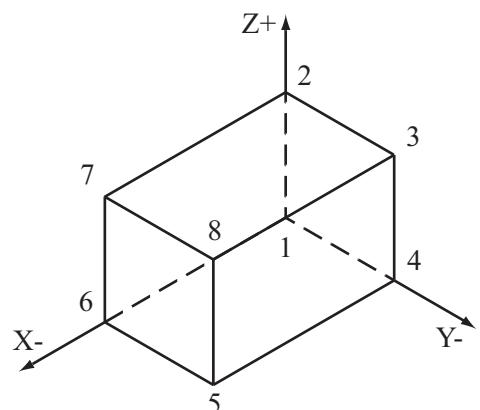
Figure 1-17
Three-Axis Part Example

Part 1 CNC Basics

Figure 1-17 illustrates a box-like object in which one vertex (point 1) is located at the origin of the coordinate system. At the side of the drawing, the coordinate signs are given for each of the numbered locations. Note the position of the coordinate system on the following machines.

On vertical milling machines, the spindle axis is perpendicular to the surface of the worktable (Figure 1-18).
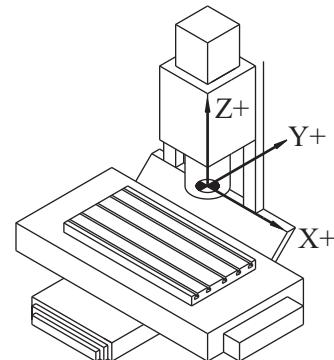
Figure 1-18
Axis Designation for a Three-Axis Mill

On horizontal milling machines, the spindle axis is parallel to the surface of the worktable (1-19).
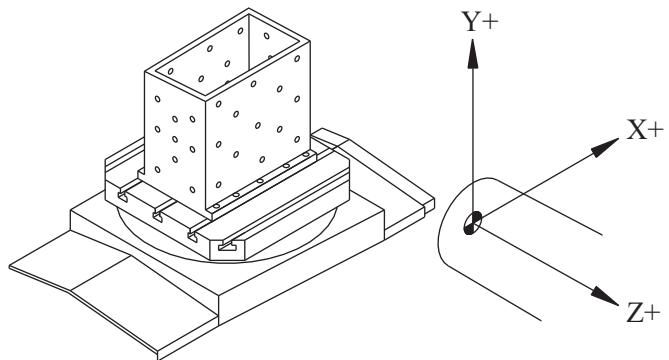
Figure 1-19
Axis Designation for a
Three-Axis Horizontal Mill

On turning centers, the spindle axis is also the workpiece axis (1-20).
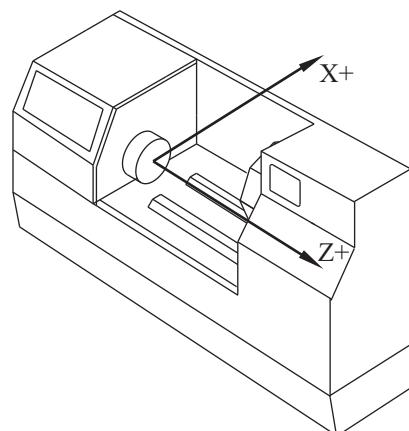
Figure 1-20
Axis Designation for a Two-Axis
Turning Center

POINTS OF REFERENCE

When using CNC machines, any tool location is controlled within the coordinate system. The accuracy of this positional information is established by specific zero points (reference points). The first is Machine Zero, a fixed point established by the manufacturer that is the basis for all coordinate system measurements. On a typical lathe, this is usually the spindle centerline in the X-axis and the face of the spindle nose for the Z-axis. For a milling machine, this position is often at the furthest end of travel in all three axes in the positive direction.

Occasionally, this X-axis position is at the center of the table travel.

This Machine Zero Point establishes the coordinate system for operation of the machine and is commonly called Machine Home (Home position). Upon startup of the machine, all axes need to be moved to this position to establish the coordinate system origin (commonly called homing the machine or Zero Return). The Machine Zero Point identifies to the machine controller where the origin for each axis is located. Some machines today are equipped with absolute encoders so that homing is no longer necessary at machine startup.

The operator's manual supplied with the machine should be consulted to identify where this location is and how to properly home the machine.

The second zero point can be located anywhere within the machine work envelope and is called Workpiece Zero; it is used as the basis for programmed coordinate values used to produce the workpiece. It is established within the part program by a special code and the coordinates are taken from the distance from the Machine Zero point. The code number in the program identifies the location of offset values to the machine control where the exact coordinate distance of the X, Y, and Z axes of Workpiece Zero is in relationship to the Machine Zero. All dimensional data on the part will be established by accurately setting the Workpiece Zero. A way of looking at the Workpiece Zero is like another coordinate system within the machine coordinate system, established by the Home position.

Tool offsets are also considered to be Zero Points as well and are compensated for with tool length and diameter offsets. The tool-setting point for a lathe has two dimensions: the distance on diameter from the tool tip to the centerline of the tool turret, and the distance from the tool turret face to the tool tip. The tool-setting point for the mill is the distance from the spindle face to the tool tip, and the distance from the tool tip to the spindle centerline.

Engineering Drawing or Blueprint Relationship to CNC

The standard called ASME Y14.5-2009 establishes a method for communicating part dimensional values, in a uniform way, on the engineering drawing or blueprint. The drawing information will be translated to the coordinate system in order for dimensional values and part features to be manufactured.

On the engineering drawing or blueprint, datum features are identified as Primary (A), Secondary (B) and Tertiary (C). Dimensions for the workpiece are derived from these datum features. On the drawing, the point where these three datum features meet is called the origin or zero point for the part. When possible, this same point should be used for Workpiece Zero. This allows the use of actual engineering drawing or blueprint dimensions

within the part program and often results in fewer calculations. Most drawings are developed using an absolute dimensioning system based on datum dimensions derived from the same fixed point (origin or zero point). Occasionally, some features may be dimensioned from the location of another feature. An example of this is a row of holes exactly one half of an inch apart. This type of dimensioning is called relative or incremental.

Note: A thorough knowledge of engineering drawing or blueprint reading is imperative for successful results using manual or CNC equipment.

Machine Zero

Each CNC machine is assigned a fixed point, which is referred to as Machine Zero (or Machine Home). For most machines, Machine Zero is defined as the extreme travel end position of main machine components that are oriented in a given coordinate system. From Machine Zero, we can determine the values of the coordinates that, in turn, determine the position of the points commanded in a CNC program. Electromechanical sensors called micro-switches (limit switches) are located in the extreme end positions of traveling machine components. These sensors send a signal to the controller when they are activated and thus setting the Home position. In the case of milling machines, Machine Zero on the table is set with respect to the X- and Y-axes. Machine Zero on the spindle is set with respect to the Z-axis, whereas Machine Zero of the tool carrier on lathes is set with respect to the X- and Z-axes. Positioning the traveling components at zero can be performed manually, as well as with the use of the control panel or directly from within the program by employing a Reference Point Return function. At the initial startup of any CNC machine, it is required that the machine be "Homed" or sent to Machine Zero before proceeding any further. From that point on, all machine components will always automatically return to the same exact position when commanded to do so in the program.

Machine Zero is frequently the position in which tool changes take place. Therefore, if you intend to change the tool before a given operation, then the machine must be positioned at Machine Zero for the Z-axis on vertical machines and the Y-axis on horizontal machines.

Workpiece Zero

So far, for all main traveling components of CNC machines, we have assigned an oriented axis within the coordinate system. Any movement of machine components must be described by points, which actually determine the traveling path of the tool. Changes in the position tool are determined with respect to the stationary reference point of Machine Zero.

In order to better understand this concept, this situation can be illustrated with a rectangular plate in which all coordinates are described at their four corners (P_1, P_2, P_3, P_4) (Figure 1-21).

$$P_1 = \text{X–15.0, Y–10.0}$$

$$P_2 = \text{X–15.0, Y–12.0}$$

$$P_3 = \text{X–20.0, Y–12.0}$$

$$P_4 = \text{X–20.0, Y–10.0}$$
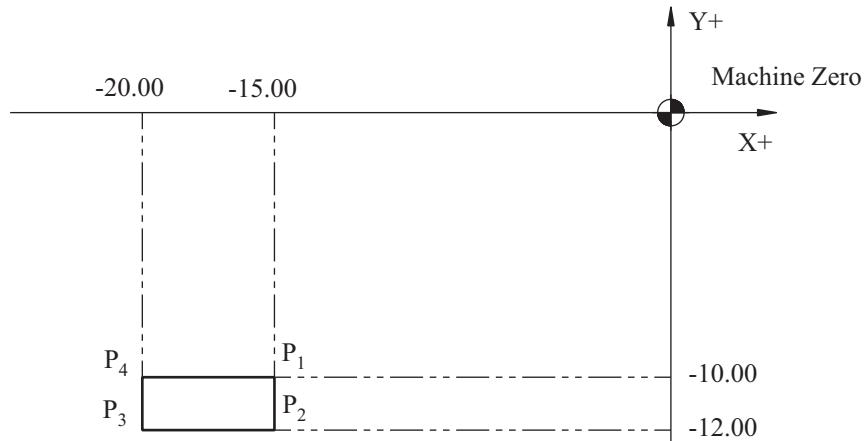
Figure 1-21
Machine Zero to Workpiece Zero

Determine the coordinates of these points. The rectangle has been placed in such a manner that each side is parallel to one axis of the coordinate system. If the distance from Machine Zero is measured to any point on the workpiece, the coordinates of the remaining points can be determined from the dimensions given on the drawing.

All programmed point coordinates (whose values are determined with respect to Machine Zero) must be calculated with respect to Machine Zero every time, which is time consuming. It may also cause errors due to the fact that all the given dimensions determining the points do not always refer to those on the drawing. As previously mentioned, in order to determine the coordinates for the four corners of the rectangular part illustrated, it is necessary to find the distance between Machine Zero and a specific point of reference on the part. Then, all the remaining dimensional data to be used are taken from the engineering drawing or blueprint.

For all CNC machines, we follow certain principles to define the method of selecting Workpiece Zero from within the part program. At the beginning of the program, we input the value of the distance between Machine Zero and the selected Workpiece Zero by employing function G92 or G54 through G59 for machining centers and function G50 or G54 for turning centers. These measured values are input either directly into the program, as in the case of G92 for mills and G50 for lathes, or in offset registers in the control for G54 through 59. Let us review the same situation as above and note the changes of the point coordinates when applying Workpiece Zero (Figure 1-22).

G92 X15.0 Y10.0   or   G54 X–15.0 Y–10.0

P_1 = X0, Y0

P_2 = X0, Y–2.0

P_3 = X–5.0, Y–2.0

P_4 = X–5.0, Y0

27
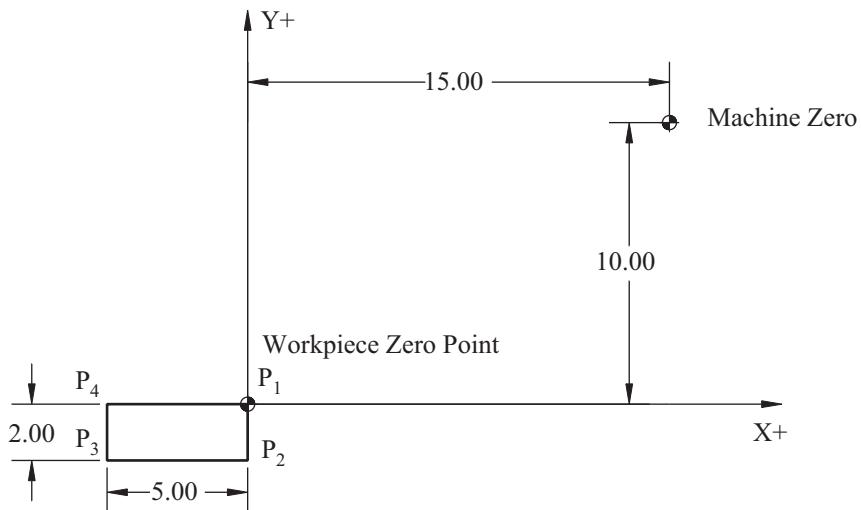
Figure 1-22
Workpiece Zero Point

The values X15.0 and Y10.0 for G92 or X–15.0 Y–10.0 for G54 through 59 are valid until they are recalled by the same function, but with different coordinates for X and Y. When programming machining centers, we place function G92 or G54 through 59 only at the beginning of the program, whereas the values assigned to function G50 for turning centers will need to be added to the program with respect to each tools position. Once this activation is read by the control, all coordinates will be measured from the new Workpiece Zero, allowing the use of part dimensions for programmed moves.

With turning centers, Workpiece Zero in the direction of the Z-axis is most often on the face surface of the workpiece, and the centerline axis of the spindle is Workpiece Zero in the direction of the X-axis (Figure 1-23).

On machining centers, Workpiece Zero is frequently located on the corner of the workpiece or in alignment with the datum features of the workpiece.

The application of Workpiece Zero is quite advantageous to the programmer because the input values of X, Y, and Z in the program can be taken directly from the drawing. If the program is used another time, the values of coordinates X and Y (assigned to functions G50 and G92 or G54 through G59) will have to be inserted again, prior to automatic operation.

Absolute and Incremental Coordinate Systems

When programming in an absolute coordinate system, the positions of all the coordinates are based upon a fixed point or origin of the coordinate system. The tool path from point P_1 to P_{10}, for example, is illustrated in Figure 1-24.
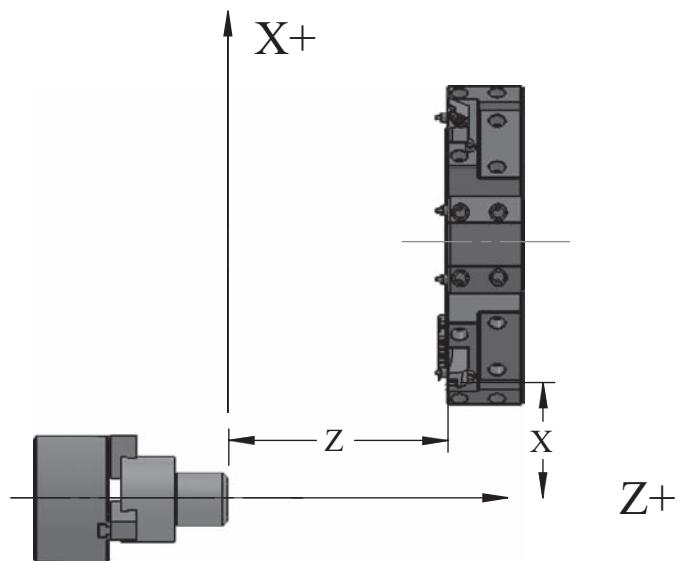
Figure 1-23
Workpiece Zero for Turning Centers
Courtesy Kennametal
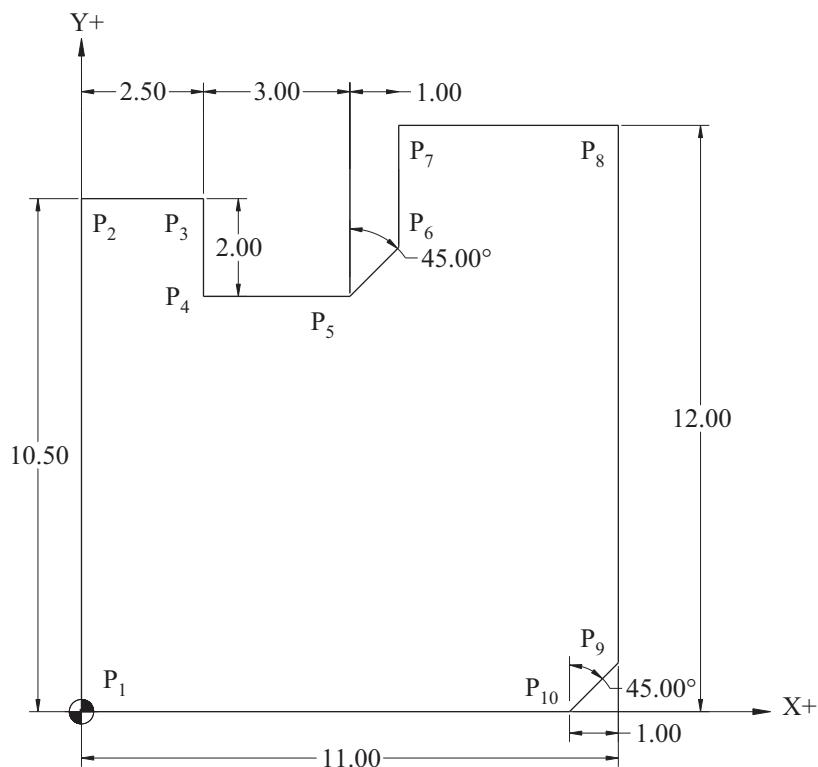
Figure 1-24
Absolute and Incremental Coordinate System Points

	X	Y
P_1	0.0	0.0
P_2	0.0	10.5
P_3	2.5	10.5
P_4	2.5	8.5
P_5	5.5	8.5
P_6	6.5	9.5
P_7	6.5	12.0
P_8	11.0	12.0
P_9	11.0	1.0
P_{10}	10.0	0.0

Programming with an incremental coordinate system is based upon the determination of the tool path from its current position to its next consecutive position and in the direction of all the axes. Sign determines the direction of motion. Based on the drawing from the previous example, we can illustrate the tool path in an incremental coordinate system, starting and ending at P_1.

	X	Y
P_2	0.0	10.5
P_3	2.5	0.0
P_4	0.0	−2.0
P_5	3.0	0.0
P_6	1.0	1.0
P_7	0.0	2.5
P_8	4.5	0.0
P_9	0.0	−11.0
P_{10}	−1.0	−1.0
P_1	−10.0	0.0

Coordinate Input Format

CNC machines allow input values of inches (specified by the command G20), millimeters (specified by the command G21), and degrees using a decimal point with significant zeros in front of (leading) or at the end (trailing) of the values. When using inch programming, the two ways distances can be specified:

Programming with a decimal point:

1 inch = 1. or 1.0

1 1/4 inch = 1.250 or 1.25

1/16 inch = 0.0625 or .0625

Programming with significant trailing zeros:

In this case, the zero furthest to the right corresponds with the ten thousandths of an inch.

$$1 \text{ inch} = 10000$$

$$1 \text{ 1/8 inch} = 11250$$

$$1 \text{ 1/32 inch} = 10313$$

These two coordinate input formats (G20 and G21) are the standard on all CNC machines.

With modern controllers, neither leading nor trailing zeros are required—the decimal placement is the significant factor. In this case, the input is as follows:

$$1 \text{ inch} = 1. \text{ or } 1.0$$

$$1 \text{ 1/4 inch} = 1.25$$

$$1/16 \text{ inch} = .0625 \text{ or } 0.0625$$

PROGRAM FORMAT

The language described in this book is used for controlling machine tools and is known informally as "G-Code". This language is used worldwide and is reasonably consistent. The standard by which it is governed was established by the Electronics Industries Association and the International Standards Organization, called EIA/ISO for short. Because of this standardization, a program created for a particular part on one machine may be used on other similar machines with minimal changes required.

Each program is a set of instructions that controls the tool path. The program is made up from blocks of information separated by the semicolon symbol (;). This symbol (;) is defined as the end of the block (EOB) character. Each block contains one or more program words. For example:

Word	Word	Word	Word	Word
N02	G01	X3.5	Y4.728	F8.0

Each word contains an address, followed by specific data. For example:

Address	Data	Address	Data	Address	Data
N	02	G	01	X	3.5

Chart 1-4 is a list for all of the letter addresses that are applicable in programming, along with brief explanations for each. Chart 1-5 then lists symbols commonly used in programs.

Part 1 CNC Basics

ADDRESS CHARACTERS	
CHARACTER	**MEANING**
A	Additional rotary axis parallel and around the X-axis
B	Additional rotary axis parallel and around the Y-axis
C	Additional rotary axis parallel and around the Z-axis
D	Tool radius offset number; (Turning) Depth of cut for multiple repetitive cycles
E	User Macro Character
F	Feed rate; (Turning) Precise designation of thread lead
G	Preparatory Function
H	Tool Length Offset number
I	Incremental X coordinate of circle center; (Turning) parameter of fixed cycle
J	Incremental Y coordinate of circle center
K	Incremental Z coordinate of circle center; (Turning) parameter of fixed cycle
L	Number of repetitions (subprogram, hole pattern); Fixed offset group number
M	Miscellaneous Function
N	Sequence or block number
O	Program number
P	Dwell time, program number, and sequence number designation in subprogram; (Turning) Sequence number start for multiple repetitive cycles
Q	Depth of cut, shift of canned cycles; (Turning) Sequence number end for multiple repetitive cycles
R	Point R for canned cycles, as a reference return value; Radius designation of a circle arc; Angular displacement value for coordinate system rotation
S	Spindle-Speed Function
T	Tool Function
U	Additional linear axis parallel to X-axis
V	Additional linear axis parallel to Y-axis
W	Additional linear axis parallel to Z-axis
X	X coordinate
Y	Y coordinate
Z	Z coordinate

Part 1 CNC Basics

COMMON SYMBOLS USED IN PROGRAMS	
SYMBOL	**MEANING**
–	Minus Sign, Used for Negative Values
/	Slash, Used for Block Skip Function
%	Percent Sign, Necessary at program beginning and end for communications only
( )	Parentheses, Used for comments within programs
:	Colon, Designation of Program Number
;	Semicolon, End-Of-Block character
.	Decimal Point, Designation of fractional portion of a number

Part 1 Study Questions

1. Programming is a method of defining tool movements through the application of numbers and corresponding coded letter symbols.

     T or F

2. A lathe has the following axes:

     a. X, Y, and Z

     b. X and Y only

     c. X and Z only

     d. Y and Z only

3. Program coordinates that are based on a fixed origin are called:

     a. Incremental

     b. Absolute

     c. Relative

     d. Polar

4. On a two-axis turning center, the diameter controlling axis is:

     a. B

     b. A

     c. X

     d. Z

5. The letter addresses used to identify axes of rotation are:

     a. U, V, and W

     b. X, Y, and Z

     c. A, Z, and X

     d. A, B, and C

Part 1 CNC Basics

6. The acronym TLO stands for:

      a. Tool Length Offsets

      b. Total Length Offset

      c. Taper Length Offset

      d. Time Length Offset

7. When referring to the polar coordinate system, the clockwise rotation direction has a positive value.

      T or F

8. In Figure 1-17, in which quadrant is the part placed?

9. A program block is a single line of code followed by an end-of-block character.

      T or F

10. Each block contains one or more program words.

      T or F

11. Using Figure 1-15, list the X and Y absolute coordinates for the part profile where Workpiece Zero is at the lower left corner. (The corner cutoff is at a 45° angle.)

12. Using Figure 1-15, list the X and Y incremental coordinates for the part profile where workpiece zero is at the lower left corner.

13. How often should the machine lubrication levels be checked?

PART 2

CNC MACHINE OPERATION

Part 2 CNC Machine Operation

Figure 2-1
FANUC 30i Machine Tool Controller
Courtesy FANUC FA America

Part 2 CNC Machine Operation

OBJECTIVES:

1. Become familiar with common CNC machine operator panel functions.

2. Become familiar with common machine control panel functions.

3. Learn common operations performed at the machine control.

4. Learn how to use the controls to input setup data including tool and work offsets.

5. Learn how to use the control to edit programs.

6. Examine some common cases of problem situations and learn how to solve them.

Every CNC Machine Tool has an operation panel and a control panel that provide the interface for the operators and programmers with the machine tool, sometimes referred to as the Human Interface (HMI). By using the operator panel, we physically manipulate the working components of the machine to do what we need; the control panel is where the program data are entered and stored. A thorough understanding of each is necessary for successful CNC machine use. First we will study an example of an operation panel.

OPERATOR PANEL FEATURES

The following descriptions for the model shown in Figure 2-2 represent the configuration for a common operator panel. Some differences exist for each manufacturer's operator panel, but they generally contain the same features. Figure 2-2 shows a panel for a three-axis mill. The panel used for a lathe would be essentially identical except for the axes keys for X and Z only. You should consult the applicable manufacturer manual for detailed descriptions that match your needs. Please note that the Handle (Manual Pulse Generator) is not shown, although it is described in the text. Another common item not shown here are those lathe-specific switches used to change the chucking direction from external to internal. The following sections describe the buttons in order from top-left to right for the three columns of buttons in Figure 2-2.
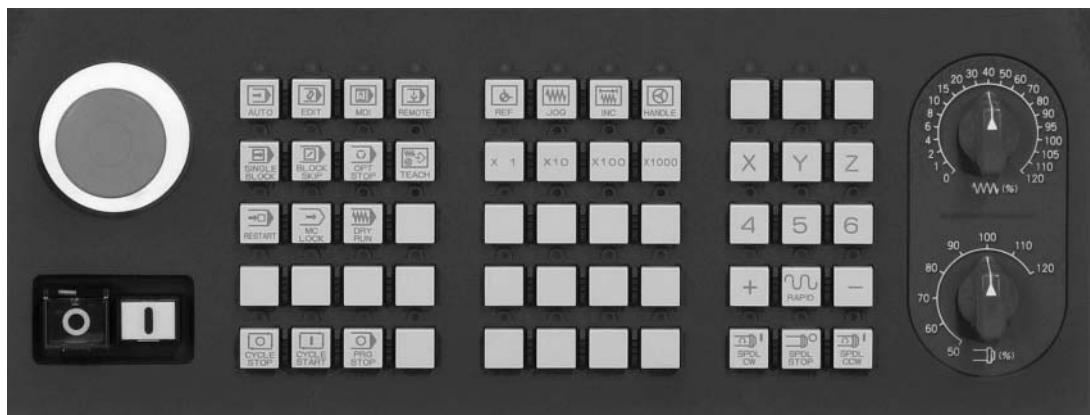

Figure 2-2
Common Operator Panel
Courtesy FANUC FA America

FEEDRATE OVERRIDE

The Feedrate Override dial allows the operator to control the feedrate by adjusting the position of the dial (Figure 2-3). During auto-cycle, when the dial is at the 100 position, the feed will occur at 100% of the programmed value. The control of the work feed is defined by the F-word in the program. You can increase the percentage of the value entered in the program to 120% or decrease it to 0%. This feature provides you the control needed to fine tune feeds. You can also use this feature to control feedrates while using the manual jog mode function. The same is true for the Spindle Override feature, which enables you to override the programmed control spindle speed between 50 and 120%.

EMERGENCY STOP

The EMERGENCY STOP is the large, red, mushroom-shaped button used to stop machine function when an emergency situation occurs. Examples of such situations are

- overloading of the machine
- the machined part has come loose
- incorrect data in the program or work/tool offsets have caused a collision (crash) between the tool and the workpiece

When this button is pressed, all program-commanded feed rates and spindle revolutions are halted immediately. To recover from an "E-Stop" condition, you must reset the program controller and Home machine axes. To reset the EMERGENCY STOP button, turn it clockwise. It should "pop out" of the depressed condition. Check the monitor for any alarm signals and take note of the Alarm # and description; then eliminate the cause that forced the use of the EMERGENCY STOP button. Press the Reset button to clear all pending commands and Home the machine axes when no interference conditions are present.

PROGRAM PROTECT

When the Program Protect key switch is in the ON condition (vertical), it prohibits any program changes to be made (Figure 2-5). The condition does not affect work or tool offset adjustments. Some shops set
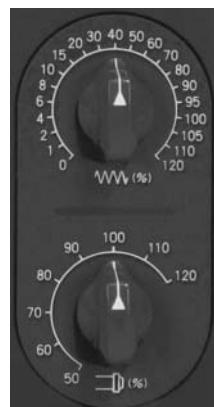
Figure 2-3
Feed and Spindle Override
Courtesy FANUC FA America
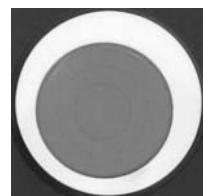
Figure 2-4
EMERGENCY STOP
Courtesy FANUC FA America

Figure 2-5
Program Protect Key Switch
Courtesy FANUC FA America

this condition to ON, remove this key, and allow only the programmer or set-up person access to the key. This is especially true in larger shops with multiple shifts and many workers. Some quality programs like AS9100 require that CNC program integrity be insured by locking out access to editing.

OPERATION PANEL KEY DESCRIPTIONS

This section introduces the function of each individual key/button (Figure 2-6).

PROGRAM SOURCE

On some operation panels, a rotary switch referred to as Mode Select is used instead of the buttons shown in Figure 2-6. This switch includes both automatic (AUTO) and manual operation functions. The position of this switch determines whether the machine utilizes the automatic or the manual control. This switch can also be positioned to allow the entry of data into the control manually (Manual Data Input or MDI) or to make changes to the program through the EDIT mode. For this example, operator panel buttons are used to specify the control or operational mode.

Note: When the buttons are pressed, they are active and remain so until another mode button is pressed that overrides it. In some conditions, multiple Light Emitting Diodes (LEDs) may be lit simultaneously. The LED above the button is lit when the mode is ON and active.

Auto

Pressing this button enables the CNC programs stored in the memory to be executed for automatic operation. When the Cycle Start button is pressed and this mode is active, automatic operation will occur.

Edit

Pressing this button selects the program edit mode. The EDIT mode enables you to enter the part program to control memory via keypad and soft-keys, enter any changes to the program, transfer program data via one of several communication interfaces (RS232, USB, Memory Card, or Ethernet) to or from an offline storage device, or check the program file memory, file locations, and storage capacity.

MDI-Manual Data Input

Pressing the MDI button selects the MANUAL DATA INPUT mode. The MDI mode enables the automatic control of the machine, using information entered in the
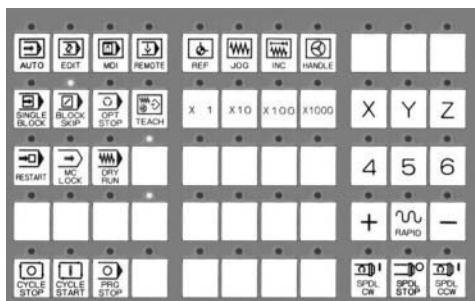

Figure 2-6
Operator Panel
Courtesy FANUC FA America

form of program blocks without interfering with the basic part program. This mode is often used when machining workpiece-holding equipment such as soft-jaws and during setup. It corresponds to single moves (milling surfaces, drilling holes), descriptions of which need not be entered to memory storage.

MDI mode can also be used during the execution of the program. Suppose the program is missing the command M03 S350 needed to turn on the spindle clockwise at 350 r/min, and the End-Of-Block character (;).To correct this omission, press the SINGLE BLOCK button and then the MDI button. Using MDI, you can enter functions M03 and S350 from the control panel keypad followed by the EOB character. Then, enter this command by pressing INPUT on the control panel. Press AUTO to reenter the program auto-cycle mode and then press CYCLE START to continue executing the program from memory. Changes of this sort (program edits) should be saved to overwrite the original program.

Remote

The Remote button is used for DNC (Direct Numerical Control) program operations when the program is too large for the controller memory and is executed (drip-fed) directly from a remote personal computer (PC) via Ethernet or memory card. USB cannot be used for DNC operations.

OPERATION SELECT

The following buttons are related to the automatic operation of the machine. Activating one of these buttons has an effect on the operation.

Single Block

The execution of a SINGLE BLOCK (SINGL BLOCK) of information is initiated by pressing this button to turn it ON. Each time the CYCLE START button is pressed, only one block of information will be executed. This switch can also be used if you intend to check the initial performance of a new program on the machine or when the momentary interruption of a machine's work is necessary.

Block Skip

When this button is pressed and is active simultaneously with the auto-cycle mode, the controller skips execution of the program blocks that are preceded by the slash (/) symbol and that end with the end of block (;) character. For instance, if a section of the part program or a particular block of the program is not presently needed—but you would like to keep this information for future use—then place the block skip symbol (/) at the beginning of each such block. The BLOCK SKIP button is located on the control panel. If it is activated, then information contained in the blocks that are preceded by the symbol (/) will not be executed.

Example:

    N100 G01 X2.810 Y3.256

    /N105 X3.253 Y2.864

    /N110 X3.800

(Blocks N105 and N110 will be skipped.)

Part 2 CNC Machine Operation

Notes: The symbol (/) should be placed at the beginning of the block. If it is not, then all the information contained in the block preceding the symbol (/) will be executed, while the information following this symbol will be omitted.

If the BLOCK SKIP is in the OFF condition, all blocks [regardless of the symbol (/)] will be executed.

When transferring the program to punched tape or external computer, all program information [regardless of symbol (/)] is transferred.

Opt Stop

When this button is pressed, the OPTIONAL STOP mode is active. The OPTIONAL STOP function interrupts the automatic cycle of the machine if the program word M01 appears in the program. Quite often, function M01 is placed in the program after the work of a particular tool is completed or before a tool change. This enables you to perform chip removal or a routine measurement directly on the machine and, if necessary, make adjustments, and then rerun the same tool to correct inaccuracies.

Teach

If the TEACH button is available, you can use it to create/edit a program, positioning the axes manually by jogging (Teach-in Jog) or handle (Teach-in Handle) mode, and inputting the other required codes while in the MDI or Edit mode. When the Teach mode is used, the movements of the axes are recorded while in either Jog or Handle mode. Machine positions along the X-, Y-, and Z-axes obtained by this manual operation are stored in memory as a program position; they are used to create a program. These movements can then be executed just as with any program. Consult the manufacturer's operator manual for detailed descriptions on the TEACH button's proper use. Not all controls have this feature.

Restart

The RESTART button allows you to restart automatic operation at a specific place in the program by entering a desired line sequence number. This sometimes becomes necessary after tool breakage or some type of collision. When the proper data are entered, the control processes through the program until the block entered is reached; it then activates any required modal G, F, D, and H commands. Any needed auxiliary codes (M, S, T, and B) must be entered during the Restart process.

MC Lock

This function is known as MACHINE LOCK. Activating this mode inhibits axis movement on all of the axes. This button is used to check a new program on the machine through the controller. All movements of the tool are locked, while a program check is run on the computer and displayed on the screen. You can observe the position display on the screen; if any program errors are encountered, an alarm will be displayed. This function is especially useful for checking very large programs requiring a long cycle time to complete. This test is normally the first in a series (Program Test, Dry Run, and Single Block) of preliminary actions to be executed before full auto cycle mode is attempted. For any program test, all offsets should be set first. During MACHINE LOCK, all axes are locked. It is possible to unlock selected axes during this mode in order to inhibit axis

movement in only one axis. This function is especially useful when inhibiting the Z-axis so that all X, Y movements can still be observed.

Dry Run

When you press the DRY RUN button during automatic cycle, all of the rapid and work feeds are changed to a feed set in the parameters instead of the programmed feed (usually rapid traverse feed rate). In this mode, you can control the feed rate by using the Feed Override switch. Consult the manufacturer manual for specific directions on the use of this function.

DRY RUN is also used to check a new program on the machine without any work actually being performed by the tool. This is particularly useful on programs with long cycle times so you can progress through the program more quickly.

Use caution when using this function. It is NOT intended for metal cutting.

EXECUTION

These three buttons are related to automatic operation of the machine. The first button temporarily stops the operation, the second starts automatic operation, and the last key merely indicates when a program stop is encountered. Their specific functions are described below:

Cycle Stop (Feed Hold)

Pressing the CYCLE STOP button during automatic operation will halt all feed movements of the machine. It will not stop the spindle r/min or affect the execution of tool changes on some machines. When the CYCLE STOP button is pressed, the LED located on the button goes on, and the LED located on the CYCLE START button goes off. This button is used when minor problems are encountered, such as coolant flow direction or when checking DISTANCE-TO-GO during setup. When the problem is remedied, press CYCLE START again to resume automatic cycle operation. It is not recommended using this button to interrupt a cut because the spindle does not stop; therefore, damage to the tool or part may occur. When pressed during the execution of the tapping or threading cycle, CYCLE STOP will take effect after the thread pass or the tap is withdrawn. If the tap breaks during the tapping cycle, the only way to stop the machine is by pressing the RESET button on the controller or the EMERGENCY STOP button.

Cycle Start

The CYCLE START button is used to start automatic operation. Use this button in order to begin the execution of a program from memory. When the CYCLE START button is pressed, the LED located above this button goes on and the active program will be executed to the end.

PRG STOP (Program Stop)

When a Program Stop is commanded in the program by the program word M00, automatic operation is stopped and the LED on this button is turned on. This button does not have an ON/OFF function that affects the program stop condition. It is merely an LED indicator lamp to indicate when a program stop condition is active.

Part 2 CNC Machine Operation

OPERATION

The keys in the Operation section (middle column) are used for manual operation of the machine during setup and initial startup. Their specific functions are described below:

REF (Reference)

Press the REF button to activate it. Then, pressing the X or Z buttons in the positive direction (X, Y or Z for machining centers) causes the machine to return to the Machine Zero position for each axis in relation to the machine coordinate system at the rapid traverse rate (maximum feedrate). The button must be held until the axis indicator LED is lit.

Jog

Pressing the JOG button activates a manual feed mode that allows the selection of manual feed movements along single axes X or Z (X, Y, or Z for machining centers). With the button activated, use the Axis/Direction (+ or –) buttons and the Feed Override dial to move the desired axis at the chosen feed rate. On some controls, Speed/Multiply is a rotary type switch that activates this function.

INC (Incremental)

Press the INC (Incremental JOG) button to activate the JOG mode in incremental steps. When the INC option is selected from the Operation Mode buttons, the incremental step selected by the Speed/Multiply buttons determines the magnitude of the displacement along the chosen axis in the selected direction. When one of the axes directional buttons is pressed and released, the movement will be as follows:

> X1 = a movement of .0001 inch or .0025 millimeters (mm)
>
> X10 = a movement of .001 inch or .0254 mm
>
> X100 = a movement of .010 inch or .254 mm
>
> X1000 = a movement of .100 inch or 2.54 mm

The buttons used to select the axis and the direction of movement (+, RAPID, and –) are located in the third column on the fourth row of the Operator Panel, as shown in Figure 2-6 for this controller.

For example, to displace the tool along the X-axis in a positive direction by the value of .010 inch, follow these steps.

- Press the INC button.
- Press the X100 button.
- Press the X button to activate the axis.
- Press the + button once.

Each time the button is pressed, a displacement of the selected value results.

If the JOG mode is selected when one of these buttons is activated, and the selected axis button is pressed and held in, the movement will occur at feed as indicated by the % Traverse Feed override dial.

Part 2 CNC Machine Operation

Handle

Pressing the HANDLE button activates the manual handle feed mode for the selected axis. This handle is known as the Manual Pulse Generator (MPG). Pressing this button places the machine in the HANDLE mode, which enables manual control of axis movements [for X, Y, or Z; or for rotational axes 4 *(A),* 5 *(B),* or 6 *(C)*] by use of the handle after activating their respective axis buttons (Figure 2-7). For instance, press HANDLE, then press X, and then use the Handle to move to the desired position along the X-axis. By turning the handle clockwise, you can move the tool in a positive direction with respect to the position of the coordinate system. By turning the handle counterclockwise, the tool is moved in a negative direction with respect to the position of the coordinate system. The handle contains 100 notches, each of which corresponds to an increment (distance to be moved). Turning the handle, you can feel the displacement from one notch to the next.

To set the magnitude for the distance to be moved, press one of the Speed/Multiply buttons as described in the INC section above.

Caution: If the handle is rotated quickly while the magnitude is set at X100 or X1000, the tool will move at a rapid feed rate and a crash could occur!

When the HANDLE mode is selected, the incremental step selected by the Speed/Multiply buttons determines the magnitude of the movement along the chosen axis in the selected direction. Each button setting corresponds with the scale of the hand-wheel. One full revolution of the hand-wheel (360º) corresponds to 100 units on the scale. On the button X1, the X means "times" the minimum increment. In manual control, you must use the + or – buttons to identify axis direction as well as to determine the axis of displacement.

Usually, X1 is used when you are precisely dialing-in the zero of the workpiece and when you are determining the tool length offset. For example, if you need to move the machine table with respect to the tool by 1.00" along the X-axis in a positive direction, follow these steps:

- Press HANDLE mode button.

- Press the axis directional button +X.

- Press the SPEED/MULTIPLY button X10.

- Turn the handle one full revolution (100 units) and then check the value of the displacement on the screen. It should indicate a movement of 1.00 inch.

Figure 2-7
Manual Pulse Generator (MPG) HANDLE
Courtesy FANUC FA America

X, Y, Z

The X, Y, and Z buttons correspond with each of the linear axes. When one is pressed, that axis becomes active for manual positioning by any of the manual modes.

4, 5, 6

The 4, 5, and 6 buttons correspond with each of the rotary axes. When one is pressed, that axis becomes active for manual positioning by any of the manual modes.

AXIS DIRECTION

+ and –

These buttons are used to select the manual feed axis direction. Pressing these buttons executes movement along the selected axis in the selected direction relative to the machine coordinate system—by Jog feed (or INC feed) or HANDLE—when the corresponding button is set to ON in the jog feed mode (or INC feed mode). The same is true for each of the linear and rotational axis buttons. As long as the button is held, the axis will move at a feed rate determined by the Feed Override dial or INC setting until released.

Rapid

After selecting the desired axis and pressing the Rapid button simultaneously with the + or – button, the machine will move along that axis at rapid traverse until released.

Caution: Prior to activation, be sure the part, fixture, or clamping devices that are in the chosen path are not going to interfere. The axis will move at a rapid feed rate and a crash could occur!

The Operator Panel in Figure 2-2 includes a Feed/Rapid % override dial you can use to control the rapid feed rate. This dial is used to reduce the rapid feed rate (G00). If it is positioned at 100, it corresponds to 100% of the rapid feed rate that the machine can generate. When override buttons are used, they are commonly incremented in steps of 10, 25, 50, and 100%.

SPINDLE

The spindle buttons are used exclusively during the machine's manual operation for setup functions. The descriptions below explain their specific function:

SPDL CW

By pressing the SPDL CW button while in one of the operation modes REF, JOG, INC, or HANDLE, the spindle will start rotation in the clockwise (CW) direction. The spindle rev/min is adjusted by using the Spindle Override dial. When set at 100%, the spindle will rotate the last r/min commanded in the program. The spindle command is retained and will restart upon pressing the SPDL CW button. When the machine is first started, there has been no value established for the r/min. Therefore, if one of these buttons is pressed while in the modes listed above, an alarm will result. An r/min must be input via MDI or by activating the program. From that point on, as long as the machine is not turned off, the r/min will be activated at the last commanded value when one of these buttons is pressed.

SPDL STOP

Pressing the SPDL STOP button stops spindle motor rotation while in one of the operation modes listed above. However, pressing this button will NOT stop the spindle while in any of the Automatic execution modes.

SPDL CCW

Pressing the SPDL CCW button is exactly the same as pressing the SPDL CW *except* that the spindle will start rotation in the counterclockwise (CCW) direction.

COOLANT

Although not shown on this operator panel, there are buttons that control coolant flow during manual or automatic operation. In some cases, there is a manual switch to turn coolant on and off.

CONTROL PANEL

The control panel described here is quite typical of the control panels used on CNC machines. The control panel switches and buttons may be distributed differently on the panel for each individual machine; however, the purpose and function of each switch and button remains the same. Some control panels are equipped with additional buttons or switches not shown here. Definitions and applications of these buttons or switches can be found in the manufacturer instruction manuals for the machines.

The control panel is located at the front of the machine and is equipped with a CRT and with various buttons and switches, as illustrated in Figure 2-8.

PC integrated controls are commonly identified with the lower case letter i after the controller series number, i.e., 0i. They allow you to use third party software like Excel, etc., and to connect to the Internet for diagnostic purposes and remote access. Modern machine tools are equipped with Ethernet connectivity to your company's network (some offer wireless), thereby offering unlimited part program and tool file storage with easy access via Windows-based operating systems.

For this controller, there is an access door on the upper left side of the display that contains a memory card slot and USB port. Some older controls *still* have a 3.5 floppy disk and possibly a PCMCI (Portable Computer Memory Card Interface) slot. All of these methods are used as a file storage medium and transfer.

There are two alpha key arrangements. QWERTY (Figure 2-1) matches standard keyboards while the ONG style, which is more popular for 0i, is shown in Figure 2-8. A detailed description for the use of each button and its purpose on the control panel are presented in the following sections.

POWER-ON AND POWER-OFF

Located in the lower left hand corner of the control in Figure 2-8, the Power-On and Power-Off buttons are used to activate/deactivate the power to the control. Press these buttons to turn CNC power ON and OFF. The ON button is white in color and when it is ON, the key is lit. The OFF button is typically black in color and when the power is turned OFF to the control, the key is lit. At startup of the main power to the machine, the OFF button is lit. On older machines, these keys will be green and red

Part 2 CNC Machine Operation
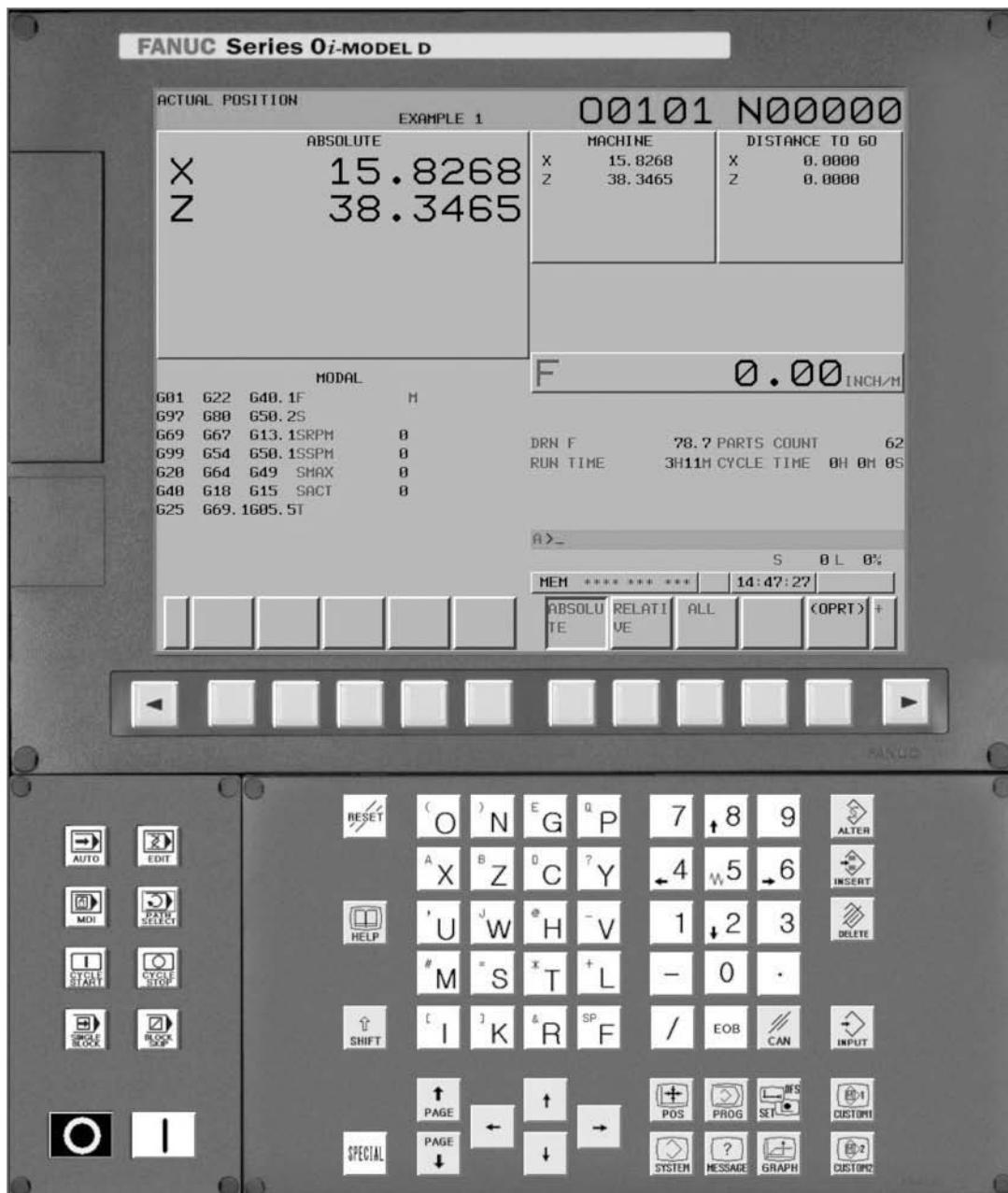
Figure 2-8
Common Control Panel
Courtesy FANUC FA America

in color respectively. Directly above these buttons are the Operation Selection buttons which are described in the same-named section of this chapter. Path Select is used for multi-path controls. For this controller, it is inactive.

Note: The control is always turned ON after turning on the MAIN POWER switch, which is located on the door of the control system, typically at the back or side of the machine. The control is always turned OFF before the MAIN POWER switch is turned OFF.

CRT Display

The CRT is the display screen on which all the program characters and data are shown. Sizes vary from around 9 inches to approximately 15 inches. Displays are color, monochrome, or liquid crystal displays (LCD). FANUC displays (shown here) offer 8.4, 10.4, and 15-inch screens.

Reset

Pressing the RESET button resets or cancels an alarm; it can also be used to cancel an automatic operation. An alarm can only be cancelled if its cause has been eliminated.

When the reset button is pressed during automatic operation, all program-commanded axis feeds and spindle revolutions are cancelled. The program will return to its starting block when this button is pressed.

Help

Pressing the HELP key gives the operator access to help screens on how to operate the machine functions such as MDI key operation, or details related to an alarm that has occurred in the control.

Figure 2-9
RESET Button
Courtesy FANUC FA America

SOFT KEYS

The soft keys have numerous functions, depending on the applications selected along with other keys. The specific functions of the soft keys are displayed in the box above the key at the bottom of the CRT screen, as shown in Figure 2-11. The purpose of the soft keys is to minimize the use of dedicated keys on the control panel.

Figure 2-10
HELP Button
Courtesy FANUC FA America
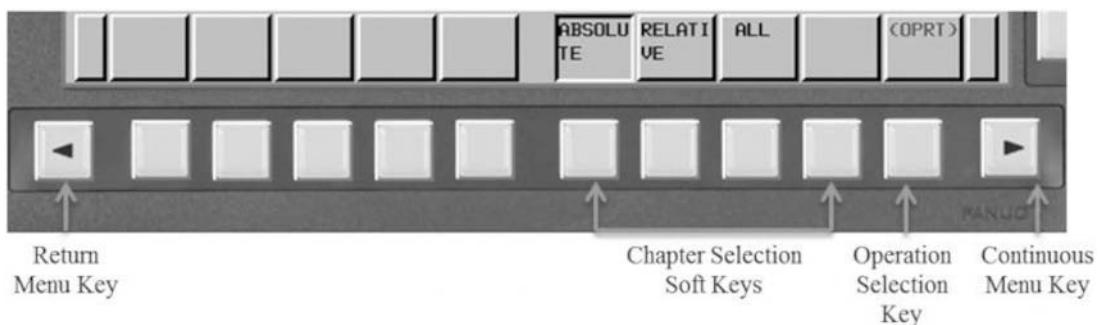
Figure 2-11
Controller Soft Keys
Courtesy FANUC FA America

The selection soft keys are the first four rectangular keys under the CRT. By pressing one of them, the function selections that belong to that key appear. These selection choices are called chapters. If the soft key for a target chapter is not displayed, you must press the continuous menu key located at the right end of the soft keys (sometimes referred to as next menu). In some cases, there are additional chapters that can be selected from within a chapter. When the desired screen is displayed, press the soft key under operation selection (OPRT) on the display in order to manipulate the data. To reverse through the chapter selection soft keys, press the return menu key located at the left end of the soft keys.

The machine ORIGIN register can be reset by use of the soft keys. READ soft keys let you enter the program to memory from a punched tape or other storage medium; the PUNCH soft key allows program readout from memory to one of several options of storage medium.

The general screen display procedure is explained above; however, the actual display procedure varies from one screen to another. For details, see the description of individual operations.

Note: The operator should consult the manufacturer's manual for more specific detailed instructions on the use of the soft keys.

ADDRESS AND NUMERIC KEYS (ALPHA-NUMERICAL KEYS)

This keypad of letters, numbers, and symbol characters is used to input data while writing or editing programs at the control. These keys are also used to enter numerical data and offsets into memory. Many of the keys are used in conjunction with other keys.

Shift

Because there is not enough space on the control for all keys necessary, the ONG keys have two characters on them. When the letter or symbol indicated in the upper left corner of the key is needed, the operator first presses the SHIFT key, which switches the key to that character. This sequence must be followed each time an alternate letter is needed. The shift key functions the same way as its equivalent on a computer keyboard.

When the Shift key is pressed a special character will be displayed in the left upper corner of the screen. Then the desired (second) character on the key may be entered.

Cancel

The CANCEL key is used while inputting data in the MDI mode. It is essentially a destructive backspace key and can be used to correct an erroneous

Figure 2-12
Alpha-Numerical Keypad
Courtesy FANUC FA America

Figure 2-13
SHIFT Key
Courtesy FANUC FA America

entry. Press this key to delete the last character or symbol input to the key input buffer. For instance, when the key input buffer displays:

N5 X12.00 Z

and then the cancel key is pressed, the address Z is erased and:

N5 Xl2.00

is displayed.

EOB

The EOB key is the END-OF-BLOCK key. When pressed while in the MDI mode, the EOB character (;) is inserted into the program at the cursor location.

Note: The (;) symbol is never part of the program manuscript. When a program is edited offline using a PC, the control system will automatically show the EOB character for each time the "Enter" key is used on the keyboard.

Input

The INPUT key is used for MDI operation and to change the offsets. After the data are entered via the keypad, the INPUT key is pressed. The data are entered into the offset register or the program for execution.

PART PROGRAM EDIT KEYS

These keys are used to enter new program data (INSERT), to make program changes (ALTER/CALC), or to delete program data in memory (DELETE). They are used while editing programs.

FUNCTION BUTTONS

The Function buttons correspond to particular display modes (active mode). By pressing any one of these buttons, the display will be switched to the corresponding screen. Then the soft keys may be used to display the needed data.

Figure 2-14
CANCEL Key
Courtesy FANUC FA America
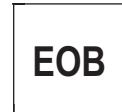
Figure 2-15
End-Of-Block Key
Courtesy FANUC FA America
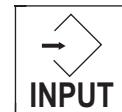
Figure 2-16
INPUT Key
Courtesy FANUC FA America

Figure 2-17
Program Edit Keys
Courtesy FANUC FA America
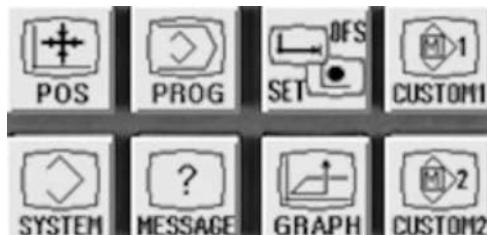
Figure 2-18
Function Buttons
Courtesy FANUC FA America

- Press the POS key to display the position screen.
- Press the PROG key to display the program list screen.
- Press the OFS/SET (Offset/Setting) key to display the screen used to set offsets or adjust parameter settings.
- Press the SYSTEM key to display the system screen.
- Press the MESSAGE key to display the message screen.
- Press the GRAPH key to display the graphics screen.
- CUSTOM1 and CUSTOM2 are keys reserved for the display of conversational macro or C Language Executor.

Cursor

The cursor shows a blinking dash on the display located below the position of a particular address while in one of the Edit modes. On many controls, the cursor highlights the whole word, for example, X7.777.

Cursor Move

In order to navigate through the program, four keys are used to move the cursor.

The right pointing arrow key moves the cursor to the right or in the forward direction. When this key is pressed, the cursor moves only one space each press of the button, in the forward direction. The left pointing arrow key moves the cursor to the left or in the reverse direction. As with the right arrow, when this key is pressed, the cursor moves only one space each press of the button, in the reverse direction.

Figure 2-19
Cursor Move Keys
Courtesy FANUC FA America

The downward pointing arrow key moves the cursor downward through the program in the forward direction. Each time this key is pressed, the cursor moves downward one full line. The upward pointing arrow key moves the cursor upward through the program in the reverse direction. Each time this key is pressed, the cursor moves upward one full line.

Use the CURSOR button with the arrow pointing up to change pages in the opposite direction. For example:

O0001

N1 G50 X7.777 Z7.777 S1000

N2 T0100 M39

N3 G96 S600 M03

In this example, the CURSOR is resting below N. By pressing the CURSOR button three times with the right-pointing arrow, the cursor moves below the letter (address) G.

By pressing and holding the CURSOR button with the up arrow, the prompt will move to the first word of program O, which corresponds to the upper limit of cursor movement. Another fast way to return to the program head is to press the RESET key.

By pressing the CURSOR button once, with the arrow pointing down, the cursor will move down one line. If the cursor must be moved over a few or many words, you need not press the button repeatedly. Just press and hold this button down; the cursor automatically jumps one word at a time in the given direction.

Page Up/Down

Usually the length of the program exceeds what the height of the screen will display. The CURSOR move keys can be used to scroll through the program line-by-line. A more effective method to move a large amount is to use the two PAGE keys. Using these keys will advance in the direction selected by the number of lines the screen can display. The last block of a given page becomes the first block of the next page. The PAGE keys allow for scrolling through long programs more effectively.
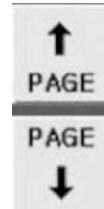
Figure 2-20
Page Up/Down Keys
Courtesy FANUC FA America

PC FUNCTION

This set of function keys are used for PC Functions.

ABC/abc

This key is used to switch from Caps-Lock (all capital letters) to lower case in the same manner as with PC functions.
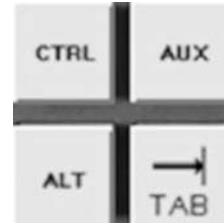
Figure 2-21
PC Function Keys
Courtesy FANUC FA America

Special

The Special Key allows selection of two keys simultaneously from within the NC Guide i software (PC version only). It is used for PC operations where pressing of multiple keys is required.

- Press the Special key.

- Press the required multiple keys in any order. The command will be executed upon final key entry.
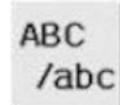
Figure 2-22
ABC/abc Key
Courtesy FANUC FA America

OPERATIONS PERFORMED AT THE CNC CONTROL

The following explanations are for operations considered routine for users of CNC machine tools and are given in their sequence of use. Please note that these procedures are specific to the type of controller depicted here (Fanuc series 0i). The procedures for another type of control may be similar. Be sure to consult the manufacturer manuals specific to your machine tool operation and control panel.

Figure 2-23
SPECIAL Key
Courtesy FANUC FA America

Part 2 CNC Machine Operation

The Machine is Turned On and Homed (Machine Zero)

Turn on the main power switch, and then press the ON Power button on the controller. Most modern machine tools will automatically start-up in the REF/ZERO RETURN mode. This means that before any automatic or manual operation may begin, you must Home the machine first.

If the LED above the REF button in the operation section of the Operator Panel is not lit, press it now to activate the mode.

Using the Axis/Direction keys, press the direction necessary to HOME the machine. Note that many machine tools will have LEDs for each axis that are lit to indicate when that axis is HOMED.

At machine start-up, a common screen displayed is ACTUAL POSITION (ABSOLUTE). If it is not displayed, press the function key labeled POS, then the soft-key ABSOLUTE. The displayed coordinate values represent the relationship between the Workpiece Zero and the Machine Zero (HOME). When the machine is HOME, press the soft-key OPRT, then ORIGIN, and then ALL AXIS to zero each of the coordinate axes.

By pressing the soft keys, you can activate other display screens. For instance, when you press the button ABSOLUTE (which corresponds to position), the digital counter appears on the screen for the X, Y and Z axes, which is the absolute coordinate system for a given workpiece (for turning centers, X and Z will be displayed). The position (POS) function is assigned four display screens and can be found by pressing the soft keys labeled ABSOLUTE, RELATIVE, ALL, and (OPRT). The first screen corresponds to a position change in the ABSOLUTE (ABS) system for X, Y, *and* Z, as illustrated in Figure 2-24. The second screen RELATIVE (REL) corresponds to position changes in the incremental system for milling machines X, Y, and Z (*U* and *W* for turning centers). The third, ALL, gives representation of all four of the displays simultaneously on one screen, as shown in Figure 2-25.

The values listed in the readout for MACHINE represent the distance from Machine Home position.

The DISTANCE TO GO readout is the most significant part of the third display. The coordinates in this quarter of the screen display correspond to the path that will be
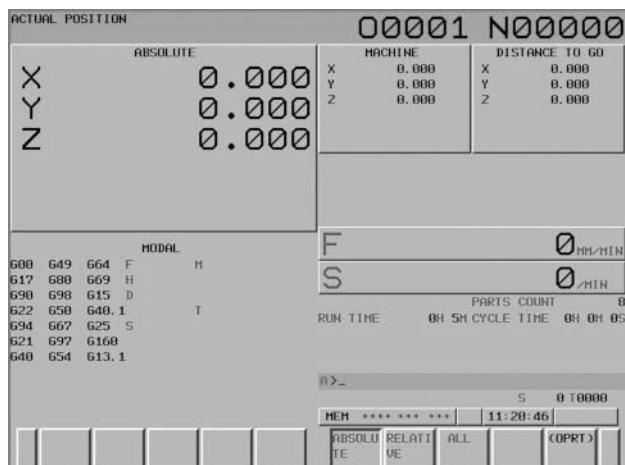
Figure 2-24
Actual Position (Absolute) Screen
Courtesy FANUC FA America
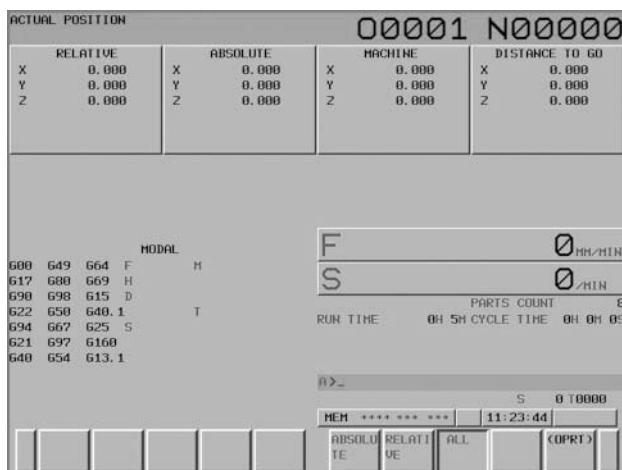
Figure 2-25
Actual Position, ALL Screen
Courtesy FANUC FA America

followed by the tool in order to complete the execution of a given block of information while under automatic operation.

Example

N20 G00 Z0.0

N22 G01 Z–12.00 F.015

When block N22 is first read by the control, the value Z–12.000 will appear under the DISTANCE TO GO readout in the upper right corner of the screen. After moving a distance of 1 inch, the value of coordinate Z changes to Z–11.000, and so on. The other displays, ABSOLUTE and RELATIVE, correspond to the first two display screens, but this time they are smaller so that all four displays may fit on one screen. All of the displays may be changed to read in millimeters, with respect to Machine Zero, by changing a machine parameter or by using a G-Code in the program.

The codes listed in the lower left corner of the display pictured in Figures 2-24 and 2-25 are the default G-Codes that are active upon startup of the machine. They are also reinstated by pressing the RESET key.

A Program is Loaded from CNC Memory

The program may be in the program directory, but not activated for automatic operation as shown in Figure 2-26. Follow these steps to activate a program.

1. Press the EDIT button to enter the EDIT mode.
2. Press the PROG function button.
3. Press the DIR soft key.
4. Key in the desired program number from the list. Soft key availability changes to include O SRH.
5. Press the O SRH soft key. The program is now activated.
6. Press the AUTO cycle key to execute the program.
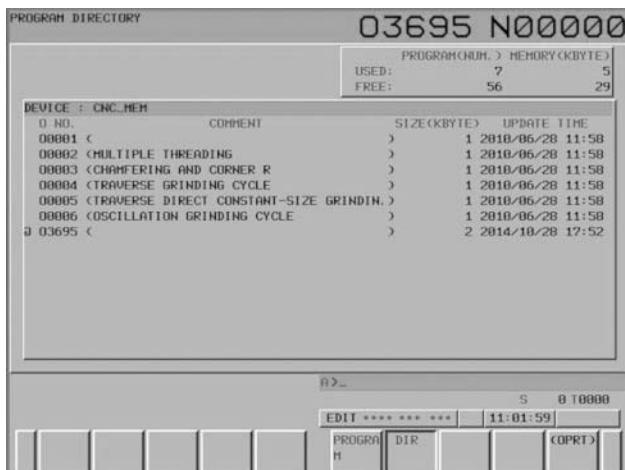
Figure 2-26
Program Folder Screen
Courtesy FANUC FA America

A Program is Loaded from an Offline Location

Examples of offline locations include memory cards (PCMIA) and USB drives. Be sure that the output device is ready.

1. Press the EDIT button.
2. Press the PROG function key.
3. Press the DIR soft key.
4. Press the OPRT soft key.
5. Press the DEVICE CHANGE soft key.
6. Select the MEMORY CARD soft key to see available the files or MEM CARD for those on USB.

Select the desired file from the list by keying in the program number and then press the EXEC soft key to activate.

For network drives connected via the Ethernet, if available:

1. Press the EDIT button.
2. Press the PROG function key.
3. Press the DIR soft key.
4. Press the OPRT soft key.
5. Press the DEVICE CHANGE soft key.
6. Select the EMBED ETHER soft key to see available the files from the EMBEDDED ETHERNET HOST FILE LIST.
7. Press the F INPUT soft key.
8. Key in the desired program file name or select desired file from the list by using the cursor move keys.
9. Press the F NAME soft key.
10. Press the EXEC soft key to activate.

Part 2 CNC Machine Operation

A Program Folder is Saved from CNC Memory

1. Press the EDIT button on the operator panel to enter the EDIT mode.

2. Press PROG function key.

3. Press the OPRT soft key.

4. Press the rightmost (continuous-menu key) soft key two times.

5. Press the F OUTPUT soft key.

6. Select F SET or P SET to identify the File name or Program name respectively.

7. Use the keypad to enter the desired program number, preceded by the letter address O, and then press the EXEC soft key.

The program will be saved to the selected offline location media.

8. To save all programs stored in memory, use the same steps above Press O–9999.

A Program is Deleted from Memory

To delete a program from the controller memory, follow these steps:

1. Enter the EDIT mode.

2. Press the PRGRM soft key.

3. Press the DIR soft key.

The program directory will be displayed.

4. Press the OPRT soft key.

5. The screen with one of the soft keys labeled DELETE will be displayed.

6. Enter the program file number (preceded by the letter address O) that you wish to delete.

7. Press the DELETE soft key.

8. Press the EXEC soft key.

The file is deleted.

MDI OPERATIONS

You may input small programs via the keypad at the control. The size of the program, which is limited to 10 lines on the control described in this book, is determined by the parameter setting from the manufacturer. Small programs provide an excellent method of executing simple commands like tool changes, controlling the spindle r/min and its rotation direction, etc. To enter the MDI mode of operations, follow these steps:

1. Press the MDI button on the Operator Panel.

2. Press the PROGRAM function key.

3. Enter the data to be executed by using the methods described later in the section Program Editing Functions.

Part 2 CNC Machine Operation

For the program number, the control assumes O0000 and the data may be entered. Each block ends with the end-of-block (EOB) character (;) so that individual blocks of information can be kept separately. For example:

N1 G50 S1000;

4. Press the EOB function key to insert the semicolon at the end of each line.

5. Press the INSERT key.

6. Press CYCLE START to execute the program information.

If you make a typographical error while entering a given block, you can eliminate it by pressing the CAN key to cancel the error and then reenter the correct value.

You may execute the MDI program as you do with automatic operation. The same control functions apply except that an M30 (tape rewind) command does not return the control to the program head; instead, M99 is used to perform this function. Please refer to the machine tool manufacturer manual for specific instructions.

You can erase an entire program created in MDI mode by the following step:

1. Press the RESET key.

The program will also be erased when the last block of the program is executed by single–block operation.

To perform an individual MDI operation, use the methods described above. For the control described, the display screen is shown in Figure 2-27.

Example 1

1. Turn on the spindle at 500 RPM in the clockwise direction.

2. Key in the following command:

3. S500 M03

4. EOB

5. INSERT

6. CYCLE START
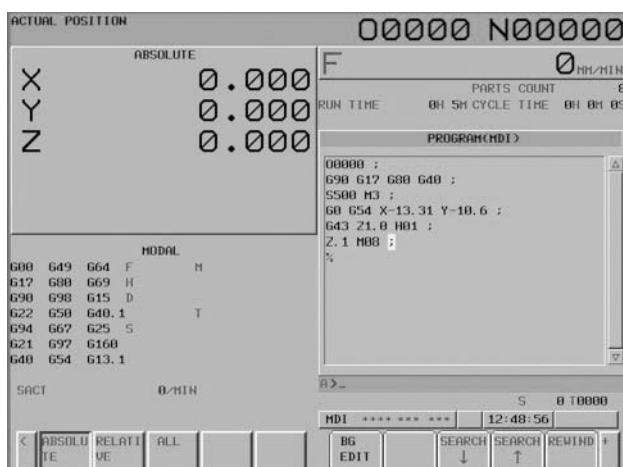
Figure 2-27
Program Screen in MDI Mode
Courtesy FANUC FA America

Example 2

Use the following directions to position tool number 5 to the active position on the turret (or to install tool 5 into the spindle on a milling machine). Key in the following commands:

1. T5 M6 (or T0500for a turning center)
2. EOB
3. INSERT
4. CYCLE START

MEASURING WORK COORDINATE OFFSETS: MACHINING CENTER

Following is the procedure for setting the Work Offsets for each workpiece coordinate system G54 to G59. When the values are known or adjustments are needed, you can:

1. Press the OFFSET/SETTING function key.
2. Press the WORK soft key.

The WORK COORDINATES setting screen is displayed as shown in Figure 2-28. Two display screens are needed to handle the six offsets G54–G59. To display a desired page, follow either of these two methods.

Method 1

1. Press the PAGE UP or PAGE DOWN keys until the desired offset is shown.
2. Use the cursor move keys to select the offset number G54–G59 and axis desired.

Alternatively, you can input an offset number 01–06 and press the NO.SRH (number search) soft key.

To change the coordinate values of the offsets, use the following method.

Method 2

1. Use the alphanumeric keypad to enter the new value for the offset.
2. Press the INPUT soft-key.
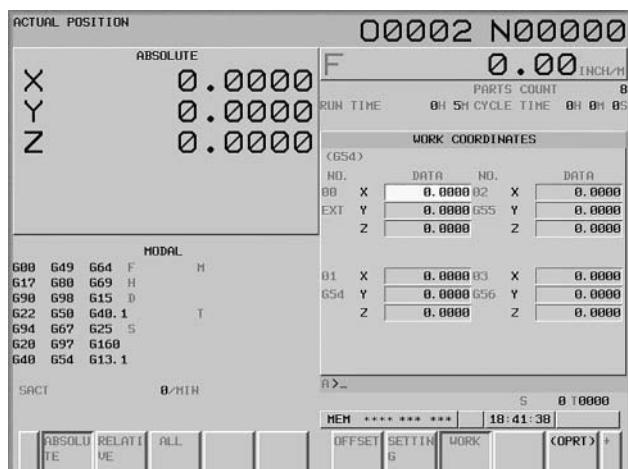
Figure 2-28
Work Coordinates Display Screen
Courtesy FANUC FA America

Part 2 CNC Machine Operation

Note: When the INPUT key is used to enter values, the amount entered will replace any amount in the register. When the +INPUT or –INPUT key is used, the existing amount in the offset register will be added or subtracted, whichever applies, by the amount entered into it.

Once the value is entered here, it is the new Workpiece Zero or origin for the workpiece coordinate system. To change an offset by a specific amount, use the alphanumeric keypad to enter the desired value; then press the +INPUT soft key.

Measured Values

Work Offsets can be measured manually by positioning an edge-finding tool to contact with the workpiece zero surface in both X and Y axes sequentially. In this procedure, which is called edge-finding, it is nearly always the perpendicular edges (secondary and tertiary datum) of the workpiece that are referenced.

Follow these steps for measuring Work Coordinate Offsets:

1. Position the machine to HOME.

2. Use the procedure above (steps 1–2) to find the Work Coordinates setting display screen.

3. Use the arrow keys to position the cursor on the offset you wish to use.

4. Press 0 INPUT for the X value.

5. Press 0 INPUT for the Y value.

6. Install an Edge-Finding tool into the spindle using MDI or manually.

7. Start the spindle RPM clockwise at approximately 1000 either manually or by using MDI.

8. Manually position the tool tip edge to contact the workpiece zero surface along the X- or Y-axis.

9. Use the cursor keys and select X or Y; then input the value of the current position related to the workpiece.

10. Press the MEASUR soft key. The absolute position value will be input to the offset.

11. Manually retract the edge-finding tool and repeat the same operation for the remaining axis. In most cases, you will be required to input the difference between the value input and the edge-finder radius (typically 0.100 or 3mm) before automatic operation can be executed.

MACHINING CENTER TOOL OFFSETS

Tool Length Offsets (TLO) are referenced in the program by words beginning with H. The values input into the corresponding T# (LENGTH) GEOM column are needed for to properly position the tool along the Z-axis. When adjustments are needed to compensate for wear, values are input into the WEAR column. Similarly, the Cutter Diameter Compensation (CDC) values are entered on the Offset display register into the (RADIUS) GEOM column and are referenced in the program words beginning with D. These compensations are important for proper radial (X, Y) positioning of the tool. If the

values are known, the following sequence can be used to input them into the offset page. When the setup values are known, you may:

1. Press the OFFSET/SETTING function button.

2. Press the OFFSET soft key to display a figure such as Figure 2-29.

3. Use the cursor move keys or page keys to position the cursor to the tool number to be set.

The search method may also be used by entering the tool number whose compensation is to be changed and then pressing the NO.SRH soft key.

Enter the numerical, value of the offset (including sign) and press the INPUT soft key.
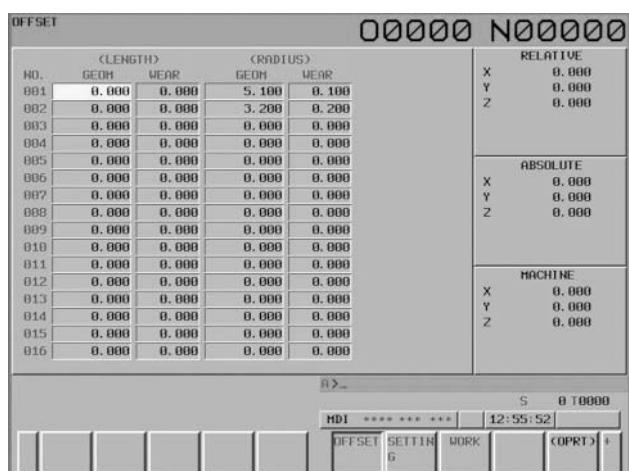
To add or subtract from an existing offset value, key in the amount (a negative value to reduce the current value) then press the +INPUT soft key.

Diameter compensation values are input as known after measuring their actual size. Depending on the parameter setting for the specific machine used, the value is entered as either tool diameter or radius. Consult the appropriate manufacturer operation manual for exact conditions.

Measured Values

Tools length offsets can be measured by manually positioning the tool tip to contact the Workpiece Zero surface (Z-axis). This procedure is called "Touching-Off" and is nearly always the top most surface, primary datum of the workpiece. All tools used in the program must have their offsets recorded in the Offset register. If there is not a value in the offset register for a programmed tool, the control will not execute for that tool call, an alarm will occur, and the machine will stop. If a value of zero is in the offset register, the control will accept the zero offset and over travel will result. Conversely, if a value in the offset register is incorrect, the control will execute the tool call as if it was correct and the result could be a collision. For this reason, it is a good idea to delete tool offset data from the offset register when the tool for which it was intended is removed. To do this, input a value of zero for the tool offset register desired.

Figure 2-29
Machining Center
OFFSET/GEOMETRY
Display Screen
Courtesy FANUC FA America

62

The following steps are needed for the tool offset measuring procedure:

1. Manually position the tool tip to contact the workpiece zero surface (Z-axis).

2. Press the POSITION function key.

3. Select the RELATIVE soft key.

4. Use the alphanumeric keypad and press Z and then INPUT to enter the axis to be measured. The axis should be blinking on the display screen and the soft key options PRESET and ORIGIN shown.

5. Press the ORIGIN soft key and then EXEC. The value in the RELATIVE position display will be changed to 0.0.

6. Press the OFFSET/SETTING function button and then press the OFFSET soft key to display the offset page for tool compensation.

7. Manually position the tool tip to contact the workpiece zero surface (Z-axis).

8. Use the arrow direction keys or the search method described above to position the cursor to the desired offset.

9. Use the alphanumeric keypad and press Z.

10. Press the INPUT soft key.

The relative Z value for the tool offset will be input to the offset register. Repeat for each tool used in the program.

Machining Center Tool Sensor Measuring

On most modern machines, a tool sensor is used as opposed to manually measuring each tool length. When this is the case, all of the programmed tools are manually or automatically positioned to contact the sensor for each tool axis, and the offset values are automatically input into the control. Review the operator manual specific to your machine for exact procedures.

ADJUSTING WEAR OFFSETS FOR MACHINING CENTERS

For machining centers, WEAR offset is assigned in the direction of the Z-axis for tool length compensation. Variations in the X- and Y-axes are compensated by adjusting the values in the (RADIUS) WEAR column. The method for inputting adjustment data is similar to adjusting wear offsets for turning centers.

MEASURING WORK OFFSETS, TURNING CENTER

It is necessary to establish a relationship between the machine coordinate system and the workpiece coordinate system. The following steps are necessary to input the measured values for the workpiece zero to the control's Work Coordinates offset page.

Measure the Z-Axis Work Coordinate

1. Identify the coordinate system G54–G59 to be used.

2. Manually position the cutting tool and make a cut on the face of the workpiece.

3. Without moving the Z-axis, stop the spindle and move the tool away from the part in the X-axis direction.

4. Identify the distance along the Z-axis from cut surface to the desired zero point.

5. Press the WORK soft key to display the WORK COORDINATES display screen.

6. Position the cursor to the desired workpiece offset to be set.

7. Use the letter address key Z to select the axis to be measured.

8. Use the value of the measurement taken to input the Z-axis work coordinate.

9. Press the MEASUR soft key.

   The work coordinate for the Z-axis will be input.

Measure the X-Axis Work Coordinate

1. Manually position the cutting tool and make a cut along the Z-axis to create a diameter on the workpiece.

2. Without moving the X-axis, stop the spindle and move the tool away from the part in the Z-axis direction.

3. Measure the diameter you just cut on the workpiece.

4. Use the value of the measurement taken to input the X-axis work coordinate (enter the diameter).

5. Follow the same procedure for setting the Z-axis work coordinate value as stated above in Steps 6 and 7.

   The work coordinate for the X-axis will be input.

TURNING CENTER TOOL OFFSETS

On turning centers, the tool offsets are measured in two directions: Z and X. These values represent the difference between the reference position (Machine Home) of the tool turret and the actual position of a tool tip used as the programmed tool point. The amount of Tool Nose Radius is input on the OFFSET display screen where R is indicated for each tool. An incorrect value here will have an effect on the finished part where tapers and radii are turned. Refer to Part 3, *Tool Nose Radius and Tip Orientation (T)*, for more details.

Measured Values

If the position register commands (G50 for turning and G92 for milling) are used, the values for each tool that have been measured will be input into the program for each tool with the G50 or G92 command.

The more commonly used method today is to input these values into the OFFSET/GEOMETRY register for each tool (Figure 2-30). Follow these steps to input the measured tool offset value.

Measure the Z-Axis Offset

1. Manually position the cutting tool and make a cut on the face of the workpiece.

2. Without moving the Z-axis, stop the spindle and move the tool away from the part in the X-axis direction.

3. Measure the distance along the Z-axis from the cut surface to the desired zero point.

Figure 2-30
Turning Center
OFFSET/GEOMETRY
Display Screen
Courtesy FANUC FA America

4. Use this value to input the Z-axis offset for the desired tool number with the following procedure:

a) Press the OFFSET/SETTING function button.

b) Press the OFFSET soft key.

c) Use one of the search methods or use the cursor keys to move the cursor to the offset number to be set.

d) Use the alphanumeric keypad to select the letter address Z.

e) Use the alphanumeric keypad to key in the value of the measurement taken.

f) Press the MEASURE soft key.

Input the difference between measured value and the coordinate as the offset value.

Measure the X-Axis Offset

1. Manually position the cutting tool and make a cut along the Z-axis to create a diameter on the workpiece.

2. Without moving the X-axis, stop the spindle and move the tool away from the part in the Z-axis direction.

3. Measure the diameter just cut on the workpiece.

4. Follow the same procedure for setting the X offset value as stated above (steps 4. a–f) to input the diameter measured in step 3.

Apply this method for all of the remaining tools used in the program. The offset values are automatically calculated and set.

Turning Center Tool Sensor Measuring

On most modern machines, a tool sensor is used as opposed to machining the diameter and face of the material. In this case, all of the programmed tools are manually or automatically positioned to contact the sensor for each axis and the offset values are automatically input to the control. The operator still must manually enter Tool Nose Radius compensation values in the "R" column of the OFFSET/GEOMETRY register and

Part 2 CNC Machine Operation

Tool Tip Orientation "T". Review the Operator Manual specific to your machine for exact procedures.

Adjusting Wear Offsets for Turning Centers

Wear-Offsets are used to correct the dimensions of the workpiece that change because of cutting tool wear. For a turning center, the X direction offset corresponds to the diameter. For example, if the X wear offset for a tool is .01, an incremental change of minus .01 refers to a decrease of the diameter by .01 and an incremental change of plus .01 refers to an increase of the diameter by .01.

To adjust the WEAR-offsets:

- Press the OFFSET/SETTING button.
- Press the OFFSET soft key. The screen display shown in Figure 2-31 appears.

Examples of Adjusting Wear Offsets

- For the following examples, the operator should display the OFFSET screen for WEAR offsets and the cursor should be positioned to the tool and axis requiring adjustment.

Example 1: The Absolute System
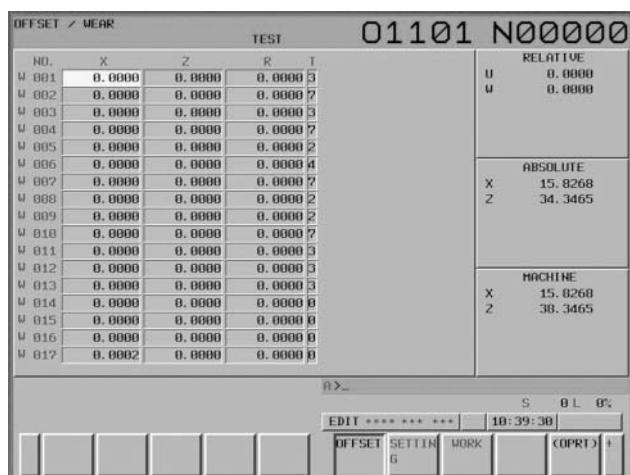
After machining the workpiece shown in Figure 2-32, if the measured external diameter exceeds the value of tolerance (for example, 1.003), enter the offset with a negative sign assigned to the value –.003 in the wear offset by following these steps.

1. Press X
2. Key in –.003
3. Press INPUT+

Then, after machining several more pieces, the diameter increases due to tool wear. If the measured diameter is 1.002, enter the offset as follows:

1. Press X
2. Key in –.005
3. Press INPUT+

Figure 2-31
OFFSET/WEAR
Display Screen
Courtesy FANUC FA America
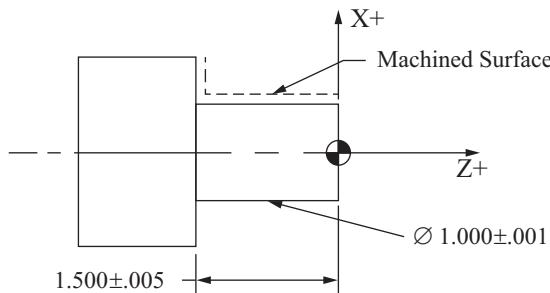
Figure 2-32
Example of a Machined Workpiece
Used for Adjusting Wear Offsets
Courtesy FANUC FA America

Please note that it was necessary to add a value of .002 into the Offset register to the previously entered value of .003. A similar approach is applicable in the direction of the Z-axis.

If the measured length is 1.492, then the value of the offset entered is −.008.

1. Press Z

2. Key in −.008

3. Press INPUT+

A new measured length of 1.494 gives an entered value of the offset of −.006.

1. Press Z

2. Key in −.014

3. Press INPUT+

Example 2: The Incremental System

To gain a better understanding, let us examine identical cases when the incremental coordinate system is used. The measured value is = 1.003.

Offset: *U*

1. Key in −.003

2. Press INPUT

Following that, the diameter is = 1.002.

3. Press *U*

4. Key in −.002   (on the screen)

5. Press INPUT (X−.005)

And Z = 1.492

Offset: *W*

1. Key in −.008

2. Press INPUT

After machining a few pieces, Z = 1.94.

Offset: *W*

3. Press *W*

4. Key in –.006 (on the screen)

5. Press INPUT (Z–.014)

TOOL PATH VERIFICATION OF THE PROGRAM

A standard feature of modern controllers helps you verify that the program is ready to use via graphic display of the programmed tool path, as shown in Figure 2-33 and Figure 2-34 (output from an NC Guide i simulation). This visual representation of the programmed tool path provides yet another check enabling you to catch any errors before machining takes place.

Follow these steps to access this display screen.

1. From the AUTO mode, press the GRAPH function button.

The graphics parameter screen will be displayed (not shown). Use the cursor to position to each parameter and the INPUT key to insert all of the required data.

2. Press the GRAPH soft key.

3. Press Cycle Start and simulation of the programmed tool paths will be displayed on the screen.

You have the added ability to adjust the magnification (ZOOM), and to CENTER the workpiece on the display.

DRY RUN OF PROGRAM

Under the DRY RUN condition, the tool is moved at the rapid traverse feed rate regardless of the feed rate in the program. The actual feed rate is determined by a parameter setting and also by a rotary dial or override buttons on the operation panel.

This function is used for checking the programmed movements of tools when the workpiece is not present in the work holding device. The rapid traverse rate can be adjusted by using the rapid traverse feed override or by using the feed override on the
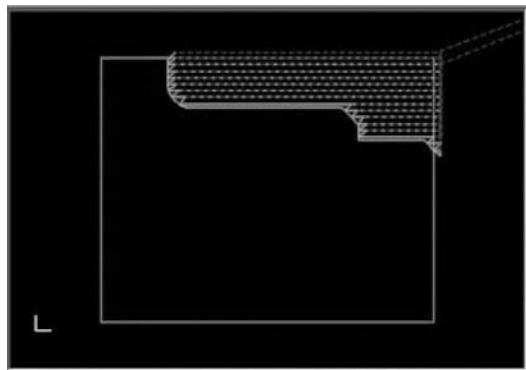
Figure 2-33
Tool Path Graphics Display
Courtesy FANUC FA America
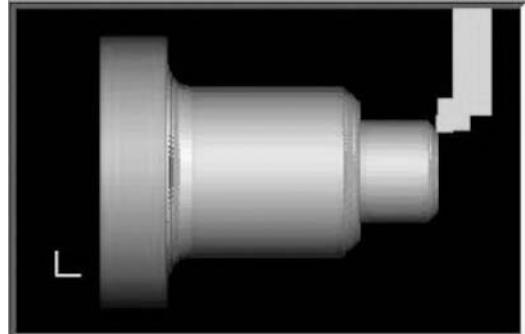
Figure 2-34
Tool Path Animated Simulation Display
Courtesy FANUC FA America

operator control (refer to FEEDRATE *OVERRIDE and Figure 2-3*). Consult the manufacturer's manual for specific instructions.

Caution: This function is not intended for metal cutting.

Another form of DRY RUN is to execute the program cycle at the actual programmed feed rates, but without a part mounted in the work holding device.

EXECUTION IN AUTOMATIC CYCLE MODE

Figure 2-35 shows the Program Check display screen with descriptions for each content area. When all the prior steps have been completed, the program is ready to be executed under automatic cycle. A helpful and informative display screen to use during this first cycle is ALL/PROGRAM. This display, shown in Figure 2-35, is convenient because you can see program contents as they are being executed: the Absolute Position display, a Distance-To-Go display, and all of the active modal commands.

To begin execution in the automatic cycle mode, follow these steps:

1. Be sure that the desired program is in the control and active and that all set-up procedures have been completed. If it is not, use the steps above to activate by following the directions as stated in the section *A Program is Loaded from CNC Memory*.

2. Press the AUTO mode.

3. Press the RESET key on the controller.

4. Press the PROG function button on the controller.

5. Press the NEXT soft key on the controller.

6. Press the ALL soft key.

7. Press the CYCLE START button on the Operator Panel.

The automatic cycle will begin.

DNC OPERATION

Occasionally, it will be necessary to run the program from a remote storage device (memory card or a computer hard disk). This is typical for instances where the program is very large and will not fit in the control system memory. Direct Numerical
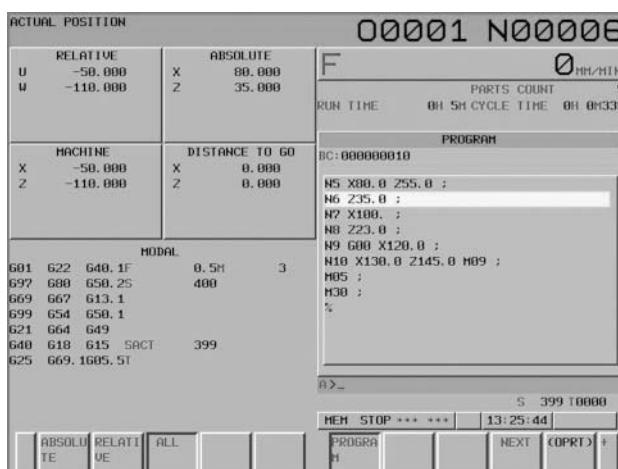
Figure 2-35
Turning Center Program
Check Display
Courtesy FANUC FA America

Part 2 CNC Machine Operation

Control (DNC) allows the program to be executed (drip-fed) from the offline storage location. When using the computer hard drive method, the offline personal computer (PC) is required to have the necessary communications software and an Ethernet configuration on the controller. Older methods still use the RS232 cabling hardware connected to a switch-box interfaced with the PC. Please consult the manufacturer's manual for specific instructions. To execute a DNC operation, follow these steps:

1. Activate the program number to be executed by one of the search methods.
2. Press the RMT button on the *Operator Panel* to set REMOTE execution mode. Note that this button is displayed in Figure 2-2.
3. Press CYCLE START to begin automatic operation.

PROGRAM EDITING FUNCTIONS

Editing part programs includes inserting, deleting, and altering program words. You must understand the techniques for program number searching, sequence number searching, word searching, and address searching before you begin any editing of the program. The control needs to be in the proper mode and the program loaded.

The following steps describe how each is accomplished.

1. Press the RESET key.
2. Press the PROG function button.
3. Press the EDIT operation button to activate the EDIT mode.
4. If the program to be edited is not active, you must load it now. Follow the directions as stated in the section *A Program is Loaded from CNC Memory*.

Setting the Program to the Beginning

By pressing the RESET key while in the EDIT or MEMORY mode, the active program will be returned to the beginning line of the program (program head).

The second method is accomplished by doing the following steps:

1. Press the OPRT soft key.
2. Press the SEARCH up soft key.

Cursor Scanning

The program may be scanned to an editing location by using the cursor and the page keys. Follow the directions as stated in this section under Cursor, Cursor Move, and PAGE UP/DOWN. If the program is very large, using cursor scanning method is not the most efficient method for searching through the program for edit locations. Figure 2-36 displays an example of how the screen might look during the editing process. For instance, the program is displayed with the cursor highlighting the N6 sequence number.

Sequence Number Searching

If the sequence number in the program that requires editing is known, you can search directly to that program location by following these steps:

1. From the EDIT mode, use the alphanumeric keypad to input the sequence number preceded by the letter address N.
2. Press the SEARCH soft key.
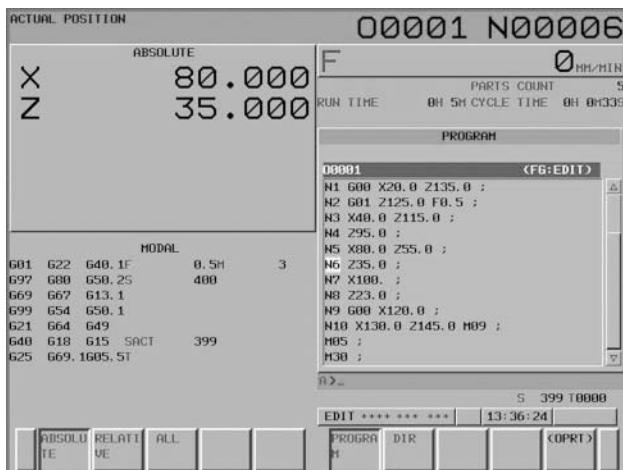
Figure 2-36
Program Scanning for Editing
Courtesy FANUC FA America

3. Input the desired line number. N??? (Sequence number, N6).

4. Press the SEARCH down or up soft key forward or reverse for the direction needed.

The cursor will be moved to the identified sequence number.

Word Searching

Much like sequence number searching, you can search to a specific word in the program. For instance, to search to a specific word in the program like 25.0, follow these steps:

1. From the EDIT mode.

2. Press the SEARCH soft key.

3. Press the program word 25.0.

4. Press the SEARCH down or up soft key forward or reverse for the direction needed.

The cursor will be moved to the identified word 25.0.

Address Searching

As with word or sequence number searching, you can search to a specific address in the program. For instance, to search to a specific address in the program like M06, follow these directions:

1. From the EDIT mode, press the SEARCH soft key.

2. Press the program word M06.

3. Press the SEARCH down or up soft key forward or reverse for the direction needed.

The cursor will move to the word address M06 or the first instance of the M-address that is found. Pressing the direction arrow again will advance to the next instance, if needed.

Part 2 CNC Machine Operation

Inserting a Program Word

From the EDIT mode, use a searching method to scan the program to the word immediately before the word to be inserted. Follow these steps to insert a program word:

1. Scan to the location desired for insertion.
2. Use the alphanumeric keypad to key the address and the data to be inserted.
3. Press the INSERT Edit key.
4. The new data are inserted.

Example

To insert the program word Z.2 on sequence number N4 of the program listed below:

1. Press the EDIT key.
2. Press the PROG function button.
3. Press the SEARCH soft key and then the down SEARCH soft key to move in the forward direction in the program.
4. Key in the word X1.2.
5. Key in the new word to insert Z.2.
6. Press INSERT

       O1234

       N1 G50 S1000

       N2 T0100

       N3 G96 S600 M03

       N4 G00 <u>X1.2</u>

The result will be as follows: N4 G00 X1.2 Z.2

Altering Program Words

From the EDIT mode, use a searching method to scan the program to the word to be altered.

1. Use the alphanumeric keypad to key the new address and the new data to be inserted.
2. Press the ALTER Edit key
3. The new data are changed.

Example

To change the program word, Z.2, in the example to Z.3, follow these steps:

1. Press the EDIT key.
2. Press the PROG soft key.
3. Key in the word Z.2.
4. Press the SEARCH soft key.

5. Enter the program word Z.2

6. Press the SEARCH in the forward direction.

7. Key in the new word, Z.3.

8. Press ALTER.

The result will be as follows: N4 G00 X1.2 Z.3

Deleting a Program Word

From the EDIT mode, use a searching method to scan the program to the word that needs to be deleted. Then press the DELETE Edit key.

To delete the program word, Z.3 from the example, follow these steps:

1. Press the EDIT key.

2. Press the PROG function button.

3. Press the SEARCH soft key.

4. Key in the word Z.3

5. Press the SEARCH in the forward direction.

6. Press DELETE.

SETTING

The SETTING soft key is accessible by first pressing the OFS SET (OFFSET SETTING) function button on the control panel. The PAGE key may be used to view multiple display screens. By pressing this key, access is gained to the dialog where the Parameter setting, sequence number comparison, run time, and parts count displays can be manipulated. By pressing the SETTING soft key, the operator has access to adjust settings to enable or disable parameter writing called Parameter Write Enable (PWE), set automatic insertion of program sequence numbers, change from inch to metric units, and set mirror image requirement.

From the MDI mode:

1. Press the OFS SET function button.

2. Press the SETTING soft key. The SETTING (HANDY) display will appear on the screen.

3. Set values as necessary for the desired results.

Note: Some of the information about the basic parameters of the machine is shown on the screen. To change any of these parameters, perform the following steps:

1. Use the PAGE keys to display the desired screen.

2. Use the arrow keys to position the CURSOR under the parameter that you wish to change.

Enter the desired new value.

3. Zero (0) or 1 is entered, where 1 indicates the ON condition and 0 the OFF condition.

4. Then press INPUT.

Part 2 CNC Machine Operation

Parameter

Some parameters can be accessed via the OFS SET function button and by pressing the SETTING soft key. By pressing the PAGE keys, you can display a set of numerical codes that control certain constants assumed during programming and operation. These codes are numerical values that usually exclude a decimal point. Also, they are sometimes hexadecimal numbers representing an ON/OFF condition for each place in the number having multiple functions. Access to most parameters is not allowed without unprotecting their access. This is done through SETTING, as described above. Consult the manufacturer's manual for specific directions to unprotect parameters.

An example of a parameter setting is the distance a drill will retract during the chip-breaking process that is assigned to canned cycle G83.

Caution: Parameters should only be changed when the results of such change are understood completely. The changes will affect all programs that are executed.

To access the Parameters for adjustments, follow these steps:

1. Enter the MDI mode
2. Press the SYSTEM function button
3. Move the cursor to the desired parameter screen by using the PAGE keys
4. Move the cursor to the desired position to change by using the arrow keys
5. Enter the new value of the desired change
6. Press the soft key INPUT

Diagnosis

By pressing the DIAGNOSIS soft key, the diagnostics screen is displayed. It defines a set of coded digits, which allow a quick determination of the cause of any machine damage or required maintenance. Maintenance personnel use this display screen to obtain needed information.

COMMON OPERATION PROCEDURES

In this book, we want to include explanations concerning situations that may arise during actual machining. We will concentrate on the procedures that should be followed when repetition of particular parts of the program for a specific tool is required. We will also review cases when there is a need to use an EMERGENCY STOP button and recovery from this condition. The following program is used for the case studies.

CNC Turning Center Program

O1234

N1 G50 S2000

N2 T0100

N3 G96 S400 M03

N4 G00 X1.25 Z.2 T0101 M08

N5 ... ...

N6 ... ...

... ...

N17 M01

N18 G50 S1000

N19 T0200

N20 G96 S200 M03

N21 G00 X.75 Z.1 T0202 M08

N22 ... ...

... ...

N39 M01

N40 G50 S2500

N41 T0300

N42 G96 S600 M03

N43 G00 X2.2 Z.05 T0303 M08

N44 ... ...

... ...

N45 M30

Using the program from this example, let us review a procedure you should follow if you need to repeat a part of the program for tools T01, T02, and T03.

Turning Center Case No. 1

Problem

Execution of the program was interrupted in block N17 and you need to repeat from the beginning all operations performed by tool T01.

Solution

1. From the AUTO or EDIT mode, press the RESET button.

This will cause a cancellation of CNC commands under control and return the program to the beginning.

2. From the AUTO mode, press CYCLE START.

Turning Center Case No. 2

Problem

Execution of the program was interrupted in block N39, and you need to execute a program for tool T02.

Solution

1. From the EDIT mode, press the RESET button.

2. Using the alphanumeric keypad on the control panel and the search methods described earlier, search to block N18.

3. Change to the AUTO mode and press CYCLE START.

Note: In both cases, if you intend to execute the program to the end without interruption, the OPTIONAL STOP button must be turned OFF. However, if you intend to execute only part of the program corresponding to work of tool T01 or T02, the procedure is as follows:

1. Press the OPTIONAL STOP button to the ON condition.

   After the work is completed by the desired tool,

2. Press the RESET key while in the AUTO or EDIT mode.

The machine is ready for automatic cycle once again from the program head. The program will stop after reading an M01 code.

Turning Center Case No. 3

Problem

Execution of the whole program is completed but you need to repeat operations performed by tool T03.

Solution

From the EDIT mode:

1. Press the RESET button (if the program is completed and at its head, the RESET is not required).

2. Using alphanumeric keypad on the control panel and the search methods described earlier, search to block N40.

3. From the AUTO mode, press CYCLE START.

Turning Center Case No. 4

Execution of the program is interrupted by the use of the EMERGENCY STOP button. In this case, follow the same procedure as mentioned above. However, you must HOME the machine to reset the machine coordinate system with respect to the X- and Z-axes.

CNC MACHINING CENTER PROGRAM

O2345

N1 G40 G80 G90

N2 G54 G00 X0.0 Y1.5 S1520 M03

N3 G43 Z1.0 M08 H01

N4 ... ...

... ...

N29 G91 G28 Z0.0

N30 M01

N31 T02

N32 M06

N33 G90 G54 G00 X.5 Y1.3 S1500 M03

N34 G43 Z1.0 M08 H02

N35 G81 G98 Z–.47 F6.0 R.1

. . .   .

. . .   .

N38 G91 G28 Z0.0

N39 M01

N40 T03

N41 M06

N42 G90 G54 G00 X–4.125 Y0.0 S2000 M03

N43 G43 Z1.0 M08 H03

N44. . .   . .

. . .   .

N55 G91 G28 Z0.0

N56 T01

N57 M06

N58 G28 X0.0 Y0.0

N59 M30

Machining Center Case No. 1

Problem

Execution of the program was interrupted in block N30, and you need to repeat operations performed by tool T01.

Solution

1. From the AUTO or EDIT mode Press the RESET button.

2. From the AUTO mode Press CYCLE START.

Machining Center Case No. 2

Problem

During the work of tool T02, the tool was damaged. You should change the tool and repeat all operations performed by this tool.

Solution

1. Press the CYCLE STOP (Feed Hold) button.

2. Press RESET to stop spindle rotations and coolant flow.

3. Change to one of the OPERATION MODES, JOG, or MPG.

4. Move the axes to a clearance point from the part.

5. Press the HOME button.

6. Use the axis jog direction keys to HOME the axes.

7. Change the tool and clear the wear offset tool 2.

8. Press the EDIT key.

9. Using alphanumeric keypad on the control panel and the search methods described earlier, search to block N33.

10. From the AUTO mode, press CYCLE START.

Note: In both cases, in order to repeat the work of the remaining tools, the OPTIONAL STOP button should be in the OFF condition.

If you need to repeat the work of only one tool, follow the steps listed below:

OPTIONAL STOP ON

After machining is completed, for tool T01:

1. From the EDIT mode, press the RESET button.

2. HOME the machine with respect to X, Y, and Z axes.

3. From the AUTO mode, press CYCLE START.

After machining is completed for tool T02 or T03:

1. From the EDIT mode, press the RESET button.

2. HOME the machine with respect to X, Y, and Z axes.

3. Using alphanumeric keypad on the control panel and the search methods described earlier, search to block N31 for T02 or N40 for T03.

4. Return to the AUTO Mode and press CYCLE START.

Machining Center Case No. 3

Problem

Execution of the whole program is completed, but you need to repeat the operations performed by tool T03.

Solution

1. Enter the EDIT mode.

2. Using alphanumeric keypad on the control panel and the search methods described earlier, search to block N40.

3. Return to the AUTO mode and press CYCLE START.

Note: Every time that you use the EMERGENCY STOP button, make sure to HOME the machine axes. Then follow the procedures listed above.

Part 2 CNC Machine Operation

In Part 2 we have covered CNC machine operation. Please note that there are a multitude of situations possible and there is not enough space to cover everything here. The intent of this section was to give a basic understanding of the Operation function for turning and machining centers. For complete details on operation features specific to your machine, consult the manufacturer's manuals.

Part 2 Study Questions

1. The counterclockwise direction of rotation is always a negative axis movement when referring to the HANDLE (pulse generator).

> T or F

2. Which display includes the programmed Distance-To-Go readouts?

3. When the machine is ON and the program check screen is displayed, there is a list group of G-Codes displayed. What does this indicate?

4. Describe the difference between the function of the Input and the +Input soft keys.

5. Which button is used to activate automatic operation of a CNC program?

> a. Emergency Stop
>
> b. CYCLE STOP
>
> c. CYCLE START
>
> d. AUTO

6. Which display screen lists the CNC program?

> a. POSITION page
>
> b. OFF SET page
>
> c. PROGRAM CHECK
>
> d. PROGRAM page

7. When the machine is turned on for the first time, it must be sent to its HOME position.

> T or F

Part 2 CNC Machine Operation

8. Which operation selection button allows for the execution of a single CNC command?

        a. DRY RUN

        b. SINGLE BLOCK

        c. BLOCK SKIP

        d. OPTIONAL STOP

9. Which mode switch/button enables the operator to make changes to the program?

        a. EDIT

        b. MDI

        c. AUTO

        d. JOG

10. What does the acronym MDI stand for?

11. Which display screen is used to enter tool information?

12. If the Reset button is pressed during automatic operation, then spindle rotations, feed, and coolant will stop.

        T or F

13. During setup, the mode switch used to allow for manual movement of the machine axes is:

        a. AUTO

        b. MDI

        c. EDIT

        d. JOG

PART 3

Programming CNC Turning Centers

OBJECTIVES:

1. Learn G-Codes associated with turning center programming.
2. Learn M-Codes associated with turning center programming.
3. Apply the proper use of feeds and speeds within turning center programs.
4. Learn how to properly use coordinate systems for programming the turning center.
5. Learn program structure.
6. Learn how to use multiple repetitive cycles.
7. Learn how to use tool nose radius compensation (G40, G41, and G42).
8. Examine several practical examples of turning center programs.

In this section, we will focus our attention on programming the CNC turning center. The programming examples given here will be limited to two-axis lathes. These machines are the base configuration for CNC turning; a solid foundation is laid for further learning about more advanced machinery by gaining full understanding of the techniques presented here. We begin by introducing the program codes in the language commonly called *G-Code*.

PREPARATORY FUNCTIONS (G-CODES)

Preparatory functions are programmed with the letter address G, normally followed by two digits, to establish the mode of operation in which the tool moves. In Chart 3-1 and throughout this text, the G-Codes listed and explained refer to the most commonly used Fanuc system, Type A, and as used on the 0i control. There are some variations in the use of the other two types, B and C, but most of the codes are identical. When system programming consult the programming manual for the specific control before selecting the type.

More detailed descriptions and application examples are given in the section *Overview of Preparatory Functions for CNC Turning Centers*.

MISCELLANEOUS FUNCTIONS (M-CODES)

Miscellaneous functions are used to command various operations. Two commonly used M-Codes—M03 and M08—are used for starting spindle rotation in the clockwise direction and activating coolant flow respectively. The code consists of the letter M typically followed by two digits. Normally, one block will contain only one M-Code function; however, up to three M-Codes may be in a block depending upon parameter settings. Most of the common M-Codes are listed in Chart 3-2, Miscellaneous Functions (M-Codes) Specific to Turning Centers. Many machine tool builders assign other codes for specific purposes relative to their equipment. Always consult the manufacturer manuals specific to the machine in use for pertinent M-Codes.

Part 3 Programming CNC Turning Centers

	Chart 3-1	Preparatory Functions (G-Codes) Specific To Turning Centers

Code	Group	Function
G00	01	Rapid Traverse Positioning
G01	01	Linear Interpolation
G02	01	Circular and Helical Interpolation CW (clockwise)
G03	01	Circular and Helical Interpolation CCW (counterclockwise)
G04	00	Dwell
G09	00	Exact Stop
G10	00	Programmable Data Setting
G11	00	Programmable Data Setting Cancellation
G20	06	Input in Inches
G21	06	Input in Millimeters
*G22	09	Stored Stroke Limit ON
G23	09	Stored Stroke Limit OFF
G25	08	Spindle Speed Fluctuation Detection ON
G26	08	Spindle Speed Fluctuation Detection OFF
G27	00	Reference Point Return Check
G28	00	Reference Point Return
G29	00	Return From Reference Point
G30	00	Return to Second, Third, and Fourth Reference Point
G32	01	Thread Cutting
*G40	07	Tool Nose Radius Compensation Cancel
G41	07	Tool Nose Radius Compensation, Left Side
G42	07	Tool Nose Radius Compensation, Right Side
G50	00	Coordinate System Setting/ Maximum Spindle Speed Setting
G52	00	Local Coordinate System Setting
G53	00	Machine Coordinate System Setting
G54-59	14	Work Coordinate System Selection
G68	04	Mirror Image for Double Turrets ON
*G69	04	Mirror Image for Double Turrets OFF
G70	00	Finishing Cycle
G71	00	Stock Removal in Turning
G72	00	Stock Removal in Facing
G73	00	Pattern Repeating
G74	00	Peck Drilling Cycle
G75	00	Groove Cutting Cycle
G76	00	Multiple Thread Cutting Cycle
*G80	10	Canned Drilling Cycle Cancellation
G83	10	Face Drilling Cycle
G84	10	Face Tapping Cycle
G86	10	Face Boring Cycle
G90	01	Outer/Inner Diameter Turning Cycle
G92	01	Thread Cutting Cycle/Maximum Spindle Speed Setting (SYS B,C)
G94	01	Face Cutting Cycle
G96	02	Constant Surface Speed Control
*G97	02	Constant Surface Speed Control Cancellation
G98	05	Feed per Minute
*G99	05	Feed per Revolution

NOTES:

1. *In the table, G-Codes marked with an asterisk (*) are Active upon startup of the machine.*

2. *At machine startup or after pressing reset, the inch (G20) or metric (G21) measuring system last active remains in effect.*

3. *G-Codes of group 00 represent "one shot" G-Codes, and they are effective only to the designated blocks.*

4. *Modal G-Codes remain in effect until they are replaced by another command from the same group.*

5. *If modal G-Codes from the same group are specified in the same block, the last one listed is in effect.*

6. *Modal G-Codes of different groups can be specified in the same block.*

7. *If a G-Code from group 01 is specified within a canned drilling cycle block, the cycle will be cancelled just as if a G80 canned cycle cancellation were called.*

More detailed descriptions and application examples are given later in the section Overview of Preparatory Functions for CNC Turning Centers.

Chart 3-2	Miscellaneous Functions (M-Codes) Specific To Turning Centers
M-Code	**Function**
M00	Program Stop
M01	Optional Stop
M02	Program End Without Rewind
M03	Spindle ON Clockwise (CW) Rotation
M04	Spindle ON Counterclockwise (CCW) Rotation
M05	Spindle OFF Rotation Stop
M08	Flood Coolant ON
M09	Coolant OFF
M10	Chuck Close
M11	Chuck Open
M12	Tailstock Quill Advance
M13	Tailstock Quill Retract
M17	Rotation of Tool Turret Forward
M18	Rotation of Tool Turret Backward
M18	Spindle Orient Cancel
M19	Spindle Orient
M21	Tailstock Direction Forward
M22	Tailstock Direction Backward
M23	Threading Finishing with Chamfering
M24	Threading Finishing with Right-Angle
M30	Program End With Rewind
M41	Spindle LOW Gear Range Command
M42	Spindle HIGH Gear Range Command
M71	Bar Feed ON – Start
M72	Bar Feed OFF – Stop
M73	Parts Catcher Advance
M74	Parts Catcher Retract
M76	Parts Counter
M98	Subroutine Call
M99	Return to Main Program From Subroutine

The following descriptions are given for many of the M-Codes in the chart. Also observe their use within each of the example programs. Please consult the appropriate operator and programming manual for machine-specific information about M-Codes for your application.

PROGRAM STOP (M00)

When the M00 code is encountered, the spindle r/min, feeds, and coolant flow stop. This function interrupts the automatic work cycle in order to allow the following:

1. In-process inspection and gauging
2. Visual inspection of tool wear and other components
3. Removal of chips
4. Interruption of the cycle, in order to relocate the workpiece when the workpiece is being machined from both sides during one operation

The function of M00 within the control system is accompanied by the following events:

1. Spindle revolutions are stopped.
2. Flood coolant flow is deactivated.
3. All feed movement is stopped.
4. The CYCLE START light will still be on.

After use of this code, the program block following must include either M03 or M04 as well as M08 to reactivate these functions. Functions G96, G97, S, and F are NOT canceled by M00.

OPTIONAL PROGRAM STOP (M01)

This function is nearly the same as M00 with one significant difference. It is applied only by pressing or switching the OPTIONAL STOP button or toggle switch ON, for example, to stop the machine so that measurements can be taken or to remove chips at the discretion of the machine operator. If the OPTIONAL STOP button is not active, the machine will ignore this command even if it is present in the program.

PROGRAM END (M02)

Function M02 cancels the automatic work cycle, interrupts revolutions, stops feed rate and coolant flow, and cancels the control system of the CNC. The CYCLE START light goes off in some types of controls and the PROGRAM END light comes on. Repetition of a programmed operation is not possible without RESET of the control. To repeat the program, the M30 function is used. The M02 command is used primarily on NC tape machines.

SPINDLE ON (CLOCKWISE) (M03)

This function signals the machine to activate the spindle with clockwise revolutions at a value indicated by the S function. M03 is cancelled by M04, M05, M00, M01, M02, and M30.

SPINDLE ON (COUNTERCLOCKWISE) (M04)

To activate or cancel this function, follow the same instructions as for M03.

Spindle Off (M05)

This function is cancelled by M03 and M04.

Flood Coolant On (M08)

Function M08 activates the flood coolant flow.

Coolant Off (M09)

Function M09 deactivates the coolant flow.

Chuck Close and Open (M10 and M11)

M10 automatically closes the spindle chuck jaws and M11 automatically opens the spindle chuck jaws. A toggle switch on the operator panel determines whether clamping is internal or external. M10 and M11 are used in certain cases when there is a special part puller or gripper used for the insertion and removal of the workpiece from the spindle chuck. Such devices are used in automated operations where mass production is the primary focus.

Tailstock Quill Advance and Retract (M12 and M13)

During the process of turning long shafts, a tailstock is often used to support the shaft. A center drilling operation must be programmed first. By applying functions M12 and M13, the workpiece can be supported with a center in the tailstock; the workpiece can then be turned. When the operation is completed, the tailstock is returned to its original position by the M13 command. All operations are performed automatically, without interruption. If the time required to extend the tail spindle is noticeably long after function M12, apply a dwell function G04.

Rotation of the Tool Turret Forward and Reverse
(M17 and M18)

Miscellaneous function M17 rotates the tool turret in the normal (clockwise) direction whereas M18 rotates the tool turret in the opposite direction (counterclockwise). These functions may be used only for some types of machines. Function M17 is valid at machine start-up. Function M18 implies a change in direction of the rotation opposite to the one set previously. These commands may be useful when special tooling is mounted in the tool turret or chuck where clearance issues must be considered during turret rotation.

Tailstock Direction Forward and Reverse (M21 and M22)

Programmable shifting of the tailstock as a whole is an added function on some types of machines, especially if the extended length of the tailstock spindle is not sufficient to perform the operation and/or the lathe is large and has a long Z-axis stroke. These codes are a factory option only.

Thread Finishing (M23 and M24)

Miscellaneous function M23 should be applied to the G92 Thread Cutting cycle when the threading tool usually exits at a 45° angle. M24 is necessary when the ending

thread is followed by a greater diameter or a recess groove. M23 is the default state after machine power is turned on. These functions are used in conjunction with the G92 Tread Cutting Cycle at a thread end for either a 90° or a 45° retract respectively.

END OF PROGRAM (M30)

This function is similar to M02. However, it returns to the beginning of the program. In the case of older NC machines, it activates the rewinding of the punched tape.

TOOL FUNCTION

Tool pockets in turrets on CNC turning centers are assigned coded numbers. The coding system is fixed and cannot be erased even if the machine is turned off. In the process of programming, tool function is commanded by the four digits that follow the letter address T. The first two digits are related to the tool number and its corresponding geometry offset. The remaining two digits signify the tools wear offset number, as illustrated below. For example T0101:

Tool pocket number Geometry offset	Tool offset number Wear offset
T01	01

Tool numbers may vary from 1 to the maximum number of pockets in the tool turret, for example, from 1 to 8, etc. When there are two tool turrets, the numbers assigned to turret No. 2 are sequenced consecutively from the No. 1 turret. For example, if turret No. 1 has 8 tools, then the first pocket in turret No. 2 would be 9. The number of the tool and wear offset available may vary; depending on the specific control, numbers 1 through 32 are common. This means that each tool number may be assigned 1 out of the 32 offset numbers. If tool offset number 01 is assigned to tool 01, this rules out the possibility of assigning the same tool offset number to another tool.

The same is true for wear offsets.

Example

Recommended	NOT Recommended
T0101	T0501
T0202	T0602

Usually, tool offset 01 is assigned to tool number 01. This convention simplifies operating procedures for the operator.

PRACTICAL APPLICATION OF TOOL WEAR OFFSET

Tool wear compensation is a procedure aimed at correcting dimensional variations along the X or Z axes caused by tool wear or deflection.

Please note that throughout the figures where a tool is depicted in relationship to the workpiece, the tool will appear above the part, as is common in practice on rear turret, slant-bed style CNC turning centers. In reality, the tool will be a right-hand style and the insert will be facing downward. The graphical representation here shows the insert facing upward.
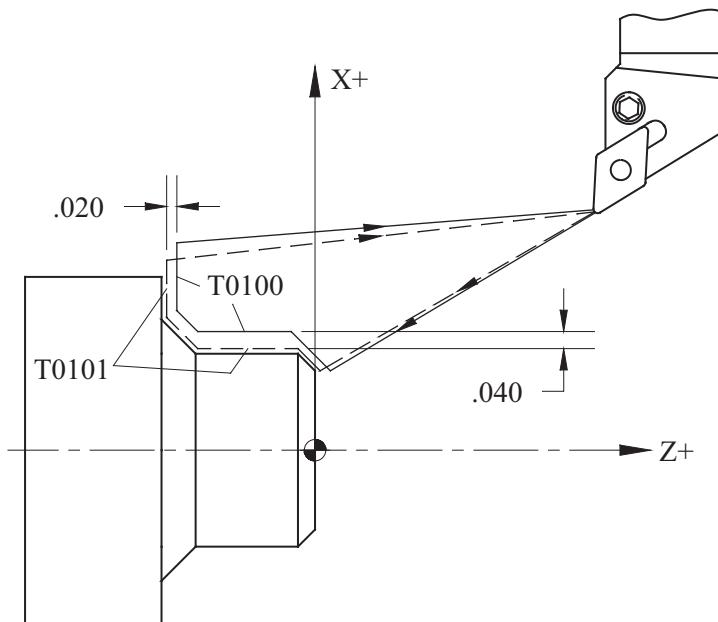
Figure 3-1
Tool Wear Compensation
by Offset

Offset is 0.040 on X axis and 0.020 on Z axis

Tool geometry offset numbers refer to certain values of X and Z, and their relationship to the Workpiece Zero and Machine Home. These geometry offsets should not be used to make adjustments for wear. Only wear offsets are used for this purpose. Small correctional values are input to a particular tool wear offset number for X or Z to compensate for any variations. In Figure 3-1, the dashed line refers to the cutting tool path, with the X and Z values for tool wear offset as follows:

$$X = 0; Z = 0$$

The solid line refers to the cutting tools path, with X and Z values for tool wear offset as follows:

$$X = -.080; Z = -.020$$

In Figure 3-1 the value of the tool offset is twice that of the dimension because X-axis dimensions in CNC programming refer to diameters on the workpiece. In most cases, both the tool function and the tool offset appear in the initial phase of the program for each individual tool. Figure 3-1 demonstrates that if tool function T0101 is used, the programmed tool path is changed by a displacement equal to the value of input to the wear offset.

Command T0100 initiates the cancellation of the tool wear offset. If the T0100 command is omitted—thereby, canceling the tool offset—the tool will return to the starting point with a displacement equivalent to the value of the tool wear offset. In this case, the displacement is indicated by X = –0.080 and Z = –0.020, shown on the drawing by the solid line. If T0100 offset cancelling is not performed the tool path for each subsequent

tool will be altered by the offset amount equal to this value. This type of programming error increases the chances for over-cutting workpieces. Changing the tool wear offset number does not require cancelling the previous tool offset number.

FEED FUNCTION

Feed function determines the amount of feed rate of the cutting tools in the machining process. Feed is programmed using the letter address "F", followed by up to four digits in the metric system and five digits in the inch system. These digits represent certain values of feed. The following examples are two methods of designating feed rate:

1. Inches per revolutions (in/rev), formerly known as IPR, or millimeters per revolution, mm/rev, of the spindle (G99). In this case, in order to obtain feed with a certain assigned value of speed, the spindle as well as the workpiece must be rotating.

Examples

> F1.1205 in/rev.
>
> F0.05   in/rev.
>
> F0.001   in/rev.

When the spindle speed is changed after the constant surface speed per revolution (G96) has been called, the feed rate will change for a certain period of time. Therefore, feed is directly coupled with spindle speed.

The following notes apply to the feed function, G99 … F.

a. The values entered into the program for feed remain active until replaced by another feed rate, or cancelled by the G00 rapid traverse call.

b. The input value of speed is equivalent to the actual speed if the feed rate override on the operation panel is set to 100%. See Part 2, *CNC Machine Operation*, for a detailed description of Feed Rate Override.

2. Feed rate per time is measured (programmed) in inches per minute, in/min (formerly known as IPM), or millimeters per minute, mm/min (G98). If the feed function in the program contains feed rate per time period, then any change in the spindle speed has no effect on the feed rate because the feed rate and the spindle speed are not coupled.

Examples

> F121.15 in/min
>
> F1.05   in/min
>
> F0.5   in/min

Generally, all CNC lathes are set to a default of feed per revolution of the spindle at machine start-up. In order to establish feed rate per minute (in/min or mm/min), the G98 function must be used. This function remains effective until cancelled by function G99 or until the machine is turned off.

The following notes apply to feed function G98:

a. The values entered into the program for feed remain active until replaced by another feed rate, or cancelled by the G00 rapid traverse call.

b. The input value of speed is equivalent to the actual speed if the feed rate override on the control panel is set to 100%. See Part 2, *CNC Machine Operation*, for a detailed description on Feed Rate Override.

c. Feed functions containing feed in inches per minute are not applicable to threading cycles.

Examples of feed functions include:

    F0.02   in/rev.
    F0.004   in/rev.
    F0.035   in/rev.

    G98
    F2.0   in/min
    F0.5   in/min

    G99
    F0.012   in/rev.
    F0.008   in/rev.

SPINDLE FUNCTION

Spindle function is commanded by the letter address S, followed by a number up to four digits, as shown below. The spindle rotation direction clockwise (M03) or counter-clockwise (M04) is typically in the same block.

    S50, S150, S3000

Some machines are assigned two ranges of speeds: low or high. Others may have three or more ranges. Depending on the given value of the rotational speed, the machine automatically adjusts to the appropriate range, as seen in the following rotational speeds. In practice, most manufacturers overlap the low and high range, for example:

    30–1200 (rev/min) low range (M41)
    80–4000 (rev/min) high range (M42)

For turning centers, two functions are applicable to the control of spindle speed. These are:

    G96:   Constant surface speed control
    G97:   Constant surface speed control cancellation (sometimes referred to as constant spindle speed)

Both functions appear together with function S, for example:

G97 S500 = 500 revolutions per minute (r/min)
G96 S400 = 400 Surface Speed (*V*) in ft/min (or m/min)

Constant spindle speed (G97) is applied in the case of threading cycles and in the machining of a workpiece, with the diameter remaining constant. It is also used for all operations on the centerline, like drilling, etc. If the situation calls for several changes of spindle speed in a given program, new values for the S function are assigned.

Example

G97S1000 is active for diameter one and sets the constant spindle speed.

S800 changes the r/min for diameter two.

S300 changes the r/min for diameter three.

CONSTANT SURFACE SPEED CONTROL (G96)

To further examine this concept, study the diagram of peripheral speed distribution in Figure 3-2. The diagram appears during a facing cut, using function G97 S1000. The following formulas calculate peripheral surface speed for each diameter of 1.00, 2.00, and 3.00 inches. The result is not desirable because of a decrease in surface speed as the diameter gets smaller.

$$V_1 = \frac{\pi \times D \times n}{12} = \frac{3.14 \times 3.0 \times 1000}{12} = 785 \text{ (SFPM)}$$

$$V_2 = \frac{\pi \times D \times n}{12} = \frac{3.14 \times 2.0 \times 1000}{12} = 523 \text{ (SFPM)}$$

$$V_3 = \frac{\pi \times D \times n}{12} = \frac{3.14 \times 1.0 \times 1000}{12} = 261 \text{ (SFPM)}$$

where:

n = RPM or r/min

D = diameter

π = 3.14

V = Cutting speed (FPM)

The diagram shows that surface speed decreases if G97 is used—as a diameter decreases and reaches zero at the centerline of the part. But is this phenomenon of any advantage to us in the process of facing? Before answering this question, look at the advice of cutting tool manufacturers who recommend specific cutting speeds for different types of machined materials. In the case of function G97, such a condition will be fulfilled only with respect to one diameter. As mentioned previously, Constant Surface Speed control (G96) is one of the factors that can be included in programs for CNC turning centers.
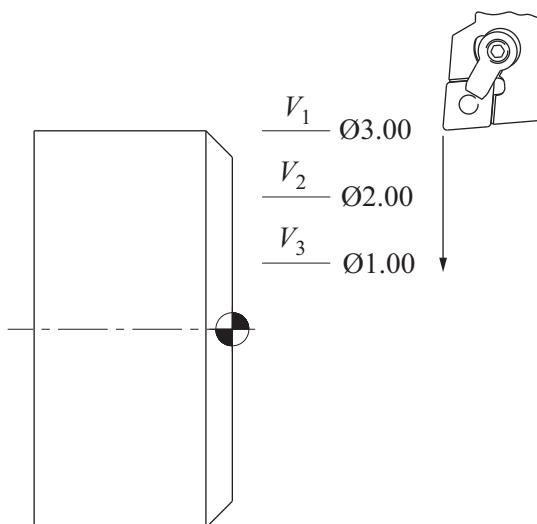
Figure 3-2
Constant Surface Speed Control (G96)

If the surface speed must remain constant, then the spindle speed has to increase with a decreasing diameter. Spindle speed for each consecutive diameter is calculated by the control, according to the formula $n = (12 \times V)/(\pi \cdot D)$. Thus, a closer look at this formula leads to the conclusion that, theoretically, as the diameter decreases to zero, the spindle speed increases to infinity. In reality, the spindle speed range is limited by the maximum r/min (RPM) capacity of the machine.

In practical terms then, function G96 is very useful during facing and also in all cutting that involves a change in the diameter of the workpiece. As the diameter of the machined workpiece decreases, spindle speed increases; conversely, as the diameter increases, the spindle speed decreases.

MAXIMUM SPINDLE SPEED SETTING (G50)

When the letter address S, is given within a program block and is preceded with function G50, it refers to the maximum spindle speed setting that can be applied in the current operation for a given tool. As mentioned above, the spindle speed is calculated according to technological metal-cutting conditions for a given tool, or for any particular material. In some cases, the workpiece holding arrangement may require special equipment, which is mounted onto the conventional holding equipment. Such workpiece holding equipment creates conditions that do not permit utilization of the full range of spindle speeds, especially maximum spindle speed for a given machine. Because of this fact, a maximum spindle speed for a particular operation can be assigned by using the function G50. Therefore, if, in the machining process, metal-cutting conditions arise that require a higher spindle speed, an increase of the spindle speed will not take place. Using function G50 is called "clamping the spindle speed" at a safe maximum r/min.

Note: If G50 is not used in conjunction with the spindle speed command (such as G50 S1250) when G96 is commanded, then the machine will increase the r/min as the diameter decreases, up to the machine's maximum capability. This condition could result in an excessive r/min and damage could occur.

COORDINATE SYSTEMS FOR PROGRAMMING OF CNC TURNING CENTERS

PROGRAMMING IN THE ABSOLUTE COORDINATE SYSTEM

When programming in an absolute coordinate system, input coordinates of programmed points always refer to a fixed zero coordinate point. The actual coordinates of traverse for the tool tip from point 0 to 10 are shown in Figure 3-3.

In order to simplify programming as well as program readout, values of X are equivalent to each particular diameter of the workpiece.

0. X0.0 Z0.0

1. X1.0 Z0.0

2. X1.0 Z–1.15

3. X2.0 Z–1.15

4. X2.2 Z–1.25

5. X2.2 Z–2.25

6. X3.04 Z–2.25

7. X3.3 Z–2.38
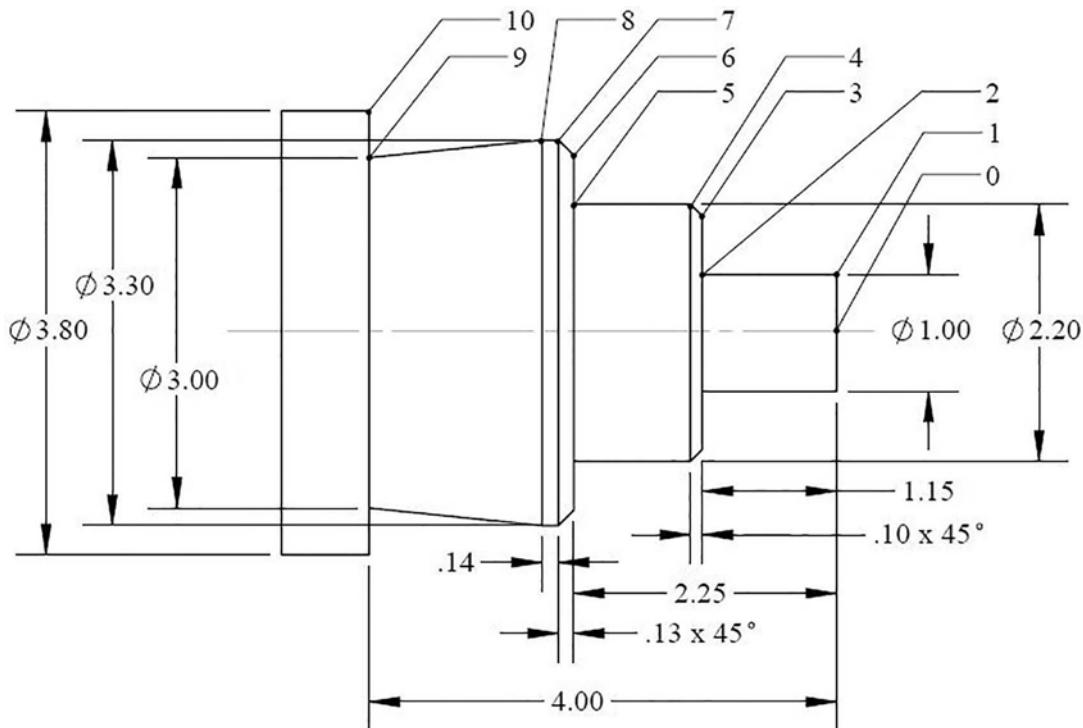
Figure 3-3
Coordinate Systems for Turning Centers

8. X3.3 Z–2.52

9. X3.0 Z–4.0

10. X3.8 Z–4.0

PROGRAMMING IN THE INCREMENTAL COORDINATE SYSTEM

When using an incremental system, the path of the tool from one position to the next is given in the direction of each axis. Using letter address U and W, the point displacements may be input in the direction of the X and Z axes respectively. The sign in front of the value determines the direction of movement. Values of U refer to motion in the X-axis direction and refer to changes in the diameter of the workpiece. This next example illustrates the tool movement as programmed using the incremental coordinate system (refer to the same drawing, Figure 3-3).

0. U0.0 W0.0

1. U1.0 W0.0

2. U0.0 W–1.15

3. U1.0 W0.0

4. U0.2 W–0.1

5. U0.0 W–1.0

6. U0.84 W0.0

7. U0.26 W–0.13

8. U0.0 W–0.14

9. U–0.3 W–1.48

10. U0.8 W0.0

If necessary, combinations of both systems (absolute and incremental) can be used. The CNC control registers the position of the tool, regardless of whichever method of programming is being used.

0. X0.0 Z0.0

1. X1.0

2. Z–1.15

3. X2.0

4. X2.2 W–0.1

5. Z–2.25

6. X3.04

7. X3.3 W–0.13

8. W–0.14

9. X3.0 Z–4.0

10. X3.8

If the value of one of the coordinates remains the same, then input only the value of the next consecutive changing coordinate.

COORDINATE SYSTEM SETTING (G50)

With machine tools of today, the (G50) setting method is seldom used. Because of the potential problems of incorrect entry of dimensional data into the actual program, the method is prone to mistakes. The more modern technique uses the geometry offset register to store the dimensional data for each tool and to use Work Coordinate System Setting (G54), which is covered next. Consult the manufacturer and operation manuals for specific instructions if needed.

WORK COORDINATE SYSTEM SETTING (G54)

With modern CNC machines, a much more reliable method of setting the coordinate system is used. Many machines today have a tool measurement sensor that is used as a fixed reference position. Each tool is measured to the electronic sensor and the coordinate values are input directly into the offset register specific to that tool. A separate measurement is taken for the X- and Z-axes. A parameter in the machine control is used to register the location of the tool sensor so that the X-axis values are related back to the centerline of the spindle and the Z-axis values to a fixed position. The setup person then merely identifies and measures the finished face of the part along the Z-axis in order to establish the work coordinate system and input the position in the WORK COORDINATES display for the G54 register. Consult the appropriate operation manual for your machine to find specific details for setting methods.

PROGRAM STRUCTURE FOR TURNING CENTERS

In order to gain a better understanding of the basic structure in CNC turning center programming, the following sample program is given.

Sample Program

O1234	**Program Number**
N10 G50 S1000	
N15 T0101	
N20 G00 G54 X4.3 Z.5 M08	
N25 M42	
N30 G96 S500 M03	
N35 Z.1 M08	
. . .	**Program for Tool No. 1**
. . .	
N75 G00 X4.3 Z.1 T0100 M09	
N80 G28 U0.0 W0.0	
N85 M01	
N90 T0202	
N95 M41	
N100 G96 S600 M03	
N105 G00 G54 X4.3 Z.1 M08	**Program for Tool No. 2**

. . .

. . .

N130 G00 X4.3 Z.1 T0200 M09

N135 G28 U0.0 W0.0

N31 M30

%

PROGRAM NUMBER

The *program number* typically consists of a four-digit integer following the letter O. This number is used to identify the programming procedures. The range of numbers that can be used as program numbers is 0001 to 9999. Be careful not to mistake the letter O, which precedes the actual program number, for a zero. The 9000 series of program numbers are reserved for macro programs and may not be used for main or subprogram naming.

Example of program numbers

O0001

O2160

O4004

O1261

BLOCK COMPOSITION

A *block* is basically one line of the part program that contains the commands used to simultaneously execute operations on CNC machines. The block is composed of program words and always ends with a semicolon known in CNC programming as the *end of block* (EOB) character. The semicolon is never part of the written or disk copy of the program. Refer to Part 1, for additional information on the block format.

Examples of program blocks:

N10 G50 S1000

N15 G00 G54 X2.125 Z.1 M08

N20 M42

N55 M30

Arrangement of the program words in a given block can be random, but the address N in the sequence number must be first. For the sake of consistency, follow the order presented in this text.

BLOCK NUMBER

The *block number* (also called the sequence number) is defined by the letter N, followed by one to four or five digits, and is limited to these four or five digits. A block number provides easier access to information contained in the program. The arrangement of block numbers in a given program can be random, but typically is sequenced in increments of some amount other than one. The most common step increments are five or ten.

Examples of block or sequence numbers

    N1000

    N10

    N15

    N20

Block or sequence numbers can be omitted in a block, or in the program itself. This saves storage space in the controller memory. It is important to note, however, that a program searching technique requires a sequence number in order to restart a program from somewhere other than its beginning or head. The logical location for sequence numbers is at tool changes, enabling the restart of that tool. It is also possible to search for a specific program word, such as T0202.

PART PROGRAM

The part program is that section of the program that contains essential information needed to control movement of the cutting tool and activation of auxiliary equipment.

SUBPROGRAM

The subprogram is a subordinate program. It is also registered in the controller memory with the letter O, followed by a four- or five-digit number, just as the main program is. In the main program, a subprogram is called by using the M98P…. function. M99 (called for in the subprogram) is then the function that ends a subprogram and returns to the main program.

Example subprogram

    O1234

    N5   . . .   .

    . . .   .

    N20   M99

The subprogram is called with function M98 in the main program, while the number of the subprogram is called with the letter address P.

Example of a subprogram call

    M98 P1300

In the following example, the program number for the subprogram is O1300.

Example of main and subprogram structure

Main Program	Subprogram	Subprogram
O1200	O1300	O1400
N5 G54 X5.0 Z.5	N10 G00 X2.0	N10 G01 Z.2
N10	N20	N20 . . . . . .
N20	N30	N30 . . . . . .

N30 M98 P1300	**N40 M98 P1400**	**N40 . . . .**
N40	**N50 . . . . . .**	**N50 M99**
N50	**N60 M99**	
N60		
N70 M30		

Consecutive subroutines may be called from a current subprogram by applying the above-mentioned methods. The number of subprograms linked together to form a program may be as high as 9999. In the program block N30 of the above example, execution of the program begins with subprograms O1300 and then O1400. After each of these subprograms is completed, further execution of the program begins with the next consecutive block in the previous program. This is the block (N40) following the block that called the subprogram.

In some cases, address L may be added to the program block; it indicates how often the subprogram is to be repeated. The amount of repeats for L can range from 1 to 999. If the address L is omitted from the program, the subprogram is executed only once.

N30 M98 P1300 L2

SAMPLE PROGRAM

Sample Program O0001

For this first example, only the finished cutting profile pass (.005 stock removal) is programmed for the part geometry. Compensation for the tool nose radius will not be accounted for in the chamfered sections even if it is set in the Tool Data register (R) of the control. Tool Nose Radius Compensation, G41 and G42, will be covered later in Part 3. It is possible to manually calculate the start and end values of the chamfers using trigonometry and input the values directly in the program. However, this method is no longer widely used.

Note: If this part were to be machined from solid 5.0" bar stock, a rough turning path must be machined first. This method will be explained later in the section on Multiple Repetitive Cycles.

> **O0001**
> **N10 G50 S1800**
> **N15 T0200 M42**
> **N20 G00 G54 X5.2 Z1.0 T0202**
> **N25 G96 S600 M03**
> **N30 X0.0 Z.1 M08**
> **N35 G01 Z0.0 F.02**
> **N40 X1.75**
> **N45 X2.0 Z–.125 F.004**
> **N50 Z–1.5 F.008**

       N55 X3.188

       N60 X3.5 W–.156 F.004

       N65 Z–3.75 F.008

       N70 U.25 Z–3.75 F.004

       N75 X5.0 F.008

       N80 X5.1 F.02

       N85 G00 Z.1 M09

       N90 T0200

       N95 G28 U0.0 W0.0 M05

       N100 M30

The following explanations are given for the individual parts of Sample Program 1:

O0001 is the program number.

N10 through N100 are sequence or block numbers.

N10 G50 S1800

Block N10 contains the program words G50 S1800. Because G50 is specified simultaneously in the block with the S1800, it sets a maximum spindle speed of 1800 r/min applicable during machining for tool No. 2, called by T0200 in the following block.

N15 T0200 M42

Tool function T0200 commands that tool No. 1 is in the work position. If the turret is in another position, it will rotate tool No. 1 into the ready position. The number 00 cancels any wear offset so that no offset compensation is used for this tool, at this time. Miscellaneous function M42 provides the machine with the information that the highest spindle speed range is applicable to this tool.

N20 G00 G54 X5.2 Z1.0 T0202

Block N20 activates the rapid traverse (G00) tool movement to the X and Z coordinates listed in relation to the G54 work offset and T0202 tool offset in the controller. Typically, this first move positions the tool near the work diameter and the part face by a safe distance.

N25 G96 S600 M03

Block N25 contains function G96 commands that the machining process will take place at a constant cutting speed of 600 ft/min and the spindle speed will be adjusted automatically, based on the diameter of the workpiece up to a maximum of 1800 r/min. Miscellaneous code M03 refers to the direction of spindle rotation and activation of the spindle, which in this case is clockwise. Please note the spindle direction in this example is pertinent to the tool orientation in the drawing provided in Figure 3-4. With most modern slant bed style CNC turning centers, the tool is mounted with the insert facing downward, which requires clockwise rotation (M03) for cutting.
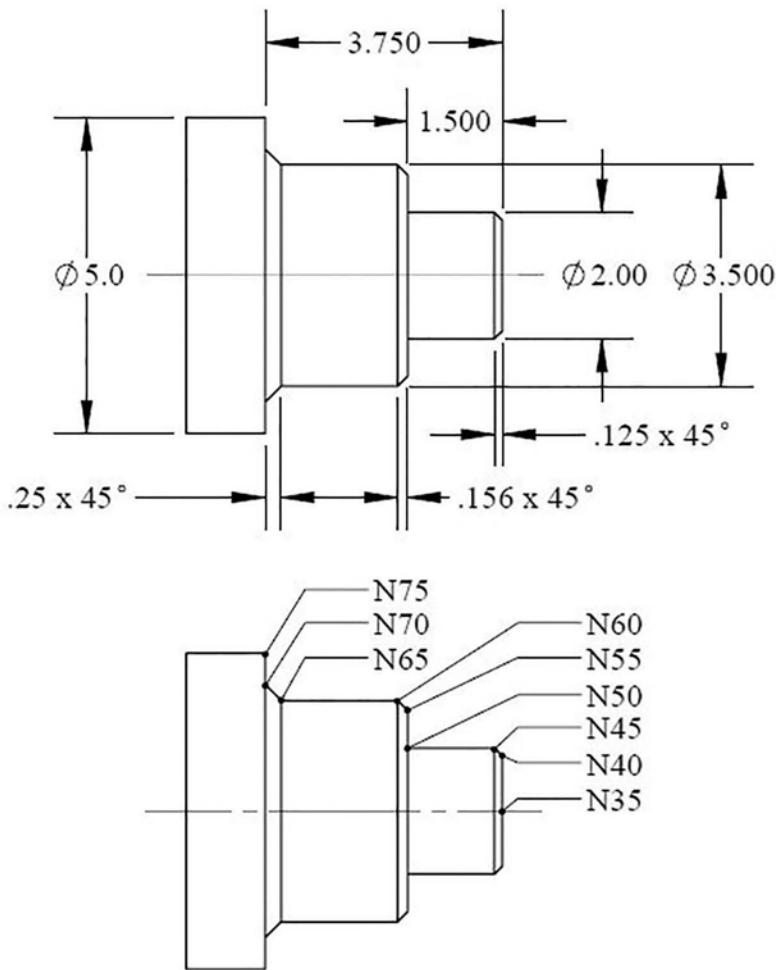
Figure 3-4
Drawing for Sample Program O0001

N30 X0.0 Z.1 M08

The information contained in this block N30 activates the execution of the commands related to tool motion and the flood coolant system, etc. First, the tool turret and carrier advances to a position specified by the coordinates X0.0 and Z.1 at rapid traverse, while simultaneously, the flood coolant system is activated.

N35 G01 Z0.0 F.02

Function G01 in block N35 refers to the linear interpolation with respect to the Z-axis, with the ending coordinate of Z = 0.0 and a value of feed .02 in/rev.

N40 X1.75

N45 X2.0 Z–.125 F.004

N50 Z–1.5 F.008

N55 X3.188

N60 X3.5 W–.156 F.004

N65 Z–3.75 F.008

N70 U.25 Z–3.75 F.004

N75 X5.0 F.020

The information found in blocks N40 to N75 refer to a change of position in the programmed points of the workpiece coordinate system that machines the part profile, including feed-rate changes in specific blocks.

N80 X5.1 F.02

In block N80, the tool feeds along the X-axis to clear the diameter of the part by .100".

N85 G00 Z.1 M09

In this block, the tool will advance at rapid traverse to Z.1 and the flood coolant system will be turned off.

N90 T0200

This block is used to cancel the active Tool Offset prior to using the G28 command in the next block.

N95 G28 U0.0 W0.0 M05

Block N95 commands the tool to return to the Reference Point identified by the G28 function and the U and W coordinate. Miscellaneous code M05 stops rotation of the spindle.

N100 M30

Miscellaneous function M30 in block N100 ends the program and resets the program to its beginning point or head.

PREPARATORY FUNCTIONS FOR TURNING CENTERS (G-CODES)

RAPID TRAVERSE POSITIONING (G00)

G00 is *Rapid Traverse Positioning*; it is used for position changes without machining. Function G00 can be combined with the M, S, or T functions. It is a modal G-Code and will remain in effect until replaced by another command from the same group.

For this example, shown in Figure 3-5, only the finished profile cutting pass is programmed for the part geometry. The following program is an example of circular interpolation utilizing I and K.
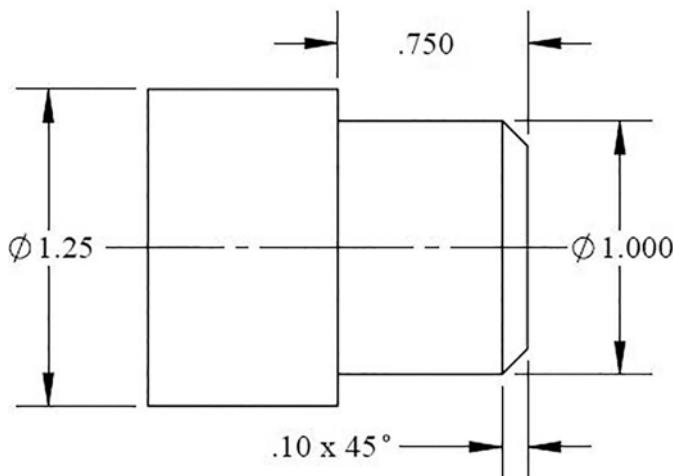
Figure 3-5
Drawing for Rapid Traverse
Function (G00)

Important Note 1: Compensation for the tool nose radius will not be accounted for in the chamfered or arc sections even if it is set in the Tool Data register (R) of the control. Tool Nose Radius Compensation, G41 and G42, will be covered later in this chapter. It is possible to manually calculate the start and end values of the chamfers using trigonometry and input the values directly in the program. However, this method is no longer widely used.

Important Note 2: If this part were to be machined from solid bar stock, a rough turning path must be machined first. This method will be explained in Multiple Repetitive Cycles later in this chapter.

See Part 2 of this text for an explanation of proper Tool Data entry.

Sample Program O0002: Using the Rapid Traverse Function (G00)

O0002

N10 G50 S2000

N15 T0200 M42

N20 G00 G54 X1.35 Z1.0 T0202

N25 G96 S500 M03

N30 X0.0 Z.1 M08

N35 G01 Z0.0 F.02

N40 X.8 F.01

N45 X1.0 Z–.1

N50 Z–.75

N55 X1.25

N60 X1.35 M09

N65 G00 Z1.0 M09

N70 T0200

N75 G28 U0.0 W0.0 M05

N80 M30

Caution: If G00 rapid positioning is programmed in the direction of both axes, note that the tool will not advance to a specified point following the shortest possible path. The tool path is determined by the speed of rapid traverse with respect to each axis. In most cases, the speed for the X-axis is much greater than that for the Z-axis. The axis that must travel the least distance will be reached first.

Figure 3-6 compares rapid and feed traverse movements when the axes are commanded simultaneously. The rapid traverse moves are indicated by dashed lines and feed traverse by solid lines.

N20 = a rapid traverse move toward the material (workpiece)

N35 = a linear feed traverse move (G01) to the zero face of the workpiece

N40 = a linear feed traverse move (G01) to the start of the chamfer

N45 = a linear feed traverse move (G01) to cut the .10 x 45° chamfer

N50 = a linear feed traverse move (G01) to the Z–.75 shoulder

N55 = a linear feed traverse move (G01) to the outer diameter

N60 = a linear feed traverse move (G01) to a positive clearance of the outer diameter
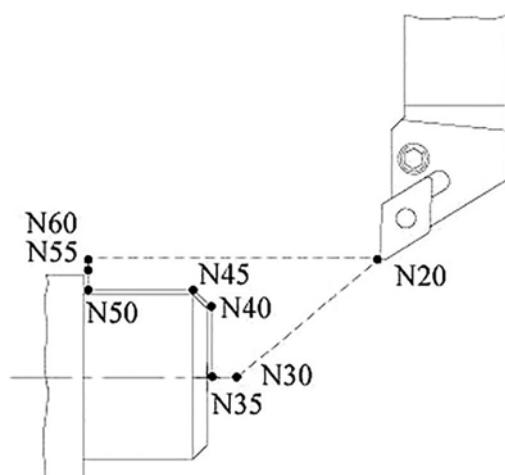
N65 = a rapid traverse move to return to the X and Z start position in line N20

Workpiece holding devices (chuck jaws) quite often extend beyond the workpiece holding equipment (chuck). Therefore, careful consideration should be given to the position of work holding devices (including tailstock centers and quill extension position), so that rapid traverse paths of the tool do not interfere with them and cause a crash that may damage the machine, clamping device, or part.

LINEAR INTERPOLATION (G01)

Linear interpolation is programmed by using the G01 function; it may be applied simultaneously for both axes. The G01 function commands the movement of the tool from a given position to the position with the assigned coordinates having feed rate specified by

Figure 3-6
Rapid Traverse Positioning and Feed
Traverse Comparison
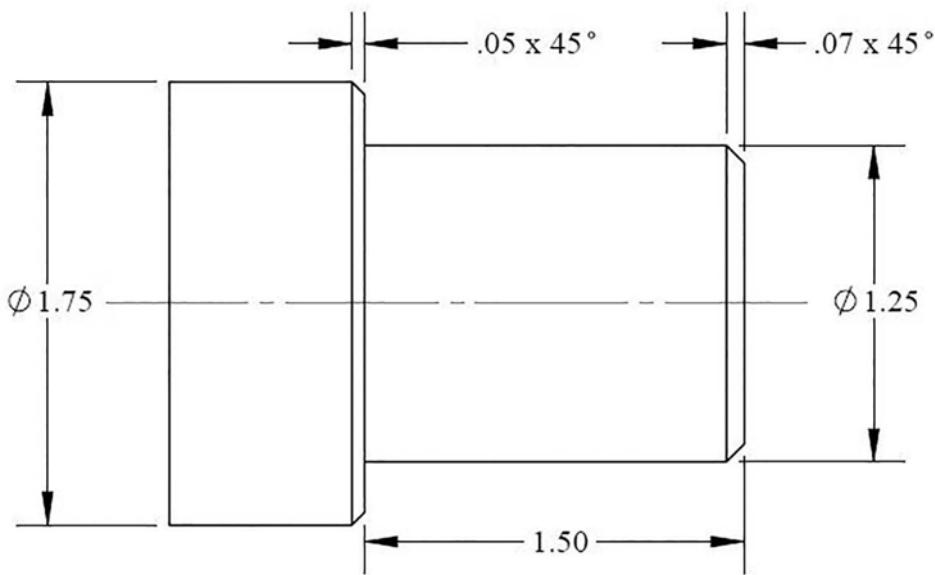
Figure 3-7
Drawing for Linear Interpolation (G01)

the F-Word. G01 is a modal command that stays in effect until replaced by another command from the same group. The block format for linear interpolation is given as follows:

G01 X(U). . . Z(W). . . F. . .

The interpolator in the control system calculates various speeds for the motion axis so that the resulting speed is equivalent to the programmed feed rate. For this example, only the finished profile cutting pass is programmed for the part geometry. See Important Notes: 1 and 2 above.

Sample Program O0003: Using Linear Interpolation Function (G01) (Figure 3-7)

O0003
N10 G50 S2500
N15 T0200 M42
N20 G00 G54 X1.95 Z.2 T0202 M08
N25 G96 S500 M03
N30 G01 Z0 F.01
N35 X1.11
N40 X1.25 Z–.07
N45 Z–1.5
N50 X1.650
N55 X1.75 Z–1.55
N60 X1.95

N65 G00 Z1.0 M09
N70 T0200
N75 G28 U0.0 W0.0 M05
N80 M30

In block N30 a Linear Interpolation (feed) move is commanded to the work face along the Z-axis at a feed rate of .01 in/rev.

In block N35, a linear feed move is made along the X-axis to the beginning of the chamfer at the same feed rate as commanded in block N30.

In block N40, a linear feed move is made along both the X- and Z-axes to create the chamfer.

In block N45, a linear feed move is made along the Z-axis to the shoulder.

In block N50, a linear feed move is made along the X-axis to the start of the second chamfer.

In block N55, a linear feed move is made along both the X- and Z-axes to create the second chamfer.

In block N60, a linear feed move is made along the X-axis to a safe distance beyond the outer diameter of the part.

In block N65, a rapid traverse move is made to the X- and Z-axes start point in front of the contour.

In blocks N30 through N60, linear interpolation moves are made along the finish contour of the part at the feedrate commanded in block N30.

CIRCULAR INTERPOLATION (G02 AND G03)

Circular interpolation allows programmed tool movements along an arc. In order to define the circular interpolation function, the following four conditions must be fulfilled:

1. Selection of the direction of interpolation:

   G02 = Clockwise

   G03 = Counterclockwise

2. Position coordinates of the starting point of the arc.

3. Position coordinates of the ending or point of the arc. The ending point coordinates may be omitted if they correspond to the coordinates of the starting point (half circle, circle).

4. The dimension corresponding to the distance between the center of the tool nose radius and the center of the arc from the starting point of each axis must be given.

The incremental distance for the X-axis is defined by the value of letter address I, and the Z-axis is defined by the value of the letter address K. Values for I and K may be omitted from the program if they are equal to zero. The block format for circular interpolation is given as follows:

G02 X(U). . . Z(W). . . I. . . K. . . F. . .
G03 X(U). . . Z(W). . . I. . . K. . . F. . .
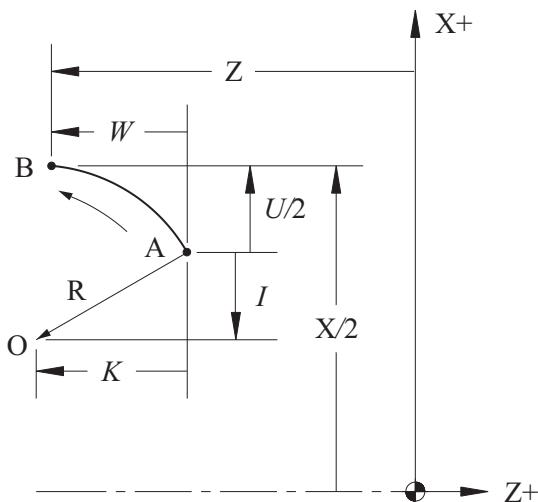
Figure 3-8
Arc Center Identification for
Circular Interpolation

Figure 3-8 graphically identifies all of the components necessary for programming arcs and their descriptions.

A = Arc starting point

B = Arc ending point

I = Incremental distance from start point (A) to the arc center (O) along the X-axis

K = Incremental distance from start point (A) to the arc center (O) along the Z-axis

O = Arc center

R = Arc radius (a negative signed value will produce a concave arc)

U/2 = Incremental distance from the arc starting point to the ending point along the X-axis

W = Incremental distance to the arc end point along the Z-axis

X/2 = Absolute coordinate for the ending point of the arc along the X-axis

Z = Absolute coordinate for the ending point of the arc along the Z-axis

In order to establish signs for I and K, consider the following directions: Imagine a line is drawn from the arc starting point to the arc center point with a direction vector toward the center of the arc. Next, project this vector onto the axes of the coordinate system with the origin at the arc start point. If the resulting projections of the vector are oriented in the same direction as the corresponding axis of the absolute coordinate system, the plus sign (+) is applicable. Otherwise, the minus sign (–) is applicable. If no sign is given in the coordinate entry, the control assumes the sign is positive.

Figure 3-9 is a pictorial representation showing how to determine the sign for vectors of I and K, while taking into account the tool nose radius.

Figure 3-10 shows how to determine the direction of the circular interpolation and signs for vectors I and K. (The tool tip in the drawing is represented by a circle for easier pictorial presentation.)
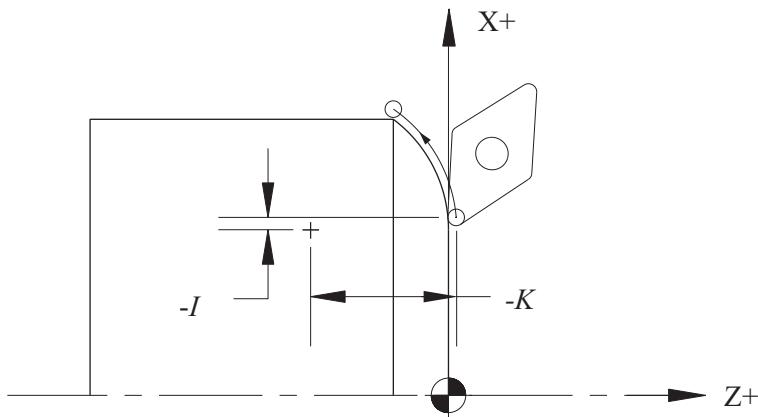
Figure 3-9
Circular Interpolation Sign Vector Selection
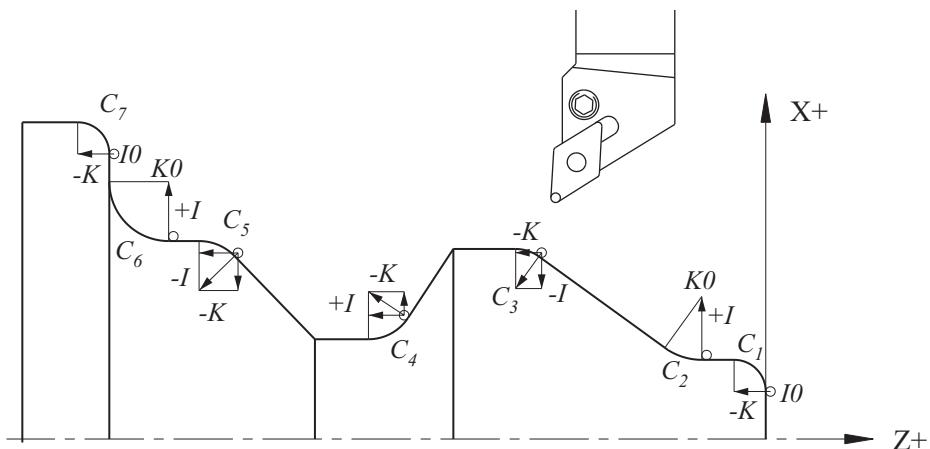
Figure 3-10
Circular Interpolation Direction Vector Selection

$C_1 = \text{G03, } I0, -K$

$C_2 = \text{G02, } +I, K0$

$C_3 = \text{G03, } -I, -K$

$C_4 = \text{G02, } +I, -K$

$C_5 = \text{G03, } -I, -K$

$C_6 = \text{G02, } +I, K0$

$C_7 = \text{G03, } I0, -K$

Part 3 Programming CNC Turning Centers

Modern CNC controls include an additional capability to use R in place of I and K. R is the distance from the center of the tool radius to the center of the following arc. If an arc is smaller than or equal to 180°, then R assumes a positive sign; if it is greater than 180°, then R assumes a negative sign. The block format for circular interpolation using R is given as follows:

G02 X(U). . . Z(W). . . R. . . F. . .

G03 X(U). . . Z(W). . . R. . . F. . .

Circular interpolation can be performed these two different ways: the first using I and K, the second using R. However, the application of address R is less difficult.

$$R = R_1 + r_1$$

where:

R1 = Radius of the arc
r1 = Radius of the tool nose

Note: If the tool nose radius compensation is used in the program (G41 or G42, covered later), the value for R is the radius of the arc only.

If I and K are applied, then machine control is fed with precise information about the position of the center of the radius of the performed arc. In this case, the coordinates of the arc ending point must correspond to a position on the programmed circle. If, however, the given values of the coordinates are incorrect, the tool will not respond by following an arc.

If the second method using R is employed, the tool will follow an arc equal to the value given, i.e., R.375, even if the values of the coordinates are incorrect (Figure 3-11). (This is true if the ending point of the arc falls within the area of the diameter of the circle.)

Sample Program O0004: Circular Interpolation (G02 or G03)

The following program is an example of circular interpolation utilizing I and K. Once again, in this example, only the finished profile is programmed for the part geometry. Compensation for the tool nose radius will not be accounted for along the radius, even if it is set in the Tool Data register (R) of the control. Tool Nose Radius Compensation G41
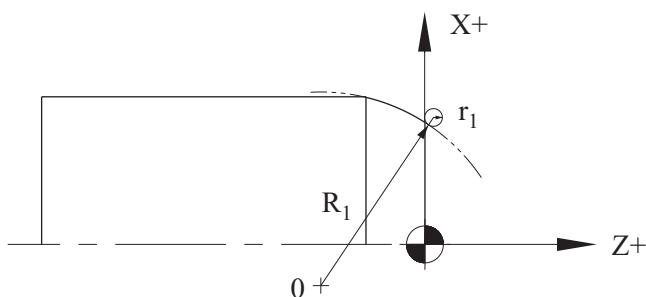
Figure 3-11
Circular Interpolation Using R
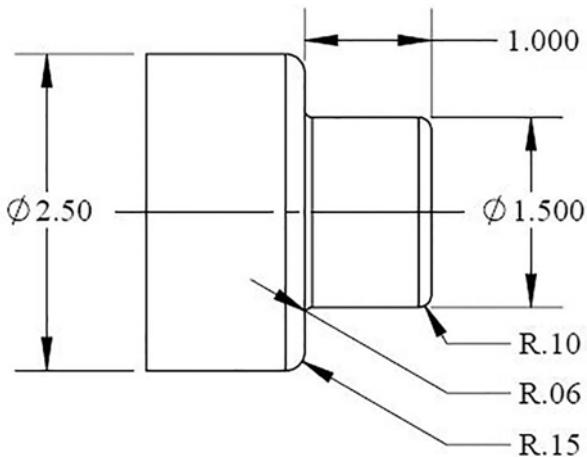
Figure 3-12
Drawing for Circular
Interpolation Program

and G42 will be covered later in this chapter. For the part geometry used in this sample program, refer to Figure 3-12. See Part 2 of this text for an explanation of proper Tool Data entry.

O0004

N10 G50 S2000

N15 T0400 M42

N20 G00 G54 X2.6 Z.2 T0404 M08

N25 G96 S500 M03

N30 G01 Z0 F.02

N35 X–.030

N40 Z.1

N45 G00 X1.3 W.1

N50 G01 Z0

N55 G03 X1.5 Z–.1 K–.1 I0

N60 G01 Z–0.94

N65 G02 U.06 Z–1.0 K0.0 I.06

N70 G01 X2.2

N75 G03 X2.5 W–.15 I0.0 K–.15

N80 X2.6

N85 G00 Z.2 M05

N90 T0400

N95 G28 U0.0 W0.0 M09

N100 M30

The following program is based on the same example, except it uses R instead of I and K.

> **O0004**
>
> **N10 G50 S2000**
>
> **N15 T0400 M42**
>
> **N20 G00 G54 X2.6 Z.2 T0404 M08**
>
> **N25 G96 S500 M03**
>
> **N30 G01 Z0 F.02**
>
> **N35 X–.03**
>
> **N40 Z.1**
>
> **N45 G00 X1.3**
>
> **N50 G01 Z0**
>
> **N55 G03 X1.5 Z–1.0 R.1**
>
> **N60 G01 Z–0.94**
>
> **N65 G02 U.12 Z–1.0 R.06**
>
> **N70 G01 X2.2**
>
> **N75 G03 X2.5 W–.15 R.15**
>
> **N80 G00 X2.6**
>
> **N85 Z.2 M05**
>
> **N90 T0400**
>
> **N95 G28 U0.0 W0.0 M09**
>
> **N100 M30**

DWELL (G04)

Dwell is initiated by use of function G04, and the length of time for the dwell is specified by P, X, or U, (depending on the control type) as follows:

> G04 P . . . (in milliseconds)
>
> G04 X . . . (in milliseconds)
>
> G04 P . . . (in milliseconds)
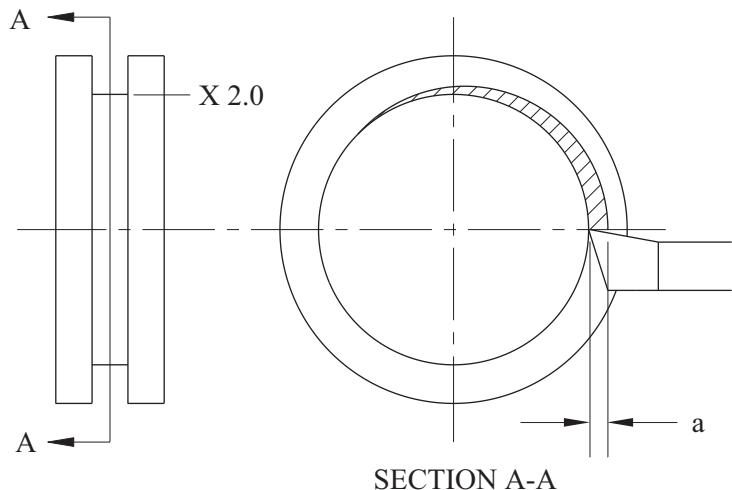
Examples

> G04 P2500
>
> G04 X2.5
>
> G04 U2.5

In the examples above, the dwell time values are the equivalent of 2 and 1/2 seconds. Also note that when using P to address the amount of time for dwell, a decimal point may not be used. The value of time is measured in milliseconds (ms), 1000 ms = 1 second. Function G04 is a "one shot" command; it is active only in the block in which it is called.

Figure 3-13
Drawing for Dwell (G04)
Example

SECTION A-A

The dwell is activated at the end of the feed move and should be the only contents of the block. Dwell is sometimes indicated by the number of revolutions as opposed to the amount of time (dependent on a parameter setting). Study the manufacturer programming manual specific to the equipment to be sure of the exact method used. A common use for dwell is in the process of machining internal or external grooves, as shown in Figure 3-13 and described below.

Examples

G01 X2.0 F.008

G04 U.25 (or G04 X0.25 or G04 P250)

G00 X2.5

In the process of making the groove, you must remove a layer of material with a thickness corresponding to the depth of the cut and equivalent to the feed per revolution. In order to avoid an egg-shaped workpiece, a certain amount of time must be allowed for the cut to be completed for the full circumference as the tool tip reaches the diameter indicated in the program. If function G00 follows function G01, the resulting shape of the workpiece will be that of an egg because the tool is removed from the groove before all of the material can be cut. In the example above, the programmed dwell is 1/4 second; which allows a sufficient pause necessary to clean up the bottom of the groove before retracting from it.

REFERENCE POINT RETURN (G28)

One of the ways to command a tool to return to the reference point after completion of an operation is through the application of function G28. The block format for Reference Point Return (G28) is given as follows.

G28 X(U). . . Z(W). . .

In this block, the values in X(U) and Z(W) are the coordinates for an intermediate point through which the tool will pass on its way to machine zero return (Figure 3-14).
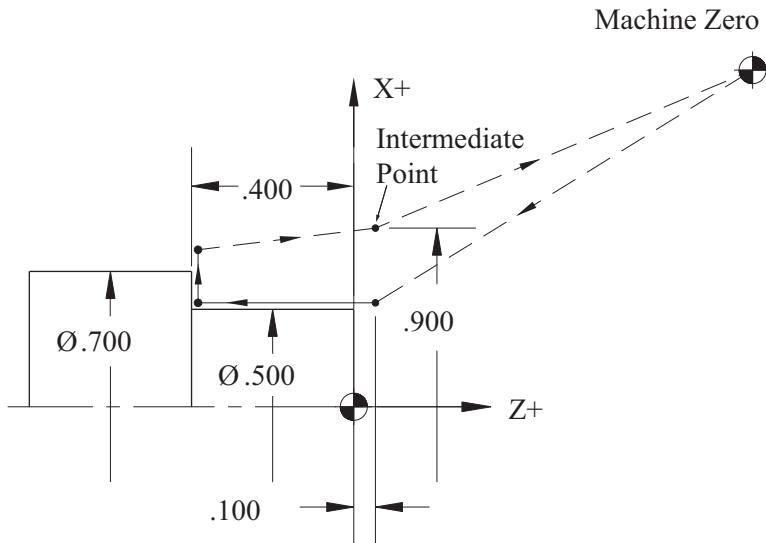
Figure 3-14
Reference Point Return
(G28)

The tool will position at a rapid traverse rate in nonlinear form. Therefore, an escape move off the part should be programmed so that the tool is clear of the part before commanding G28. When using G28, tool offsets should be cancelled in a prior block.

T0200 = Cancellation of tool offset

Sample Program O0005: Reference Point Return (G28)

O0005
N10 G50 S800
N15 T0200 M42
N20 G00 G54 X.5 Z.1 T0202 M08
N25 G96 S200 M03
N30 G01 Z–.4 F.008
N35 X.8
N40 T0200
N45 G28 X.9 Z.1 M05
N50 M30

Block N45 is programmed in the absolute coordinate system. To change the command into the incremental system, follow this format:

N7 G28 U.1 W.5 M05

If no obstacles are present on the tool's path as the tool proceeds to Reference Point Return, then most often the intermediate point is merely a directional point.
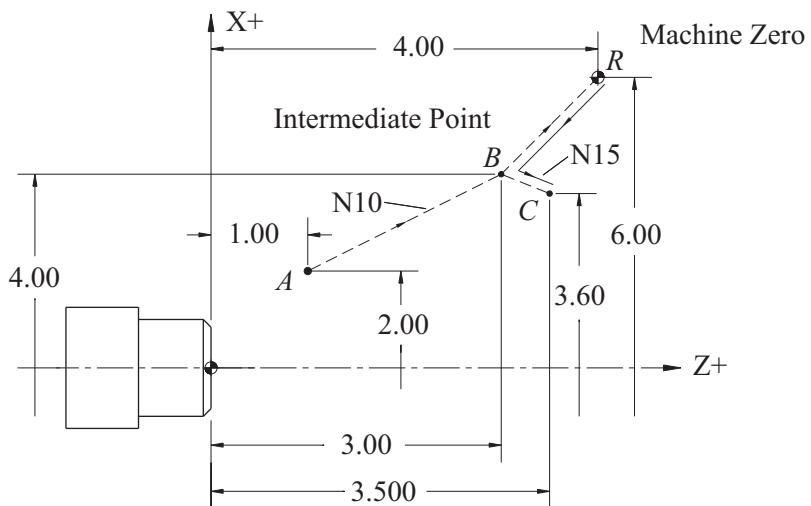
N7 G28 X.8 Z–.4 M05

or

N7 G28 U0 W0 M05

Figure 3-15
Return from
Reference
Point (G29)

RETURN FROM REFERENCE POINT (G29)

Function G29 generally follows G28; it MUST NOT be programmed without using G28 first. Using this command *returns* the tool, at rapid traverse, to a programmed point by way of the same intermediate point as given in the G28 command. The block format used for G29 is as follows:

G29 X(U). . . Z(W). . .

N10 G28 U2.0 W2.0

N15 G29 U-0.4 W0.5

As shown in Figure 3-15, the tool has moved from point *A* to point *B* (the intermediate point) and then on to point *R* (Machine Zero), all of which is determined in the first block with function G28. In block N2, the tool automatically returns from point *R* toward point *B*. Programming is limited to the calculation of the distance between points *A* and *C*. Point *R* is the Machine Zero position. Point *C* refers to the first position after tool change.

OUTER/INNER DIAMETER TURNING CYCLE (G90)

The outer/inner diameter turning cycle is a cylindrical cutting function (to cut diameters). The block format used for G90 is as follows:

G90 X(U). . . Z(W). . . F. . .

Using function G90 in a program is a convenience. However, using function G90 will result in some loss of time because, after each pass, the tool returns by the motion *d*, as shown in Figure 3-16. This figure also shows the cycle execution is performed with four movements of the tool.

The drawing in Figure 3-17 is used to create sample program O0006.

Sample Program O0006: Outer/Inner Diameter Turning Cycle

O0006

N10 G50 S2000
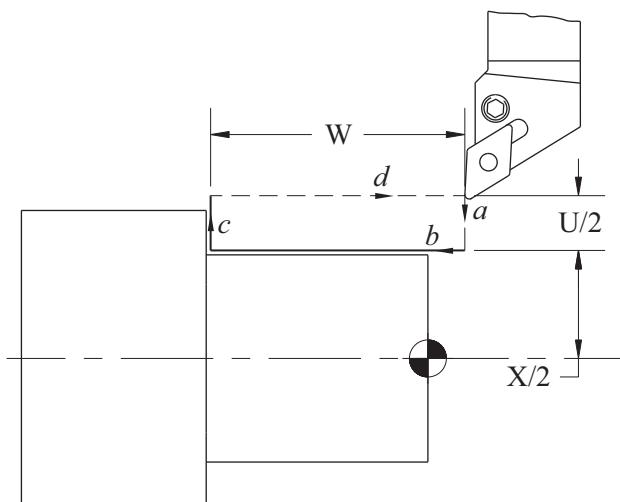
Figure 3-16
Outer/Inner Diameter
Turning Cycle (G90)
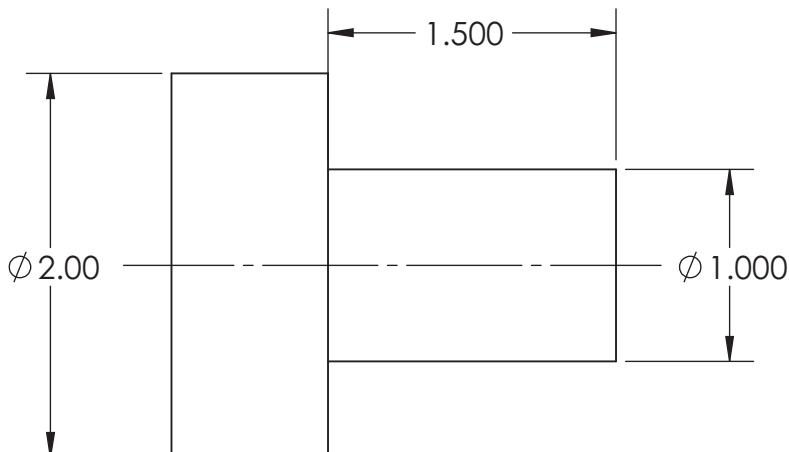
Figure 3-17
Drawing for
Outer/Inner
Diameter Turning
Cycle (G90)

N15 T0200 M42

N20 G00 G54 X2.1 Z.2 T0202 M08

N25 G96 S500 M03

N30 G90 X1.8 Z–1.5 F.015

N35 X1.6

N40 X1.4

N45 X1.2

N50 X1.0

N55 T0200 M09

N55 G28 U0.0 W0.0 M05

N60 M30

Note: Depending on the control type, tapered cuts may be programmed for function G90 by including letter address I or R. Where either is input, there is a radial value of the difference between the starting and ending diameters.

THREAD CUTTING CYCLE (G92)

By using G92, a cycle of four individual movements of the tool can be obtained in one block of information. These movements are:

1. Rapid traverse to a given diameter

2. Thread cutting with programmed feed rate

3. Rapid withdrawal

4. Rapid traverse return to the starting point

Notes: Changing the spindle speed or the feed rate override while within the threading cycles is not effective.

A "dry run" condition is applicable and effective.

The use of constant rotational speed programming, G97 S . . ., is required!

The cycle execution is performed with four movements of the tool, the same as shown in Figure 3-16. Types of threads available are as follows:

1. Straight thread (cylindrical) (Figure 3-18)

   G92 X(U). . . Z(W). . . F or E. . .

2. Taper thread (Figure 3-19)
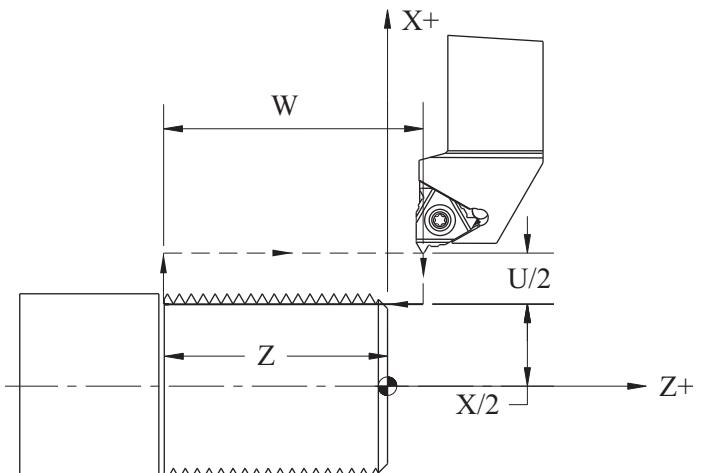
   G92 X(U). . . Z(W). . . I. . . F or E. . .

By activating SINGLE BLOCK, the introduced movements of a, b, c, and d can be executed by pressing CYCLE START.

Note: By using function G92, it is possible to end the thread with a 45° chamfer or simply at a right angle.

In Figure 3-20, a 45° chamfer is obtained by using function M23 and a 90° thread end is obtained by using function M24, as seen in Figure 3-21.

Figure 3-18
Thread Cutting Cycle (G92)
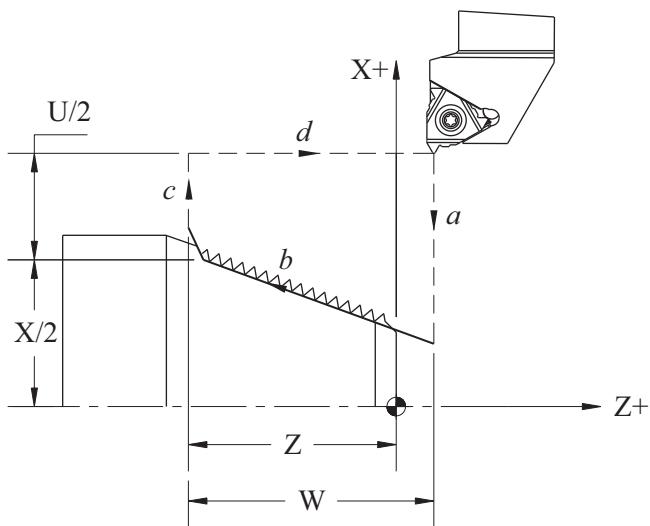
Figure 3-19
Thread Cutting Cycle (G92) for
Tapered Threads
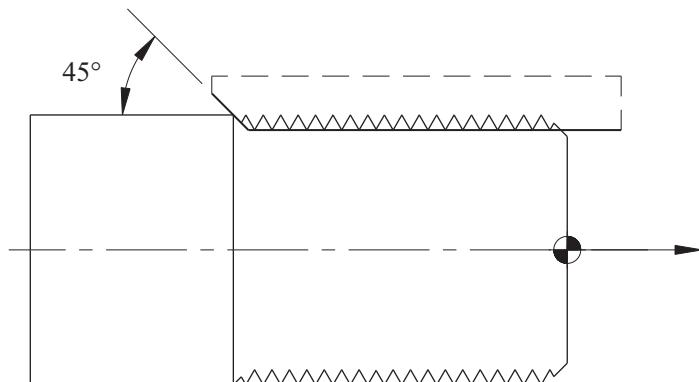
Figure 3-20
Thread End
Function (M23)
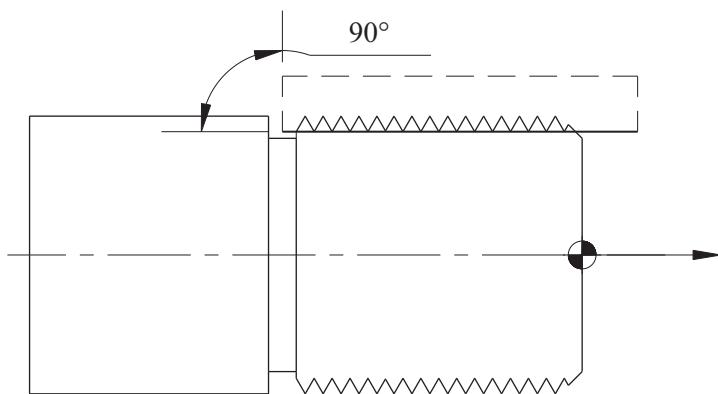
Figure 3-21
Thread End Function (M24)

At the end of the threading routine, the G92 function must be cancelled by a G00 move. If it is not, the next programmed movement will continue as if it were still threading.

Note: On some older control models, function G92 is a position register setting code. In these cases, when G92 is used for threading, the G50 position register command must be used.

119
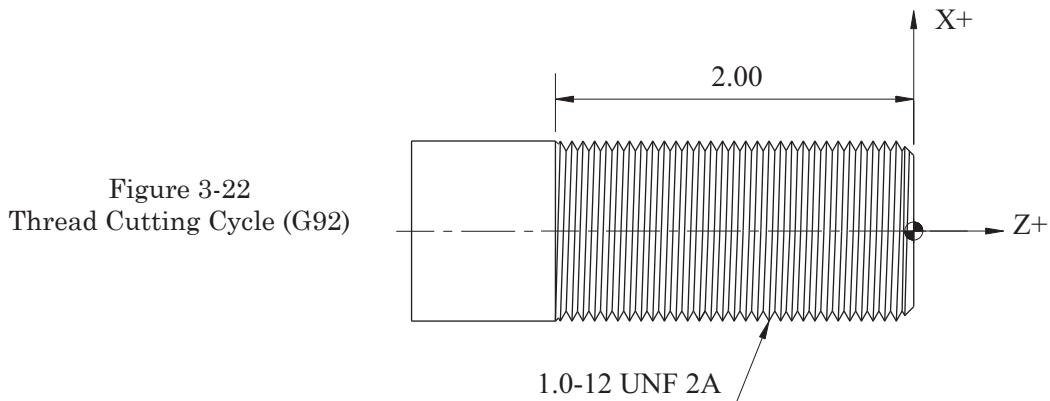
Figure 3-22
Thread Cutting Cycle (G92)

Sample Program O0007: Thread Cutting Cycle (Figure 3-22)

O0007
N10 G50 S2000
N15 T0800 M42
N20 G97 S800 M03
N25 G00 G54 X1.1 Z.5 T0808 M08
N30 G92 X.97 Z–2.0 F0.083333
N35 X.95
N40 X.92
N45 X.90
N50 T0800 M09
N55 G28 U0 W0 M05
N55 M30

Note: Using this cycle with work requiring repetitive threading and cutting operations is very convenient, especially for large thread sizes. In such cases, after executing the block containing function G92, a certain number of consecutive blocks may be omitted and, instead, only the last few blocks may be executed.

It is convenient to use function G92 in programs that result in much shorter length. For the above program, blocks N35, N40, and N45 deal with diameters for particular passes, while the preceding data that form block N30 are valid until block N50. The value of feed F (or E for some types of controls) may be expressed as shown in the preceding example with programmable accuracy reaching .000001 inch. In this example, the value for Z in block N25 is .5. As far as the programmer is concerned, thread cutting may be initiated from a point positioned much closer to the material.

Threading could start as close as Z = 0.0. Practically speaking however, it is not possible for the tool to begin the operation with the feed rate given in a program. Thus, part of the tool path is followed by the tool, with acceleration, until the tool reaches the value of the feed rate equivalent to thread lead (Figure 3-23). A similar situation occurs at the end of the threading process when the tool decelerates. A certain distance is traveled by the tool after some delay.
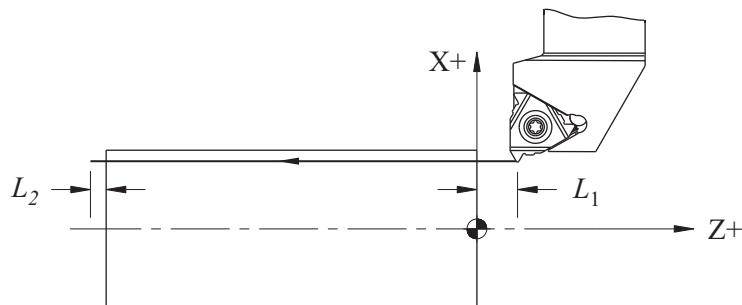
Figure 3-23
Threading Acceleration
and Deceleration

The lengths of $L1$ and $L2$, in Figure 3-23 depend on the thread pitch and rotational speed of the spindle. Theoretical analysis of the mathematical formulas applied in calculating such lengths is rather complicated. For this reason, we will limit our explanation to the more simple calculations that are based on the following equations.

$$L_2 = \frac{P \times S}{1800} \left(\text{in}\right)$$

where:

P = lead (in)

S = revolutions per minute

$$L_1 = L_2 \times K \quad K = \ln\frac{1}{a} - 1 \quad a = \frac{\Delta P}{P}$$

where:

ΔP = lead error

P = lead

The amount of lead error P depends on the design of the machine, as well as its servo motor systems. Values of the coefficient K, for a few specified error allowances, are contained in Chart 3-3.

Example

Given:

P = 0.0833

S = 1200

a = 0.010

Chart 3-3 Error Allowance Coefficients

a	K
.02	2.91
.015	3.2
.01	3.605
0.005	4.298

$$L_2 = \frac{P \times S}{1800} = \frac{0.0833 \times 1200}{1800} = 0.0555$$

$$L_1 = L_2 \times K = 0.0555 \times 3.605 = 0.200$$

In most cases, these values are based on experience.

TAPERED THREAD CUTTING USING CYCLE (G92)

In order to cut a tapered thread using function G92, use the same block format shown in item two in the previous section. The additional letter address I represents the value of the radius or difference per side between the thread diameters at the end of the cut to the thread diameter at the start of the cut. Depending on the sign for I in the taper threading command G92, the cutting tool will move as shown in Figure 3-24 and 3-25.
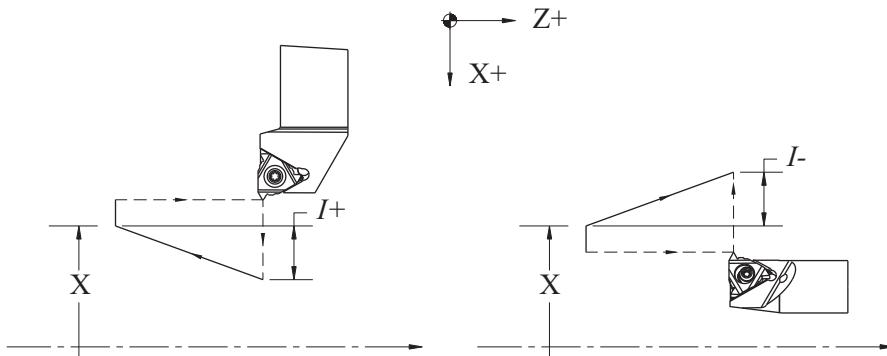
Figure 3-24
Tapered Thread Cutting Using Cycle (G92)
Front Turret
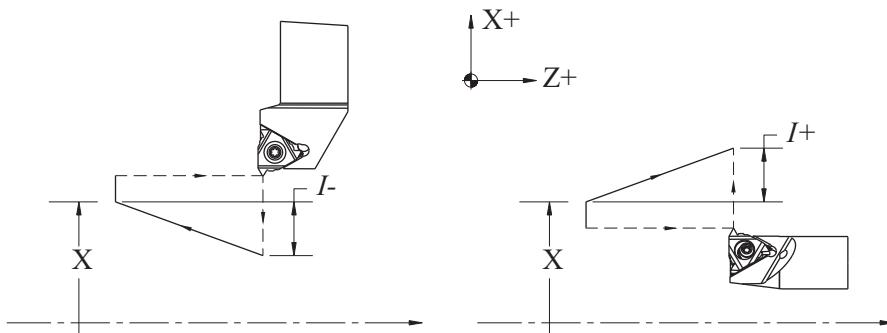
Figure 3-25
Tapered Thread Cutting Using Cycle (G92)
Rear Turret

FACE CUTTING CYCLE (G94)

The face cutting cycle is a function similar in application to function G90 except that it is used for facing.

G94 X(U). . . Z(W). . . F. . .

As with function G90, the tool always returns after each pass to the starting point by motion d, as shown above in Figure 3-26. This is one reason why both methods of programming, G90 and G94, are not widely used in practice today. Multiple repetitive cycles are a much better choice and are discussed next.

Sample Program O0008: Face Cutting Cycle (Figure 3-27)

> **O0008**
>
> **N10 G50 S2000**
>
> **N15 T0200 M42**
>
> **N20 G96 S500 M03**
>
> **N25 G00 G54 X2.6 Z.1 T0202 M08**
>
> **N30 G94 X1.5 Z–.1 F.015**
>
> **N35 Z–.2**
>
> **N40 Z–.3**
>
> **N45 Z–.4**
>
> **N50 Z–.5**
>
> **N55 Z–.6**
>
> **N60 T0200 M09**
>
> **N65 G28 U0 W0 M05**
>
> **N65 M30**

Note: Tapered cuts may be programmed for function G94 by inclusion of address K, where input is a radial value of the difference between the starting and ending diameters.

Figure 3-26
Face Cutting Cycle (G94)
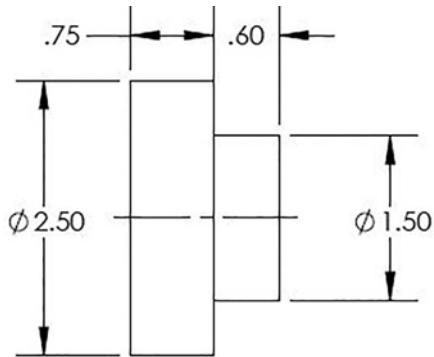
Figure 3-27
Drawing for Face Cutting
Cycle (G94)

MULTIPLE REPETITIVE CYCLES

Stock Removal Turning Cycle (G71)

Function G71 is the stock removal cycle for turning that removes metal along the direction of the Z-axis. In a case where there is a lot of material to be removed, such as bar stock, this cycle provides an easy method for programming.

In Figure 3-28, the dotted lines refer to the initial shape of the workpiece, whereas the solid line refers to the final product. In the programming described so far, it has been necessary to employ many blocks of information to perform

Figure 3-28
Stock Removal Turning Cycle (G71)

all the individual cuts for roughing. By using function G71, programming of the final shape of the workpiece is defined. Material is removed automatically in each pass.

There are two types of program format for stock removal using function G71: single block and double block. The CNC control model used determines which type will be needed. Consult the manufacturer programming manual specific to the machine to determine the required method.

Function G71, Double Block

There are two program blocks required for function G71 when using the double block method. The finished profile of the part, as shown in Figure 3-29, is machined starting at point *a* and proceeding to points *b* and *c*. The metal removal amount along the X-axis is defined by the parameter U (depth of cut), in the first program block. A finish allowance for the X-axis is defined by the parameter U in the second block. Be careful not to get the two confused, as they do different things. The finish allowance for the Z-axis is defined by W in the second block.

The following is the block format for function G71:

G71 U. . . R . . .

G71 P. . . Q. . . U. . . W. . . F. . . S. . .

where:

in block one:

    U = the depth of each roughing cut per side, to be used in consecutive passes (no sign)

    R = the amount of retract, along the X-axis, for each cut

in block two:

    P = the sequence number of the first block in the program, which defines the finish profile

    Q = the sequence number of the last block of the program, which defines the finish profile

    U = the stock allowance to be left for a finishing pass in the X-axis direction (diameter sign is + or –)
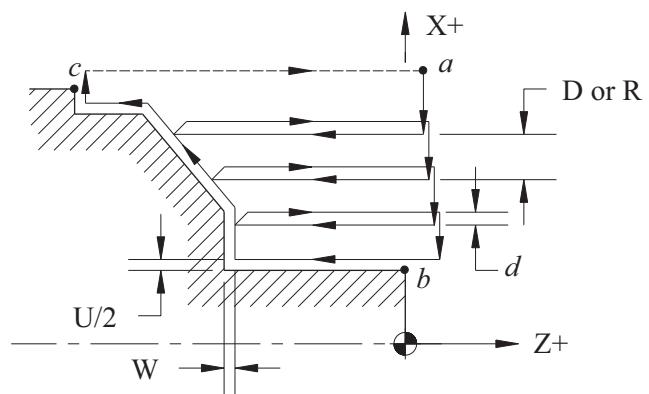
Figure 3-29
Stock Removal Turning
Cycle (G71) Diagram

W = the stock allowance to be left for finishing in the Z-axis direction (sign is + or–)

F = Cutting feed rate (in/rev or mm/rev) for blocks defined from P to Q

S = Spindle speed (ft/min or m/min) for blocks defined from P to Q

The signs attached to symbols U and W may have negative or positive values, depending on the orientation of the coordinate system and the direction in which the allowance is assumed.

In Figure 3-29, *d* represents the amount of X-axis retract programmed for clearance by R in the first program block. This amount may also be set by parameter.

For Figure 3-29:

a = the starting point of the given cycle

b = the sequence number of the first programmed point for the finish contour, which corresponds with P number of the second G71 block

c = the sequence number of the last programmed point for the finish contour

Notes for the double block command:

1. *Changes in the feed between blocks P and Q will be ignored in G71. Only feed F, indicated by function G71, is valid.*

2. *The first tool path move of the programmed cycle from point a to point b cannot include any displacement in the direction of the Z-axis.*

3. *The tool path between point b and point c must be a steadily increasing or steadily decreasing pattern in both axes.*

4. *Both linear and circular interpolation are allowed.*

5. *The value for R must be noted as in the following examples:*

    R2000

    R2500

    R1500

6. *For some controls, use of a decimal point may be allowed.*

7. *Figure 3-30 illustrates that if the allowance for finishing is located on the positive side (in the direction of the X and Z axes) with respect to the programmed contour, no sign is used; if on the negative side, the negative sign (–) is used.*

Function G71, Single Block
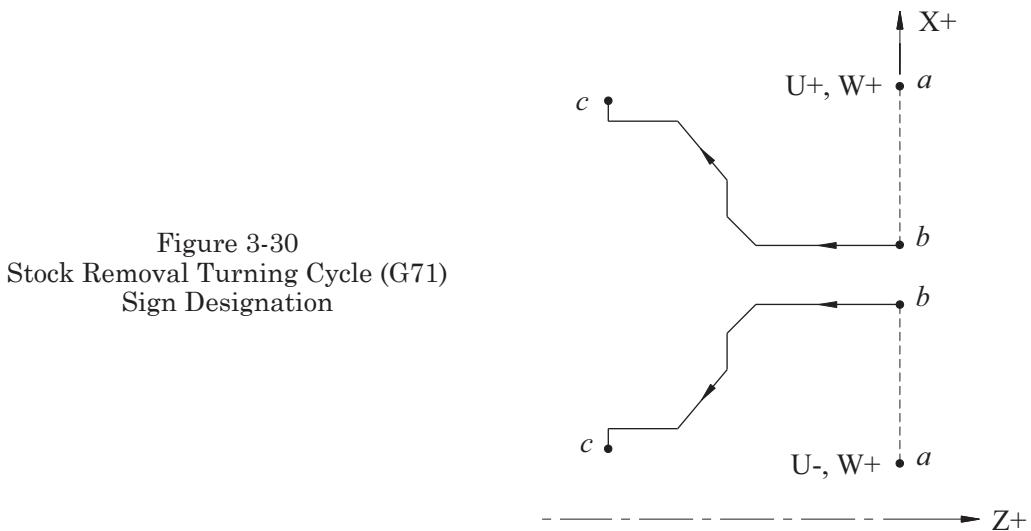
There is only one program block required for function G71 when using this method. Much more freedom is allowed in regards to programmable shapes. In this case, it is not necessary to program a steadily increasing or decreasing pattern in both axes, it is only required along the Z-axis, and up to ten concave figures are allowed.

Noteworthy differences between double and single block function G71:

1. Block format for a single block is as follows:

    G71 P. . . Q. . . I. . . K. . . U. . . W. . . D. . . F. . . S. . .

Figure 3-30
Stock Removal Turning Cycle (G71)
Sign Designation

where:

All parameters are identical as stated above except:

      I = Radial distance and direction of the rough cut along the X-axis
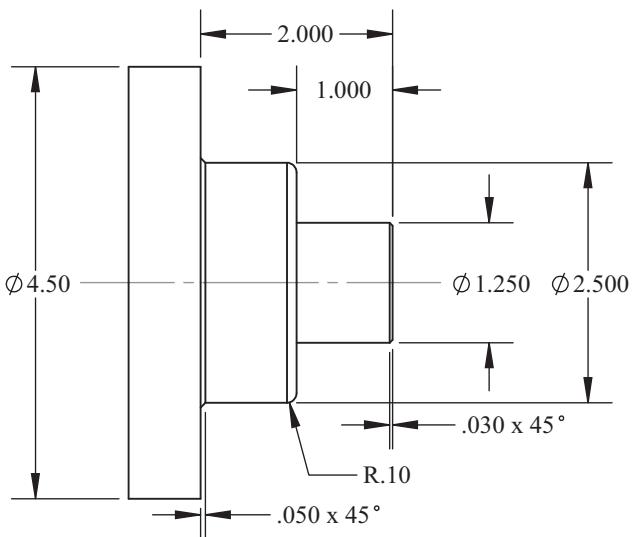
      K = Distance and direction of the rough cut along the Z-axis

      D = Depth of each roughing cut

2. In the single block format, two axes may be programmed in the first block identified by the parameter P.

3. If single block format programming is used and the first block does not include any Z displacement, 0 must be input for parameter W in the G71 block.

Sample Program O0009: Stock Removal Turning Cycle

Figure 3-31
Drawing for Stock Removal
Turning Cycle (G71)

The following is a program example for a double block call of G71:

O0009

N10 G50 S2000

N15 T0200 M42

N20 G00 G54 X4.75 Z.2 T0202 M08

N25 G96 S600 M03

N30 G71 U.12 R.05

N35 G71 P40 Q80 U.03 W.015 F.019

N40 G00 X1.19

N45 G01 Z0

N50 X1.25 Z–.03

N55 Z–1.0

N60 X2.3

N65 G02 X2.5 W–.1 I0.0 K–.1

N70 G01 Z–1.95

N75 X2.6 W–.05

N80 X4.75

N85 T0200 M09

N90 G28 U0 W0 M05

N95 M30

The following is a program example for a single block call of G71:

O0009

N10 G50 S2000

N15 T0200 M42

N20 G00 G54 X4.75 Z.2 T0202 M08

N25 G96 S600 M03

N30 G71 P35 Q75 U.03 W.003 D1200 F0.019

N35 G00 X1.19

N40 G01 Z0.0

N45 X1.25 Z–.03

N50 Z–1.0

N55 X2.3

N60 G02 X2.5 W–.1 I0.0 K–.1

N65 G01 Z–1.95

N70 X2.6 W–.05

N75 X4.75

N80 T0200 M09

N85 G28 U0.0 W0.0 M05

N90 M30

Comments: Block N35 appears right after the block containing function G71. Despite the fact that the last programmed cycle block N75 refers to the location of tool tip Z–2.0, the tool will automatically return to the starting point indicated as Z.2 in the block N20.

STOCK REMOVAL FACING CYCLE (G72)

The properties for function G72 are similar to G71 (Figure 3-29). The only difference is in the change of the cutting direction to facing as detailed in Figure 3-32. The following is a function block diagram:

Function G72, Single Block:

G72 P... Q... I... K... U... W... D... F... S...

Function G72, Double Block:

G72 W... R...

G72P... Q... U... W... D... F... S...

The parameters for this function have the same meaning as those described for function G71.

Notes:

1. *The first block of the programmed cycle should not include any displacement in direction of the X-axis.*

2. *The remaining notes for this function are identical to those for function G71.*

3. *The principle defining the choice between a positive or negative sign for U and W is identical to function G71.*

Figure 3-32
Stock Removal Facing Cycle (G72)

Sample Program O0010: Stock Removal Facing Cycle (Figure 3-33)

O0010

N10 G50 S2000

N15 T0200 M42

N20 G96 S500 M03

N25 G00 G54 X4.4 Z.2 T0202 M08

N30 G72 P35 Q60 U.03 W.012 D1000 F.012

N35 G00 Z–1.5

N40 G01 X3.5

N45 X3.1 Z–1.3

N50 X2.0

N55 Z–.5
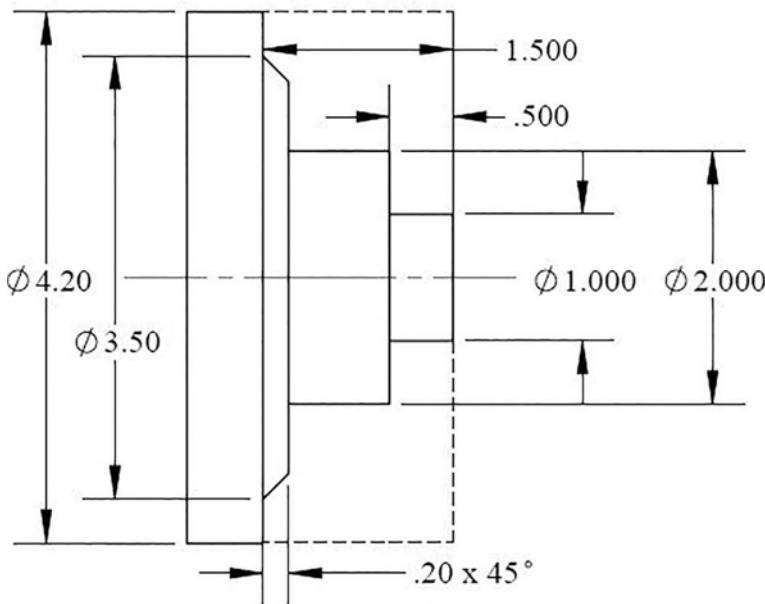
Figure 3-33
Drawing for Stock Removal Facing Cycle (G72)

N60 X1.0

N65 T0200 M09

N70 G28 U0 W0 M05

N75 M30

PATTERN REPEATING FUNCTION (G73)

Function G73 permits the repeated cutting of a fixed pattern, with displacement of the axes position by an amount determined by the total material to be removed, divided by the number of passes desired. This cycle is well suited for previously formed castings, forgings, or rough machined materials. This machining method assumes that an equal amount of material is to be removed from all surfaces. It can still be used if the amounts are not equal. However, caution should be applied concerning excessive depths of cut, and there may also be occasions of air cutting. Basically, the best scenario is when the finished contour closely matches the casting, forging, or rough material shape.

The following is the block diagram for function G73 using the double block format:

G73 U. . . W. . . R. . .

G73 P . . . Q . . . U. . . W. . . F. . . S. . .

where:

in block one:

U = total displacement in the direction of the X-axis (sign + or –)

W = total displacement in the direction of the Z-axis (sign + or –)

R = the number of rough cutting passes

in block two:

P = number of the first block of the finished profile (given in the following figure as position *b*)

Q = number of the last block of the finished profile (given in the following figure as position *c*)

U = finish stock allowance in the direction of the X-axis (sign + or –), referred to the diameter

W = finish stock allowance in the direction of the Z-axis (sign + or –)

F = feed rate, effective for blocks P through Q

S = spindle speed, effective for blocks P through Q

Notes on function G73 using the double block format:

1. *Do not confuse the function of U and W with those in the single block format.*

2. *The letter address D is not used in the G73 double block format. In this case, the depth of cut is automatically calculated by the control based on values input for: U and W, the stock removal amount entered in the X and Z axes, and the number of cutting passes identified with R.*

The following is a block diagram for function G73 using a single block:

G73 P. . . Q. . . U. . . I. . . K. . . U. . . W. . . D. . . F. . . S. . .

where:

P = number of the first block of the finished profile (given in the following figure as position *b*)

Q = number of the last block of the finished profile (given in the following figure as position *c*)

I = total displacement in the direction of the X-axis (sign + or –)

K = total displacement in the direction of the Z-axis (sign + or –)

U = finish stock allowance in the direction of the X-axis (sign + or –), referred to stock left on the diameter

W = finish stock allowance in the direction of the Z-axis (sign + or –)

D = the number of rough cutting passes

F = feed rate, effective for blocks P through Q

S = spindle speed, effective for blocks P through Q

Point *a* on the drawing in Figure 3-34 is the starting point. In executing this cycle, the tool travels from point *a*
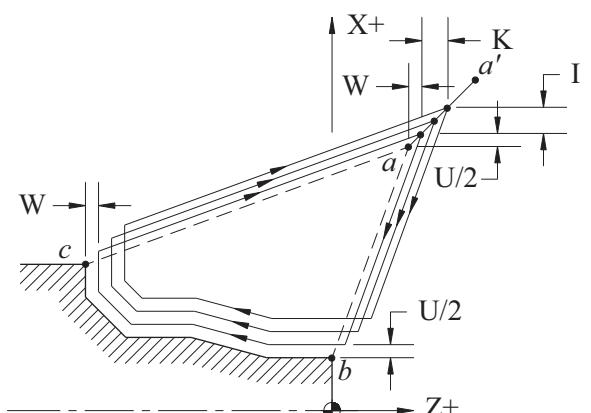
Figure 3-34
Pattern Repeating Cycle (G73) Diagram

to point *a*'. Axis displacement amounts are defined by the values of I and K in Figure 3-35. Finish stock allowances are defined with U and W. The cycle begins equivalent cutting passes with the number of passes determined by *D*. At the end of the cycle, the tool automatically returns to point *a*. Points *b* to *c* define the finished profile to be machined by function G70 as described in the next section.

Address I and K are measured on the machined workpiece. The principle defining choice of signs is similar to that for U and W.

Sample Program O0011: Pattern Repeating Cycle (Figure 3-36)

The following is a program example for a double block call of G73:

O0011
N10 G50 S2000
N15 T0200 M42
N20 G96 S500 M03
N25 G00 G54 X2.2 Z.3 T0202 M08
N30 G01 Z0.0 F.03
N35 X–.03 F.012
N40 G00 X3.0 Z.2
N45 G73 U.156 W.156 R3
N50 G73 P50 Q85 U.04 W.02 F.012
N50 G00 X1.59
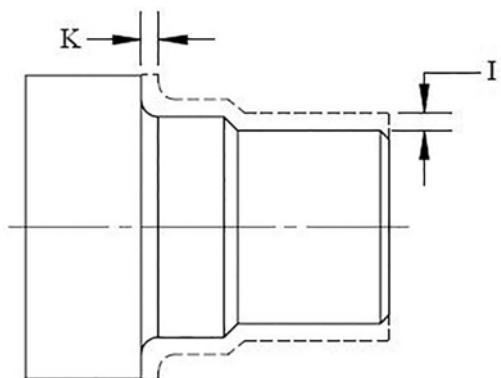
Figure 3-35
Pattern Repeating Cycle (G73) Equal Material Diagram
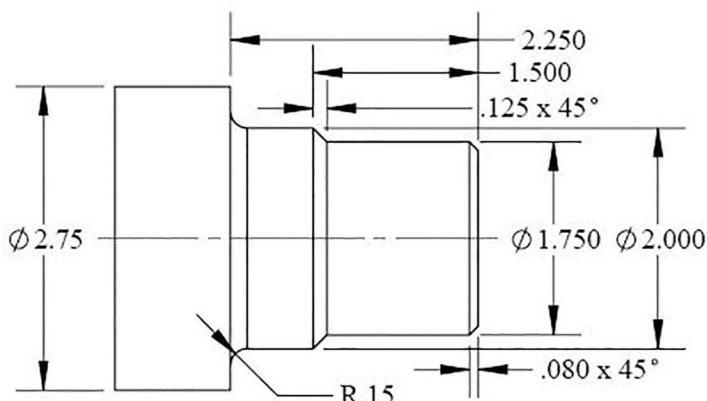
Figure 3-36
Drawing for Pattern Repeating Cycle (G73)

N55 G01 Z0.0

N60 X1.75 Z–.08

N65 Z–1.375

N70 X2.0 W–.125

N75 Z–2.1

N80 G03 U.3 Z–2.25 I–.15 K0.0 F.004

N85 G01 X2.85

N90 T0200 M09

N950 G28 U0 W0 M05

N100 M30

The following is a program example for a single block call of G73:

O0011

N10 G50 S2000

N15 T0100 M42

N20 G96 S500 M03

N25 G00 G54 X2.2 Z.3 T0101 M08

N30 G01 Z0.0 F.03

N35 X–.03 F.012

N40 G00 X3.0 Z.2

N45 G73 P50 Q85 I.156 K.156 U.04 W.02 D3 F.012

N50 G00 X1.59

N55 G01 Z0.0

N60 X1.75 Z–.08

N65 Z–1.375

N70 X2.0 W–.125

N75 Z–2.1

N80 G03 U.3 Z–2.25 I–.15 K0.0 F.004

N85 G01 X2.85

N90 T0100 M09

N950 G28 U0 W0 M05

N100 M30

FINISHING CYCLE (G70)

Stock allowances left for finishing (U, W) may be removed by the same tool used in rough cutting. However, it is a common practice to use a different tool for the finishing pass. Application of function G70 allows for removal of the remaining stock allowance (with the previously applied cycles G71, G72, and G73, without repetitive passes, along the contour).

The following is a block diagram for function G70:

G70 P. . . Q. . . F. . . S. . .

where:

P = number of the first block of the finished profile (given in the above figure as position *b*)

Q = number of the last block of the finished profile (given in the above figure as position *c*)

F = feed rate, effective for blocks P through Q

S = spindle speed, effective for blocks P through Q

Notice from the block diagram, that it is only necessary to enter position coordinates of the first (b) through last block (c) of the previous rough cycle, which define the finished profile. This will cause an automatic return to the earlier part of the program for the coordinates needed for the completion of the process removing allowances U and W.

N10 G50 S1000

....

....

N30 G71 P35 Q65 U.02 W.01 D1000 F.014

....   F.010

....   F.006

....   F.008

N50 M01

N55 G50 S1500

....

....

N70 G70 P35 Q65

....

N80 M30

Attention: In the above program, values of the feed used in information blocks following function G71 are ignored for the roughing cycle, but they are valid for function G70.

Sample Program O0012: Finishing Cycle (Figure 3-37)

O0012

N10 G50 S2000

N15 T0200 M42

N20 G96 S600 M03
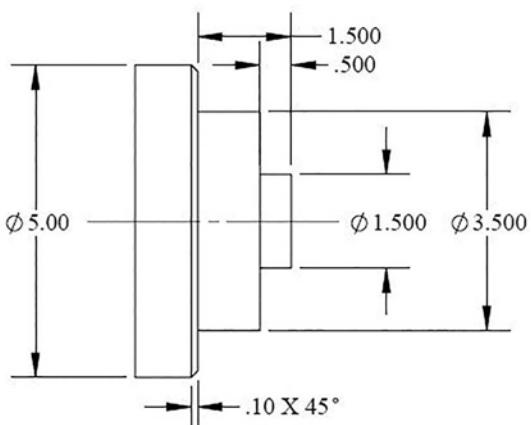
Figure 3-37
Drawing for Finishing Cycle (G70)
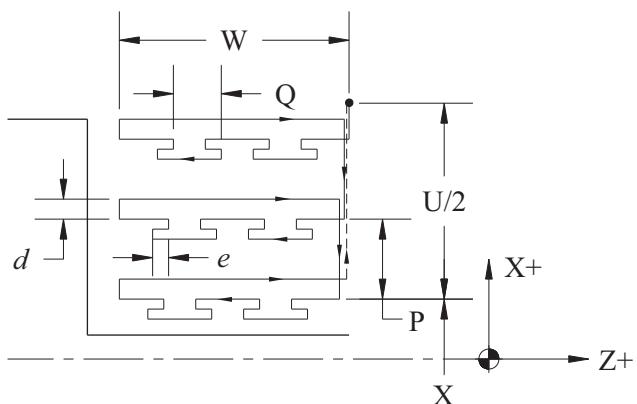
N25 G00 G54 X5.2 Z.2 T0202 M08

N30 G71 U.12 R.05

N30 G71 P35 Q65 U.04 W.005 F.012

N35 G00 X1.5

N40 G01 Z–.5 F.010

N45 X3.5 F.008

N50 Z–1.5 F.010

N55 X4.8 F.009

N60 X5.1 Z–1.7 F.004

N65 X5.2 F.010

N70 G00 X7.75 Z.5 M09

N75 T0200 M05

N80 G28 U0 W0

N85 M01

N90 T0400 M42

N95 G50 S2000

N100 G96 S700 M03

N105 G00 G54 X5.2 Z.2 T0404 M08

N110 G70 P35 Q65

N115 T0400 M09

N120 G28 U0.0 W0.0 M05

N125 M30

PECK DRILLING CYCLE (G74)

The most common use of this function is for drilling deep holes that require an interruption in the feed in order to break long stringy chips. In spite of its name, however, this function may be applied to cylindrical or face cutting (along the Z-axis) of grooves that exhibit hard breaking chips as well. A block diagram of this function, as well as the movements of the tool, is illustrated in Figure 3-38.

Figure 3-38
Peck Drilling Cycle (G74)
Diagram

The amount of the clearance (indicated by d in Figure 3-38) is set by a system parameter. The amount of the return (indicated by e in the Figure 3-38) is also set by a system parameter. The following is a block diagram for function G74 using a double block:

G74 R . . .

G74 X(U). . . Z(W). . . P. . . Q. . . R. . . F. . . S. . .

where:

in the first block:

R = retract amount of the tool after each cut

in the second block:

X = diameter of the workpiece at the bottom of the groove

U = distance between the starting and end points (in an incremental system)

Z = final Z cut depth in the absolute system

W = Z distance from start to finish cut depth in the incremental system

P = depth of each cut for X-axis (no sign)

Q = depth of each cut for Z-axis (no sign)

R = retract amount of the tool at the bottom of the cutting

F = cutting feed rate

S = spindle speed

The following is a block diagram for function G74 using a single block:

G74 X(U). . . Z(W). . . I. . . K. . . D. . . F. . . S. . .

X = diameter of the workpiece at the bottom of the groove

U = distance between the starting and end points along the X-axis (in an incremental system)

Z = final Z cut depth in the absolute system

W = Z distance from start to finish cut depth in the incremental system

I = depth of cut per side in X direction (no sign)

K = depth of cut per side in Z direction (no sign)

D = retract amount of the tool at the bottom of the cutting

F = cutting feed rate

S = spindle speed

In the following example program, this function is applied for drilling a deep hole using the double block format.

Sample Program O0013: Peck Drilling Cycle (Figure 3-39)
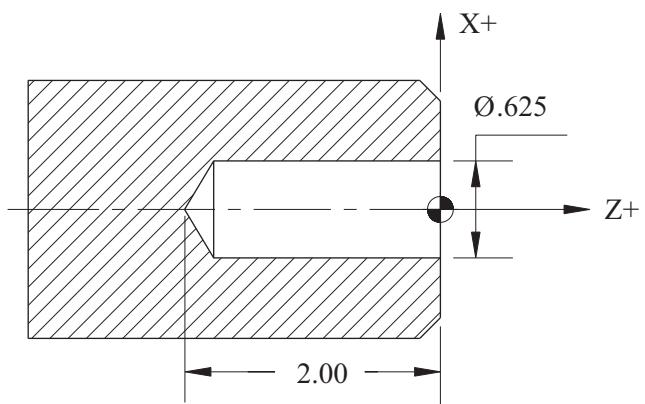
O0013

N10 G50 S1000

N15 T0700 M42

Figure 3-39
Drawing for Peck Drilling Cycle
(G74)

N20 G97 S800 M03

N25 G00 G54 X0.0 Z.2 T0707 M08

N30 G74 R.1

N35 G74 Z–2.0 K.550 F.007

N40 T0700 M09

N45 G28 U0.0 W0.0 M05

N50 M30

At the end of drilling (i.e., block N35), the tool automatically returns to the starting position of Z.2. Advantages of the peck drilling cycle include chip-breaking and cooling of the cutting drill tip.

GROOVE CUTTING CYCLE (G75)

Function G75, in its form, is very similar to function G74. Its difference is seen in the direction of the tool movement, which is opposite that in function G74. Function G75 is used for cutting grooves that require an interrupted cut along the X-axis (Figure 3-40).

The following is a block diagram for function G75 using a double block:

G75 R. . .
G75 X(U). . . Z(W). . . P. . . Q. . . R. . . F. . . S. . .

The following is a block diagram for function G75 using a single block:

G75 X(U). . . Z(W). . . I. . . K. . . D. . . F. . . S. . .

All notations assumed for this function are defined exactly as in function G74.

Sample Program O0014: Groove Cutting Cycle (Figure 3-41)

O0014

(RIGHT SIDE OF INSERT CUTS)

N10 G50 S2000

N15 T0600 M42
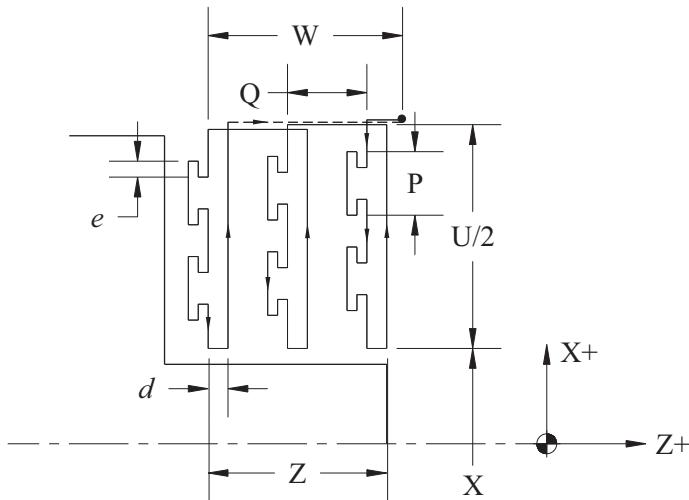
Figure 3-40
Groove Cutting Cycle (G75)
Diagram
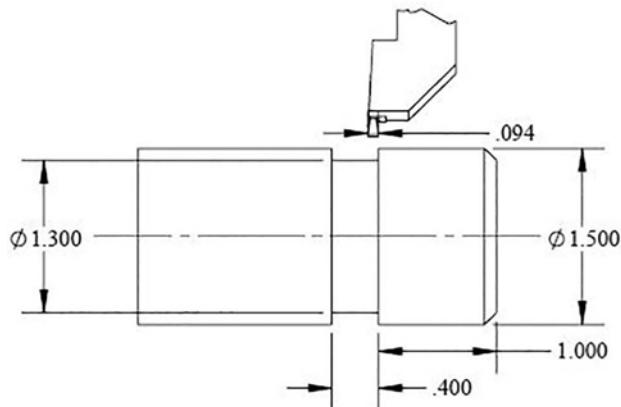
Figure 3-41
Drawing for Groove Cutting
Cycle (G75)
Tool, Courtesy Kennemetal Inc.

N20 G96 S400 M03

N25 G00 G54 X1.55 Z–1.156 T0606 M08

N30 G75 R.15

N30 G75 X1.3 Z–1.4 P.1 Q.075 F.005

N35 G00 X1.75 Z.5

N40 T0600 M09

N45 G28 U0.0 W0.0 M05

N50 M30
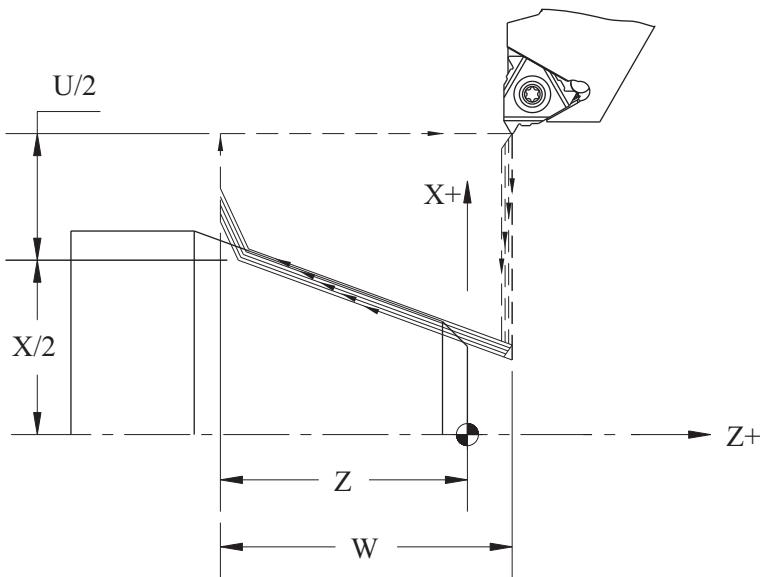
Note: At the end of the cycle, the tool returns to the starting point in both functions G74 and G75.

MULTIPLE THREAD CUTTING CYCLE (G76)

The G76 multiple thread cutting cycle is used on most modern controls, in place of the outdated G32 and G92. All of the information needed to complete the desired

Figure 3-42
Multiple Thread
Cutting Cycle (G76)
Diagram

thread is input in either one or two blocks, depending on the control, rather than multiple blocks in the former. By inputting the appropriate data for a particular type of thread in the program blocks, the number of cutting passes is automatically calculated by the control. Because of the limited number of blocks required, this method is very easy to program and edit. Figure 3-42 represents the basic functions of the cycle.

The following is a block diagram for function G76 using a double block:

G76 P. . . Q. . . R. . .

G76 X. . . Z. . . R. . . P. . . Q. . . F. . .

where:

in the first block:

P = uses a six digit entry (P010000) of three pairs as follows:

- the first two digits specify the number of finishing cuts (01–99)
- three and four specifies the number of thread leads required for gradual pull-out (0.0–9.9 times the lead) without a decimal point entry (00–99)
- five and six denote the angle of the thread (only 00, 29, 30, 55, 60 and 80 degrees are allowed).

Q = the minimum cutting depth

R = finishing allowance (allows a decimal point)

in the second block:

X = Final diameter of the thread

Z = Final end position of the thread along the Z-axis, (can be specified as an incremental distance using address W)

R = incremental distance from the thread starting to ending, as a radial value (used for tapered threads)

    P = single thread height (always a positive radial value, without a decimal point)

    Q = depth of cut for the first threading pass (always a positive radial value, without a decimal point)

    F = feed rate, lead of thread

The following is a block diagram for function G76 using a single block:

    G76 X(U). . . Z(W). . . I. . . K. . . D. . . F. . . A. . . P . . .

where:

    X(U) = diameter of thread core (last diameter cut)

    Z(W) = full length of the cut thread (end of thread position)

    I = difference of thread radius (+ or −) from start to finish (for tapered threading)

    K = single thread height (always positive)

    D = depth of first threading cut (always positive)

    A = thread angle (matches the included angle of the threading insert and is always positive)

    F = feed rate, lead of thread

Notes:

1. *Changing the spindle speed or feed rate via override, while within the threading cycle, is not effective.*

2. *A dry-run condition is applicable and effective.*

3. *The use of constant rotational programming G97 S…. is required!*

4. *The depth of a first cut D is approximately .003 to .018, depending on machining conditions.*

5. *With a small value of first cut, the number of passes increases and, inversely, with a greater value of the first cut, the number of passes decreases.*

6. *The selection of factor D depends on the type of thread, the material, and the tool tip.*
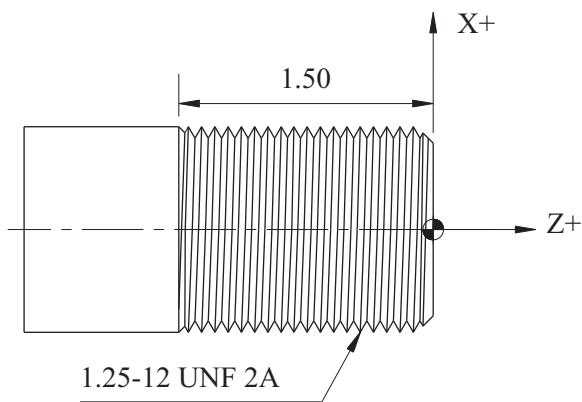
Figure 3-43
Drawing for Multiple Thread Cutting Cycle (G76)

Sample Program O0015: Multiple Thread Cutting Cycle (Figure 3-43)

> **O0015**
> **N10 G50 S2000**
> **N15 T0800 M42**
> **N20 G97 S800 M03**
> **N25 G00 G54 X1.35 Z.5 T0808 M08**
> **N30 G76 X1.2 Z–1.5 I0.0 K.055 D.012 A60 E083333**
> **N35 G00 X1.5 Z.5**
> **N40 T0800 M09**
> **N45 G28 U0 W0 M05**
> **N50 M30**

Note: Depending on the value of the first cut specified by D, a certain number of passes will be obtained, as defined by the parameters that are set for the machine.

PROGRAMMING FOR THE TOOL NOSE RADIUS

In all of the programming examples examined thus far, the effect of the tool nose radius was not compensated, even if value was entered to the R column of the Tool Data register. In the case of straight facing along the X-axis or straight turning along the Z-axis, the tool contact point is the same as the measured point on the tool nose. Tool Nose Compensation is required, however, for chamfers, tapers, and arcs. Without using compensation, the aforementioned would be undercut or overcut by a small amount that varies depending on the radius of the tool nose. Before the use of Geometry Offsets became common on controllers, these small differences had to be calculated manually (a cumbersome process that was error prone) and input into the program code. The most advantageous and effective method used today is described in the next section.

APPLICATION OF TOOL NOSE RADIUS COMPENSATION (TNRC) G40, G41, AND G42

Tool nose radius compensation is the most commonly used programming method today. TNRC adds ability to control the dimensional quality of geometry features, when using indexable inserts or any tool with a nose radius. It also aids in programming by making it necessary only to program the workpiece profile without shifting the tool path to compensate for the tool nose radius. When a program is properly written using functions G41 or G42, setting the values correctly in the offset register of the control will produce a dimensionally accurate workpiece. TNRC also makes corrections to the tool path when wear compensation has been adjusted for the affected tool. See Part 2, *Adjusting Wear Offsets for Turning Centers* for details. The following explanations are given for the critical information needed for using function G41 and G42.

TOOL NOSE RADIUS AND TIP ORIENTATION

In Figures 2-30 and 2-31 of Part 2, *CNC Machine Operation*, the Offset Display Screen and Offset/Geometry Display Screen for tool and geometry offsets are shown.

The last two columns in this register are used to input the values for the Tool Nose Radius (R) and the Tool Tip Orientation (T). These data are setup related, but have a very direct effect on the use of Tool Nose Radius Compensation (TNRC) in programming. If TNRC (G41 or G42) is used in the program, these data must be set accurately; otherwise, the programmed tool path will not generate the expected geometry. Straight facing or turning cuts (parallel to either the X- or Z-axes) do not require the use of TNRC, but in the case of tapered or circular contouring cuts and radii, TNRC is essential.

Figures 3-44 and 3-45 detail the necessary information to select the proper setting for the Tool Tip Orientation (T), in the offset register, for a rear turret lathe.

In Figures 3-44 and 3-45, the tool tip is represented and the contact point is identified. The drawing should help with the selection of the appropriate Tool Tip Orientation, (T) based on the direction the contact point is pointing. In Figure 3-45, the same concept is superimposed onto the actual tool nose and the arrows indicate the contact point and direction. Thus, the selection of the Tool Tip Orientation number is determined by the direction the tool nose is pointing. Use number 0 or 9 for programming the center of the tool nose.

Figure 3-44
Tool Tip Orientation Guide

CALLING G41 OR G42 IN THE PROGRAM

Selecting Which Code to Use

Selection of either G41 or G42 is based on the side of the part profile that the tool needs to be on in order to create the desired results. Think of the part profile as the centerline of a highway. Then, based on the direction of travel, decide which side of the road to drive on. For the left side, G41 is selected and, for the right side, G42 is used. This procedure may be applied whether the cut is internal or external. Facing uses G41 and Turning the outside diameter uses G42. In the case of Turning the inside diameter/bores, G41 is used.

The block format is:

G41 X(U). . . Z(W). . .

G42 X(U). . . Z(W). . .

Tool Nose Radius

Figure 3-45
Superimposed Tool Tip
Orientation Guide

Initiating Tool Nose Radius Compensation with G41 or G42

To initiate the use of tool nose radius compensation, the G41 or G42 should occur on a G00 rapid positioning move that is at least .100 of an inch away from the part profile. This move need only be in one axis direction, but it can include both. Please note how the G42 is initiated in the following example program for the part shown in Figure 3-46.

Figure 3-46
Drawing for Tool Nose
Radius Compensation
(G41 and G42)

Ending Use of G41 or G42 with Function G40

    In order to end the use of G41 or G42, TNRC may be cancelled by using function G40. When the machine is first started, the G40 command is active by default. To program the cancellation of tool nose radius compensation, the command is generally input on a move that is in a departing vector from the machined profile. This move may be either G00 or G01, and cancellation may be initiated with the G28 command, where the compensation will be cancelled upon reaching the intermediate point. Please note how the G40 is used to cancel tool nose radius compensation in the following example program for the part shown in Figure 3-46. In this example, only the finished profile is programmed.

    Note: Tool Nose Radius Compensation G41 and G42 will be used for the remaining examples where applicable.

    Sample Program O0016: Tool Nose Radius Compensation (TNRC) Using G41 or G42

        O0016
        N10 G50 S2000
        N15 T0400 M42
        N20 G96 S500 M03
        N30 G54 G00 G54 X3.0 Z.5 T0404 M08
        N35 G41 X1.1 Z.2
        N40 G01 Z0.0 F.02
        N45 X–.04 F.01
        N50 G00 Z.03
        N55 G42 X.940
        N60 G01 Z0.0
        N65 X1.0 Z–.03 F.005
        N70 Z–1.2 F.008

> N75 X2.1
>
> N80 X2.399 Z–1.24 F.005
>
> N85 X2.5 Z–1.29
>
> N90 Z–2.5 F.008
>
> N95 X2.850
>
> N100 G00 G40 X4.0 Z.5 M09
>
> N105 T0400 M05
>
> N110 G28 U0.0 W0.0
>
> N115 M30

Notes on using G41 and G42:

If the values for Tool Nose Radius (R) and the Tool Tip Orientation (T) are omitted in the offset register, the desired results will not be obtained (0 radius is assumed in this case). Note the change from G41 for facing, to G42 for the profile.

PROGRAMMING EXAMPLES FOR TURNING CENTERS

CUTTING TOOL DESCRIPTIONS FOR TURNING CENTER PROGRAMMING EXAMPLES

The charts on pages 144 and 145 describe the tools used in the remaining Turning Center program examples:

APPLICATION OF G00 AND G01 IN BOTH ABSOLUTE AND INCREMENTAL SYSTEMS

Preliminary Considerations:
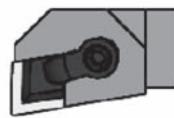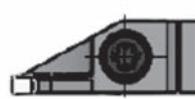
1. For the following example, the workpiece is rough turned with an allowance given for finishing; therefore, the use of only one finishing tool is programmed.

2. The part profile is programmed using Tool Nose Radius Compensation (G41 or G42).

3. The left side of the workpiece was faced in a previous operation.

4. The 55-degree Finish Turning Tool (T0404) is used.

Sample Program O0017: Application of G0 and G01 Using the Absolute System Coordinate System (Figure 3-47)

> O0017
>
> N10 G50 S1500
>
> N15 T0400 M42
>
> N20 G96 S600 M03
>
> N30 G00 G54 X4.25 Z.2 T0404 M08
>
> N35 G41 Z0.0 F.03
>
> N40 G01 X–.05 F.01
>
> N45 G00 Z.03
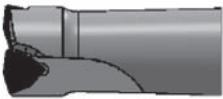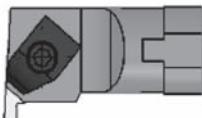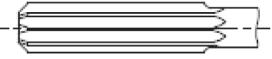
Part 3 Programming CNC Turning Centers

CUTTING TOOL DESCRIPTIONS FOR TURNING CENTER EXAMPLES

Tool Graphic	Tool Number	Tool Description
	T0101	#5 High Speed Steel Center Drill
	T0202	O.D. Rough Turning Tool 80 Degree Diamond Insert .031 Tool Nose Radius
	T0303	I.D. Rough Boring Tool 80 Degree Diamond Insert .031 Tool Nose Radius
	T0404	O.D. Finish Turning Tool 55 Degree Diamond Insert .015 Tool Nose Radius
	T0505	I.D. Finish Boring Tool 55 Degree Diamond Insert .01531 Tool Nose Radius
	T0606	O.D. Grooving Tool .005 Tool Nose Radius .118 Wide
	T0707	Standard Drill Sizes Vary as Needed
	T0808	O.D. Threading Tool 60 Degree
Tool Graphics, Courtesy Kennemetal NOVO		

CUTTING TOOL DESCRIPTIONS FOR TURNING CENTER EXAMPLES

Tool Graphic	Tool Number	Tool Description
	T0909	Inserted Drill Tool .625 Minimum Diameter
	T1010	Alternate O.D. Tool
	T1111	I.D. Groove Tool .005 Tool Nose Radius .118 Wide
	T1212	Part-Off Blade .125 Wide
	Alternate T0909	I.D. Threading 60 Degree Inserted
	Alternate T1111	Reamer Any Required Size
	Alternate T1111	Tap Any Required Size
Tool Graphics, Courtesy Kennemetal NOVO		
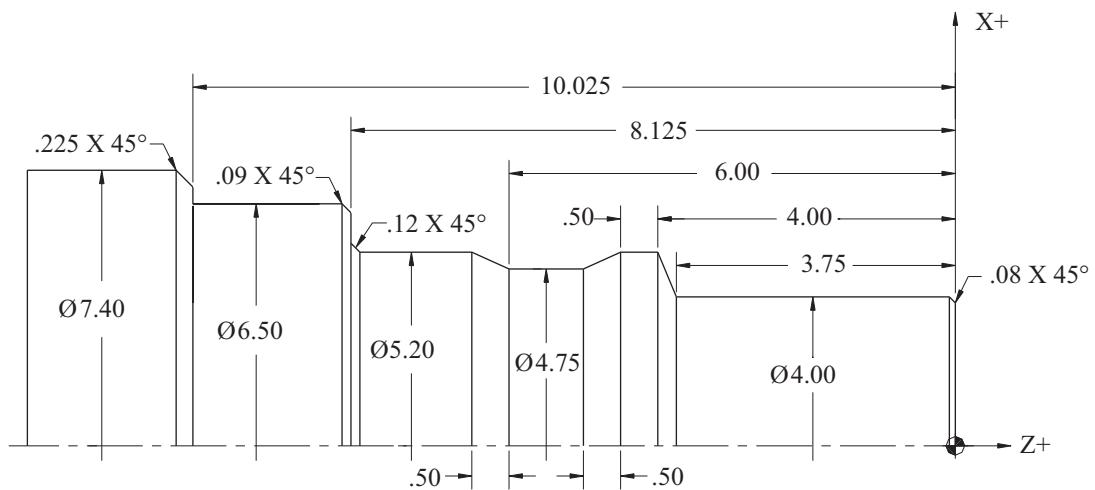
Figure 3-47
Drawing for Sample Program O0017

N50 G42 X3.84
N55 G01 Z0.0
N60 X4.0 Z–.08
N65 Z–3.75
N70 X5.2 Z–4.0
N75 Z–4.5
N80 X4.75 Z–5.0
N85 Z–6.0
N90 X5.2 Z–6.5
N95 Z–8.005
N100 X5.44 Z–8.125
N105 X6.32
N110 X6.5 Z–8.215
N115 Z–10.025
N120 X6.95
N125 X7.4 Z–10.35
N130 G00 G40 X7.5
N135 Z.1 M09
N140 T0400 M05
N145 G28 U0 W0
N150 M30

Part 3 Programming CNC Turning Centers

Sample Program O0171: Application of G00 and G01 Using the Incremental Coordinate System

O0171

N10 G50 S1500

N15 T0400 M42

N20 G96 S600 M03

N30 G00 G54 X4.25 Z.2 T0404 M08

N35 G41 Z0.0 F.03

N40 G01 U–4.3 F.01

N45 G00 W.1

N50 G42 U3.89

N55 G01 W–.03

N60 U.16 W–.08

N65 W–3.67

N70 U1.2 W–.25

N75 W–.5

N80 U–.45 W–.5

N85 W–1.0

N90 U.45 W–.5

N95 W–1.505

N100 U.24 W–.12

N105 U.88

N110 U.18 W–.09

N115 W–1.81

N120 U.45

N125 U.55 W–.325

N130 G00 G40 X7.5

N135 Z.1 M09

N140 T0400 M05

N145 G28 U0 W0

N150 M30

Sample Program O0172: Application of G0 and G01, Using both Absolute and Incremental Coordinate Systems

O0172

N10 G50 S1500

N15 T0100 M42

N20 G96 S600 M03

N30 G00 G54 X4.25 Z.2 T0404 M08

N35 G41 Z0.0

N40 G01 X–.03 F.01

N45 G00 W.03

N50 G42 X3.84

N55 G01 Z0

N60 X4.0 W–.08

N65 Z–3.75

N70 X5.2 W–.25

N75 Z–4.5

N80 X4.75 W–.5

N85 Z–6.0

N90 X5.2 W–.500

N95 Z–8.005

N100 U.24 W–.12

N105 X6.32

N110 X6.5 W–.09

N115 Z–10.025

N120 X6.95

N125 X7.4 W–.325

N130 G00 G40 X7.5

N135 Z.1 M09

N140 T0400 M05

N145 G28 U0

N150 M30

SUBPROGRAM APPLICATION

By using subprograms, we can easily program parts that have identical repetitive features. An advantage of subprograms is that fewer lines of programming code are needed. The following is a simple example of how a subprogram might be used for cutting multiple grooves:

Sample Program O0018: Subprogram Application (Figure 3-48)

O0018

N10 G50 1600

N15 T0600 M03

N20 G97 S833 M41

N35 G00 G54 X1.3 Z–.625 T0606 M08
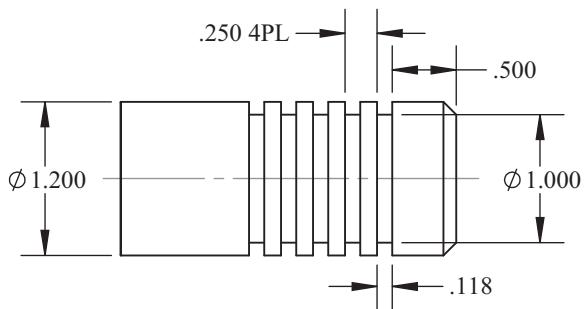
Figure 3-48
Drawing for Sample Program O0018

N40 M98 P0181 L5
N45 T0600 M9
N50 G28 U0 W0 M05
N55 M30

Subprogram for Sample Program 18: Subprogram Application

O0181
N10 G01 X1.0 F.004
N15 G00 X1.3
N20 W–.25
N25 M99

In block N40 of the main program, the subprogram O0181 is called and executed 5 times. When N25 of the subprogram is reached, the execution returns to the main program at block N40 and proceeds to its end.

EXAMPLE OF MAKING A TAPER PIPE THREAD

For this example, a 3/4-inch National Pipe Thread (NPT) will be programmed on a 4140 steel part (Figure 3-49). Before programming can begin, several calculations are necessary, as shown below:
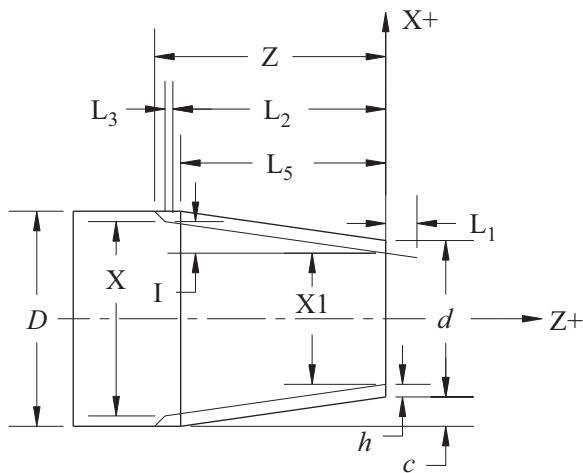
Figure 3-49
Drawing for Making a Taper
Pipe Thread

The thread dimensional data are drawn from *Machinery's Handbook.*

$D = 1.05$, diameter of pipe

$L_2 = .5457$, effective thread length

$L_5 = .4029$, length from the end of the pipe to the intersection of the taper and the outside diameter of the straight turned portion $h = .0571$, height of the thread

Lead = 14

The angle of the thread is equivalent to 60° (Figure 3-50).

Taper of the thread on the diameter is 3/4 of an inch per foot (Figure 3-51). The calculation below is needed to determine the value used in calculating the diameters for the part in Figure 3-49.

$$\tan \alpha = \frac{.375}{12.0} = .03125$$

$$\alpha = 1.7899° = 1°47'$$

Figure 3-50
Thread Angle

The small diameter at the right end of the part is calculated as diameter d (Figure 3-52).

$$d = D - 2_c$$

$$\tan \alpha = \frac{c}{L_5}$$

$$c = L_5 \times \tan \alpha = .4029 \times .03125 = .0126$$

$$d = 1.5 - 2 \times .0126 = 1.0248$$

The following is a calculation of the resulting length L for the programmed section during threading (Figure 3-53).

$$L = L_1 + L_2 + L_3$$

$$L_3 = \frac{S \times P}{1800} = \frac{727 \times .0714}{1800} = .028$$

$$a = 3.2$$

$$L_1 = L_3 \times a = .028 \times 3.2 = .0896 \approx .090$$

$$L = .09 + .5457 + .028 = .6637$$

Figure 3-51
Taper per Foot

To determine the incremental displacement along the X-axis from the start to the end of the taper calculate the value of I (Figure 3-54):

$$\tan \alpha = \frac{I}{L}$$

$$I = L \times tg \, \alpha = .6637 \times .03125 = .0207$$

$$I = .0207$$

To determine the large diameter of the taper, calculate the diameter X (Figure 3-55):
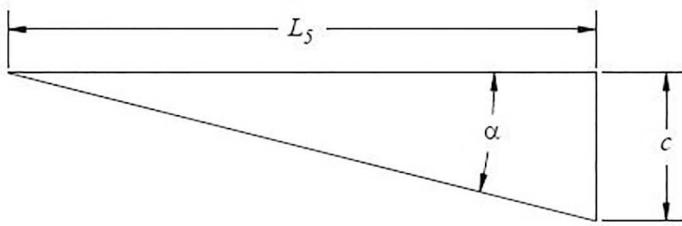
Figure 3-52
Small Diameter Taper
Calculation
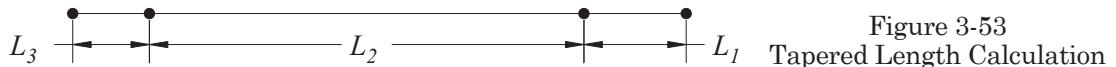
Figure 3-53
Tapered Length Calculation
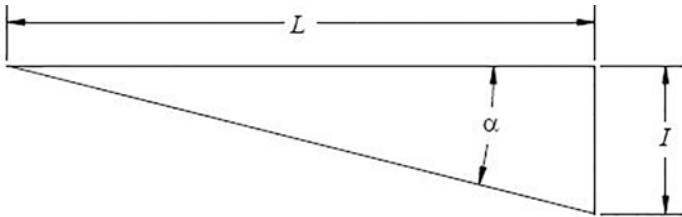
Figure 3-54
Drawing for the
Calculation of I
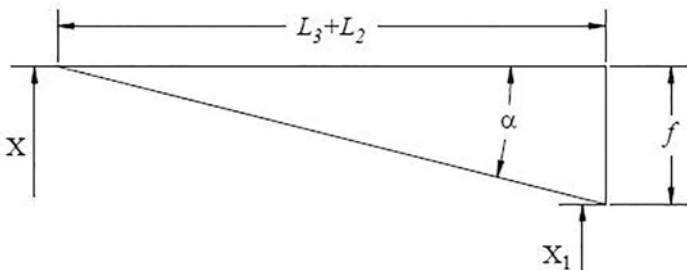
Figure 3-55
Drawing for the
Calculation of X

$$X = X_1 + 2f$$

$$X_1 = d - 2h = 1.0248 = 2 \times .0571 = .9106$$

$$\tan \alpha = \frac{f}{L_2 + L_3}$$

$$f = (L_2 + L_3) \times \tan \alpha = (.5457 + .028) \times .03125 = .0179$$

$$X = X_1 + 2f = .9106 + 2 \times .0179 = .9464$$

$$X = .9464$$

In the examples that follow, an abbreviated version of the CNC Setup Sheet is used (as described in Part I of this text). This provides an aid in identifying multiple tools and other specific setup information.

Part 3 Programming CNC Turning Centers

Machine: Turning Center			Program Number: O0019	
Workpiece Zero: X, <u>Centerline</u> Z, <u>Part Face</u> Setup Description: Material = 4140 Alloy Steel AISI 1300				
Tool #	Tool Orientation #	Description	Insert Specification	Comments
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	400 SFM
T0808	8	O.D. Threading Tool	60 Degree Thread Forming Insert	300 sfm

Sample Program O0019: Making a Taper Pipe Thread (Chart 0019)

O0019
(OD TURNING TOOL)
N10 G50 S1500
N15 T0200 M41
N20 G96 S400 M03
N25 G00 G54 X1.2 T0202 M08
N30 G41 Z.2
N35 G01 Z0.0 F.03
N40 X−.06 F.008
N45 G00 W.03
N50 G42 X1.0266
N55 G01 Z0.0
N60 X1.050 Z−.4336 F.007
N65 G00 G40 U.06 Z.2
N70 T0200 M09
N75 G28 U0 W0
N80 M01
(OD THREADING TOOL)
N85 G50 S1000
N90 T0800 M41
N95 G97 S1000 M03
N100 G00 G54 X1.15 Z.09 T0808 M08
N105 G76 X.9464 Z−.5737 I−.0207 K.0571 A60 D.012 F.071428
N110 T0800 M09
N115 G28 U0.0 W0.0
N120 M30

Note: Function G00 is used in block N25 and is valid until block N35, where it is replaced by a linear feed move.

EXAMPLE OF TURNING BAR STOCK

In this example a simple outside diameter is to be turned with a .03 radius and a .04 chamfer (Figure 3-56). After turning is completed, a part-off tool cuts the part to length.

Sample Program O0020: Turning Bar Stock (Chart O0020)

O0020

(OD TURNING TOOL)

N10 G50S 4000

N15 T0200 M42

N20 G96 S600 M03

N25 G00 G54 X1.1 Z.2 T00202 M08

N30 G01 G41 Z0 F.03

N35 X–.06 F.007

N40 G00 W.030

N45 G42 X1.1 Z.2

N50 G71 P55 Q85 U.03 W.003 D0800 F0.016

N55 X.565 Z0.0

N60 G03 X.625 Z–.03 I0 K–.03

N65 G01 Z–1.0

N70 X.92

N75 X1.0 Z–1.04

N80 Z–2.375

N85 G00 G40 X1.1
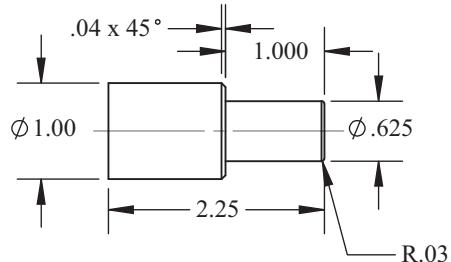
Figure 3-56
Drawing for Turning Bar
Stock Example

Machine: Turning Center		Program Number: O0020		
Workpiece Zero: X, <u>Centerline</u> Z, <u>Part Face</u> Setup Description: Material = 1" Diameter Carbon Steel 1100 bar stock Stock projection from collet face 2.5"				
Tool #	Tool Orientation #	Description	Insert Specification	Comments
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	600 SFM
T1212	8	Part-Off Blade	.118 Wide	300 SFM

N90 G70 P55 Q85 F.007

N95 T0200 M09

N100 G28 G40 U0 W0 M05

N105 M01

(OD PART-OFF)

N110 G50 S3000

N115 T1200 M42

N120 G96 S300 M03

N125 G00 G54 X1.1 Z.2 T1212 M08

N130 Z–2.3750

N135 G01 X.10 F.008

N140 X0.0

N145 M05

N150 G00X1.1

N155 T0500 M09

N155 G28 U0 Z2.0 M05

N160 M30

PROGRAMMING EXAMPLE FOR MAKING A BUSHING

The diagram in Figure 3-57 is for an aluminum bushing that will be machined using the following steps.

First operation

In the first operation, the left side of the part is machined (Figure 3-58). This operation roughs the 2.0" diameter hole and turns the O.D. to a length of 4.22" inch. The 2.10" bore is finished also. The length is finish machined in the next operation.

Sample Program O0021: Making a Bushing, Operation 1 (Figure 3-58 and Chart O0021)

O0021

(ROUGH OD TURNING TOOL)

N10 G50 S1000

N15 T0200 M41

N20 G96 S800 M03

N25 G00 G54 X4.4 Z.5 T0202 M08

N30 G41 Z.01

N35 G01 X–.03 F.014

N40 G00 G40 W.15

N45 G42 X3.69

N50 G01 Z.01

SECTION A-A

DETAIL B

DETAIL A
SCALE 1 : 1
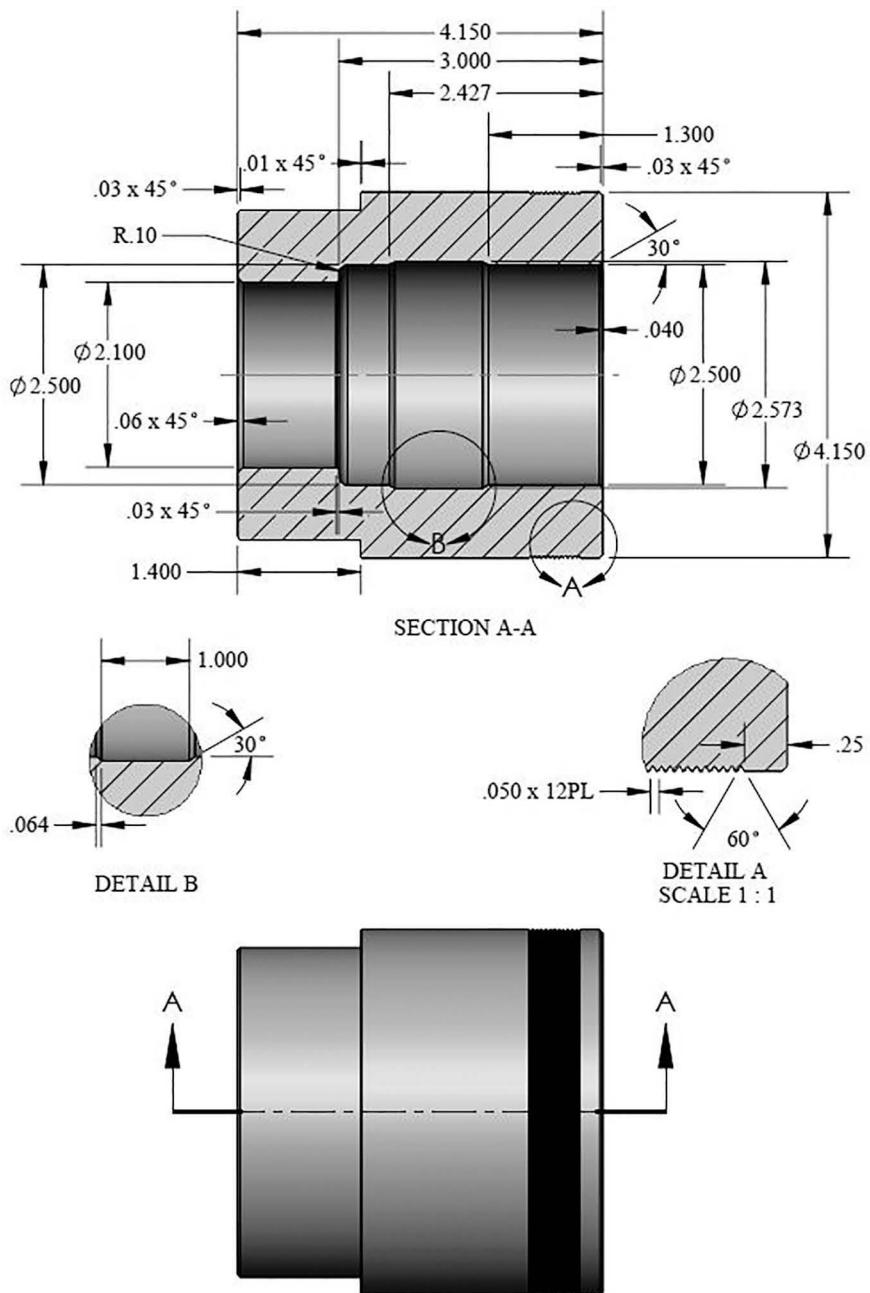
Figure 3-57
Drawing for Making a Bushing Example

Part 3 Programming CNC Turning Centers
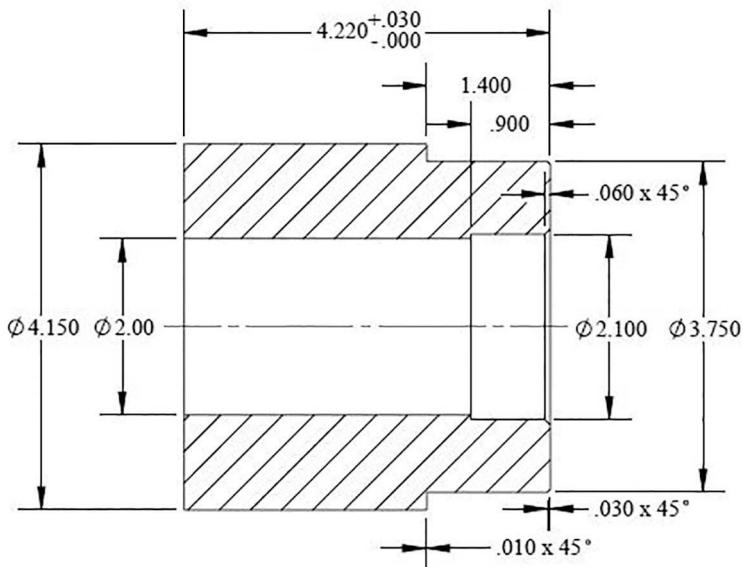
Figure 3-58
Drawing for Making a
Bushing, Operation 1

Machine: Turning Center			Program Number: O0021	
Workpiece Zero: X, Centerline Z, Part Face				
Setup Description OP 1: Material = 4.25 Diameter x 4.5 long 6061 Aluminum bar stock				
Stock projection from chuck face 2.5"				
Tool #	Tool Orientation #	Description	Insert Specification	Comments
T0101	7	#5 HSS Center Drill		200 SFM
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	800 SFM
T0303	2	I.D. Boring Tool	80 Degree Diamond .031 Nose Radius	800 SFM
T0404	3	O.D. FINISH Turning Tool	55 Degree Diamond .015 Nose Radius	1200 SFM
T0707	7	2.0 Diameter Drill		200 SFM 5.0 Minimum Extension

N55 G01 X3.77 W–.03 F.012
N60 Z–1.39
N65 X4.13
N70 X4.17 W–.01
N75 Z–1.5
N80 G00 G40 X4.35 M09
N85 T0200

N90 G28 U0 W0 M05

N95 M01

(#5 CENTER DRILL)

N100 T0100 M42

N105 G97 S1600 M03

N110 G00 G54 X0.0 Z.2 T0101 M08

N115 G01 Z–.4 F.006

N120 G00 Z1.0 M09

N125 T0100

N130 G28 U0 W0 M05

N135 M01

(2-INCH DIAMETER DRILL)

N140 T0700 M41

N145 G97 S400 M03

N155 G00 G54 X0.0 Z.2 T0707 M08

N160 G74 Z–4.8 K1.125 F.008

N165 G00 Z1.0 M09

N170 T0700

N175 G28 U0 W0 M05

N180 M01

(FINISH OD TURNING TOOL)

N185 T0400 M42

N190 G96 S1200 M03

N200 G00 G41 X3.85 Z.1 T0404 M08

N205 G01 Z0.0 F.02

N210 X1.8 F.01

N215 G00 G40 W.15

N220 G42 X3.69

N225 X3.75 W–.03 F.004

N230 Z–1.4 F.001

N235 X4.13

N240 X4.15 W–.01 F.004

N245 G00 G40 Z–4.35

N250 T0400 M09

N255 G28 U0 W0 M05N260 M01

(ID BORING BAR)

N265 T0300 M41

N270 G96 S800 M03

N280 G00 G41 X2.16 Z.1 T0303 M08

N285 G01 Z0 F.004

N290 G01 X–2.1 W–.06

N295 Z–.9

N300 X1.9

N305 G00 G40 Z.1 M09

N310 T0300

N315 G28 U0 W0 M05

N320 M30

Second Operation

In the second operation, the remaining portion of the bushing will be machined from the opposite end (Figure 3-59).

Sample Program O0212, Making a Bushing Operation 2 (Chart O0212)

O00212

(ROUGH OD TURNING TOOL)

N10 T0200 M42

N15 G96 S800 M03

N25 G00 G54 X4.5 Z.2 T0202 M08

N30 G01 Z.02 F.03

N35 X1.8 F.01

N40 G00 G42 X4.09 W.05

N45 G01 Z–0.0

N50 X4.15 W–.03

N55 Z–2.85

N60 G00 G40 X4.35 Z.1

N65 T0200 M09

N70 G28 U0 W0 M05

N75 M01

(ROUGH ID BORING BAR)

N80 T0300 M42

N85 G96 S800 M03

N95 G00 G54 X4.35 T0303 M08

N100 G41 X1.9 Z.1

N105 G71 U.08 R.05
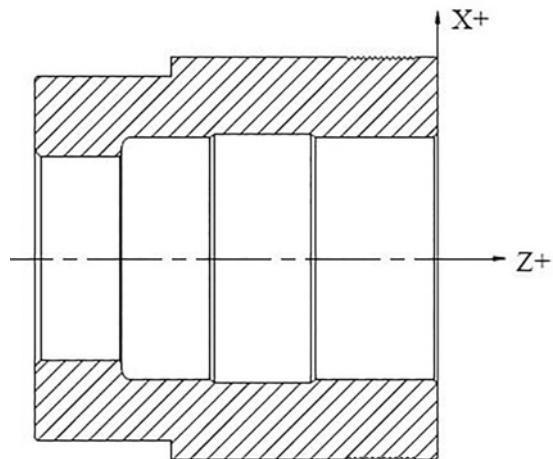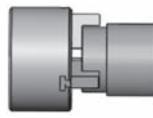
Figure 3-59
Drawing for Making a Bushing,
Operation 2

Part 3 Programming CNC Turning Centers

Machine: Turning Center		Program Number: O0212	

Workpiece Zero: X, <u>Centerline</u> Z, <u>Part Face</u>
Setup Description OP 2: Soft Jaws pre-cut to match 3.75 diameter .800 Deep

Tool #	Tool Orientation #	Description	Insert Specification	Comments
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	800 SFM
T0303	2	I.D. Boring Tool	80 Degree Diamond .031 Nose Radius	800 SFM 3.3 minimum projection
T0404	3	O.D. FINISH Turning Tool	55 Degree Diamond .015 Nose Radius	1200 SFM
T0505	2	I.D. Boring Tool	55 Degree Diamond .015 Nose Radius	1200 SFM 3.3 minimum projection
T0808	8	O.D. Thread Tool 60 Degree		300 SFM

N110 G71 P115 Q170 U.03 W.015 F.019
N115 G01 Z0.0 F.01
N120 U–.04 Z–.04
N125 Z–1.3
N130 X2.5 W–.064
N135 W–1.0
N140 X2.5 W–.064
N145 Z–2.9
N150 G03 X2.3 Z–3.0 I–.1 K0.0
N155 X2.04
N160 X2.1 W–.03
N165 G00 G40 X1.9
N170 Z.1
N175 T0300 M09
N180 G28 U0 W0 M05
N185 M01
(FINISH OD TURNING TOOL)
N190 T0200 M42
N195 G96 S1200 M03

N205 G00 G54 X4.35 T0202 M08

N210 G41 Z0.0

N215 G01 Z0.0 F.03

N220 X2.35 F.008

N225 G00 G40 W.1

N230 G42 G00 X4.09

N235 X4.15 W–.03 F.004

N240 Z–2.85 F.008

N245 G00 G40 X4.5 Z.1

N250 T0200 M09

N255 G28 U0.0 W0.0 M05

N260 M01

(FINISH ID BORING BAR)

N265 T0500 M42

N270 G96 S1200 M03

N275 G00 X2.0 T0505 M08

N275 G41 X1.9 Z.1

N280 G70 P115 Q170 F.012

N285 G00 Z1.

N290 T0500 M09

N295 G28 U0.0 W0.0 M05

N300 M01

(SPECIAL V-FORM TOOL)

N305 T0800 M42

N310 G96 S300 M03

N315 G00 X4.25 Z–.267 T0808 M08

N320 M98 P5 L12

N325 G00 Z1.

N330 T0800 M09

N335 G28 U0.0 W0.0 M05

N340 M30

Subprogram for Operation 2

O0005

N10 G01 X–4.09 F.003

N15 G00 X–4.25

N20 W–.05

N25 M99

Notes for the second operation of the bushing program:

Due to the use of TNRC in the I.D. sections of the program, there are minimal calculations required and ultimately fewer program blocks. If dimensions are not given in the engineering design drawing, some calculations will be necessary for programming chamfers and tapers.

EXAMPLE ILLUSTRATING THE APPLICATION OF FUNCTIONS G72 AND G75

Program O0022 demonstrates the use of functions G72 and G75 for the diagram in Figure 3-60. The rough diameters are turned (G72) and then the groove is cut in the 3.1" diameter (G75).

Figure 3-60
Drawing for Example of Application
of G72 and G75

Sample Program O0022: Application of G72 and G75 (Chart O0022)

> **O0022**
> **(ROUGH OD TURNING TOOL)**
> **N10 G50 S1500**
> **N15 T0200 M41**
> **N20 G96 S250 M03**
> **N25 G00 G54 X4.8 Z.5 T0202**
> **N30 G41 Z.1 M08**

Machine: Turning Center		Program Number: O0022		
Workpiece Zero: X, Centerline Z, Part Face Setup Description OP 1: Material = 4.75 Diameter x 2.45 long 316 Stainless Steel bar stock Stock projection from chuck face 3.0"				
Tool #	Tool Orientation #	Description	Insert Specification	Comments
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	250 SFM
T0404	3	O.D. FINISH Turning Tool	55 Degree Diamond .015 Nose Radius	400 SFM
T0606	8	O.D. Grooving Tool	.118 Wide .015R	250 SFM

N35 G01 Z0.0 F.04

N40 X–.06 F.012

N45 G00 Z.1

N50 G42 X4.8

N55 G72 U.04 W.005 P60 Q115 D.12 F.012

N60 G00 Z–1.47

N65 G01 X4.75

N70 U–.02 Z–1.45 F.004

N75 X3.5 F.008

N80 Z–.53

N85 U–.03 Z–.5 F.004

N90 X1.75 F.008

N95 G01 Z–.025 F.008

N100 U–.025 Z0.0 F.004

N110 Z.1

N115 G00 G40 X4.85

N120 T0200 M09

N125 G28 G40 U0.0 W0.0

N130 M01

(FINISH OD TURNING TOOL)

N135 G50 S2200

N140 T0400 M41

N145 G96 S400 M03

N150 G00 G54 X2.0 Z.5 T0404

N155 G41 Z.1

N165 G01 Z0.0 F.04

N170 X–.06 F.008

N175 G00 Z.1

N180 G42 X4.8

N185 G70 P60 Q115

N190 T0400 M09

N195 G28 G40 U0.0 W0.0 M05

N200 M01

(OD GROOVING TOOL)

N205 G50 S2000

N210 T0600 M41

N215 G96 S250 M03

N220 G00 G54 X3.55 Z.2 T0606 M08

N225 Z–1.118

N230 G75 X3.1 I.05 F.004

N235 G00 X3.55

N240 Z–1.15

N245 G75 X3.1 I.05 F.004

N250 G00 X3.55

N255 Z–1.16

N260 G01 U–.01 W.01 F.003

N265 G00 X3.55

N270 Z–1.108

N275 G01 X3.1 F.008

N280 U–.01 W–.01 F.003

N285 Z1.0 M09

N290 T0600 M09

N295 G28 U0.0 W0.0 M05

N300 M30

In block N230, when the tool reaches a diameter of X3.1, the tool position automatically returns to the starting point because of the G75 cycle. As a result, consecutive block N235 (X3.55) is actually a safety clearance block because on some older types of controls, the automatic return may not appear. In cycle G75, some symbols such as K, D, and Z may be omitted if any are equal to zero (K = 0, D = 0). The value of Z does not change and is omitted as well.

COMPLEX PROGRAM EXAMPLE

Program O0023 and O0232 demonstrate the use of several programming functions covered in this chapter for the multiple operation part shown in Figure 3-61.

FIRST OPERATION

   a. The order calls for 1,025 pieces of material with a 3.125 diameter and a length of 6.625.

   b. Complete all the dimensions on the left side of the part, along with a length of 6.60 inch, which leaves .100 inch for the finish pass on the second operation on the opposite end.

   The following are the initial program data for the first operation:

1. The center drill has a maximum diameter of .3 inch and the r/min is calculated as follows:

   Cutting speed for the steel is:

    $V = 80$ ft/min

$$n = \frac{12 \times V}{\pi \times d} = \frac{12 \times 80}{3.14 \times .3} = 1019$$
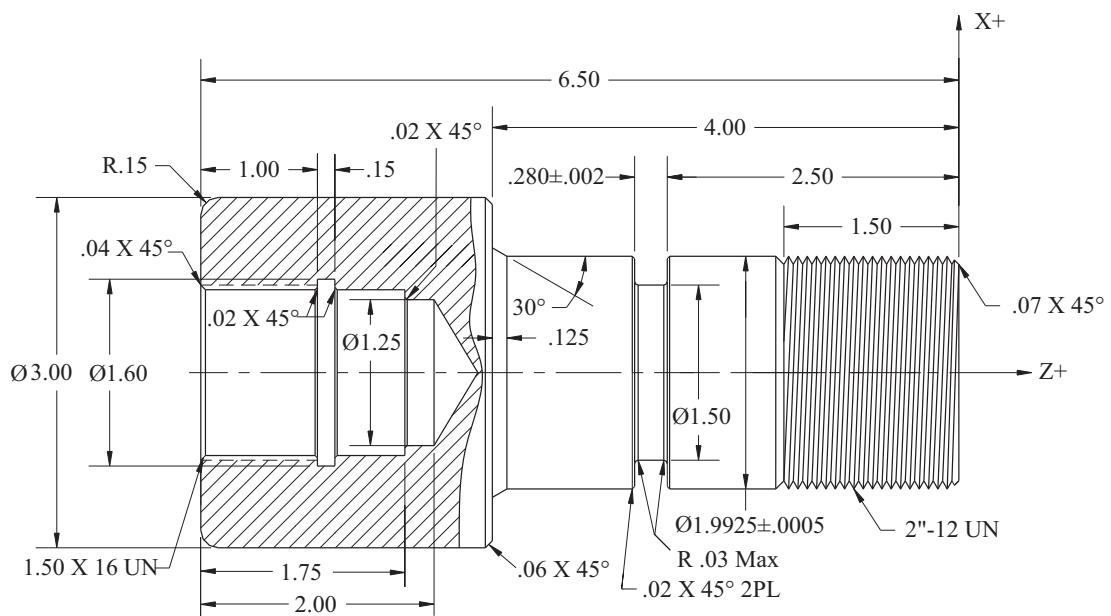
Figure 3-61
Drawing for Complex Program Example

2. The 1.25 inch diameter drill bit tool r/min is calculated as follows:

$V = 80$ ft/min

$$n = \frac{12 \times V}{\pi \times d} = 244$$

3. The cutting speeds for the boring bar, I.D groove, and the internal thread are $V = 300$ ft/min due to the unfavorable cutting conditions and the likelihood that vibrations will be present.

4. The minor diameter for the 1.5-16 internal thread is obtained from the *Machinery's Handbook* and is from 1.432 minimum to 1.446 maximum.

Sample Program O0023: Complex Program, Operation 1 (Chart O0023)

O0023

(ROUGH OD TURNING TOOL)

N10 G50 S2000

N15 T0200 M42

N20 G96 S500 M04

N25 G00 G54 X3.2 Z.5 T0202 M08

N30 G41 Z.1

N35 G01 Z.02 F.05

N40 X–.03 F.014

N45 G00 W.03

Part 3 Programming CNC Turning Centers

Machine: Turning Center			Program Number: O0023	
Workpiece Zero: X, <u>Centerline</u> Z, <u>Part Face</u> Setup Description OP 1: Machine the left end including all interanl work Material: Brass				
Tool #	**Tool Orientation #**	**Description**	**Insert Specification**	**Comments**
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	500 SFM
T0101	7	#5 HSS Center Drill		80 SFM
T0707	7	1.25 Diameter Drill		80 SFM
T0303	2	I.D. Boring Tool	80 Degree Diamond .031 Nose Radius	400 SFM
T0404	3	O.D. FINISH Turning Tool	55 Degree Diamond .015 Nose Radius	600 SFM
T0505	2	I.D. Boring Tool	55 Degree Diamond .015 Nose Radius	300 SFM
T1111	2	I.D. Grooving Tool	.118 wide .005 Radius	300 SFM
T0909	2	I.D. Thread Tool 60 Degree		300 SFM

N50 G42 X2.7

N55 G01 Z0.0 F.014

N60 G03 X3.0 W–.15 I0.0 K–.15

N65 Z–2.8

N70 G00 G40 X3.2 Z1.0 M09

N75 G28 U0.0 W0.0 T0200

N80 M01

(#5 CENTER DRILL)

N85 T0100 M42

N90 G97 S1019 M03

N95 G00 G54 X0.0 Z.2 T0101 M08

N100 G01 Z–.3 F.006

N105 G00 Z.5

N110 T0900 M09

N115 G28 U0.0 W0.0 M05

N95M01

(1.25 DIAMETER DRILL)

N120 T0707 M42

N125 G97 S244 M03

N130 G00 G54 X0.0 Z.1 T0707 M08

N135 G74 Z-2.360 K.150 F.008

N140 T0700 M09

N140 G28 U0.0 W0.0

N145 M01

(ROUGH ID BORING BAR)

N150 G50 S2000 M03

N155 T0300 M42

N160G96 S400 M04

N165 G00 G54 X3.1 Z.1 T0303 M08

N170 G42 X1.125

N175 G71 U–.03 W.005 P175 Q215 D1000 F.012

N180 G00 X1.519

N185 G01 Z0.0

N190 X1.439 W–.04

N195 Z–1.75 F.007

N200 X1.21

N205 1.29 W–.02 F.004

N210 G00 X1.125

N215 G40 Z.1

N220 T0300 M09

N225 G28 U0.0 W0.0 M05

N230 M01

(FINISH OD TURNING TOOL)

N235 G50 S2000

N240 T0400 M42

N245 G96 S600 M03

N250 G00 G54 X3.1 Z.1 T0404 M08

N255 G42 X2.7

N260 G01 Z0.0 F.014

N265 G03 X3.0 W–.15 I0.0 K–.15

N270 Z–2.8

N275 G00 G40 X3.2 Z1.0 M09

N280 T0400 M09

N285 G28 U0.0 W0.0 M05

N290 M01

(FINISH ID BORING BAR)

N295 G50 S2000

N300 T0500 M42

N305 G96 S600 M03

N310 G00 G54 X3.1 Z.1 T0505 M08

N315 G00 G41 X1.125

N320 G70 P175 Q215

N325 T0500 M09

N330 G28 U0.0 W0.0 M05

N335 M01

(ID GROOVING TOOL)

N340 G50 S2000

N345 T1100 M42

N350 G96 S300 M04

N355 G00 G54 X3.1 Z.2 T1111M08

N360 X1.125

N365 G01 Z–1.15 F.03

N370 X1.6 F.004

N375 G00 X1.4

N380 Z–1.118

N385 G01 X1.6 F.008

N390 G00 X1.4

N395 Z–1.17

N400 G01 U–.02 W–.02 F.004

N405 G00 X1.6

N410 Z–1.012

N415 G01 U–.02 W.02 F.004

N420 G00 X1.4

N425 Z1.0

N430 T1100 M09

N435 G28 U0.0 W0.0 M05

N440 M01

(ID THREADING TOOL)

N445 T0900 M42

N450 G97 S764 M03

N455 G00 G54 X1.340 Z.5 T0909 M24

N460 G76 X1.5 Z–1.075 A60 I0 K.040 D.0140 F.0625

N465 T0900 M23

N465 G28 U0.0 W0.0 M9

N470 M30

Notes: During the process of drilling, notice that function G97 is applied, which corresponds to a constant spindle speed. It is used in blocks N90 and N125, because when a centerline tool such as a drill is used, the assigned coordinate for its position is X0.0. In block N20 function G96, constant cutting speed is instated. If G96 were to remain active, the spindle speed for the X position of zero would calculate to infinity and the machine would be limited to its maximum programmable r/min; obviously, this is not practical. In this program, the value of the spindle speed is limited, with the value of S assigned to function G50.
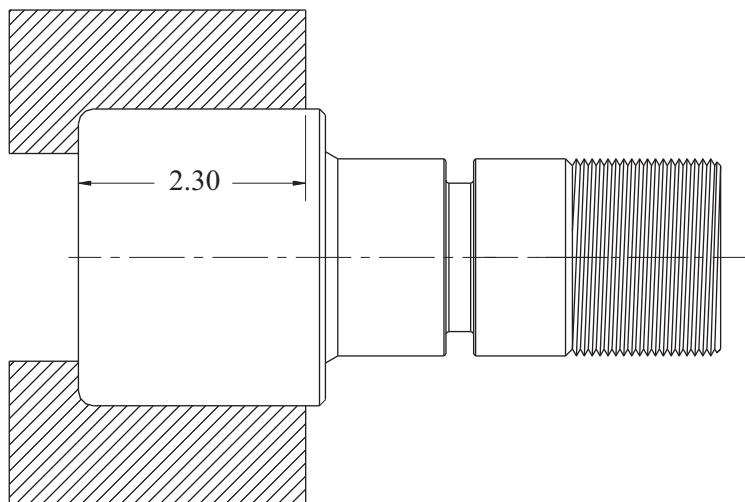
Second Operation

The setup for operation 2 shown in Figure 3-62 is needed for the remaining OD work on the part. Program O0232 uses the functions to rough and finish turn the OD, complete the groove and threaded portion of the part.

Sample Program O0232: Complex Program Operation 2 (Chart O0232)

O0232

(ROUGH OD TURNING TOOL)

N10 G50 S2000

N15 T0200 M42

N20 G96 S500 M03

N25 G00 G54 X3.2 Z.2 T0202 M08

Figure 3-62
Drawing for Complex
Program, Operation 2

2.30

Part 3 Programming CNC Turning Centers

Machine: Turning Center			Program Number: O0232	
Workpiece Zero: X, <u>Centerline Z</u>, <u>Part Face</u> Setup Description OP 2: Machine the right end including all external work Material: Brass				
Tool #	Tool Orientation #	Description	Insert Specification	Comments
T0202	3	O.D. Turning Tool	80 Degree Diamond .031 Nose Radius	500 SFM
T0404	3	O.D. FINISH Turning Tool	55 Degree Diamond .015 Nose Radius	600 SFM
T0606	8	O.D. Grooving Tool	.118 Wide .005 Nose Radius	400 SFM
T0808	8	O.D. Threading Tool	60 Degree	285 SFM

N30 G42 X3.1 Z.1

N35 G01 Z.005 F.05

N40 X–.03 F.014

N45 G00 Z.1

N50 G42 X3.2

N55 G71 U.03 W.005 P60 Q100 D1500 F.014

N60 G00 X1.86

N65 G01 Z0.0 F.008

N70 X1.9925 W–.07

N75 Z–3.875

N80 X2.1105 Z–4.0

N85 X2.88

N90 X3.0 W–.06

N95 X3.0

N100 G00 G40 X3.2 Z.1

N105 T0200 M09

N110 G28 U0.0 W0.0 M05

N115 M01

(FINISH OD TURNING TOOL)

N120 G50 S2000

N125 T0400 M42

N130 G96 S600 M03

N135 G00 G54 X3.2Z.5 T0404 M08

N140 G41 Z.1

N145 G01 Z0.0 F.05

N150 G00 X–.03 F.008

N155 G40 X3.2 Z.1

N160 G60 P Q100

N165 T0400 M09

N170 G28 U0.0 W0.0 M05

N175 M01

(OD GROOVING TOOL)

N180 G50 S2000

N185 T0600 M42

N190 G96 S400 M03

N195 G00 G54 X2.1 Z.2 T0606 M08

N200 Z–2.78

N205 G01 X1.50 F.006N210 G00 X2.1

N215 Z–2.687

N220 G01 X1.50 F.006

N225 G00 X2.1

N230 Z–2.618

N235 G01 X1.50 F.006

N240 G00 X2.1

N245 Z–2.598

N250 G01 X1.9925 F.008

N255 U–.02 W–.02

N260 G00 2.1

N265 Z–2.8

N270 G01 X1.9925 F.008

N275 U–.02 W.02

N280 G00 X2.1

N285 T0600 M09

N290 G28 U0.0 W0.0 M05

N295M01

(OD THREADING TOOL)

N300 T0800 M42

N305 G97 S573 M03

N310 G00 G54 X2.1 Z.5 T0808 M08

N315 G76 X1.896 Z–1.5 I0 K.0485 A60 D140 F.083333

N320 T0800 M09

N325 G28 U0.0 W0.0 M05

N330 M30

EXAMPLE OF CUTTING A THREE-START THREAD
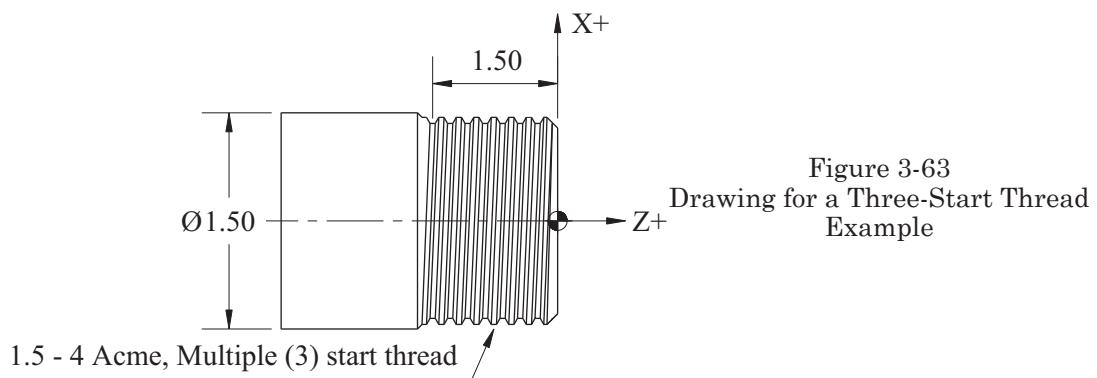
This example requires a pre-machined major thread diameter. The example uses a special Acme thread form insert in the T0808 turret location. The following example is given for machining a three-start 1.5-4 Acme thread, as illustrated in Figure 3-63.

Sample Program O0024: Cutting a Three-Start Thread

O0024

N10 G50 S1500

N15 T0800 M41

N20 G97 S750 M03

N25 G00 G54 X1.58 Z.7 T0808

N30 M98 P1235 L3

N35 T0800 M09N40 G28 U0.0 W0.0 M05

N45 M30

Subprogram for Program O0024

O1235

N10 G00 W–.0833

N15 G76 X1.395 Z–1.5 I0 K.0525 A60 D140 F.250000

N20 M99

Figure 3-63
Drawing for a Three-Start Thread
Example

1.5 - 4 Acme, Multiple (3) start thread

Part 3 Study Questions

1. In NC (tape) controlled machines, the M02 command rewinds the tape to its start. How is a CNC machine commanded to return the program to its start?

    a. M03

    b. M08

    c. M30

    d. M05

2. The term *modal commands* means that once the command is initiated, it stays in effect until cancelled or replaced by another command from the same group.

    T or F

3. In programming, tool function is commanded by the four digits that follow the letter address T (example T0404). What do these two sets of numbers refer to?

4. Of the following choices, which is the best method for compensating for dimensional inaccuracies caused by tool deflection or wear?

    a. Geometry Offsets

    b. Wear Offsets

    c. Tool Length Offsets

    d. Absolute Position Register

5. When the rough turning cycle G71 is used, which letters identify the amount of stock to leave for the finish pass, X-axis, and Z-axis respectively?

    a. U and V

    b. X and Z

    c. P and Q

    d. U and W

6. Sequence (N) numbers in programs may be omitted entirely and the program will execute without any problem.

    T or F

Part 3 Programming CNC Turning Centers

7. The default (at machine start-up) feed rate on lathes is typically measured in

   a. Cutting speed

   b. Inches per revolution in/rev

   c. Inches per minute in/min

   d. Constant surface speed

8. The advantage of using G96 Constant Cutting Speed in turning is that as the diameter changes (position of the tool changes in relation to the centerline), the r/min increases or decreases to accomplish the programmed cutting speed.

   T or F

9. The preparatory function G50 relates to two things in programming:

   a. Absolute positioning setting and maximum spindle r/min setting

   b. Absolute position setting and constant spindle r/min

   c. Work offset and tool offsets

   d. Absolute position setting and constant surface speed

10. When incremental programming is required in turning diameters and facing, which letters identify the axis movements respectively?

   a. I and J

   b. U and W

   c. I and K

   d. U and V

11. Which M Codes are used to activate and deactivate a subprogram respectively?

   a. M03 and M04

   b. M2 and M30

   c. M98 and M99

   d. M41 and 42

Part 3 Programming CNC Turning Centers

12. What M-Code is listed in the last line of a subprogram?

      a. M30

      b. M99

      c. M98

      d. M02

13. What two letters identify the incremental distance from the starting point to the arcs center in G02/G03 programming?

      a. I and J

      b. I and K

      c. J and K

      d. X and R

14. When programming arcs with modern CNC controllers I, J and K can be omitted and replaced by R. If the arc is greater than 180°, what must be added to the R command?

15. Fixed cutting cycle G90 is limited to orthogonal tool movements (no contouring or chamfers are allowed).

      T or F

16. When using the Multiple Repetitive Cycle, Rough Cutting Cycle, G71, U and W represent the stock allowance for finishing. What cycle is required to remove the stock allowance?

      a. G90

      b. G73

      c. G76

      d. G70

Part 3 Programming CNC Turning Centers

17. What is the major reason for selection of the Pattern Repeating Cycle, G73, as opposed to the Rough Cutting Cycle, G71?

      a. It is required to make the finish allowance cut on X and Z.

      b. This cycle is well suited where an equal amount of material is to be removed from all surfaces.

      c. G73 is limited to orthogonal cuts while G71 can cut radius and chamfers.

      d. G71 is limited to rough cutting only while G73 is required to remove stock allowances.

18. When programming arc and angular cuts using tool nose radius compensation G41 and G42, which is used for facing and turning when the tool is mounted above the part centerline?

      a. G41 facing, G42 turning

      b. G42 facing, G41 turning

      c. Neither. The tool nose radius amount must be calculated and programmed to compensate.

      d. G41 for facing if contours or angles are involved, G42 for turning if contours or angles are involved.

19. Tool tip orientation needs to be identified in the controller when TNRC is used to program functions G41 or G42. How is this information input?

      a. The number is input by R into the program

      b. The number is input by T into the offset page

      c. The number is input by R into the offset page

      d. The number is input by T into the program

20. Using the tool figures 89, 91, 95, and 96, identify the tool tip identification numbers.

Part 3 Programming CNC Turning Centers

21. When programming the G76 multiple thread cutting cycle, the letter address I allows for programming:

        a. Tapered thread cutting

        b. Elimination of taper in thread cutting

        c. Both a and b

        d. only a

22. Name one reason that writing programs using Tool Nose Radius Compensation (TNRC) are advantageous.

PART 4

Programming CNC Machining Centers

OBJECTIVES:

1. Learn G-Codes associated with machining center programming.

2. Learn M-Codes associated with machining center programming.

3. Apply the proper use of feeds and speeds within machining center programs.

4. Learn how to properly use coordinate systems for programming the machining center.

5. Learn program structure.

6. Learn how to use canned cycles.

7. Learn how to use Cutter Diameter Compensation (G40, G41, and G42).

8. Examine several practical examples of machining center programs.

Programming of CNC machining centers will be presented in this section. In Part 1 of this text, the basic steps necessary for programming were identified. These steps will be followed to create programs. Individual programming words and codes will be defined and demonstrated with examples. This section is written as if the program manuscript is being created manually, line-by-line. This experience will help you understand the programs and how they are made, which will make it easier for you to edit existing programs. In reality, though, the most common method for creating programs today is by using CAD/CAM software, which is introduced in Part 5 and Part 6. In some cases, conversational programming (see Part 7 and Part 8) is done on the shop floor. In the programming process, we need to input the cutting tool information.

TOOL FUNCTION (T-WORD)

Proper selection and application of cutting tools and work holding ensure that the programs you create produce desired results. Refer to Part 1 "Cutting Tool Selection", *Machinery's Handbook*, and the cutting tool and insert manufacturers' ordering catalogs and online resources—like Kennametal's "NOVO" application—for detailed tool information. The tool function is utilized to prepare and select the appropriate tools from the tool magazine. In order to describe the tool in the program, the address T is followed by one or more digits that refer to the pocket numbers in the tool magazine.

Example

T05 = tool number 5

Please note that on most modern controllers, it is not necessary to use the leading zero in a tool call; thus, T5 has the same meaning as T05.

TOOL CHANGES

A tool change is specified in the program by the miscellaneous function M06. To initiate a tool change, first call for the desired tool number. Then use the miscellaneous function M06 to execute the change.

Example

N10 T01 (TOOL IN THE READY POSITION)
N15 M06 (ACTUAL TOOL CHANGE)
N20 T02 (NEXT TOOL IN THE READY POSITION)
N25 . . .
N30 M06 T03
N35 . . .

In block N10, the requested tool is positioned in the tool magazine to a ready state (waiting position). In block N15, tool T01 is automatically installed into the spindle and, in the meantime, tool T02 is positioned to a ready state in the tool magazine for the next tool change. In block N20, tool T01 performs programmed work. In block N30, a tool change takes place. Tool T02 is installed into the spindle, while tool T01 is returned to the tool magazine and tool T03 is positioned to a ready state in the tool magazine for the next tool change. In block N35, tool T03 performs the programmed work.

In cases where Random Access Tool Changers are used, it is not necessary to stage the next tool. When a tool is finished, it is replaced with the next required tool, which is positioned in the same (now empty) pocket. Assigned tool pockets are not required except at initial job setup. The system keeps track of where each tool is placed in the magazine. These systems attain very fast tool changes because no wait is involved in tool magazine rotation.

Please note that on most modern controllers, it is not necessary to use the leading zero in a miscellaneous function call; thus, M6 has the same meaning as M06. Also note that on machines with umbrella style tool magazines, the tool call (T01) and tool change call (M06) must be stated together.

Example

T01 M06

FEED FUNCTION (F-WORD)

The F-word is utilized to determine the work feed rates. This program word, which is used to establish feed rate values, precedes a numeric input for the feed amount in inches per minute (in/min), millimeters per minute (mm/min), inches per revolution (in/rev), or millimeters per revolution (mm/rev). The value that is set by this command stays effective until changed by reentering a new value for the F-word.

Example

F20 = a feed rate of 20 inches per minute (in/min)
F.006 = a feed rate of .006 inches per revolution (in/rev)

If the function G20 (data in inches) is active, the notation F20 refers to the feed speed of 20 in/min, whereas, with the function G21 (data in millimeters), the notation F20 refers to a feed rate of 20 mm/min.

With rapid traverse G00, the machine traverses at the highest possible feed rate that is specified in control memory (actual rates depend on the design of the machine). In the case of feed rate motion G01, the value of the feed rate must be accurately specified. The machine default setting for feed rate is inches per minute in the United States.

Examples

F20.0 = 20.0 inches of feed per minute

F500. = 500 millimeters of feed per minute

F2.0 = 2.0 inches of feed per minute

F50. = 50 millimeters of feed per minute

F.02 = twenty thousandths inch of feed per revolution

F0.50 = millimeters of feed per minute

F.002 = two thousandths inch of feed per revolution

F0.050 = millimeters of feed per minute

The proper methods for selecting and calculating feed rates were discussed in Part 1, "Metal Cutting Factors". Also, *Machinery's Handbook,* the cutting tool and insert manufacturers' ordering catalogs, and online resources—like Kennametal's "NOVO" application for detailed tool information—provide excellent feed and speed data.

SPINDLE SPEED FUNCTION (S-WORD)

The letter address S is followed by a specified value in revolutions per minute (rev/min).

Example

S2100 (specifies 2100 r/min)

One or more digits following the letter address S are used for the value of the rotational speed. If S0 is input, this command deactivates the spindle rotation and leaves it in a neutral position so that the spindle can be rotated manually, depending on the machine tool. Having this ability is quite useful, especially when using a coaxial indicator for dialing-in the X and Y locations of the workpiece to establish Workpiece Zero coordinates. The value of S is specified in revolutions per minute (r/min).

PREPARATORY FUNCTIONS (G-CODES)

G-codes are the preparatory functions that identify the type of activities the machine will execute. A program block may contcain one or more G-codes.

The letter address G and specific numerical codes allow communication between the controller and the machine tool. This combination of letters and numerical values is commonly called G-Code. In order to perform a specific machining operation, a G-Code must be used. There are two types of G-Codes: modal and non-modal. Modal commands

remain in effect, in multiple blocks, until they are changed by another command from the same group. Non-modal commands are in effect only for the block in which they are stated.

Examples

Group 00 (non-modal, one-shot commands)

Group 01 (modal commands)

There are several different groups of G-codes as indicated in column 2 of Chart 4-1. One code from each group may be specified in an individual block. If two codes from the same group are used in the same block, the first will be ignored by the control and the second will be executed. Those G-Codes that are active upon startup of the machine are indicated by an asterisk (*) in the chart.

A Safety Block is commonly placed in the first line of the program where cancellation codes are used to cancel all G-Codes that have been in effect in prior programs (described later in this chapter, *Explanation of the Safety Block*). Typically they are: (G40) cutter compensation cancel; (G49) tool length compensation cancel; (G80) canned cycle cancel; and (G17) XY plane selection. These cancellations are important because of modal commands that stay in effect until either cancelled or replaced by a command from the same group. It is also a good idea to insert a Safety Block after tool change blocks, in case you need to rerun a single operation from within the program. By inserting the Safety Block, there is no chance of modal commands remaining active. The digits following the letter address G identify the action of the command for that block.

If the measurement system is changed—for example, from the G20 inch to G21 metric system—then G21 will be in effect at the next startup of the machine or until a program call of G20 is executed. On machines sold in the United States, the parameters are set to default to the G20-inch system upon startup.

MISCELLANEOUS FUNCTIONS (M-CODES)

Miscellaneous functions or M-Codes (Chart 4-2) control the working components that activate and deactivate coolant flow, spindle rotation, the direction of the spindle rotation, and similar activities.

PROGRAMMING OF CNC MACHINING CENTERS IN ABSOLUTE AND INCREMENTAL SYSTEMS

These two coordinate measuring systems—absolute and incremental—are used to determine the values that are input into the programming code for the X, Y, and/or Z program words. They can also be used in the same manner for rotary axes A, B, and/or C.

ABSOLUTE COORDINATE PROGRAMMING (G90) OF THE MACHINING CENTER

In absolute programming, all coordinate values are relative to a fixed origin of the coordinate system. Axis movement in the positive direction does not require inclusion of the sign whereas negative movements do require signs.

Part 4 Programming CNC Machining Centers

Chart 4-1 Preparatory Functions (G-Codes) Specific To Machining Centers

Code	Group	Function	Code	Group	Function
*G00	01	Rapid Traverse Positioning	*G80	09	Canned Cycle Cancellation
*G01	01	Linear Interpolation	G81	09	Drilling Cycle, Spot Drilling
G02	01	Circular and Helical Interpolation CW (clockwise)	G82	09	Drilling Cycle, Counter Boring
G03	01	Circular and Helical Interpolation CCW (counterclockwise)	G83	09	Deep Hole Peck Drilling Cycle
G04	00	Dwell	G84	09	Tapping Cycle
G09	00	Exact Stop	G85	09	Reaming Cycle
G10	00	Programmable Data Setting	G86	09	Boring Cycle
G11	00	Programmable Data Setting Cancellation	G87	09	Back Boring Cycle
*G15	17	Polar Coordinate Cancellation	G88	09	Boring Cycle
G16	17	Polar Coordinate System	G89	09	Boring Cycle
*G17	02	XY Plane Selection	*G90	03	Absolute Programming
G18	02	ZX Plane Selection	*G91	03	Incremental Programming
G19	02	YZ Plane Selection	G92	00	Setting for the Work Coordinate System or Maximum Spindle r/min
G20	06	Input in Inches	*G94	05	Feed per Minute
G21	06	Input in Millimeters	G95	05	Feed per Revolution
*G22	04	Stored Stroke Limit ON	G96	02	Constant Surface Speed Control
G23	04	Stored Stroke Limit OFF	*G97	02	Constant Surface Speed Control Cancel
G27	00	Reference Point Return Check	*G98	10	Canned Cycle Initial Level Return
G28	00	Reference Point Return	G99	10	Canned Cycle R=Level Return
G29	00	Return From Reference Point			
G30	00	Return to Second, Third, and Fourth Reference Point			
G33	01	Thread Cutting			
G37	00	Automatic Tool Length Measurement			
*G40	07	Cutter Compensation Cancel			
G41	07	Cutter Compensation, Left			
G42	07	Cutter Compensation, Left			
G43	08	Tool Length Offset Compensation positive (+) direction			
G44	08	Tool Length Offset Compensation negative (-) direction			
G45	00	Tool Offset Increase			
G46	00	Tool Offset Decrease			
G47	00	Tool Offset Double Increase			
G48	00	Tool Offset Double Decrease			
G49	08	Tool Length Offset Compensation Cancel			
*G50	11	Scaling Cancel			
G51	11	Scaling			
G52	00	Local Coordinate System Setting			
G53	00	Machine Coordinate System Setting			
*G54-59	14	Work Coordinate System Selection (G54 default)			
G60	00	Single Direction Positioning			
G63	15	Tapping Mode			
G64	15	Cutting Mode			
G68	16	Rotation of Coordinate System			
*G69	16	Cancellation of Corrdinate System Rotation			
G73	09	Peck Drilling Cycle			
G74	09	Reverse Tapping Cycle			
G76	09	Fine Boring Cycle			

NOTES:

The items marked with an asterisk (*) are active upon startup of the machine or are reinstated when the RESET button has been pressed. Check the specific manufacturer Operation Manual for your application.

For G00, G01, G90 and G91 the initial code that is active is determined by a parameter setting. These are typically G01 and G90 for startup condition.

G-Codes from groups 00 are one-shot G-Codes.

Multiple G-Codes from different groups can be specified in the same block. If more thatn one from the same group is specified only the last G-Code listed will be active.
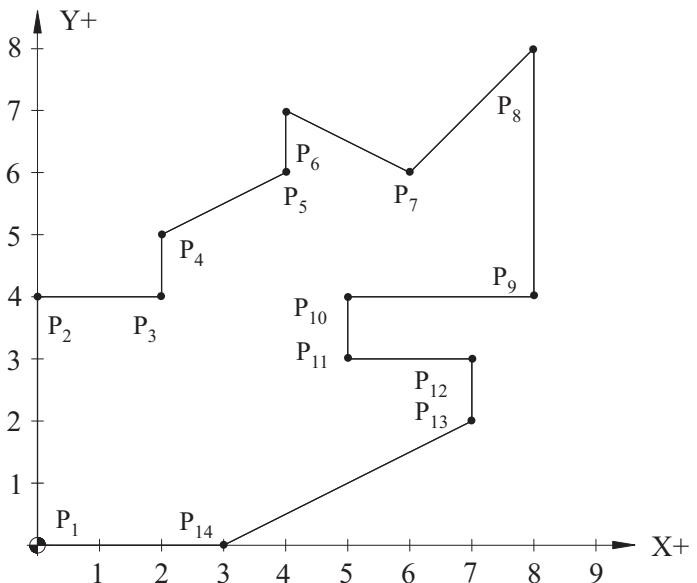
Chart 4-2 Miscellaneous Functions (M-Codes) Specific To Machining Centers

M-Code Function

M00   Program Stop

M01   Optional Stop

M02   Program End Without Rewind

M03   Spindle ON Clockwise (CW) Rotation

M04   Spindle ON Counterclockwise (CCW) Rotation

M05   Spindle OFF Rotation Stop

M06   Tool Change

M07   Mist Coolant ON

M08   Flood Coolant ON

M09   Coolant OFF

M10   Work Table Rotation Locked

M11   Work Table Rotation Unlocked

M13   Spindle ON CW with Coolant

M14   Spindle ON CCW with Coolant

M16   Change of Heavy Tools

M19   Spindle Orientation

M21   Mirror Image, X-axis

M22   Mirror Image, Y-axis

M23   Mirror Image Cancelaltion

M30   Program End With Rewind

M98   Subroutine Call

M99   Return to Main Program From Subroutine

Figure 4-1
Absolute and Incremental
Coordinates for CNC
Machining Centers

Absolute and Incremental Coordinates for Figure 4-1			
For G90 Absolute		**For G90 Incremental**	
P_1	G90 X0.0 Y0.0	P_1	G91 X0.0 Y0.0
P_2	G90 X0.0 Y4.0	P_2	G91 X0.0 Y4.0
P_3	G90 X2.0 Y4.0	P_3	G91 X2.0 Y0.0
P_4	G90 X2.0 Y5.0	P_4	G91 X0.0 Y1.0
P_5	G90 X4.0 Y6.0	P_5	G91 X2.0 Y1.0
P_6	G90 X4.0 Y7.0	P_6	G91 X0.0 Y1.0
P_7	G90 X6.0 Y6.0	P_7	G91 X2.0 Y−1.0
P_8	G90 X8.0 Y8.0	P_8	G91 X2.0 Y2.0
P_9	G90 X8.0 Y4.0	P_9	G91 X0.0 Y−4.0
P_{10}	G90 X5.0 Y4.0	P_{10}	G91 X-3.0 Y0.0
P_{11}	G90 X5.0 Y3.0	P_{11}	G91 X0.0 Y−1.0
P_{12}	G90 X7.0 Y3.0	P_{12}	G91 X2.0 Y0.0
P_{13}	G90 X7.0 Y2.0	P_{13}	G91 X0.0 Y−1.0
P_{14}	G90 X3.0 Y0.0	P_{14}	G91 X4.0 Y−2.0
P_1	G90 X0.0 Y0.0	P_1	G91 X−3.0 Y0.0

INCREMENTAL COORDINATE PROGRAMMING (G91) OF THE MACHINING CENTER

In incremental systems, every measurement refers to a previously dimensioned position (point-to-point). Incremental dimensions are the distances between two adjacent points.

The coordinate notations for the points in Figure 4-1 (in absolute and incremental systems) appear in Chart 4-3.

WORK COORDINATE SYSTEMS (G54, G55, G56, G57, G58, G59)

Identifying where the workpiece is located in X, Y, and Z within the machining envelope is necessary in every program. G92 was used on older systems to set the Absolute Position Register for the part. It required the positional data for the workpiece to be entered directly into the program and was used on less complicated programs in which only one Workpiece Zero was needed. Because G92 is seldom used today, the focus of this chapter will be on the offsets method of Functions G54 through G59. It is highly effective when multiple coordinate systems are required on one or multiple parts to establish Workpiece Zero coordinates. This method is described now.

In order to specify multiple Workpiece Zeros for Figure 4-2, use the following functions:

G54: first Workpiece Zero
G55: second Workpiece Zero

When using functions G54 through G59, the coordinates of the program zeros are entered into controller memory on the work offset page for the coordinate systems.

Part 4 Programming CNC Machining Centers
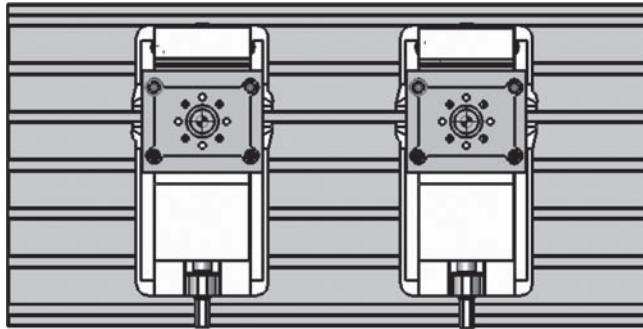
Figure 4-2
Multiple Work Coordinate Systems

Under the position 01 on the offset page, enter the coordinates of Workpiece Zero in the function G54, and so on. See specific instructions on the process for setting Workpiece Zero in Part 2, *Measuring Work Offsets, Machining Center*.

The following offsets are available. See Figure 2-28 in Part 2.

01   G54 X. . .Y. . .Z. . .

02   G55 X. . .Y. . .Z. . .

03   G56 X. . .Y. . .Z. . .

04   G57 X. . .Y. . .Z. . .

05   G58 X. . .Y. . .Z. . .

06   G59 X. . .Y. . .Z. . .

Workpiece Zero coordinates G54 through G59 are measured from Machine Zero to Workpiece Zero. Figures 4-3 and 4-4 illustrate the application of the functions G54 through G59:

Example: Work Coordinate Systems, Program No. 401

In the following program, which utilizes the functions G54 through G59, a .5" diameter HSS drill is used with a cutting speed of 80 ft/min to calculate the r/min and feed rate.

Part No. 1: (A) G54 X–10.0 Y–7.0 Z0.0

Part No. 2: (B) G55 X–14.0 Y–7.75 Z0.0

Part No. 3: (C) G56 X–18.5 Y–6.55 Z0.0

O0401

N1 0G90 G80 G20 G40 G49

N15 G00 G54 X.75 Y.5 S611 M03

N20 G43 Z1.0 H01 M08

N25 G00 Z.1

N30 G01 Z–.65 F3.6

N35 G00 Z.1
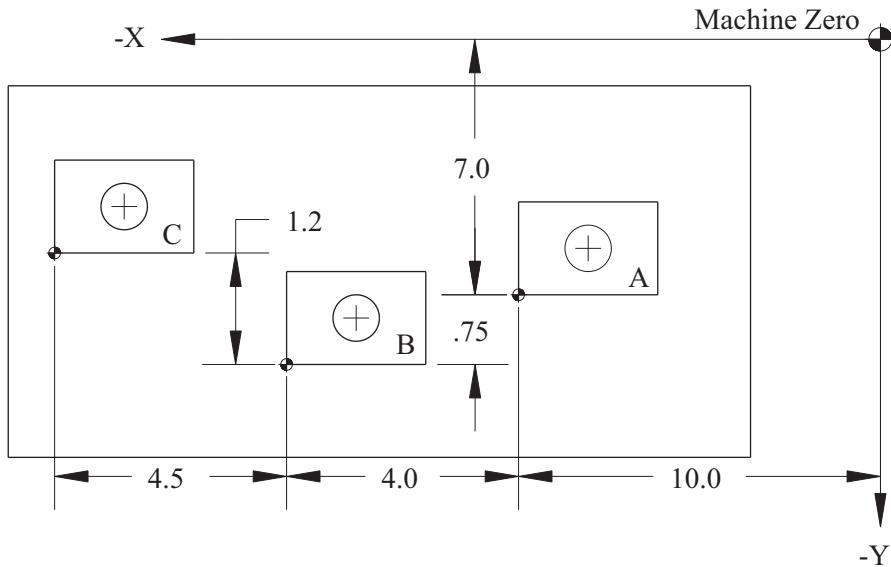
Figure 4-3
Work Coordinate Systems G54–59

N40 G55 G00 X.75 Y.5

N45 G01 Z–.65

N50 G00 Z.1

N55 G56 G00 X.75 Y.5

N60 G01 Z–.65

N65 G00 Z.1

N70 G91 G28 Z1.0 M09

N75 G91 G28 X0 Y0 M05

N80 M30

Figure 4-4
Drawing for Work Coordinate
Systems Example
Vises, Courtesy Kurt Industrial
Products

Note that the values in the program are in direct relation to the part zero. The only difference is where they are located within the work envelope. The G54, G55, and G56 offsets are input at the setup of the machine in the corresponding offset registers.

Note: G92 should <u>NOT</u> be used in conjunction with functions G54 through G59 because the results are unpredictable.

PROGRAM STRUCTURE FOR MACHINING CENTERS

PROGRAM NUMBER

Each program is assigned a number. The capital letter O is reserved for the program number identification and is usually followed by four digits, which specify the actual program number. For example, to create a program with the number 1234, the programmer

must input the letter address O, and then the number 1234 (O1234). All programs require this format.

Note: Refer also to Part 1, Program Format.

Examples

    O0001 = program number 1

    O0014 = program number 14

A common mistake made here, is to enter zero (0) instead of the letter O, resulting in an alarm on the control system.

COMMENTS

Comments that help the operator may be added to the program by using parentheses. The comments or data inside parentheses will not affect the execution of the program in any way. A very common place to add comments is at the program number, to identify a part number or at a tool change or program stop, in order to direct the operator in some way. An End-of-Block character (EOB) is typically required after the parenthesis if entered via MDI. No EOB character is needed if the data is entered via an offline text editor.

Examples of Comments

O0001

(PN587985-B)

N90 M00

(REMOVE CLAMPS FROM OUTSIDE OF PART)

BLOCK NUMBER

The letter *N* is reserved to identify the program line sequence numbers (block numbers), and precedes any other data in a program line. For each line in a program, a block number is assigned sequentially. For example, the first block of a program is labeled N10, the second N15, etc. Typically the program block numbering system is sequenced by an increment other than one. An example of this is sequencing by five, where line one is labeled N5 and line two is labeled N10. The original intent of an increment of other than one is to allow for inserting additional blocks of data between the increments, as needed. These additions can sometimes be advantageous when editing programs. Note that block numbering is not necessary for the program to be executed. The program blocks, even if not numbered, will be executed sequentially during the machine's automatic cycle. Removing block numbers is sometimes helpful when a program is too large to fit into the resident controller memory on older machines—each character of a program takes memory space and large programs can have block numbers into five digits.

It is a common practice to place block numbers only at a tool change command. The operator can then restart the execution of the machining program at a specific tool change; otherwise, the entire program may have to be executed in order to rerun a specific operation and time would be wasted. Block numbers enable movement within the program in order to enter offsets, verify data, or search for a block in the program. They are often referenced by the control in case of a programming error that causes an alarm,

enabling the programmer to search to the problem directly by block number. Block numbers are not required at all for the program to work, but they are necessary for restarting the program at a specific place.

SUBPROGRAM (M98–M99)

If a certain part of the program can be used repeatedly within the program, assign a number for this part and list it separately. Then call for it whenever it is needed. This part of the program is called a subprogram. Subprograms greatly simplify programming, decreasing the amount of data that must be placed into the controller memory. A subprogram is called up from control memory by the function M98 and the letter P, both of which are entered in the main program. The letter P refers to the program number of the subroutine call. As with the main program, in order to enter the subprogram number, you must first write the letter O and then the number. For each subprogram number, the four spaces after the letter O are reserved. Ordinarily, no workpiece coordinate system is established within the subprogram because of its dependence (subordinate) on the main program.

Example
O0002
N10
. . .
. . .
N50 M99

Function M99 refers to the end of the subroutine. It returns to the main program block following the block in the main program that contains the subroutine call.

Example
M98 P0002

This is a call for subprogram No. 2.

Main program	Subprogram	Subprogram
O0001		
N10 G54 X . . . Y . . .		
N15 . . .		
N20 . . .		
. . .		
. . .		
. . .		
. . .	**O0002**	**O0003**
. . .	**N10 . . .**	**N10 . . .**
. . .	**N15 . . .**	**. . .**

N55 M98 P0002	**. . .**	**. . .**
N60 . . .	**. . .**	**. . .**
. . .	**N30 M98 P0003**	**. . .**
. . .	**N35 . . .**	**. . .**
. . .	**. . .**	**. . .**
N80 M30	**N45 M99**	**N45 M99**

In the example above, enter the subroutine call in the program block N55 of the main program; then use function M98 to call up subprogram O0002. Next, work is performed according to the commands in the subprogram until block N30 is reached, at which point the processing is transferred to subprogram O0003. After the execution of subprogram O0003, the program is returned to block N35 of subprogram O0002, which executes the remaining information blocks of this subprogram. From block N45 of this subprogram, the program is returned to block N60 of the main program, which then executes the remaining part of the main program. As many as four levels of subprograms can be linked together. This process is called nesting. To repeat a given subprogram twice, enter the following:

> **M98 P0002 L2**
>
> **L2**   repeat subprogram twice

Some differences apply for this input, depending on the controller. It is highly recommended that the programmer study the manufacturer's operation and programming manuals for specific instructions on using subprograms.

PROGRAM END (M30)

The difference between M02 and M30 is that M02 refers to the program end, while M30 refers to the program end and a simultaneous return to the program beginning (head). Both commands are found in the final line of the main program only.

Note: on some controls, M02 behaves the same as M30 for compatibility with older programs.

TOOL LENGTH COMPENSATION (G43, G44, G49)

Workpiece Zero for the X, Y, and Z axes is placed within the machining envelope in relationship to Machine Zero. The Z-axis is typically on the top-most surface of the machined part. Workpiece Zero for the Z-axis should be in the same position for all tools. Due to different tool lengths, in order to transfer Workpiece Zero along the Z-axis from Machine Zero to the surface of the workpiece, you must apply function G43, a Tool Length Compensation function (Tool Length Offset).

Offset number H. . . (H01, H02) is always assigned in the same line with function G43. As a rule, in order to simplify program execution, offset number H should be the same as the tool number for each corresponding tool. The measured value of offset H is entered into the offset registers in the computer memory (for example, H01 = −11.1283). The value of the Tool Length Offset for a given tool corresponds to the distance between the tool tip and the surface of the workpiece, as shown in Figure 4-5.
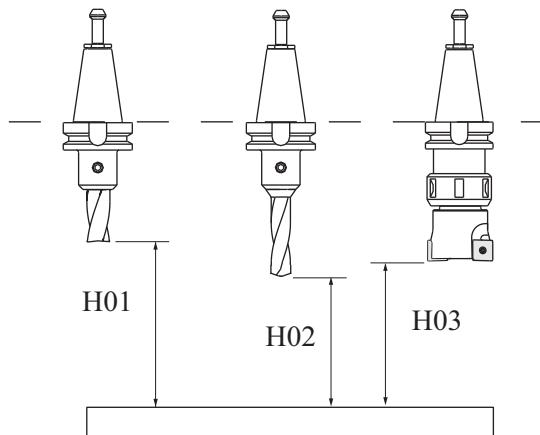
Figure 4-5
Tool Length Offset

In order to determine the value of offset H for a specific tool, zero (Home) the machine with respect to the Z-axis with the tool to be measured in the spindle. Zero the position readout for Z; then, manually move the tool along the Z-axis to the surface of the workpiece in such a way that the tool tip touches the surface of the workpiece. The value in the position register is the distance that determines the value of offset H for a given tool. This value is registered in the offset table with the number corresponding to the offset number in the program.

Function G43, with the assigned offset number H, must be entered in the program before the tool does any work. If the offset number H . . . is not entered, the tool will perform the work with the previous tool length offset. If the value of the tool length offset for the previous tool is smaller than the value of the current tool's length offset, then the tool will not approach the material.

Caution: If, however, the value of the previous tool length offset is greater than the working tool's value, the working tool will rapidly advance toward the workpiece and crash into it, causing damage to the tool, the workpiece, and the holding equipment.

This condition can be avoided by employing the Tool Length Compensation Cancel function (G49) in a Safety Block at the beginning of the next tool sequence. See *Explanation of the Safety Block* later in this section.

G43 = positive tool length offset

Note: the entry in the offset register is negative because the offset represents the distance from the tool tip to the workpiece Z zero.

G44 = negative tool length offset (not often used today)
G49 = cancellation of tool length offset

Functions G43 and G44 are used to read the tool length offset amount from the offset registers. Numbers in the offset registers correspond to the tool length in

the zero position from the surface of the part to be machined. The tool length offset is called in the program by the letter address H. The tool length offset value may be added (G43) to the programmed value of Z or subtracted (G44) from the programmed value of Z.

The tool offset number used is identified by the letter H and two digits.

Example
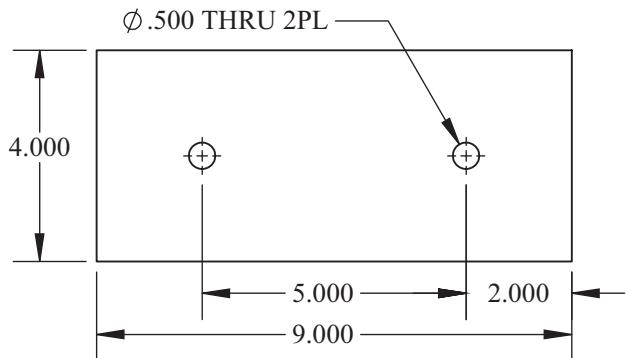
   H01 = offset number one

When entering the new offset value of H to the offset register, the previous offset value is automatically canceled; the machine reads the new value without considering the previous value. To cancel functions G43 and G44, use function G49. This cancellation code (G49) is placed in the Safety Block.

Example: CNC Machining Center, Program 402

Two holes, with diameters of .500 inch each, are to be drilled in a one inch thick steel plate having the dimensions 9 x 4 inches. The position of the holes can be seen in Figure 4-6.

Figure 4-6
Drawing for Machining Center
Example Program 402

⌀.500 THRU 2PL

4.000

5.000   2.000

9.000

Machine: Machining Center	Program Number: O0402

Workpiece Zero: X = Upper Right Corner Y = Upper Right Corner Z= Top Part Surface
Setup Description: Material = Steel 1018
The plate is placed on 1.5" parallels

Tool #	Description	Offset	Comments
1	Number 3 Center Drill		
2	.500 Diameter Drill		

Part 4 Programming CNC Machining Centers

The use of the CNC Setup sheet was described in Part 1. Throughout the remainder of Part 4, an abbreviated version will be used for the examples. The top portion (including the Title, Date, Prepared By, Part Name, and Part Number) will be omitted to save space. However, each tool will be identified.

```
O0402
N10 G90 G20 G80 G40 G49
(NO. 3 CENTER DRILL)
N15 T1 M6
N20 G90 G00 G54 X–2.0 Y–2.0 S900 M03
N25 G43 Z1.0 H01 M08
N30 G81 G99 X–2.0 Z–.219 R.1 F4.6
N35 X–7.0
N40 G80 Z1.0 M09
N45 G91 G28 Z1.0
N50 M01
(.500 DIAMETER DRILL)
N55 T2 M06
N60 G90 G00 G54 X–2.0 Y–2.0 S620 M03
N65 G43 Z1.0 H02 M08
N70 G81 G99 X–2.0 Z–1.25 R.1 F7.0
N75 X–7
N80 G80 Z1.0 M09
N85 G91 G28 Z0.0 M05
N90 G28 X0.0 Y0.0
N95 M30
```

The following descriptions are given for Machining Center program 402 above:

- In block N10, functions G90, G20, G80, G40, and G49 are defined as follows:

  - G90 establishes absolute coordinate system dimensioning.
  - G20 establishes the inch system of measurement.
  - G80 cancels all canned cycle functions and is entered at the beginning of the program to ensure that all cyclic functions from the previous program are no longer in effect.
  - G40 cancels cutter diameter compensation functions G41 and G42.
  - G49 cancels the tool length compensation functions G43 and G44.
- At N15, tool one (the number 3 center drill) is mounted in the spindle.
- Block N20 positions the tool to the proper position for drilling the first hole and sets the spindle speed (S900) and direction (M03) of the rotation.

- In block N25, the values of tool length offset H01 is read (center drill), while rapid traverse of the spindle along the Z-axis (with a value of Z = 1.000) is performed to a position above the workpiece. The miscellaneous function M08 activates the flood coolant flow.

- In block N30, the first hole is center drilled with a feed equivalent to 4.6 in/min.

- In block N35, the second hole is center drilled.

- Block N40 cancels canned cycle function G81, raises the tool to the initial reference plane position of Z1.0, and deactivates the coolant flow with M09.

- In line N45 the machine returns to zero with respect to the Z-axis.

- In block N50, the programmed machine work is stopped by function M01, but only if the "optional stop" button on the operator control panel is on. The purpose of this block is primarily to confirm whether tool T01 has performed the work that has been assigned in the program. This block usually appears before the tool change. This stoppage of the program execution gives the operator an opportunity to perform in-process inspection of the completed operation.

- In block N55, tool 1 is replaced by tool 2 in the spindle. Up to now, only one program segment for one tool has been executed. Now, by comparing the remaining part of the program, notice that it contains many similar elements for the second tool, the .500 diameter drill.

Generally speaking, the program segment consists of a few characteristic elements that play the following roles:

- Establish the absolute or incremental coordinate system.
- Establish the Work Offset used.
- Set the spindle r/min and rotation direction.
- Execute the tool length offset given and activate the flow of the coolant.
- Determine the drilling canned cycle.
- Determine the tool's work path to consecutive holes in the pattern to be drilled.
- Positioning of the tool to the consecutive locations.
- Cancel any canned cycles and deactivate the coolant flow.
- Command of the Z, X and Y axes to return to Machine Zero position.
- Optional Program Stop.
- Tool change if needed.
- Repeat sequence if needed.
- Program Ending.

Example: Machining Center, Program 403

The half-inch step and drilled hole are machined on the part in the drawing given in Figure 4-7; the tools and setup information are listed on Chart 403.
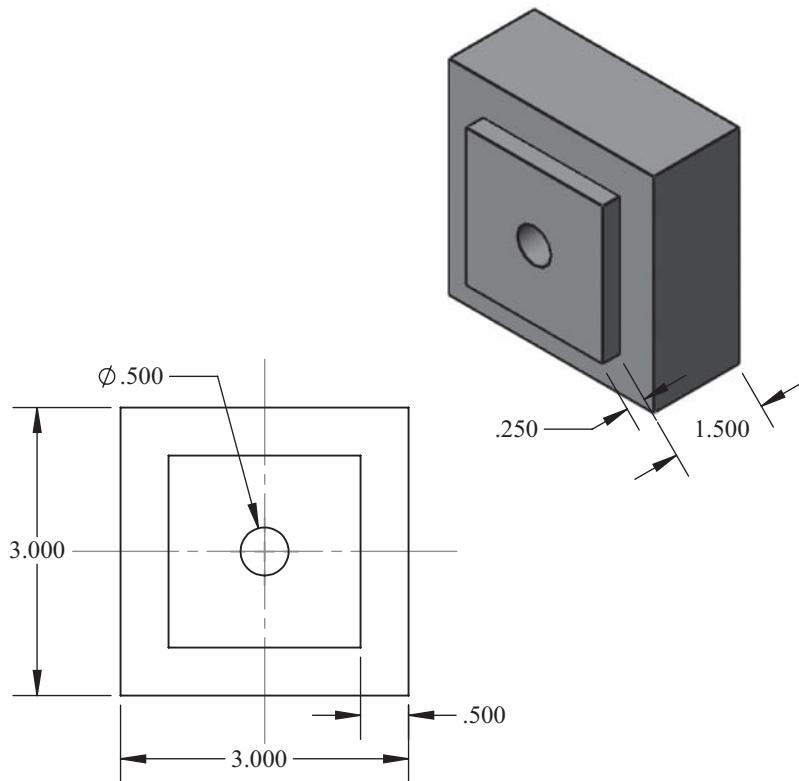
Figure 4-7
Drawing for Machining Center Example Program 403

Machine: Machining Center		Program Number: O0403	
Workpiece Zero: X = <u>Upper Right Corner</u> Y = <u>Upper Right Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = Aluminum 6061 The plate is placed on 1.0" parallels			
Tool #	Description	Offset	Comments
T1	5/8 Endmill 2 Flute HSS		SFM= 400
T2	Number 6 Center Drill		SFM= 400
T3	.500 Diameter Drill		SFM= 400

O0403
N10 G90 G20 G80 G40 G49
N15 G00 G54 X.4 Y−.1875 S2445 M03 T02
N20 G43 Z1.0 M08 H01
N25 G01 Z−.25 F20.0

N30 X–2.8125

N35 Y–2.8125

N40 X–.1875

N45 Y.4

N50 G91 G28 Z1.0 M09

N55 M01

N60 M06

N65 G90 G80 G40 G49

N70 G00 G54 X–1.5 Y–1.5 S3057 M03 T03

N75 G43 Z1.0 H02 M08

N80 G81 G98 Z–.438 R.1 F12.0

N85 G80 M09

N90 G91G28 Z1.0

N95 M01

N100 M06

N105 G90 G80 G40 G49

N110 G00 G54 X–1.5 Y–1.5 S3057 M03 T01

N115 G43 Z1.0 H03 M08

N120 G81 G98 Z–1.664 R.1 F18.0

N125 G80 Z1.0 M09

N130 G91 G28 Z0.0 M05

N135 G28 X0.0 Y0.0

N140 M06

N145 M30

EXPLANATION OF THE SAFETY BLOCK

Block N10 is called the Safety Block; the name refers to its role in the program. This Safety Block consists of G-Codes that establish the type of coordinate system used (G90 Absolute or G91 Incremental), establish the measurement system used (G20 inch or G21 metric), cancel all canned cycles (G80), cancel cutter diameter compensation (G40), and cancel tool length compensation (G49).

In order to define the canceling function of G80 more accurately, study the Machining Center Program 402. In block N80, the canned drilling cycle G81 is activated. If the canned drilling cycle is interrupted for some reason (the drill or the workpiece becomes damaged), the machining must be stopped in order to exchange the tool and or the workpiece. Then by pressing the RESET button, the program is returned to its beginning or head.

However, in the sequence of events described above, the canceling function G80 (included in block N90) has not occurred. If machining is started from the beginning of the program, tool one (an end mill of .625 inch) will need to be returned to the spindle and the new drill returned to the tool magazine prior to starting. In the above-described

procedure, if function G80 is omitted from block N10, then function G81 is still valid. Thus, all position changes of the end mill (along the X and Y axes) will be executed as if the canned drilling cycle is still active (including the assigned parameters of Z and R from block N80). On many machines, RESET will cancel canned cycles. Check the operator and programming manuals specific to your machine to be sure.

To avoid the possibility of this kind of issue, use the Safety Block. Include the canceling functions in the first block of the program and immediately following all tool changes.

Consider this similar situation in which tool radius compensation was applied: In programming the milling process, you may establish the movement of the tool by determining the movement of the center of the tool. This approach applies to standard programming. However, you may also program the actual contour of the workpiece. From that, the computer will calculate the path of the center of the mill (or other tool) with a given radius. This means that the actual machine movements will differ from the programmed movements by the value of the offset placed in the offset register; this value is the radius of the tool. Such an arrangement can be applied by using functions G41 (cutter compensation to the left) and G42 (cutter compensation to the right). If for any reason the milling process with the assigned tool radius compensation is interrupted, then the next tool used will be positioned with a displacement value equal to the mill radius compensation previously used, for example, a drill. In order to avoid such situations, employ the cancellation function G40 in the Safety Block.

The same scenario can be applied to tool length compensation values that are activated in the program by the letter address H followed by the number of the tool, as shown in block N20. If this offset value is not cancelled by G49 in the safety block, it too will remain valid. Note: If function G28 is used prior to tool change, as in line N60, the tool length offset is cancelled.

To avoid such mistakes, use the Safety Block, including the canceling functions, in the first block of the program and immediately following all tool changes.

PREPARATORY FUNCTIONS FOR MACHINING CENTERS (G-CODES)

Preparatory functions, often called G-Codes, are a major part of the programming puzzle. They identify to the controller what type of machining activity is needed. For example, if a hole needs to be drilled, function G81 may be used, or if programming in the incremental coordinate system is required, function G91 is used. These codes, along with other data, control machine motion.

The motion of the axes of a machine may be performed along a straight line, an arc, or a circle.

Codes G00 and G01 allow axes movement along a straight line.

Codes G02 and G03 allow axes movement along a circular path of motion.

Rapid Traverse Positioning (G00)

The rapid traverse function is entered to relocate the tool from position A to position B along a straight line at the fastest possible traverse. The shortest axis movement distance will be accomplished first. Therefore, you must be aware of the workpiece

holding the equipment in order to avoid any collision between the tool and the holding equipment. The path traveled by the tool for G00 X30.0 Y20.0 is as follows:

If you are uncertain of the tool path, then position the RAPID TRAVERSE OVERRIDE to a reasonable percentage (see Part 2, *Rapid Traverse Override*). This adjustment reduces the traverse speed and increases the time allowance for a possible manual interruption of the motion. Programmers should position the Z-axis to an acceptable clearance plane of 1.0 inch or any amount necessary to move over clamps or obstructions to reduce the chance of collision.

By adjusting a system parameter, the movement described in Figure 4-8 can be changed to a simultaneous or diagonal move of both axes. Consult the manufacturer's manual for specific instructions.

LINEAR INTERPOLATION (G01)

Function G01 is used to move the tool from point P1 to point P2 (Figure 4-9) along one or all of the axes simultaneously, along a straight line of motion, and at a given feed rate specified by the F-word.

Example

G01 X10.0 Y20.0 F8.0

Linear interpolation may be performed along three axes simultaneously. The control system calculates the particular speeds for each axis so that the resulting speed is equivalent to the programmed feed rate.

Example: Linear Interpolation, Program 404

The half-inch steps need to be machined on the ends of the part in Figure 4-10; the tools and setup information are listed on Chart 404.
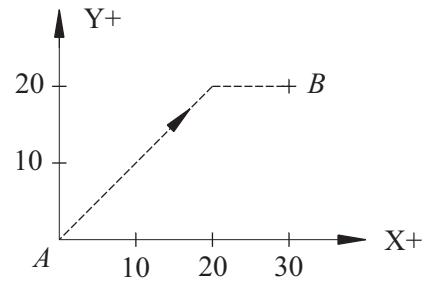
Figure 4-8
Rapid Traverse Tool Movement
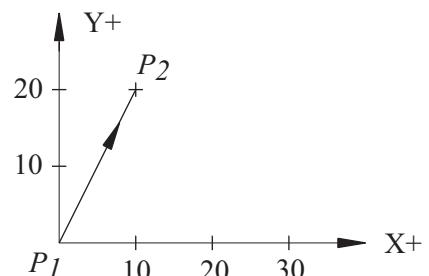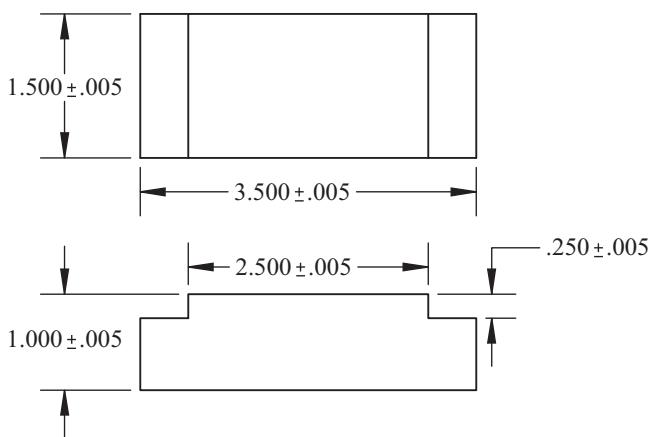
Figure 4-9
Linear Interpolation

Figure 4-10
Drawing for Linear
Interpolation Example

Machine: Machining Center		Program Number: O0404	
Workpiece Zero: X = <u>Lower Left Corner</u> Y = <u>Lower Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = Aluminum 6061-T6, Blank dimensions 3.5 x 1.5 x 1.0 The plate is placed on 1.50" parallels			
Tool #	Description	Offset	Comments
T1	.625 Endmill 2 Flute HSS		SFM= 400 Feed = .003 inch per tooth

O0404

N10 G90 G20 G80 G40 G49

N15 G00 G54 X.1875 Y–.4 S2445 M03

N20 G43 Z1.0 H01 M08

N25 G01 Z–.250 F50.0

N30 Y1.9 F29.0

N35 G00 Z1.0

N40 X3.3125

N45 G01 Z–.250 F50.0

N50 Y–.4 F29.0

N55 G91 G28 Z1.0 M09

N60 G28 X0.0 Y0.0 M05

N65 M30

CIRCULAR INTERPOLATION (G02, G03)

Circular interpolation allows tool movements to be programmed to move along the arc of a circle. When applying the circular interpolation, the plane in which the arc is positioned must be determined initially. To do this, employ preparatory function G17, G18, or G19. Then, depending on the direction of the machining, select function G02 to make a clockwise movement along the arc and function G03 to make a counterclockwise movement along the arc. In order to describe the movements of the tool along the arc (Figure 4-11), apply the following two methods:
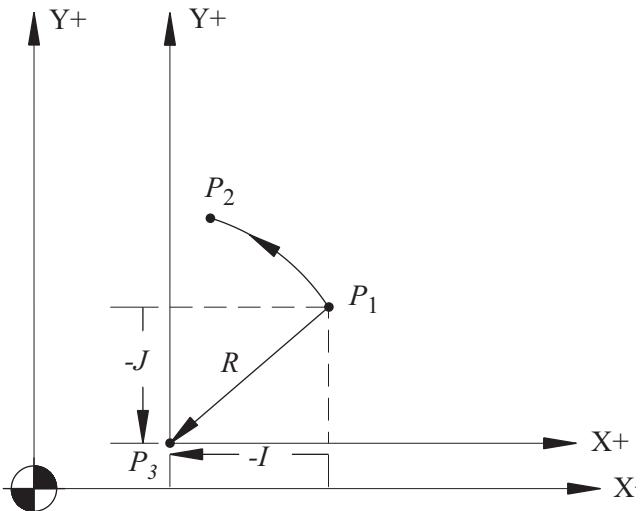
1. Determine the radius R and the values of the end point coordinates in the given plane.

2. Determine the value of the endpoint of the arc and the values of the incremental distance to the arc center in a given plane.

P_1 = the start of an arc

P_2 = the end of an arc

P_3 = the center of the arc

Figure 4-11
Required Data for Circular
Interpolation

R = radius vector

I = the incremental distance to the arc center along the X-axis

J = the incremental distance to the arc center along the Y-axis

The incremental distance is defined as a radius projection onto a given axis. The radius incremental distance is always attached. It begins at the starting point and ends at the center of the circle. It is always directed toward the center of the circle.

- Vector projections onto the X-axis are identified by use of the letter address I.
- Vector projections onto the Y-axis are identified by use of the letter address J.
- Vector projections onto the Z-axis are identified by use of the letter address K.

In the following example, a vector projection is illustrated. The radius is positioned at the origin of the coordinate system, as shown in Figure 4-12.

Note: in Figure 4-12, the signs (+, −) of the incremental distances I, J, and K depend on the position of the starting point of the arc with respect to the center of the arc, that is, with respect to the coordinate system. If the direction of the vector is consistent with the direction of the assigned axis of the coordinate system, then we apply the positive sign. If not, then we apply the negative sign.

Most modern controllers do not require the use of the positive sign. If no sign is present, the value is considered to be positive.

In Figure 4-13, the tool path begins in the lower left corner of the part. Each arc is identified C_1 through C_{10} with the center point of the arc shown. The arrows indicate direction vector to the arc center. The arc cutting directions and the I and J values are:

$$C_1 = G02, \; I, \; J0$$
$$C_2 = G03, \; I, \; -J$$

$C_3 = \text{G02}, I, J0$

$C_4 = \text{G03}, I, -J$

$C_5 = \text{G02}, I, -J$

$C_6 = \text{G03}, I, -J$

$C_7 = \text{G02}, -I, -J$

$C_8 = \text{G03}, -I, J$

$C_9 = \text{G02}, -I, J$

$C_{10} = \text{G02}, -I, J$
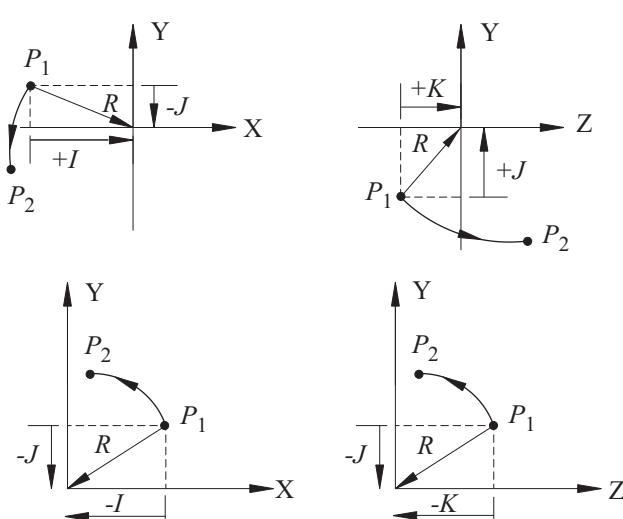
Figure 4-12
Vector Projection for Circular
Interpolation
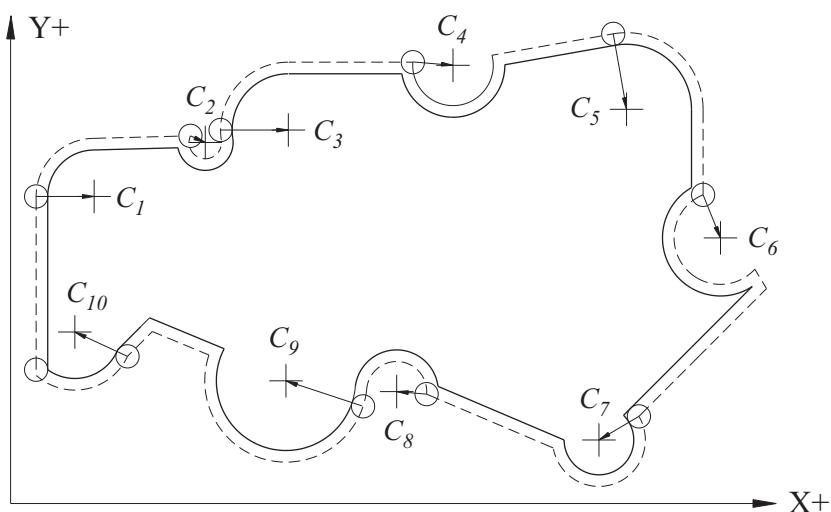
Figure 4-13
Example of Circle Center Vector Direction

Part 4 Programming CNC Machining Centers

Notes: When circular motion is described with radius function R, no sign is required if the arc is less than or equal to 180° (the system defaults to positive unless otherwise specified). Assign a negative value to R (–) if the arc is greater than 180°. The maximum rotation of an arc using R is 359.9°.

A full circle may be accomplished using R by linking two 180° arcs. If a full circle of 360° is to be performed, then it is necessary to employ the incremental distances of the arc center points for I, J, and K—not radius R—in the program.

Example

Do not use I, J, or K with R in the same block, because I, J, or K will be ignored by the control of they are used; the tool will follow the arc with the assigned radius of R. If the value of an entered radius R is zero, an alarm will result.
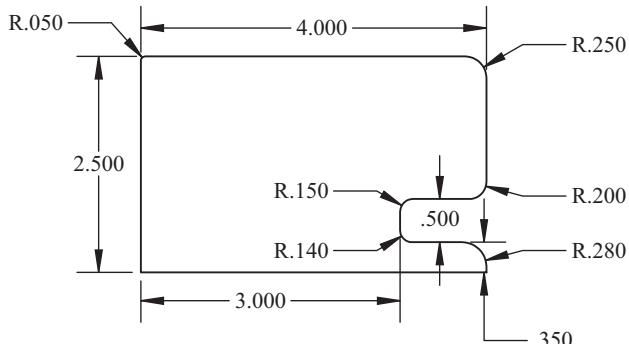
Cutter radius compensation may be used for circular interpolation. However, it must be initiated in a G00 or G01 block preceding the G02/G03 information.

In the next example, the tool path centerline is programmed. Because of this, you must add the radius of the tool to all of the radii and set the Offset Register for each tool to zero. A more effective and easier programming method will be introduced later in *Cutter Compensation*.

Example: Circular Interpolation, Program 405

The profile of the part needs to be machined for the part in Figure 4-14. The tools and setup information are listed on Chart 405.

Figure 4-14
Drawing for Machining
Center Circular
Interpolation Example

Machine: Machining Center	Program Number: O0405

Workpiece Zero: X = Lower Left Corner Y = Lower Left Corner Z= Top Part Surface
Setup Description: Material = Steel, Blank dimensions 4.0 x 2.5 x .150
The plate is placed mounted to a fixture plate by existing .250 diameter holes, not shown on the drawing.
The program is limited to a finish path.

Tool #	Description	Offset	Comments
T2	.250 Endmill 4 Flute HSS		SFM= 90 Feed = .002 inch per tooth

O0405
N10 G90 G20 G80 G40
N15 G00 G54 X–.125 Y–.2 S1375 M03
N20 G43 Z1.0 H02 M08
N25 G01 Z–.16 F50.0
N30 Y2.45 F11.0
N35 G02 X.05 Y2.625 I.175 J0.0
N40 G01 X3.75
N45 G02 X4.125 Y2.25 I0.0 J–.375
N50 G01 Y1.05
N55 G02 X3.8 Y.725 I–.325 J0.0
N60 G01 X3.15
N65 G03 X3.125 Y.7 I0.0 J–.025
N70 G01 Y.49
N75 G03 X3.140 Y.475 I.015 J0
N80 G01 X3.72
N85 G02 X4.125 Y.07 I0.0 J–.405
N90 G01 Y–.125
N95 X–.125
N100 G91 G28 Z1.0 M09
N105 G28 X0.0 Y0.0 M05
N110 M30

The following is the same program but utilizing radius R:

O0405
N10 G90 G20 G80 G40
N15 G00 G54 X–.125 Y–.2 S1375 M03
N20 G43 Z1.0 H02M08
N25 G01 Z–.1 F50.0
N30 Y2.45 F11.0
N35 G02 X.05 Y2.625 R.175
N40 G01 X3.75
N45 G02 X4.125 Y2.25 R.375
N50 G01 Y1.05
N55 G02 X3.8 Y.725 R.325
N60 G01 X3.15
N65 G03 X3.125 Y.7 R.025

N70 G01 Y.49

N75 G03 X3.14 Y.475 R.015

N80 G01 X3.72

N85 G02 X4.125 Y.07 R.0405

N90 G01 Y–.125

N95 X–.125

N100 G91 G28 Z1.0 M09

N105 G28 X0.0 Y0.0 M05

N110 M30

HELICAL INTERPOLATION USING G02 OR G03

Helical interpolation allows movements of the tool to be programmed so that it travels along a circular path in the XY plane with a simultaneously straight-line motion along the Z-axis. Such a combination of tool movements with respect to the three axes creates a helix contour and may be used to machine threads (Figure 4-15 and 4-16). For all practical purposes, this function is limited to machining threads with large diameters because of the required tool diameter.

Block Format:

G02 X . . . Y . . . I . . . J . . . Z . . . F . . . (R . . . may be used in place of I and J)

G03 X . . . Y . . . I . . . J . . . Z . . . F . . . (R . . . may be used in place of I and J)

where

X = arc ending point

Y = arc ending point

I = incremental X-axis distance of the arc center from the start point

J = incremental Y-axis distance of the arc center from the start point

Z = arc ending point (at depth)

R = radius of the arc

F = feed rate

As stated before, function G02 refers to clockwise motion (CW) whereas function G03 refers to counterclockwise motion (CCW).

Notes on Helical Interpolation

- *This function may only be applied under the following conditions:*
  - *The minor diameter for internal threads or major diameter for external threads must be machined in a prior operation.*
  - *The XY plane is in a circular interpolation.*
  - *The Z-axis is in a linear interpolation.*
  - *The tool must be positioned in the Z-axis prior to the helical interpolation.*
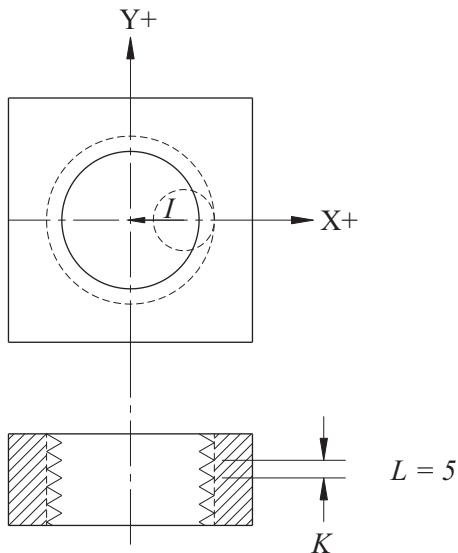
Figure 4-15
Thread Pitch for Helical
Interpolation
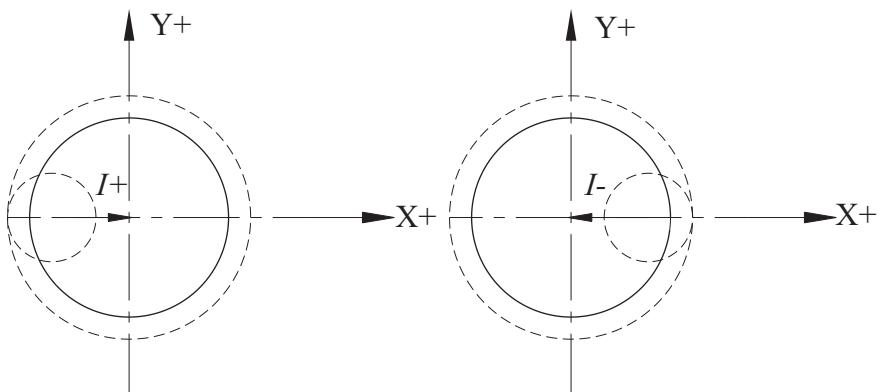
Figure 4-16
Helical Interpolation

- *The feed rate designated by the F-word is the circular interpolation for the XY plane.*
- *Climb milling is the preferred machining method.*
- *If one pass of the tool does not complete the full thread, depending on machining starting from the bottom or top of the thread, you must adjust the Z-axis coordinates of the starting or ending point for another pass.*
- *If cutter radius compensation is used in the program, G41 or G42 may not be called in the same line as the G02 or G03. For an internal thread, it is typically called in a linear approach move from the center of the diameter moving outward, then cancelled at the end of interpolation moving back towards the center. This move must be equal to, or greater than, the radius of the tool used.*

Part 4 Programming CNC Machining Centers

Figure 4-17 is an example of a tool that may be utilized during threading with the application of helical interpolation.

Example: Helical Interpolation, Program 406

The 1.125-14 thread shown in Figure 4-18 is machined using helical interpolation. The setup and tool information is listed on chart 406.

Figure 4-17
Helical Threading Tool

.0714

.625
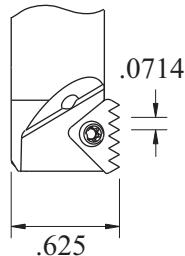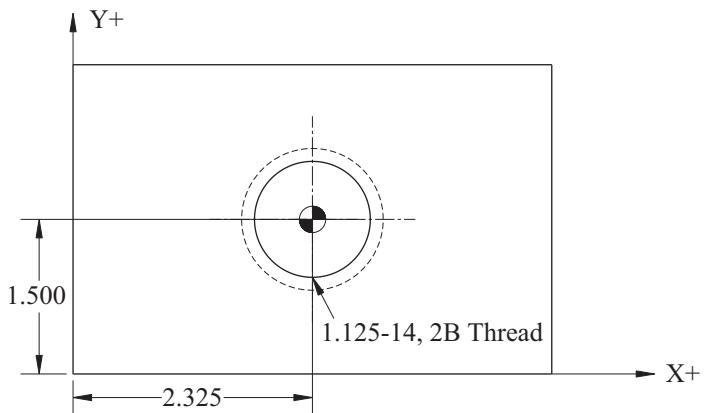
Figure 4-18
Drawing for Helical
Interpolation Example

Y+

1.500

2.325

1.125-14, 2B Thread

X+

Machine: Machining Center	Program Number: O0406

Workpiece Zero: X = <u>Part Centerline</u> Y = <u>Part Centerline</u> Z= <u>Top Part Surface</u>
Setup Description: Material = Material used is .50 thick CD 12L14 steel plate
The plate is placed on 1.625 inch parallels in a vise.
Preparation of the minor diameter was completed in a prior operation.

Tool #	Description	Offset	Comments
T1	Special Carbide Threading Tool with .0714 lead. See Figure 17		SFM= 70 Feed = 1.5 IPM

O0406

N10 G90 G20 G80 G40 G49

N15 G00 G54 X0.0 Y0.0 S428 M03

N20 G43 Z1. H01 M08

N25 G01 Z–.575 F50.0

N30 G01 X.25 F3.5

N35 G03 X.25 Y0.0 I–.25 Z–.5036 F1.5

N40 I–.25 Z–.4322

N45 G01 X0.0 F50.0

N50 G00 Z1.0 M09

N55 G91 G28 Z0.0 M05

N60 G28X0Y0

N65 M30

Note: The value X.25 in block N35 was calculated in the following manner:

$$X = \frac{1.125}{2} - \frac{.625}{2} = .250$$

The difference between the Z value on line N40, –.5036, and on line N45, –.4322, is equal to the lead of the thread, .0714.

Dwell (G04)

Dwell is determined by the preparatory function, G04 and by using the letter address P or X, which correspond to the dwell's time duration.

Block Format
    G04 P . . .
    G04 X . . .
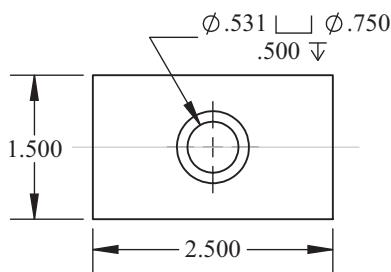
Example
    G04 X1.0 or G04 P1000

The length of time for the dwell is stated by X or P; in this example, it is equivalent to 1 second. The P address is programmed in milliseconds (ms) 1 second = 1000 ms. When the letter address P is used to determine time, do not use the decimal point. Drilling or counter-boring holes, with a specifically defined depth, are two examples in which to apply function G04. In these cases, time P and X determine a time period in which the drill, having full rotational speed, hesitates at the bottom of the hole long enough to fully and accurately remove the excess material from the bottom of the hole.

Part 4 Programming CNC Machining Centers

Figure 4-19
Drawing for Dwell Example

Machine: Machining Center			Program Number: O0407	
Workpiece Zero: X = <u>Lower Left Corner</u> Y = <u>Lower Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = Steel 4140, 1.0" Thick The plate is placed on 1.625" parallels				
Tool #	Description	Offset	Comments	
T1	#4 Center Drill		SFM= 75 FPT = .001	
T2	17/32 HSS Drill		SFM= 75 FPT = .002	
T3	.750 Diameter HSS End Mill		SFM= 75 FPT = .002	

Example: Using Dwell, Program 407

The counter-bored hole shown in Figure 4-19 requires the tool to dwell for a period of time to ensure full clean-up. The setup and tool information is listed on Chart 407.

O0407

N10 G90 G20 G80 G40 G49

N15 T1 M06

N20 G00 G54 X1.25 Y.750 S960 M03 T02

N25 G43 Z1.0 H01 M08

N30 G00 Z.1

N35 G01 Z–.269 F1.9

N40 G91 G28 Z0.0 M09

N45 M01

N50 M06

N55 G90 G80 G40 G49

N60 G00 G54 X1.25Y.750S564 M03 T03

N65 G43 Z1.0 H02 M08

N70 Z.1

N75 G01 Z–1.35 F2.25

N80 G91 G28 Z0.0 M09

N85 M01

N90 M06

N95 G90 G80 G40 G49

N100 G00 G54 X1.25 Y.750 S400 M03

N105 G43 Z1.0 H03 M08

N110 Z.1

N115 G01 Z–.5 F3.2

N120 G04 P300

N125 G00 Z1.0 M09

N130 G91 G28 Z0.0 M05

N135 G28 X0.0 Y0.0

N140 M30

In block N120, a dwell of .3 seconds is utilized to fully cut the bottom of the counterbore.

EXACT STOP (G09)

When this function is used, the machine responds by decelerating to a feed of zero in the axis commanded before executing the next line, thus, checking the programmed end point. This function is used only if you want to obtain a sharp edge around corners in the cutting feed mode. It refers only to the block to which it was assigned.

PROGRAMMABLE DATA SETTING (G10)

If the number of work, tool geometry, tool radius, or tool wear offsets needed exceeds the limit of the machine's computer memory, then an additional offset may be entered directly into the program by the use of Programmable Data Setting function G10.

Block format for work offsets:

G10 L2 P2 X. . . Y. . . Z. . .

where

L2 = work offsets

P2 = offset number (G55)

X, Y, or Z = value of offset

If all the work coordinate systems (functions G54 through G59) are being used, then by entering the new coordinate values (X, Y, and Z) in the program, along with the function G10, six additional systems are gained.

Example

G10 L2 P6 X. . . Y. . . Z. . .

In the case of the sixth coordinate system (G59), the previous coordinate values (X, Y, and Z) are replaced by new ones.

G10 L2 P1 = offset G54

G10 L2 P2 = offset G55

G10 L2 P3 = offset G56

G10 L2 P4 = offset G57

G10 L2 P5 = offset G58

G10 L2 P6 = offset G59

Block format for tool offsets:

G10 L10 P2 R. . .

where

L10 = tool offsets

P2 = offset register number (tool 2)

R = value of offset

Block format for cutter radius offsets:

G10 L12 P5 R. . .

where

L12 = cutter radius

P5 = offset register number (tool 5)

R = value of offset

Block format for wear offsets:

G10 L13 P6 R. . .

where

L13 = wear offsets

P6 = offset register number (tool 6)

R = value of offset

Note: The function G10 is used to change the work coordinate values and other offsets. Therefore, if you plan to use the new coordinate system, call directly for the newly entered system in the consecutive block (in the example, G59). Before G10, you should use function G90 to replace the existing offset value and G91 to increase or decrease the existing offset value by the amount input. This G10 format varies with control models and may be an option. Consult the specific manufacturer control manuals.

POLAR COORDINATES CANCELLATION (G15)

This command is used to cancel use of the polar coordinate system called by function G16. This command must be programmed in a line, by itself.

POLAR COORDINATE SYSTEM (G16)

The polar coordinate system may be used to program the locations for holes in a bolt circle. By locating the bolt circle center, using the Local Coordinate System function G52, a rotation angle and circle radius can be programmed to locate the holes (unless the center is X0.0 Y0.0). The programmed values in the X-axis represent the circle radius; in the Y-axis, they represent the angular. The angular values may be programmed as either positive or negative. Positive rotation is counterclockwise and negative rotation is clockwise.

PLANE SELECTION (G17, G18, G19)

G17 = XY plane

G18 = XZ plane

G19 = YZ plane

By declaring the selection of plane G17, G18, and G19, the machine selects the given plane (Figure 4-20). The default plane on a vertical machining center is the G17 XY plane. Therefore, it is not necessary to input G17, but it should be part of the Safety Block, especially when other planes are selected within the program.
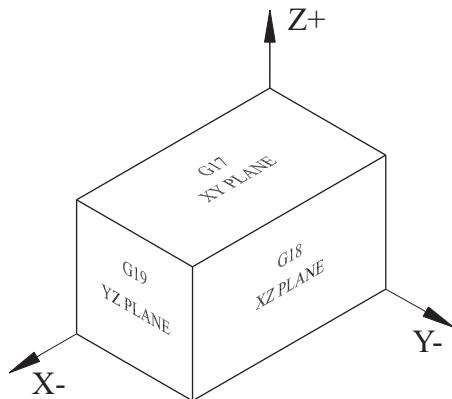
Figure 4-20
Plane Selection

INPUT IN INCHES (G20) AND INPUT IN MILLIMETERS (G21)

Function G20 or G21 is entered at the beginning of the program in the Safety Block to establish the measurement system. Either applies to the whole program. Functions G20 and G21 cannot be interchanged during programming. When using functions G20 or G21, the values are the same units of measure for:

F = feed rate

Position of X, Y, and Z

Offset values

The default measurement system for most American machines is inch. Therefore, it is not necessary to use the above functions unless the opposite system is required. The desired system should always be stated within the Safety Block.

STORED STROKE LIMIT ON/OFF (G22, G23)

Function G22 identifies the stored stroke limit for the tool area outside the working envelope. Function G23 identifies the stored stroke limit for the tool area inside the working envelope.

In order to define the work envelope for the machining center, the manufacturer enters these stroke limits of the axis travel into the controller memory. In a case where the tool is positioned beyond the limited travel area (outside the limits of the stroke), the machine will stop and an alarm is displayed on the screen.

REFERENCE POSITION RETURN CHECK (G27)

This function checks for the accurate return to the reference point and confirms the programmed position has been reached. Entering function G27 causes rapid traverse of the tool to the reference point. If the tool is positioned accurately at the reference point, the axis LEDs will light up. If the LEDs do not go on, the tool is displaced from the reference point and an alarm will result. This might occur if, prior to the G27 command, cancellation of the tool radius compensation has been omitted (i.e., function G40 is omitted following the use of functions G41 and G42). This will cause a position displacement of the tool equal to the value of the compensation offset and the execution of the following program block with the same displacement.

REFERENCE POSITION RETURN (G28)

This function is the automatic reference point return through an intermediate point programmed in the X, Y, and/or Z axes. The machine will position at rapid traverse (G00) to the programmed intermediate point coordinate values and then to the reference point of machine zero. This function is commonly used before an Automatic Tool Change (ATC). After using function G28, the axis LEDs on the control panel should light up to indicate successful return to zero for the axis. When you use function G28, you must specify the point through which the tool passes on its way to zero. If the command, G28 X0.0 Y0.0 Z0.0 is entered in the incremental mode (G91), the machine will position at rapid traverse to the reference position in all three axes simultaneously. Caution must be exercised so as not to interfere with the work holding device or part. It is a good practice to use G28 with a Z-axis positioning move in a prior block to ensure clearance.

The following example demonstrates the best method for using function G28 by first positioning the Z-axis to a level for clearance of the work holding (Figure 4-21).

Figure 4-21
Reference Position Return (G28)

Example

Function G90 is active.

N50 G28 Z3.00

N55 G28 X6.00 Y7.00

where

    X, Y, and Z = the coordinates of the point through which the tool passes on its way to Machine Zero

P_1 = present position

P_2 = point through which the tool passes

P_3 = Machine Zero position

Function G91 is activated.

N50 G91 G28 Z0.0

N55 G28 X0.0 Y0.0
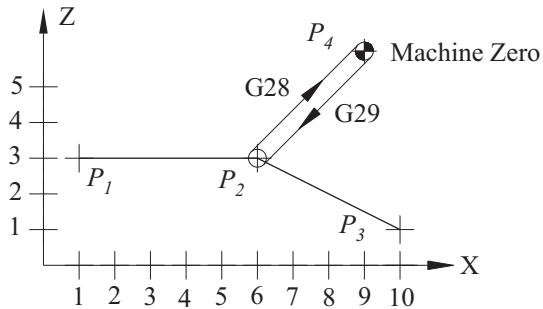
Figure 4-22
Return from the Reference Position

where

In line N50, the tool will retract along the Z-axis until it reaches the Machine Zero position. Then in the next block N55, the machine will move to the X and Y Machine Zero coordinates.

RETURN FROM THE REFERENCE POSITION (G29)

To use function G29, it must follow function G28 or G30. It can be applied after an automatic tool change so that the tool will return to the work position at rapid traverse, as it does similarly does in G00. The tool will travel through the intermediate point programmed (Figure 4-22).

Example

G91 = incremental positioning command

N50 G28 X5.0 Y0.0$P_1 - P_2 - R$

N55 M06

N60 G29 X4.0 Y–1.5 R $- P_2 - P_3$

where

P_1 = initial work point of tool 1

P_2 = end work point of tool 1

P_2 = initial work point of tool 2

P_3 = end work point of tool 2

P_4 = Machine Zero

During the execution of the block containing function G29, the tool automatically follows the return path from point P_4 to P_2 and the programming is limited only to entering the difference between points P_2 and P_3.

RETURN TO SECOND, THIRD, AND FOURTH REFERENCE POSITIONS (G30)

Usually Machine Zero coordinates are coincidental to the reference point assigned to function G28. However, if the reference point assigned to function G28 does

213

not correspond to Machine Zero, function G30 must be used in order to place the tool in this position. (This may be the case if the tool change position is not the same point as is specified in function G28.) Before applying function G30, apply function G28. The movement determined by function G30 is rapid traverse. Functions G28, G29, and G30 can be used to program in both absolute and incremental systems. Cutter compensations (length and diameter) must be cancelled prior to this command.

CUTTER COMPENSATION (G40, G41, G42)

The use of cutter compensation allows the programmer to use part geometry that exactly matches the engineering drawing/blueprint for programmed coordinates. Without using compensation, the programmer must always know the cutter size and offset the programmed coordinates for the geometry by the amount of the radius. In this scenario, if a different size cutter is used, the part will not be machined correctly. Cutter compensation adds the advantage for using any size cutter as long as the offset amount is input accurately into the offset register. It is also very effectively used for fine-tuning of dimensional results by minor adjustments to the amount in the offset register.

G40 = Cutter Compensation Cancel

G41 = Cutter Compensation Left

G42 = Cutter Compensation Right

Cutter Compensation Cancellation (G40)

Function G40 is used to cancel cutter radius compensation initiated by G41 or G42. It should be programmed after the cut using the compensation is completed by moving away from the finished part in a linear (G01) or rapid traverse (G00) move by at least the radius of the tool. Care should be taken here because if the cancellation is on a line without movement, the cutter will move unpredictably in the opposite direction and may damage the part.

Cutter Radius Compensation Left and Right (G41 and G42)

Functions G41 and G42 offset the programmed tool position to the left (G41) or right (G42) respectively by the value of the tool radius entered into offset registers and called in the program by the letter address D. For each tool, enter the corresponding offset amount in the Radius Geometry column of the Tool Data register. In the program, the letter D and the number of the offset (two digits) are input to initiate the compensation call.

The direction the tool is offset, to the left or the right, depends upon which direction the tool is traveling. To accomplish climb cutting with right-hand tools, always use G41; for conventional cutting, use G42. Consider which direction of offset is needed by facing the direction the tool is going to travel, then observe which side of the part the cutter will be—to the left for G41 or to the right for G42 (Figure 4-23).
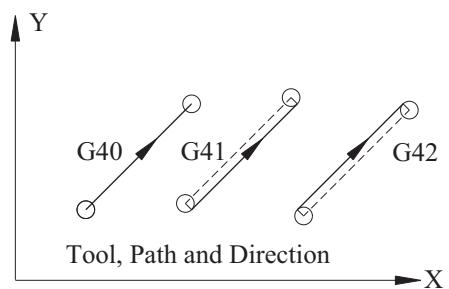
Figure 4-23
Cutter Compensation

Part 4 Programming CNC Machining Centers

Procedure for Initiating Cutter Compensation

Position the tool in the X and Y axes to a point away from the required finished geometry. Then program a linear move that is larger than the radius of the cutter to feed into the part, e.g., G01, G41, or G42 offset direction, X or Y absolute coordinate, and D offset number. To use the cutter compensation properly, there needs to be one full line of movement to position the tool on the proper vector to cut the part. Once this is accomplished, program the part geometry per print.

The tool will not be positioned to the actual programmed point on the geometry. Rather, it will be positioned to a point plus or minus the offset value of the cutter called by D, and the edge of the cutter will be aligned with finished part geometry.

Rules for Cutter Compensation Use

When cutter compensation codes are encountered in a program, the control does what is called "look-ahead". To set up the appropriate vector needed to position the offset amount required, the control looks ahead two program lines for each move.

Once G41 or G42 commands have been called up, movement must be maintained. By following this rule, over-cutting of the part can be avoided. If two lines of non-movement commands are placed consecutively after cutter compensation is called up, the control will ignore functions G41 or G42; the part will then be cut incorrectly.

Do not start cutter compensation G41 or G42 when either G02 or G03 is in effect.

Before a change from left to right compensation is made, or right to left, you must cancel the first compensation. The second compensation may then be called and; thus, the transition of the tool position vector will not conflict.

When machining an inside radius, the radius must be larger than the offset of the tool. Otherwise, the control will stop the program and an alarm will be displayed.

The move used to call up cutter compensation must be larger than the radius of the tool used.

Example: Cutter Compensation, Program 408

Offset for tool number one is D01.

Offset for tool number two is D02.
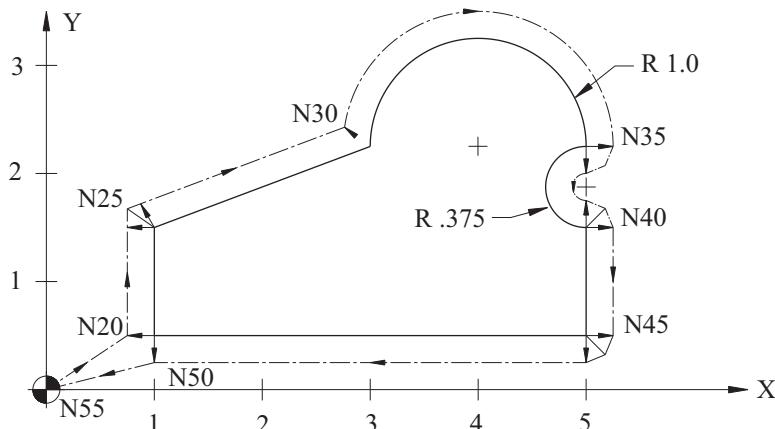
Figure 4-24
Drawing for Cutter
Compensation
Example

Part 4 Programming CNC Machining Centers

In the following program, note there are no Z-axis moves. The intent here is to demonstrate the usage of cutter radius compensation only. The program line sequence numbers in Figure 4-24 correspond with Program 408. The arrows along the tool center-line path indicate the travel direction. At each intersection point on the geometry, the arrows indicate the tool offset direction.

> **O0408**
>
> **N10 G90 G17 G20 G80 G40 G49**
>
> **... .. .. ..**
>
> **... .. .. ..**
>
> **N20 G01 G41 X1.00 Y.50 D01 F10.0**
>
> **N25 G01 Y1.5**
>
> **N30 G01 X3.00 Y2.25**
>
> **N35 G02 X5.00 Y2.25 I1.00**
>
> **N40 G03 X5.00 Y1.50 J−.375**
>
> **N45 G01 Y.50**
>
> **N50 X1.00**
>
> **N55 G00 G40 X0.0 Y0.0**
>
> **N60 M05**
>
> **N65 M30**

Notes: At the end of the tools work, function G40 must be applied to cancel any previously entered compensation value of offset D.

Using cutter compensation functions G41 and G42 in the same block along with functions G02 and G03 causes displacement of the center of an arc by the amount in the offset register. This action will result in a controller alarm and the actual path would resemble Figure 4-25.
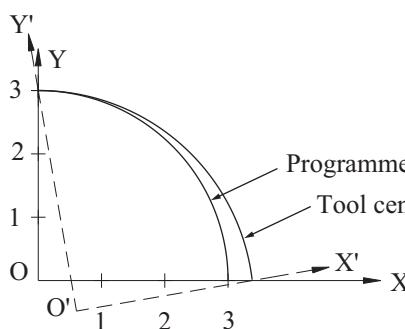
Example

> G41 D01 G02 X3.0Y−3.0J−3.0
>
> O = programmed center of an arc
>
> *O'*= displaced center of an arc

Figure 4-25
Error Caused by Using Cutter
Compensation in an Arc Move

Part 4 Programming CNC Machining Centers

Radial Offset Vector

The radial offset vector is defined as the distance between the center of the tool and the programmed contour. The offset vector is always perpendicular to the programmed line or arc. The offset vector is two-dimensional (for two axes) and depends on the choice of plane (G17, G18, or G19).

The programming manuals supplied with the machine tool provide examples of the tool path followed when the execution of functions G41 and G42 are initiated and then cancelled by function G40. When the angles of the machined part are less than or equal to180 degrees, two types of tool approaches, A and B (to and from the machined part) can be found when using functions G41 and G42. Types A or B are permanently stored in the machine's control parameters.

Special Cases

The radius of the machined workpiece is smaller than the tool radius.

The control has a look-ahead feature that always reads one or more lines of the program in advance. For Figures 4-26 and 4-27, at the end of the execution of block N1, an alarm will be displayed on the screen. The execution will then be interrupted. If the execution of the program proceeds line by line (the SINGLE BLOCK button is ON) after the completion of operation N1, the control will not read the information contained in N2. Therefore, it cannot offset the tool path to compensate for the condition and will undercut the edge in U1.

In both cases, a similar situation will be encountered with undercut U2. The width of the undercut is smaller than the tool diameter.

Example: Cutter Compensation, Program 409

We will use cutter compensation to machine the 3/4 depth step and 2.0 diameter hole for the part shown in Figure 4-28. The tools required are listed on Chart 409.

Machine: Machining Center		Program Number: O0409	
Workpiece Zero: X = <u>Lower Left Corner</u> Y = <u>Lower Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = 303 Stainless Steel, 3.5 x 3.5 x 1.25 The blank is placed on 1.0" parallels			
Tool #	Description	Offset	Comments
T1	#5 Center Drill		SFM= 100 FPT = .002
T2	1.25 Diameter Drill		SFM= 100 FPT = .003
T3	1.25 Diameter 6-Flute HSS Roughing Cutter	Geometry Offset Radius Column .650 D3	SFM= 100 FPT = .003
T4	1.25 Diameter, 6-Flute HSS Finishing End Mill	Geometry Offset Radius Column .625 D4	SFM= 100 FPT = .002

Figure 4-26
Cutter Compensation for Smaller Radius
than Tool (Case 1)
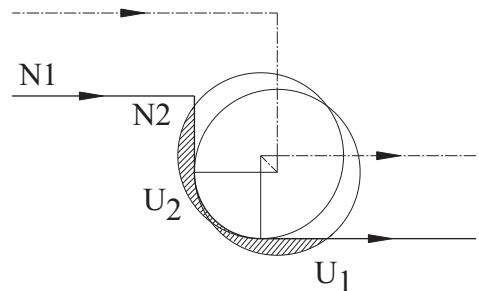
Figure 4-27
Cutter Compensation for Smaller
Radius than Tool (Case 2)
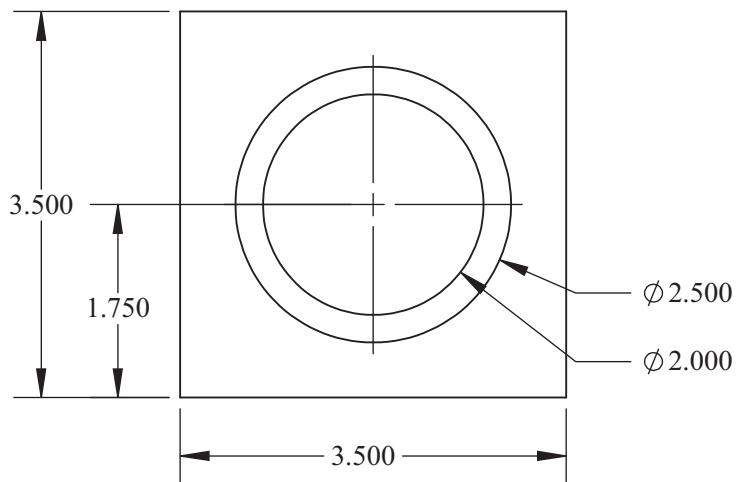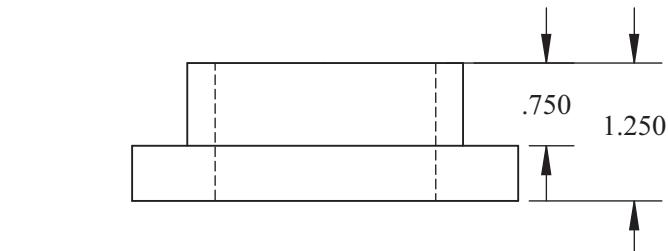
Figure 4-28
Drawing for Cutter Compensation Example 2

O0409

(#5 CENTER DRILL)

N10 G90 G20 G80 G40 G49

N15 T01 M06

N20 G00 G54 X1.75 Y1.75 S915 M03 T02

N25 G43 Z.1 H01 M08

N3 G81 G98 Z–.382 R.1 F4.0

N35 G00 G80 Z1.0 M09

N40 G91 G28 Z0.0

N45 M01

(1.25 DIAMETER DRILL)

N50 T02 M06

N55 G90 G80 G40 G49

N60 G00 G54 X1.75 Y1.75 S320 M03 T03

N65 G43 Z.1 H02 M08

N70 G83 G98 Z–1.625 R.1 Q.625 F2.0

N75 G00 G80 Z1.0 M09

N80 G91 G28 Z0.0

N85 M01

(1.25 DIAMETER 6-FL ROUGHING END MILL, CUTTER COMP #D3)

N90 T03 M06

N95 G90 G80 G40 G49

N100 G00 G54 X1.75 Y1.75 S320 M03 T04

N105 G43 Z.1 H03 M08

N110 Z–1.3 F20.0

N115 G01 G42 Y2.750 D03 F5.76

N120 G02 J–1.0

N125 G01 G40 Y1.750 F10.0

N130 G00 Z.1

N135 Y–1.0

N140 G01 Z–.73 F20.0

N145 G42 Y.5 D03

N150 G03 J1.25 F6.0

N155 G01 G40 Y–1.0 F20.0

N160 G00 Z.1 M09

N165 G91 G28 Z0.0

N170 M01

(1.25 DIAMETER 6-FL FINISHING END MILL, CUTTER COMP #D4)

N175 T04 M06

N180 G90 G80 G40 G49

N18 G00 G54 X1.75 Y1.75 S320 M03 T01

N190 G43 Z.1 H04 M08

N195 G41 Y2.75 F10.0 D04

N200 G01 Z–1.3 F50.0

N205 G03 J–1.0 F3.84

N210 G01 G40 Y1.75 F20.0

N215 G00 Z.1

N220 Y–1.0

N225 Z–.75

N230 G01 G41 Y0.50 F10.0 D04

N235 G02 J1.25 F4.0

N240 G01 G40 Y–1.0 F20.0

N245 G00 Z.1 M09

N250 G91 G28 Z0.0 M05

N255 G28 X0.0 Y0.0
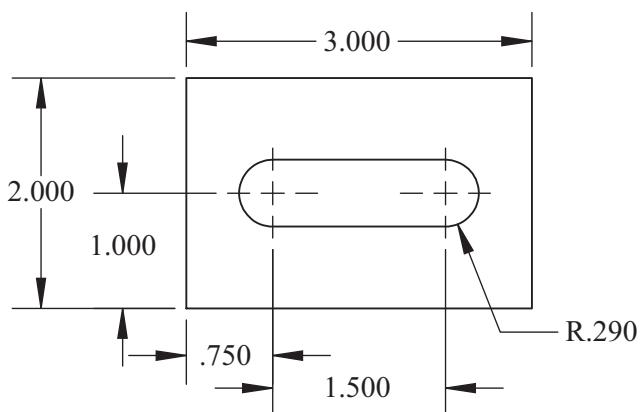
N260 T01 M06

N265 M30

Example: Cutter Compensation, Program 410

For this example we will use cutter compensation to machine the .580 width slot for the part shown in Figure 4-29. The tools required are listed on Chart 410.

Figure 4-29
Drawing for Cutter
Compensation Example 3

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0410	
Workpiece Zero: X = Lower Left Corner Y = Lower Left Corner Z= Top Part Surface Setup Description: Material = Aluminum 6061 The blank is placed on 1.250" parallels			
Tool #	Description	Offset	Comments
T1	#5 Center Drill		SFM= 300 FPT = .003
T2	17/32 Diameter HSS Drill		SFM= 300 FPT = .003
T3	.500 Diameter 2-Flute HSS Roughing Cutter	Geometry Offset Radius Column .260 D3	SFM= 850 FPT = .003
T4	.500 Diameter 2-Flute HSS Finish Cutter	Geometry Offset Radius Column .250 D4	SFM= 850 FPT = .002

O0410

N10 G90 G80 G20 G40 G49

(#5 CENTER DRILL)

N15 T01 M06

N20 G00 G54 X.750 Y1.0 S2200 M03 T02

N25 Z.1 H01 M08

N3 0 G81 G98 Z–.3 R.1 F8.8

N35 G80 Z1.0 M09

N40 G91 G28 Z0.0

N45 M01

(17/32 DRILL)

N50 T02 M06

N55 G90 G80 G20 G40 G49

N60 G00 G54 X.75 Y1.0 S2150 M03 T03

N65 G43 Z1. H02 M08

N70 G81 G98 Z–.70 R.1 F8.0

N75 G80 Z1.0 M09

N80 G91 G28 Z0.0

N85 M01

(1/2 2-FL ROUGHING END MILL, CUTTER COMP #D3)

N90 T03 M06

N95 G90 G80 G40 G49

N100 G00 G54 X.75 Y1.0 S2200 M03
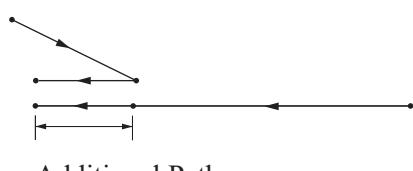
N105 G43 Z1.0 H03 M08

N110 Z.1N115 G01 Z–.52 F50.0

N120 G41 Y.71 F4.5 D03

N125 X2.25

N130 G03 Y1.29 J.29

N135 G01 X.75

N140 G03 Y.71 J–.29

N145 G01 G40 Y1.0

N150 G00 Z1.0 M09

N155 G91 G28 Z0.0

N160 M01

(1/2 2-FL FINISHING END MILL, CUTTER COMP #D04)

N165 T04 M06

N170 G90 G80 G40 G49 M03

N180 G43 Z1.0 H04 M08

N185 Z.1

N190 G01 Z–.52 F50.0

N195 G41 Y1.29 F8.8 D05

N200 G03 Y.71 J–.29

N205 G01 X2.25

N210 G03 Y1.29 J.29

N215 G01 X.75

N220 G40 Y1.0

N225 G00 Z1.0 M09

N230 G91 G28 Z0.0 M05

N235 G28 X0.0 Y0.0

N240 M30

SINGLE-DIRECTION POSITIONING (G60)

Usually, function G60 is applied when accuracy is the prime factor in determining distances between points. The tool will always approach the programmed point from one direction only (Figure 4-30).

The value of the additional path is set by parameter in the control. Function G60 eliminates the undesirable influence of gear and feed-screw play (backlash) on the accurate positioning of the programmed point.

Additional Path

Figure 4-30
Single-Direction Positioning

CANNED CYCLE FUNCTIONS

The function of a Canned Cycle is defined as a set of operations assigned to one block and performed automatically without any possibility of interruption. Usually, it is a set of six operations, as follows:

1. Positioning of the X and Y axes at rapid traverse

2. A rapid traverse move to an initial clearance level plane (G98)

3. The machining cycle is executed (drill, bore, etc.).

4. A dwell or other operation is executed at the bottom of the hole.

5. A rapid traverse return to the R level plane along the Z-axis (G99)

6. A rapid traverse return to the initial level plane along the Z-axis

The block format is as follows:

N. . . G. . . G. . . X. . .Y. . . Z. . . R. . . Q. . . P. . . F. . . K. . . L. . .

where

N = the block number

G = the type of cycle function

G = initial or R level return G98/G99

X, Y = the hole position (positioning is carried out by rapid traverse)

Z = the depth of the hole

R = the distance between plane R and the surface of the material

Level R refers to the horizontal plane, positioned closely above the material on which the tool tip moves (commonly .100 in. or .08 mm). The programmed value of R is valid until the new value is entered. It does not have to be included in every block. The tool will return to this level at the end of each hole drilled.

Q = the depth of cut (drilling) for individual pecks (not used for every drilling cycle)

P = the dwell time for the drill while rotating at the bottom of the hole (not used for every drilling cycle)

The dwell in seconds is for the purpose of complete removal of excess material.

F = feed rate in inches per minute (in/min)

K = the number of repeats

L = the number of holes incrementally spaced

When K is used with G91, L represents the number of holes incrementally spaced by the amount entered as the X or Y position coordinate. (If L does not appear in the block, that means machining of only one hole, L = 1.) On some modern controls, the letter address K is used in the same manner. Figure 4-31 shows tool movements as follows:

1. Positioning with rapid traverse

2. Rapid traverse to level R along Z-axis

3. Feed traverse to Z depth

4. Operations performed on the bottom of the hole

5. Return to level R (G99)

6. Return to plane level (G98)
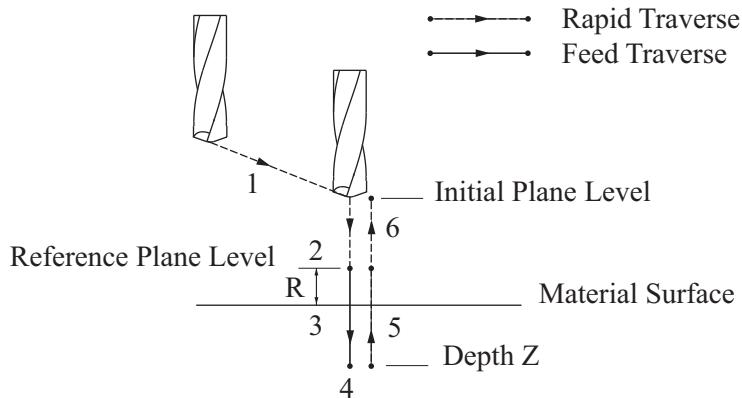
Figure 4-31
Canned Cycle Drilling Functions

HIGH SPEED PECK DRILLING CYCLE (G73)

Block format:

G73 X. . . Y. . . Z. . . R. . . Q. . . F. . .

Figure 4-32 explanations:

1. G00—rapid traverse to a position in X or Y axes

2. G00—rapid traverse to plane R
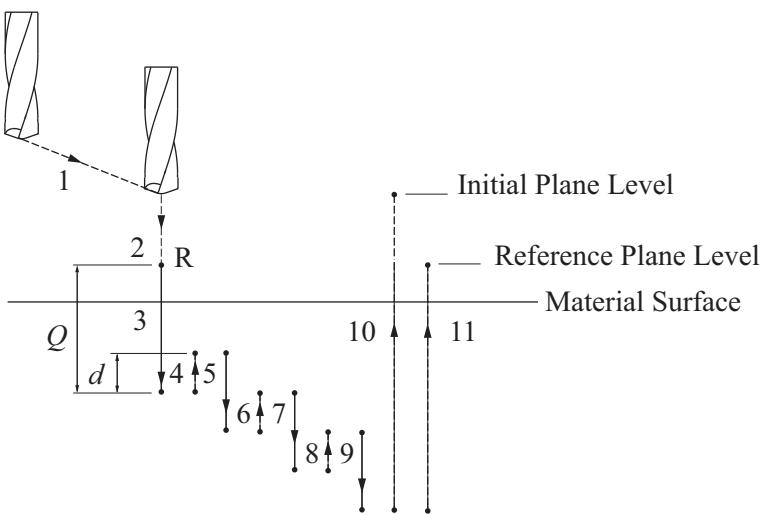
Figure 4-32
High Speed Peck Drilling Cycle, G73

3, 5, 7, 9.   G01—feed traverse along Z to depth Q

4, 6, 8.   G00—rapid traverse upwards by a value specified
Established by parameter and shown here as amount "d".

10. G00—rapid traverse to a plane level assigned to function G98

11. G00—rapid traverse to plane R assigned to function G99

Q. —the depth of cut for each drilling peck.

d. —the value of the upward traverse that is used to accomplish momentary inter-
ruption, removal of chips, and the delivery of coolant to the bottom of the
hole. The value of d is entered into the parameters of the machine's control.

LEFT-HANDED TAPPING CYCLE (G74)

Block format:

G74 X. . . Y. . . Z. . . R. . . P. . . F. . .

The spindle is rotating in the counterclockwise direction by using the M04 com-
mand with a value of S.

Figure 4-33 explanations:

1. G00—rapid traverse to a position in X or Y axes

2. G00—rapid traverse to plane R

3. G01—feed traverse along Z to depth

P. —dwell time in seconds at the bottom of the hole

4. —spindle direction is reversed to clockwise

5. G01—feed traverse along the Z to the R plane

6. —spindle direction is reversed to counterclockwise

7. G00—rapid traverse to initial plane if G98 is used
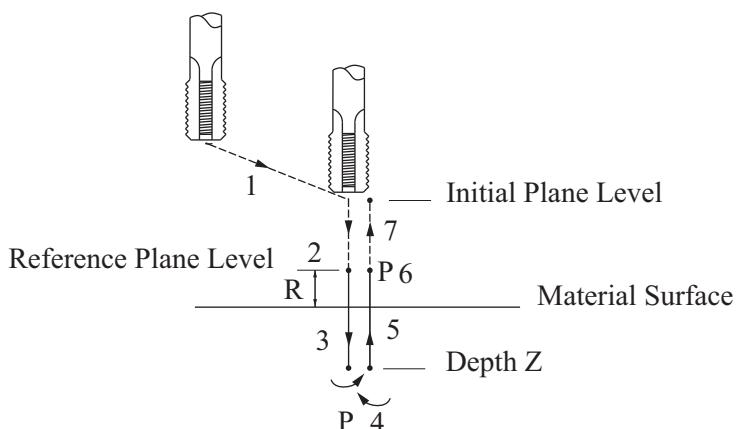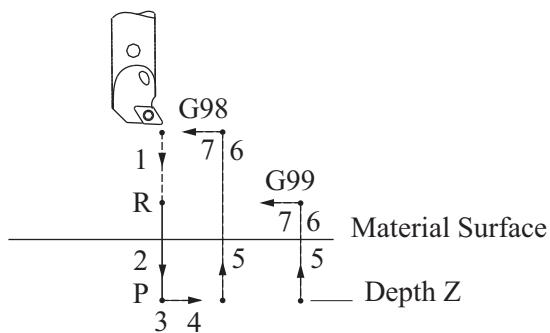
Figure 4-33
Left Handed Tapping Cycle, G74

Figure 4-34
Fine Boring Cycle, G76

FINE BORING CYCLE (G76)

Block format:

G76 X. . . Y. . . Z. . . R. . . Q. . . P. . . F. . .

Figure 4-34 explanations:

1. G00—rapid traverse to point R

2. G01—feed traverse to Z depth

P. —dwell time in seconds

3. M19—miscellaneous function M19

Figure 4-35
Spindle Orientation

Miscellaneous function M19 initiates orientation of the spindle position and the interruption of revolutions. At function M19, the cutting edge always rotates and stops in the same position. The cutting edge stops so that it is perpendicular to the X- or Y-axis. This activity takes place without inserting the miscellaneous function M19 into the program. It is a part of the G76 canned cycle that is built-in.

4. G00—rapid displacement along the X or Y-axis

The displacement may be in the Y-axis if the cutting edge is perpendicular to the X-axis, with the value of Q (Figure 4-35). The value of Q is entered into the parameters of the control. The value of Q must be known in order to avoid a collision between the tool and the back wall of the hole.

5. G00—rapid traverse from the hole to the reference point R for function G99, or rapid traverse to the initial plane level for function G98

6. G00—rapid displacement of the tool along the X (or Y) axis, with the value Q

7. —M03 activate clockwise spindle rotation in preparation for the boring of the next hole

This boring cycle is usually applied for finishing in which it is intended to obtain a smooth surface, free of scratches. Because of the retract amount specified in Q, there will be no tool mark caused on the finished surface at exit from the hole.

CANNED DRILLING CYCLE CANCELLATION (G80)

This command is used to cancel all canned cycles. It should be entered at the end of each canned cycle machining sequence. This code is also entered in the Safety Block.

Notes for cycles G73 through G89:

- *Revolutions of the spindle (right or left), during the cycle, must be entered in the block preceding the canned cycle block.*
- *Never press the ORIGIN button (zero set) during the canned cycle execution because unpredictable actions may result.*
- *In order to execute the cycle (the drilling of one hole with the SINGLE BLOCK button being ON, after the execution of each block when the machine stops), you must press the CYCLE START button three times to continue.*
- *If feed hold is pressed during a threading canned cycle operation, the cycle will be completed before stopping.*

CANNED CYCLE, SPOT DWELLING (G81)

Block format:

G81 X. . . Y. . . Z. . . R. . . F. . .

Figure explanations:

1. G00—rapid traverse to R
2. G01—feed traverse to Z depth
3. G00—rapid traverse to R (G99) or
4. G00—rapid traverse to initial level plane (G98)

Example: Canned Cycle, Spot Drilling, Program 411

This is an example of drilling a hole with a diameter of .152 and a countersink with a diameter of .279 inch (Figure 4-37 and Chart 411).
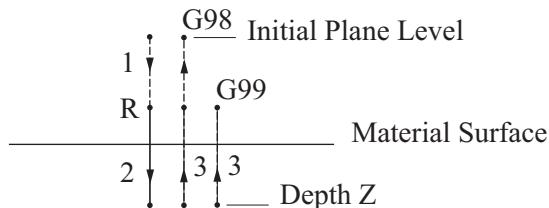
Figure 4-36
Canned Cycle, Spot Drilling, G81

Program

    O0411

    N10 G90 G80 G20 G40 G49

    N15 T01 M06

    N20 G00 G54 X.5 Y–.5 S2000 M03 T02

N25 G43 Z1.0 H01 M08

N30 G81 G98 Z–.279 R.1 F12.0

N35 G80 Z1.0 M09

N40 G91 G28 Z0.0

N45 M01

N50 M06

N55 G90 G80 G40 G49

N60 G00 G54 X.5 Y–.5 S2894 M03

N65 G43 Z1.0 H02 M08

N70 G81 G98 Z–.5 R.1 F11.5

N75 G80 Z1.0 M09

N80 G91 G28 Z0.0

N85 G28 X0.0 Y0.0 M05

N90 M30
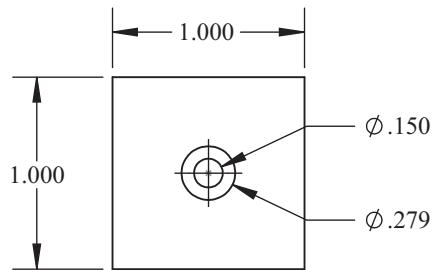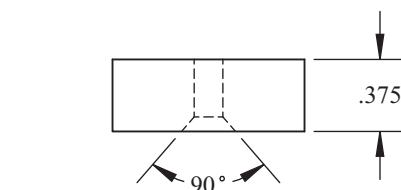
Figure 4-37
Drawing for Canned Cycle,
Spot Drilling Example

Machine: Machining Center		Program Number: O0411	
Workpiece Zero: X = <u>Upper Left Corner</u> Y = <u>Upper Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = 1018 Low Carbon Steel Mount the workpiece in a vise with a part stop on the left side The blank is placed on 1.250" parallels			
Tool #	Description	Offset	Comments
T1	.5 Carbide 90 Degree Spot Drill		SFM= 250 FPT = .006
T2	#24 .152 Diameter HSS Drill		SFM= 110 FPT = .002

COUNTER BORING CYCLE (G82)

In function G82, the feed is interrupted in the drilling cycle (while the spindle is ON) at the bottom of the hole for time specified by P.

Block format:

G82 X. . . Y. . . Z. . . R. . . P. . . F. . .

Process explanations:

1. G00—rapid traverse to reference level R
2. G01—feed traverse to Z depth
3. —interruption of the feed for time duration P, in order to remove the material at the bottom of the hole
4. G00—rapid traverse to initial plane level for G98
5. G00—rapid traverse to reference plane level R for G99

Example: Counter Boring Cycle, Program 412

In this example a counter bored (step hole) bolt hole is drilled as shown in Figure 4-38. Tool and setup information are given in Chart 412.

O0412
N10 G90 G80 G20 G40 G49
N15 T1 M06
N25 G00 G54 X1.0 Y−.75 S4800 M03 T02
N30 G43 Z1.0 H01 M08
N35 G81 G98 Z−.4 R.1 F24.0
N37 X5.0
N40 G80 Z1.0 M09
N45 G91 G28 Z0.0
N50 M01
N55 M06
N60 G90 G80 G40 G49
N65 G00 G54 X1.0 Y−.75 S2015 M03 T03
N70 G43 Z1.0 H02 M08
N75 G81 G98 Z−.7 F8.06 R.1
N77 X5.0
N80 G80 Z1.0 M09
N85 G91 G28 Z0.0
N90 M01
N95 M06
N100 G90 G80 G40 G49
N105 G00 G54 X1.0 Y−.75 S1280 M03 T01

N110 G43 Z1.0 H03 M08

N115 G82 G98 Z–.375 R.1 P200 F5.12

N117 X5.0

N120 G80 Z1.0 M09

N125 G91 G28 Z0.0

N130 G28 X0.0 Y0.0 M05

N135 M30

Note: On some controls, if no value is assigned for P to function G82, its value will be automatically selected by the control. If you do enter a value for dwell P (for example, P1000 = 1 second), then the constant value included in the parameters of the machine will be ignored.
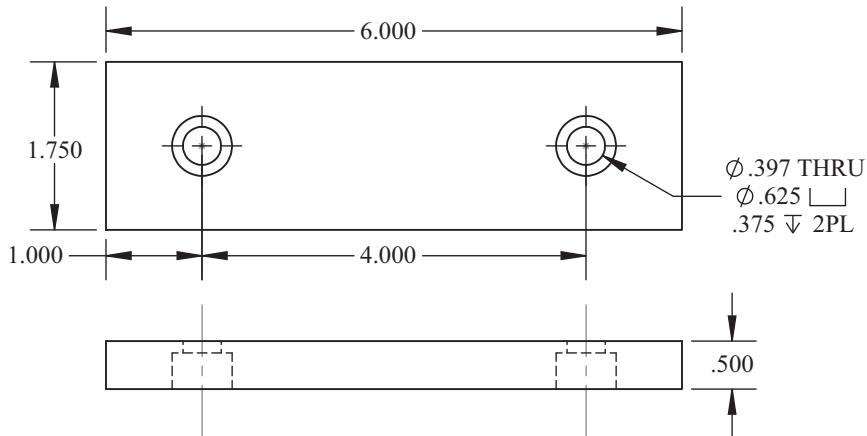
Figure 4-38
Drawing for Counter Boring Cycle, G82 Example

Machine: Machining Center		Program Number: O0412	
Workpiece Zero: X = Upper Left Corner Y = Upper Left Corner Z= Top Part Surface Setup Description: Material = 6061 Aluminum The blank is placed on 1.50" parallels			
Tool #	Description	Offset	Comments
T1	.5 Carbide 90 Degree Spot Drill		SFM= 600 FPT = .020
T2	Letter X .397 Diameter HSS Drill		SFM= 200 FPT = .002
T3	.625 HSS End Mill		SFM= 200 FPT = .003

DEEP HOLE PECK DRILLING CYCLE (G83)

Block format:

G83 X. . . Y. . . Z. . . R. . . Q. . . F. . .

Figure 4-39 explanations:

1.   G00—rapid traverse to R level

2.   G01—feed traverse with length Q (peck amount)

3, 6, 9. G00—rapid return traverse to R

4, 7, 10. G00—rapid traverse to the depth previously drilled, less the value of d

5, 8, 11. G01—feed traverse increased by the value of d

12. G00—rapid traverse to initial plane level for G98

12'. G00—rapid traverse to reference plane level R for G99

This drilling cycle is used to drill exceptionally deep holes. As the drill reaches the depth identified by Q, the drill then returns at rapid traverse to the R level point, allowing the removal of chips and the delivery of coolant to the bottom of the hole. Entering a given depth of Z into the control enables it to calculate the number of feed traverses necessary for Q. Q can be any incremental amount desired smaller than the total Z-axis travel. The value of Q does not need to have a common factor with the dimension Z. The value of d is set by parameter.

Example: Deep Hole Peck Drilling Cycle, Program 413

In this example, the hole in Figure 4-40 is drilled. Tool and setup information are given in Chart 413.
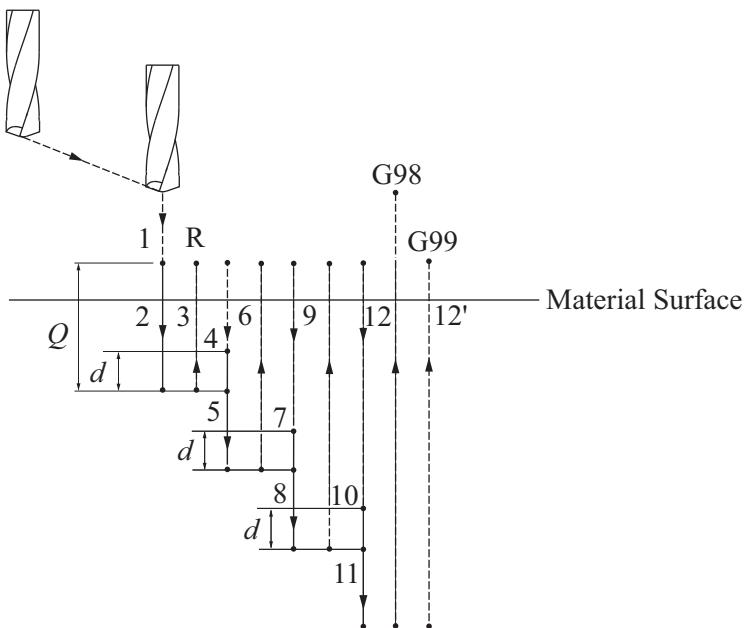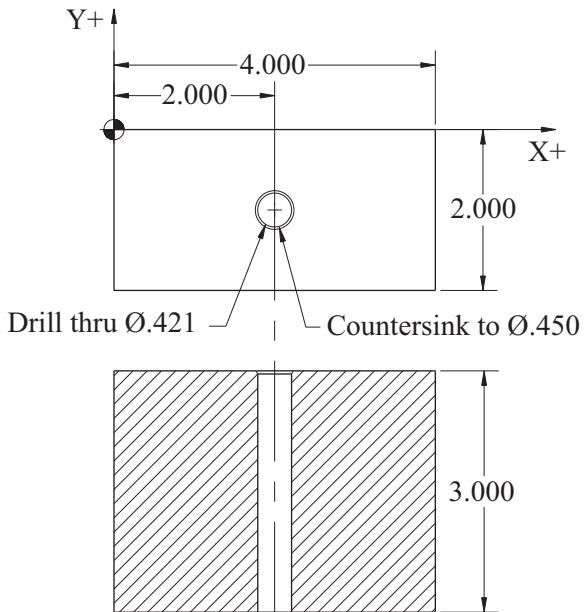
Figure 4-39
Deep Hole Peck Drilling Cycle, G83

Figure 4-40
Drawing for Deep Hole Peck
Drilling Example

Drill thru Ø.421 — Countersink to Ø.450

Machine: Machining Center		Program Number: O0413	
Workpiece Zero: X = Upper Left Corner Y = Upper Left Corner Z= Top Part Surface Setup Description: Material = 1018 Low Carbon Steel The blank is placed on .50" parallels			
Tool #	Description	Offset	Comments
T1	#5 HSS Center Drill		SFM= 110 FPT = .002
T2	.421 Diameter HSS Drill		SFM= 110 FPT = .0035

O0413

N10 G90 G80 G20 G40 G49

N15 T1 M06

N20 G00 G54 X2.0 Y–.1 S1006 M03 T02

N25 G43 Z1.0 H01 M08

N30 G81 G99 Z–.38 F4.03 R.1

N35 G80 Z1.0 M09

N40 G91 G28 Z0.0

N45 M01

N50 M06

N55 G90 G80 G40 G49

N60 G00 G54 X2.0 Y–1.0 S1045 M03

N65 G43 Z1.0 H02 M08

N70 G83 G99 Z–3.15 R.05 Q.45 F7.31

N75 G80 Z1.0 M09

N80 G91 G28 Z0.0

N85 G28 X0.0 Y0.0 M05

N90 M30

Note: By using function G83 in block N70, the deep hole, peck drilling cycle is initiated. Starting at level R, the drill feeds by the amount of Q.45 and then, at rapid traverse, returns to the starting point R. The next move advances by the depth Q, decreased by the value of d, which is entered in the parameters of the control. This cycle repeats itself until the drill reaches a depth of Z–3.15. Also, after each feed traverse with the value Q, the tool returns to level R.

TAPPING CYCLE (G84)

Block format:

G84 X. . . Y. . . Z. . . R. . . P. . . F . . .

Figure 4-41 explanations:

1. G00—rapid traverse to R level

2. G01—feed traverse to Z

3. M05—revolutions stop

P. —dwell time at the bottom of the hole

4. M04—counterclockwise revolutions are ON

5. G01—feed traverse to R

6. M05—spindle stop

7. M03—clockwise revolution is ON

8. G00—rapid traverse to a level plane for the function G98
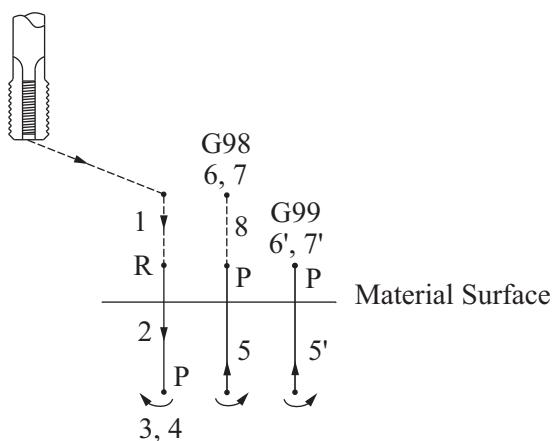
Figure 4-41
Tapping Cycle, G84

Example: Tapping Cycle, Program 414

In this example the 3/8-16 threaded hole in Figure 4-42 is machined. Tool and setup information are given in Chart 414.
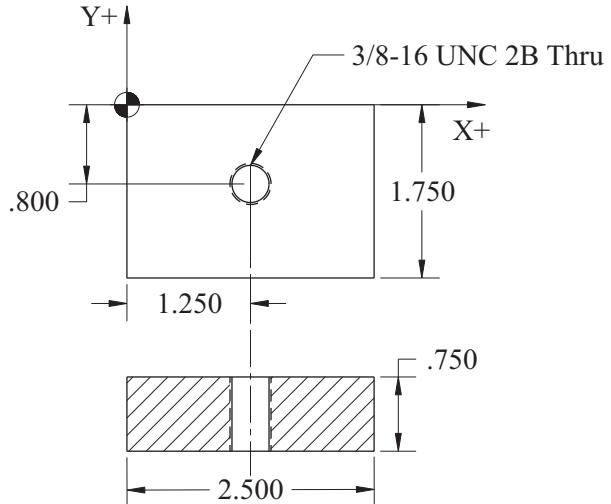
Figure 4-42
Drawing for Tapping Cycle, G84

Machine: Machining Center		Program Number: O0414	
Workpiece Zero: X = <u>Upper Left Corner</u> Y = <u>Upper Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = 1018 Low Carbon Steel The blank is placed on 1.0" parallels			
Tool #	Description	Offset	Comments
T1	#5 HSS Center Drill		SFM= 110 FPT = .002
T2	.3125 Diameter HSS Drill		SFM= 110 FPT = .0035
T3	3/8-16 Tap		SFM = 15 Tap mounted in a floating tap holder

O0414
N10 G90 G80 G20 G40 G49
N15 T1 M6
N20 G00 G54 X1.25 Y–.8 S1045 M03 T02
N25 G43 Z1.0 H01 M08
N30 G81 G99 Z–.38 R.1 F4.18
N35 G80 Z1.0 M01

N40 G91 G28 Z0.0

N45 M01

N50 M06

N55 G80 G40 G49

N60 G54 G00 X1.25 Y–.8 S1408 M03 T03

N65 G43 Z1.0 H02 M08

N70 G81 G99 Z–.85 R.1 F9.85

N75 G80 Z1.0 M09

N80 G91 G28 Z0.0

N85 M01

N90 M06

N95 G80 G40 G49

N100 G00 G54 Z1.25 Y–.8 S152 M03 T01

N105 G43 Z1.0 H03 M08

N110 G84 G99 Z–1.0 R.1 F9.5

N115 G80 Z1.0 M09

N120 G91 G28 Z0.0 M05

N125 G28 X0.0 Y0.0

N130 M30

Note: The feed in block N110 (F9.5) was calculated as follows:

F = 1/16 x 152 = 9.5

1/16 = *the lead of the thread in inches*

152 = *the spindle speed, expressed in revolutions per minute(r/min)*

BORING CYCLES

REAMING CYCLE (G85)

Block format:

G85 X. . . Y. . . Z. . . R. . . F. . . K . . .

Figure 4-43 explanations:

1. G00—rapid traverse to R

2. G01—feed traverse to Z depth

3. G01—feed traverse to the R level plane and then rapid to the level plane assigned to function G98

3'. G01—feed traverse to R for function G99

4. and 4'. M03—the clockwise revolution is ON
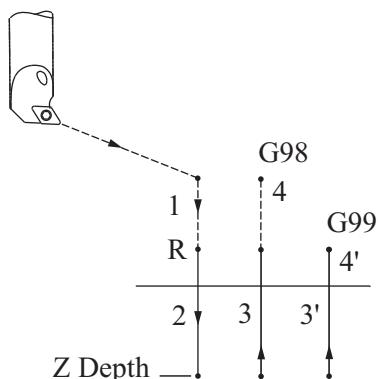
Figure 4-43
Reaming Cycle, G85

BORING CYCLE (G86)

Block format:

G86   X. . . Y. . . Z. . . R. . . F. . .

Figure 4-44 explanations:

1. G00—rapid traverse to R

2. G01—feed traverse to Z depth

3. M05—spindle revolution is stopped

4. G00—rapid traverse to the level plane assigned to function G98

4'. G00—rapid traverse to the R level plane for function G99

5. M03—the clockwise revolution is ON
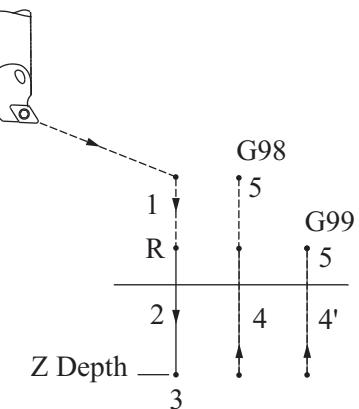
Figure 4-44
Boring Cycle, G86

Example: Canned Drilling and Boring Cycles, Program 415

Figures 4-45, 4-46, 4-47, and 4-48 illustrate Program 415. Tool and setup information are given in Chart 415.
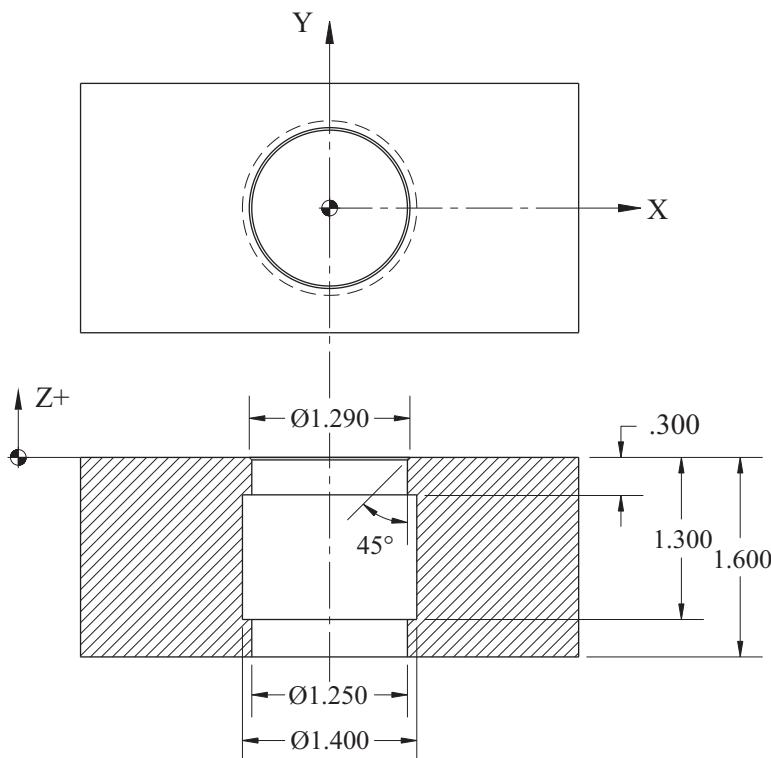
Figure 4-45
Drawing for Canned Drilling and Boring Cycles Example

236

Machine: Machining Center		Program Number: O0415	
Workpiece Zero: X = <u>Upper Left Corner</u> Y = <u>Upper Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = 6061 Aluminum The blank is placed on 1.0" parallels			
Tool #	Description	Offset	Comments
T1	#6 HSS Center Drill		SFM= 300 FPT = .002
T2	1.125 Diameter HSS Drill		SFM= 300 FPT = .0035
T3	Carbide Boring Bar Set to 1.25 Diameter		SFM= 800 FPT = .001
T4	Carbide 45 Degree Chamfer Cutter		SFM= 800 FPT = .001 X = .625-.3-.05 = .545
T5	Carbide Back Boring Bar .125 Wide Insert		SFM= 800 FPT = .001

O0415

N10 G90 G80 G20 G40 G49

N15 T1 M06

(T1 CENTER DRILL FOR BORES)

N20 G00 G54 X0.0 Y0.0 S2291 M03

N25 G43 Z1.0 H01 M08

N30 S2291 M03

N35 G81 G98 Z–.35 F9.16 R.1

N40 G80 Z1.0 M09

N45 G91 G28 Z0.0

N50 M01

N55 T2 M06

(T2 DRILL 1.125 DIAMETER FOR BORES)

N60 G90 G80 G40 G49

N65 G00 G54 X0.0 Y0.0 S1182 M03 T3

N70G43Z1.0H02M08

N75 G73 G98 Q.3 Z–2.0 F8.27 R.1

N80 G80 Z1.0 M09

N85 G91 G28 Z0.0
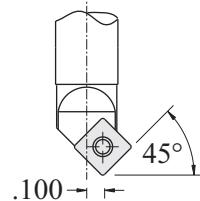
Figure 4-46
Chamfer Cutter
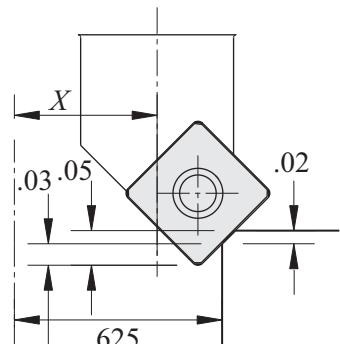
Figure 4-47
Chamfer Cutter Detail

N90 M01

N95 T3 M06

(T3 BORE 1.25 DIAMETER HOLE THROUGH)

N100 G90 G80 G40 G49

N105 G54 G00 X0.0 Y0.0 S2444 M03 T4

N110 G43 Z1.0 H3 M08

N115 G86 G98 Z–1.65 F2.44 R.1

N120 G80 Z5.0 M09

N125 G91 G28 Z0.0

N130 M01

N135 T4 M06

(T4 CHAMFER 45° X .02 ON 1.25 DIAMETER HOLE)

N140 G90 G80 G40 G49

N145 G00 G54 X0.0 Y0.0 S3086 M03 T5

N150 G43 Z1.0 H4 M08

N155 S3086 M03

N160 G01 Z–.05 F50.0

N165 X.545 F3.08

N170 G03 I–.545

N175 G01 X0.0

N180 G00 Z1.0 M09

N185 G91 G28 Z0.0

N190 M01

N195 T5 M06

(T5 BACK BORE 1.400 DIAMETER UNDERCUT)

N200 G90 G80 G40 G49

N205 G00 G54 X0.0 Y0.0 S2182 M03

N210 G43 Z1.0 H05 M08

N215 S2182 M03

N220 G01 Z–.425 F50.0

N225 G91 G41 Y.7 F6.5 D5

(D5 = .500)

N230 G03 J–.7 F2.18

N235 M98 P0002 L14

N240 G01 G40 Y–.7 F50.0
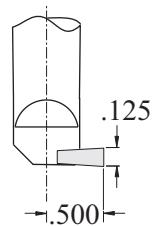
Figure 4-48
Back Boring Bar

.125

.500

  N245 G90 G00 Z5.0 M09

  N250 G91 G28 Z0.0 M05

  N255 G28 X0.0 Y0.0

  N260 M30

  The following subprogram is repeated 14 times to attain the full depth for the 1.400 diameter bore. Boring cycle G87 could be substituted in place of the subroutine (see the following description).

 Subprogram

  O0002

  N10 G01 Z–.1 F2.0

  N15 G03 J–.7 F3.0

  N20 M99

BORING CYCLE (G87)

  Function G87 is used to bore that part of the hole or chamfer on the bottom of the hole (cutting is along the Z-axis in the positive direction).

  Block format:

  G87 X. . . Y. . . Z. . . R. . . Q. . . P. . . F. . .

  Figure 4-49 explanations:

1. G00—rapid traverse of oriented boring bar with a value of Q

2. Non-programmed M19—orientation of boring bar (same as for G76)

3. G00—rapid traverse to Z (bottom of the hole)

4. G00—rapid traverse of boring bar with a value of Q

5. M03—the clockwise rotations are ON

6. G01—work traverse to programmed Z value necessary to attain bore

7. Non-programmed M19—orientation of boring bar

8. G00—rapid traverse of boring bar with a value of Q

9. G00—rapid traverse to the R level plane

10. G00—rapid traverse with a value of Q

11. M03—the clockwise rotation is ON
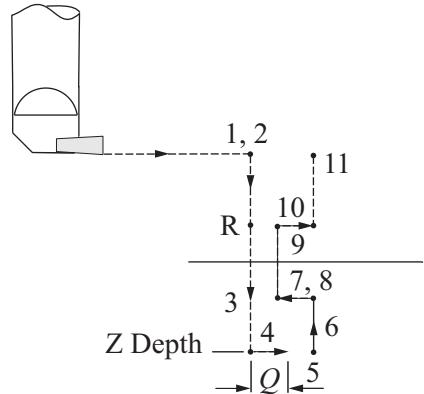
Figure 4-49
Boring Cycle, G87

BORING CYCLE (G88)

Block format:

G88 X. . . Y. . . Z. . . R. . . P. . . F. . . K. . .

Figure 4-50 explanations:

1. G00—rapid traverse to R
2. G01—feed traverse to Z depth
3. G04—temporary interruption of feed for the time period P

   Dwell—for the purpose of complete removal of material from the bottom of the hole
4. M05—the revolutions stop
5. 5, 5'—Manual or mechanical withdrawal of the tool

Figure 4-50
Boring Cycle, G88

After the revolutions are stopped, the tool may be removed from the holder while still at the bottom of the hole.

P—time period of the temporary interruption is given in seconds

BORING CYCLE (G89)

Block format:

G89 X. . . Y. . . Z. . . R. . . P. . . F. . .
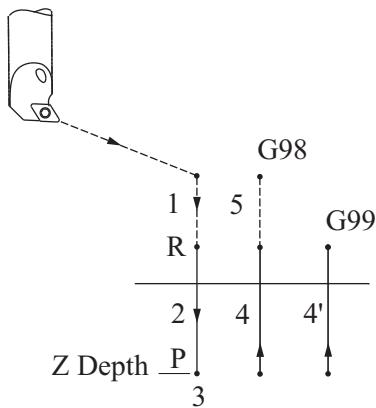
Figure 4-51
Boring Cycle, G89

Figure 4-51 explanations:

1. G00—rapid traverse to R
2. G01—feed traverse to Z depth
3. G04—temporary interruption of feed
4. 4, 4'—G01—feed traverse to R
5. G00—rapid traverse to a level plane for function G98

EXAMPLES OF PROGRAMMING CNC MACHINING CENTERS

COMPARISON OF ABSOLUTE (G90) AND INCREMENTAL (G91) PROGRAMMING

In the absolute system, all dimensions are relative to the fixed origin of the workpiece coordinate system. In the incremental system, every measurement refers to the previously dimensioned position. Incremental dimensions are distances between the adjacent points (from the current tool location).

Part 4 Programming CNC Machining Centers

The following example also provides application of the Initial Level Return function (G98) and R Level Return function (G99) in the Canned Drilling Cycles for the part in Figure 52.

Example: Absolute and Incremental Programming and G98/99, Program 416
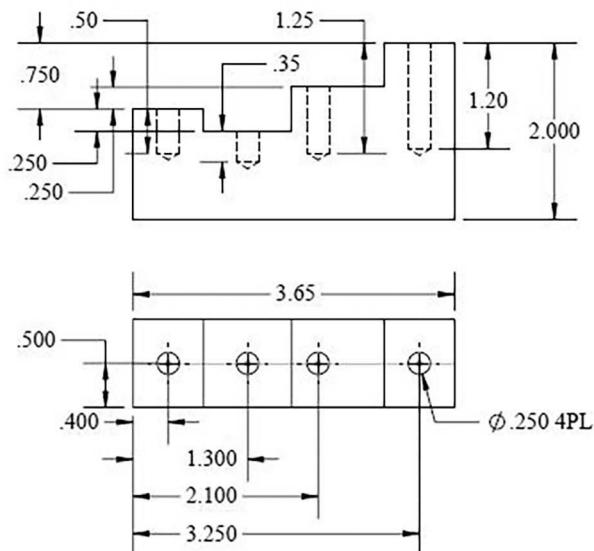
Figure 4-52
Drawing for Comparison of Absolute and
Incremental Programming Example

Machine: Machining Center		Program Number: O0416	
Workpiece Zero: X = Lower Left Corner Y = Lower Left Corner Z= Top Part Surface Setup Description: Material = Steel The blank is placed on .500" parallels			
Tool #	Description	Offset	Comments
T1	#4 HSS Center Drill		SFM= 70 FPT = .002
T2	.250 Diameter HSS Drill		SFM= 70 FPT = .0035

Part 4 Programming CNC Machining Centers

The following program uses an absolute coordinate system and function G98, Initial Level Return. The steps and overall dimensions are pre-cut (Chart 416).

> **O0416**
> **N10 G90 G80 G20 G40 G49**
> **N15 S1070 M03**
> **N20 G00 G54 X.4 Y.5**
> **N25 G43 Z1.0 H01 M08**
> **N30 G81 G98 Z–.575 F7.48 R.1**
> **N35 X1.3 Z–.675 R–.15**
> **N40 X2.1 Z–.825 R.35**
> **N45 X3.25 Z–.525 R.85**
> **N50 G80 Z1.0 M09**
> **N55 G91 G28 Z0.0 M05**
> **N60 G28 X0.0 Y0.0**
> **N65 M30**

The following program is the same as the one above (O0416), except it uses an incremental coordinate system.

> **O0416**
> **N10 G90 G80 G20 G40 G49**
> **N15S1070M03**
> **N20 G00 G54 X.4 Y.5**
> **N25 G43 Z1.0 H01 M08**
> **N30 G91 G81 G98 Z–.675 F6.4 R–.9**
> **N35 Z–.525 R–1.15**
> **N40 X.9 Z–1.175 R–.65**
> **N45 X.8 Z–1.375 R–.15**
> **N50 G80 X1.15 Z5.0 M09**
> **N55 G28 Z0.0 M05**
> **N60 G28 X0.0 Y0.0**
> **N65M30**

Notes:

- *The sign (+ or –) is determined with respect to the axis of the coordinate system.*
- *Function G90 or G91 must be entered at the beginning of the program, where it remains valid until replaced by function G91 or G90. Also, for Initial and R-Level Return that are applied within the Canned Drilling Cycles note the position values of the tool.*

COMPLEX PROGRAM EXAMPLE 1

To machine the part represented by Figure 4-53, a lathe is used first to prepare the bronze bushings dimensioned, as seen in Figure 4-54.

Consecutive operations will be performed on the vertical machining center. This part will be machined in two operations. First, three holes with a diameter of .052 will be drilled and reamed. In the second operation, the step notch will be milled on the outside
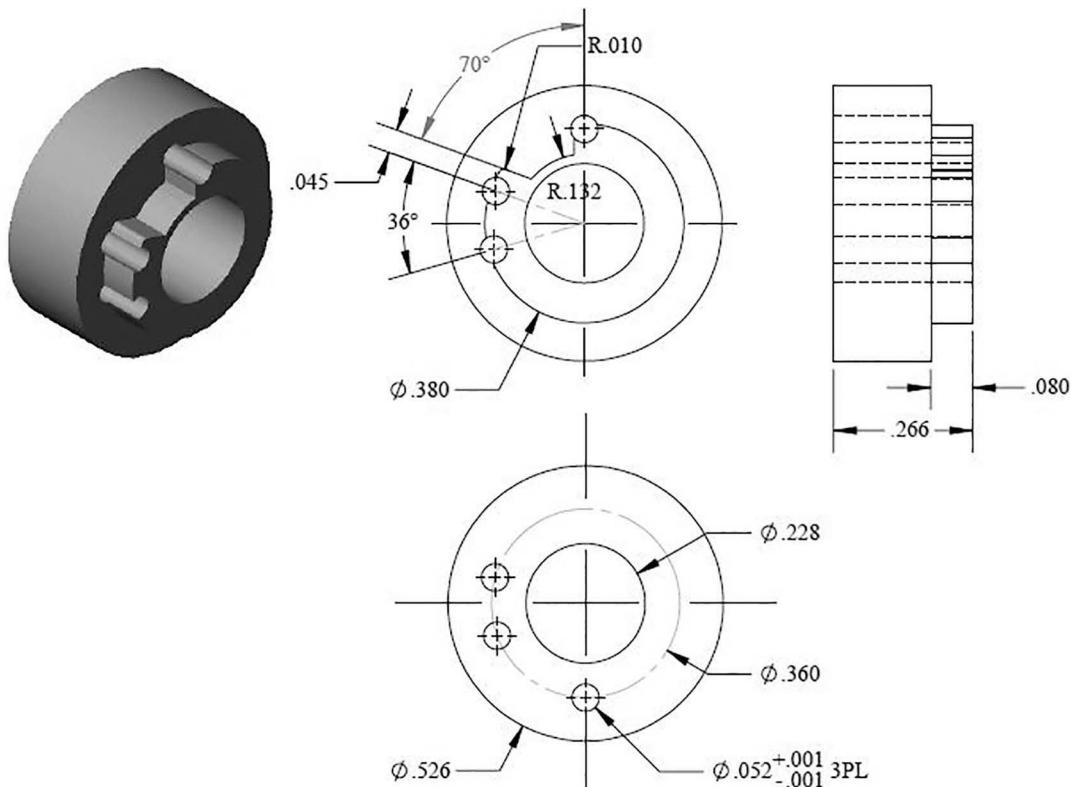
Figure 4-53
Drawintg for Complex Program Example 1
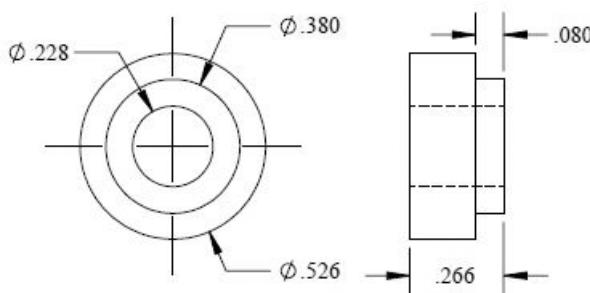
Figure 4-54
Bronze Bushing Blanks for
Complex Program Example 1

of the workpiece. Machining speeds, simplicity of fastening, and maximum accuracy are the decisive factors that determine the selection of the number of operations made and the sequence they follow.

Fixture Preparation

Prepare the work-holding fixture for the machined parts from a piece of rectangular steel, dimensioned as follows: $1.5 \times 6.5 \times .375$ thick. Two copies will be required, one for Operation 1 and another for Operation 2 [Figure 4-55].

The blank is clamped in a vise on 1.5 inch parallels. The longer side of the part is aligned with the back jaw of the vise to establish parallelism with the part seats, which will later be machined. Afterwards, the 20 seats in the steel plate in which the parts will be fastened are machined.

The seats have the following dimensions: diameter .528, depth .100. At the center of each seat, drill and then tap holes for the part clamping screws (6-32 Flat Head Cap Screws). For this example, the work-holding fixture has been prepared earlier.

Complex Program Example 1 Setup Description

The fixture is placed in the "dialed-in" machining center vise and is positioned so that the long side is parallel to the X-axis. Next, the values of X and Y for the functions G54 and G55 work offsets are determined.
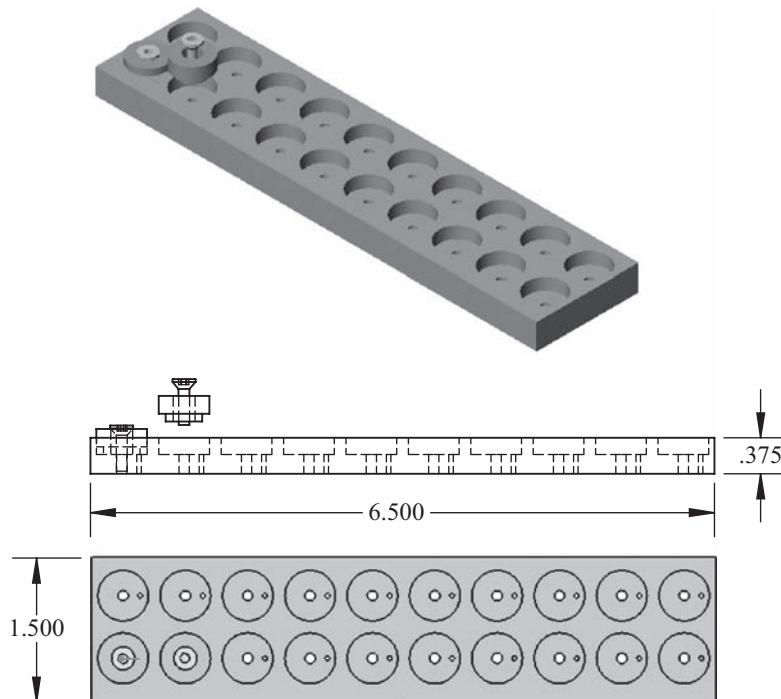
Figure 4-55
Fixture for Setup 1

Part 4 Programming CNC Machining Centers

Procedure for Measuring Work Offsets

Detailed descriptions for set-up procedures are given in Part 2 of this book. Remember that the procedures used are specific to the type of control described in this text and are for the FANUC 0i controller. Other control types may use similar procedures. Check the specific manufacturer operation manuals for their procedures.

Find the center of the first left-seat hole that is positioned in the upper row of holes. The values of X and Y are read from the absolute display and entered under the offset values of the coordinate system, of which position 01 corresponds to the function G54. The position of the left-seat hole in the lower row is determined in a similar manner; the values of the coordinates corresponding to function G55 are entered under 02, in the coordinate system register. Once Workpiece Zeros are set, place the previously prepared bronze bushings in the fixture seats and fasten them with machine screws.

Procedure for Measuring Tool Length Offsets

Using the skills you learned in Part 2 of this text, set the tool lengths for each of the tools in the setup sheet in Chart 417. The programs consist of two operations with tools given in Setup sheets 417 and 418.

The following example provides the main program for the first operation.

Machine: Machining Center		**Program Number: O0417**	
Workpiece Zero: G54 = X = <u>Upper Left Seat CL</u> Y = <u>Upper Left Seat CL</u> Z= <u>Top Part Surface</u> G55 = X = <u>Lower Left Seat CL</u> Y = <u>Lower Left Seat CL</u> Z= <u>Top Part Surface</u> Setup Description: Material = Bronze The fixture is placed on 1.500" parallels			
Tool #	**Description**	**Offset**	**Comments**
T1	#0 HSS Center Drill	H1	SFM= 70 FPT = .002
T2	.250 Diameter HSS Drill	H2	SFM= 70 FPT = .0035
T3	.052 Diameter Reamer	H3	SFM = 30 FPT = .001

Example: Complex Program Example 1, Operation 1, Program 417

O0417

N10 G90 G80 G20 G40 G49

N15 T01 M06

N20 G00 G54 X.1787 Y0.0 S4000 M03 T02

N25 G43 Z1.0 H01 M08

N30 G81 G99 Z–.06 R.05 F4.0

N35 M98 P2

N40 G90 G00 G55 X.1787 Y0.0

N45 G81 G99 Z–.055 R.05 F4.0

N50 M98 P2

N55 G90 G00 G80 Z5.0 M09

N60 G91 G28 Z0.0

N65 G28 X0.0 Y0.0

N70 M01

N75 M06

N80 G90 G80 G40 G49

N85 G00 G54 X.1787 Y0.0 S4000 M03 T03

N90 G43 Z1.0 H02 M08

N95 G83 G99 Z–.35 R.05 Q.05 F4.0

N100 M98 P2

N105 G90 G00 G55 X.1787 Y0.0

N110 G83 G99 Z–.35 R.05 Q.05 F4.0

N115 M98 P2

N120 G80 G00 Z1.0 M09

N125 G91 G28 Z0.0

N130 G28 X0.0 Y0.0

N135 M01

N140 M06

N145 G90 G80 G40 G49

N150 G54 G00 X.1787 Y0.0 S4000 M03

N155 G43 Z1.0 H03 M08

N160 G85 G99 Z–.31 R.05 F6.0

N165 M98 P2

N170 G80 G90 G00 G55 X.1787 Y0.0

N175 G85 G99 Z.31 R.05 F6.0

N180 M98 P2

N185 G80 G00 Z1.0 M09

N190 G91 G28 Z0.0

N195 G28 X0.0 Y0.0

N200 M05

N205 M30

The following is the subprogram O0002 for the first operation.

O0002
N2 X.0407 Y.1753
N3 X.0701 Y–.1658
N4 X.8287 Y0.0
N5 X.6907 Y.1753
N6 X.7201 Y–.1658
N7 X1.4787 Y0.0
N8 X1.3407 Y.1753
N9 X1.3701 Y–.1658
N10 X2.1287 Y0.0
N11 X1.9907 Y.1753
N12 X2.0201 Y–.1658
N13 X2.7787 Y0.0
N14 X2.6407 Y.1753
N15 X2.67011 Y–.1658
N16 X3.4287 Y0.0
N17 X3.2907 Y.1753
N18 X3.3201 Y–.1658
N19 X4.0787 Y0.0
N20 X3.9407 Y.1753
N21 X3.9701 Y–.1658
N22 X4.7287 Y0.0
N23 X4.6201 Y.1753
N24 X4.5907 Y–.1658
N25 X5.3787 Y0.0
N26 X5.2407 Y.1753
N27 X5.2611 Y–.1658
N28 X6.0287 Y0.0
N29 X5.8907 Y.1753
N30 X5.9201 Y–.1658
N31 M99

The following explanations are for some blocks of program O0417, operation one.

N10 G90 G80 G20 G40 G49

Block N10 is the Safety Block. It contains the functions G90, G80, G20, G40, and G49 where: G90 refers to programming in an absolute coordinate system (all dimensions

correspond to a fixed origin that is the Workpiece Zero); G80 is entered in the beginning of the program to assure that all canned cycle functions are cancelled; G20 initiates the inch measurement system; G40 assures that all cutter diameter compensation functions are cancelled; and, G49 cancels all tool length compensations.

N15 T01 M06

In this block, Tool T01 is called from the magazine and is inserted into the spindle through miscellaneous function M06.

N20 G00 G54 X.1787 Y0.0 S4000 M03 T02

In this block, the control reads the coordinates of the Workpiece Zero (defined in G54) from the coordinate system offset registers. It then moves at rapid traverse (G00), with respect to that zero, the distance whose value is given by X and Y (in this case, X.1787 and Y0.0). The block also defines and activates the spindle rotation of 4000 (S4000) revolutions per minute (r/min) in the clockwise direction (M03) and calls tool number 2 (T02) into the ready position in the magazine.

N25 G43 Z1.0 H01 M08

In this block, the control reads the value of the positive tool length offset (G43) for the center drill as is registered in offset number 1 and called in the program by H01. The control then positions the tool at rapid traverse along the Z-axis corresponding to the values of one inch above the part. Miscellaneous function M08 activates the flood coolant flow.

N30 G81 G99 Z–.060 R.05 F4.0

Functions G81 and G99 refer to the canned drilling cycle to a depth of −.060. At the completion of the cycle, the drill rapidly traverses to the R-level reference plane (identified by the letter R) of .050 inch above the material and for positioning to the remaining holes. The drilling begins from this point and feeds down to a depth of Z−.060 with a feed rate of F4.0 inches per minute (in/min). After completing all of the spot drilling, the center drill withdrawal occurs with a rapid traverse to the Z value, which is given in block N25 and is Z1.

N40 M98 P2

Function M98 calls up subprogram O0002 (P2 identifies the subprogram program number) from the controller memory. From this point on, execution of subprogram O0002 will be in progress until its completion.

The following explanations are for a few blocks of subprogram O0002 called from the main program O0417, operation one.

N2 X.0407 Y.1753

In block N2, the center drill performs the spot drilling of the second hole with the assigned coordinates X and Y.

N3 X.0701 Y–.1658

In this block, the third hole is drilled in the first bushing.

N4 X.8287 Y0.0

The center drill is displaced along a horizontal straight line (along the X-axis) by the value X.8287 over to the second bushing, and the first hole is spot drilled. Similar moves are performed in consecutive blocks and consecutive holes are spot drilled. In block N30 of the subprogram, spot drilling of the holes in the 10th bushing is completed.

N31 M99

Block N31 ends the subprogram, and function M99 commands the return to the main program O0417 at block N40.

The explanations that follow return to the main program O0417, at block N40:

In block N40 of the main program, the new program zero G55 is assigned in the center of the first bushing in the lower row. In block N45, the drilling command is repeated; then in block N50, the function M98 P2 commands the repeated execution of the subprogram O0002. Executing the subprogram causes the machine to spot-drill holes in the 9 remaining bushings (the lower row). When the work is completed, there is a return to block N55 of the main program, a deactivation of coolant flow M09 and, simultaneously, a withdrawal of the spotting drill to Z5.0 along the Z-axis at rapid traverse. In block N60, the tool is moved to the Reference Position Return (G28) with respect to the Z-axis. Then in block N65, the X and Y axes are moved to the same position as well. Function M01 is entered in block N70 and work is interrupted in this block, but only if the OPTIONAL STOP button on the controller is ON. In block N75, a tool change is commanded by M06 and tool two is placed in the spindle.

N80 G90 G80 G40 G49

This block contains the Safety Block information required after each tool change.

N85 G00 G54 X.1787 Y0.0 S4000 M03 T03

Function G54 refers to the return from the previous coordinate system, i.e., the functions G54 to G55 (Workpiece Zero (G54) is located in the center of the first bushing in the upper row). The tool then rapidly traverses (G00) to the position above the first hole with coordinates X.1878 and Y0.0.

N90 G43 Z1.0 H02 M08

The negative value of the tool length offset from the offset register for T02 is read by the control. The tool rapidly traverses along the Z-axis to a position one inch above the material. The coolant flow is activated by the function M08. Block N90 is similar to block N25. They differ only by the value of the tool length offset H02, because tool T02 is used instead.

Comparing blocks N15 through N70 with blocks N75 through N135 and N140 through N200, notice the similarities:

> N15 through N70—work of the first tool
>
> N75 through N135—work of the second tool
>
> N140 through N200—work of the third tool

In block N205, function M30 stops the execution of the program and commands the program to return to its beginning. After completion of block N205, the completed bushings are replaced in the fixture by new blanks and then by pressing the CYCLE START button, machining of the next 20 pieces is initiated.

Example: Complex Program Example 1, Operation 2, Program 418

The following is the main program for the second operation.
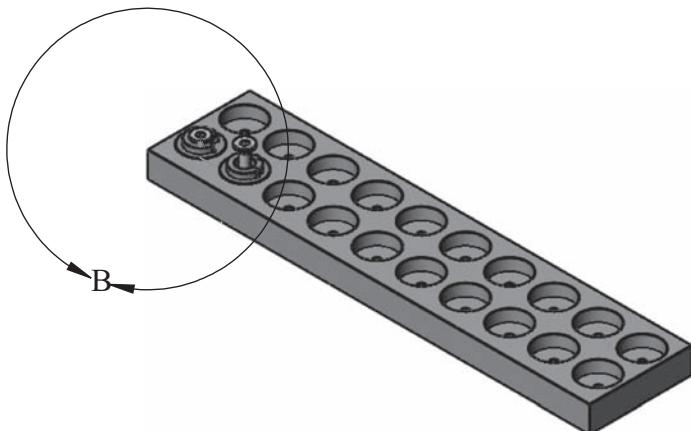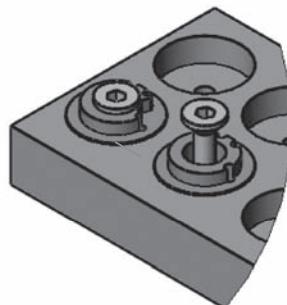
Figure 4-56
Fixture for Setup 2

DETAIL B
SCALE 1 : 1

O0418

N10 G90 G80 G20 G40 G49

N15 T04 M06

N20 G00 G54 X0.0 Y0.0 S3000 M03 T05

N25 G43 Z1.0 H04 M08

N30 M98 P6 L10

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0418	
Workpiece Zero: G54 = X = <u>Upper Left Seat CL</u> Y = <u>Upper Left Seat CL</u> Z= <u>Top Part Surface</u> G55 = X = <u>Lower Left Seat CL</u> Y = <u>Lower Left Seat CL</u> Z= <u>Top Part Surface</u> Setup Description: Material = Bronze The fixture with the pins installed for the second operation is placed on 1.500" parallels			
Tool #	**Description**	**Offset**	**Comments**
T4	.125 Diameter HSS Roughing End Mill	H4	SFM= 70 FPT = .002
T5	.0625 Diameter HSS 4-Flute Finishing End Mill	H5	SFM= 70 FPT = .002

N35 G90 G00 G55 X0.0 Y0.0

N40 M98 P6 L10

N45 G00 G90 Z1.0 M09

N50 G91 G28 X0.0 Y0.0 Z0.0

N55 M01

N60 M06

N65 G90 G80 G40 G49

N70 G00 G54 X0.0 Y0.0 S2300 M03

N75 G43 Z1.0 H05 M08

N80 M98 P7 L10

N85G00 G90 G55 X0.0 Y0.0

N90 M98 P7 L10

N95 G00 G90 Z1.0 M09

N100 G91 G28 X0.0 Y0.0 Z0.0

N105 M01

N110 M30

Subprogram O0006 is for the Roughing end mill tool path.

O0006

N1 G90 G00 X0.0 Y.25

N2 Z0.0

N3 G41 X.095 Y.1875 D04

N4 G01 Z–.08 F4.0

N5 X.098 Y.163

N6 G02 X.102 Y.151 R.01

N7 G01 X.085 Y.101

N8 G02 X.130 Y.02 R.132

N9 G01 X.180 Y.020

N10 G40 X.250

N11 G00 Z1.0

N12 G91 X.65

N13 M99

Subprogram O0007 is for the Finish end mill tool path.

O0007

N1 G90 G00 Z0.0

N2 G41 X.095 Y.1875 D05

N3 G01 Z–.08 F4.0

N4 X.098 Y.163

N4 G02 X.102 Y.151 R.01

N5 G01 X.085 Y.101

N6 G02 X.130 Y.02 R.132

N9 G01 X.180 Y.020N10 G40 X.250

N11 G00 Z1.0

N12 G91 X.65N13 M99

The Second Operation

In the second operation, the drilled bushings are milled using two end mills—first, a rough pass with a tool diameter of .125 and, second, a finish pass with a tool diameter of .0625 inch. The bushings are placed in a fixture similar to the one in the previous operation. The only difference in the second fixture is that each seat has a pin forced into the bottom with a diameter of .0515 onto which the previously reamed hole in the bushing is placed. Such positioning of the bushings assures the proper position of the hole and prevents rotation of the bushing in the seat during milling.

Main Program Number O0418

The following explanations are for some blocks of program O0019 operation two. Sub-program O0006 is for the first tool and subprogram O0007 is for the second tool.

N10 G90 G80 G20 G40 G49

This is a Safety Block where G90 sets the program in the absolute coordinate system. G80 cancels all canned cycle functions. G20 initiates the inch measurement system. G40 cancels all radius compensation functions. G49 cancels all tool length compensation functions.

N15 T04 M06

This block commands the preparation of the tool T04 in the magazine and changes it into the spindle.

N20 G54 G00 X0.0 Y0.0 S3000 M03 T05

Workpiece Zero coordinates are referenced by function G54; there is a rapid traverse of the tool to the position of zero. The spindle is started in the clockwise direction by function M03, at a speed (S) of 3000 r/min, and tool number 5 (T05) is moved to the ready position in the magazine.

N25 G43 Z1.0 H04 M08

In this block, the machine is commanded to read the tool length offset for T04 from the offset register while the tool positions at rapid traverse to 1 inch above the material along the Z-axis. The flood coolant in activated by M08.

N30 M98 P6 L10

In block N30, the subprogram O0006 is called up and executed ten times (L10) for the 10 bushings. At this point, the tool is controlled by the subprogram. In order to simplify the program notation, the coordinate position characteristics are given on the drawing in Figure 4-57. Note that the part is mounted in the mill –90.0 degrees from the view given.

Subprogram O0006

The following explanations are for a few blocks of subprogram O0006 called from the main program O0418, operation two.
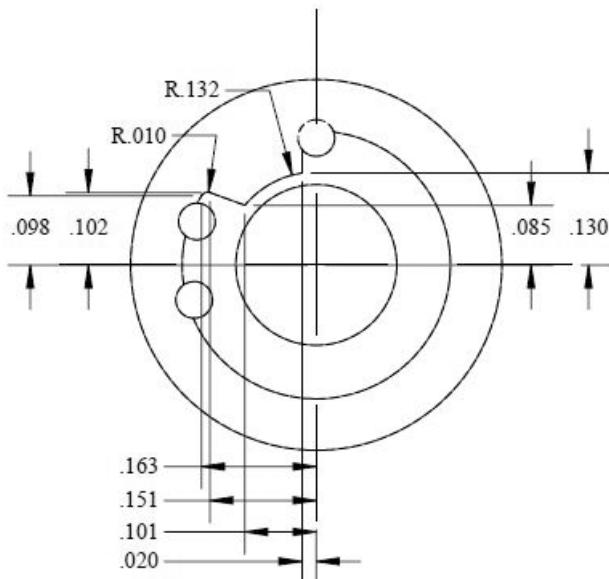
Figure 4-57
Coordinate Locations
for the Cutout

N1 G90 G00 X0.0 Y.25

The block refers to programming in the absolute coordinate system and traverses along the X and Y axes at rapid to the positions given.

N2 Z0.0

This block moves the tool to the Z-axis 0.0 position.

N3 G41 X.095 Y.1875 D04

The tool traverses with a previously assigned feed rate to a position of Y.1225 while Cutter Compensation (G41) is activated, with a simultaneous reading of the radius offset (D04) from the offset register. In this case, the value of the offset is equivalent to .005 inch to allow material to remain for the finishing operation in which the finishing end mill is used.

N4 G01 Z–.08 F4.0

This block feeds the tool to the full depth of –.08 and 4.0 inches per minute.

N5 X.098 Y.163

In this block, the tool is positioned at linear feed to the starting point of the .010 R.

N6 G02 X.102 Y.151 R.01

The tool traverses along a circular path, in a clockwise direction with an endpoint specified by the values of X and Y. The radius value of R.01 is used.

N7 G01 X.085 Y.101

The tool traverses along a straight line, specified by the value of X and Y at the feed rate listed earlier.

N8 G02 X.130 Y.02 R.132

Traverse along the circular path to the endpoint coordinates given with a radius of .132.

N9 G01 X.180 Y.020

The end mill is fed along a linear path to a clear point in both the X and Y axes.

N10 G40 X.250

The active function G41 is cancelled by G40, and the tool is moved along a straight line to X.250.

N11 G00 Z1.0

The tool is positioned at rapid traverse, 1.0 inch above the material (use G90 to assure that the tool position is above the material).

N12 G91 X.65

A rapid traverse is made along the X-axis by a value of 1 in (toward the center of the second bushing).

N13 M99

This block commands a return to the main program O0418. However, because function M98 L10 was entered previously in the main program, the subprogram will be executed nine more times (ten bushings will be machined in exactly the same manner). After the machining of the last bushing in the upper row is completed, control is returned to block N35 of the main program.

N35 G90 G00 G55 X0.0 Y0.0

In this block, the new Workpiece Zero, G55 is introduced with its assigned coordinates given in the offset register. G00 refers to a rapid traverse to the new position of the new Workpiece Zero. (Position of the new Workpiece Zero is set to the center of the first bushing of the lower row.)

N40 M98 P6 L10

Function M98 calls up subprogram O0006 and executes it 10 times (subprogram O0006 was described earlier). After the 10 executions of the subprogram, control is returned to block N45 of the main program for the second operation.

N45 G00 G90 Z1.0 M09

A rapid traverse movement is made to a position 1 inch above the surface of the material based on the absolute coordinate system, and the deactivation of the coolant flow (M09) occurs.

N50 G91 G28 X0.0 Y0.0 Z0.0

Function G28 returns the machine to the Machine Zero position in the X, Y, and Z axes.

N55 M01

Miscellaneous command M01 provides an additional interruption of the program (only if the OPTIONAL STOP button is ON). The purpose of this interruption is to check the dimensions and condition of the tools.

N60 M06

This is the tool change for T05 that was positioned in the tool magazine in block N20 and then in block N60 transfers the tool (T05) into the spindle of the machine.

N65 G90 G80 G40 G49

This is a safety block.

N70 G00 G54 X0.0 Y0.0 S2300 M03

A new Workpiece Zero (G54) is introduced; the tool rapidly travels to the position of the new zero and clockwise spindle rotation is initiated at 2300 r/min.

N75 G43 Z1.0 H05 M08

The tool length offset H05 obtained from the offset register is entered, allowing the traverse of the tool to a position 1 inch above the surface of the material. Flood coolant flow is also activated.

N80 M98 P7 L10

The subprogram O0007 (P7) is called up and then executed 10 (L10) times.

The coordinates of the individual characteristic points are the same as given before with the difference being tool diameter and offset (D05) of .0625. Subprogram O007 executes in exactly the same as O0006 with the new tool.

N95 G00 G90 Z1.0 M09

When all of the sequences are complete, the tool is positioned 1.0 inch above the part at rapid traverse.

N100 G91 G28 X0.0 Y0.0 Z0.0

There is a return to Machine Zero for the X, Y, and Z axes.

N105 M01

There is an additional interruption of the machine work, provided that the OPTIONAL STOP switch/button is ON.

N110 M30

This is the end of the program and the control returns to the beginning. Completion of N110 results in a machine stop. The finished bushings must be removed and be replaced with unfinished ones. Press the CYCLE START button and the work cycle is initiated once again.

EXAMPLE ILLUSTRATING THE APPLICATION OF MIRROR IMAGE

The next example illustrates the use of the mirror image function (Figure 4-58). This function makes it possible to duplicate patterns by projecting a mirror image of the pattern in the X- or Y-axis directions. The main purpose for using the mirror image function is to shorten and simplify the part program. Because the program illustrated here uses only one tool, a setup sheet is not used. The program starts with tool one already in the spindle.

> M21 = the mirror image in the direction of the X-axis
>
> M22 = the mirror image in the direction of the Y-axis
>
> M23 = the cancellation of the mirror image
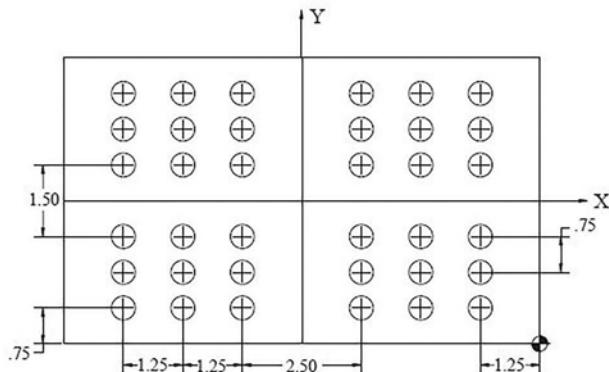
Figure 4-58
Drawing for Mirror Image Example

Example: Mirror Image, Program 419

O0419

N10 G90 G80 G20 G40 G49 M23

N15 G00 G54 X–1.25 Y.75 S1000 M03

N20 G43 Z1.0 H01 M08

N25 G81 G98 Z–.35 R.1 F6.0

N30 M98 P8

N35 G00 G80 X0.0

N40 M21

N45 G00 X–1.25 Y.75

N50 G81 G98 Z–.35F6.0 R.1

N55 M98 P8

N60 G00 G80 X0.0 Y0.0

N65 M23

N70 M22

N75 G00 X–1.25 Y.75

N80 G81 G98 Z–.35 F6.0 R.1

N85 M98 P8

N90 G80 G00 X0.0 Y0.0

N95 M21

N100 G00 X–1.25 Y.75

N105 G81 G98 Z–.35 F6.0 R.1

N110 M98 P8

N115 G80 Z1.0 M09

N120 M23

N125 G91 G28 X0.0 Y0.0 Z0.0

N130 M30

Subprogram for Mirror Image, Program 419

    O0008

    N1 X–2.5

    N2 X–3.75

    N3 Y1.5

    N4 X–2.5

    N5 X–1.25

    N6 Y2.25

    N7 X–2.5

    N8 X–3.75

    N9 M99

EXAMPLE ILLUSTRATING THE APPLICATION OF AXIS ROTATION

Occasionally, vertical milling machines are used to perform operations on cylindrical or other workpieces that require the use of a rotational axis. In the following case, there are four drilled and tapped holes, and a slot that must be machined by using this method (Figure 4-59). Rotation about the X-axis is required. This is an additional axis and is identified as the A axis. Chart 420 lists all of the tools used in this program.
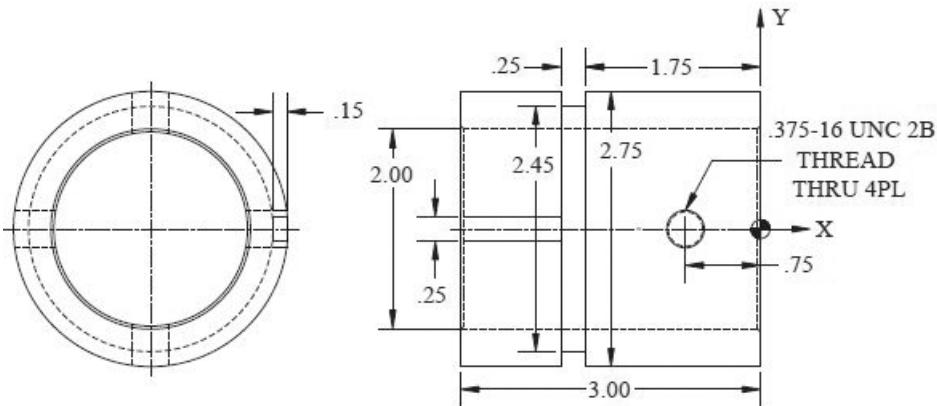
Figure 4-59
Drawing for Application of Axis Rotation Example

Program 420

    O0420

    N10 G90 G80 G20 G40 G49

    N15 T1 M06

    N20 G00 G54 X–.75 Y0.0 A0.0 S1520 M03

N25 G43 Z1.0 H01 M08

N30 G98 G81 Z–.375 R.1 F6.0

N35 M98 P9

N40 G80 Z1.0 M09

N45 G91 G28 X0.0 Y0.0 Z0.0

N50 M01

N55 T2 M06

N60 G90 G80 G40 G49

N65 G00 G54 X–.75 Y0.0 A0.0 S1152 M03

N70 G43 Z1.0 H02 M08

N75 G98 G81 Z–.450 R.1 F8.0

N80 M98 P9

N85 G80 Z1.0 M09

N90 G91 G28 X0.0 Y0.0 Z0.0

N95 M01

N100 T3 M06

N105 G90 G80 G40 G49

N110 G00 G54 X–.75 S120 M03

N115 G43 Z1.0 H03 M08

N120 G98 G84 Z–.55 R.2 F7.5

N125 M98 P9

N130 G80 Z5.0 M09

N135 G91 G28 X0.0 Y0.0 Z0.0

N140 M01

N145 T4 M06

N150 G90 G80 G40 G49

N155 G00 G54 X–3.3 Y0.0 A0.0

N160 G43 Z1.0 H4 M08

N165 S1520 M03

N170 G01 Z–.15 F50.0

N175 X–1.875 F6.0

N180 A360.0

N185 G00 Z1.0 M09

N190 G91 G28 X0.0 Y0.0 Z0.0 A0.0

N200 M30

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0420	
Workpiece Zero: X = <u>Right End</u> Y = <u>Center Line</u> Z= <u>Top of Part Surface</u> Setup Description: Material = Steel Mount the part in a 3-jaw chuck attached to an indexing head or 4th axis			
Tool #	**Description**	**Offset**	**Comments**
T1	#3 HSS Center Drill	H1	SFM= 95 FPT = .002
T2	.3125 Diameter HSS Drill	H2	SFM= 95 FPT = .0035
T3	Tap 3/8-16	H3	SFM= 15
T4	.250 HSS 4-Flute End Mill	H4	SFM= 95 FPT = .002

Subprogram for Program 420

> **O0009**
> **N1 A90.0**
> **N2 A180.0**
> **N3 A270.0**
> **N4 M99**

Note: If, in a given block, there is a rotation during the work traverse, then the feed is defined in degrees per revolution.

EXAMPLE OF PROGRAMMING A HORIZONTAL MACHINING CENTER

Thus far, in this part of the text, only vertical machining center programming has been described. In this example, some of the programming techniques covered earlier will be applied to a horizontal machining center. With the horizontal machine, the Z-axis is horizontal. Another feature that will be useful on the illustrated part is its indexing capability.

Example: Programming a Horizontal Machining Center, Program 421

The material is 8-inch thick hexagonal steel plate with the following dimensions: 8 x 14 in. There are also two positioning holes in the plate having a .500 inch diameter.

The workpiece is placed on the fixture that has two positioning pins that are .500 inch in diameter. The workpiece is held down by a holding mechanism and a cap screw that is placed in the center of a previously made hole of 1.5 inch. To simplify programming, assume that the rotation axis of the table is aligned with the symmetry axis of the object.

Operations are performed on a four-axis horizontal milling machine. Chart 421 lists all of the tools used in this program.

Part 4 Programming CNC Machining Centers
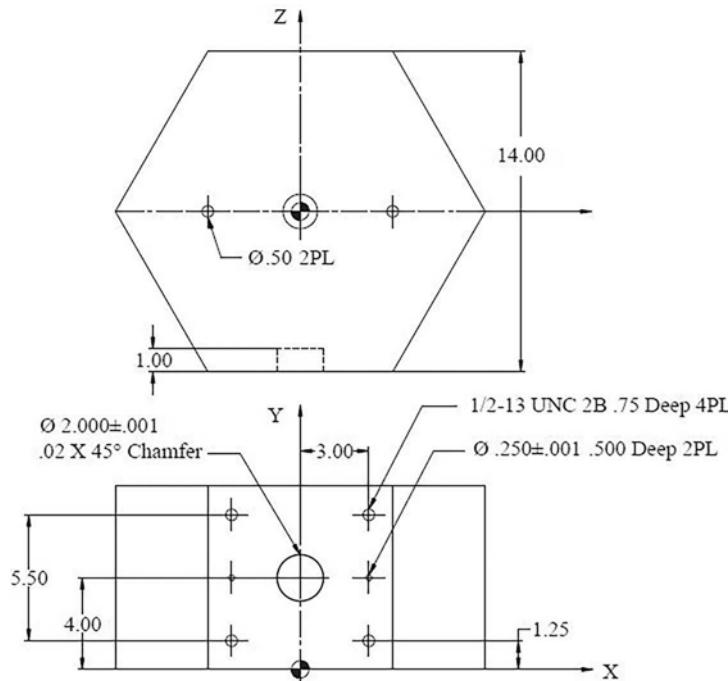
Figure 4-60
Drawing for Horizontal
Machining Center
Example

Machine: Machining Center		Program Number: O0421		
Workpiece Zero: G54 X = <u>Center Line</u> Y = <u>Bottom of Part Surface</u> Z= <u>Center Line</u> Setup Description: Material = Steel				
Tool #	Description	Offset	Comments	
T1	#3 HSS Center Drill	H1		
T2	.125 Diameter HSS Drill	H2		
T3	1.0 Diameter 4-flute HSS End Mill	H3		
T4	27/64 (.4219) Diameter HSS Drill	H4		
T5	1/2-13 UNC Tap	H5		
T6	15/64 (.2344) Diameter HSS Drill	H6		
T7	.240 Diameter 4-flute HSS End Mill	H7		
T8	.250 HSS Reamer	H8		
T9	Boring Bar for 2.00 Diameter	H9		
T10	45 Degree Cutter	H10		

Program 421

O0421

N10 G90 G80 G20 G40 G17

(T1 #3 HSS CENTER DRILL)

N15 M11 T1

N20 M06

N25 G00 G54 X0.0 Y4.0 B0.0 S600 M03

N30 M10

N35 G43 Z8.5 M08 H1

N40 M98 P0050 L6

N45 G80 Z10.0 M09

N50 M11

N55 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N60 M01

(T2 .125 DIAMETER HSS DRILL)

N65 T2

N70 M06

N75 G90 G80 G20 G40 G17

N80 G00 G54 X0.0 Y4.0 B0.0 S244 M03

N85 M10

N90 G43 Z8.5 M08 H2

N95 M98 P0052 L6

N100 G80 Z10.0 M09

N105 M11

N110 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N115 M01

(T3 1.0 DIAMETER 4-FLUTE HSS END-MILL)

N120 T3

N125 M06

N130 G90 G80 G20 G40 G17

N135 G00 G54 X0.0 Y4.0 B0.0 S305 M03

N140 M10

N145 G43 Z8.5 M08 H3

N150 M98 P0053 L6

N155 G80 Z10.0 M09

N160 M11

N165 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N170 M01

(T4 27/64 DIAMETER HSS DRILL)

N175 T4

N180 M06

N185 G90 G80 G20 G40 G17

N190 G00 G54 X3.0 Y6.75 B0.0 S726 M03

N195 M10

N200 G43 Z8.5 M08 H4

N205 M98 P0054 L6

N210 G80 Z10.0 M09

N215 M11

N220 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N225 M01

(T5½-13 UNC TAP)

N230 T5

N235 M06

N240 G90 G80 G20 G40 G17

N245 G00 G54 X–.30 Y6.75 B0.0 S100 M03

N250 M10

N255 G43 Z8.5 M08 H5

N260 M98 P0055 L6

N265 G80 Z10.0 M09

N270 M11

N275 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N280 M01

(T6 15/64 DIAMETER HSS DRILL)

N285 T6

N290 M06

N295 G90 G80 G20 G40 G17

N300 G00 G54 X3.0 Y4.0 B0.0 S1300 M03

N305 M10

N310 G43 Z8.5 M08 H6

N315 M98 P0056 L6

N320 G80 Z10.0 M09

N325 M11

N330 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N335 M01

(T7 .240 HSS 4-FLUTE END-MILL)

N340 T7

N345 M06

N350 G90 G80 G20 G40 G17

N355 G00 G54 X–3.0 Y4.0 B0.0 S1200 M03

N360 M10

N365 G43 Z8.5 M08 H7

N370 M98 P0057 L6

N375 B180.0 Z10.0M09

N380 M11

N385 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N390 M01

(T8 .250 HSS REAMER)

N395 T8

N400 M06

N405 G90 G80 G20 G40 G17

N410 G00 G54 X–3.0 Y4.0 B0.0 S800 M03

N415 M10

N420 G43 Z8.5 M08 H8

N425 M98 P0058 L6

N430 G80 Z10.0 M09

N435 M11

N440 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N445 M01

(T9 BORING BAR FOR 2.00 DIAMETER)

N450 T9

N455 M06

N460 G90 G80 G20 G40 G17

N465 G00 G54 X0.0 Y4.0 B0.0 S500 M03

N470 G43 Z8.5 M08 H9

N475 G76 G98 Q.01 Z6.0 F2.0 R7.2

N480 B60

N485 B120

N490 B180

N495 B240

N500 B300

N505 G80 Z10.0 M09

N510 G91 G28 X0.0 Y0.0 Z0.0 B0.0

N515 M01

(T10 45° CHAMFER CUTTER)

N520 T10

N525 M06

N530 G90 G80 G20 G40 G17

N535 G00 G54 X0.0 Y4.0 S500 M03

N540 G43 Z8.5 M08 H10

N545 G82 G98 Z6.9 F2.5 R7.1

N550 B60

N555 B120

N560 B180

N565 B240

N570 B300

N575 G80 Z10.0 M09

N580 G91 G28 X0.0 Y0.0 Z0.0 B0.0 M05

N585 M10

N590 M06

N595 M30

Notes: There are two types of horizontal milling machines. On milling machines equipped with an indexing table, rotation of the table only by 1° or more is allowed. When programming the changes of angular position of the table, as a rule, a decimal point is not used (but will do no harm).

Example

B120, B180, etc.

Special functions are used to determine the direction of rotation of the table. On a milling machine equipped with a four-axis table, rapid rotation or regular feed rotation can be applied. The table can be rotated by .001°, or even more accurately. On these machines, performance of all kinds of curved contours may be accomplished while machining the object. Plus (+) or minus (–) signs determine the direction of rotation. The table should not be locked if a rotation is to be performed. At the time tools 1 to 8 perform their operations, the table is locked by function M10. When the remaining two tools perform their operations, however, the table is unlocked by function M11. Whether the table

should be locked or not is determined by the conditions of machining (light cutting/table unlocked or heavy cutting/table locked).

Subprograms for example O0421

O0050

N1 G81 G98 Z6.5 F2.5 R7.1

N2 X–3.0 Y6.75

N3 M98 P0051

N4 X–3.0 Y4.0 Z6.73

N5 X3.0

N6 G80 X0.0 Y4.0 M11

N7 G91 G00 A60.0

N8 G90 M10

N9 M99

O0051

N1 X3.0

N2 Y1.25

N3 X–3.0

N4 M99

O0052

N1 G81 G98 Z6.0 F1.5 R7.1

N2 G80 M11

N3 G91 G00 A60.0

N4 G90 M10

N5 M99

O0053

N1 G90 G00 Z7.1

N2 G01 Z6.0 F3.0

N3 G42 Y5.0 D3 F2.5

N4 G02 J1.0

N5 G01 G40 Y4.0

N6 G00 Z8.5 M11

N7 G91 A60.0

N8 G90 M10

N9 M99

O0054
N1 G83 G98 Q.15 Z5.8 F4.5 R7.1
N2 M98 P0051
N3 G80 X–3.0 Y6.75 M11
N4 G91 G00 A60.0
N5 G90 M10
N6 M99

O0055
N1 G84 G98 Z6.0 F7.2 R7.3
N2 M98 P0051
N3 G80 X–3.0 Y6.75 M11
N4 G91 G00 A60.0
N5 G90 M10
N6 M99

O0056
N1 G83 G98 Q.100 Z6.25 F6.5 R7.1
N2 X3.0
N3 G80 X–3.0 Y4.0 M11
N4 G91 G00 A60.0
N5 G90 M10
N6 M99

O0057
N1 G81 G98 Z6.4 F8.0 R7.1
N2 X3.0
N3 G80 X–3.0 Y4.0 M11
N4 G91 G00 A60.0
N5 G90 M10
N6 M99

O0058
N1 G81 G98 Z6.47 F4.8 R7.1
N2 X3.0
N3 G80 X-3.0 Y4.0 M11
N4 G91 G00 A60.0
N5 G90 M10
N6 M99

Part 4 Programming CNC Machining Centers
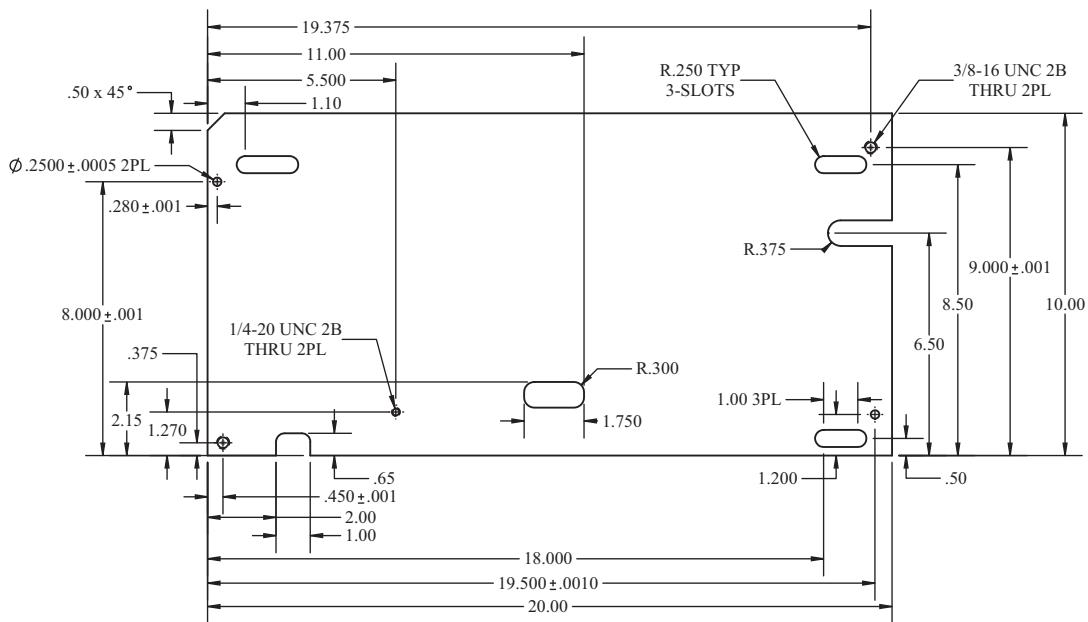
Figure 4-61
Drawing for Complex Program Example 2

Machine: Machining Center		Program Number: O0422	
Workpiece Zero: G54 X = <u>Lower Left Corner</u> Y = <u>Lower Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = 1/4 inch thick Steel			
Tool #	**Description**	**Offset**	**Comments**
T1	#3 HSS Center Drill	H1	
T2	.3125 Diameter HSS Drill	H2	
T3	3/8-16 UNC Tap	H3	
T4	.201 Diameter HSS Drill	H4	
T5	1/4-20 UNC Tap	H5	
T6	.242 Diameter HSS Drill	H6	
T7	.250 Diameter HSS Reamer	H7	
T8	7/16 Diameter HSS Drill	H8	
T9	.437 Diameter Roughing HSS End Mill	H9/D9	D9 Offset = .240
T10	.437 Diameter Finishing HSS End Mill	H10/D10	D10 Offset = .2187

COMPLEX PROGRAM EXAMPLE 2

Example: Complex Program 2, Program 422

The drawing in Figure 61 is machined using the tools listed in Chart 422.

O0422

(T1 #3 CENTER DRILL)

N10 G90 G80 G20 G40 G49

N15 T1 M06

N20 G00 G54 X.45 Y.375 S611 M03

N25 G43 Z1.0 H1 M08

N30 G81 G98 Z–.375 R.1 F2.4

N35 X19.375 Y9.0

N40 X19.500 Y1.2 Z–.26

N45 X.28 Y8.0

N50 X5.5 Y1.275 Z–.25

N55 X1.1 Y8.5 Z–.437

N60 X18.0

N65 Y.5

N70 X10.5 Y1.775

N75 G80 Z1.0 M09

N80 G91 G28 Z0.0

N85 M01

(T2 5/16 DIAMETER TAP DRILL)

N90 T2 M06

N95 G90 G80 G40 G49

N100 G54 G00 X.450 Y.375 S733 M03

N105 G43 Z1.0 H02 M08

N110 G81 G98 Z–.35 R.1 F4.3

N115 X19.375 Y9.0

N120 G80 Z1.0 M09

N125 G91 G28 Z0.0

N130 M01

(T3 3/8-16 UNC 2B TAP)

N135 T3 M06

N140 G90 G80 G40 G49

N145 G54 G00 X.450 Y.375 S150 M03

N150 G43 Z1.0 H3 M08

N155 G84 G98 Z–.45 R.2 F9.0

N160 X19.375 Y9.0

N165 G80 Z1.0 M09

N170 G91 G28 Z0.0

N175 M01

(T4 .201 DIAMETER TAP DRILL)

N180 T4 M06

N185 G90 G80 G40 G49

N190 G00G54 X5.5 Y1.275 S1140 M03

N195 G43 Z1.0 H4 M08

N200 G81 G98 Z–.35 R.1 F6.8

N205 G80 Z1.0 M09

N210 G91 G28 Z0.0

N215 M01

(T5 1/4-20 UNC 2B TAP)

N220 T5 M06

N225 G90 G80 G40 G49

N230 G00 G54 X5.5 Y1.275 S220 M03

N235 G43 Z1.0 H5 M08

N240 G84 G98 Z–.42 R.2 F11.0

N245 G80 Z1.0 M09

N250 G91 G28 Z0.0

N255 M01

(T6 .242 DIAMETER "C" DRILL)

N260 T6 M06

N265 G90 G80 G40 G49

N270 G00 G54 X.28 Y8.0 S996 M03

N275 G43 Z1.0 H6 M08

N280 G81 G98 Z–.350 R.1 F5.0

N285 X19.5 Y1.2

N290 G80 Z1.0 M09

N295 G91 G28 Z0.0

N300 M01

(T7 .250 DIAMETER REAMER)

N305 T7 M06

N310 G90 G80 G40 G49

N315 G00 G54 X.28 Y8.0 S458 M03

N320 G43 Z1.0 H7 M08

N325 G85 G98 Z–.29 R.1 F2.7

N330 X19.5 Y1.2

N335 G80 Z1.0 M09

N340 G91 G28 Z0.0

N345 M01

(T8 7/16 DRILL TO OPEN FOR SLOTS)

N350 T8 M06

N355 G90 G80 G40 G49

N360 G00 G54 X1.1 Y8.5 S524 M03

N365 G43 Z1.0 H8 M08

N370 G81 G98 Z–.40 R.1 F3.1

N375 X18.0

N380 Y.5

N385 X10.5 Y1.775

N390 G80 Z1.0 M09

N395 G91 G28 Z0.0

N400 M01

(T9 7/16 DIAMETER ROUGHING END-MILL)

(RADIUS COMPENSATION D9 = .240)

N405 T9 M06

N410 G90 G80 G40 G49

N415 G00 G54 X1.1 Y8.5 S524 M03

N420 G43 Z1.0 H9 M08

N425 Z.2

N430 M98 P0060

N435 X18.0 Y.5

N440 M98 P0060

N445 X18.0 Y18.5

N450 M98 P0060

N455 X10.5 Y1.775

N460 G01 Z–.260 F10.0

N465 G41 X11.0 D9 F2.0

N470 Y1.85

N475 G03 X10.7 Y2.15 I–.3

N480 G01 X9.55

N485 G03 X9.25 Y1.85 J–.3

N490 G01 Y1.7

N495 G03 X9.55 Y1.4 I.3

N500 G01 X10.7

N505 G03 X11.0 Y1.7 J.3

N510 G01 Y1.775

N515 G40 X10.5

N520 G00 Z.2

N525 X20.25 Y6.5

N530 G01 Z–.26 F10.0

N535 G41 Y6.875 D9 F2.0

N540 X18.25

N545 G03 Y6.125 J–.375

N550 G01 X20.25

N555 G40 Y6.5

N560 G00 Z.2

N565 X2.5 Y–.25

N570 G01 Z–.26 F10.0

N575 G41 X3.0 D9 F2.0

N580 Y.4

N585 G03 X2.75 Y.65 I–.25

N590 G01 X2.25

N595 G03 X2.0 Y.4 J–.25

N600 G01 Y–.25

N605 G40 X2.5

N610 G00 Z1.0 M09

N615 G91 G28 Z0.0

N620 M01

(T10 7/16 DIAMETER FINISHING END-MILL)

(RADIUS COMPENSATION D10 = .2187)

N625 T10 M06

N630 G90 G80 G40 G49

N635 G54 G00 X1.1 Y8.5 S524M03

N640 G43 Z1.0 H10 M08

N645 Z.2

N650 M98 P0061

N655 X18.0 Y8.5

N660 M98 P0061
N665 X1 8.0 Y.5
N670 M98 P0061
N675 X10.5 Y1.775
N680 G01 Z–.26 F10.0
N685 G41 X11.0 D10 F2.0
N690 Y1.85
N695 G03 X10.7 Y2.15 I–.3
N700 G01 X9.55
N705 G03 X9.25 Y1.85 J–.3
N710 G01 Y1.7
N715 G03 X9.55 Y1.4 I.3
N720 G01 X10.7
N725 G03 X11.0 Y1.7 J.3
N730 G01 Y1.775
N735 G40 X10.5
N740 G00 Z.2
N745 X 20.25 Y6.5
N750 G01 Z–.26 F10.0
N755 G41 Y6.875 D10 F2.0
N760 X18.25
N765 G03 Y6.125 J–.375
N770 G01 X20.25
N775 G40 Y6.5
N780 G00 Z.2
N785 X2.5 Y–.25
N790 G01 Z–.26 F10.0
N795 G41 X3.0 D10 F2.0
N800 Y.4
N805 G03 X2.75 Y.65 I–.25
N810 G01 X2.25
N815 G03 X2.0 Y.4
N820 G01 Y–.25
N825 G40 X2.5
N830 G00 Z1.0 M09
N835 G91 G28 Z0.0
N840 M01

(T10 7/16 DIAMETER FINISHING END-MILL)

(RADIUS COMPENSATION D10 = .2187)

N845 G90 G80 G40 G49

N850 G00 G54 X–.5 Y7.5 S229 M03

N855 G43 Z1.0 H10 M08

N860 G01 Z–.26 F10.0

N865 G41 X0.0 D10

N870 X.5 Y10.0 F1.3

N875 G40 X1.0 Y10.5

N880 G00 Z1.0 M09

N885 G91 G28 Z0.0

N890 G28 X0.0 Y0.0 M05

N895 M30

Subprograms for Complex Program Example O0422

O0060

(SUBPROGRAM OF PROGRAM O0422 FOR ROUGHING SLOTS)

N1 G00 Z.2

N2 G91 G01 Z–.260 F10.0

(D9 = .4375 ÷ 2 + .01)

N3 G41 Y.25 D9 F1.5

N4 G03 Y–.5 J–.25

N5 G01 X.5

N6 G03 Y.5 J.25

N7 G01 X–.5

N8 G40 Y–.25

N9 G90 G00 Z.2

N10 M99

O0061

(SUBPROGRAM OF PROGRAM O0422 FOR FINISHING SLOTS)

N1 G00 Z.2

N2 G91 G01 Z–.460 F10.0

(D10 = .4375 ÷ 2)

N3 G41 Y.25 D10 F1.5

N4 G03 Y–.5 J–.25

N5 G01 X.5

N6 G03 Y.5 J.25

N7 G01 X–.5

N8 G40 Y–.25

N9 G90 G00 Z.2

N10 M99

MILLING EXAMPLE

Example: Milling, Program 423

Figure 4-62 is the drawing for the milling example. The tooling for this project is listed in Chart 423.

O0423

N10 G90 G80 G20 G40 G49

(T1 #6 CENTER DRILL)

N15 T1 M06

N20 G00 G54 X0.0 Y0.0 S687 M03

N25 G43 Z1.0 H01 M08

N30 G81 G98 Z–.5 F2.7 R.1

N35 X–2.5 Y–2.0

N407 M98 P0070

N45 X0.0 Y–1.5

N50 M98 P0071

N55 G80 Z1.0 M09

N60 G91 G28 Z0.0 M19
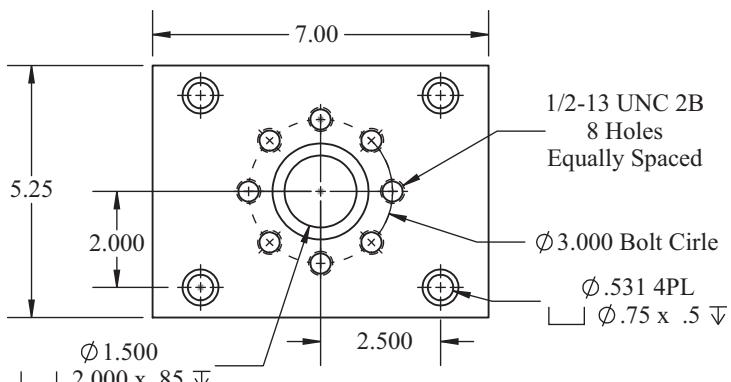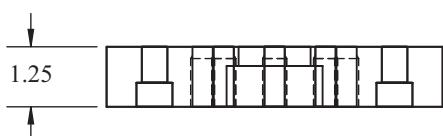
Figure 4-62
Drawing for Milling
Example

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0423	
Workpiece Zero: G54 X = <u>Part Center</u> Y = <u>Part Center</u> Z = <u>Top Part Surface</u> Setup Description: Material = Steel			
Tool #	**Description**	**Offset**	**Comments**
T1	#6 HSS Center Drill	H1	
T2	17/32 Diameter HSS Drill	H2	
T3	27/64 Diameter HSS Drill	H3	
T4	1.0 Diameter HSS Drill	H4	
T5	.75 Diameter 2-Flute HSS End Mill	H5	
T6	1.0 Diameter HSS Roughing End Mill	H6	D6 =.490
T7	1.0 Diameter 4-Flute HSS Finishing End Mill	H7	D7 =.500
T8	1/2-13 Tap	H8	

N65 M01

(T2 .531 DIAMETER DRILL)

N70 T2 M06

N75 G90 G80 G40 G49

N80 G00 G54 X–2.5 Y–2.0 S648 M03

N85 G43 Z1.0 H2 M08

N90 G73 G98 Z–1.45 R.1 F3.8

N95 M98 P0070

N100 G80 Z5.0 M09

N105 G91 G28 Z0.0 M19

N110 M01

(T3 .4218 DIAMETER DRILL)

N115 T03 M06

N120 G90 G80 G40 G49

N125 G00 G54 X0.0 Y–1.5 S816 M03

N130 G43 Z1.0 H03 M08

N135 G73 G98 Z–1.43 R.1 F4.9

N140 M98 P0071

N145 G80 Z 1.0 M09

N150 G90 G28 Z0.0 M19

N155 M01

(T4 1.0 DIAMETER DRILL)

N160 T4 M06

N165 G90 G80 G40 G49

N170 G00 G54 X0.0 Y0.0 S275 M03

N175 G43 Z1.0 H4 M08

N180 G73 G98 Z–1.65 F1.65 R.1

N185 G80 Z1.0 M09

N190 G91 G28 Z0.0 M19

N195 M01

(T5 .750 2-FLUTE END MILL)

N200 T5 M06

N205 G90 G80 G40 G49

N210 G00 G54 X–2.5 Y–2.0 S458 M03

N215 G43 Z1.0 H5 M08

N220 G82 G98 Z–.75 R.1 F3.6

N225 M98 P0070

N230 G80 Z1.0 M09

N235 G91 G28 Z0.0 M19

N240 M01

(T6 1.0 DIAMETER ROUGHING END MILL)

N245 T6 M06

N250 G90 G80 G40 G49

N255 G54 G00 X0.0 Y0.0 S343 M03

N260 G43 Z1.0 H6 M08

N265 G01 Z–1.26 F50.0

N270 G42 X–.5 F2.7 D6

N275 G02 X–.5 I.5 J0.0

N280 G01 G40 X0.0 Y0.0 F50.0

N285 Z–.83

N290 G42 X–1.0 Y0.0 F2.7

N295 G02 I1.0 J0.0

N300 G01 Y0.0 F50.0

N305 G00 Z1.0 M09

N310 G91 G28 Z0.0 M19

N315 M01

(T7 1.0 DIAMETER 4-FLUTE FINISHING END MILL)

N320 T7 M06

N325 G90 G80 G40 G49

N330 G54 G00 X0.0 Y0.0 S343 M03

N335 G43 Z1.0 H7 M08

N340 G01 Z–1.26 F50.0

N345 G42 X–.5 F2.7 D6

N350 G02 X–.5 I.5 J0.0

N355 G01 G40 X0.0 Y0.0 F50.0

N360 Z–.85

N365 G42 X–1.0 Y0.0 F2.7

N370 G02 I1.0 J0.0

N375 G01 Y0.0 F50.0

N380 G00 Z1.0 M09

N385 G91 G28 Z0.0 M19

N390 M01

(T8 .500–13 TAP)

N395 T8 M06

N400 G90 G80 G40 G49

N405 G00 G54 X0.0 Y–1.5 S114 M03

N415 G43 Z1.0 H8 M08

N420 G84 G98 Z–1.3 F8.7 R.2

N425 M98 P0071

N430 G80 Z1.0 M09

N435 G91 G28 Z0.0 M19

N440 G28 X0.0 Y0.0

N445 M30

Subprograms for Milling Example Program O0423

O0070

(SUBPROGRAM FOR COUNTERBORES)

N1 X–2.5 Y–2.0

N2 Y2.0

N3 X2.5

N4 Y–2.0

N5 M99

O0071

(SUBPROGRAM FOR BOLT CIRCLE)

N1 X0.0 Y–1.5

N2 X–1.0607 Y–1.0607

N3 X–1.5 Y0.0

N4 X–1.0607 Y1.0607

N5 X0.0 Y1.5

N6 X1.0607 Y1.0607

N7 X1.5 Y0.0

N8 X1.0607 Y-1.0607

N9 M99

Example: Calculation Needed for Tool Path Geometry

Figure 4-63 is the drawing for the calculations example and the tooling for this project is listed in Chart 424.

Note: The example given here is limited to a finishing pass only. In order to write a program, follow the necessary mathematical calculations shown in Figure 4-64.

$$Y = 1.6 - Y_1 \quad Y_1 = b - a$$

$$\tan \alpha = \frac{b}{1.0} \quad b = 1.0 \times \tan \alpha = 1.0 \times \tan 30° = .5773$$

$$\cos \alpha = \frac{R}{a} \quad a = \frac{R}{\cos \alpha} = \frac{.25}{\cos 30°} = .2886$$

$$Y_1 = b - a = .5773 - .2887 = .2887$$

$$Y = 1.6 - Y_1 = 1.6 - .2887 = \underline{1.3113}$$

$$Y_2 = 1.6 + d$$

$$\cos \alpha = \frac{d}{R} \quad d = R \times \cos \alpha = .2165$$

$$Y_2 = 1.6 + d = 1.6 + .2165 = \underline{1.8165}$$

$$X_2 = 1.0 - c$$

$$\sin \alpha = \frac{c}{R} \quad c = R \times \sin \alpha = .125$$

$$X_2 = 1.0 - c = 1.0 - .125 = \underline{.875}$$

Example: Offsetting Tool Path, Program 424

O0424

N10 G90 G80 G20 G40 G49

N15 G00 G54 X–.2 Y–.2 S6100 M03

N20 G43 Z1.0 H1 M08

N25 G00 Z–.1

N30 G01 G41 X0.0 D1 F36.0

N35 Y1.3113

N40 X.875 Y1.8165

N45 G02 X1.0 Y1.85 I.185 J–.2165

N50 G01 X1.55

N55 X1.6 Y1.8

N60 Y1.3

N65 G03 X1.7 Y1.2 I.0

N70 G01 X2.125

N75 G03 X2.225 Y1.3 J.1

N80 G01 Y1.8

N85 X2.275 Y1.85

N90 X3.1

N95 X3.25 Y1.7

N100 Y.2

N105 G02 X3.05 Y0.0 I–.2

N110 G01 X–.02

N115 G40 Y–.2

N120 G00 Z1.0 M09

N125 G91 G28 Z0.0 M05

N130 G28 X0.0 Y0.0

N135 M30
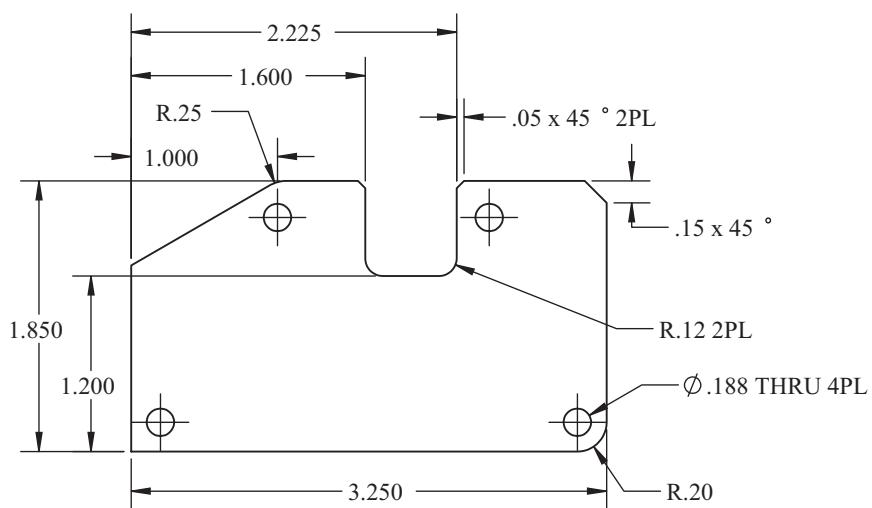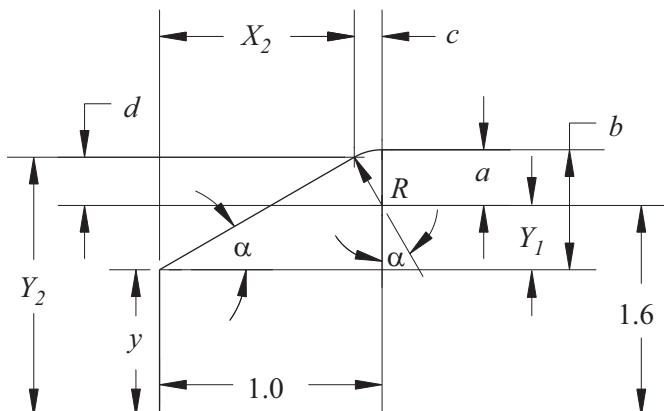
Figure 4-63
Drawing for Offsetting Tool Path Example

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0424	
G54 = Workpiece Zero: X = <u>Lower Left Corner</u> Y = <u>Lower Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = .100 Thick free cutting Brass The part is clamped to a fixture plate using the 4 holes machined in a prior operation			
Tool #	Description	Offset	Comments
T1	.1875 Diameter 4-Flute HSS End Mill	D1	SFM= 300 FPT = .0015

Figure 4-64
Diagram for Mathematical
Calculations
for Tool Path Geometry

Example: Drilling 1000 Holes Using Only Six Blocks of Code

In Figure 4-65, the grid of holes shown represent only a portion of those on the part. The actual part contains 4 rows of 250 holes each. The tool is placed in the spindle prior to beginning.
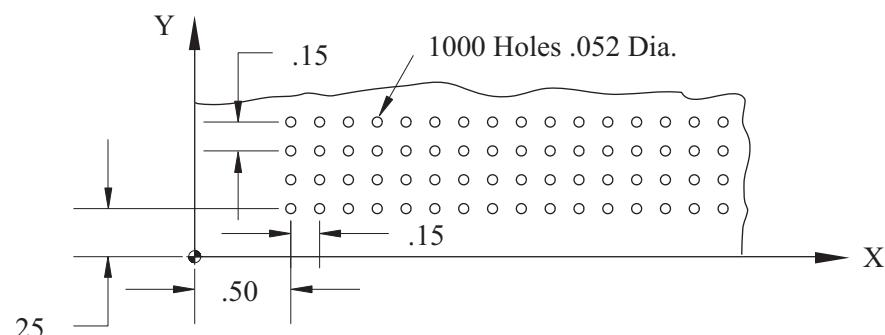
Figure 4-65
Drawing for Drilling 1000 Holes Example

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0425	
G54 = Workpiece Zero: X = <u>Lower Left Corner</u> Y = <u>Lower Left Corner</u> Z= <u>Top Part Surface</u> Setup Description: Material = 4140 Steel			
Tool #	Description	Offset	Comments
T1	.052 Diameter HSS Drill		SFM = 45 FPT = .001

Example: Drilling 1000 Holes, Program 425 (Chart 425)

O0425

N10 G90 G00 G80 G40 G49 G54 X.5 Y.25 S3307 M03

N15 G43 M98 P0080 L4 Z1.0 H1

N20 G91 G28 X0.0 Y0.0 Z0.0 M30

O0080

N1 G91 G81 G98 X.15 Z–.05 L250 R.05 F6.6 M08
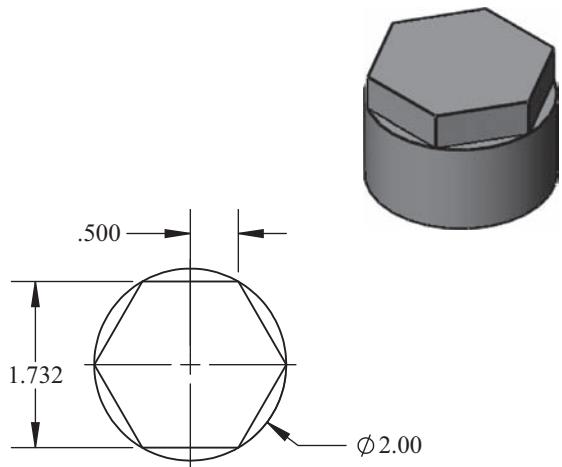
N2 G80 G00 X–36.15 Y.15

N3 M99

Example Illustrating Application of Coordinate System Rotation

In this example, the coordinate system rotation function is used to machine the part in shown in Figure 4-66. The tool used in this project is listed in Chart 426.

Figure 4-66
Drawing for Coordinate System
Rotation Example

.500

1.732

⌀2.00

Part 4 Programming CNC Machining Centers

Machine: Machining Center		Program Number: O0426	
G54 = Workpiece Zero: X = <u>Part Center</u> Y = <u>Part Center</u> Z= <u>Top Part Surface</u> Setup Description: Material = Low Carbon Steel Use a V-Block to hold the part in a vise with.5 inch minumum above the jaws			
Tool #	**Description**	**Offset**	**Comments**
T7	1/2 Diameter 4-Flute HSS End Mill	D7 = .250	

Example: Coordinate System Rotation, Program 426

O0426

(1/2 DIA HSS 4FL END MILL)

N10 G90 G20 G80 G40 G49

N15 T7 M6

N20 G00 G54 X.75 Y1.366 S720 M3

N25 G43 Z1.0 H7 M08

N35 G68 X0.0 Y0.0 R0.0

N40 M98 P291

N45 G68 X0.0 Y0.0 R60.0

N50 M98 P291

N55 G68 X0.0 Y0.0 R120.0

N60 M98 P291

N65 G68 X0.0 Y0.0 R180.0

N70 M98 P291

N75 G68 X0.0 Y0.0 R240.0

N80 M98 P291

N85 G68 X0.0 Y0.0 R300.0

N90 M98 P291

N95 G69

N100 G00 Z1.0

N105 G40 G80 Z5.0 M09

N110 G91 G28 Z0.0

N115 G91 G28 X0.0 Y0.0

N120 M30

Subprogram for Coordinate System Rotation Example Program O0028

O0291

(Rotation Subprogram)

N1 Z.1

N2 G1 Z–.4 F20.

N3 G2 X.5 Y1.116 R.25 F7.0

N4 G1 X–.5

N5 G2 X–.75 Y1.366 R.25

N6 G0 Z.1

N7 M99

Part 4 Study Questions

1. What other program word is necessary when programming G01 linear interpolation?

      a. S

      b. F

      c. T

      d. H

2. Which code activates the positive tool length offset?

      a. G54

      b. G40

      c. G41

      d. G43

3. When programming an arc, which letters identify the arc center location?

      a. X, Y, and Z

      b. A, B, and C

      c. I, J, and K

      d. Q and P

4. Which of the following are modal commands?

      a. G01

      b. G00

      c. F

      d. All of the above

Part 4 Programming CNC Machining Centers

5. Cutter diameter compensation G41 and G42 offset the cutter to the left or the right. Which command is used for climb (down) milling?

      a. G40

      b. G41

      c. G42

      d. G43

6. When programming an arc, an additional method exists that does not use I, J, and K. Which program word is used?

      a. A

      b. B

      c. C

      d. R

7. A block of codes at the beginning of the program are used to cancel modal commands and are called the "Safety Block". They are:

      a. G90 G54 G00

      b. G20 G90 G00

      c. G40 G80 G49

      d. G91 G28 G00

8. When programming an arc using the R address, a negative sign must be used with the radius value in order to create a full circle.

      T or F

9. When using the canned drilling cycle G83, which letter identifies the peck amount?

      a. Z

      b. P

      c. K

      d. Q

Part 4 Programming CNC Machining Centers

10. When a subprogram is used, which miscellaneous code is used to call it?

      a. M06

      b. M99

      c. M98

      d. M19

11. What character is used in the program to instate an optional block skip?

      a. (

      b.)

      c. ;

      d. /

12. What character is inserted at the End of a Block when the program is loaded into the controller?

      a. ;

      b. /

      c. (

      d.)

13. The letter address, O, is used to identify the program number and has no other use in programming.

      T or F

14. Sequence or line numbers are identified by the letter address N. The program will not execute if they are omitted.

      T or F

15. The letter address H is used to indicate a tool length offset register number. Which preparatory function is it used in conjunction with?

      a. G54

      b. G43

      c. G42

      d. G41

Part 4 Programming CNC Machining Centers

16. When using canned drilling cycles, which of the following codes are used to return the drill to the initial plane?

      a. G99

      b. G98

      c. G90

      d. G92

17. When using canned drilling cycles, which of the following codes are used to return the drill to the reference plane?

      a. G99

      b. G98

      c. G90

      d. G92

18. When using canned drilling cycles, what other letter address is necessary to identify the reference plane position?

      a. P

      b. Q

      c. R

      d. S

19. When using cutter diameter compensation in a program, what letter address is used to identify the location of the value of the offset?

      a. A

      b. R

      c. H

      d. D

20. Which G-Code is used to cancel Cutter Diameter Compensation?

      a. G40

      b. G41

      c. G42

      d. G43

Part 4 Programming CNC Machining Centers

21. Match the following definitions with the proper M-Code.

Program Stop	M30 ___
Optional Stop	M06 ___
End of Program	M03 ___
Spindle on Clockwise	M00 ___
Spindle on Counterclockwise	M01 ___
Spindle Off	M19 ___
Flood Coolant On	M05 ___
Flood Coolant Off	M98 ___
Spindle Orientation	M08 ___
Subprogram Call	M04 ___
Subprogram End	M09 ___
Tool Change	M99 ___

22. Match the following definitions with the proper G-Code.

Linear Interpolation	G90 ___
Circular Interpolation Clockwise	G00 ___
Rapid Traverse	G81 ___
Dwell	G40 ___
Absolute Programming	G28 ___
Incremental Programming	G41 ___
Canned Cycle Cancellation	G91 ___
Peck Drilling Cycle	G42 ___
Drilling Cycle	G43 ___
Cutter Diameter Compensation Left	G01 ___
Cutter Diameter Compensation Right	G83 ___
Cutter Diameter Compensation Cancellation	G84 ___
Zero Return Command	G04 ___
Inch Programming	G54 ___
Metric Programming	G92 ___
Tapping Cycle	G21 ___
Absolute Program Zero Setting	G02 ___
Fixture Offset Command	G80 ___
Positive Tool Length Compensation	G20 ___

Part 4 Programming CNC Machining Centers

23. Match the following definitions with the proper letter address.

Program Number	A ___
Sequence Number	B ___
Preparatory Function	C ___
Miscellaneous Function	D ___
X Coordinate	F ___
Y Coordinate	G ___
Z Coordinate	H ___
Reference Plane Designation	I ___
Feed rate	J ___
Spindle Function	K ___
Subprogram Repeats	L ___
Tool Function	M ___
Tool Length Compensation Register	N ___
Diameter Compensation Register	O ___
Rotational Axis about the X	P ___
Rotational Axis about the Y	Q ___
Rotational Axis about the Z	R ___
Dwell in Seconds	S ___
Peck Amount in Canned Drilling	T ___
Arc Center X-axis	X ___
Arc Center Y-axis	Y ___
Arc Center Z-axis	Z ___

24. If a linear move is programmed G01 X1.5 Y1.5, what is the angle of the resulting cut?

       a. 30°

       b. 180°

       c. 45°

       d. 90°

25. If a rapid traverse positioning move is programmed along the X and Y axes and the distances are unequal, the shortest distance will be achieved first.

       T or F

26. When using fixture offset programming G54–G59, it is possible to have multiple offsets in one program.

T or F

27. In the Tool Sheet Chart for Program 421, the Feeds and Speeds are missing. Identify the SFM and FPT for each tool.

PART 5

Computer-Aided Design and Computer-Aided Manufacturing (CAD/CAM)

OBJECTIVES:

1. Explain common CAD/CAM capabilities.

2. Describe the Mastercam X8 CAD/CAM Graphical User Interface.

3. Learn terminology specific to CAD/CAM.

4. Apply geometry creation techniques common to CAD/CAM.

5. Explain how to use the CAD/CAM software to create toolpath and CNC program code.

6. Complete a toolpath from a solid model for a lathe.

7. Complete a toolpath from a solid model for milling.

WHAT IS CAD/CAM?

Computer-aided design and computer-aided manufacturing (CAD/CAM) utilize computers to design drawings of part feature boundaries in order to develop cutting toolpath and CNC machine code (a part program). By using CAM, specific cutting tool data are defined. Toolpaths are then created by selecting drawing geometry that identify feature shapes and how they are going to be machined. Drawing in CAD is simply constructing a drawing using lines, arcs, circles, and points and then positioning them relative to each other on the screen. One of the major benefits of CAD/CAM is the time saved. It is much more efficient than writing CNC code line-by-line.

CAD/CAM is now the conventional method of creating mechanical drawings and computer numerical control (CNC) programs for machine tools. CAD is the standard throughout the world for generating engineering drawings. The personal computer is a powerful tool used by manufacturing and many other divisions within an organization. Engineers seldom use the drafting board to design their projects; they now use computers extensively. Designers can create the drawings needed and share them electronically with the manufacturing department. Drawings are exported/saved to a common file format, such as the Initial Graphics Exchange Specification (IGES), Standard for the Exchange of Product model data (STEP), Stereo Lithographic (STL), and Drawing Exchange Format (DXF) to name a few. There continue to be huge advancements in the design field and solid models are prevalent over two-dimensional drawings.

Many CAM systems import solid model files directly for toolpath creation (drag-and-drop functionality), allowing the manufacturing engineer/CNC programmer to create the toolpath and assign cutting tool information relative to the desired results. CAD is limited in nature to the generation of engineering drawings, whereas CAD/CAM combines both design and manufacturing capabilities.

When using CAD/CAM, the drawing may be created from scratch or imported from a CAD program using one of the file formats mentioned earlier. It is not necessary to have the drawing dimensioned for this operation, but the full scale of the part is required. The CAD/CAM operator assigns the tools and their order of usage while creating the toolpath. There are many CAD/CAM programs on the market today. The most popular ones are easy to use; they have excellent service and support as well as a strong background of reliability. To make good use of this computing power, it is important to fully understand the machining processes. Just as CNC doesn't change the actual machining, the same is true of CAD/CAM for programming.

Remember: the overall objective of CAD/CAM is to generate a toolpath for a CNC machine in the form of a CNC program. It is imperative to have a full understanding of the rectangular and polar coordinate systems. It is also necessary to have a complete understanding of cutting tool selection, speeds, and feeds. Nearly all CAD/CAM programs will automatically develop speed and feeds data based on the tool and work material selection saved in a database, however, adjustments are commonly made. This database of information can and should be updated to match the requirements for your shop for the best results.

When constructing the part geometry, consider the type of machining operation. For instance, if the desired result is to drill a hole using a standard drill, construction of only the point that represents the holes center location in the coordinate system is necessary.

In this chapter, Mastercam X8 is featured as the CAD/CAM software used for the programming examples. This chapter is intended only as an introduction to CAD/CAM. To cover the full extent of its capabilities would require an entire text, if not volumes. Many other CAD/CAM programs available today use similar techniques to accomplish the same result.

ASSOCIATIVITY

The concept of associativity is inherent to most modern CAD/CAM programs. The information input to the program regarding the toolpath, tool, material, and parameters specific to each are linked to the geometry. This means that if any of the parameters for the parts mentioned above are changed, the other related data can be regenerated to take these changes into account without recreating the entire operation.

CAD/CAM is the tool of choice for creating CNC programs. The power it has now will only be magnified in the future. The basic concepts that are demonstrated here are merely a taste of the capabilities CAD/CAM has to offer.

PERSONAL COMPUTERS

A personal computer or Apple Mac has important minimum requirements to run this type of software. Normally, a large screen or even two is desirable for ease of viewing the geometry created. CAD/CAM programs require a lot of hard disk space so a large hard drive is also recommended. Because CAD/CAM is used to create complex drawings and perform graphic simulations, the computer has the following basic needs:

- The memory the computer uses to access files while working on them is called RAM (Random Access Memory). A large amount of RAM is highly recommended (individual software manufacturers have recommended minimums).
- The computer's processing speed is listed in MHZ (Megahertz). Again, the higher the number, the better.
- The computer's graphics card and monitor controls the screen resolution. A powerful graphics card is strongly advised. Remember to consult the specific minimum requirements for the CAD/CAM program you choose.

WINDOWS

To be successful using CAD/CAM, it is necessary to understand the use of a personal computer's operating systems and software programs. Microsoft Windows is the

most widely used operating system on personal computers. The software presented in this chapter was installed on a PC with the current Windows operating system. With CAD/CAM, the operator must understand the operating system and have basic skills, including double-click, right mouse button, and the mouse pointer. Just like most computer programs in use today, CAM programs use a Graphical User Interface (GUI) for ease of input. CAM programs use icons, toolbars, and menu systems to guide the user through program use. Many functions can be accessed via short-cut keystrokes and mouse button clicks. Some functions require a single-click with the left mouse button to activate a command.

The following short description uses Mastercam X8 to create geometry, toolpath, and program code for CNC machines.

MASTERCAM X8, PROGRAM STARTUP

From the Windows main screen, look at the desktop to find a shortcut icon for Mastercam X8, then double-click the left-click mouse button on it (Figure 5-1). If there is no shortcut icon on the desktop, press the start button in the lower left corner of the task bar. Select All Programs at the bottom of the list and use the scroll bar to locate and select Mastercam X8. Once the program is started by default, Mastercam is in the Design mode. This example uses Milling, so change to the Mill system by selecting **MACHINE TYPE** from the Menu Bar. To activate the Mill system, single click on **MACHINE TYPE**, select Mill, and then select Default for the machine configuration. Figure 5-2 displays the Mastercam main interface screen.

Figure 5-1 Mastercam X8
Shortcut Icon
Courtesy CNC Software Inc.
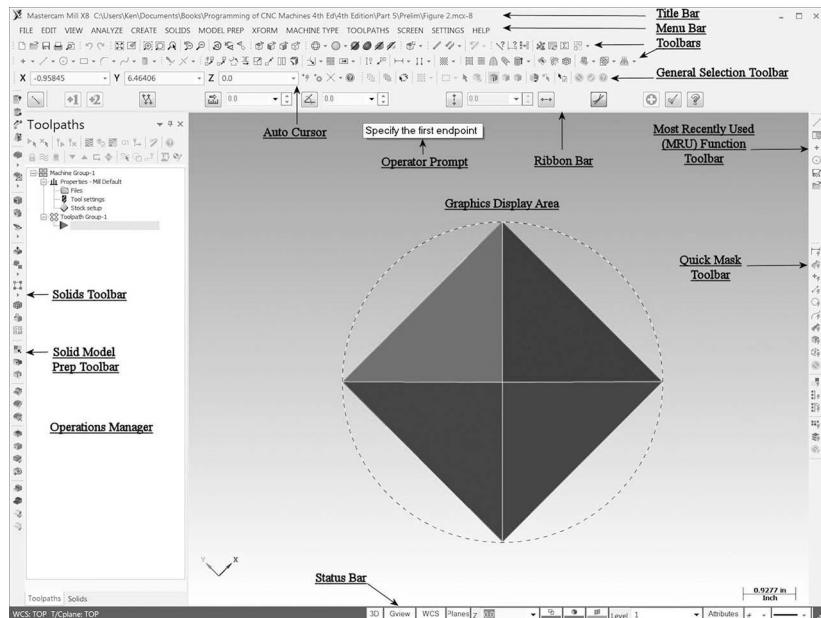
Figure 5-2 Mastercam X8 Graphical User Interface
Courtesy CNC Software Inc.

MASTERCAM X8 USER INTERFACE

Getting familiar with the user interface is imperative for efficient use. A brief description of the main components of the interface is given here. The more you use the functions, the more natural it will become. Don't forget that there is extensive help available within the program by selecting **Help** from the Menu Bar or the question mark (?) icon for context sensitive questions and use Alt+h for the keyboard shortcut.

Title Bar

The title bar lists the software name, version, current mode (e.g., Design, Mill, Lathe, Wire, or Router), and the file name and location. For example:

Mastercam Mill X8 C:\MCAMX8\MCX\KEN.MCX

Menu Bar

Many of the program functions are accessible by left clicking while over the icon on the tool bars. A list of the same functions is available on the Menu Bar. The functions from the Menu Bar can be activated by using the mouse or by using short-cut keystrokes. Press the Alt key to activate access to these keystrokes. They are generally the first letter of the word or, if not, they are identified by an underscore of the letter needed to activate the command. In the following example, when the Menu selection method is used, the Menu item used will be **bolded** in the instructions and the short-cut keystroke underscored to match the Mastercam menu bar. Specific directions will be included in the instructions.

Graphics Display Area

This is where the part geometry and toolpath are displayed during creation and verification. The Graphics Display Area is set to Gradient (start color #101, end color#15) by default. It can be changed to another color if necessary, by selecting **SETTINGS** from the Menu Bar, then Configuration, then Colors from the tree, then Gradient background start/end color or Graphics background color for a solid color. Use caution when changing the background colors because some of the other color combinations will not allow visibility of selection, chaining, and toolpath, etc.

Toolbars

The Toolbars tab contains icons that act as short-cut buttons to all types of functions and are accessible via mouse clicks. A fly-out descriptor appears when the mouse selection arrow is used to hover over one of the tool bar icons. The user can activate/deactivate the on-screen tool bars by accessing the Customize dialog box (Figure 5-3), right-clicking near the Menu bar and choosing either Customize or Load Workspace.
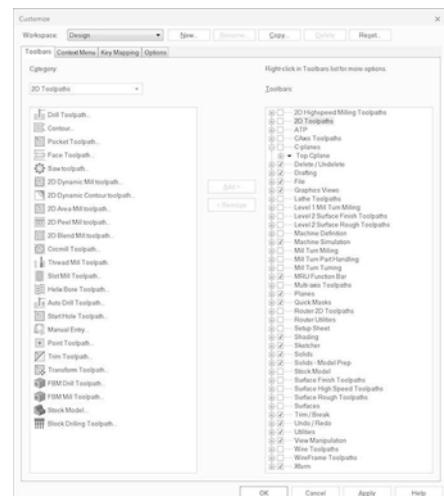
Figure 5-3 Customize Dialog Box
Courtesy CNC Software Inc.

Part 5 Computer-Aided Design and Computer-Aided Manufacturing

Auto Cursor Ribbon Bar

Use this tool bar to set coordinate values for X, Y, or Y-axis input, set Auto Cursor selection types, activate Fast Point mode, and change the configuration for Auto Cursor (Figure 5-4).

General Selection Ribbon Bar

Use this tool bar to set general selection masking functions for chaining and deleting of entities (Figure 5-5).

Active Function Ribbon Bar

Once a drawing command is active, the Active Function Ribbon Bar becomes available for input that is specific to the type of entity being created (Figure 5-6). Some examples in the line creation mode are: Multi-line, Line length, Polar Angle, Horizontal or Vertical, and Tangent.

Status Bar

The Status Bar (Figure 5-7) is placed at the bottom of the screen and gives the user feedback about the current drawing status with information about 2D/3D, Geometry views (Gview), Construction and Tool Planes (CPlane and TPlane), Colors, Levels, Attributes, Point Styles, Line Styles, Line Weights, World Coordinate Systems (WCS), and Groups. The user can set any of the aforementioned data from the Status Bar as well.

Figure 5-4 Auto Cursor Ribbon Bar
Courtesy CNC Software Inc.

Figure 5-5 General Selection Ribbon Bar
Courtesy CNC Software Inc.

Figure 5-6 Active Function Ribbon Bar
Courtesy CNC Software Inc.

Figure 5-7 Status Bar
Courtesy CNC Software Inc.

Most Recently Used (MRU) Toolbar

The Most Recently Used (MRU) Toolbar (Figure 5-8) is usually docked vertically on the right side of the User Interface. It is a temporary tool bar that contains the commands most recently used. It is made handy for quick access to the last 10 commands used.

Figure 5-8 Most Recently Used (MRU) Toolbar
Courtesy CNC Software Inc.

Quick Masks Tool Bar

The Quick Masks tool bar allows selection control by left-clicking of the type of mask desired (Figure 5-9). By default, the types of masks available on the Quick Mask tool bar are Points, Lines, Arcs, Splines, Surfaces, Solids, Wireframe, Result, Group, Color, Level, and Drafting. By selecting a function button on the Quick Masks tool bar, masking is set to select all entities of the type chosen. Pressing the right button while hovering over the desired Quick Mask type button sets masking to single selection.

For example, if you are working on a complex drawing and need to select Points only, you can left-click on the Quick Masks Points button; only Points will be selected. By right-clicking the button, points can be selected one at a time until selection is complete.

Operations Manager

The Operations Manager contains information about Toolpaths, Solids, and Art (Figure 5-10). Under View, each can be accessed by selecting Toggle

Figure 5-9 Quick Masks Tool Bar
Courtesy CNC Software Inc.
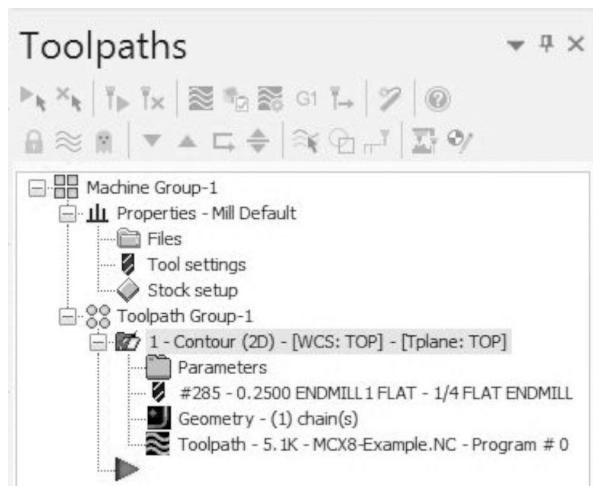
Figure 5-10 Operations Manager
Courtesy CNC Software Inc.

Toolpaths or Solids Manager. The Toolpaths Manager describes the Machine Group and Toolpath Group, its Properties and parameters. At the top of the Toolpath Manager are the icons specific to Toolpath Selection, Backplot, Verification, Post Processing, and Editing.

Note: When the escape key (esc) is pressed, the software accepts the current command and steps back to the main level in the menu, essentially clearing the active command. If the Enter key is pressed after creation of a feature of the geometry, the current command is accepted and the mode of geometry entry is retained.

MACHINE GROUP SETUP AND GEOMETRY CREATION

In many cases, the machinists/programmers are not provided with electronic files for the part geometry they must create. CAD/CAM programs provide the ability to recreate the drawing for the purpose of making the toolpath program. For this example, all of the geometry shown in Figure 5-11 must be recreated. The first consideration when recreating geometry is where to set the Workpiece Zero or origin. The zero location for this part is in the lower left corner. It makes good sense to use this same location to start drawing. The part will be clamped with cap screws to a holding fixture that is held in a vise. In this example, only the contour will be machined with a 3/16 2-Flute end mill. The part blanks provided are 1/4 thick by 2-inch wide 2024 Aluminum that is pre-machined to 3.35 inches in length.

MACHINE TYPE

When you begin the process planning steps to write any CNC program, you must select the type of machine required to perform the job. The software configuration purchased dictates the types of machines available to select, including Mill, Lathe, Wire, Router, or Design. The information entered here establishes the data needed for the program, such as stock size tools and part origin.

Note: When Mastercam is started, the Design mode is active. You can create the design and then activate the Machine Type before toolpath creation can begin but it is a matter of preference.
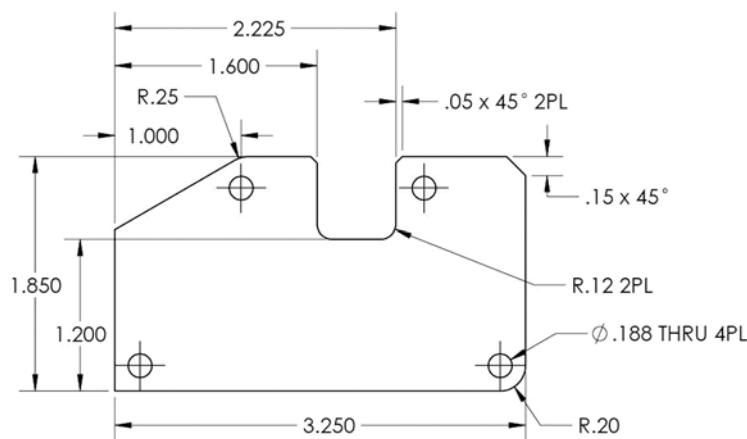
Figure 5-11 Drawing for Mastercam CAD Example
Courtesy CNC Software Inc.

- From the Menu Bar, single click the left button on **Machine Types**. With the mouse pointer, select <u>M</u>ill and then <u>D</u>efault from the drop down list for this example.

Note: To access keyboard menu shortcuts, press the ALT key. Then the letter of the menu item will be underlined to indicate which key is used to activate the command.

If you choose to use an alternate machine, select Manage List to access the Machine Definition Menu Management dialog (Figure 5-12). Machine Group 1 will appear in the Operations Manager.

Figure 5-12 Machine Definition Menu Management
Courtesy CNC Software Inc.

STOCK SETUP

- To access the Stock setup dialog, left-click on the plus sign next to Properties—Mill Default in the Operations Manager, as shown in Figure 5-13. Then left-click on Stock setup to access the dialog displayed in Figure 5-14.

Explanations are given below for each item in the Machine Group Properties dialog in Figure 5-14 that will be used in this example.

- <u>Stock Plane</u> should be set to correspond with the standard geometry view to be used for the toolpath needed. *In this case, the Top is correct.*

Figure 5-13 Machine Group Properties
Courtesy CNC Software Inc.

Figure 5-14 Machine Group Properties, Stock Setup
Courtesy CNC Software Inc.

- <u>Shape</u> should be set to match the raw material shape or it can be set to an existing solid by selecting the model or any file by identifying the location. *In this case, the part blank information is input to the fields for X (3.4), Y (2.0), and Z (.25).* To set the stock size, key in the dimensions for the X, Y, and Z, including raw material (the Z value must be positive).

- <u>Display</u> establishes the way the material is displayed in the graphics window. Whether the stock is included when Fit to the Screen function is used, the part file is displayed as a Wire Frame or as a Solid. Use Wire Frame.

- <u>Stock Origin</u> identifies the coordinate locations of the raw material zero values. The origin for the part may be moved to a desired corner location by left-clicking once on the black arrow and then left-clicking over the new location. *For our example, left-click on the arrow in the center of the part, then on the lower left corner.* The arrow indicating the part origin will move to the corner selected. The exact numerical locations may also be input into the X, Y, and Z to accomplish this. The Y and X coordinates define the outer boundary of the raw stock of the part. When the arrow button below the Stock Origin section is pressed, the display will revert to the drawing file, allowing manual selection of the stock origin directly from the drawing. In this example, there is .150 excess material in the length and width. *By inputting half the difference in the length and width into the Stock Origin fields for X (.075), Y (−.075), and Z (.25), the graphical verification will display the appropriate amount of metal removal for each side of the part.*

- <u>Select corners</u> … This button allows for manual selection of the stock corners from the active drawing. Once the selection is made, the values are input automatically into the fields that identify the part's boundaries.

- <u>Bounding box</u>. This button allows the user to return to the drawing to manually make a boundary around the drawing and expand it in each axis to represent the stock.

- <u>NCI Extents</u> … This button compares and calculates all of the toolpaths in the NCI file; it creates stock boundaries that include all tool movement extents.

TOOL SETTINGS

Explanations are given below for each item in the Machine Group Properties dialog, Tool Settings tab (Figure 5-15) that will be used in this example.

- Enter the desired number <u>Program #</u> in the first field.
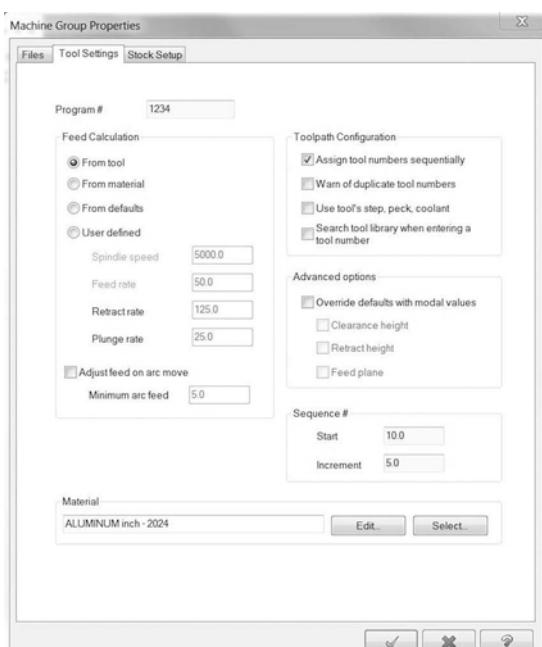
Figure 5-15 Machine Group Properties, Tool Settings
Courtesy CNC Software Inc.

- Feed Calculation may be based from the tool, material, system defaults, or User defined. *In this case, select From Tool.*

- Toolpath Configuration contains four optional settings related to tools. Select the check box to Assign tool numbers sequentially (otherwise tool library numbers will be used).

- Advanced Options contain an Override of default values for the three check boxes.

- Sequence #'s for the program code are set here. Sequence # Start is the starting number for the line sequence "N" numbers in the program. Any starting value can be entered here. Common starting numbers are 1, 2, 5, 10, or 100. Increment is the sequence number increments for the line sequence numbers. Any value may be entered here. Set the values of 10 and 5.

- Material type is assigned by selecting from the materials library after pressing the Select… button. By pressing the Edit … button, the user can change feeds and speeds settings by gaining access to the Material Definition Manager.

When all of the settings are completed, press the checkmark button to accept them. The stock will be displayed in the graphics display.

From the Menu Bar, select **TOOLPATHS** and then Tool Manager … from the drop down menu. The dialog in Figure 5-16 will then be displayed. The upper section is for the Part and the lower has the Library tools. Use the dropdown to select either Both, Part, or Library.

Place a check in the Filter Active check box in the Library section of the dialog; then press the Filter … button to activate the filters. If this check box is not checked, then all 427 tools in the standard database will be displayed in the list. Activate this filter so the number and type of tools displayed can be refined.

From the Tool List filter dialog (Figure 5-17), press the button labeled None. Then press the button for flat end mills only to be displayed and press the check mark
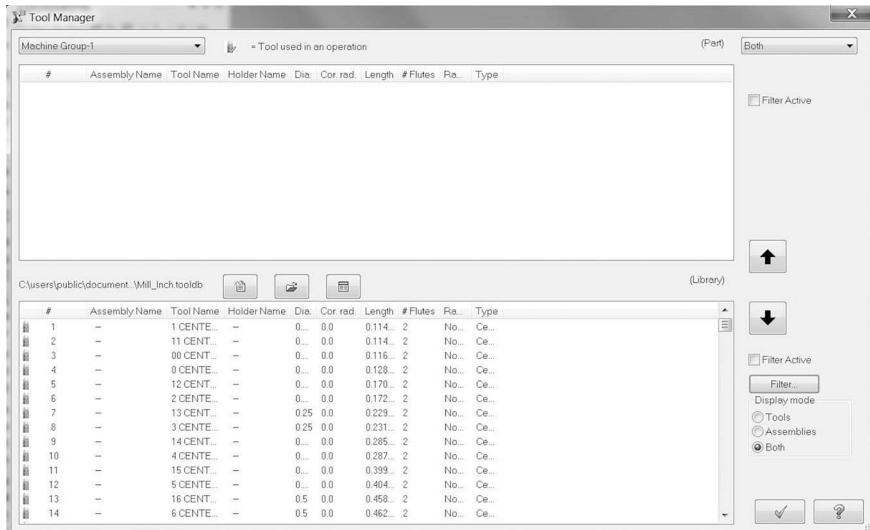
Figure 5-16 Tool Manager Dialog
Courtesy CNC Software Inc.
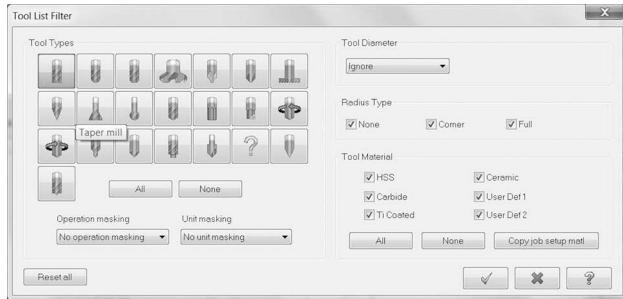
Figure 5-17 Tool List Filter Dialog
Courtesy CNC Software Inc.

to accept. A refined list of only end mill tools will be displayed. Select the 3/16 inch diameter end mill from the list (#284) by double-clicking on it. The chosen end mill will now display in the Part section of the Tools Manager.

This is the only tool needed for the program in our example. If additional tools are needed, the same process may be followed to add them. Now all the Machine Group setup information is complete and the geometry creation may begin. Any of the items that have been set so far can always be edited if changes are required.

MASTERCAM X8 GEOMETRY CREATION STEP-BY-STEP

Please note the steps listed here are by no means the only method by which this geometry may be created. Individual preferences and speed will ultimately be the determining factor. The Menu Bar method will be listed as the method of geometry creation.

To create the drawing in Figure 5-11, follow these steps:

Note: To activate the keystroke short-cuts, press the Alt key first.

1. From the Menu Bar, position the mouse pointer over **<u>C</u>REATE** and press the left-click button, or press the letter "C" to activate the create menu.

Note: Even though the short-cut keystroke letter is in uppercase, a lower case "c" will accomplish the same thing.

2. By the same method, choose **<u>L</u>ine**, or press the letter "L" to activate the line menu. Select, **Create Line <u>E</u>ndpoint** …

In the Operator Prompt area, the prompt reads: "Specify the first endpoint". (The system defaults to the sketch method of point entry).

3. Key in the coordinates for the start point of the line 0, 0 and press Enter (where 0, 0 = X value, Y value). The Z value is zero unless otherwise specified via the Status Bar or keyed here.

4. Press the letter H on the keypad or the Horizontal Icon on the Active Function Ribbon Bar to lock the line into the horizontal line mode.

In the Operator Prompt area, the prompt reads: "Specify the second endpoint." The first point is drawn and a horizontal line is attached to it. As the mouse is moved, the line stretches like a rubber band to the point. The default mode is Sketch, so whenever the mouse is moved, the line length changes correspondingly in the Ribbon Bar readouts.

5. The specific line **<u>L</u>ength** (value of the line length) may be entered by pressing L on the keyboard or by selecting the Length Icon on the Active Function

Ribbon Bar; either one locks the line into the line length mode. Then enter the signed value of the length. To accept the input, left-click the button, then press Enter again to accept the Y value for the line.

6. To adjust the value of the Y-axis, press the Enter key. Then enter the new Y value into the Active Function Ribbon Bar to set the Y location and press Enter to accept the input.

Note: Alternatively, if the second endpoint coordinate values are known, press the spacebar to activate the Fast Point mode and begin typing in the values; the line will be created when Enter is pressed a second time. If the coordinate value for a point stays the same, no entry is necessary for that axis. For example: Key in: 3.250 for the second end point and press Enter, Enter.

The Line creation mode is still active, except now change to a vertical line type.

7. Press the letter V to activate the Vertical Line mode or left-click on the Vertical icon in the Active Function Ribbon Bar to lock the line into the vertical line mode.

In the Operator Prompt area, the prompt reads "Specify first endpoint".

8. Use the mouse to select the endpoint of the line created last. The Auto Cursor Configuration allows easy selection of the line endpoint. Left-click anywhere near the line endpoint to select it.

9. Press the letter L to set the specific Length and then key in 1.850. Left-click the button to accept the length. Press Enter to accept the X coordinate of 3.25.

The Line creation mode is still active, except now change back to a horizontal line type.

10. Use the mouse to again select the endpoint of the line created last.

11. Key in 2.225, 1.85, and Enter, then Enter again to accept, and another Enter to accept the Y coordinate.

12. Use the mouse to again select the endpoint of the line created last. Activate the vertical line type.

13. Key in 1.6 and press Enter, Enter, and Enter.

14. Use the mouse to again select the endpoint of the line created last. Activate the horizontal line type.

15. Press the spacebar and key in 1.85; then press Enter, Enter.

Note: Be sure to enter a comma first in order to accept the current location for the X-axis value.

16. Use the mouse to again select the endpoint of the line created last.

17. Press the spacebar and key in 1.0 and press Enter, Enter.

Note: Because the start of the .25 radius arc is known to be at that location in X and Y, a construction line must now be created in order to obtain the proper endpoint location for the .25 radius arc (Figure 5-18).
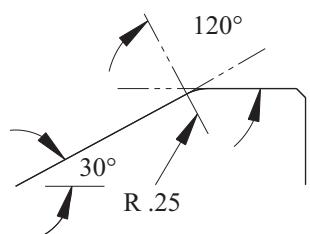
Figure 5-18
Construction Line for
Arc Creation
Courtesy CNC Software Inc.

18. Key in 1.0, 1.6. (The coordinate value of Y is obtained by subtracting the known radius value of .250 from 1.85 = 1.6.)

19. Press the letter A on the keyboard to select the Angle line type.

In the Operator Prompt area, the prompt reads "Enter the angle in degrees."

20. Key in 120° and press Enter. (The angle of 30° is given on the print; when the 30° is added to 90°, it equals 120°.)

In the Operator Prompt area the prompt reads "Enter the line length."

21. Press the letter L on the keyboard and enter .25, Enter, and Enter to accept the line.

22. From the Menu Bar, select **CREATE**, <u>A</u>rc, and Create Arc En<u>d</u>points.

23. Select the upper endpoint of the construction line last created.

In the Operator Prompt area the prompt reads "Enter the second point."

24. Select the endpoint of the horizontal line that was last created and then press Enter.

25. Press the letter R on the keyboard to set the radius of the arc to .25 and press Enter.

Upon doing this, two full circles will be drawn. In the Operator Prompt area the prompt reads "Select an Arc."

26. Carefully position the mouse selection point over the arc you wish to keep and left-click to accept the selection; press Enter to accept.

Notes: It may be necessary to zoom in on the area of the drawing for easier selection of the correct arc.

If the Auto Cursor is enabled, a small box will be displayed when the cursor is positioned over the endpoints. The Auto Cursor may be enabled or disabled by placing a check mark to Enabled or removing the check mark to disable. When this function is ON, it is useful for simple drawings, but may be annoying on complex drawings.

27. From the Menu Bar select **CREATE,** <u>L</u>ine**,** Create Line <u>E</u>ndpoint …

Notice in the Active Function Ribbon Bar whether the tangent function is active. If it is not, press the icon to activate it. It may once again be useful to zoom in on the area of the drawing for easier selection of the line arc tangency point.

28. Right-click the mouse while anywhere within the graphics display area and select Auto Cursor from the list (Figure 5-19). Press the Disable All button. Then place a checkmark to Enable only the Tangent setting; press the checkmark button to accept the changes.
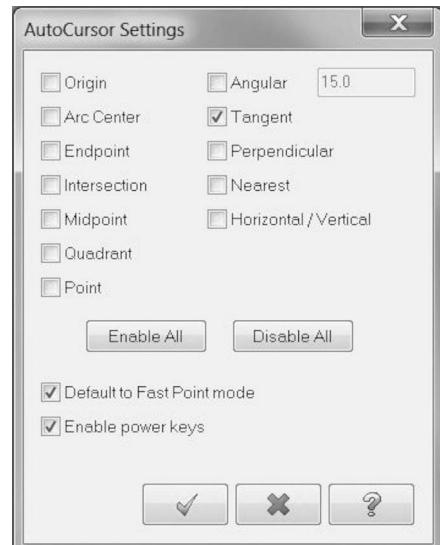
Figure 5-19 Auto Cursor Settings Dialog
Courtesy CNC Software Inc.

29. Use the mouse to position the selection point over the .25 radius arc and press Enter to accept the selection.

30. Press the letter A on the keyboard to activate the Angle mode.

31. Key in an angle of 210° and then press Enter (because 180° + 30° = 210°).

32. Press the letter L on the keyboard to activate the Length mode and enter a value of 2.0 inches. This length will be sufficient to intersect with the vertical line rising from the X, Y zero location we will draw next. A line 2-inches long will be drawn in both directions from the tangency point.

33. Select the line you wish to keep (left side of the tangent arc) by left-clicking on it and then pressing Enter, Enter.

34. Right-click the mouse anywhere within the graphics display area; then select Auto Cursor again to adjust the settings to include Endpoints.

35. While the line command is still active, select the endpoint of the line at the X, Y zero location.

36. Press the letter V on the keyboard to activate the vertical line function.

37. Key in Y1.5, Enter, Enter, and Enter to accept the entry. (If the drawing exceeds the display area of the screen, press the F2 function key to Un-zoom the current display by .5.)

Now take the steps necessary to trim and delete the excess lines, chamfer, and fillet the corners as specified on the print.

38. From the Menu Bar, select **Create, Fillet,** Fillet Entities

In the Operator Prompt area the prompt reads "Fillet: Select an entity." (The default fillet size is a .250 radius). Press the letter R on the keyboard and enter the desired value.

39. Press the letter R on the keyboard and enter the .2 for the radius.

Note: The Style drop-down, should be set to Normal on the Active Function Ribbon Bar.

40. Use the mouse to select an entity, pick line 1, then line 2 as shown in Figure 5-20.

41. While still in fillet mode, change the radius to .12 for the two other fillets by pressing the letter R on the keyboard, entering .12, and pressing Enter.

42. Pick each vertical and horizontal line, as shown in Figure 5-21. Press Enter after all four picks to accept the fillets.

All fillets are now done. Now trim the excess lines off from the left-click angular and vertical lines.

43. From the Menu Bar select **EDIT,** Trim/Break, then Trim/Break/Extend.

44. On the Active Function Ribbon Bar, press the "Trim 2 entity" button.
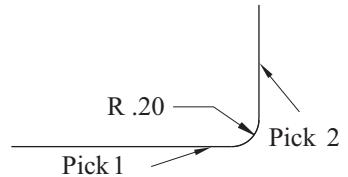
Figure 5-20 Selection
Points for the
.2 Radius Fillet
Courtesy CNC Software Inc.
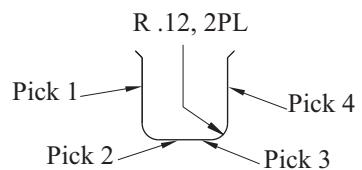
Figure 5-21 Selection
Points for the
.12 Radius Fillets
Courtesy CNC Software Inc.

In the Operator Prompt area, the prompt reads "Select the entity to trim/extend."

45. Choose the angular line, then the last vertical line (as shown in Figure 5-22), and press Enter to accept.

The excess lines will be trimmed. Next the chamfers must be created.

46. From the Menu Bar, select **CREATE**, Chamfer, Chamfer Entities …

Figure 5-22 Selection Points for Trimming
Courtesy CNC Software Inc.

47. Set the value for the chamfer to .05 by pressing the number 1 on the keyboard, entering .05, and pressing Enter. The default value for distances is .25; change as needed.

48. Pick the necessary lines, as shown in Figure 5-23, and press Enter to accept.

49. Press the number 1 on the keyboard again to change the chamfer distance setting to .15 for the last chamfer.

50. Pick the necessary lines, as shown in Figure 5-24, and press Enter to accept.

Figure 5-23 Selection Points for Chamfering
Courtesy CNC Software Inc.

The construction line that was used for the creation of the .25 arc, tangent to the angular line, should be deleted now.

51. From the Menu Bar, select **EDIT,** Delete, Delete entities …

52. Use the mouse to position the cursor over the construction line. Then left-click to select it, press Enter, and the line will be deleted.

The drawing is complete and the file should be saved.
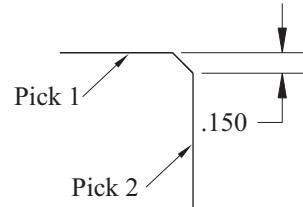
Figure 5-24 Selection Points for Chamfering the .15 Distance
Courtesy CNC Software Inc.

54. From the Menu Bar, select **FILE**, Save As …

The "Save As" dialogue box will appear.

55. Choose the folder location where you wish to save. Then Key in the File name desired and press the Save button. *For this example, use the File Name: CAD-EX.*

The completed drawing should be displayed in the graphics area, as shown in Figure 5-25.

TOOLPATH CREATION

Once the part geometry has been created, toolpaths can be generated from it. Choose the type path that applies to the geometry. For this example, use contour.

1. From the Menu Bar, select **TOOLPATHS,** Contour …

At this point, a dialog box will appear. Enter new NC name. The name CAD-EX will be present because the file was named earlier when saved. Leave it as is or change

Figure 5-25 Finished Geometry
Courtesy CNC Software Inc.

it to whatever is desired. Press the OK checkmark to accept the entry and the Chaining dialog will appear.

2. In the Operator Prompt area, the prompt reads "Select Contour chain 1."

Note: The default setting is Chain, so the contour may be picked without first acti-vating the chaining mode. There are many options available for setting chaining options via the Chaining dialog displayed. Take a moment to review each setting and refer to the online help for detailed descriptions.

3. Left-click on the vertical line geometry just above the Workpiece Zero location.

This selection determines where the actual toolpath will begin, so take care to select an appropriate point. An arrow on the drawing contour will be displayed indicating the direction of the chain. The chain direction determines the tool travel direction. Tool offset direction is always dependent on chaining. For instance, if the arrow is pointing up, as shown in Figure 5-26, the tool will travel in the Y positive direction on the part geometry and to the left of the line.
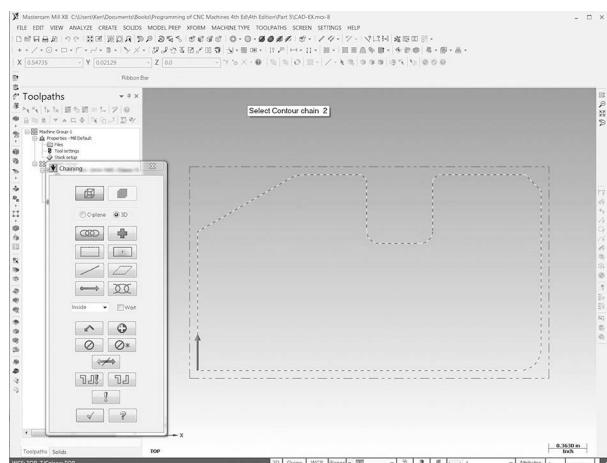
Figure 5-26 Contour Toolpath Chain
Courtesy CNC Software Inc.

If the arrow direction is not correct, press **Reverse** on the Chaining dialog to change the direction. The offset direction from the contour determines climb or conventional cutting. Press **Reverse** again and the direction toggles. If the starting location is not the desired location, press the Expand dialog button (down arrow in upper left corner of the dialog) to expose the Start buttons of the dialog. Move Forward or Back until the desired starting position for the tool start is found.

4. Press the green check mark to accept the toolpath chain.

The 2D Toolpaths—Contour dialog will appear as shown in Figure 5-27 (Contour type is preselected).

Notice that the .1875 diameter flat end mill identified earlier from the Tool Manager is present. If an optional tool is required, it can be added from this dialog by pressing the "Select library tool …" button. Follow the instructions for filtering and selection described earlier for the Tool Manager. A tool parameter can be edited by right-clicking on it and selecting, "Edit tool …." By right-clicking in the white-space area of the tool list, you can create an entirely new tool by selecting "Create new tool …." Adjust the feeds and speeds parameters for a selected tool by right-clicking on the tool in the list and then selecting the "Feed speed calculator …" from the list. Press OK to accept the settings.

2D Toolpaths—Contour Dialog

The 2D Toolpaths—Contour dialog has a Parent/Child Tree structure in the left-click panel and specific parameters section for each on the right.

Toolpath Type

Choose from the options of Contour, Pocket, Facing, or Slot Mill. Each will have unique parameter requirements.

Tool

When "Tool" is selected from the tree, a dialog displays that contains the following fields.
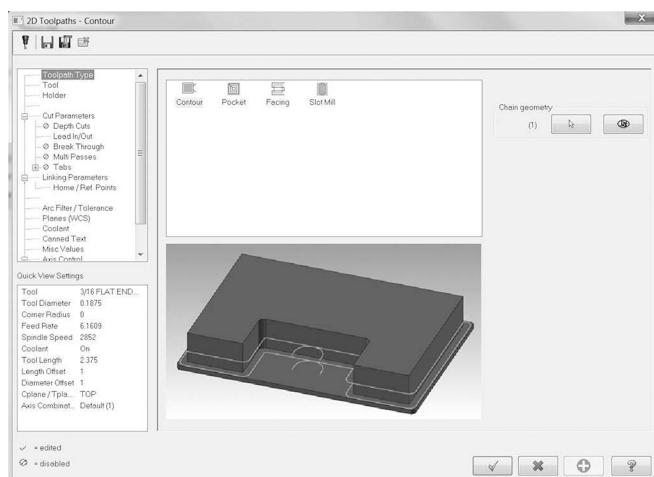
Figure 5-27 2D Toolpaths—
Contour Dialog
Courtesy CNC Software Inc.

Tool dia: Defaults to the diameter of the selected tool.

Corner radius: Allows the input of the corner radius of a bull or ball-nose style cutter as described in the Tool Library.

Tool #: Defaults to the number that matches the library tool number but it can be changed to any number desired. In this example, tool numbers are assigned sequentially based on the setup information provided earlier.

Head #: Refers to the magazine where the tool is mounted in the case of multiple tool magazines.

Len. offset: Defaults to correspond with the number selected from the tool library on the first selected tool or, in this case, sequentially. The length-offset number will normally correspond with the tool number, but any number may be used.

Dia. Offset: Defaults to match the tool selected from the tool library, but it also can be changed to any number, dependent upon the available pockets in the tool changer of the machine being programmed. Because of the common use of geometry offsets today, the tool diameter offset will have the same number as the tool; the value of the diameter or radius will be entered in the appropriate column of the offset register at the machine controller.

Feed rate: The linear feedrate at which the tool will travel. It is determined by the tool and workpiece material selected. The feed rate can be modified in this field.

Note: When any of the six fields in this section of the dialog are adjusted, the others are updated automatically.

Spindle speed: Allows for setting of the r/min desired.

FPT: Feed per Tooth field allows for setting of the desired value.

SFM: Surface Feet per Minute field allows for setting of the desired value.

Plunge rate: The linear Z-axis feedrate at which the tool will travel determined by the tool and workpiece material that is selected. This feed can be modified in this field.

Retract rate: The speed at which the tool will retract to the reference plane at completion of the cutting cycle can be modified in this field.

At the bottom of the dialog there are two self-descriptive checkboxes available: one that forces a tool change and one that allows use of rapid retract.

Comment: This section of the dialog allows for the input of program comments that are intended to aid the operator and will be inserted within the program for operator guidance.

Holder

The tool holder type and details can be set via this dialog.

Cut Parameters

Select Cut Parameters to open the dialog shown in Figure 5-28. As each tree item is selected, a unique dialog is displayed. The drop-down selection box for Compensation type allows selection of Computer, Control, Wear or Reverse Wear, or Off. If Control is selected, the post processor will develop a diameter compensation number specific to the tool, then insert function G41 or G42 and a D# into the code of the program. For this example, accept Computer for the type.
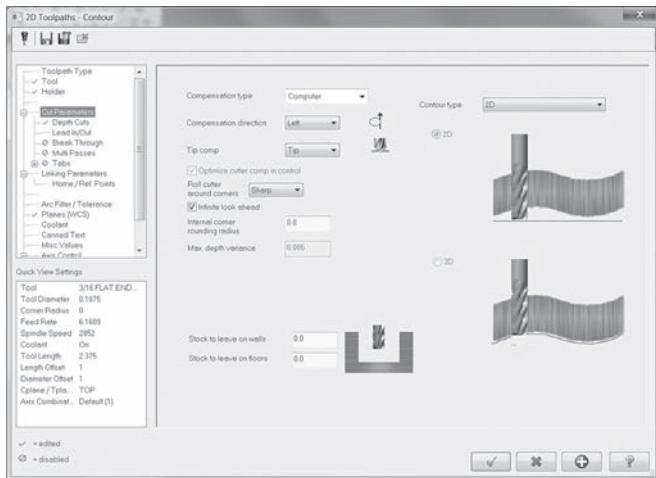
Figure 5-28 Cut Parameters
Dialog
Courtesy CNC Software Inc.

The Compensation direction selection button determines the offset direction of the cutter. (Be sure the correct direction is indicated; for this example, it should be Left-click.)

Press the OK checkmark to accept the settings.

Depth Cuts: Enables multiple Z-step passes. The number of finish cuts, finish steps, and a maximum roughing step can be established when selected.

Lead In/Out: Enables control of toolpath entry and exit moves by lines and arcs relative to the direction of the chain mentioned earlier.

Break Through: Enables the entry of an amount the tool should break through the material.

Multi Passes: When checked, enables roughing and finishing passes in the X Y direction with control of finish passes at final depth or all depths.

Tabs: Allows for the creation of tabs along the contour.

Linking Parameters

Select the Linking Parameters tree item to open the dialog shown in Figure 5-29.

Input the appropriate values for Clearance …, Retract …, Feed plane …, Top-of-stock …, and Depth … as required. In this example: Clearance is set to an Absolute value of 1 inch; Feed plane is set to an Absolute value of .1 inch; Top-of-stock is set to an Absolute value of 0.0; and Depth is set to an Absolute value of −.25.

Home/Ref. Points: This dialog allows for setting of a specific tool change and Reference positions. Ref. points are used to adjust Approach and Retract toolpath intermediate points.

Arc Filter/Tolerance: By converting motions within a set tolerance amount, this dialog allows control of toolpath tolerances that ultimately affect surface finish.

Planes: Opens the Work coordinate system, Tool plane, Comp/construction plane dialog. By using this box, the planes for construction and machining, origins, and work offset for the toolpath can be set.
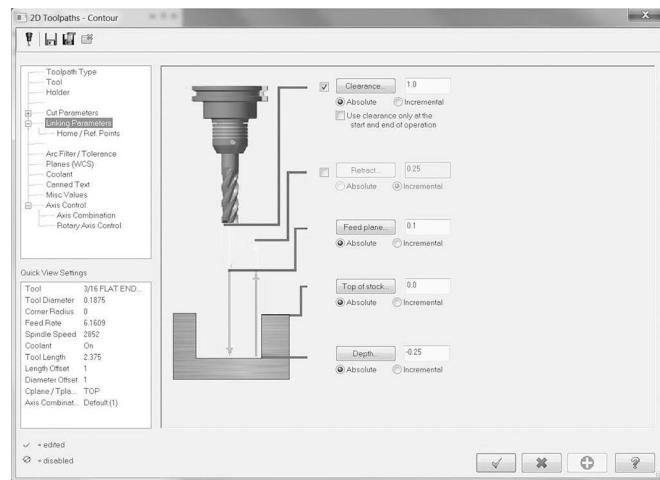
Figure 5-29 Linking Parameters
Dialog
Courtesy CNC Software Inc.

Coolant: Activates the use of flood coolant, mist coolant (if available), and thru-tool coolant On/Off setting.

Canned Text: Allows addition of program comments from a list of some commonly used text statements.

Misc Values: Can be used to set up to ten integer values. The most commonly adjusted integers are Work Coordinates (0–1 = G92 or 2 = G54); Absolute/Incremental (0 = ABS or 1 = INC); and Reference Return (0 = G28 or 1 = G30).

Axis Control: Contains Axis Combination and Rotary axis. When selected, these parameters are used to create toolpath motion where a rotating axis is used for cylindrical parts.

When all the settings are adjusted, press OK and a display of the toolpath will appear, as in Figure 5-30. To display the part geometry and toolpath, right-click in the graphics display area and pick the Isometric (WCS). Fit the screen (Alt+F1) and then Un-zoom .8 (Alt+F2).
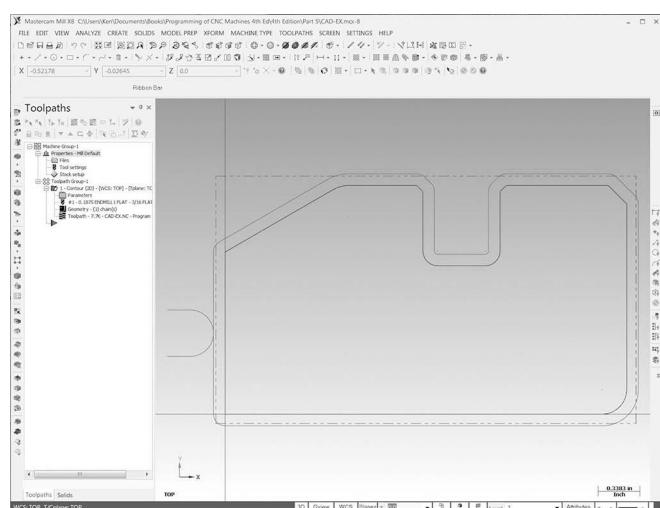
Figure 5-30 Tool Path Display
Courtesy CNC Software Inc.

1. From the Operations Manager, select the "Backplot selected operations" button (at the middle of the bar), as shown in Figure 5-31. Be sure that the Contour (2D) … folder has a green check mark on it before proceeding. If it does not have a green checkmark on the folder, choose "Select all operations" from the button bar and then select "Regenerate all selected operations".

2. Press Play (R) from the player buttons, to observe a simulation of the cutter's actual path. Backplot Display Option settings within the Backplot dialog

Figure 5-31 Toolpath Backplot Icon
Courtesy CNC Software Inc.

that are used to manipulate the display of the toolpath during playback include Display tool; Display Holder; Display rapid moves; Display endpoints, Quick verify; Options; and more (See Figure 5-32). Playback controls are located in the upper left corner of the display and include normal playback functions plus Trace mode and Run mode. Also, the Visible Motion Position slider shows toolpath progress. It can be moved either direction by selecting with the left mouse button and holding it to dynamically display the tool position. Experiment with the settings in the dialog to observe the results.

TOOLPATH VERIFICATION

To display a solid model style representation of the finished toolpath, follow these instructions.

1. From the button bar, press "Verify selected operations" for a solid model simulation of the part being cut.

2. Press the Play (or key R) button to play the simulation. The speed with which the simulation runs can be controlled by adjusting the Speed slider. The progress of
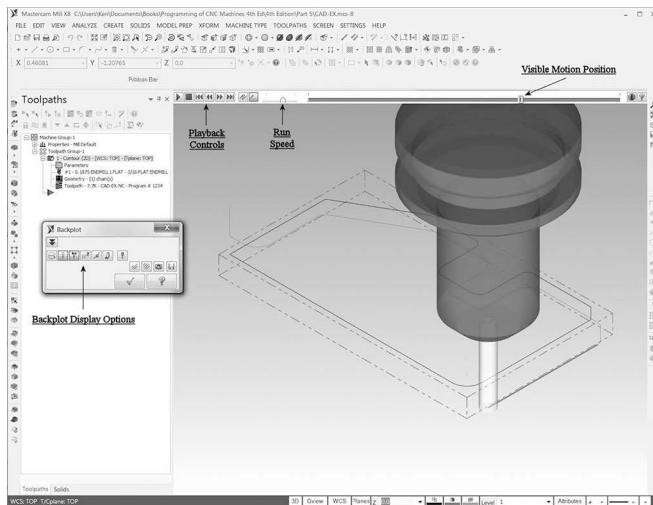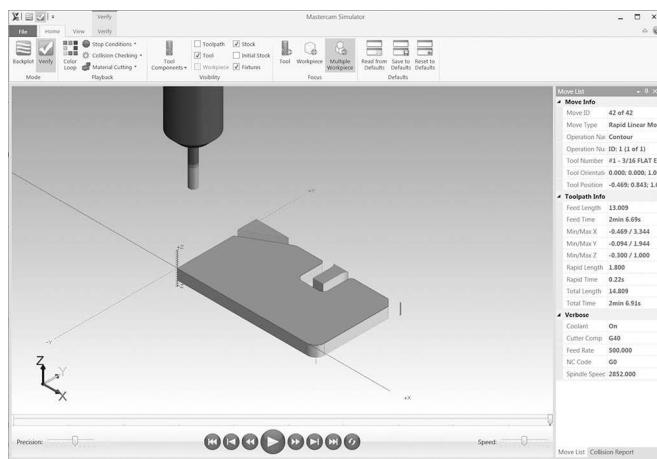
Figure 5-32 Toolpath Backplot Display
Courtesy CNC Software Inc.

Figure 5-33 Verify Toolpath
Simulation
Courtesy CNC Software Inc.

the toolpath is displayed just above the playback controls and can be dynamically moved forward and backward to display the tool position. When the toolpath is simulated, it will look similar to Figure 5-33. Experiment with the settings in the Verify dialog to see the results.

After viewing the Backplot and Verify simulations, you can still make any needed adjustments to the toolpath by selecting the Parameters folder for the Toolpath Group-1 and making the changes. Be sure to select and regenerate any operations you have adjusted before Backplotting or Verifying again. At this point, if all looks well, click on the (x) in the upper right corner of the Verify floating dialog box to close it.

POST PROCESSING

In order to actually machine the toolpath contour generated, it must first be post processed into machine-usable CNC code. This step converts the toolpath just created into CNC code that the machine tool can read in order to machine the part.

1. Press "Post selected operations" (labeled G1) from the Toolpaths toolbar.

2. Press the green checkmark Post Processing dialog to post the file.

3. When the Save As dialog appears, identify the program number to save the file. The default file name is the same as the toolpath file name. If a different file name is desired, it should be changed now. The machine tool requires a program number and is established in the Machine Group Properties, Tool Setting tab dialog (described in Tool Setting).

4. Press Save. In a matter of seconds, the program will be post processed and will open the program file editor, as shown in Figure 5-34.

Specific post processors are designed for each type of machine controller. The default controller type is a Fanuc-style G-code (MPFAN.pst) and is used here.

- Close the Mastercam Code Expert screen by pressing the close window button (x) or by choosing File from the menu; then Exit.

- To close Mastercam X, select **FILE** from the Main menu bar; then Save.

- Once the file is post processed and saved, it is ready to be sent to a specific machine controller. Check the manufacturer's manual for specific directions for this procedure.

After completing these steps, the programmer can print a set-up sheet by right-clicking within the Toolpath manager and selecting Setup sheet from the list. The style of this setup sheet can be changed by selecting options through the Settings, Configuration, Reports, and Setup Sheet. In effect, this is a form of automated process planning and this document can be used by CNC setup persons to aid in the machine setup.

Figure 5-34 Post Processed File
Courtesy CNC Software Inc.

SOLID MODEL MILL PROGRAM EXAMPLE

A solid model is commonly provided to the programmer. The steps for programing the milled part shown in Figure 5-35 are presented next.

Basic Steps for a Mastercam Mill Program

1. Open the file in Mastercam X8
2. Position the part
3. Define Machine Type
4. Define Stock
5. Define Tools
6. Program, Generate, Backplot, and Verify Toolpaths
7. Verify all paths
8. Post
9. Reports, Setup Sheets

Open the File

1. Begin by opening the file in Mastercam. File types available are those commonly used in industry (IGES, STEP, STL, etc.).

Figure 5-35 Mill Example Part Design
Courtesy CNC Software Inc.

Note: Prior to opening the file, select __SETTINGS__ from the Menu Bar; choose Configuration from the list, and Converters from the Tree. Be sure Edge Curves is checked. This will load edge curves with the Solid file and make it easier to chain for toolpaths later.

For this example, a model created in SolidWorks is used. The file can be opened using the **F**ILE, Open dialog by setting the file type to SolidWorks Files (*.sldprt;*.sldasm*) and then selecting it. Alternatively, the file can be added by dragging-and-dropping it from Windows Explorer into the Mastercam graphics display.

Position the Part

2. The part file will be oriented in the same fashion that it was created. It will commonly be necessary to translate the model into the Mastercam Work Coordinate System (WCS) frame.

• Identify where the part zero location will be on the part model.

• Press the F9 key (again to toggle off when done) to display the System Coordinate Axes of the modeled part.

• Left-click on the Dynamic XFORM … icon: follow the Operator Prompt, then Select entities to Move/Copy**.**

• Press enter and follow the prompt ctrl-right-click for gnomon settings. Uncheck the box "switch to manipulate geometry after initial gnomon placement" (Figure 5-36), then press OK.

• A gnomon is a graphical representation of three connected axes at the origin that allows the programmer the ability to manipulate transformations dynamically.

Figure 5-36 Gnomon Settings
Courtesy CNC Software Inc.

• Pick the model origin point, as shown in Figure 5-37. Follow the prompt to manipulate the axes so that the X, Y, and Z align with the direction of the intended setup. Use the left-mouse-button to accept the position.

• Toggle the Gnomon/Geometry Switch to the Manipulate Geometry mode in the graphics display (Figure 5-38). This Manipulate Geometry function will allow the model move to align with the new settings.

• On the Ribbon Bar (Figure 5-39), press the Move icon, select WCS origin from the drop-down and press the Move to origin button. Lastly, press the Align to axes button, then press OK if the

Figure 5-37 Gnomon XForm
Courtesy CNC Software Inc.

preview is what was expected. Right-click and select clear colors from the list to remove the result color.

• The model is now placed in relation to the wcs. Ḟile, Ṡave.

Define Machine Type

3. From the Menu bar, select **MACHINE TYPE**—Ṁill—Ḋefault.
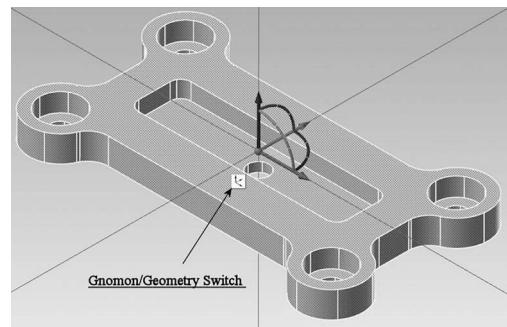
Define Stock

Figure 5-38 Gnomon/Geometry Switch
Courtesy CNC Software Inc.

4. In the operations manager, expand the Properties Mill-Default to expose Stock setup and left-click to open it. Complete the required information per the blank stock dimensions as in Figure 5-40.

• Select Bounding Box and then enter .05 to expand the X and Y stock boundaries. Press OK.

Figure 5-39 XForm Ribbon Bar
Courtesy CNC Software Inc.
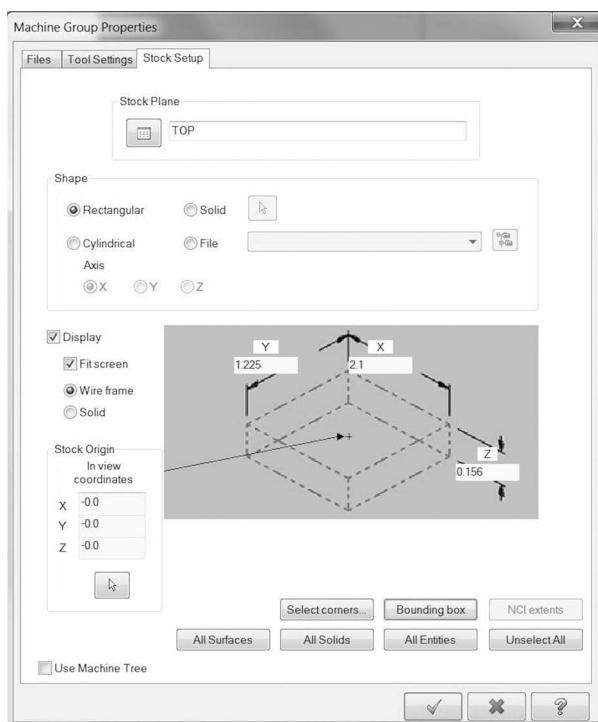
Figure 5-40 Mill Example Stock Setup
Courtesy CNC Software Inc.

- Open the Tool Setting dialog and complete, per Figure 5-41.

Note: The radio button, From Material, is set for Feed & Speed Calculations.

Define Tools

5. From the Menu Bar select **TOOLPATHS**—Tool Manager. Set the filters to minimize the displayed tools and press OK. From the library list, select each tool determined (based on the part geometry) and double-click to add to the used list (see Figure 5-42).

Program Toolpaths

6. Right-click in the empty space of the tool bar area and select Load Workspace. From the list, select the 2D Toolpaths. The Toolbar (Figure 5-43) will be placed to the left of the display area.
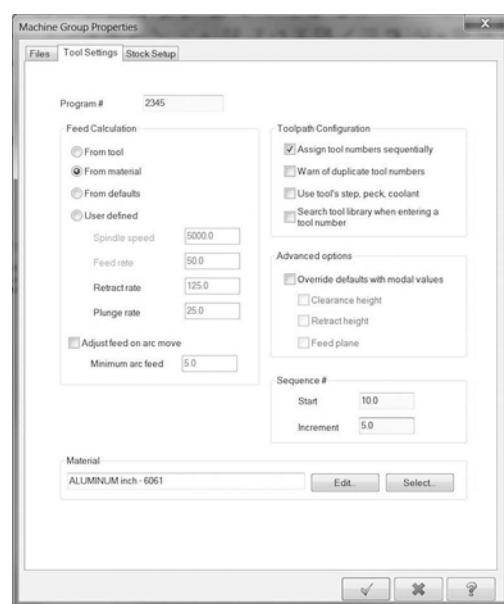
Figure 5-41 Tool Settings Dialog
Courtesy CNC Software Inc.

Figure 5-42 Mill Example Tool Manager
Courtesy CNC Software Inc.

Figure 5-43 2D Toolpaths Toolbar
Courtesy CNC Software Inc.

- Also, set the Coolant to Flood On.
- Generate, Backplot, and Verify. Make any adjustments required prior to continuing.
- Key the letter T to erase the toolpath display and again to redisplay.

Contour Toolpath

b. From the 2D Toolpaths toolbar choose the Contour… icon. Set the Chaining dialog to Solids with Edge and Loop active.

- Select the bottom edge of the contour of the part. If the entire bottom edge of the contour highlights, press OK. If the vertical face where selected highlights, press the "Other face" button from the Pick Reference Face dialog. An arrow indicating the start point of the chain and the direction of tool travel will display on the model. Use the Forward/Back buttons to change the start point and the Reverse button to switch the direction.
- Press OK to continue.
- Complete the settings within the 2D Toolpaths—Contour dialog, as displayed in Figures 5-46 and 5-47.
- Select the 1/4 Flat End Mill for this operation. Also set the Coolant to Flood On.
- In the Cut Parameters dialog, set the Compensation type to Control and the Compensation direction to Left. Setting this will output G41 and a D# to the machine code, allowing for adjustments via the Offset page at the machine controller. Generate, Backplot, and Verify. Make any adjustments required prior to continuing.

Pocketing Toolpath

c. From the 2D Toolpaths toolbar, choose the Pocket Toolpath… icon. Select the bottom edge of the contour of the pocket. Press OK when the entire bottom edge of the pocket highlights.
- Set the Cut Parameters to Climb; then change Stock to leave on walls and floors to zero (0.0) stock.
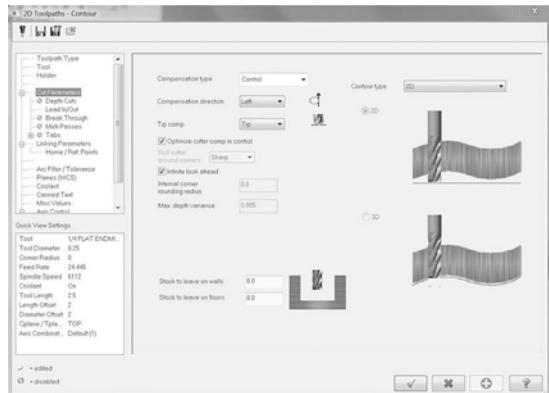
Figure 5-46 Contour—Cut Parameters
Courtesy CNC Software Inc.
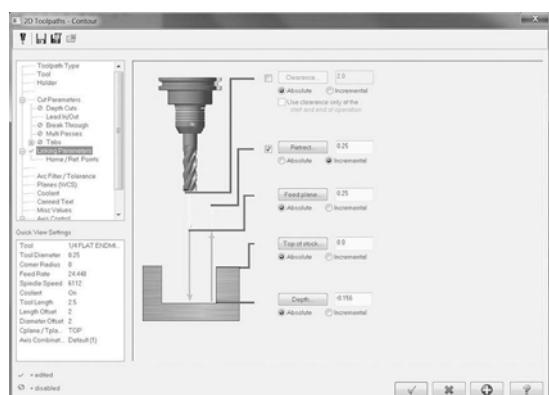
Figure 5-47 Contour—Linking Parameters
Courtesy CNC Software Inc.
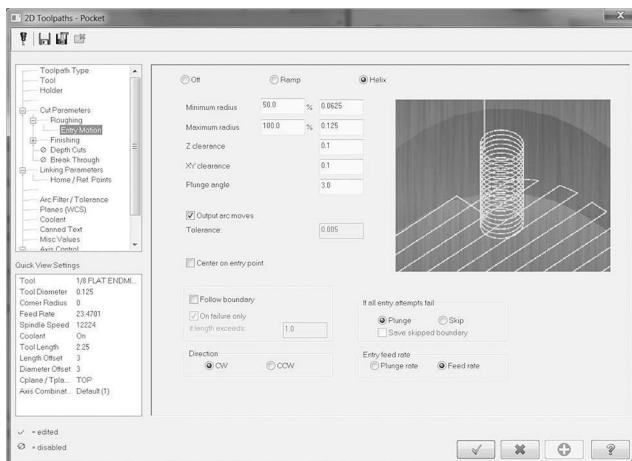
Figure 5-48 Pocket Roughing
Entry Parameters
Courtesy CNC Software Inc.
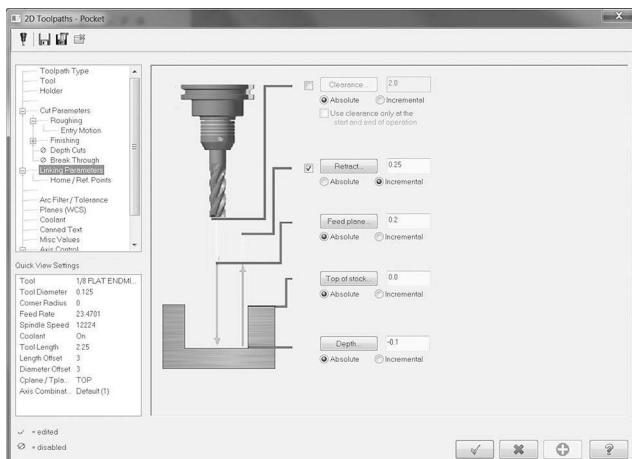
Figure 5-49 Pocket Linking
Parameters
Courtesy CNC Software Inc.

- Input the Pocket Roughing Entry Parameters as shown in Figure 5-48.

- Adjust the Pocket Linking Parameters as shown in Figure 5-49.

- Also set the Coolant to Flood: On.

- Generate, Backplot, and Verify. Make any adjustments required prior to continuing.

Spot Drilling Toolpath

d. From the 2D Toolpaths toolbar, choose the Drill Toolpath… icon. The Drill Point Selection dialog will display (Figure 5-50).

- Press the Mask on Arc button, carefully select the top of the four .25 diameter counterbores, and hit Enter when finished. The cutting order can be changed
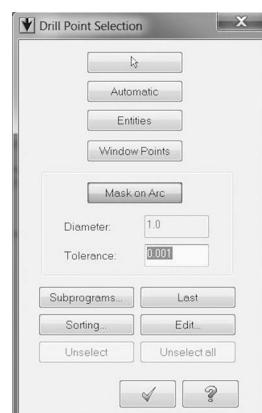
Figure 5-50 Drill Point
Selection Dialog
Courtesy CNC Software Inc.
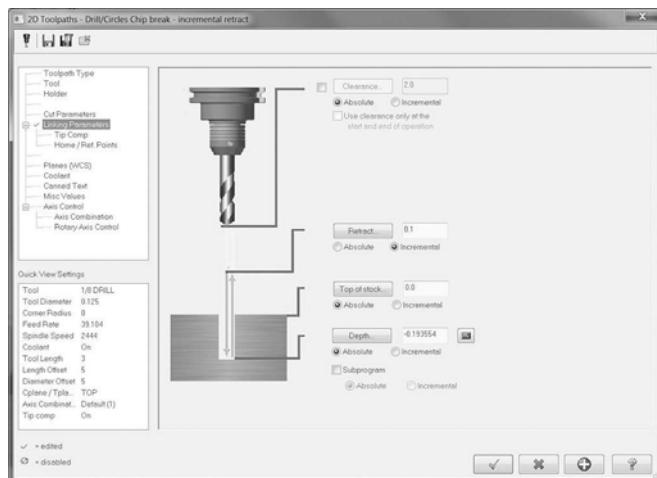
Figure 5-51 Drill Linking
Parameters
Courtesy CNC Software Inc.

by pressing the Sorting … button within the dialog. Experiment with the different settings to get the most efficient travel movement. Press OK.

- Select the 3/8 Spot drill for the tool.
- Leave the default Cut Parameters as Drill/ Counterbore with no Dwell.
- For Linking Parameters, set per Figure 5-51, and set the Depth values per Figure 5-52.

   Note: Press the calculator icon next to the Depth field to open the Depth Calculator dialog.

   Input the desired Finish diameter, then press OK to add the value to the depth.

- Also set the Coolant to Flood: On.
- Generate, Backplot, and Verify. Make any adjustments required prior to continuing.

Figure 5-52 Drill Depth
Calculator Dialog
Courtesy CNC Software Inc.

Drilling Toolpaths

e. The same four Spot-Drilled holes will be drilled with the 1/8" Drill.

- Right-click on the Drill/Counterbore path just created and select Copy from the options.
- Right-click again and select Paste from the options to paste an exact copy of the path after the last.
- Left-click to open the Parameters 2D Toolpaths—Drill/Circles Simple drill—no peck dialog.
- Change to the 1/8" Drill for the Tool.
- For Cut Parameters, change the Cycle to Chip Break and set the Peck amount to .06".
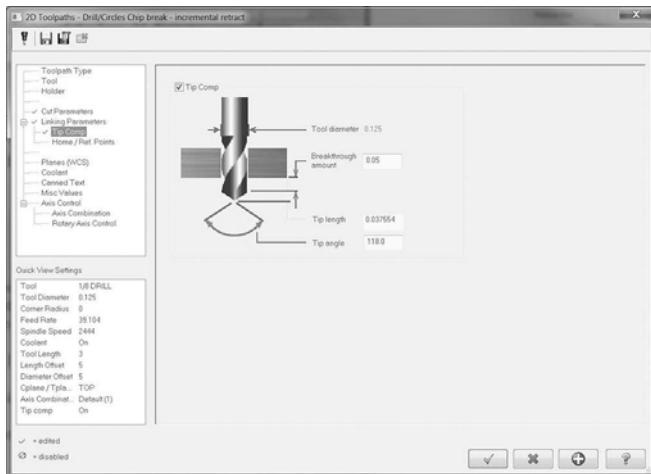
Figure 5-53 Tip Compensation
Dialog
Courtesy CNC Software Inc.

- For Linking Parameters, set the Depth to –.156".
- Activate Tip Comp and adjust the Breakthrough amount to .05" as in Figure 5-53.
- Generate, Backplot, and Verify. Make any adjustments required prior to continuing.

Counterbore Toolpath
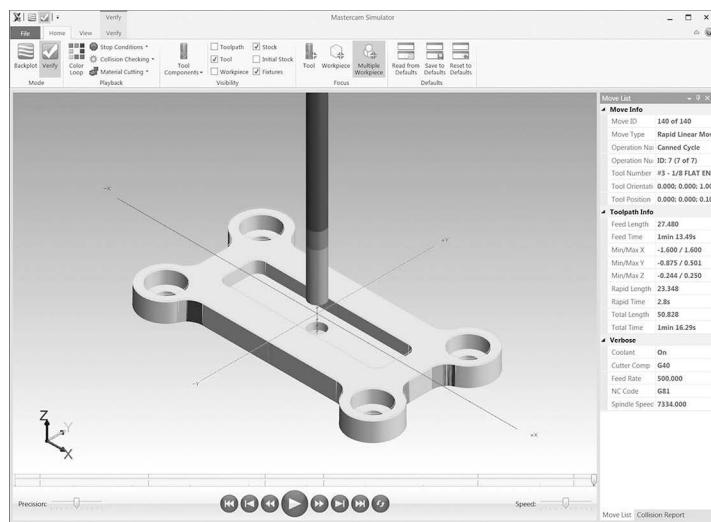
f. From the 2D Toolpaths toolbar, choose the Circ Mill Toolpath … icon. Use Mask on Arc in the Drill Point Selection dialog to select the bottoms of each of the four counterbores and press OK.

- Use Sorting to control path movements.
- Set the tool to the 1/8" Flat Endmill.
- In the Cut Parameters dialog, set the "Stock to leave on walls" and the "Stock to leave on floors" fields to zero (0).
- Also set the Coolant to Flood: On.
- Generate, Backplot, and Verify. Make any adjustments required prior to continuing.

Drill Toolpath

g. From the 2D Toolpaths toolbar, choose the Drill Toolpath… icon.

- Select the arc at the top of the .125 diameter hole at the center of the pocket and press OK.
- In the 2D Toolpaths—Drill/Circles Simple drill—no peck dialog, set the Tool to the 1/8 Flat End Mill.
- Set the Cut Parameters to Drill/Counterbore with no Dwell.
- Set the Depth to –.156.
- Activate Tip Comp and set the Breakthrough amount to .03" and the Tip angle to 180. Also set the Coolant to Flood: On
- Generate, Backplot, and Verify. Make any adjustments required prior to continuing.

Figure 5-54 Verify Display
Courtesy CNC Software Inc.

VERIFY ALL TOOLPATHS

7. Select all Operations, press the Verify icon, and press play (Figure 5-54). Adjust, if necessary, and re-verify. Confirm the result is what was expected and continue.

POSTPROCESS CNC CODE

8. Select all Operations and press the Post selected operations icon (G1). Figure 5-55 displays the posted tool paths.

TOOL LIST, SETUP SHEETS

9. The posted path does contain some of the information needed for the setup, including tool numbers and brief descriptions.

• A detailed setup sheet can be generated that includes graphic display tools, toolpaths, and cycle times. To create a setup sheet, right-click in the Operation Manager and select Setup Sheet from the options listed. Complete the General Information fields as desired. Press OK. A multi-paged report will be generated.

Figure 5-55 Mill Example Posted Paths
Courtesy CNC Software Inc.

• Multiple style options are available for the type of setup sheet output (Figure 5-56). In the Report Templates section of the Setup Sheet dialog, highlight the report listed and press F2. Select from the list. Experiment with the options and choose the one best suited to your needs.

• For a simple Tool List only, right-click in the Operations Manager and select Tool List from the options listed
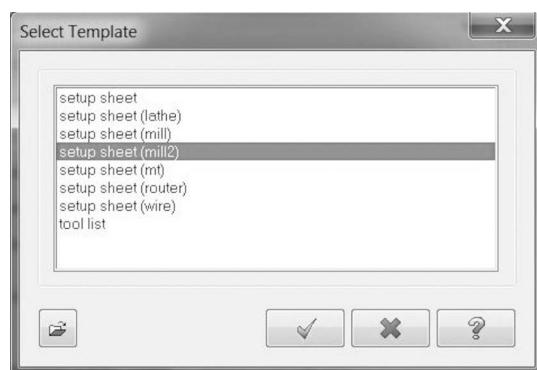
Figure 5-56 Setup Sheet Template Styles
Courtesy CNC Software Inc.

SOLID MODEL LATHE PROGRAM EXAMPLE

A solid model is commonly provided to the programmer. The steps to create the program for the turned part in Figure 5-57 are given next.

BASIC STEPS FOR A MASTERCAM LATHE PROGRAM

1. Open the file in Mastercam X8
2. Define Machine Type
3. Position the part
4. Define Stock
5. Define Tools
6. Program, Generate, Backplot, and Verify Toolpaths
7. Post
8. Reports, Setup Sheets
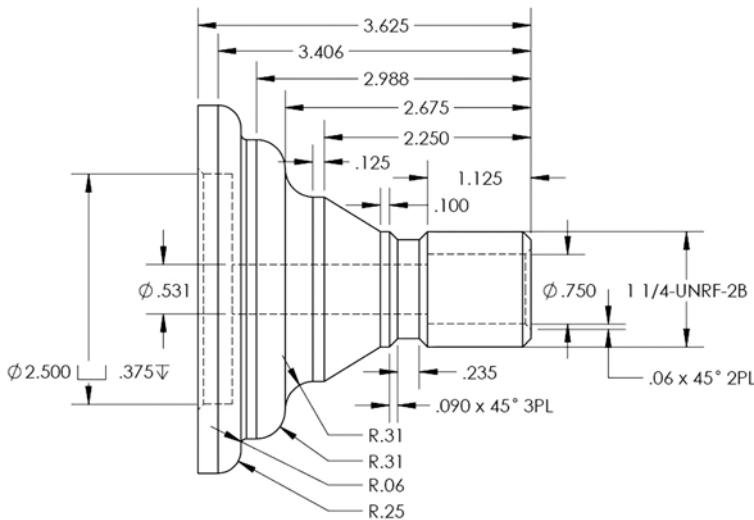
Figure 5-57 Lathe
Example Part Drawing
Courtesy CNC Software Inc.

Part 5 Computer-Aided Design and Computer-Aided Manufacturing

Open File

1. Begin by opening the file in Mastercam. File types available are those commonly used in industry (IGES, STEP, STL, etc.).

   Note: Prior to opening the file, select __SETTINGS__ from the Menu Bar, choose Configuration from the list, and Converters from the Tree. Be sure Edge Curves is checked. This will load edge curves with the Solid file and make it easier to chain for toolpaths later.

   For this example, a model created in SolidWorks is used. The file can be opened using the **FILE**, Open dialog by setting the file type to: SolidWorks Files (*.sldprt; *.sldasm*) and then selecting it. Alternatively, the file can be added by drag-and-drop from Windows explorer into the Mastercam graphics display.

Define Machine Type

2. From the Menu bar, select MACHINE TYPE—Lathe—Default. Setting the machine type will activate the two-axes lathe mode. Notice in the lower left corner of the display that the coordinate system symbol changes to D+ (diameter) and Z. Also notice the coordinate tracking fields on the Auto Cursor Ribbon Bar change to D, Z, and Y.

Position the Part

3. The part file will be oriented in the same fashion that it was created. It will commonly be necessary to translate the model into the Mastercam Work Coordinate System (WCS) frame. If it is oriented correctly, proceed to step 4.

- Identify where the part zero location will be on the part model. Typically, this will be the front finished face for the Z-axis and centerline for the X-axis for turned parts.

- Press the F9 key (again to toggle off when done) to display the System Coordinate Axes of the modeled part.

- Left-click on the Dynamic XFORM… icon: follow the Operator Prompt, Select entities to Move/Copy**.**

- Press Enter and follow the prompt Ctrl-right-click for Gnomon settings. Uncheck the box: "Switch to Manipulate geometry after initial gnomon placement" (see example in Figure 5-36) and press OK.

- Pick the model origin point and follow the prompt to manipulate the axes so that the X, Z axes align with the direction of the intended setup. Use the left-mouse-button to accept the position (see example in Figure 5-37).

- Toggle the Gnomon/Geometry switch to the Manipulate Geometry mode in the graphics display. This Manipulate Geometry function will allow the model move to align with the new settings (see example in Figure 5-38).

- On the Ribbon Bar press the Move icon, select WCS origin from the drop-down, and press the Move to origin button. Lastly, press the Align to axes button and press OK if the preview is what was expected (see example in Figure 5-39). Right-click and select clear colors from the list to remove the result color.

- The model is now placed in relation to the WCS. **FILE**, Save.

   Note: Remember to save often throughout the programming process.

Figure 5-58 Stock Setup: Lathe
Courtesy CNC Software Inc.
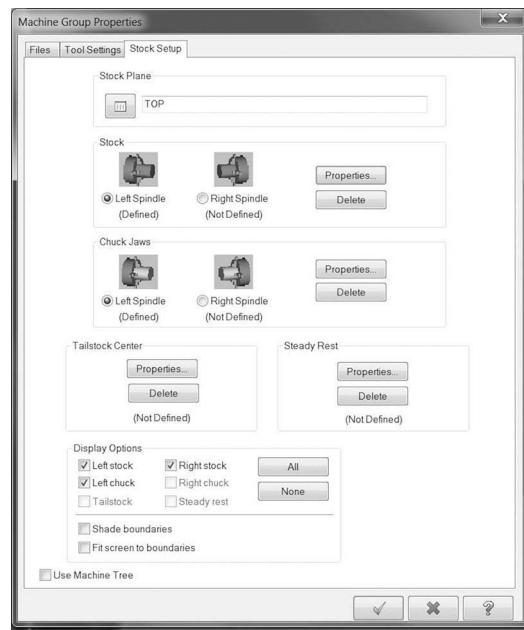
Figure 5-59 Machine Component
Manager: Stock
Courtesy CNC Software Inc.

Define the Stock

4. In the Operations Manager, expand the Properties Mill-Default to expose Stock setup and left-click to open it. Complete the required information per the blank stock dimensions, as in Figure 5-58 and Figure 5-59.

- For the OD: (X), press the Select … button to select directly from the model or enter the value, if known.

- For the Length, press the Select … button to select directly from the model or enter the value, if known.

- Press OK.

- Open the Tool Settings tab and complete the dialog per Figure 5-60.

  Note: The radio button, From Material, is set for Feed & Speed Calculations based on the part material.

- Press OK.
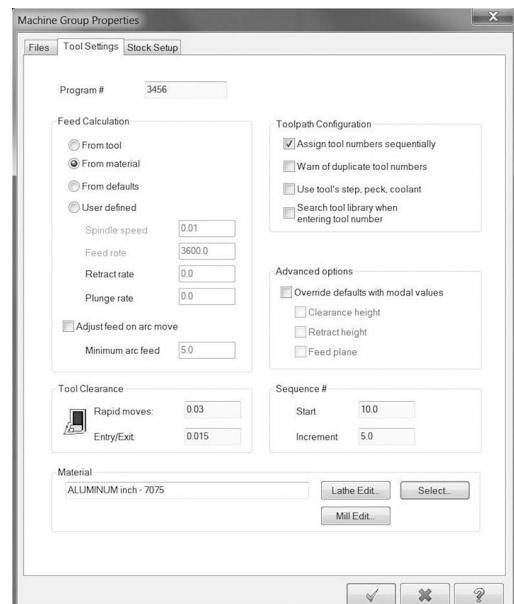
Figure 5-60 Machine Group Properties:
Tool Settings
Courtesy CNC Software Inc.

Define Tools

5. From the Menu Bar, select **TOOLPATHS**—Lathe Tool Manager …. Set the filters to minimize the displayed tools to display General Turning only and press OK (Figure 5-61). From the library list, select each tool determined (based on the part geometry) and double-click to add to the used list (Figure 5-62). Press OK when all tools are selected.

Program Toolpaths

6. Right-click in the empty space of the tool bar area and select Load Workspace. From the list, select the Lathe. The Tool bar (Figure 5-63) will be placed to the left of the display area.
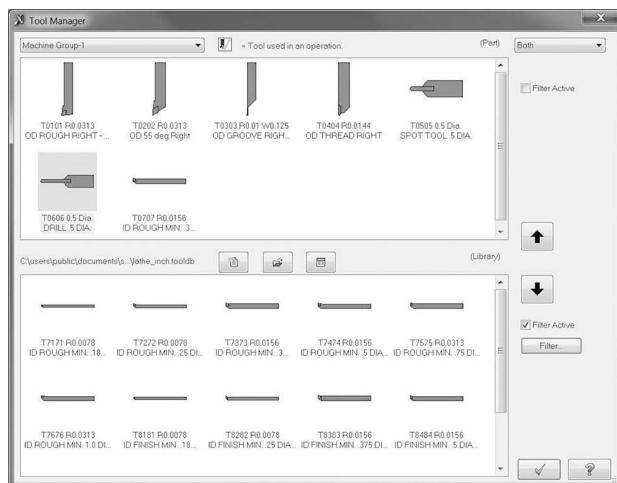
Figure 5-61 Lathe Tool
Manager Dialog
Courtesy CNC Software Inc.
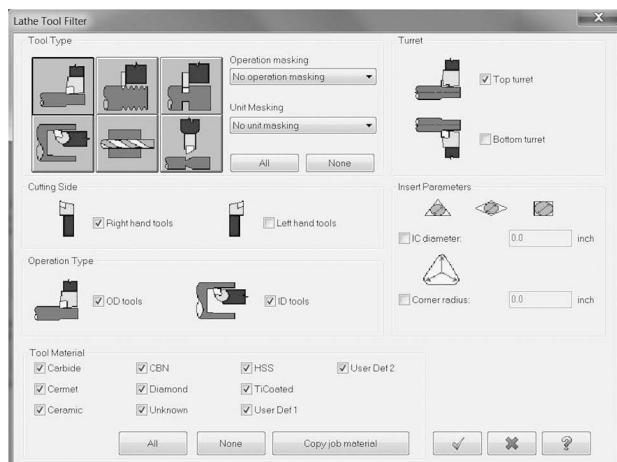
Figure 5-62 Lathe
Tool Filter
Courtesy CNC Software Inc.

Figure 5-63 Lathe Toolpaths Toolbar
Courtesy CNC Software Inc.

Part 5 Computer-Aided Design and Computer-Aided Manufacturing

As each toolpath is completed, you should Generate and Backplot to verify tool-paths and make any required adjustments.

Facing Toolpath

a. From the Lathe Toolpaths toolbar, choose the Create Lathe Face Toolpath… icon. Accept the path name or change as desired and press OK.

• In the Lathe Face Properties dialog, toolpath parameters tab, press the Coolant button and set Flood to On and Press OK.

• From the tools displayed, select the OD Rough Right from the tool list (Figure 5-64).

• In the Face parameters tab press the Finish Z … change the value in the field to 0.0 (Figure 5-65) and Press OK.
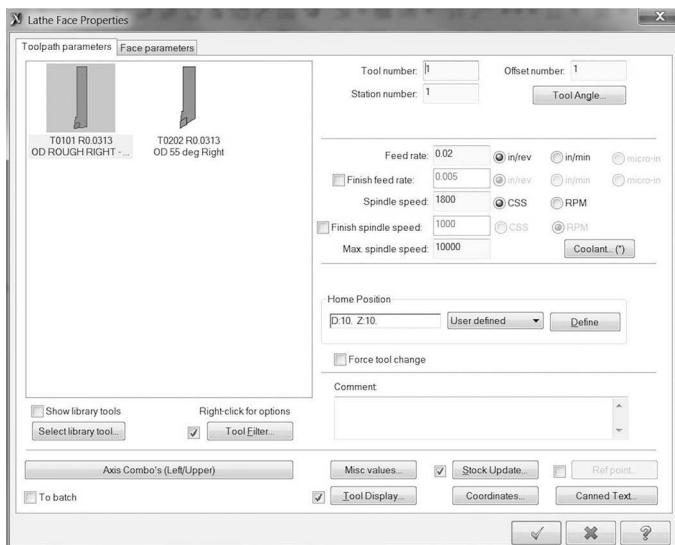
Figure 5-64 Lathe Face Properties Dialog
Courtesy CNC Software Inc.
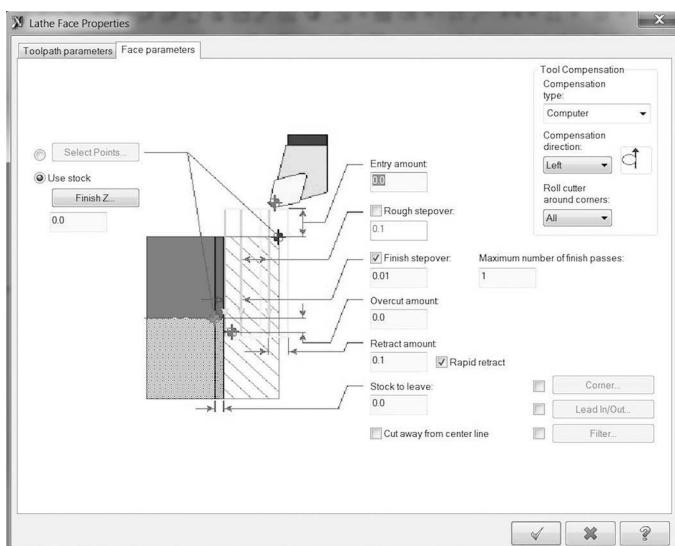
Figure 5-65 Lathe Face Parameters
Courtesy CNC Software Inc.

- Backplot to verify the toolpath results are correct. Make any adjustments required prior to continuing.

- Key the letter T to erase the toolpath display and again to redisplay.

Lathe OD Rough Toolpath

b. From the Lathe Toolpaths toolbar, choose the Create Lathe Canned Rough Toolpath … icon. Set the Chaining dialog to Solids.

- From the Chaining dialog, select the Solids button.

- Follow the prompt Partial chain: select the first entity. Pick the chamfer at the beginning of the threaded portion of the OD. The start point of the chain and the direction of tool travel will display on the model. Use the Forward/Back buttons to change the start point and the Reverse button to switch the direction if needed.

- Follow the prompt to select the last entity in the chain, as shown in Figure 5-66.

- Press OK to continue.

- In the Lathe Canned Rough Properties dialog, select the same OD Rough tool (T0101) and settings as used in the Facing toolpath.

- Press the Coolant button, set Flood to On, and Press OK.

- For the Canned rough parameters tab, change the entries to match Figure 5-67.
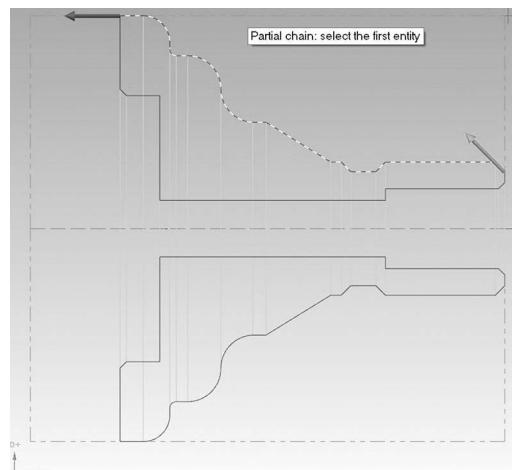
Figure 5-66 Lathe Rough OD Chain
Courtesy CNC Software Inc.
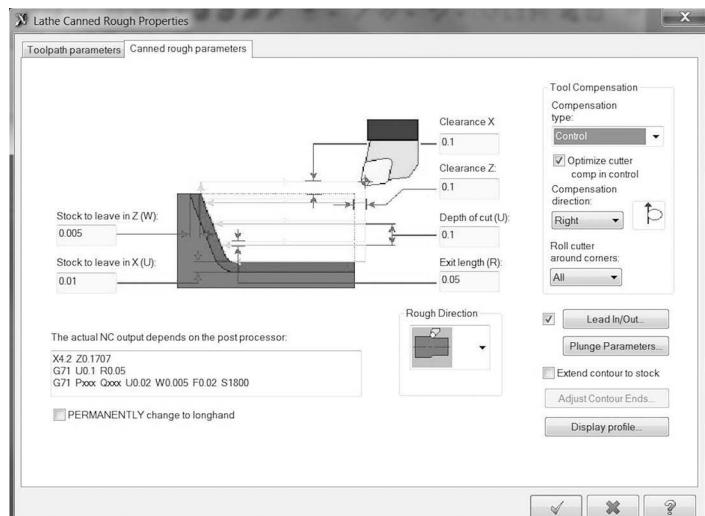
Figure 5-67 Lathe Canned
Rough Parameters
Courtesy CNC Software Inc.

- Set the Compensation type to: Control and the Compensation direction to: Right. Setting these this will output G42 to the machine code, allowing for adjustments via the Offset page at the machine controller for the finish path.

- Press OK. The Rough OD toolpath will process and the result will be displayed, as seen in Figure 5-68.

- For Backplot, activate the Quick verify mode (simulates metal removal) by pressing the icon in the Backplot dialog. Make any adjustments required prior to continuing.
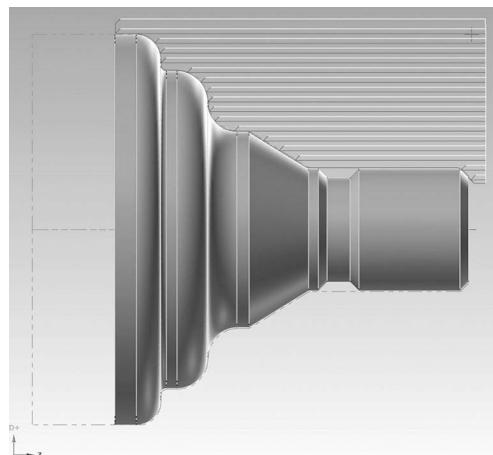
Figure 5-68 Rough OD Toolpath Display
Courtesy CNC Software Inc.

Lathe OD Finish Toolpath

c. From the Lathe Toolpaths toolbar, choose the Create Lathe Canned Finish Toolpath … icon.

- In the Lathe Quick Finish Properties dialog, set the Quick tool parameters tab to the OD 55-degree tool (T0202).

- Press the Coolant button, set Flood to On, and Press OK.

- In the Canned finish parameters tab, ensure the settings match those in Figure 5-69.
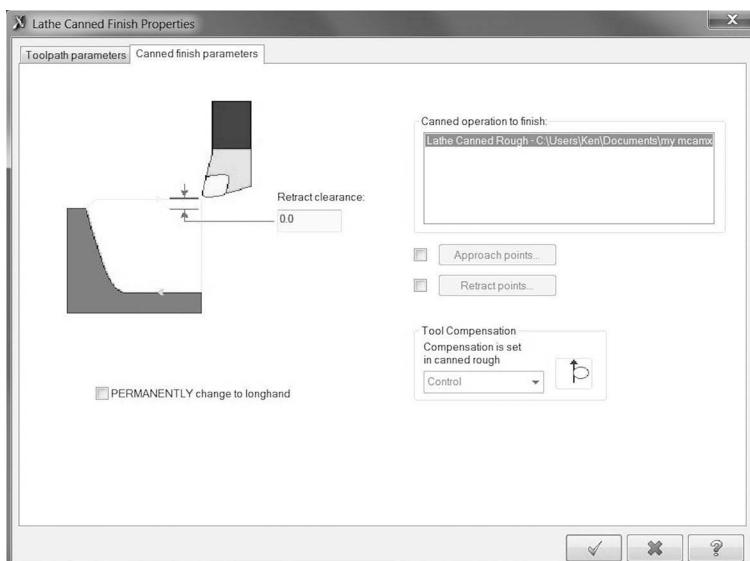
Figure 5-69 Canned Finish Parameters
Courtesy CNC Software Inc.

Note: The Canned operation to finish section in the dialog references the roughing path just completed, including the Tool Compensation.

- Backplot and Quick verify. Make any adjustments required prior to continuing.

OD Grooving Toolpath

d. From the Lathe Toolpaths toolbar, choose the Create Lathe Groove Toolpath … icon.

- Set the Grooving Options dialog box radio button to Chain and press OK.

- Follow the prompt, Partial chain: select the first entity. Pick the right end of the groove portion of the OD. The start point of the chain and the direction of tool travel will display on the model. Use the Forward/Back buttons to change the start point and the Reverse button to switch the direction, if needed.

- Follow the prompt to select the last entity in the chain, as shown in Figure 5-70.

- Press OK to continue. The Lathe Groove (Chain) Properties dialog will display.

- Because the needed tools were selected at the beginning of programming within the Tool Manager, the only grooving tool displayed for the Toolpath parameters tab will be T0303.

Figure 5-70 Groove Chain
Courtesy CNC Software Inc.

- Set the Coolant to Flood: On. Press OK.

- For Groove shape parameters tab, leave the default settings as is.

- Set the Groove rough parameters tab, as shown in Figure 5-71.

- Set the Groove finish parameters tab, as shown in Figure 5-72.

- Backplot and Quick verify. Make any adjustments required prior to continuing.

OD Thread Toolpath

e. From the Lathe Toolpaths toolbar, choose the Create Lathe Thread Toolpath … icon.

- In the Lathe Thread Properties dialog, the Toolpath parameters tab will display only the OD Thread Right tool (T0404).

- Set the Coolant to Flood: On. Press OK.

- For the Thread shape parameters tab, press the End Position … button (Figure 5-73); on the model, select the bottom front end of the groove chamfer.
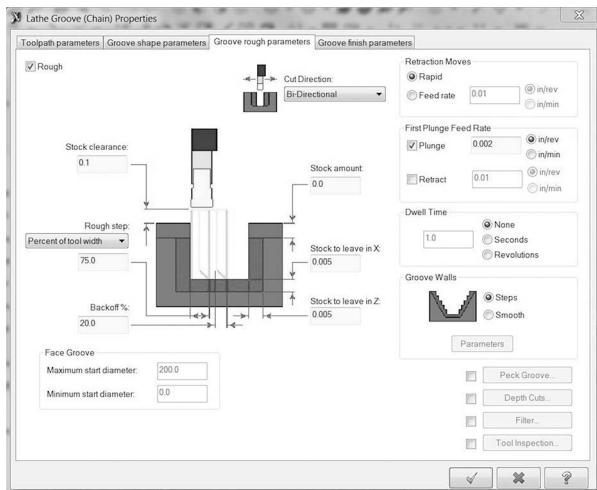
Figure 5-71 Groove Rough Parameters
Courtesy CNC Software Inc.
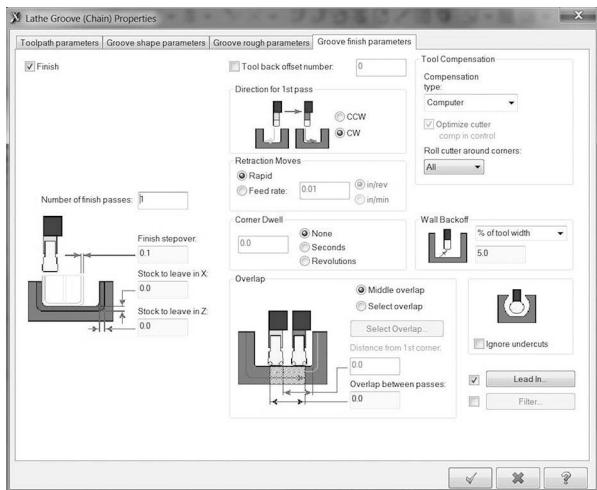
Figure 5-72 Groove Finish Parameters
Courtesy CNC Software Inc.
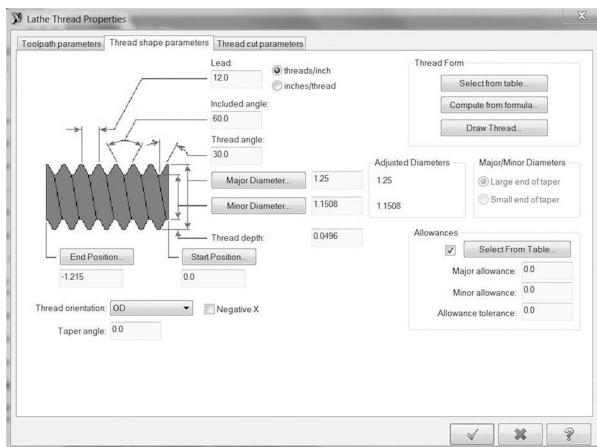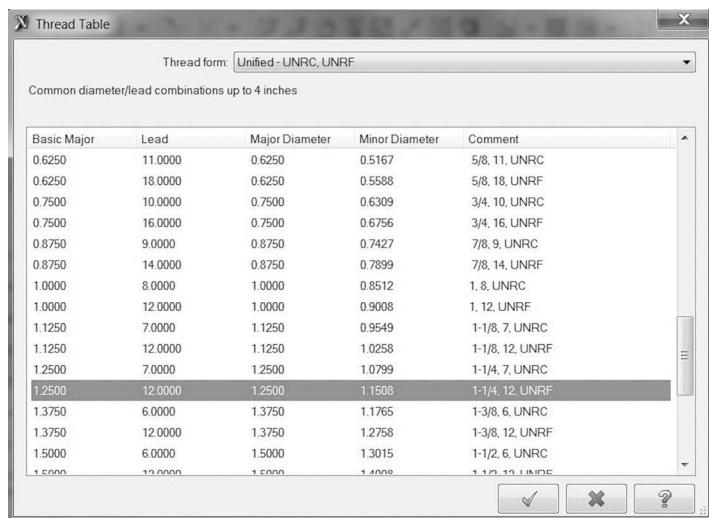
Figure 5-73 Thread Shape Parameters
Courtesy CNC Software Inc.

Figure 5-74 Thread Table
Courtesy CNC Software Inc.

- Press the Select from table … button in the Thread Form section of the dialog.
- For the Thread cut parameters tab, set the values to match Figure 5-74.
- Backplot and Quick verify. Make any adjustments required prior to continuing.

Spot Drill Toolpath

f. From the Lathe Toolpaths toolbar, choose the Create Drill Toolpath … icon.

- In the Lathe Drill Properties dialog, select the Spot Tool .5 diameter, (T0505), from the tools listed.
- Set the Coolant to Flood: On. Press OK.
- Select the Simple drill—no peck tab and adjust the settings to match Figure 5-75. Use the Calculator button next to the Depth field to access the Depth Calculator
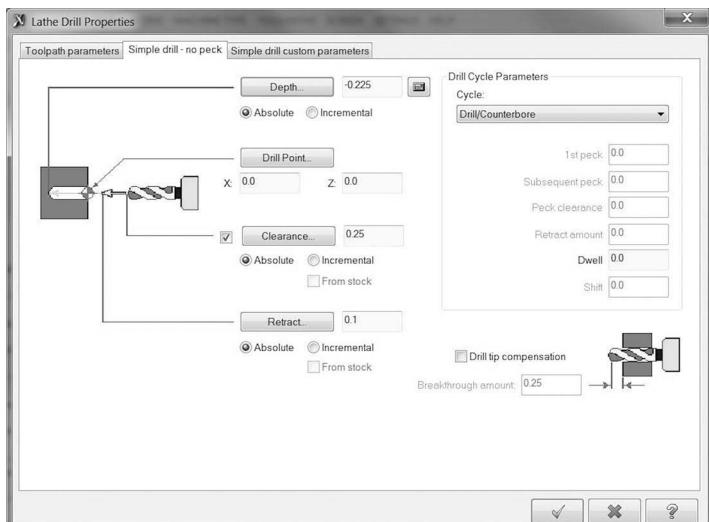
Figure 5-75 Simple Drill Parameters
Courtesy CNC Software Inc.

dialog. Set the desired Finish diameter to: .45 and the radio button to: Add to depth and Press OK.

- Press OK.
- Backplot and Quick verify. Make any adjustments required prior to continuing.

Drilling Toolpath

g. From the Lathe Toolpaths toolbar, choose the Create Drill Toolpath … icon.

- In the Lathe Drill Properties dialog, select the Drill .5 Dia., (T0606), from the tools listed.
- Double-click on the listed tool to open the Define Tool—"Machine Group-1" dialog (Figure 5-76) to set the tool data for the 17/32 diameter drill required.
- Set the Coolant to Flood: On. Press OK.
- Select the Simple drill—no peck tab and change the Cycle to Peck Drill in the Drill Cycle Parameters section of the dialog.
- Set the First peck value as 66% of the drill diameter.
- Set the Subsequent peck amount to .3 and the Peck Clearance to .1.
- Activate the Drill tip compensation with the checkbox and set the Breakthrough amount to .15.
- Use Depth … button to select the bottom of the drilled hole from the model.
- Press OK when all settings match Figure 5-77.
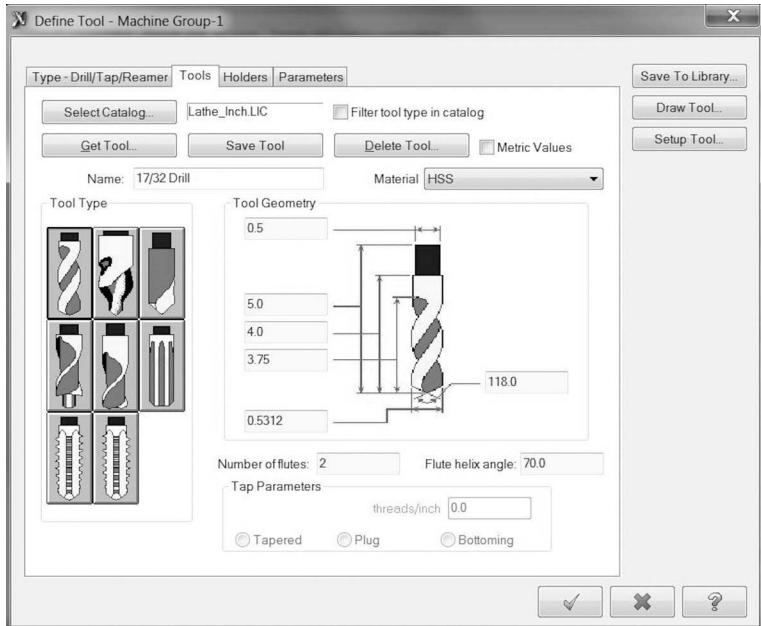
Figure 5-76 Define Tool Dialog
Courtesy CNC Software Inc.
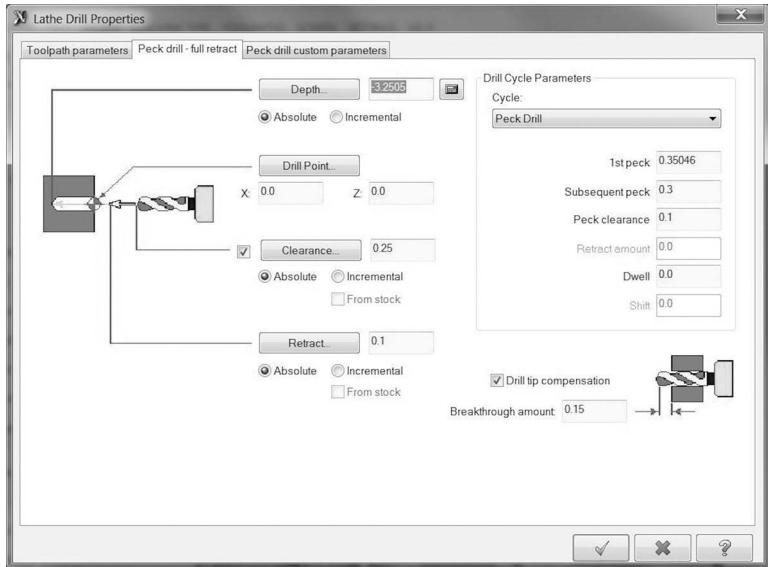
Figure 5-77 Drill Peck Parameters
Courtesy CNC Software Inc.

Note: if the tool extension is insufficient, it may be necessary to adjust the overall length in the Holder Geometry from the Holders tab of the Define Tool—Machine Group dialog.

- Press OK.
- Backplot and Quick verify. Make any adjustments required prior to continuing.

Bore Rough Toolpath

h. From the Lathe Toolpaths toolbar, choose the Create Lathe Canned Rough Toolpath … icon.

- Follow the prompt, Partial chain: select the first entity. Pick the chamfer at the beginning of the bore I.D. The start point of the chain and the direction of tool travel will display on the model. Use the Forward/Back buttons to change the start point and the Reverse button to switch the direction, if needed.

- Follow the prompt to select the last entity in the chain, as shown in Figure 5-78.

- Press OK to continue.

- In the Lathe Canned Rough Properties dialog on the Toolpath parameters tab, select the ID Rough tool (T0707).
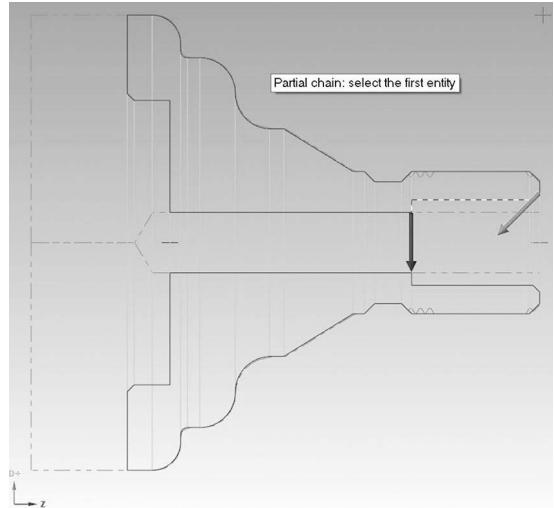
Figure 5-78 Bore Chain
Courtesy CNC Software Inc.
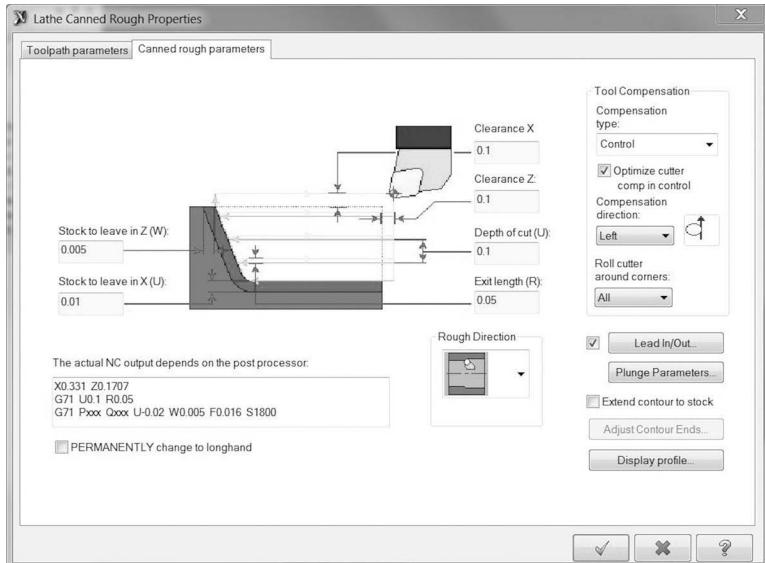
Figure 5-79 Bore
Rough Parameters
Courtesy CNC Software Inc.

- Press the Coolant button, set Flood to On, and Press OK.
- For the Canned rough parameters tab, change the entries to match Figure 5-79.
- Press OK. The Rough ID toolpath will process and the result will be displayed.
- For Backplot, activate the Quick verify mode (simulates metal removal) by pressing the icon in the Backplot dialog. Make any adjustments required prior to continuing.

Bore Finish Toolpath

i. From the Lathe Toolpaths toolbar, choose the Create Lathe Canned Finish Toolpath … icon.
- In the Lathe Canned Finish Properties dialog, select the ID Rough tool (T0707).
- Press the Coolant button and set Flood to On and Press OK.
- On the Canned Finish Parameter tab, select the second path in the list of the dialog section: Canned operation to the finish.

Note: The Tool Compensation direction is determined by the settings in the rough path.

- Press OK to continue.
- Press OK. The Rough ID toolpath will process and the result will be displayed.
- Backplot the finish bore operation. Make any adjustments required prior to continuing.
- Select all paths and Backplot the entire program.
- Make any final adjustments and save.

Postprocess Lathe CNC Code

7. Select all Operations; press the Post selected operations icon (G1). Figure 5-80 displays the posted paths for the Lathe Example.

Tool List, Setup Sheets

8. The posted path does contain some of the information needed for the setup including tool numbers and brief descriptions.

• A detailed setup sheet can be generated that includes graphic display tools, toolpaths, and cycle times. To create a setup sheet, right-click in the Operation

Figure 5-80 Lathe Example
Posted Paths
Courtesy CNC Software Inc.

TOOL LIST			1/17/2015	3:20 PM
OPERATIONS MANAGER			SORTED:	NO
#	Tool Type	Tool Name	Insert Info	
T0101	General Turning Tool	OD ROUGH RIGHT - 80 DEG.	R0.0313	
T0202	General Turning Tool	OD 55 deg Right	R0.0313	
T0303	Grooving Tool	OD GROOVE RIGHT - NARROW	R0.01 W0.125	
T0404	Threading Tool	OD THREAD RIGHT	R0.0144	
T0505	Drilling Tool	SPOT TOOL .5 DIA.	0.5 Dia.	
T0606	Drilling Tool	DRILL .5 DIA.	0.5312 Dia.	
T0707	Boring Tool	ID ROUGH MIN. .375 DIA. - 75 DEG.	R0.0156	

Figure 5-81 Lathe Example Tool List
Courtesy CNC Software Inc.

Manager and select Setup Sheet from the options listed. Complete the General Information fields as desired. Press OK. A multi-paged report will be generated.

- There are multiple options for the style of setup sheet output. In the Report Templates section of the Setup Sheet dialog, highlight the report listed and press F2. Select from the list. Experiment with the options and choose the one best suited to your needs.

- For a simple Tool List only, right-click in the Operations Manager and select Tool List from the options listed (Figure 5-81).

SUMMARY

In this section, the most prevalent method for programming using CAD/CAM was introduced. Mastercam X8 is widely used in shops and is one of many software programs available to meet the CNC Programming task. The skill gained using one CAD/CAM program is easily applied to similar programs. Solid Models and electronic data are commonly transferred from the customer to the shop to aid in the process. Apply the knowledge you have learned within the workplace and you will be rewarded with the output of successful CNC programs and quality parts more quickly.

Part 5 Study Questions

1. The acronym CAD/CAM stands for Computer-Aided Design and Computer-Aided Manufacturing.

       T or F

2. Toolbars can be docked, undocked, docked vertically, docked horizontally, or turned off completely.

       T or F

3. The Machine Group is displayed in the

       a. Graphic display area

       b. Status Bar

       c. Verify dialog

       d. Operations Manager

4. What is a fillet?

5. How is the fillet command accessed within Mastercam?

6. What is a chamfer?

7. How is the chamfer command accessed within Mastercam?

8. Geometry creation consists of

       a. CAD

       b. Lines, Arcs, Points, etc.

       c. Both a and b

       d. Toolpath creation and Post Processing

Part 5 Computer-Aided Design and Computer-Aided Manufacturing

9. Where do you establish the stock size for your part drawing?

       a. From the Machine Type menu item

       b. Toolpath

       c. Machine Group Properties—Stock Setup

       d. Operations Manager

10. Parameters specific to the toolpath can be adjusted by accessing the

       a. Contour (2D) dialog

       b. Operations Manager

       c. Tool Parameters

       d. XForm dialog

11. The general steps to creating a Mastercam program for a CNC Machine are selection of the Machine Type, Stock Setup, Design, Backplot, Verification, and Posting.

       T or F

12. To create a line by inputting the coordinates for its endpoints, how do you enter the "Fast Point" mode?

       a. Press the space bar.

       b. Mouse click in the coordinate fields in the Auto Cursor Ribbon bar.

       c. Press the letter X, Y, or Z and enter the values for each, separated by a comma.

       d. All of the above are methods.

13. What is the function of the AutoCursor?

14. Describe the meaning of Compensation in computers when referring to cutter compensation?

Part 5 Computer-Aided Design and Computer-Aided Manufacturing

15. As part geometry is created, lines sometimes cross over each other, forming an intersection and extending beyond. How do you remove these line extensions to make a clean corner?

        a. Fillet set to zero radius

        b. Chamfer

        c. Delete

        d. Trim/Break

16. When chaining part geometry, the direction the chaining arrow points determines whether the tool will travel on the left or right of the selected line.

        T or F

17. A line can be created from polar coordinates if the _____ function button is selected.

        a. <u>A</u>ngle

        b. <u>S</u>ketcher

        c. <u>T</u>angent

        d. <u>M</u>ulti-line

18. The Most Recently Used (MRU) toolbar stores the last several commands that you have accessed. You can use these buttons to reactivate any command on the MRU button bar.

        T or F

19. Why would Compensation Type be set to Control in tool parameters?

20. Describe the method used to position the model within the Work Coordinate System in Mastercam when opening a solid model file from another CAD program.

PART 6

Introduction to Feature-Based Machining
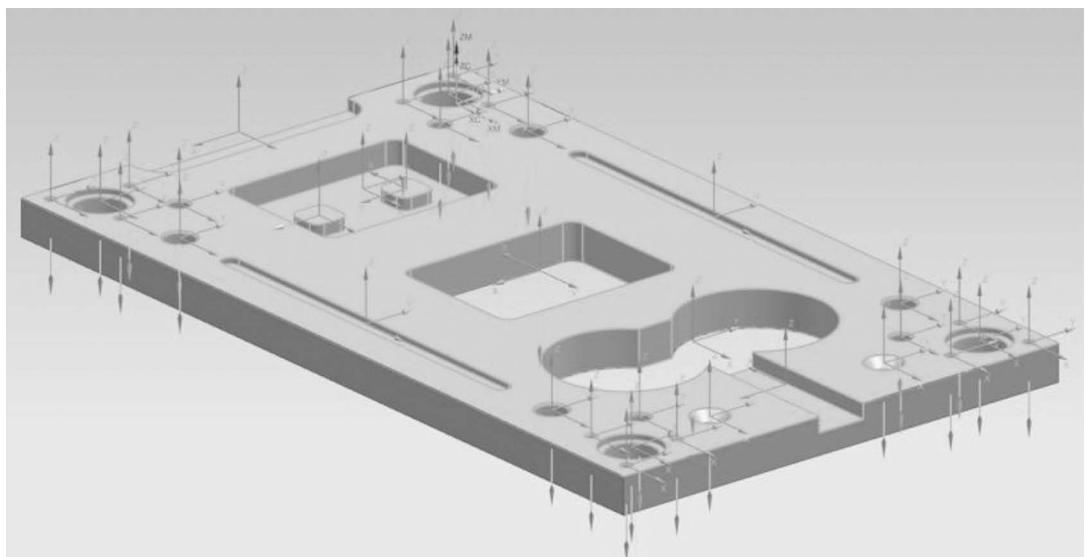
Figure 6-1 NX CAM 9.0 Feature-Based Machining
Courtesy Siemens PLM

Part 6 Introduction to Feature-Based Machining

OBJECTIVES:

1. Become familiar with Feature-Based Machining (FBM) best uses.
2. Identify procedures in and use the User Interface for FBM.
3. Use terminology specific to FBM.
4. Practice steps on how to use NX CAM Feature-Based Machining to create program code.
5. Use FBM-created program for CNC machine setup instructions.

DEFINITION

FBM is the automated process of defining machining operations from modeled features based on best practices that are established in a Machining Knowledge Library. The customizable system uses Rules that control the selection of tools and cutting parameters, and take dimensional tolerances into consideration.

WHY USE FBM?

The automation of CNC programming significantly minimizes time-consuming manual programming efforts and reduces repetitive tasks. The result is faster program development by reducing the number of steps to complete programs and output CNC program code. By limiting the chance for input and selection errors during programming, increased program accuracy is attained. FBM promotes the use of standardized programming and machining best practices. FBM is especially well-suited for programming families of parts with similar features. Primary uses are hole-making, simple repetitive and redundant tasks, 2½-D milling, drilling, reaming, and tapping.

FBM TERMS

- Machine Coordinate System (MCS)
- Machining Center Tool (MCT)
- In-Process Workpiece (IPW)
- User Defined Features (UDF)
- Machining Knowledge Library (MKL)

  This is the database where machining processes are defined.

- Machining Knowledge Editor (MKE)

  This is the database where the rules for process definition can be edited and saved by those with proper licensure and administrative rights.

- Product Manufacturing Information (PMI)

  Annotations in the part design that are considered automatically during process creation such as tolerance holes or surface finishes.

MODELING BASICS

Programming from solid models represents the current method of choice for design to manufacture. It has great power and flexibility to aid the implementation of the model to finished product concept. The designer has freedom to modify designs with ease

and all related dimensional characteristics will automatically be updated to include the changes (associative). This carries over to the CAM program as well. Once the adjusted file is opened after any design changes, the program recognizes that changes have been made to the original; it then asks the user if the existing file should be updated to reflect these changes. The software for solid modeling is intuitive and user friendly. Machinists, programmers, and design engineers learn to use its power fairly quickly.

Solid models are feature-based parametric designs. A feature is any entity of a part that has volume or area. Some basic features are holes, pockets, steps, notches, and surfaces. Parametric means that the entity has parameters that define its shape and size. For instance, a prismatic part may have a length of 3.0 inches, a width of 2.0 inches, and a height of 1.0 inches. These values are parameters that can be adjusted, resulting in a similar prismatic part.

A properly constructed solid model is required for the efficient use of FBM. Features must be "water tight", meaning there must be no holes or gaps in the solid.

The machine tool you are programming has specific physical characteristics such as tooling pockets. It is important to establish and match this fact within the setup for programming. There may also be unique tools within your shop. Create or retrieve these tools into NX and make them available as options to those generated by NX FBM.

When the design model is imported into the working NX environment, it will be necessary to establish the Machine Coordinate System (MCS), place the workpiece in relation to it, and identify the part blank and any fixtures.

OVERVIEW OF THE NX CAM USER INTERFACE

When using any software, the first thing users have to get accustomed to is the User Interface (UI). This is the driver's seat from which you will create your product designs, manipulate the Solid Model files, create toolpaths, and post-process CNC code for manufacturing the part. The more comfortable you are with the user interface, the faster and more accurately your programming output will be.

Some brief descriptions are given below for the NX CAM 9.0 User Interface (Figure 6-2).
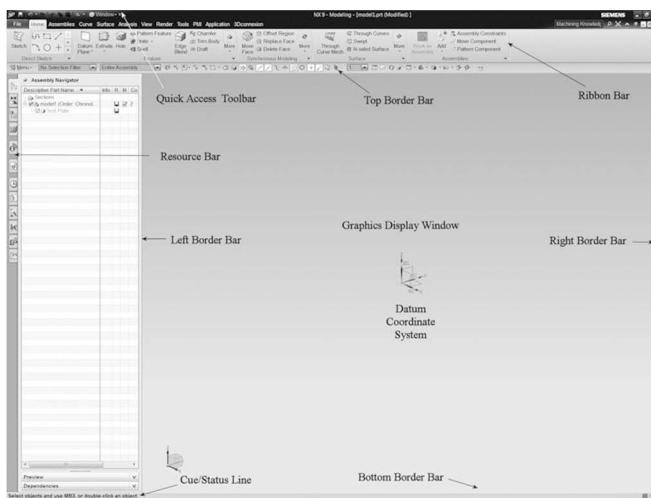
Figure 6-2 NX CAM 9.0
User Interface
Courtesy Siemens PLM

Part 6 Introduction to Feature-Based Machining
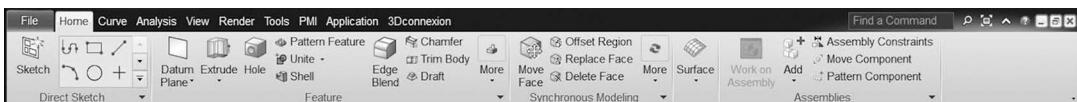
Figure 6-3 Ribbon Bar
Courtesy Siemens PLM

Graphics Display Window

The graphics display window is the large open area at the center of the interface where the part model is displayed when created, imported, or opened. By default, the screen color is a graduated grey to a very light grey, but it can be changed to any color desired by selecting File tab of the Ribbon Bar, then All Preferences, then Background from the fly-out menu. Use the Edit Background dialog to make the changes you prefer.

Datum Coordinate System

The Datum Coordinate System is located in the center of the window and represents the Absolute Origin for the model.

Ribbon Bar

The Ribbon Bar is located above the Graphics Display Window and consists of tabs that contain common commands for groups such as Direct Sketch, Feature, Synchronous Modeling, and Assemblies for the Home Tab. Tabs can be accessed by left-clicking or by placing the cursor over the open space of the Ribbon Bar, then using the MB2 to scroll through the options. Another powerful feature is the Command Finder in the upper right section of the display. Simply type in a word that represents the command you want to find and the Command Finder dialog-box will display a list of matched options. Choose from the list by left-clicking to start the command or use the down arrow to select from the list of options to place the command on the Border Bar, Ribbon Bar, or Quick Access Toolbar. You can also right-click in the open space of the Ribbon Bar to customize it to your desired functions. To save any changes to the user interface, select Roles from the Resource Bar, right-click in the white-space under the User section, and select New User Role from the list. Complete the Role Properties dialog-box and press OK. Each time you return to an NX session the Role last active will be displayed.

Resource Bar, Navigator Windows

The Resource Bar located on the left border (by default) contains tabs for Navigators such as Assembly, Constraints, Parts, and Roles for Modeling. In the following example, we will focus our attention on the navigation tabs specific to manufacturing: Operation Navigator and Machining Feature Navigator.

Quick Access Tool Bar

The Quick Access Tool Bar is located in the Top left of the display and allows access to commonly used commands in NX such as: Save; Undo; Cut; Copy; Past and Repeat Last Command by default. The tool bar can be customized to meet your needs.

Top, Bottom, Right, and Bottom Border Bars

The Top Border Bar is active by default. It contains the Menu drop-down, Selection, and View tools. Commands can be added to the Top, Right, Bottom, or Left Border Bars to enhance user functionality as needed. Remember to save any changes to the role as described above.

Cue/Status Line

The Cue/Status Line is present in the lower left-hand corner of the display beneath the Bottom Border Bar, if present. User cues relevant to needed actions are displayed here.

CONVENTIONS

In the example that follows, there will be descriptive graphic figures of the User Interface. Specific directions will be included in the numbered steps.

GENERAL STEPS FOR FEATURE-BASED PROGRAMMING

1. Specify the Part and the raw material Blank.
2. Use Feature Finder to find parametric features.
3. Create Feature Processes and Operations.
4. Generate Toolpaths.
5. Review and Edit as needed.
6. Verify Toolpaths.
7. Find Features again for any UDFs.
8. Repeat above steps.
9. Post Process.
10. Complete Setup Documentation.
11. Release to production.
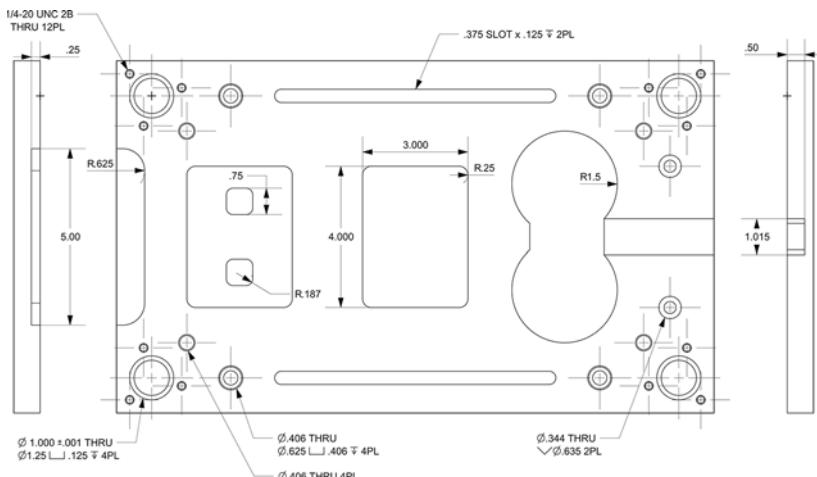
Figure 6-4 Test-Plate Design Drawing
Courtesy Siemens PLM

The following step-by-step process is an example using some of the programming capabilities of NX CAM 9.0 Feature-Based Machining. The goal is to create the program as efficiently and accurately as possible. FBM will output suggested operations based on the information in the Machining Knowledge Library. The operations may be edited to meet specific needs. For the sake of detailed presentation, the steps listed in this example may differ slightly from the steps you use in the end. The model we will use is shown in Figure 6-4.

NX CAM 9.0 STEPS FOR FEATURE BASED PROGRAMMING

A. Complete the Design Model and save it in NX CAM 9.0. If the model is not native to NX, add the model via the Add Component icon, then place it using the Move Component icon on the Ribbon Bar. Enter the Manufacturing application via the File Tab or use the keyboard shortcut, Ctrl+Alt+M.

B. Select the Machining Environment from the list in the dialog box as seen in Figure 6-5 and press OK. The Program Group Folders (placeholders for the upcoming operations) will be created and listed in the Operation Navigator in the Program Order view (Figure 6-6). Later, we will copy and rename these folders specifically for this project. As processes are created, they will be placed either in the applicable folder for tool-path generation or in the Unused Items section for your subsequent action.

CUTTING TOOLS

C. Enter the Machine Tool View of the Operation Navigator by pressing the icon in the Top Border Bar or right-clicking in the white space of the Operation Navigator and choose Machine Tool from the list. From the Ribbon Bar, press the Create Tool icon to begin identifying the Machining Center Tool (MCT) pockets so they correspond with the machine pockets (Figure 6-7).

Figure 6-5 Machining Environment
Courtesy Siemens PLM
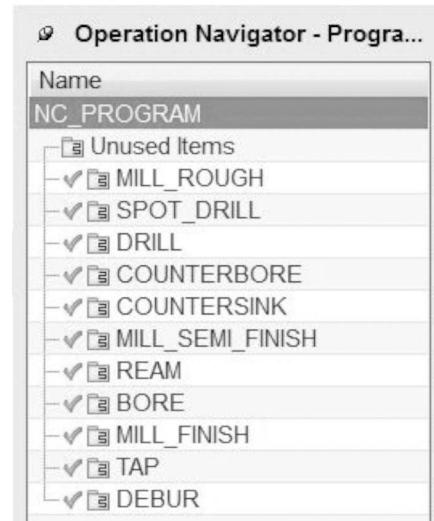
Figure 6-6 Operation Navigator, Program Groups
Courtesy Siemens PLM

As you proceed, NX FBM will develop tools based on the largest diameter possible within the dimensional limits of the feature, including the required depth of the machined feature. The tools will be placed in available Machine Tool Pockets. The tools developed can be changed to meet specific needs, if desired (Figure 6-8). Then create any cutting tools unique to the part being machined or retrieve them from the Library and place them in the Unused Items folder for now.

SPECIFY THE PART AND BLANK

D. From the Manufacturing Application, access the Geometry View of the Operation Navigator. Double click on the Workpiece under the MCS to specify the Part and the Blank (adding material to the Top surface that will be machined to establish the zero stock surfaces).

Identify any fixture components that could cause toolpath interference by pressing the Specify Check icon and selecting the components.

Establish the part material by pressing the wrench (edit) icon in the dialog box and then selecting from the list of materials. Press OK.

Double click on the MCS in the Operation Navigator to adjust the placement representative of your part zero, if necessary; then set the Safe Clearance Distance to an appropriate value.

FIND FEATURES

E. Select the Machining Feature Navigator from the Resource Bar (Figure 6-9).

Pick the Find Features icon from the Ribbon Bar, as shown in Figure 6-10. Alternatively, you can right-click in the white space of the Feature Navigator window and select Find Features from the list. Use Parametric Recognition for any solid

Figure 6-7 Machining Center Tool Pockets
Courtesy Siemens PLM
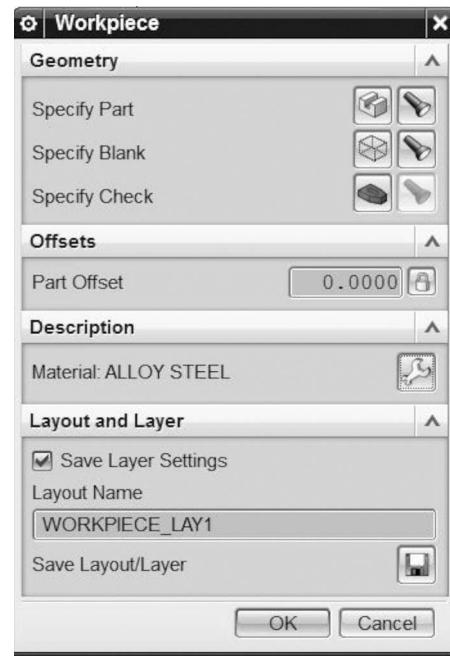
Figure 6-8 Workpiece Dialog
Courtesy Siemens PLM

Figure 6-9 Resource Bar
Courtesy Siemens PLM

model, including those created outside NX. Set the Search Method to Workpiece, as identified in the Geometry view of your setup. You can filter what Features to Recognize by selecting/deselecting those from the list. For instance, notice that Turning and Wire EDM are unchecked. It may also be helpful to limit the search to find features from only the specified vector. Experiment with the other settings and choose the one best suited to your needs. Press the Find Features Icon in the lower right corner of the dialog box below the

Figure 6-10 Find
Features Icon
Courtesy Siemens PLM

Recognized Features section (Figure 6-11). Review the findings by left-clicking on them in the Recognized Features section of the Find Features dialog-box. Delete any extraneous findings. Press OK and the list will populate in the Machining Feature Navigator—Features (Figure 6-12).

CREATE FEATURE PROCESS

Pressing the Create Feature Process icon accesses the Machining Knowledge Library (MKL) and uses pre-configured best practices to develop the operation process. This library file can be edited using the Machining Knowledge Editor (MKE) and saved with your company's own best practices by a senior programmer (Knowledge Engineer) who has full administrative rights with the proper Licensing.
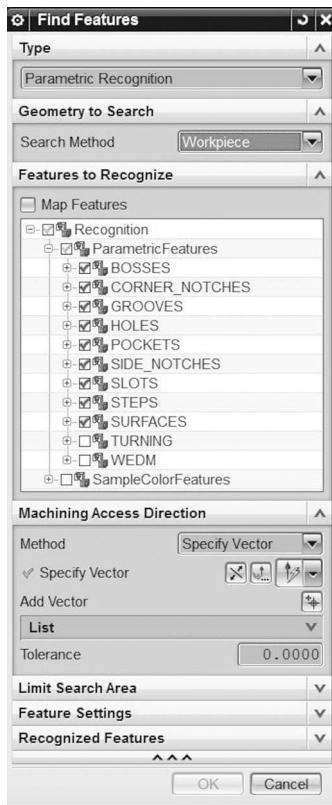
Figure 6-11 Find Features Dialog
Courtesy Siemens PLM

Figure 6-12 Machining Feature Navigator,
Features Recognized
Courtesy Siemens PLM

F. First individually select SURFACE_PLANAR_RECTANGULAR_34 from the list and press the Create Feature Process icon from the Ribbon Bar, as shown in Figure 6-13.

Select Rule-Based from the drop-down options in the Create Feature Process dialog-box (Figure 6-14). The Type selected will automatically create the machining operations needed for the features (e.g., milling, drilling, and tapping), based on best practices established in the MKL. Use MillDrill_NX9 and select/deselect any of those from the Knowledge Libraries list as appropriate. Press OK. Processing may take a few minutes depending on how many or how complex the features are that you have selected. A progress bar will be present in the Cue/Status Line.

Now, use the Resource Bar to go to the Operation Navigator, Program Order view. Notice that what you have processed is now available under the MILL_ROUGH folder with a red circle and slash through it. Select it and press the Generate Toolpath icon (Figure 6-15). Alternatively, you can select from the list, right-click on the path, and choose Generate from the list. Examine the results and edit the paths to any specific needs.

Create a new Program Group to contain all of the paths that will be processed. Left-click to select NC Program (highlighted in Figure 6-6). Right-click and choose Insert from the menu, and then Program Group. Change the Name in the dialog-box to FBM_EXAMPLE and press OK twice (Figure 6-16). Then Drag-and-Drop the Operation Group folder MILL_ROUGH—which contains the generated MILL_SURFACE_PLANAR_RECT_FACE_MILL path into the newly-created FBM_EXAMPLE Program Group.

As processes are completed and toolpaths are generated, the software keeps track of the finished material condition in what is called the In Process Workpiece (IPW). In this case, the blank is now machined to the zero stock condition after having taken .100 off the face of the part. If you attempt to place a machining element of the program out of order, a warning will be generated.
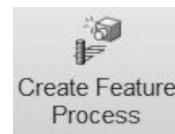
Figure 6-13 Create Feature Process Icon
Courtesy Siemens PLM
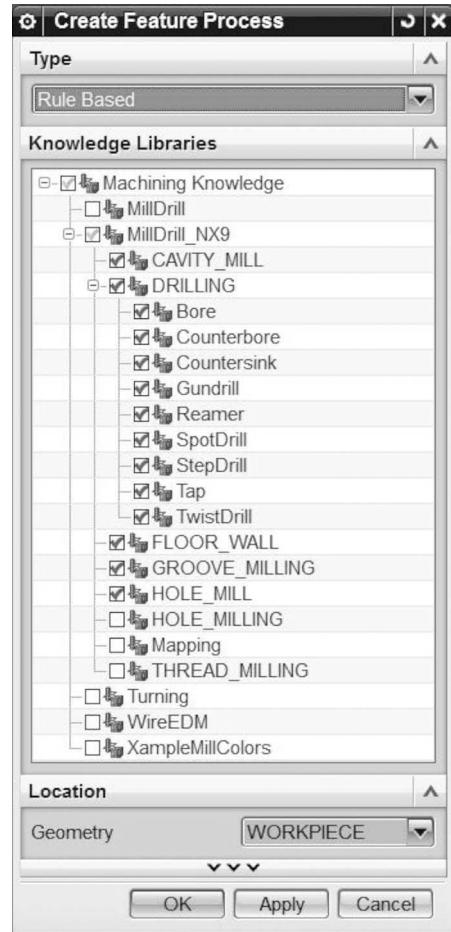
Figure 6-14 Create Feature Process Dialog
Courtesy Siemens PLM

Figure 6-15 Generate Tool Path Icon
Courtesy Siemens PLM
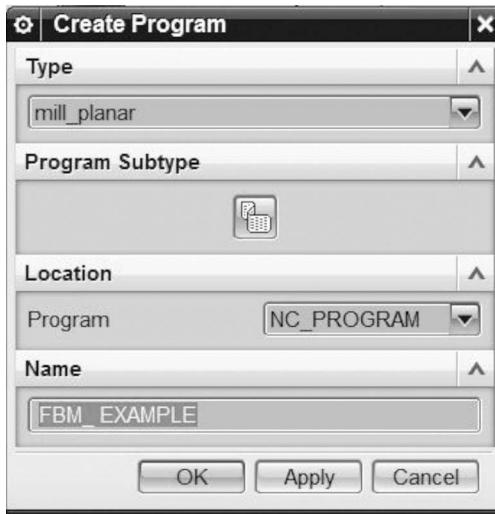
Figure 6-16 Create Program Dialog
Courtesy Siemens PLM

CREATE NEW PROGRAM GROUPS

The generated paths (Operations) for the remainder of the features will need to reside in tool-specific Program Group folders.

G. To start, left-click to highlight the MILL_ROUGH Program Group from the list. Right-click and choose Rename from the options. Type in F-MILL_3.0.

Select all of the automatically created Program Folders (Figure 6-6), then drag-and-drop them into the FBM_EXAMPLE Program Group.

Make several copies of the existing folders for all the different size drilling tools needed (.242, .296, .343, .404 and .953). To do this, right-click on the DRILL folder, select Copy from the list of options, then right-click again and press paste from the list of options. Continue until you have enough folders for each tool size, including milling tools. Lastly, left-click to select one of the folders; then right-click and choose Rename from the list of options. Rename the Program folder to match the tool size, as shown in Figure 6-17.

Continue creating Operation Groups to match the tools used in the program, as seen in Figure 6-6. Delete any Operation Groups that will not have contents, e.g., REAM, DEBURR, and MILL-SEMI-FINISH.

Return to the Machining Feature Navigator view via the Resource Bar and select everything in the list, except, SURFACE_PLANAR_RECTANGULAR_34, which has already processed. Go through the list and multiple select the recognized features (use Ctrl-click).

Figure 6-17 Program and
Operation Groups
Courtesy Siemens PLM

Press the Create Feature Process icon from the Ribbon Bar, as shown in Figure 6-13. Use the same settings shown in Figure 6-14 in the Create Feature Process dialog box.

FEATURE GROUPS

Feature Groups (Figure 6-18) are automatically created as Feature Processes are generated. They consist of those features that have identical attributes. Return to the Machining Feature Navigator Feature View via the Resource Bar; then right-click in the white space and choose Group View from the list. For example, expand STEP1HOLE_THREAD to see the grouped features.

OPTIMIZE TOOL CHANGES

H. If paths were generated and posted in the order shown in Figure 6-19, numerous redundant tool changes would be created. We will make adjustments to alleviate this issue. Occasionally, the tools suggested from the library will not be available and require attention as well.

Figure 6-18 Feature Groups
Courtesy Siemens PLM

Figure 6-19 Feature Operations Processed
Courtesy Siemens PLM

Double-click on the MILL_HOLE_FREE_ STR, Operation Group, and the Floor Wall toolpath dialog-box will display Figure 6-20. Expand the Tool section. The software has suggested a 2 and 1/2" End Mill. Our shop does not have this tool, so we will select the (End Mill 1") from the list of tools and press OK.

A similar situation exists for the MILL_ SURFACE_PLANAR_RECT_END_MILL. Change it to the End Mill 1" as well.

Double-click on the MILL_HOLE_RECT_ STR, Operation Group, change the tool to the (Carbide End Mill .495), and press OK.

Double-click on the CHAMFER_S2H_ CSINK_D1, Operation Group and the Countersinking dialog-box will display. Expand the Tool section. The software has suggested a Countersink 2" 90-degree tool. We need to use a smaller tool because our shop has only a 3/4" diameter CSK with the correct angle.

In the Tool section, choose the Create new icon next to the tool description and press OK to dismiss the warnings. From the New Tool dialog-box, change the Type to hole-making. Then select the COUNTER_SINK Tool Subtype and press the Retrieve Tool from Library icon (Figure 6-21). From the Library Class Selection, expand the Drilling class and pick Countersinking, as shown in Figure 6-22.

In the Search Criteria dialog box, press the Question mark at the bottom right corner, then OK. Lastly, from the Search Result (Figure 6-23), pick the 3/4" x 90-degree tool and press OK. Dismiss the warning.

NOTE: These tool library results are the most basic library results available—a full-featured, managed library, the Manufacturing Resource Library, is also available.

Because the tool was created, we will need to ensure it is placed in the correct pocket manually. Go to the Operation Navigator and ente;r te Machine Tool view. The tool in Pocket 20 is the 2" CSK. Left-click to highlight it; then right-click and choose Delete from the list. Now drag the new tool just created from the Unused Items folder to Pocket 8.

Drag the path into the COUNTERSINK program group folder. Regenerate the toolpaths and proceed.
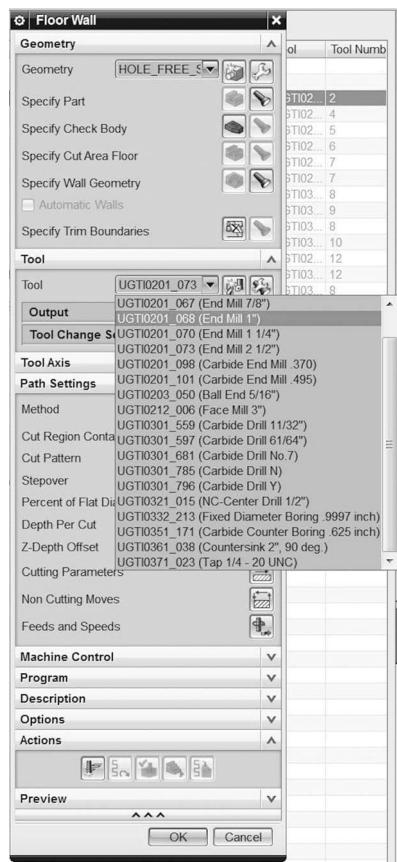
Figure 6-20 Floor Wall Dialog,
Tool Dropdown
Courtesy Siemens PLM
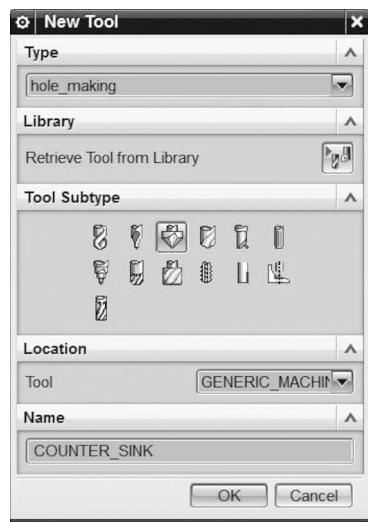
Figure 6-21 Create Tool Dialog
Courtesy Siemens PLM
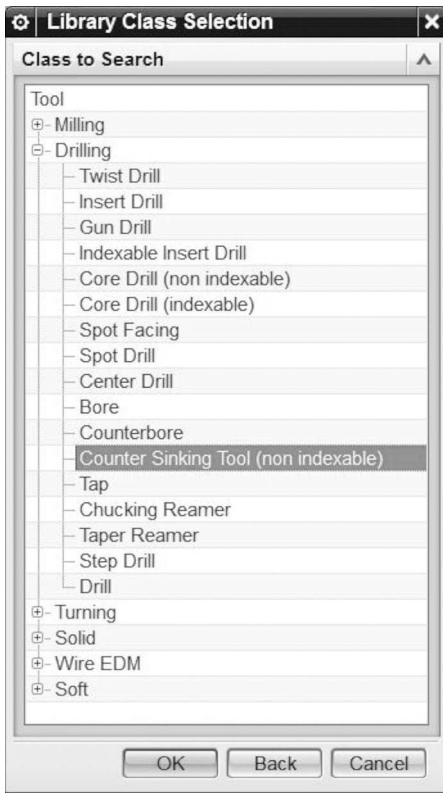
Figure 6-22 Tool Library Class Selection
Courtesy Siemens PLM
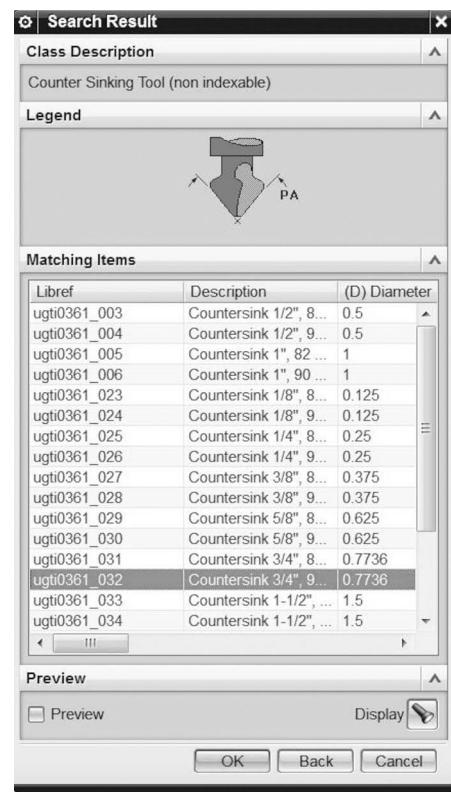
Figure 6-23 Tool Search Result
Courtesy Siemens PLM

Highlight NC_Program in the Operation Navigator, Program Order view. Right-click and choose Object from the list and then Optimize. In the Optimize Tool Paths dialog box, uncheck Create Optimization Group and press OK (Figure 6-24). This action will sort the Operations so those with the same tool are grouped together.

Generate the paths sequentially, noting any required changes. Remember that you always have the choice of modifying the output.

Figure 6-24 Optimize Tool Paths Dialog
Courtesy Siemens PLM

One such instance here is the MILL_SLOT_PARTIAL_U_SHAPED needs to have the containment adjusted. Double-click on the Operation Group and select the Cutting Parameters icon in the Floor_Wall dialog box. Pick the Containment tab and change it to None (Figure 6-25).

Drag-and-drop the generated paths into the Mill_1.0 Operation Group in the FBM_EXAMPLE Program Group you created earlier and regenerate the toolpaths.

Continue placing the related paths in the proper Operation Group and generating toolpaths. Make adjustments, if needed.

PRODUCT MANUFACTURING INFORMATION

I. Product Manufacturing Information (PMI) represents annotations on the part design that are taken into consideration automatically during manufacturing process creation; these annotations affect the operations that are developed. In the example in Figure 6-26, a bilateral hole-tolerance is given to each of the 1.0 diameter holes at the corners of the part. Another commonly used annotation is for surface finish requirements.

Figure 6-25 Cutting
Parameters Dialog
Courtesy Siemens PLM

If you look at Feature Group STEP2HOLE, notice the symbol is included to indicate PMI for Radial Dimension on the holes (Figure 6-27). The toolpaths created for this feature will include a boring operation that is needed to attain the tolerance requirement for the holes.

GENERATING MACHINING TOOLPATHS

J. Go to the Operation Navigator, Program Order view, and select the desired toolpath (for example, the out-of- Date, Counterbore_1.25 in Figure 6-28). Press the Generate Tool Path icon (Figure 6-16). Alternatively, you can select from the list and right-click on the path, then choose Generate from the list. Examine the results and edit the paths to any specific needs.

Figure 6-26 Product Manufacturing
Information
Courtesy Siemens PLM

Figure 6-27 PMI Group
Courtesy Siemens PLM

In every case, it is possible to adjust the created process to meet your expected outcome. Company best practices will override and; therefore, the MKL might need to be edited by your senior programmer.

Repeat these steps until all toolpaths are generated.

VERIFY TOOLPATHS

K. To verify all toolpaths, highlight the Program Group, FBM_EXAMPLE, and press the Verify Tool Paths icon from the Ribbon Bar (Figure 6-29). In the Tool Path Visualization dialog box, select the 2D Dynamic tab. Use the Animation Speed slider to slow down the playback speed to 6; then press the Play button.

Note: Once 2D Dynamic playback is displayed (Figure 6-30), the view cannot be manipulated to review. Choose 3D Dynamic and press Play for that capability (Figure 6-31).

Experiment with the many options within the dialog box.

OUTPUTTING CNC PROGRAM CODE

One of the last steps in the process is creating the G-code that controls the Machine Tool. The postprocessor selected depends on the type of machine tool controller being programmed. For this example, the post "mill_3axis_Sinumerik_840D_in." is used.

Figure 6-28 Generate Out-of-Date Toolpath
Courtesy Siemens PLM

Figure 6-29 Verify Tool Path Icon
Courtesy Siemens PLM
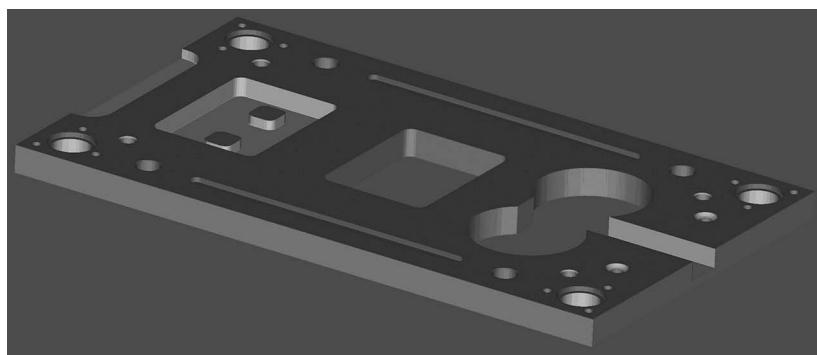
Figure 6-30 2D Dynamic Toolpath Visualization
Courtesy Siemens PLM
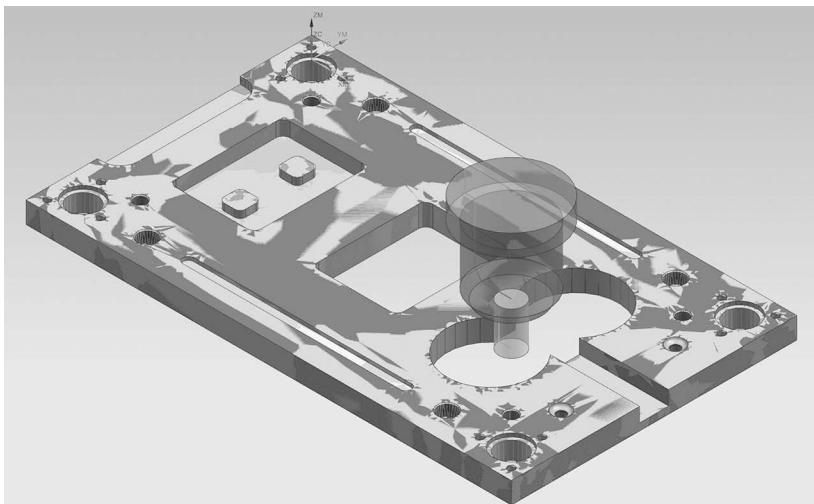
Figure 6-31 3D Dynamic Toolpath Visualization
Courtesy Siemens PLM

Figure 6-32 Post
Process Icon
Courtesy Siemens PLM

L. Highlight the FBM_EXAMPLE, Program Group, and press the Post Process icon from the Ribbon Bar (Figure 6-32). From the Postprocess dialog box, choose the mill_3axis_ Sinumerik_840D_in. from the list (Figure 6-33). If not available, press the Browse for a Postprocessor icon in the dialog box. Be sure to set the Output File location to a folder where you want the posted path to reside. This file is then sent to the Machine Tool controller.

CREATING AN ELECTRONIC SETUP SHEET

The shop will need direction on how the part is to be machined. The next step is communicating this information to the machinist on the shop floor. This can be done via either hardcopy or the intranet. By using NX CAM Shop Documentation, the output can be formatted in HTML or Excel, and include the tools used in the program, the tools' unique characteristics, images of the finished part, and fixture requirements. Tooling and specific setup information, including images, can be viewed on any computer with an internet browser, leading to the ultimate goal of a "paperless" factory.
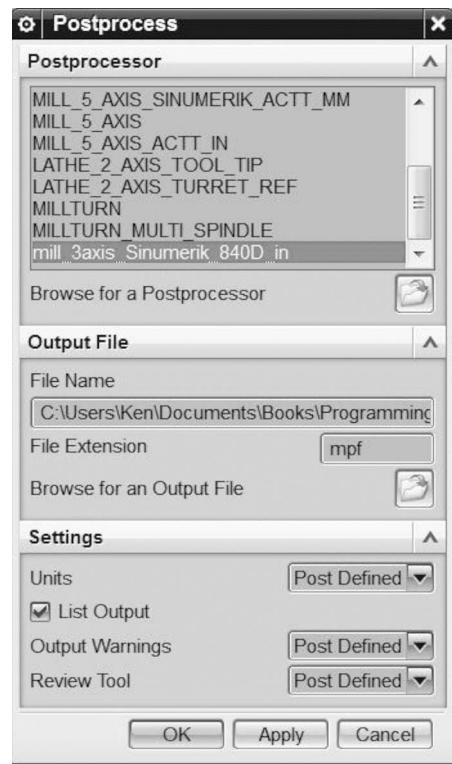
Figure 6-33 Post Process Dialog
Courtesy Siemens PLM

Part 6 Introduction to Feature-Based Machining
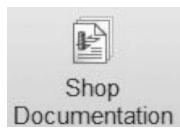
Figure 6-34 Shop
Documentation Icon
Courtesy Siemens PLM

Figure 6-35 Shop
Documentation Dialog
Courtesy Siemens PLM

SIEMENS

Tool Sheet

Part name: Test-Plate_FBM   Drawing name: --
Unit: IN   Part number: --
Pictures :   Description : --

Tool Number	Tool Name	Tool Description	Tool Diameter	Adjust Register	Cutcom Register	Flute Length	Tool Ext. Length	Holder Description	Tool Path Time in Minutes	Operation Name
1	UGTI0212_006	Face Mill 3"	3.5000	1	1	0.2500	2.5945	Small facemill adaptor	5.70	MILL_SURFACE_PLANAR_RECT_FACE_MILL
4	UGTI0201_101	Carbide End Mill .495	0.4950	4	4	0.2780	2.0000	Small weldon	21.47	MILL_HOLE_RECT_STR MILL_POCKET_FREE_STR
3	UGTI0201_098	Carbide End Mill .370	0.3700	3	3	0.2780	2.0000	Small weldon	4.02	MILL_POCK_OBROUND_STR
2	UGTI0201_068	End Mill 1"	1.0000	2	2	1.5000	1.7953	Large weldon	6.42	MILL_HOLE_FREE_STR MILL_SLOT_PARTIAL_RECT MILL_SLOT_PARTIAL_U_SHAPED MILL_SURFACE_PLANAR_RECT_END_MILL COUNTERMILL_S2H
5	UGTI0321_015	NC-Center Drill 1/2"	0.5000	5	5	0.0000	2.2441	Large collet	2.53	SPOT_DRILL_CSINK SPOT_DRILL_1 SPOT_DRILL_CSINK_1 SPOT_DRILL_2 SPOT_DRILL_3
14	UGTI0203_050	Ball End 5/16"	0.3125	14	14	0.8125	1.0827	Medium weldon	0.76	MILL_POCKET_ROUND_TAPERED
6	UGTI0301_681	Carbide Drill No.7	0.2010	6	6	5.5118	6.1818	Morse taper	2.40	DRILL_IN_CENTER_CHAMFER_S1H_2
7	UGTI0371_023	Tap 1/4 - 20 UNC	0.2500	7	7	0.4330	1.7126	Medium collet	4.86	TAP_S1H_THREAD
12	UGTI0332_213	Fixed Diameter Boring .9997 inch	0.9997	12	12	0.2362	4.9213		1.88	DRILL_UP_S1H BORE_S1H
10	UGTI0351_171	Carbide Counter Boring .625 inch	0.6250	10	10	1.9583	3.9370	Large collet	0.49	COUNTERBORE_S2H
11	UGTI0361_032	Countersink 3/4", 90 deg.	0.7736	11	11	0.0000	1.1811	Large collet	0.41	CHAMFER_S2H_CSINK_D1
15	UGTI0301_081	Twist Drill 11/32"	0.3437	15	15	3.4375	3.5162	Medium compression	0.42	DRILL_IN_CENTER_CHAMFER_S1H_1
8	UGTI0301_078	Twist Drill 19/64"	0.2969	8	8	3.0625	3.1412	Large compression	0.39	DRILL_IN_CENTER_S1P
9	UGTI0301_525	Twist Drill Y	0.4040	9	9	2.2441	5.1755	Medium collet	2.72	DRILL_IN_CENTER_CHAMFER_S1H DRILL_IN_CENTER_S1H_1 DRILL_IN_CENTER_S1H

Author : Ken   Checker : Ken   Date : Sat Jan 03 16:00.28 2015

Figure 6-36 FBM Setup Sheet
Courtesy Siemens PLM

M. Highlight the FBM_EXAMPLE, Program Group in the Operation Navigator. Press the Shop Documentation icon in the Ribbon Bar (Figures 6-34 and 6-35). The resultant output is shown in Figure 6-36.

SUMMARY

The intent of FBM is to incorporate the maximum amount of technology and automation into building the CNC program in order to save time and to output quality part programs efficiently. This chapter has only touched on the full capabilities of NX CAM 9.0 FBM. The programmer has the added ability to incorporate User Defined Features (UDFs) by using the Teach Feature dialog box and more.

Part 6 Study Questions

1. Define FBM.

2. What is the primary use for FBM?

3. Name three common formats for Solid Model files.

4. List the basic steps used in FBM to create a program.

5. FBM will output suggested operations based on information in the Machining Knowledge Library.

       T or F

6. Annotations on the model are called.

       a. Dimensions

       b. Hatch

       c. Product Manufacturing Information (PMI)

       d. GD&T

7. NX FBM will develop tools based on the largest diameter possible within the dimensional limits of the feature, including the required depth of the machined feature.

       T or F

8. Feature Groups are automatically created as Feature Processes are generated and consist of those features that have identical attributes.

       T or F

Part 6 Introduction to Feature-Based Machining

9. During toolpath verification, the systems stores the resultant stock condition and is called:

   a. MCS

   b. CSYS

   c. MKL

   d. IPW

10. The toolpath generated from FBM will not be editable.

   T or F

PART 7

FANUC NC GUIDE PROGRAMMING

Figure 7-1 FANUC Control Panel
Courtesy FANUC FA AMERICA

OBJECTIVES:

1. Learn about the advantages of FANUC NC Guide conversational programming.

2. Become familiar with NC Guide programming.

3. Learn terminology specific to NC Guide programming.

4. Become familiar with display screens and dialogue menus typical to NC Guide for the FANUC 0i controller.

5. Demonstrate how to create a CNC turning center program using NC Guide.

6. Demonstrate how to create a CNC machining center program using NC Guide.

NC GUIDE

NC Guide is PC software created by FANUC FA AMERICA to aid CNC programmers and machinists during input of program data via guided dialogs. Simulation of a completed program makes it possible to find and correct errors prior to machining. Finally, the program may be saved to a memory card and transported to the machine tool for use. The software installed on the actual machine controller is called *Manual Guide i*. It can be used in a practically identical way as NC Guide.

START NC GUIDE

To start NC Guide on the PC, go to the Windows Start button, left-click All Programs, scroll to find FANUC NC Guide, and select the folder FSOi-D from the list choose NC Guide (FSOi-D).

SET INITIAL PARAMETERS

There are certain parameters that, if set properly, will aid in the programming process. In order to change any parameters, you must first enable the Parameter Write function. For instance, use SETTINGS to input unit system desired.

1. Enter the MDI mode and press the OFS/SET function button.

2. Adjust the first field in the SETTING (HANDY) dialog, Parameter Write Enable (PWE) to 1, scroll down to input unit system, and change as desired (Figure 7-2). When changes are completed, turn OFF PWE. The controller must be restarted to take effect.

3. While in the MDI mode, adjust any other specific parameters by pressing the SYSTEM function button and then selecting the PARAMETERS soft key.

4. Use the keyboard arrow and page keys to position to the desired parameter input location (Figure 7-3). To search to a specific parameter number, enter the number and press the NO. SRH soft key.
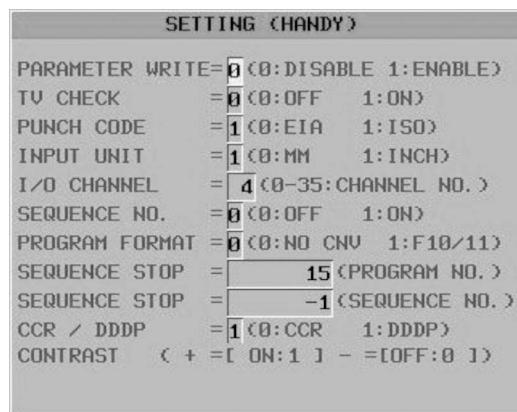
Figure 7-2 SETTING (HANDY)
Courtesy FANUC FA AMERICA
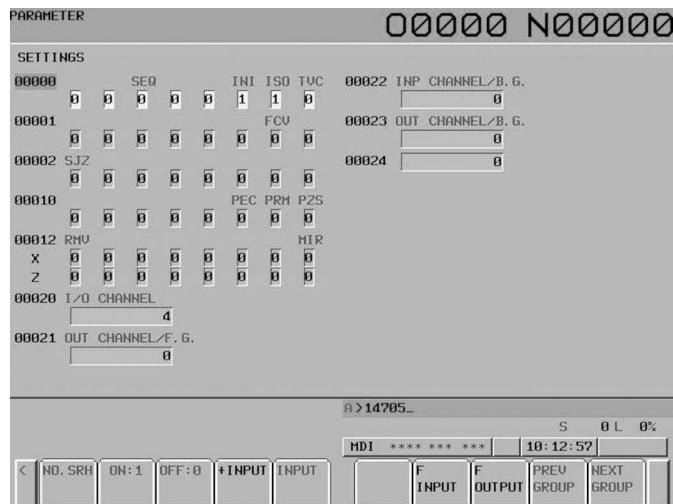
Figure 7-3 SYSTEM
PARAMETERS
Courtesy FANUC FA AMERICA

After making the required changes, you must turn OFF PWE and restart the controller for the changes to take effect.

Caution: Any changes to parameters will affect the actual machine operation and could result in unexpected results if their function is not fully understood. Consult the manufacturer's Parameter Manual for detailed information.

5. Below are some suggested settings:

Note: the bit arrangement is from right to left starting at zero. For example, bit 7 is the left most in the series.

14705 bit#7 (1) Tool Offsets display

27120 bit #2 (1) End Face Machining

27120 bit #1 (1) Residual Machining

3401 bits #6 and #7 (1) Programming Language B and C

START NEW NC GUIDE PROGRAM

1. Start NC Guide by entering the EDIT Mode function button.

2. Press the GRAPH function button and then the CUSTOM 1 key.

Use the Continuous Menu button (right arrow under soft keys) to advance to the set of soft keys displayed in Figure 7-4.

Figure 7-4 NC Guide Soft Keys
Courtesy FANUC FA AMERICA

Figure 7-5 TOOL OFFSET (T-OFS) Soft Key
Courtesy FANUC FA AMERICA

BUILD TURNING CENTER TOOL DATA

Study the engineering drawing/blueprint (Figure 7-19) and identify all cutting tools needed to complete the part. Following CNC Programming best practices establish the order of their use.

1. Press the Continuous menu arrow (4 times) until the T-OFS soft key is visible (Figure 7-5). Then press it to acess the Tool Offset Geometry Tab.

The GEOMETRY OFFSET tab (Figure 7-6) includes the X and Z measured values, the tool tip nose RADIUS, and VIRT.TIP (virtual tip) or Tool Tip Orientation. For simulation only, no data entry is needed for the X and Z values; however, they must be measured before automatic operation.

2. Use the arrow keys to move the cursor over the desired data entry point or tab. The RADIUS must be entered to match the tool nose exactly (Figure 7-6). This information should be available from the insert box or from catalog information.

3. Enter the Virtual Tip (VIRT.TIP) information. Use the graphical soft key guide, as shown in Figure 7-7, to identify the proper orientation setting. For example, OD Turning will be direction 3. Data for each tool used in the program must be completed or an error may result.

Figure 7-6 GEOMETRY OFFSET Display
Courtesy FANUC FA AMERICA

Figure 7-7 VIRTUAL TOOL TIP Soft Keys
Courtesy FANUC FA AMERICA

Part 7 FANUC NC Guide Programming

Values for the WEAR OFFSET tab are required only to correct for inaccuracies during machining (Figure 7-8). Tool Tip Orientation is carried over from entry on the Geometry page.

The TOOL DATA tab (Figure 7-9) requires entry of the Tool Cutter Type (TOOL), the Tool Description (SET), the Cutting Angle (CUT AN), and the Tool Nose Angle (NOS AN). Details on cutting tool geometries can be obtained for the vendor catalogs and online applications.

The Tool Cutter Type may be selected from the options available when the cursor is moved over the first column (Figures 7-10 and 7-11).

Figure 7-8 WEAR OFFSET Display
Courtesy FANUC FA AMERICA

Figure 7-9 TOOL DATA Display
Courtesy FANUC FA AMERICA
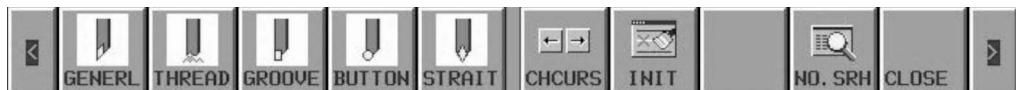
Figure 7-10 Tool Type Soft Keys
Courtesy FANUC FA AMERICA

Figure 7-11 Tool Type Additional Soft Keys
Courtesy FANUC FA AMERICA

The Tool Description may be input or edited by positioning the cursor over that column. Press the CHCURS to input new characters for the label. Press the INIT soft key to delete the entire item. An alarm will display (010 WILL BE INITIAL, ARE YOU SURE?) and YES/NO soft keys will force a decision. Press enter to accept any changes.

When the cursor is over the SET column, the graphic in Figure 7-12 is displayed. Choose the appropriate number to represent the machining to be done. For example, for OD turning and Facing, select 1. For facing only, select 6.

When the cursor is positioned over the CUT AN (cut angle) column, the angle from the spindle axis to the face of the insert (indicated by A in Figure 7-13) is required. Details on cutting tool geometries can be obtained for the vendor catalogs and online applications.

The NOS AN (Nose Angle) is the actual insert shape geometry angle (Figure 7-14). For example, a typical roughing insert has an 80-degree angle.

When the data for all tools used in the program is entered, press the CLOSE soft key.

REGISTER TURNING CENTER FIXED FORM SENTENCES

Fixed Forms sentences are repeating segments of a program that make up a sort of template and thus aid in speedy program input. In some cases, Machine Tool Builders will supply these with the purchase of the machine and the programmer need only become familiar with their contents for understanding and use. In the case of NC Guide software for the PC, it will be necessary to construct these from scratch.

1. From the Edit mode, Press the continuous menu key (right arrow in soft keys) until SETING is available and select it (Figure 7-15). Use the cursor to select item 2. REGISTER FIXED FORM SENTENCE FOR TURNING in the SETTINGS dialogue (Figure 7-16). Then Press the SELECT soft key.

As a rule, the items listed in the tab labeled FORM 1 are related directly to the START soft key; those in tab FORM 5 are related to the END soft key while programming. Only five tabs are allowed. For each tab, 10 items can be added to choose from while programming.
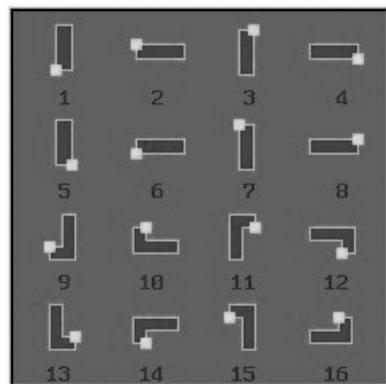
Figure 7-12 TOOL OFFSET, TOOL DATA, SET
Courtesy FANUC FA AMERICA
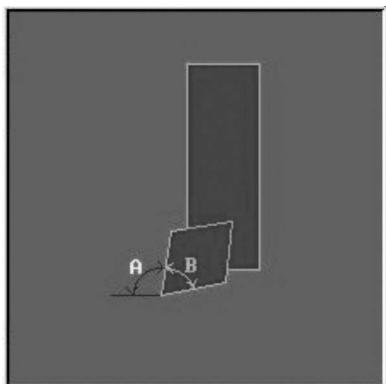
Figure 7-13 TOOL OFFSET, TOOL DATA, CUT ANGLE
Courtesy FANUC FA AMERICA
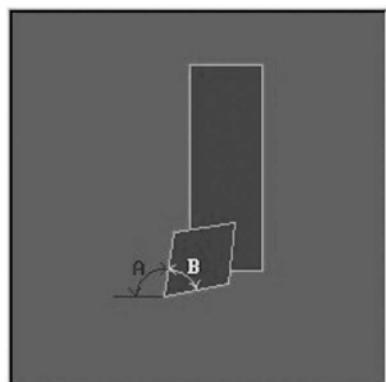
Figure 7-14 TOOL OFFSET, TOOL DATA, NOSE ANGLE
Courtesy FANUC FA AMERICA

Figure 7-15 SETING Soft Key
Courtesy FANUC FA AMERICA

- Press the NEW soft key to enter a new Fixed Form Sentence.
- Press the ALTER soft key to edit an existing Fixed Form Sentence.
- Press the NEW soft key to enter a new Fixed Form Sentence.
- Press the DELETE soft key to delete a Fixed Form Sentence. (You will be prompted to confirm whether you want to delete the file with the Yes or No soft key.)
- Press the STAND. Key to delete all Fixed Form Sentences. (You will be prompted to confirm, with the Yes or No soft key, whether you want to overwrite the current data).

Common items listed in FORM 1 are shown in Figure 7-17 and Figure 7-18 for FORM 5 below.

Figure 7-16 SETTINGS Dialog
Courtesy FANUC FA AMERICA

Figure 7-17 FIXED FORM TURNING, FORM 1
Courtesy FANUC FA AMERICA
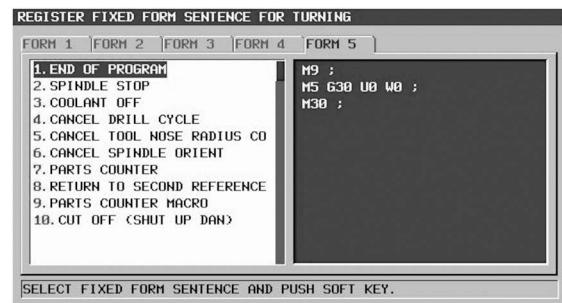
Figure 7-18 FIXED FORM TURNING, FORM 5
Courtesy FANUC FA AMERICA

Place question marks where the actual data will be variable, such as spindle speed and feed-rates. This will force input during the construction of the program that match the conditions required.

Form 2 can be used for cutting conditions specific to the materials commonly used in your shop. Form 3 can be used for Canned Cycles such as drilling and tapping and Form 4 can be used for Subroutine call and cancellation.

Follow the programming conventions covered in Chapter 3 and always consult the manufacturer programming manuals for reference. Remember, the program data you set in these forms will aid in programming input.

NC GUIDE TURNING CENTER PROGRAM CREATION

NC Guide Turning Center Program Example

For this example, we will use FANUC NC Guide software for the PC to create the NC program. It is virtually identical to Manual Guide i (MGi) installed on many Machine Tools with FANUC controls. For this example, we are using the popular FANUC Oi-D control. The sequence of events that happen when programming at the controller and the output are essentially the same. The figures are screen shots from the NC Guide Academic program.

Steps to Create an NC Guide Turning Program

Notes on data entry:

For fields with an asterisk, no data need be entered. Use the Input function button or the keyboard Enter key to enter data in dialog fields. It is possible to use math within numerical fields following the Algebraic Order System (AOS).

The part in Figure 7-19 is Mild Steel.
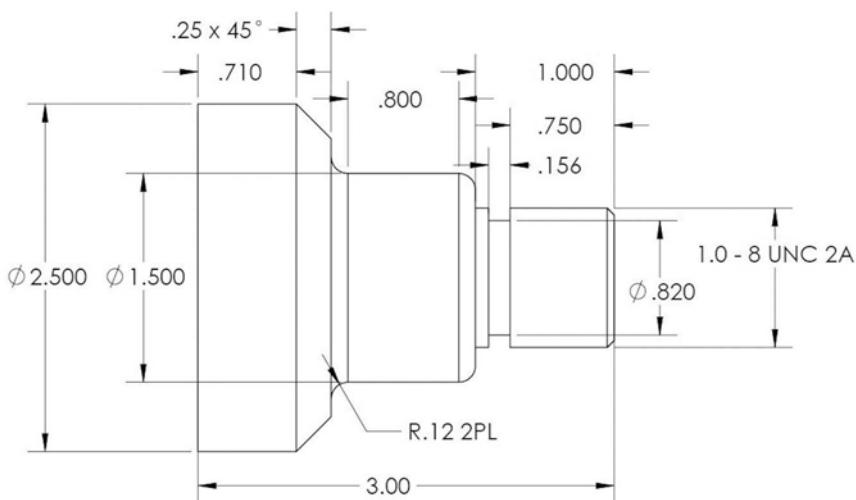
Figure 7-19 Drawing for NC Guide
Turning Center Program

1. From the EDIT mode, press the NEWPRG soft key (see Figure 7-4). **Enter the desired program number** and press the CREATE soft key. For this example, 101 is used (Figure 7-20). The program number you just created will be highlighted in the Program List area of the screen next to the graphic display area.

2. Press the O LIST and then EDTCOM soft keys to add a descriptive program name (e.g., Example 1, part #) and press Enter.

3. Press the OPEN soft key to return to the program display and open the new program for editing.

4. To **define the part blank** type and size (required for simulation only), press the Continuous menu key (right arrow of soft keys) one time.

5. Then press the START soft key (Figure 7-21). Use the cursor key to advance to the BLANK tab and then down to select 2, CYLINDER BLANK FIGURE, from the list. Press the SELECT soft key.

6. From the drawing provided in Figure 7-22, input the data required in the fields: Diameter (D), Length (L), and Work Origin Z (K) amounts. Remember that the value placed in the K field is the amount to be faced off within the program. Press the INSERT soft key when the data are ready.

7. Press the START soft key. From the list on the START tab, select the item input as your START LINES and press the INSERT soft key (Item 3 in Figure 7-17 above).

Figure 7-20 CREATE NEW PROGRAM (TC)
Courtesy FANUC FA AMERICA
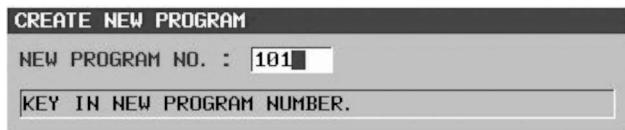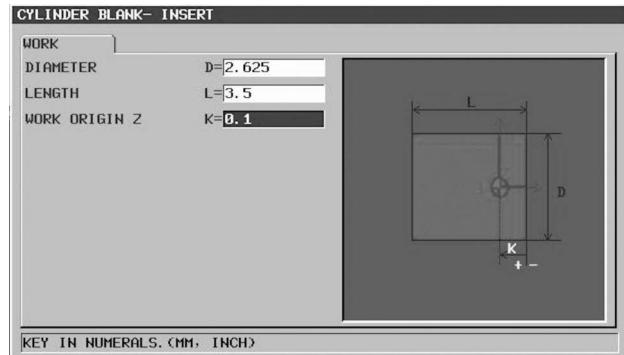
Figure 7-21 START Soft Key
Courtesy FANUC FA AMERICA

Figure 7-22 CYLINDER BLANK
Courtesy FANUC FA AMERICA

Upon insertion, the alarm IMPERFECT WORD MUST BE CHANGED is listed under the graphics display. Edit the required items noted by red question marks (???) in the program code. Use the ALTER function button to make the entry. Enter the appropriate Feeds & Speeds based on workpiece material from technical manuals, your prior learning, and/or experience.

8. Press the START soft key and select the item input as your TOOL CHANGE from the list on the START tab (Item 1 in Figure 7-17 above); then press the INSERT soft key. Upon insertion, the alarm IMPERFECT WORD MUST BE CHANGED is listed under the graphics display. Edit the required items noted in the program code using the same steps as given in Step 7 above. Enter the appropriate Tool number based on the part geometry and the data entered in the Tool Data. In this case, the first tool is number T0101. Give a brief tool description within the parentheses.

9. Press the START soft key and select the item input as your POSITIONING MOVE from the list on the START tab; then press the INSERT soft key (Item 6 in Figure 7-17 above). Edit the required items noted. Enter the appropriate values to position the tool to a clearance of at least .100 of an inch in X and Z. For this example, we use X2.75 Z.2.

10. To **Set Feeds & Speeds,** press the FIXED FORM soft key and arrow over to the FORM 2 tab to use predetermined settings based on materials (Figure 7-23). Edit the required items noted. Set a spindle limiter to maximum r/min of S2200. Establish Constant Surface Speed Control and the appropriate feed in IPR.

Rough and Finish Profile

Now **establish** the **Cycle Type** for machining of the rough Outer Diameter of the part.

1. Press the CYCLE soft key and use the arrow key to position over the TURNING tab. Select the TURNING (OUTER ROUGH) as shown in Figure 7-24; then press the SELECT soft key.

   a. Complete the required data for the CUT COND. Tab, as shown in Figure 7-25 where the values are based on the material type.

   b. When complete, press the arrow key to display the DETAIL tab and review the settings that are carried over; adjust as needed.

   c. With the cursor over the CUT RISE METHOD field, notice

Figure 7-23 FIXED FORM, FORM 2
Courtesy FANUC FA AMERICA

Figure 7-24 TURNING (OUTER ROUGH)
Courtesy FANUC FA AMERICA

the two soft keys labeled SPEED and CUT. Use the CUT soft key to select the CUT RISE method. This sets each pass feed up the back wall on all passes; otherwise, when SPEED is selected, the movement in X and Z will be at rapid traverse, leaving a rough back wall surface.

Figure 7-25 TURNING (OUTER ROUGH) CUT CONDITIONS
Courtesy FANUC FA AMERICA

d. For the APROCH MOTION field, there are three choices as indicated by the soft-keys. Select the one most appropriate to the conditions.

e. Use the cursor to advance to the ED FACE MC tab. Set the values as shown in Figure 7-26. This will remove the .100 inch of stock on the front face of the part to establish the zero face.

f. Press the INSERT soft-key.
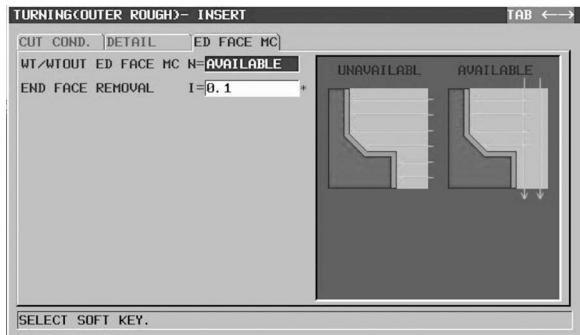
Figure 7-26 TURNING (OUTER ROUGH) ED FACE MC
Courtesy FANUC FA AMERICA

PROFILE COORDINATE DATA ENTRY

1. The START POINT dialog will display. Enter the values that represent the beginning point definition for the finished profile of the part geometry. Input X0.0 and Z0.0, then press the OK soft key. Now begin to describe part geometry by selecting the LINE soft key.

2. Select the LINE soft key and change LINE DIRECTION to UP. Set the END POINT DX= value at 1.0. Press OK to continue.

3. Select the CC soft key. Then set the CHAMFER value C=.05 and press OK. The chamfer will appear when the next line entry is completed.

4. Select the LINE soft key and change to LEFT, LINE DIRECTION. Input the END POINT Z point value of Z= –1.0 and press OK.

5. Select the LINE soft key and change the LINE DIRECTION to UP. Set the END POINT DX value at DX= 1.5 and press OK.

6. Select the CR soft key. Then set the CORNER RADIUS value of R=.2 and press OK. The corner radius will appear when the next line entry is completed.

7. Select the LINE soft key and set the LINE DIRECTION to LEFT. Input the END POINT Z point value of Z= –2.2 and press OK.

8. Select the CR soft key. Then set the CORNER RADIUS value of R=.2 and press OK.

9. Select the LINE soft key. Change LINE DIRECTION to UP. Set the END POINT DX value at DX= 2.5. Press OK to continue.

10. Select the CC soft key. Then set the CHAMFER value C=.25 and press OK.

11. Select the LINE soft key and change LINE DIRECTION to LEFT. Input the END POINT Z point value of Z= –3.0 and press OK.

12. Before entry for this last point, change the ELEMENT TYPE to BLANK. Select the LINE soft key and change LINE DIRECTION to UP. Set the END POINT DX value at: DX= 2.625 and press OK.

13. Press the continuous menu arrow four times to find the BL CONT soft key and press it.

14. Select the FIG. 1 soft key to set the figure to the OD of the contour.

15. Answer the prompt, SELECT BLANK IS CONNECTION. Select YES using the soft key to accept the command.

16. Press the CREATE soft key.

17. Choose the INSET IN CURRENT PROGRAM radio button. Press the OK soft key.

Your completed geometry profile will display as in Figure 7-27.

18. Press the FIXFRM soft key and choose POSITIONING MOVE from the list; then press OK. Correct the alarm, IMPERFECT WORD MUST BE CHANGED, by inputting X3.0 and Z.5 M9, and pressing the ALTER function button.

19. Press the FIXFRM soft key. Choose TOOL CHANGE from the list and press OK. Correct the alarm, IMPERFECT WORD MUST BE CHANGED, by inputting T0303 (93 x 55 FIN OD), and pressing the ALTER function button.

Figure 7-27 Geometry Profile
Courtesy FANUC FA AMERICA

20. Press the FIXFRM soft key. Choose POSITIONING MOVE from the list and press OK. Correct the alarm, IMPERFECT WORD MUST BE CHANGED, by inputting X3.0 and Z.5 M8, and pressing the ALTER function button.

21. Press the CYCLE soft key and choose TURNING (OUTER FINISH) from the list. Then press the SELECT soft key.

22. Set the CUT COND. dialog to –Z CUTTING DIRECTION and the FEED RATE to .010. Press the INSERT soft key. Press the CANCEL soft key twice to return to the program.

Rather than re-enter the profile coordinate data of the finish profile again, copy and paste it instead.

23. Position the cursor using the arrow keys over the line starting with G1450.

24. Press the Continuous Menu arrow until the COPY soft key is present; then press it.

25. Using the arrow key, move the cursor down in the program until the line starting with G1456 is reached (Figure 7-28).

26. Press the COPY soft key.

27. Using the arrow keys, position the cursor at the EOB character at the very last entry for the POSITION MOVE after the TOOL CHANGE.

28. Press the PASTE soft key. The entire profile will be inserted for the Finishing operation.

29. SIMULATE the toolpath to confirm correctness. The results should resemble Figure 7-29

GROOVE

Program the escape groove for the threaded portion of the part.

30. Press the FIXFRM soft key, select TOOL CHANGE from the list, and set the Tool Number for Grooving tool to T0505 with the comment of (.125 x .3 GRV).

Figure 7-28 COPY Profile Contents
Courtesy FANUC FA AMERICA
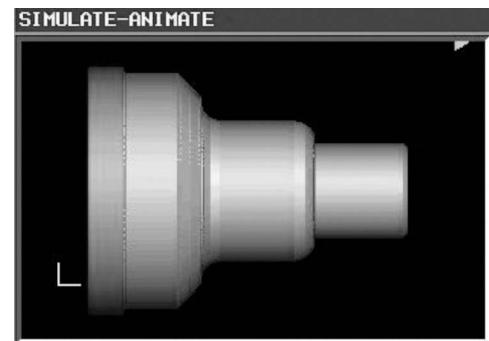
Figure 7-29 SIMULATE
Geometry Profile
Courtesy FANUC FA AMERICA

31. Position the cursor at the EOB character and key-in G97 S750 M3; this will lock the spindle speed at 750 r/min for the grooving operation.

32. Press the FIXFRM soft key and arrow to the FORM 2 tab.

33. Press the CYCLE soft key, use the arrow key to position over the T-GROOVING (OUTER ROUGH) tab and press the SELECT soft key.

34. Complete required data for the CUT COND. tab as shown in Figure 7-30 where the values are based on the material type.

35. When complete, press the arrow key to display the DETAIL tab. Input the settings given in Figure 7-31 and press the INSERT soft key when done.

Figure 7-30 GROOVING CUT CONDITIONS
Courtesy FANUC FA AMERICA

36. From the INSERT TURNING FIGURE dialog, select ZX-GROOVE (OUTER NORMAL).

37. From the ZX-GROOVE (OUTER NORMAL) dialog, set the POS/SIZE tab values per Figure 7-32.

38. Leave the CORNER INF and REPEAT tabs as default settings.

39. Press the INSERT soft key.

40 Press the FIXFRM soft key and select POSITION MOVE from the list. Alter the required coordinates to: X4.0 Z.1 M09.

41. SIMULATE the toolpath to confirm correctness.

OUTER DIAMETER THREAD

42. Press the FIXFRM soft key and select TOOL CHANGE from the list. Alter the data to read T0707 (60 DEG THD).

43. Press the FIXFRM soft key and select POSITION MOVE from the list. Alter the required coordinates to X4.0 Z.1 M08.

44. Press the CYCLE soft key and use the arrow key to move to the THREADING tab. Select THREADING (OUTER) from the list.

45. Complete the THREADING dialog CUT COND. tab per Figure 7-33.

46. Complete the THREADING dialog DETAIL tab per Figure 7-34.

47. From the INSERT TURNING FIGURE dialog, select ZX-THREADING (GENERAL) from the list.

48. Complete the POS/SIZE dialog per Figure 7-35. Use the CALC soft key to input the THREAD DEPTH, H=.
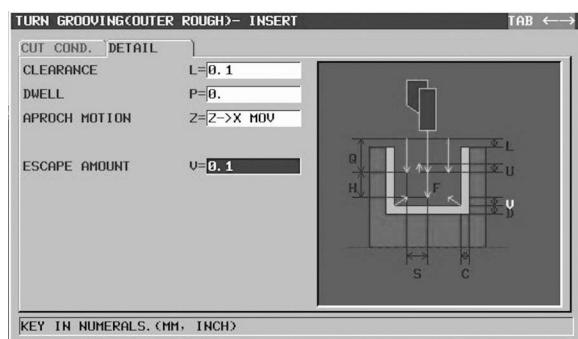
Figure 7-31 GROOVING DETAIL
Courtesy FANUC FA AMERICA
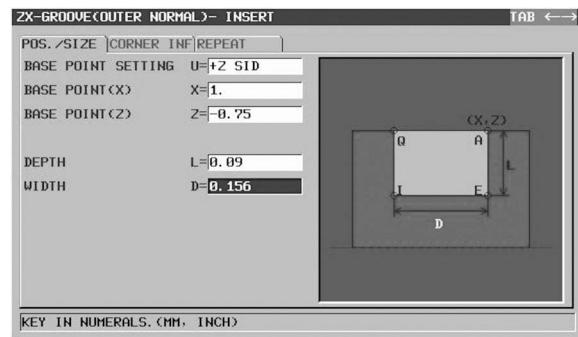
Figure 7-32 GROOVING POS./SIZE
Courtesy FANUC FA AMERICA
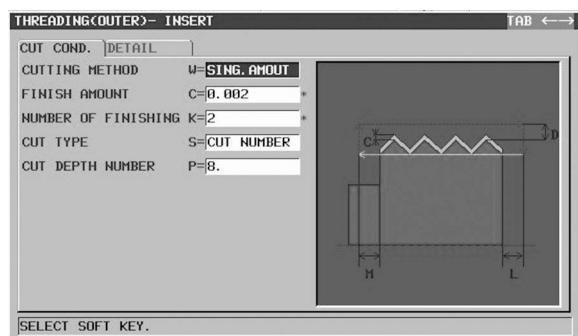
Figure 7-33 THREADING CUT CONDITIONS
Courtesy FANUC FA AMERICA

49. Press the FIXFRM soft key and select POSITION MOVE from the list. Alter the required coordinates to X4.0 Z.1 M09.

50. SIMULATE the toolpath to confirm correctness (Figure 7-36).

END PROGRAM

51. Press the FIXFRM soft key, use the arrow key to advance to the FORM 5 tab, and select END OF PROGRAM from the list.
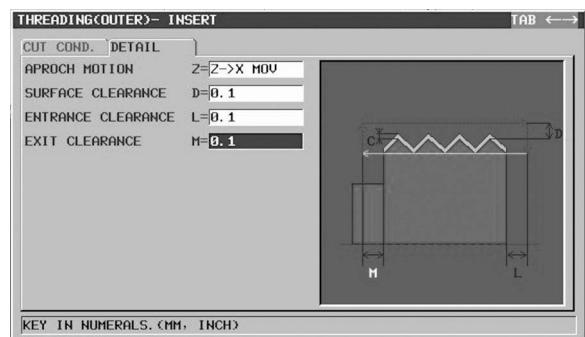
Figure 7-34 THREADING DETAIL
Courtesy FANUC FA AMERICA
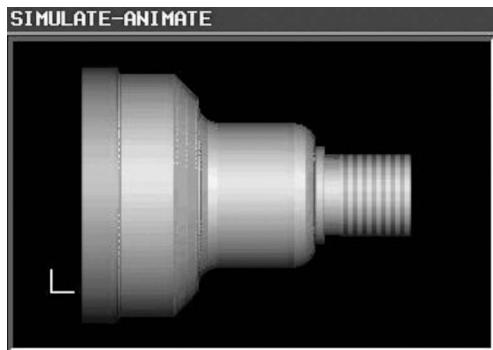
Figure 7-36 Finished Turning Example Part
Courtesy FANUC FA AMERICA
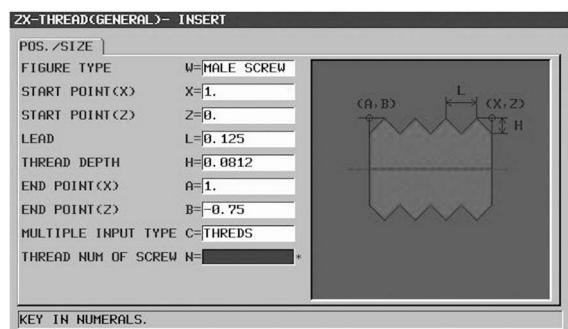
Figure 7-35 THREADING POS./SIZE
Courtesy FANUC FA AMERICA

NC GUIDE MACHINING CENTER PROGRAM EXAMPLE

BUILD MACHINING CENTER TOOL DATA

Study the engineering drawing/blueprint (Figure 7-37) and identify all cutting tools needed to complete the part. Following CNC Programming best practices, establish the order of their use.

- Start NC Guide, press the GRAPH function key, and enter the Edit mode.

- Press the Continuous menu arrow (3 times) at the right end of the soft keys until the T-OFS soft key is visible; then press the soft key to enter the Tool Offset tab. The cursor will highlight the 001 tool under the Geometry column. The value in this column may be left at zero until the machine is set up and the tool length is measured.

- Use the right arrow key to move the cursor to the CUTTER COMPENSATION, GEOMETRY column. Enter 3.0 for the Facemill diameter and press Enter.

- Press the arrow key to the right 2 more times to display the TOOL OFFSET, TOOL DATA tab.

- Press the continuous menu button once to display the tool type, FACE, and press the soft key to select it (Figure 7-38). Enter 1 in the SET column to identify the tool axis direction.
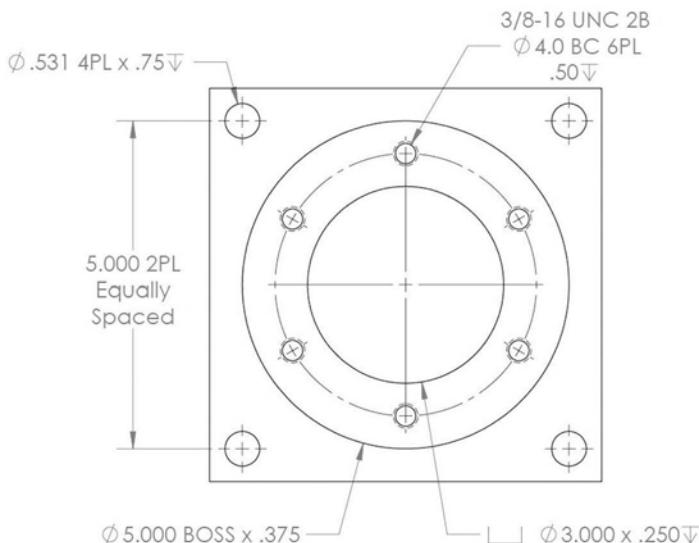
Figure 7-37 Drawing for NC Guide Machining Center Program

Figure 7-38 FACE Soft Key
Courtesy FANUC FA AMERICA

• Continue entering the data for each tool used in the program, as shown in Figures 7-39 and 7-40.

REGISTER MACHINING CENTER FIXED FORM SENTENCES

The same rules apply as in Registering Turning Center Fixed Form Sentences; refer to that section of this chapter for a refresher, if needed. You may look ahead in this next step-by-step section for some ideas on what will be needed.

• From the Edit mode, press the continuous menu arrows until the SETING soft key is available, and choose it (same as Figure 7-15). Use the cursor in the SETTINGS dialog to select item 1, REGISTER FIXED FORM SENTENCE FOR MILLING, Figure 7-16. Then Press the SELECT soft key.

Figure 7-39 TOOL OFFSET, CUTTER COMPENSATION
Courtesy FANUC FA AMERICA

Figure 7-40 TOOL OFFSET, TOOL DATA Descriptions
Courtesy FANUC FA AMERICA

381

Remember that the question marks added to the fixed-form sentences will require the programmer to input data manually in their place.

- Enter the data listed below for tab FORM 1.

1. POSITION MOVE

   G?? G0 X???? Y????;

   G43 Z1.0 H?? D??;

   Z1.0 M08;

2. PROGRAM START

   G90 G80 G40 G49;

   G28 X0 Y0 Z0;

3. TOOL CHANGE

   T?? M6;

   S???? M3;

4. CYCLE END

   G0 Z????;

   X???? Y????

- Enter the data listed below for tab FORM 5.

1. PROGRAM END

   G91 G28 X0 Y0;

   M30;

   TOOL END

   G80 Z1.0 M9;

   G91 G28 Z0;

   M01;

- The data you choose for tabs 2, 3, and 4 can be entered later.

STEPS TO CREATE AN NC GUIDE MILLING PROGRAM

1. From the Edit mode, press the NEWPRG soft key (Figure 7-41). **Enter the desired program number** and press the CREATE soft key.

   The program number you just created will be highlighted.

   - Press the O LIST and then EDTCOM soft keys to add a descriptive program name (e.g., MC Example 1, part #) and press enter. Press the CLOSE soft key to return to the program display. The program number you just created will be highlighted.

   - Press the O LIST and then EDTCOM soft keys to add a descriptive program name


```
CREATE NEW PROGRAM
NEW PROGRAM NO. : █
KEY IN NEW PROGRAM NUMBER.
```
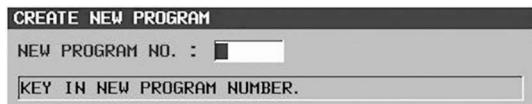
Figure 7-41 CREATE NEW PROGRAM (MC)
Courtesy FANUC FA AMERICA

Figure 7-42 Start Soft Key (MC)
Courtesy FANUC FA AMERICA

(e.g., MC Example 1, part #) and press enter. Press the OPEN soft key to return to the program display and open the file for editing.

2. To **define the part blank** type and size (required for simulation only).

   • Press the continuous menu key (right arrow of soft keys) one time.

   • Then press the START soft key (Figure 7-42). Use the cursor key to advance to the BLANK tab and then select 1. RETANGULAR BLANK FIGURE from the list (Figure 7-43). Press the SELECT soft key.

   • Key in the values as shown in Figure 7-44 for the stock blank.

   • With the last entry still highlighted, press the FIXFRM soft key and use the arrow keys to select the PROGRAM START Fixed Form. Press the INSERT soft key to add to the program.

   • With the last entry highlighted, use the arrow keys to select the TOOL CHANGE Fixed Form. Press the INSERT soft key to add to the program.

Upon insertion, the error IMPERFECT WORD MUST BE CHANGED is displayed. Use the arrow
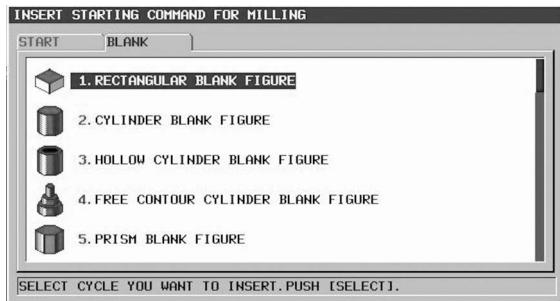
Figure 7-43 RECTANGULAR
BLANK FIGURE
Courtesy FANUC FA AMERICA
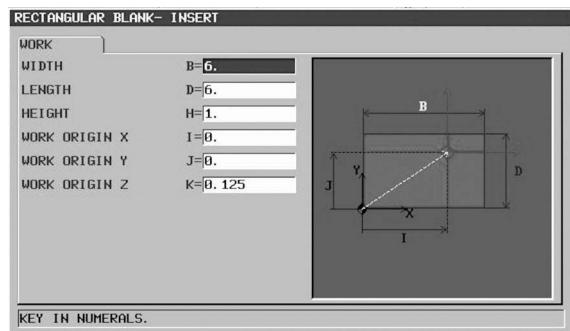
Figure 7-44 RECTANGULAR BLANK
WORK Dialog
Courtesy FANUC FA AMERICA
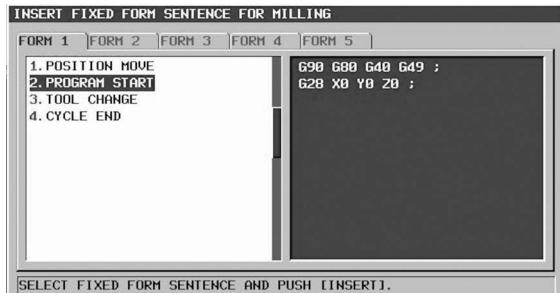
Figure 7-45 FIXED FORM, FORM 1,
PROGRAM START (MC)
Courtesy FANUC FA AMERICA

keys to highlight the word, as shown in Figure 7-47, and type in T1. Then press the ALTER function button on the controller. Position the cursor to the EOB character. Now follow the same procedure for the Spindle Speed (S????) based on a part material of medium carbon steel. Press the INSERT soft key to add to the program.

Figure 7-46 FIXED FORM, FORM 1, TOOL CHANGE (MC)
Courtesy FANUC FA AMERICA

- Position the cursor to the EOB character of the last line entered and press the FIXFRM soft key to select the POSITION MOVE Fixed Form.

- Press the INSERT soft key to add to the program. Use the arrow keys to highlight the first word G??. Type in G54 to identify the fixture offset used. Then press the ALTER function button on the controller. Position the cursor over the X and then the Y coordinate to enter starting X, Y values. Press the ALTER function button. Now follow the same procedure for the word H??. Enter H1 to represent the length offset used for Tool 1, the 3.0 Facemill; then press the ALTER function button. Lastly, ALTER the word D?? to D1 to represent the diameter offset for the 3.0 Facemill. Press the INSERT soft key to add to the program.

NOTE: The D# call in the program is required for proper Graphic display as well as Cutter Diameter Compensation, if used.

3. **Begin to program** with the Face Machining by pressing the CYCLE soft key. Use the arrow key to change to the FACE MACH. tab and choose 1. FACING (ROUGH) (Figure 7-48). Then press the SELECT soft key.

- Complete the CUT COND. tab of the dialog, as shown in Figure 7-49.

Note: Those fields marked with an asterisk are not required entries.

- Complete the DETAIL tab of the dialog, as shown in Figure 7-50, and press the INSERT soft key.

Figure 7-47 IMPERFECT WORD MUST BE CHANGED
Courtesy FANUC FA AMERICA
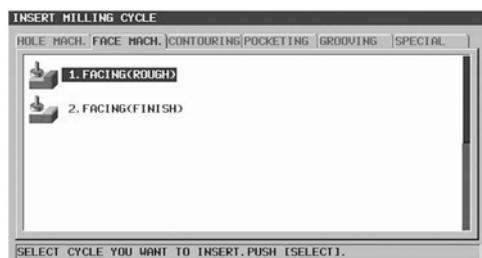
Figure 7-48 FACING (ROUGH) Cycle (MC)
Courtesy FANUC FA AMERICA
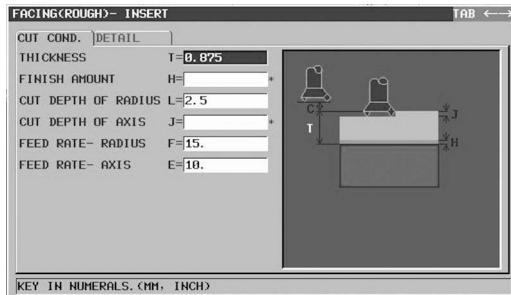
Figure 7-49 FACING (ROUGH) CUT
CONDITIONS (MC)
Courtesy FANUC FA AMERICA
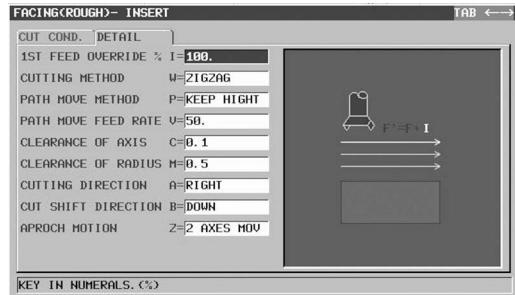
Figure 7-50 FACING (ROUGH) DETAIL
Dialog (MC)
Courtesy FANUC FA AMERICA

- Choose 1. XY-SQUARE FACING from the figure shapes available (Figure 7-51) and press the SELECT soft key.

- Complete the POS/SIZE dialog as shown in Figure 7-52 and press the INSERT soft key.

- To close the entry for the Facing Operation, place the cursor at the EOB character on the last line entered and press the END soft key. This action will automatically bring up the items stored in the END tab of the FIXED FORM SENTENCES stored earlier. Choose 2. TOOL END and press the INSERT soft key.

Figure 7-51 MILLING FIGURE, FACING
Courtesy FANUC FA AMERICA

To ensure accuracy and make corrections, each path can be simulated prior to continuing. To do this, press the SIMULATE soft key. Press the REWIND soft key. Press the START soft key and observe the toolpath. Press the STOP soft key to release the simulate mode and press the PROG function button to continue programming.
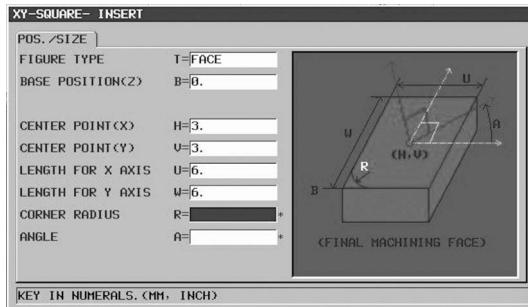
Figure 7-52
XY-SQUARE POS./SIZE
Courtesy FANUC FA AMERICA

PROGRAM THE STEP

For this and the STEP and POCKET portions of the program that follow, the 1.0" diameter Flat End-Mill (Tool #2) will be used.

- With the last entry highlighted, use the arrow keys to select the TOOL CHANGE Fixed Form. Press the INSERT soft key to add to the program.

- Use the same techniques used before to change the Tool Number and Spindle Speed values based on the program requirements.

- Use the same techniques used before to insert a POSITION MOVE to prepare for machining.

- Replace the IMPERFECT WORD statements for Work Coordinate System, XY coordinates, and to Tool #2 and D#2.

- Press the CYCLE soft key. Use the cursor keys to activate the CONTOURING tab of the MILLING CYCLE dialog. Choose 1. OUTER WALL CONTOURING (ROUGH) from the list of options and press the SELECT soft key (Figure 7-53).

- In the OUTER WALL CONTOURING (ROUGH) dialog complete the CUT CONDITIONS tab, as shown in Figure 7-54, and the DETAIL tab, as shown in Figure 7-55. Press the INSERT soft key when complete.

- To define the contour shape of the milling Figure, select item 2. XY-CIRCLE CONVEX from the options in the INSERT MILLING FIGURE dialog CONT. FIG. tab (Figure 7-56).
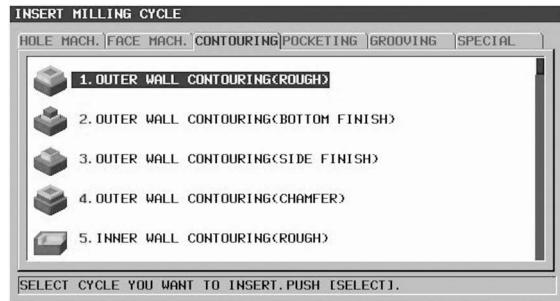
Figure 7-53 OUTER WALL CONTOURING
Courtesy FANUC FA AMERICA
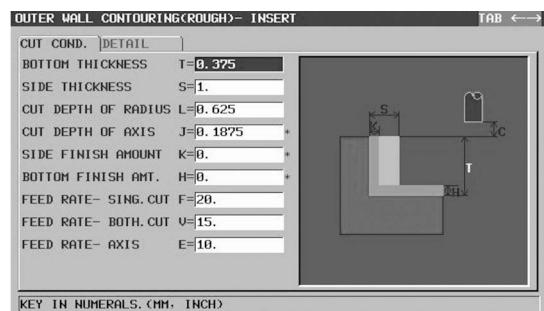
Figure 7-54 OUTER WALL CUT CONDITIONS
Courtesy FANUC FA AMERICA
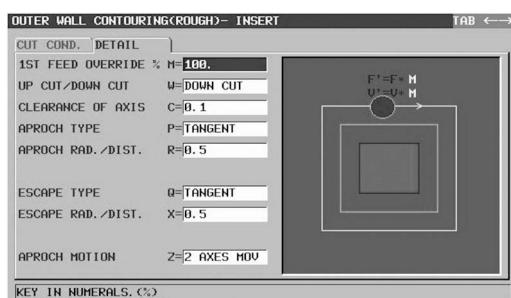
Figure 7-55 OUTER WALL DETAIL Dialog
Courtesy FANUC FA AMERICA
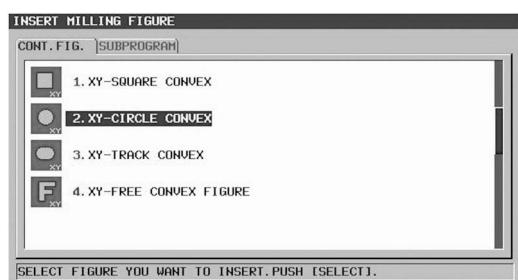
Figure 7-56 CONTOUR FIGURE, XY-CIRCLE CONVEX
Courtesy FANUC FA AMERICA

- Press the SELECT soft key to input the Position and Size. Complete the entries per Figure 7-57. Press the INSERT soft key when done.

- With the cursor on the just finished block, press SIMULATE and then the REWIND soft key. Press the START soft key and observe the toolpath. Press the STOP soft key to release the simulate mode and press the PROG function button to continue programming.

PROGRAM THE POCKET

Because the same tool will be used for the pocket, there will be no need for a tool change. Instead, a CYCLE END move will suffice.

- Place the cursor at the end of the last block entered and press the FIXFRM soft key. Select CYCLE END from the list.

- Use the same techniques used before to ALTER axes values based on the program requirements. In this case, enter Z1.0 and 3.0, 3.0 for the X and Y axes.

- Press the CYCLE soft key and use the arrow key to select the POCKETING tab. From the list, choose 1. POCKETING (ROUGH) and press the SELECT soft key.

- Complete the CUT COND. Tab, per Figure 7-58.

- Complete the POCKETING (ROUGH) DETAIL tab per Figure 7-59, Set the CUT DEPTH METHOD in the POCKETING dialog on the DETAIL tab to X=HELICAL. Note the two soft key choices of STRAIGHT or HELICAL.
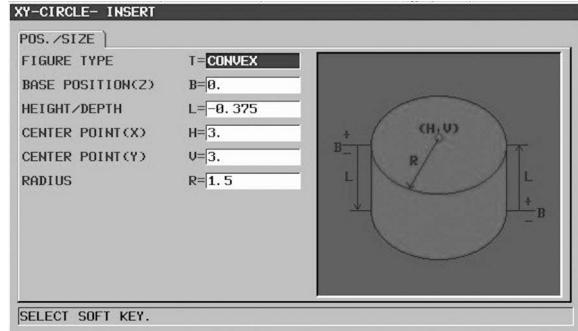
Figure 7-57 XY-CIRCLE, POS./SIZE
Courtesy FANUC FA AMERICA
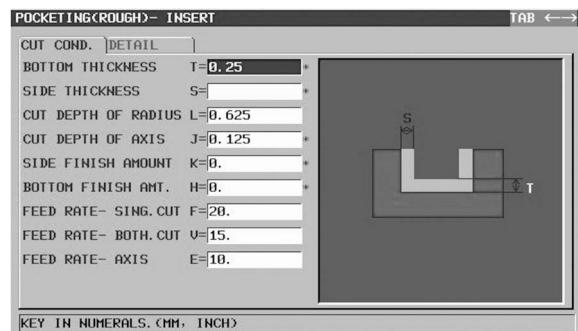
Figure 7-58 POCKETING (ROUGH),
CUT CONDITIONS
Courtesy FANUC FA AMERICA
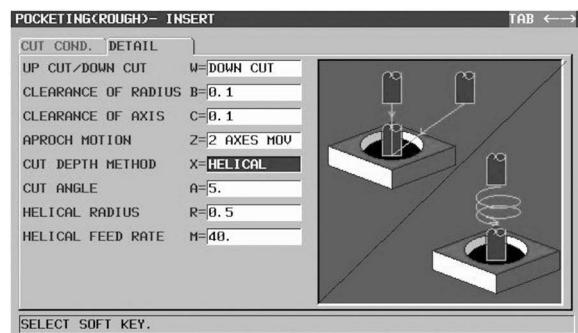
Figure 7-59 POCKETING (ROUGH),
CUT DEPTH METHOD
Courtesy FANUC FA AMERICA

- Set the CUT ANGLE A=5. per Figure 7-60.

- Set the HELICAL RADIUS R=0.5 per Figure 7-61.

- The feed rate for the HELICAL engagement may be adjusted also. Press the INSERT soft key when the data are set.

- Establish the POCKET FIGURE shape, size, and POSITION. Select item 2. XY-CIRCLE CONCAV from the options in the INSERT MILLING FIGURE dialog CONT. FIG. tab (Figure 7-62).

- With the cursor on the just finished block, press the SIMULATE soft key. Press the START soft key and observe the toolpath. Press the STOP soft key to release the simulate mode and press the PROG function button to continue programming.

- With the cursor at the EOB character of the last line entered, use the Continuous Menu button to find the FIXFRM soft key and select it. Use the arrow key to select the Form 5 tab. From the list, select item 2. TOOL END; then press the INSERT soft key.

PROGRAM THE SPOT DRILL

To perform the drilling operations, we first must spot drill all of the holes locations. The following steps outline how to do this.

Figure 7-60 POCKETING (ROUGH),
CUT ANGLE
Courtesy FANUC FA AMERICA
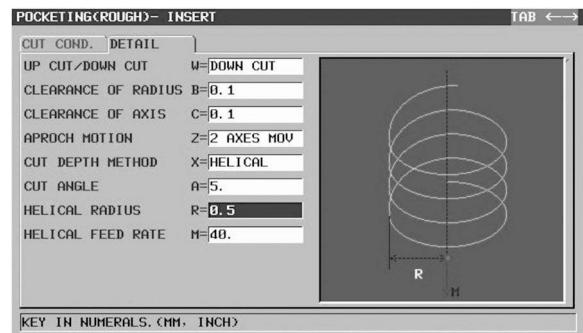
Figure 7-61 POCKETING (ROUGH),
HELICAL RADIUS
Courtesy FANUC FA AMERICA
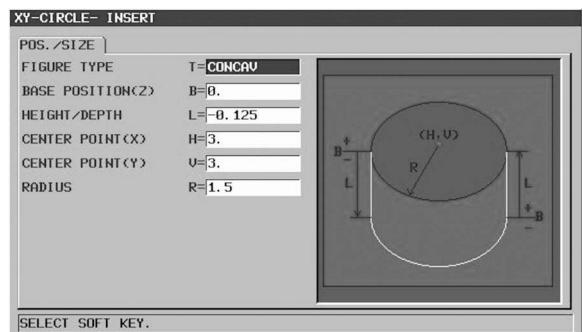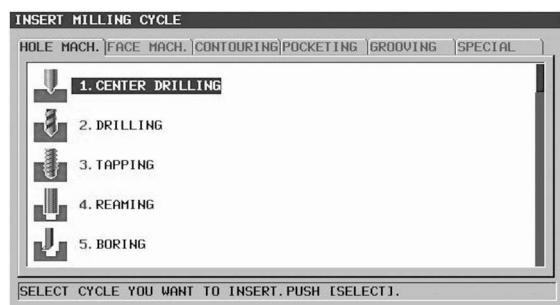
Figure 7-62 POCKETING, XY-CIRCLE,
POS./SIZE
Courtesy FANUC FA AMERICA

- Select the FIXFRM soft key and select item 3. TOOL CHANGE from the FORM 1 list. Press the INSERT soft key.

- Use the skills you learned earlier to change the IMPERFECT WORD, T?? to T3, and the S???? to S2000.
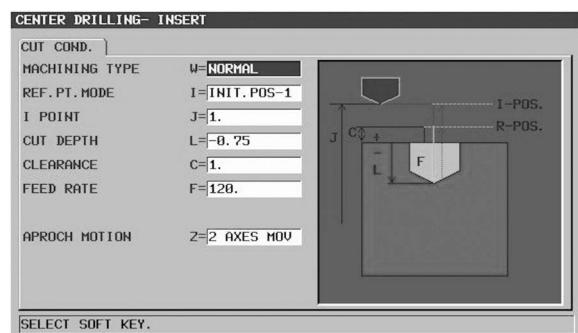
- Select the FIXFRM soft key and select item 1. POSITION MOVE. Then press the INSERT soft key.

- Use the skills you learned earlier to change the IMPERFECT WORD values to: G54, X0, Y0, H3, and D3 respectively.

- Use the Continuous Menu button to find the CYCLE soft key and press it.

- Under the HOLE MACH. tab make sure that item 1. CENTER DRILLING (Figure 7-63) is highlighted and press the SELECT soft key.

- Complete the CUT CONDI-TIONS dialog, per Figure 7-64.

Note: Select the NORMAL soft key to produce a simple G81 drilling cycle. If DWELL is selected, then a G82 cycle will be output that is ideal for Spot-Facing.

- Press the INSERT soft key.

Figure 7-63 HOLE MACHINING, CENTER DRILLING
Courtesy FANUC FA AMERICA

Figure 7-64 CENTER DRILLING CUT CONDITIONS
Courtesy FANUC FA AMERICA

- Choose item 5. XY-RECTANGLE POINTS from the HOLE POSI. tab (Figure 7-65) and press the SELECT soft key.

- Complete the XY-RECTANGLE POINTS dialog, as shown in Figure 7-66, and press the INSERT soft key.

- Check the toolpath for accuracy by simulating prior to continuing. Press the SIMULATE soft key. Press the REWIND soft key. Press the START soft key.

Figure 7-65 MILLING FIGURE, HOLE POSITION
Courtesy FANUC FA AMERICA

Figure 7-66 XY-RECTANGLE POINTS HOLE POSITION
Courtesy FANUC FA AMERICA

For the set of tapped holes, a different pattern and depth is required using the same tool.

- Place the cursor at the EOB character for the last entry and select the Continuous Menu button until CYCLE is available again; then select it.

- Choose item 1. CENTER DRILLING and press the SELECT soft key.

- For the Cutting Conditions tab, change the CUT DEPTH to –.250 and press the INSERT soft key.

- Choose item 6. XY-CIRCLE POINTS from the list of options and press the SELECT soft key (Figure 7-67).

- Complete the XY-CIRCLE POINTS dialog per Figure 7-68. Press the INSERT soft key.

- Place the cursor at the EOB character and press the START soft key. Choose item 4. CYCLE END from the list of options and press the INSERT soft key.

- Replace the IMPERFECT WORD items with Z1.0, X6.0, Y6.0.
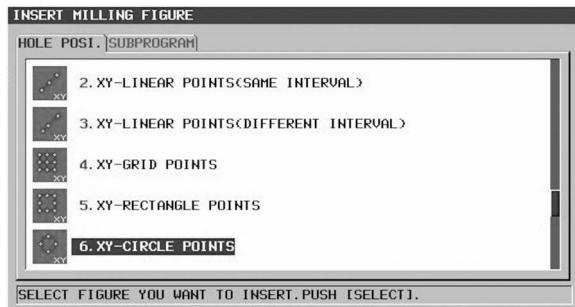
Figure 7-67 HOLE POSITION Dialog
Courtesy FANUC FA AMERICA
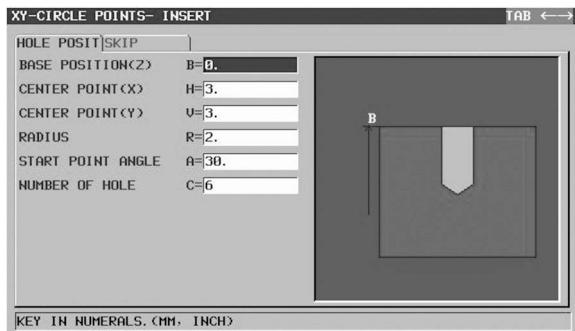
Figure 7-68 XY-CIRCLE POINTS, HOLE POSITION
Courtesy FANUC FA AMERICA

PROGRAM THE DRILL

The next set of instruction will be for the four, .531 diameter corner holes. With the cursor at the EOB character of the last block entered, press the START soft key.

- Select item 3. TOOL CHANGE and press the INSERT soft key.

- Replace the IMPERFECT WORD items with T4 and S1888.

- Use the Continuous Menu button to return to the START soft key and select it.

- From the options, select 1. POSITION MOVE and press the INSERT soft key.

- Replace the IMPERFECT WORD items with G54, X0.0 Y0.0, H4, and D4; then press the INSERT soft key.

- Press the CYCLE soft key and choose item 2. DRILLING from the list of options (Figure 7- 69). Press the SELECT soft key.

- Select PECK for the MACHINING TYPE using the soft key shown in Figure 7-70.
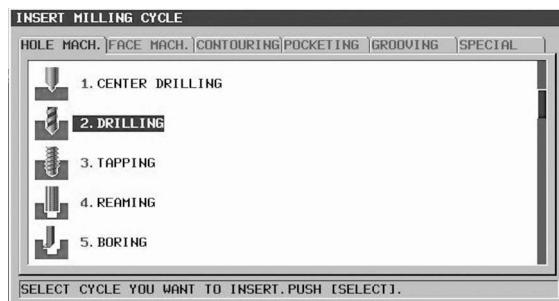
Figure 7-69 HOLE MACHINING,
DRILLING
Courtesy FANUC FA AMERICA

Figure 7-70 PECK DRILLING Soft Keys
Courtesy FANUC FA AMERICA

- Complete the DRILLING CUT CONDITIONS dialog, as shown in Figure 7-71. Do not change anything in the DETAIL tab. Press the INSERT soft key.

- From the HOLE POSI tab, choose item 5. XY-RECTANGLE POINTS pattern.

- Complete the fields exactly as you did for Spot Drilling the hole pattern in Figure 7-66

Figure 7-71 DRILLING CUT CONDITIONS
Courtesy FANUC FA AMERICA

(**XY-RECTANGLE POINTS Hole Position**). Then press the INSERT soft key.

- Place the cursor at the EOB character and press the START soft key. Choose item 4. CYCLE END from the list of options and press the INSERT soft key.

- Replace the IMPERFECT WORD items with Z1.0, X6.0, Y6.0.

PROGRAM THE TAP DRILL

The next set of instructions will be for the six-hole bolt circle with 3/8-16 threaded holes. With the cursor at the EOB character of the last block entered, pre s the START soft key.

- Select item 3. TOOL CHANGE and press the INSERT soft-key.

- Replace the IMPERFECT WORD items with T5 and S3200.

- Use the Continuous Menu button to return to the START soft key and select it.

- From the options, select 1. POSITION MOVE and press the INSERT soft key.

- Replace the IMPERFECT WORD items with G54, X0.0 Y0.0, H5, and D5 and press the INSERT soft key.

- Press the CYCLE soft key and choose item 2. DRILLING from the list of options (Figure 7-72). Press the SELECT soft key.

- Select PECK for the MACHINING TYPE using the soft key shown in Figure 7-70.

- Complete the entries in the Cutting Condition tab per Figure 7-72.
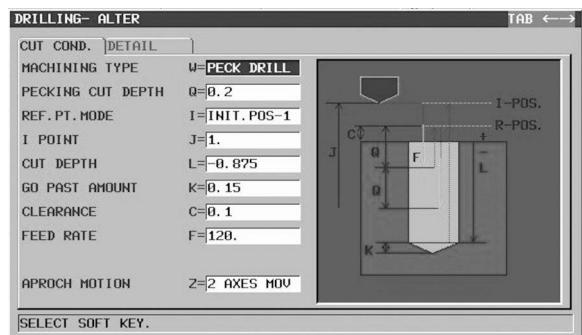
Complete the entries in the XY-CIRCLE POINTS dialog HOLE POSIT tab per Figure 7-73 and press the INSERT soft key.

Figure 7-72 TAP DRILLING
CUT CONDITIONS
Courtesy FANUC FA AMERICA

Figure 7-73 TAP DRILLING,
HOLE POSITION
Courtesy FANUC FA AMERICA

- Place the cursor at the EOB character and press the START soft key. Choose item 4. CYCLE END from the list of options and press the INSERT soft key.

- Replace the IMPERFECT WORD items with Z1.0, X6.0, Y6.0.
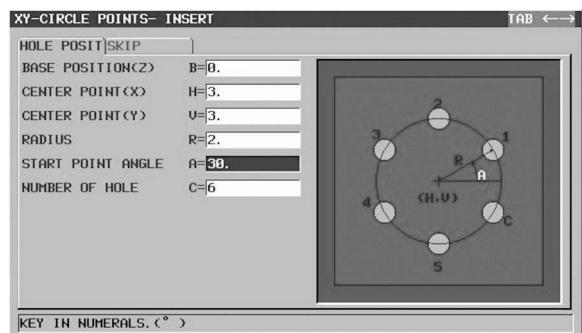
PROGRAM THE TAP

The last set of instructions will be for tapping of the six 3/8-16 threaded holes. With the cursor at the EOB character of the last block entered, press the START soft-key.

- Select item 3. TOOL CHANGE and press the INSERT soft key.

- Replace the IMPERFECT WORD items with T6 and S1000.

- Use the Continuous Menu button to return to the START soft key and select it.

- From the options, select 1. POSITION MOVE and press the INSERT soft key.

- Replace the IMPERFECT WORD items with G54, X0.0 Y0.0, H6, and D6; then press the INSERT soft key.

- Press the CYCLE soft key and choose item 3. TAPPING from the list of options (Figure 7-74). Press the SELECT soft key.

- Select NORMAL for the MACHINING TYPE, using the soft key shown in Figure 7-75.

- Complete the entries in the Cutting Condition tab per Figure 7-76.

- Complete the entries in the XY-CIRCLE POINTS dialog HOLE POSIT with the same values as for the Tap Drilling in Figure 7-73; then press the INSERT soft key.

- With the cursor at the EOB character of the last line entered, use the Continuous Menu button to find the END soft key and select it. Use the arrow key to select item 2. TOOL END from the list and press the INSERT soft key.

- Press the END soft key once again and select 1. PROGRAM END from the list. Then press the INSERT soft key.

- Check the toolpath for accuracy by simulating. Press the SIMULATE soft key. Press the REWIND soft key. Press the START soft key.

This example demonstrates one way that a simple Milling part can be programmed within the FANUC NC Guide software. At the control using MANUAL Guide i, the steps will be nearly identical. Check the Operator and Programming manuals specific to the machine being used for exact details.
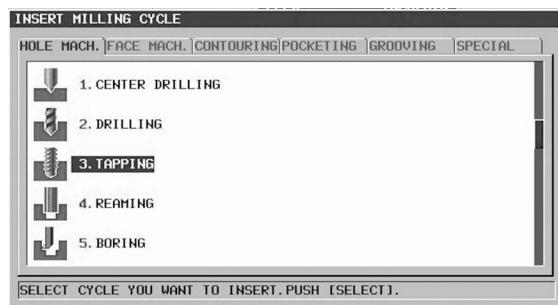
Figure 7-74 TAPPING CYCLE
Courtesy FANUC FA AMERICA
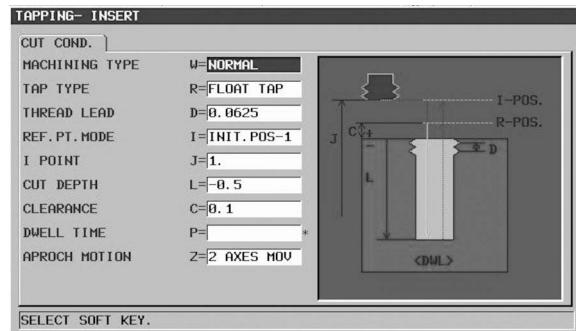
Figure 7-75 TAPPING, CUT CONDITIONS
Courtesy FANUC FA AMERICA
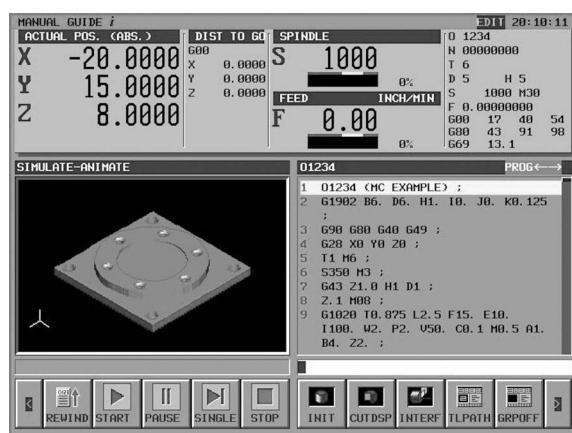
Figure 7-76 Milling Example SIMULATED
Courtesy FANUC FA AMERICA

Part 7 Study Questions

1. What does the acronym PWE stand for?

2. What is the difference between NC Guide and Manual Guide i?

3. When defining VIRTUAL TIP Tool Data for turning, entry of the required information sets the cutting direction of the tool.

      T or F

4. Why are the CUT ANGLE and NOSE ANGLE important when defining turning tools?

5. What is the purpose for FIXED FORM SENTENCES?

6. Why are question marks present in some of the FIXED FORM SENTENCES?

7. When the alarm IMPERFERCT WORD MUST BE CHANGED is encountered, what key or button is used to change code in the program?

8. When defining the paths for roughing of an Outer Diameter, each depth-of-cut pass must be programmed manually.

      T or F

9. Which dialog is used to enter the tool diameter for milling tools?

10. Portions of a program can be copied and pasted into later parts of the program to save data entry steps.

      T or F

Part 7 FANUC NC Guide Programming

11. SIMULATION on the NC Guide software requires that the part BLANK be identified at the beginning of the program.

       T or F

12. What dialog is used to set the location of a programmed figure?

13. What soft key is used to access the settings to control each depth-of-cut when drilling a deep hole on the mill?

14. For practice, use the skills learned in this chapter to program examples from Chapter 3 of this book.

15. For practice, use the skills learned in this chapter to program examples from Chapter 4 of this book.

PART 8

Mazatrol Conversational Programming

Figure 8-1 Mazatrol Control Panel
Courtesy Innovative Precision LLC

OBJECTIVES:

1. Explain the advantages of conversational programming.

2. Become familiar with the Mazatrol conversational programming language.

3. Learn terminology specific to Mazatrol conversational programming language.

4. Become familiar with typical MAZAK controller screens and menus.

5. Demonstrate how to create a CNC Turning Center program using Mazatrol conversational programming language.

6. Demonstrate how to create a CNC Machining Center program using Mazatrol conversational programming language.

7. Become familiar with offline programming software used to create Mazatrol programs.

WHAT IS CONVERSATIONAL PROGRAMMING?

For many years, the concept of programming the CNC machine tool at the controller was thought of as inefficient and tedious. When orders of a small lot size were to be produced, the choice was almost always manual machines. This thinking is not the case today, largely because of the advances in conversational programming.

Conversational programming is becoming used more widely throughout the industry and is available as standard on most recently designed machine tool controllers. Its major advantage is that it gives machinists the ability to write programs at the machine quickly and easily. Typically, the process includes a sequence of questions the machinists or programmers must answer, sometimes called *question answer format* or *prompting*. As these questions are answered, the program is constructed. Most controls with this capability also allow machinists and programmers to graphically check the tool path to verify the program. If the program has flaws or missing information, the controller will not execute the tool path and the programmers must remedy the problem. When program errors do occur, an alarm number will appear on the screen indicating what the problem is and where it occurs in the program. This method is obviously better for finding errors than actually cutting a part.

Another capability of conversational programming is its feature to perform calculations when programming data are missing from the engineering drawing. The programmer constructs intersection points or tangency points, and with this information, the controller computes the desired geometry. On some controllers, Feeds and Speeds can be calculated automatically based on workpiece material and cutting tool material. The data needed for this calculation is stored in the controller memory in the cutting condition parameters.

Conversational "shop floor" programming uses the concept of operator prompting combined with a Graphical User Interface (GUI). Questions throughout the programming process prompt the user for information necessary to complete the part program. Icons accessed through the function buttons on the controller identify machining operations, for example, Point Machining, Line Machining, and Face Machining (Figure 8-2).

Much like CAD/CAM, the programming process resembles recreation of the part geometry by constructing the shapes using lines, arcs, and points combined with other features.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	SHAPE CHECK

Figure 8-2 Mazatrol Function Buttons

This information is used in combination with tool identification parameters and cutting condition parameters to generate the tool path code needed to control the machine. The programmer has the added functions of the controller's ability to calculate for unknown coordinate values and have them automatically inserted into the program where they are needed. In most cases, no calculator is needed for trigonometric calculations. These constructions resemble the formerly popular Automatic Programmed Tool (APT) method of programming. The actual program the machine executes is still a G-Code format, but the operator may never see the actual code on the display.

This type of programming, combined with the ability to access G-Code subprograms, has tremendous power. Mazatrol, a conversational programming system offered on all MAZAK machine tools, allows "G-Code similar" programming within its conversational language, in what is called a Manual Programming Process. The acronym used to call this type of program process for Mazatrol is MNP, where MN stands for Manual and P for Programming.

There are many different conversational languages available for programming. One of the main complaints has been the lack of standardization among the various machine tool builders. MAZAK's Mazatrol language has been at the forefront of the industry in conversational programming for decades and has a proven track record of success. The focus of this chapter will be on this language only. Other languages contain similar techniques that accomplish nearly the same result.

Just as with any programming endeavor, you must be well prepared. To create a Mazatrol program, the steps used by the programmer are very similar to those used in manual programming. Careful examination of the technical part drawing for work holding considerations and tool selection must take place prior to preparation of the part program. With that in mind, it is most efficient to establish a tool file within the Mazak controller prior to programming. This file should contain a representation of the tools that are available in your shop. From this information, one of the more powerful aspects of Mazatrol conversational programming can be used to automatically develop tools used in the program. This tool file, when properly defined, can also be used to automatically calculate the proper feeds and speeds used for machining. Once this file is set up, programming may begin.

When the program is complete, the tool path must be verified by graphical simulation. At this point, if all checks well, the operator takes over for the measuring of tool and work offsets as well as one final program test by dry run. Finally, the first part of CNC machining begins.

TURNING CENTER PROGRAM CREATION

During the programming process, many unique abbreviations and acronyms are used to simplify prompting and input. Chart 8-1 summarizes some of the acronyms encountered in the sequence of creating Mazatrol conversational Turning Center programs.

Chart 8-1 Turning Center Abbreviations and Acronyms

Abbreviation/Acronym	Description
WKNO	Workpiece Number
MAT	Material
FC	Ferrous Cast Iron
FCD	Ferrous Cast Ductile Iron
S45C	Low Carbon Steel
SCM	Alloy Steel
SUS	Stainless Steel
AL	Aluminum
CU	Copper
CB ST	Carbon Steel
ALOY	Alloy Steel
CASIR	Cast Iron
9310	9310 Alloy Steel
BRASS	Brass
A2	Tool Steel
MAX	Maximum
MIN	Minimum
OD	Outside Diameter
ID	Inside Diameter
RPM	Revolutions per Minute
FIN-X	Finish Allowance — X-axis
FIN-Z	Finish Allowance — Z-axis
BAR	Bar Machining, e.g., solid Barstock
CPY	Copy Machining, e.g., net shape material, casting, etc. Uniform material all around, all surfaces
CNR	Corner Machining, e.g., re-machining of corners where the tool cannot reach due to tool geometry and more
EDG	Edge Machining
FCE	Face
BAK	Back
THR	Threading Inside Diameter (ID) or Outside Diameter (OD)
GRV	Grooving, ID, OD, Face or Back
MTR	Workpiece Shape, a user-defined arbitrary shape that is other than bar or net shape and requires non-uniform material removal
DRL	Drill
MNP	Manual Program Unit
M-Code	Miscellaneous codes, e.g., coolant M8
FCE	Face, e.g., Edge FCE or BAR FCE
CPT-X	Cutting Point — X-axis
CPT-Z	Cutting Point — Z-axis
RV	Surface Speed for Rough Cut (V = Velocity)
FV	Surface Speed for Finish Cut (V = Velocity)
V ROUGHNESS	Surface Roughness determined by in/rev setting
R-FEED	Roughing Feed rate in/rev or mm/rev
R-DEP	Roughing Maximum Depth of Cut
R-TOOL	Rough Tool No.
F-TOOL	Finish Tool No.
ID CODE	Tool Identification Code for Spare Tool Usage
LIN	Linear Feed Move
TPR	Tapered Feed Move
S-CNR	Start <CNR-C> or <CNR-R> This means Start Corner where –C = Chamfer and –R = Radius
SPT-X	Geometry Starting Point — X-axis
SPT-Z	Geometry Starting Point — Z-axis
FPT-X	Geometry Final Point — X-axis
FPT-Z	Geometry Final Point — Z-axis
F-CNR	Final <CNR-C> <CNR-R> Necking, Final corner chamfer or radius where –C = Chamfer and –R = Radius
CTR	Center Point for Radius Programming
BAK	Back Machining
CHAMF	Chamfer for thread ending
ANG	Angle of thread
HGT	Thread Height
V	Velocity Cutting Speed Threading
END	End Unit
SHIFT	Second/third part, etc., shift amount
TPC	Temporary Parameter Change / Toolpath Control

CONVENTIONS

For this section, the following text format convention is used. For the MENU selection, the letters will be in CAPITALS while the user prompts will be in capital *ITALICS*. Mazatrol acronyms are given in capital letters and **BOLD TYPE**.

The following discussion provides brief descriptions of the general programming process for turning centers. In order to begin, the control must be in the program-editing mode and a work number (program number) must be identified.

- Press the soft key labeled "Work No." and key in the desired program number; then press Input.

Note: There is no need for the letter address O to precede the program number with Mazatrol programs.

Before any programming can take place, you must determine the type of program needed: EIA/ISO or Mazatrol. All MAZAK machines use Mazatrol as their standard program type, with EIA/ISO (G-Code) on some older generation machines as an optional

feature. Turning center programs are made up of these four basic parts: a Common Data Process, Machining Process, Sequence Data, and an End Process.

COMMON DATA PROCESS

The information at the head or top line of the program applies to the entire program. The programmer is prompted to answer the following questions for this common data.

Workpiece Material <Menu>

The controller memory is preset with standard materials of Carbon Steel, Alloy Steel, Cast Iron, Aluminum, and Stainless Steel available in the menu. The choice of material affects the automatic calculation of cutting feeds and speeds throughout the program. It is possible to add user-defined materials to the cutting condition parameters if the material needed is not available.

Max. Outer Dia. Of Workpiece

This value input is dependent upon the diameter of the raw workpiece.

Note: If you input a value that exceeds this diameter, an alarm will result on the controller display, which will prevent execution of tool path verification and automatic operation.

Min. Outer Dia. Of Workpiece

If the workpiece geometry is of solid bar stock, this value may be set to zero. If an inner diameter exists, such as with tubing, the programmer must input this value. Doing this prevents the generation of tool path where material is nonexistent.

Workpiece Length

The overall length of the workpiece along the Z-axis, including the clamping amount, should be entered for this value. It must be at least the maximum machined length. If a programmed value exceeds this length, the controller will set off an alarm on the display, preventing execution of tool path verification and automatic operation. This value is not meant to represent the extension value for the setup of the part in the chuck jaws.

Max. Spindle RPM Limit (rpm)

Inputting a specific r/min in this field enables the programmer to limit the spindle RPM to a predetermined amount (G50 in G-Code programming). If no value is input into this data field, the controller will execute the maximum spindle RPM when the centerline of the X-axis is reached. In some cases, this maximum RPM may be undesirable.

Finish Allowance

The amount of material to be left for a finishing pass in the X-axis is input at this time. This value is input in consideration of the diameter of the workpiece. For example, if a value of .040 inch is input, the amount taken off the diameter equals .080 inch for the finishing pass.

Finish Allowance-Z

The amount of material to be left for a finishing pass in the Z-axis is input at this time.

Stock Removal of Workface

It is common to machine material from the face of the workpiece in order to attain a finished surface that establishes the Z-axis Workpiece Zero for the part. This amount is dependent upon the condition of the material and programmer preference.

Machining Process

In this section of the program, the individual machining process data are identified in order to complete the workpiece definition—in other words, the type of machining that is to be done. In Figure 8-3, the choices are **BAR, CPY, CNR, EDG, THR, GRV, WORKPIECE SHAPE,** and **END**.

Bar (**BAR**) machining is used for outside diameter (OD) or inside diameter (ID) turning and boring. Copy (**CPY**) machining is used for OD or ID machining of existing geometries like castings or forgings, where a uniform amount of material is to be removed and is other than solid bar stock. Corner (**CNR**) machining is used when additional cutting tools are needed to finish corners that cannot be cut because of tool geometry limitations. Edge (**EDG**) machining is used to perform machining on the face of the workpiece. Thread (**THR**) is for machining of external and internal screw threads. Grooving (**GRV**) is for machining of external and internal grooves. Workpiece shape machining is similar to **CPY** except the material removal shape does not need to be uniform. **END** is used to end the program. The arrow keys at the right of Figure 8-3 offer additional options of Drilling, Tapping, and Manual Programming (i.e., G-Code within the Mazatrol program).

Once a selection is made for the type of machining operation, more information is needed to identify how to apply it. In Figure 8-4, **BAR** machining has been selected and a new set of menu choices are displayed. Those items in bold are captured-type cuts. For example, **OUT** is used to perform general OD machining and **IN** is used for general ID machining like boring.

Once the type of machining is selected (**BAR)**, the necessary related information is as follows: Feeds and Speeds are automatically calculated by pressing a soft key (AUTOSET); they are based on parameter information directly associated to the selected cutting tool and workpiece material identified in the Common Data Process. Other information includes tool selection for roughing and finishing cycles; the Starting Point in X (**SPT-X**); the Starting Point in Z (**SPT-Z**); the Finish Point in X (**FPT-X**) and; the Finish Point in Z (**FPT-X**).

Sequence Data

The finished workpiece shape is identified by the input of point data until the desired geometry exists using lines (**LIN**), tapers (**TPR**), arcs, chamfers, and fillets, limited

Figure 8-3 Machining Process Menu

Figure 8-4 BAR Machining Menu

LIN ▼	TPR ▼	⌒	⌣		CENTER			SHAPE END	
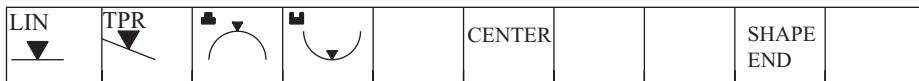
Figure 8-5 Sequence Data Menu

only by the tool geometry configuration. The same type data are necessary for internal bar machining. In Figure 8-5, the two types of arcs shown represent convex and concave shapes, respectively, and the CENTER menu selection is needed to identify the arc center point.

When all the geometric data are entered and the shape is defined properly, the SHAPE END menu key is pressed to end the process. The remainder of the program is constructed in the same manner until the workpiece geometry is complete.

END PROCESS

This process ends the program (similar to M30 in G-Code programming). It offers the opportunity to the programmer to set a counter for the number of workpieces to be machined; set the return position of the turret after machining ends; identify the next program number to machine (including whether to continue the same program repeatedly or not) and how many repetitions; and, set the shift amount for the coordinates system.

Shape Check

Once the program data are entered, you may perform a Shape Check of the geometry to verify its accuracy. The controller will not allow the shape check if there are serious problems with the geometry definition. An alarm will be displayed on the screen identifying the program number, process number, and sequence number of the mistake. Performing a shape check will draw the finished workpiece geometry on the screen in two-dimensional form.

Tool Path Verification

Another step completed before running the part is Tool Path Verification, which may be used to check the part geometry and tool path (Figure 8-6). The newest controllers
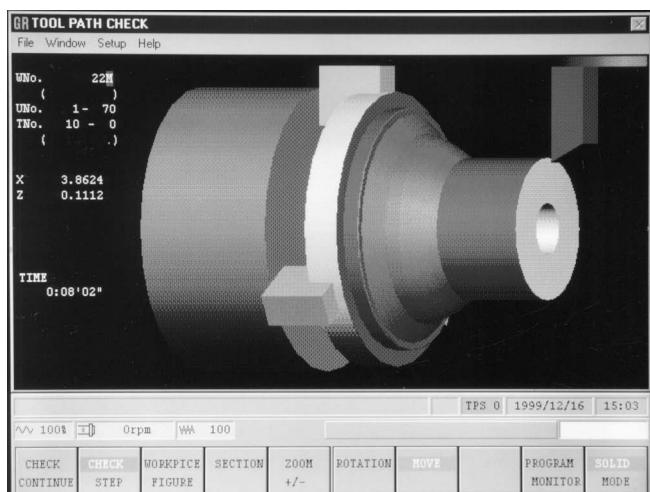
Figure 8-6 Tool Path Check
Courtesy MAZAK Corp

are equipped with solid model rendering of the workpiece raw stock configuration. The programmer can simulate on-screen the actual machining of the workpiece in real-time by pressing CHECK CONTINUE from the menu.

If an area of the graphic is hard to see because of its size, you can ZOOM into that area and magnify it for better viewing. Tool shapes are graphically simulated as well, allowing an excellent visual aid for correcting any program problems.

Often, a workpiece has internal features that are difficult to see even with solid modeling. The newest controls have

Figure 8-7 Tool Path Section View Check
Courtesy MAZAK Corp

the graphical capability to section the workpiece, allowing a full visual representation that offers even more assistance to the programmer for verifying programs (Figure 8-7).

Once these verification steps are complete, you may begin the first article of CNC automatic operation.

MACHINING CENTER PROGRAM CREATION

During the programming process, many unique abbreviations and acronyms are used to simplify prompting and input. Chart 8-2 summarizes some of the acronyms encountered in the sequence of creating Mazatrol conversational Machining Center programs. When creating Mazatrol programs, the program display will look similar to Figure 8-8.

This section provides brief descriptions of the programming process for machining centers: The control must be in the program-editing mode and a work number (program number) must be identified.

Note: There is no need for the letter address O to precede the program number with Mazatrol programs.

Before any programming can take place, determine the type of program to create—either EIA/ISO or Mazatrol. All MAZAK machines use Mazatrol as their standard with EIA/ISO (G-Code) and on some older generation machines as an optional feature. The construction of a Mazatrol machining center program contains these basic parts: a Common Data Unit, identification of a Coordinate System, the Machining Units and their Sequence Data, and an End unit.

Figure 8-8 Mazatrol Program Display
Courtesy MAZAK Corp

Chart 8-2 Machining Center Abbreviations and Acronyms

Abbreviation/Acronym	Description
CST IRN	Cast Iron
DUCT IRN	Ductile Iron
CBN STEEL	Carbon Steel
ALY STEEL	Alloy Steel
STNLESS	Stainless Steel
AL	Aluminum
COPPER	Copper
WPC	Workpiece Coordinate
ATC	Automatic Tool Change
UNo.	Unit Number
ARBITRY	Arbitrary
FACE MIL	Face Mill
TOP EMIL	Top End Mill
PCKT MT	Pocket Mountain
PCKT VLY	Pocket Valley
SRV-Z	The amount of stock removal in Z
SRV-R	The amount of radial stock removal in X-Y
BTM	Bottom
RGH	Surface roughness
WAL	Wall roughness (Same number system applies)
WID-R	Width of Cut (radial)
CHMF	Chamfer amount
FIN-Z	Finish allowance for the Z-axis
FIN-R	Finish allowance for the X-Y axis
NOM Ø	Nominal diameter of the tool
PRI-NO.	Priority number for the tool
APPROCH-X	Approach point for the tool in the X-axis
APPROCH-Y	Approach point for the tool in the Y-axis
CW CUT	Clockwise cutting direction
CCW CUT	Counterclockwise cutting direction
ZFD*	Z Feed; Move to depth of cut at linear federate in the Z-axis. Rapid may be selected here, but the default setting is G01. **Caution should be used here because a rapid movement into solid material will break the tool and possible damage the machine.**
DEP-Z	Depth of Cut
C-SP	Cutting Speed
FR	Feedrate
SNo. 1	Sequence Number
X BI-DIR	BI-Directional cutting both directions along the X-axis
Y UNI-DIR	UNI-Directional (Single) cutting along the Y-axis
CN1	Corner 1
LNE	Line
CHMF	Chamfer
CTR-DR	Center Drill
PT	Point
INIT	Initial
R	Reference or Radius

COMMON DATA UNIT

The information at the head of the program applies to the entire program. The programmer is prompted to answer the following questions for this common data.

Material <Menu>?

The controller is preset with standard materials of Cast Iron, Ductile Cast Iron, Carbon Steel, Alloy Steel, Stainless Steel, Aluminum, and Copper Alloy. Your choice, combined with tool material, affects the automatic calculation of cutting feeds and speeds throughout the program. It is possible to add other materials to the cutting condition parameters if the material needed is not available.

Initial Point Z (CLEARANCE)?

This value identifies the initial Z-clearance position where all of the tools will move, at rapid traverse, before machining begins. In G-Code programming, this value represents the same thing as the Initial Reference Plane. A common reason for setting the initial point at a particular height is to provide for clearance of work holding clamps.

ATC Mode Zero Return <Z.X+Y:0, X+Y+Z:1>?

The choice of the Zero Return method establishes how the tool is returned to the Automatic Tool Change (ATC) position for a tool change. For example, a selection of 0 returns the Z-axis to the tool change position and then, the X and Y simultaneously. This choice is the safest in most cases; however, if no clearance issues are evident, then simultaneous movement of X, Y, and Z may be chosen by selection of 1 here.

Note: this movement is always at rapid traverse.

MULTI Mode <Menu>?

There are three choices for establishing work coordinate systems: MULTI OFF, MULTI 5 × 2, AND OFFSET TYPE. When MULTI OFF is selected, the next unit is started and this unit is commonly used for the Workpiece Coordinate system or WPC. Values are set for the location of the origin of the workpiece coordinate system just as with G-Code programs. Additionally, offsets G54–59 may be used.

MULTI 5 × 2

When several workpieces need to be machined in the same setup selection of MULTI 5 × 2, allows for multiple repetitions (up to ten) of the same program. Basically you are creating rows and columns. The use of MULTI FLAG is required, in conjunction with this program call, to identify if each position in the grid is or is not to be machined (allows for omitting some positions). Use of the Multi 5 × 2 technique is limited to each of the linear repetitions having corresponding distances from part to part. For example, the distance between all X-axis repetitions must be equal. Also although Y-axis distances may be different from the values in X, they must be equal in Y (forming a grid.)

Offset Type

Using this type of coordinate system arrangement allows the arbitrary location of the Workpiece Zero or origin for multiple workpieces within the working envelope of the machine. The locations may be random and polar rotation of the coordinate system is allowed. Up to 10 individual offsets are allowed.

COORDINATE SYSTEM

In this unit, the actual physical locations for the coordinate system axis zero points are entered. As mentioned earlier, this can be in the form of a **WPC** or work offset using G54–G59, as is common in G-Code programs. The operator measures the distances in *X, Y,* and *Z* in relation to the Machine Zero, then enters these values into the program in this unit by using the WPC MEASURE function.

MACHINING UNIT

Here, the individual machining units are identified in order to complete the workpiece. In Figure 8-9, the choices are POINT MACHINING, LINE MACHINING, FACE

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	SHAPE CHECK	

Figure 8-9 Machining Process Menu

MACHINING, MANUAL PROGRAM, OTHER, WPC, OFFSET, END, and SHAPE CHECK. Additional Machining Units may be entered until the final part geometry is completed as needed. Some examples are: POINT MACHINING, used for drilling, tapping, reaming, etc.; LINE MACHINING, used for Line Center, Line Left, Chamfer Left, etc.; and FACE MACH-ING, used for Face Mill, Top End Mill, Pocket, Step, Slot, etc. The following descriptions will be limited to point machining and line machining.

Point Machining

Point machining constitutes a large percentage of machining center work. Figure 8-10 shows the different choices for types of point machining.

When a selection is made from one of these choices, unit data are required and automatic tool development is completed, based on this information. The required information for drilling is the diameter, the depth, and whether the hole is to be chamfered. The basic required tooling is developed based on this information. For example, a center drill or spot drill, a drill of the size stated, and a chamfering cutter will be developed. This tooling information is taken from a predetermined tool file in the control. The tool file should be constructed (for all other types of machining units) by the machinist/programmer prior to programming, but can be done as the program is completed.

SEQUENCE DATA

Tool Sequence Data

Each individual tool has specific sequence data that are required as follows:

- definition of the actual size of the tool
- the priority in which this tool is to be used
- the diameter of the hole
- the hole depth
- pre-existing hole diameter
- pre-existing hole depth
- the desired surface finish
- the type of drilling cycle (e.g., drilling, pecking)
- the cutting speeds and feeds
- the use of any M-Codes

Shape Sequence Data

Finally, we come to the shape sequence data set for the machining unit. In other words, the actual Figure pattern or shape is identified. In the case of drilling,
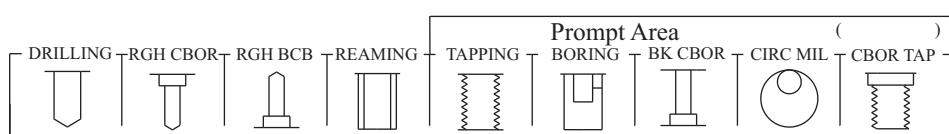
Figure 8-10 Point Machining Menu

POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD		
							SHAPE END	CHECK
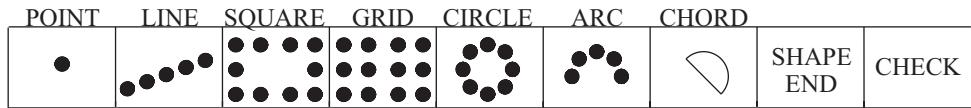
Figure 8-11 Point Sequence Data Menu

the choices are: POINT, **LNE**, SQUARE, GRID, CIRCLE, ARC, and CHORD, shown in Figure 8-11. As soon as the pattern is completed, SHAPE END is pressed to end the unit.

Just as with all units, CHECK allows the shape to be checked graphically for each individual unit. The remainder of the program is constructed in the same manner until all geometry shapes are complete.

LINE MACHINING

Line machining (linear contouring) is another very common activity performed by machining centers. Figure 8-12 shows choices for types of line machining.

When a selection is made, unit data are required and, based on this information, automatic tool development is completed. The required information for line machining is: the depth of cut; the amount of stock removal in Z (**SRV-Z**); the amount of radial stock removal in X-Y (**SRV-R**); the desired finished surface roughness; chamfer width, if required; the allowance for the finish Z depth cut; and the allowance for the radial finish width cut. The values input to these items determine the automatic tool development.

Tool Sequence Data

Each individual tool developed has specific sequence data that are required, as follows: the nominal diameter of the tool; the priority in which the tool is to be used; the approach point along the X-axis; the approach point along the Y-axis; the cutting direction of either **CW** or **CCW**; the plunge cutting feedrate along the Z-axis; the depth of cut, the cutting speed and the cutting feedrate; and M-Codes as required.

Shape Sequence Data

Finally, the shape sequence data is created in the machining unit where the actual Figure pattern or shape is identified. In the case of line machining, the choices are: SQUARE, CIRCLE, and **ARBITRY**, shown in Figure 8-13.

LINE CTR	LINE RGT	LINE LFT	LINE OUT	LINE IN	CHMF RGT	CHMF LFT	CHMF OUT	CHMF IN
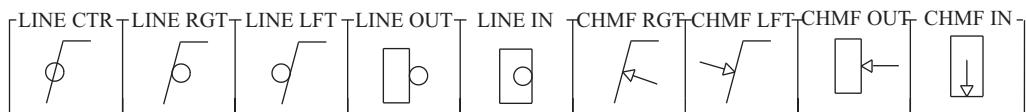
Figure 8-12 Line Machining Menu

SQUARE	CIRCLE		ARBITRY	PATTERN OF FIGURE <MENU>? ( )				
							SHAPE END	CHECK
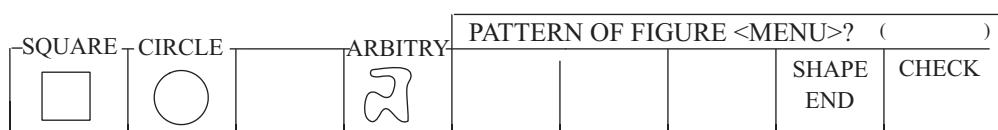
Figure 8-13 Shape Sequence Data Menu

The geometric shape is constructed by inputting point data that describe each feature of the square, circular, or arbitrary shape. As soon as the pattern is completed, SHAPE END is pressed to end the unit.

As with all similar units, the CHECK menu button allows the shape to be checked graphically for each individual unit. The remainder of the program is constructed in the same manner until all geometry shapes are complete.

END UNIT

This unit ends the program in the same manner that M30 does in a G-Code program. In this unit, you have the option of continuing the program for a number of repetitions and to control the Automatic Tool Change positioning.

Shape Check

When the entire program is written, it is beneficial to verify its accuracy by performing first a SHAPE CHECK and then a TOOL PATH CHECK. The SHAPE CHECK verifies the geometry whereas the TOOL PATH CHECK verifies the actual relationship between the geometry and the tool's actual cutting path. You can change from two-dimensional to three-dimensional views, or split the display screen to show the X-Y and X-Z views, and zoom in on features that are hard to see. Once these checks are completed, without errors, the program is ready for set-up of the tool and work offsets. You may then begin automatic operation.

MAZATROL TURNING CENTER PROGRAM EXAMPLE

For this example, we will use an offline program called MazaCAM to create the Mazatrol program. It very closely emulates the programming process that happens when programming at the controller and the output is essentially the same. Some of the Figures are screen shots from the MazaCAM program.

Start the Geopath software by double clicking the GeoPath Icon (Figure 8-14) or by selecting Start, All Programs, SolutionWare CAD-CAM, and GeoPath from the Windows start menu system.

Figure 8-14 GeoPath Icon
Courtesy SolutionWare Corp.

From the SolutionWare window, select File-Utilities from the Menu and then select MazaCAM CAD/CAM.

From the MazaCAM Utilities screen, select Make Mazatrol from the menu and then select Mazatrol Editor from the drop down list. The Select Program File dialog will come up. Input the file name of the program you wish to create and press Open. Answer yes to create the file and select the controller type you wish to program. In this case, use the Fusion 640T controller type.

For the turning center program example that follows, we will use the drawing in Figure 8-15 and the setup sheet, Chart 8-3.

To write a program in Mazatrol at the machine controller, you must use the soft keys located under the CRT monitor. There are 14 soft keys. The twelve in the middle are considered Menu selection keys while the left-most key is considered the Display Key and the right-most key is considered the Page key. Titles are given above each of the menu keys which aid in the program creation process.

Part 8 Mazatrol Conversational Programming
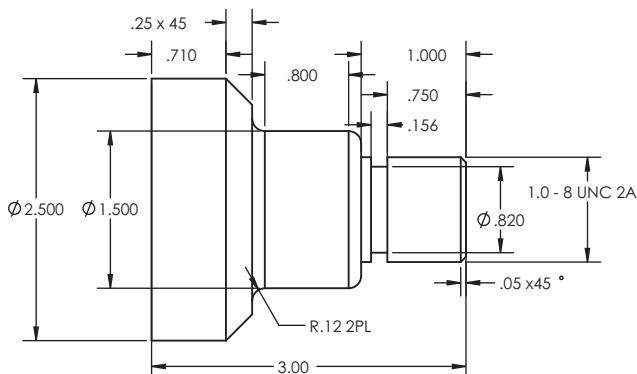
Figure 8-15 Drawing for Mazatrol Turning Center Program

Your Company
CNC Setup Sheet

Date: Today	Prepared By: You
Part Name: Turning Center Project	Part Number: 4321
Machine: CNC Turning Center	Program Number:

Workpiece Zero: X = Centerline Y = NA Z = Finished Face

Setup Description:

Clamp the part in a 3-jaw chuck with 3.25" extended out of the chuck

Tool #	Tool Description	Offset #	Comments
1	Rough Turning Tool .031 TNR	1	SFPM 600
2	Finish Turning Tool .015 TNR	2	SFPM 850
5	O.D. Grooving Tool	5	.156 W ide .005 TNR
7	O.D. Threading Tool	7	SFPM 150

All Mazatrol programs consist of: a Common Data Unit; one or more Workpiece Coordinate Units; one or more Machining Units; and an End Unit.

STEPS TO CREATE A MAZATROL TURNING PROGRAM

Turn on the machine and Home all of the axes.

- Press the left most Display key.
- Select PROGRAM from the Menu keys.
- Press the WORK NO. Menu key.

Note: During programming, there is a user prompt area in the lower right corner of the CRT monitor; this area provides a brief description that leads you through the programming process. Simply answer the questions.

The following message will be displayed in the prompt area:

WORKPIECE NO. ? ( ).

Part 8 Mazatrol Conversational Programming

- Key in the desired work number (program number), e.g., 2525, and press Input. It is a good idea to check the Program File prior to this step in order to identify program numbers that have been used.

At this point, there is a Menu option to program in either EIA/ISO (G-Code or MAZATROL).

Note: On some older generation controls (M2, M32, and M+), the EIA/ISO selection was an option and may not be available.

- Select MAZATROL from the Menu keys

You are now ready to create the Common Data Unit which will be listed on the screen as Process #0. The first prompting question encountered is:

WORKPIECE MATERIAL <MENU>? ( )

From the Menu options you should select the material that most closely matches the material you are using. This selection, along with the tool file information, affects the automatic calculation of feeds and speeds later in the programming process.

- Select **AL** from the Menu keys and Press Input.

The material choices are given above the menu keys on the control. Abbreviations for materials are: **FC** = Cast Iron; **FCD** = Cast Ductile Iron; **S45C** = Low Carbon Steel; **SCM** = Alloy Steel**; ALY STEEL** = Alloy Steel; **SUS** = Stainless Steel; **AL** = Aluminum and, **CU** = Copper. The last Menu key on the right has arrows pointing to the right ($\rightarrow \rightarrow \rightarrow \rightarrow$) to access a second page of additional material selections. These can be used for user-defined materials.

Note: All Cutting Condition Parameters are based from Carbon Steel (e.g. Plain Carbon Steel 1018)—all other materials are a percentage factor of the Carbon Steel settings. With this criteria in mind, you may develop new materials specific to your application. For details on assigning Cutting Condition Parameters, see the Mazak User Manual provided with your machine.

MAXIMUM OUTSIDE DIAMETER <MENU>? ( )

- Input 2.5 inches

MIN INSIDE DIAMETER <MENU>? ( )

- Input 0

WORKPIECE LENGTH? ( )

- Input 5.0 inches

This amount allows for 2 inches of the material to be clamped in the chuck.

MAX SPINDLE RPM LIMIT (rpm)? ( )

- Input 3000

FINISH ALLOWANCE X? ( )

- Input .01

FINISH ALLLOWANCE Z? ( )

- Input .005

STOCK REMOVAL OF WORKFACE? ( )

Part 8 Mazatrol Conversational Programming

- Input .1

  MODE MENU?

  From the Menu keys, notice the arrow keys at the left end (F9). When pressed, the next menu screen is displayed.

- Choose M-Code (F5)
- Input 08 for Flood Coolant (F6)
- Use cursor down key to start a new machining process

  MODE MENU?   (   )
- Choose EDGE

  SECTION TO BE MACHINED <MENU>
- Choose **FCE** from the Menu keys

  ROUGHING PERIPHERAL SPEED?   (   )
- Press AUTO SET (F1)

  FINISH PERIPHERAL SPEED?   (   )
- Press AUTO SET (F1)

  ROUGHING FEEDRATE?   (   )
- Input .016

  ROUGHING DEPTH OF CUT PER PASS   (   )
- Input .08

  ROUGHING TOOL TYPE?   (   )

  Choose from tools that are identified in the tool file and the turret (at the machine). In our case, we will use tool number 2 for roughing and tool number 4 for finishing, these have been predefined in the software.

- Input 2 for rough turning tool
- Input tool 4 for the finish turning tool

  STARTING POINT X?   (   )
- Input 2.5

  STARTING POINT Z?   (   )
- Input .1

  FINAL POINT X?   (   )
- Input 0

  FINAL POINT Z?   (   )
- Input 0

  SURFACE ROUGHNESS or FINISHING FEEDRATE? <MENU>   (   )
- Select Roughness (F1) and then choose 6 (F6)

  Once this data is entered, a new process begins.

  MODE MENU?   (   )
- Choose **BAR** from the Menu keys (F1)

- Choose **BAR** OUT from the Menu keys (F1)

    CUTTING PATTERN MENU?   *(   )*

- Chose #1 for the cut style (F2)

    The difference between #0 and #1 is the way the tool moves at the end of the cut sequence. For BAR OUT 1, the tool feeds up the back wall, out to the OD, at the end of the cut on every pass; whereas on BAR OUT 2, the tool rapids away from the cut at 45º to a clearance point set by parameter. Consecutive passes are then completed and the back wall is finished on the last pass only.

    CUTTING POINT X?   *(   )*

- Input 2.5

    CUTTING POINT Z?   *(   )*

- Input 0

    SURFACE SPEED FOR ROUGH CUT?   *(   )*

- Select AUTO SET from the Menu keys (F1)

    FINISHING PERIPHERAL SPEED?

- Select AUTO SET from the Menu keys (F1)

    ROUGHING FEEDRATE?

- Input .016

    ROUGHING DEPTH OF CUT PER PASS?

- Input .080

    ROUGHING TOOL TYPE?

- Input 2, for the rough turning tool

    FINISHING TOOL TYPE?

- Input 4, for the finish turning tool

    SHAPE PATTERN?

- Choose **LIN** (F1)

    CHAMFERING (C) vs. ROUNDING R AT START POINT

- Input .05 to create the .05 x 45º chamfer (C)

    Note: if a Radius is required, you must press the CORNER R Menu key (F1), prior to entry of the amount.

    X END POINT?

- Input 1.0 to create the 1.0 diameter

    Z END POINT?

- Input 1.0 to create the length of the1.0 diameter

    CHAMFERING vs. ROUNDING AT END POINT?

- Press Input to omit this setting

    RADIUS OF ARC OR TAPER ANGLE?

- Press Input to omit this setting

FINISHING FEEDRATE FOR SURFACE ROUGHNESS?

- Input 6 (F6)

   SHAPE PATTERN?

- Choose **LIN** (F1)

   CHAMFERING (C) vs. ROUNDING R AT START POINT?

- Press CORNER R (F1)
- Input .1 to create the .1 radius (R)

   X END POINT?

- Input 1.5 to create the 1.5 diameter

   Z END POINT?

- Input 2.0 to create the length of the 1.5 diameter

   CHAMFERING vs. ROUNDING AT END POINT?

- Press CORNER R (F1)
- Input .1 to create the .1 radius (R)

   RADIUS OF ARC OR TAPER ANGLE?

- Press Input to omit this setting

   FINISHING FEEDRATE FOR SURFACE ROUGHNESS?

- Input 6 (F6)

   SHAPE PATTERN?

- Choose **LIN** (F1)

   CHAMFERING (C) vs. ROUNDING R AT START POINT

- Input .25 to create the .25 x 45º chamfer (C)

   X END POINT?

- Input 2.5 to create the 2.5 diameter

   Z END POINT?

- Input 3.0 to create the length of the 2.5 diameter

   CHAMFERING vs. ROUNDING AT END POINT?

- Press Input to omit this setting

   RADIUS OF ARC OR TAPER ANGLE?

- Press Input to omit this setting

   FINISHING FEEDRATE FOR SURFACE ROUGHNESS?

- Input 6 (F6)
- Select SHAPE END from the Menu keys (F9)

   A new Process Number is started. Select the Machining Unit type next.

- Select **GRV** for Groove from the Menu keys (F6)

   SECTION TO BE MACHINED?

- Choose OUT (F1)

MACHINING PATTERN?
- Choose type #1 (F2)

    NUMBER OF GROOVES?
- Input 1

    SPACING AMOUNT OF MULTIPLE GROOVES?
- Press Input to omit this setting

    GROOVE WIDTH?
- Input .156 for the groove width

    FINISH REMOVAL ALLOWANCE?
- Input .01 for the groove finish allowance

    ROUGHING PERIPHERAL SPEED?
- Press the AUTO SET Menu key (F1)

    FINISHING PERIPHERAL SPEED?
- Press the AUTO SET Menu key (F1)

    ROUGHING FEEDRATE?
- Input .010

    ROUGHING DEPTH OF CUT PER PASS?
- Input .04

    ROUGHING TOOL TYPE?
- Input 8 for the rough grooving tool

    FINISHING TOOL TYPE?
- Input 8 for the finish grooving tool

    CHAMFERING (C) vs. ROUNDING R AT START POINT?
- Input .04 for chamfering at the start of the groove

    X START POINT?
- Input 1.0

    Z START POINT?
- Input .75

    X END POINT?
- Input .9

    Z END POINT?
- Input .75

    CHAMFERING vs. ROUNDING AT END POINT?
- Press Input to omit this setting

    FINISHING FEEDRATE FOR SURFACE ROUGHNESS?
- Input 6 (F6)

A new Process Number is started. Select the next Machining Unit type.

- Select **THR** for Threading from the Menu keys (F6)

  SECTION TO BE MACHINED?
- Choose OUT (F1)

  MACHINING PATTERN?
- Choose #0 STANDARD (F1)

  CHAMFER ANGLE <0: NO CHAMFER, 1:45 DEGREES, 2:60 DEGREES>?
- Choose #0 for no chamfer

  THREADING LEAD?
- Input .125 for the thread lead

This amount is determined by dividing one by the number of threads per inch. In this case, 1/8 = .125.

  THREADING ANGLE?
- Input 59 for the threading angle

  NUMBER OF THREADS?
- Input 1 for the number of thread starts

  THREADING HEIGHT?
- Input .0801 for the thread height

  NUMBER OF TIMES THREADING?
- Input AUTO SET (F1)

  SPINDLE PERIPHERAL SPEED?
- Input AUTO SET (F1)

  FIRST THREADING AMOUNT?
- Input .010

  TOOL TYPE?
- Input 6 for the threading tool number.

  X START POINT?
- Input 1.0

  Z START POINT?
- Input 0

  X END POINT?
- Input 1.0

  Z END POINT?
- Input .85
- Select SHAPE END from the Menu keys

A new Process Number is started. Select the next Machining Unit type.

- Select END from the Menu keys (F8)

COUNT NUMBER OF MACHINED WORKPIECES <YES = 1, NO = 0>?

• Input 0

TOOL RETURN POSITION <0 = CHANGE POSITION, 1 = HOME, 2 = FIXED POINT>?

• Input 1

Inputting this value will send the turret to the Home position at the end of the program.

WORK NUMBER OF FOLLOWING PROGRAM?

• Press Input to omit this setting

EXECUTE PERPETUALLY = 1; EXECUTE NUMBER OF TIMES IN NUM = (0)

• Press Input to omit this setting

NUMBER OF TIMES TO REPEAT PROGRAM?

• Press Input to omit this setting

Z SHIFT AMOUNT OF PROGRAM ORIGIN?

• Press Input to omit this setting

This concludes programming of the part. A sample of the program shape plot is displayed in Figure 8-16 and the program output is displayed in Figure 8-17.
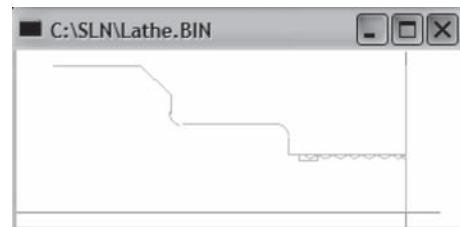
Figure 8-16 Program Shape Plot
Courtesy SolutionWare Corp

Figure 8-17 Finished Turning Program
Courtesy SolutionWare Corp

MAZATROL MACHINING CENTER PROGRAM EXAMPLE

For this example, we will give the steps to input the program as you would at the controller. The graphics for some of the Menu keys, Part Shape, and Program output are taken from the offline program called MazaCAM, which can also be used to create the Mazatrol program. It very closely emulates the programming process that happens when programming at the controller and the output is essentially the same. Some of the Figures used are screen shots from the program and are labeled as such.

For the machining center program example that follows we will use the drawing in Figure 8-18 and the setup sheet, Chart 8-4.
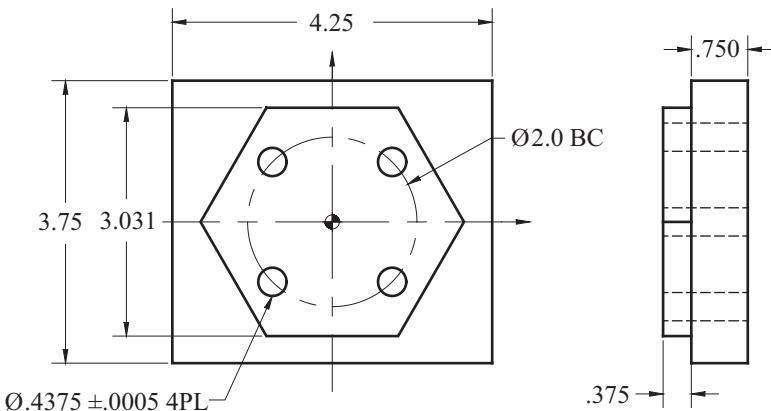
Figure 8-18 Drawing for Mazatrol CNC Machining Center Program

Your Company
CNC Setup Sheet

Date: Today	Prepared By: You
Part Name: Machining Center Project	Part Number: 5432
Machine: Machining Center	Program Number:

Workpiece Zero: X = Centerline Y =Centerline Z Top Most Finished Surface

Setup Description:

Clamp the part in a vise on parallels.
A minimum of 3/16" must be above the vise jaws.

Tool #	Tool Description	Offset #	Comments
1	3.0" Ø 5 Tooth Face Mill	1	SFPM 1800, in/rev = .012
2	.750" Ø 2 Flute End Mill	2	SFPM 200, in/rev = .002
3	.50" Ø x 90° Spotting Drill	3	SFPM 200 in/rev = .002
4	.4219" Ø 27/64 Drill	4	SFPM 200, in/rev = .002
5	.4375" Ø Reamer	5	r/min = 1/3 of lowest drill speed
			feed = 1/2 of lowest drill feed

Part 8 Mazatrol Conversational Programming

To write a program in Mazatrol at the machine controller, you must use the soft keys located under the CRT monitor, (there are 14 soft keys). The twelve in the middle are considered Menu selection keys while the left most key is considered the Display Key and the right most key is considered the Page key. Titles are given above each of the menu keys which aid in the program creation process.

All Mazatrol programs consist of a: Common Data Unit; one or more Workpiece Coordinate Units; one or more Machining Units; and, an End Unit.

Steps to Create a Mazatrol Milling Program

Turn on the machine and Home all of the axes.

- Press the left most Display key.
- Select PROGRAM from the Menu keys.
- Press the WORK NO. Menu key.

Note: During programming, there is a user prompt area in the lower right corner of the CRT monitor that gives a brief description that leads you through the programming process. Just answer the questions.

The following message will be displayed in the prompt area:

WORK NO. (NAME SEARCH <?INP>   (   )*

- Key in the desired work number (program number) e.g., 2525, and press Input. It is a good idea to check the Program File prior to this step in order to identify program numbers that have been used.

At this point, there is a Menu option to program in either EIA/ISO (G-Code or MAZATROL).

Note: On some older model controls (M2, M32, and M+) the EIA/ISO selection was an option and may not be available.

- Select MAZATROL from the Menu keys

You are now ready to create the Common Data Unit, which will be listed on the screen as Unit #0. The first prompting question encountered is:

MATERIAL <MENU>?   (   )*

From the Menu options, you should select the material that most closely matches the material you are using. This selection, along with the tool file information, affects the automatic calculation of feeds and speeds later in the programming process.

- Select **AL** from the Menu keys and Press Input

The material choices are given above the menu keys on the control. Abbreviations for materials are: **CST IRN** = Cast Iron; **DUCT IRN** = Ductile Iron; **CBN STEEL** = Carbon Steel; **ALY STEEL** = Alloy Steel; b = Stainless Steel; **AL** = Aluminum; **COPPER** = Copper. The last Menu key on the right has arrows pointing to the right ($\rightarrow \rightarrow \rightarrow \rightarrow$) for a second page of additional material selections. These can be used for user defined materials.

Note: All Cutting Condition Parameters are based from Carbon Steel (e.g., Plain Carbon Steel 1018). All other materials are a percentage factor of this. You may develop new materials specific to your application with this criteria in mind. For details on assigning Cutting Condition Parameters, see the section in the Mazak User Manual provided with your machine.

INITIAL POINT Z (CLEARANCE)?   (   )

For specific details, see the preceding text in this section for a description of Initial *Z* Clearance question to establish the *Z* Plane.

- Key in 1.0 and press Input

ATC MODE ZERO RETURN <Z.X+Y:0, X+Y+Z:1>?   (   )

Activating the ATC mode sets the path the spindle is to take for the Automatic Tool Change Mode. See existing text for a more detailed description of ATC MODE.

- Key in 0 to select the *Z* first and then the *XY* movement; Press Input

MULTI MODE <Menu>?

See the existing text in this section for details regarding whether or not there are multiple parts on the table.

- Select MULTI OFF from the Menu keys

MACHINING UNIT <MENU>?   (   )

The following choices are available to choose: POINT; LINE; FACE; MANUAL PROGRAM; OTHER; WPC; OFFSET; END, and SHAPE CHECK (See Figure 8-8).

- Choose WPC from the Menu select keys

This is UNo.1 WPC= Workpiece Coordinate. Use this unit to identify the location for the part zero of the geometry from machine home. These values are usually measured using an edge-finding device or probe system, then input within the program or the off-set registers. You may also set this item to use G54–G59 and additional offsets of A–K. Consult the Operation Manual of your machine for specific instructions on setting these values. A number can be assigned in order to include multiple WPCs within the same program.

- Press Input or use the right pointing cursor key to accept a value of 0 for our WPC number
- Press Input again to pass over the Additional Offsets

At this point, you will be prompted as follows:

WORKPIECE COORDINATE, WPC-X?   (   )

WORKPIECE COORDINATE, WPC-Y?   (   )

WORKPIECE COORDINATE, WPC-θ?   (   )

WORKPIECE COORDINATE, WPC-Z?   (   )

WORKPIECE COORDINATE, WPC-4?   (   )

- Input 0 in each case for now

Setup Notes: The exact coordinate values for each of these offsets will be measured during the setup process and entered via WPC-MEASURE, WPC SEARCH. Position an edge-finding device in order to find the workpiece edge along the X-axis. Press the TEACH Menu key and key in a value (0) that represents the location of the spindle in relation to the Workpiece Zero, and Press Input. Don't forget to compensate for the radius of the edge finder. Repeat these steps for the Y-axis. Input 0 for the workpiece rotation angle of theta. All cutting tools used in the program should be installed in the magazine and measured to

the tool sensor prior to workpiece coordinate setting. For WPC-Z, position any tool that is used in the program and touch off the top most surface of the part. Remember, sometimes this top-most surface is above the finished surface zero. This is the case in our example, so once the tool is touched off along the Z-axis, Press TEACH, key in –.05 and Press Input. Input 0 for the 4th axis workpiece coordinate.

When all these data are entered, a new unit (UNo.2) is started.

MACHINING UNIT <MENU>? ( )

Often, it is necessary to machine the work surface to establish the finished Z0.0 work face. For this reason, it is commonly established at the beginning of the program. The following choices are available to choose for Face Machining: FACE MIL; TOP EMIL; PCKT MT; PCKT VLY (Figure 8-19).

· Select FACE MIL from the Menu keys

Once the Face Machining menu key is pressed, there are columns with the following headings displayed on the screen: Depth, **SRV-Z**, **BTM** and **FIN-Z**.

DIST: WPC Z0 TO FIN SURFACE? ( )

· Input 0 here because this represents our finished surface for *Z*

Z AXIS STOCK REMOVAL? ( )

· Input .05 here because there is that much excess material to remove

BOTTOM ROUGHNESS <MENU>? ( )

· Choose 6 from the Menu keys

Selecting 6 from the menu choices will automatically set the Z depth of cut for the roughing (.0402) and finishing (.0098) passes. A number system of 1–9 is used here that relates to surface finish (Figure 8-20). The higher the number, the more fine the finish. This number affects the rough and finish depths of cut.

· Press Input

Figure 8-21 identifies the approximate surface Micro Finish that is created when each of the numbers are selected. For example, selection of Menu key 6 will produce a 32 Micro Finish.

On the left hand side of the screen, the acronym **SNo**. will be displayed and the columns will have the headings: **NOM-Ø**, **No**., **APRCH-X, APRCH-Y, TYPE, WID-R,**
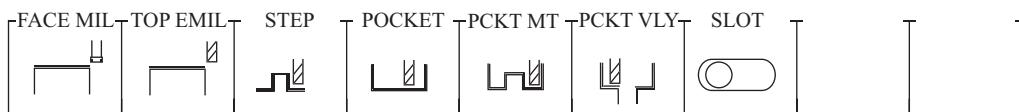
Figure 8-19 Face Machining Menu Keys

Figure 8-20 Bottom and Wall Roughness Menu Keys

1000	500	250	125	63	32	16	8	4	

Figure 8-21 Surface Micro Finish Chart

C-SP, FR, and **M.** Because we selected Face Machining earlier, the rough and finish tools will be automatically developed. On the next line, R 1 F-Mill will be displayed.

    NOMINAL DIAMETER?   (   )

- Input 3.0 for the Face Mill diameter

    TOOL FILE CODE?   (   )

- Press Input to omit setting of this special identification code

    MACHININING PRIORITY NUMBER?   (   )

- Press Input to omit

    APPROACH POINT X, AUTO → <MENU>   (   )

- Press the AUTO SET menu key

This action will automatically set the approach point a safe distance from the part geometry in the X-axis. This value is controlled by parameter setting and is typically 66% of the tool diameter.

Note: When AUTO SET is pressed, question marks are temporarily entered until the part geometry is entered and the values are changed relative to this geometry. The programmer can manually enter these values, if known, or other values are preferred.

    APPROACH POINT Y, AUTO → <MENU>?   (   )

- Press the AUTO SET menu key

    CUTTING DIRECTION <MENU>?   (   )

- Select **X BI-DIR** from the Menu keys (F1 from the function keys)

X BI-DIR stands for Bi-Directional cutting along the *X-axis,* whereas **X** or **Y UNI-DIR** stands for cutting along either axis in a single direction. **X** or **Y BI** or **UNI-DIR SHORT** means that the tool will not position completely off the part, but by only 33% (Figure 8-22).

Note: the subheadings F1–F9 are not present on a MAZAK controller. These are function keys on a standard keyboard and are used within the MazaCAM Editor only.

    DEPTH OF CUT?   (   )

- Press the AUTO SET Menu key

X BI-DIR F1	Y BI-DIR F2	X UNI-DIR F3	Y UNI-DIR F4	X BI-DIR SHORT F5	Y BI-DIR SHORT F6	F7	F8	F9

Figure 8-22 Face Machining Cutting Direction
Courtesy of SolutionWare Corporation

By selecting the AUTO SET function, the controller calculates the difference between the full depth required and the finish allowance determined earlier in the program, and inputs this amount. In our case, it is .0402, because .0098 was the allowance and .050 is the stock removal amount.

WIDTH OF CUT?   *(   )*

· Press the AUTO SET Menu key

This value is based on 66% of the diameter of the tool selected, which in our case is the three-inch face mill. Therefore, by pressing AUTO SET, the control will output approximately 2.7 for the width of cut.

CUTTING SPEED, AUTO→<MENU>?   *(   )*

· Press the AUTO SET Menu key

Pressing the AUTO SET Menu key uses the tool and material data previously entered into the program to determine the cutting speed (sf/min).

FEEDRATE, AUTO→<MENU>?   *(   )*

· Press the AUTO SET Menu key

Pressing the AUTO SET menu key uses the tool and material data previously entered into the program to determine the cutting feed rate (in/min).

M CODE?   *(   )*

Coolant flow is activated here. Two M-Codes may be activated pre-tool.

· Select 08 from the Menu keys (F6 from the function keys) for Flood Coolant (Figure 8-23)

· Press Input to omit the second M-Code selection

On the next line, F 2 F-Mill will be displayed for the Finish Face Mill.

· Repeat the entries in a similar fashion, as above, for the required data.

After identifying what style of machining is to take place, the next information needed is the Figure Pattern. This is the coordinate data that makes up the part geometry. When the entire shape has been input, you must end the unit by pressing the SHAPE END menu key. This action closes the unit and starts a new one. At this point, it is also a good idea to complete a SHAPE CHECK to ensure the data that has been input is correct. The shape of the Figure Pattern will be displayed on the screen.

PATTERN OF Figure <MENU>?   *(   )*

· Choose Square from the Menu keys

01 OPT. STOP	03 SPNDL FWD	04 SPNDL REV	05 SPNDL STOP	07 MIST COOLANT	08 FLOOD COOLANT	09 OFF COOLANT	50 AIR BLAST	->->->
F1	F2	F3	F4	F5	F6	F7	F8	F9

Figure 8-23 M-Codes Menu Keys
Courtesy SolutionWare Corp.

SQUARE	CIRCLE		ARBITRY	PATTERN OF FIGURE <MENU>? ( )				
□	○		⌐⌐				SHAPE END	CHECK
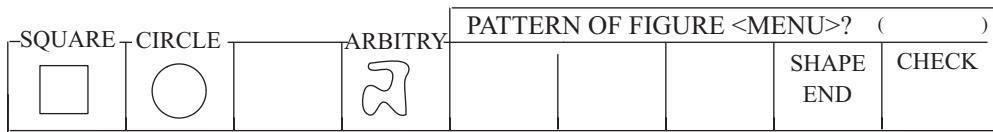
Figure 8-24 Pattern of Figure Menu Keys

The display in Figure 8-24 has columns with the headings: P1X/CX, P1Y/CY, P3X/R, P3Y, CN1, CN2, CN3, CN4.

CORNER 1 COODINATE X? ( )

- Key in –2.125 and press Input

This value represents the coordinate for the lower left corner of the part geometry for the *X-axis*.

CORNER 1 COODINATE Y? ( )

- Key in –2.125 and press Input

This value represents the coordinate for the lower left corner of the part geometry for the *Y-axis*.

CORNER 3 COODINATE X? ( )

- Key in 2.125 and press Input

This value represents the coordinate for the upper right corner of the part geometry for the X-axis.

CORNER 3 COODINATE Y? ( )

- Key in 2.125 and press Input

This value represents the coordinate for the upper right corner of the part geometry for the Y-axis. This entry completes the required geometry for the Face Milling operation.

Note: CN1 through CN4 allow the entry of chamfers or radii at each of the individual corners of the square.

- Press Input 4 times to omit entries for CN1 through CN4
- Press the SHAPE END Menu key

    A new Unit is started and is labeled UNo.3.

    MACHINING UNIT <MENU>?

- Select FACE MACHINING
- Select STEP from the Menu selection keys

The column headings displayed in Figure 8-25 are the same for STEP machining as in facing. The acronyms described at the beginning of this section are the same. Focus your attention on the prompting question in the lower right corner of the display for data entry.

DIST: WPC 0 TO FIN. SURFACE? ( )

- Because our finished Z-zero surface is our reference, we will Input zero (0) here

425

| FACE MIL | TOP EMIL | STEP | POCKET | PCKT MT | PCKT VLY | SLOT | | |
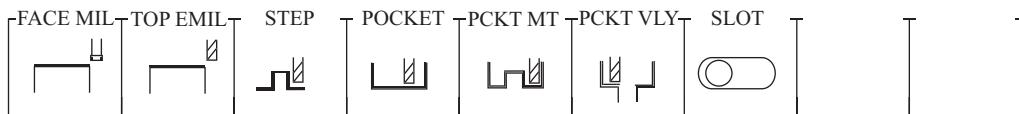
Figure 8-25 STEP Machining Unit Menu Keys

Z AXIS STOCK REMOVAL?   (   )

- Because we know there is .375 of material to remove in our STEP, Input .375; no sign is necessary

BOTTOM FACE ROUGHNESS <MENU>?   (   )

- Input 6 here for the surface roughness control for the bottom (**BTM**) of the STEP

WALL ROUGHNESS <MENU>?   (   )

- Input 6 here for the surface roughness control for the walls (**WAL**) of the STEP

FINISH ALLOWANCE Z ?   (   )

This value is automatically calculated based on the bottom (**BTM**) surface roughness number selected. It can be modified, at any time, however:

- Accept the pre-calculated value (.0098) and press Input

FINISH ALLOWANCE R ?   (   )

This value is automatically calculated based on the wall (**WAL**) surface roughness number selected. However, it can be modified at any time:

- Accept the pre-calculated value (.0098) and press Input

The tool data information is input for the unit now. The column headings are titled the same as in the face machining unit for SNo.

WHICH TYPE OF TOOL <MENU>?   (   )

- Select ENDMILL from the Menu keys (F1 function key) and press Input (Figure 8-26)

Complete the roughing (R1) tool data as follows:

NOMINAL DIAMETER?   (   )

- Input .750 for the roughing end mill diameter

TOOL FILE CODE?   (   )

Selecting a tool file code allows further identification of the tool by a letter code— for example, roughing, carbide, and high speed steel. The shop where you work should establish a system to follow.

- Press Input to omit this setting

ENDMILL	FACE MILL	CHAMFER CUTTER	BALL ENDMILL		OTHER			
F1	F2	F3	F4	F5	F6	F7	F8	F9

Figure 8-26 Type of Tool Menu Keys
Courtesy SolutionWare Corp

MACHINING PRIORITY NO.? ( )

By setting a machining priority number the sequence in which the tool is used is further controlled. If no value is input, the tool will be used by program priority.

- Press Input to omit this setting

APPROACH POINT X, AUTO → <MENU> ( )

- Press the AUTO SET menu key

APPROACH POINT Y, AUTO → <MENU> ( )

- Press the AUTO SET menu key

CUTTING DIRECTION <MENU>? ( )

The choices here are clockwise (**CW-CUT**) cutting, or counterclockwise (**CCW-CUT**). This input controls whether the cut is climb or conventional cutting.

- Press the CW-CUT Menu key

FEEDRATE Z, <MENU>/DATA<INPUT>? ( )

The menu choices are: CUT-G01 or a RAPID-G00. This value is set by default for CUT G01.

Caution: If this value is set to rapid and material exists at the penetration point, a collision will occur.

- Press the CUT-G01 Menu key

DEPTH OF CUT? ( )

- Press AUTO SET

WIDTH OF CUT? ( )

- Press AUTO SET

CUTTING SPEED, AUTO → <MENU>? ( )

- Press AUTO SET

FEEDRATE, AUTO → <MENU>? ( )

- Press AUTO SET

M-CODE? ( )

- Press 08 and press Input

Complete the finishing tool sequence (F2) data in a similar fashion as for the rough tool. A different finishing tool may be selected or the same tool can be used. The depth of the finishing pass depends on the amount remaining from the roughing tool that was identified in the prior information.

PATTERN OF Figure <MENU> ( )

Here is where we identify the shape of our STEP geometry; there are two shapes involved in every case. Always identify the outside shape first when performing a STEP geometry.

- Select SQUARE from the Menu keys

CORNER 1 COORDINATE X? ( )

- Key in –2.125 and press Input

> *CORNER 1 COORDINATE Y?* ( )

- Key in –2.125 and press Input

> *CORNER 3 COORDINATE X?* ( )

- Key in 2.125 and press Input

> *CORNER 3 COORDINATE Y?* ( )

- Key in 2.125 and press Input

This concludes the first shape for the STEP machining unit. The second pattern is neither SQUARE nor CIRCLE so we must use arbitrary (**ARBITRY**). We have pre-calculated all of the necessary coordinate points for creation of the geometry.

- Select **ARBITRY** from the Menu keys

Once ARBITRY type geometry has been selected, the following choices become available (Figure 8-27):

- Select **LINE** from the Menu keys (F1 function key)

> *COORDINATE X OF Figure ?* ( )

- Key in .875 and press Input

> *COORDINATE Y OF Figure ?* ( )

- Key in –1.5515 and press Input
- Press Input five more times to omit the other entries or use the cursor down button to skip to the next line.
- Select LINE from the Menu keys (F1 function key).

Repeat the same entry technique for the following remaining coordinates:

- *3LINE*   *X–.875*   *Y–1.5515*
- *4LINE*   *X–1.75*   *Y0*
- *5LINE*   *X–.875*   *Y1.5515*
- *6LINE*   *X.875*   *Y1.5515*
- *7LINE*   *X1.75*   *Y0*

As units are completed, it is wise to perform a SHAPE CHECK to ensure that you have entered everything correctly. If all looks well, proceed to the next unit.

- Press SHAPE END twice to end the unit

At this point the Machining Unit menu is displayed again (Figure 8-28).

> *MACHINING UNIT <MENU>?* ( )

- Choose POINT MACH-ING

LINE	CW ARC	CCW ARC	SHAPE ROTATE	SHAPE SHIFT	REPEAT END	STATING POINT	SHAPE END	
F1	F2	F3	F4	F5	F6	F7	F8	F9

Figure 8-27 Arbitrary Shape Definition Menu Keys
Courtesy SolutionWare Corp.

428

Part 8 Mazatrol Conversational Programming

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	SHAPE CHECK	

Figure 8-28 Machining Unit Menu Keys
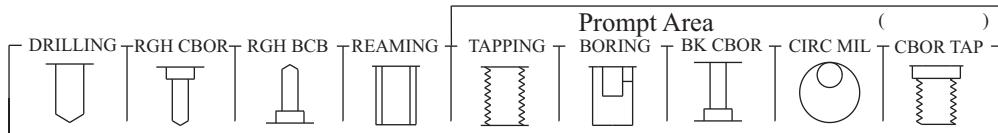
Figure 8-29 Point Machining Menu Keys

From the newly displayed Machining Unit menu (Figure 8-29), select Reaming.

MACHINING UNIT <MENU>?   (   )

• Choose REAMING

HOLE DIAMETER?   (   )

• Input .4375

HOLE DEPTH?   (   )

• Input 1.125

Note: When drills are identified in the tool data register, the drill point compensation is added so as to assure full drilling depth.

CHAMFER WIDTH?   (   )

• Input 0

PRE-REAMING OPERATION <MENU>?   (   )

Choices are: DRILL, BORING BAR or END MILL.

• Select the DRILL Menu key

CHIP VAC. CLEANER<Y:1, N:0>?   (   )

• Press Input to omit entry here

At this time, the tools are automatically developed for the reaming operation and listed based on the prior input for the tool sequence, SNo.

Sequence 1 is the **CTR-DR** of .3 diameter.

Sequence 2 is the DRILL of .43 diameter.

TYPE OF DRILLING CYCLE <MENU>?   (   )

For Sequence 2, the Drilling Cycle needs to be changed to PECKING CYCLE 2 and the DEPTH OF CUT to .43

Note: Changing the Drilling Cycle type will initiate the PECK DRILLING CYCLE for the deep holes in order to break and clear chips. Each peck will penetrate the material .43 deep until the finished depth is reached.

Sequence 3 is the REAM or .44 diameter.

POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD		
•	••••	••• ••• •••	••• ••• •••	○	•• ••	◯	SHAPE END	CHECK
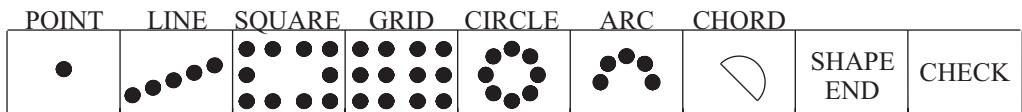
Figure 8-30 Point Cutting Pattern Menu Keys

The only thing that needs completion for the remaining tool information is the feeds and speeds and coolant.

- Press HSS AUTO
- Input 08 for the M-Code
- Repeat the same entries steps for each tool.

    POINT CUTTING PATTERN <MENU>?   (   )

- Choose CIRCLE from the Menu keys (Figure 8-30)

    Z VALUE OF THE WORK SURFACE?   (   )

- Input zero for the work surface value

    CIRCLE CENTER X?   (   )

- Input 0 for the circle center coordinate for the *X-axis*

    CIRCLE CENTER Y?   (   )

- Input 0 circle center coordinate for the *Y-axis*

    ANGLE OF START PT, FROM X-AXIS?   (   )

    By looking at the drawing, we can calculate that the first hole in the pattern is 45° from the polar coordinate reference.

- Input 45°

    CIRCLE RADIUS?   (   )

- Input 1 inch

    NUMBER OF HOLES?   (   )

- Input 4

    RETURN POSITION <INIT: 0, R:1>?   (   )

- Input 1 for the tool to return to the R-Plane
- Select SHAPE END

    That concludes the creation of our part program geometry.

- Select END from the menu keys

    CONTINUE <Y: 1, N: 0>?   (   )

- Input 0 for No

    PARTS COUNTER <Y: 1, N: 0>?   (   )

- Input 0 for No

    AUTO TOOL CHANGE <Y: 1, N: 0>?   (   )

- Input 0 for No

Part 8 Mazatrol Conversational Programming

End coordinates for *X, Y, Z,* and the fourth axis may be input to control positioning of the spindle away from the part for easy access. These values are in relation to Machine Zero.

- Press the Menu Select soft key
- Press the PROGRAM COMPLETE Menu key
- Press the TOOL PATH Menu key

The choices are: PATH CONTINUE, PATH, STEP, PART SHAPE, SHAPE ERASE, PATH TRACE, PROGRAM, STORE, PLANE CHANGE, and SCALE CHANGE.

- Choose PART SHAPE Menu key

The part geometry shape will be displayed similar to Figure 8-31. If no errors occur and all looks well, proceed to tool path checking.

- Press PATH CONTINUE

Figure 8-31 Finished Mill Program Shape Display
Courtesy SolutionWare Corp.

The tool path will now be displayed. Again, if all looks well, a first piece can be machined. The graphics capabilities allow for zooming, rotating, scaling, etc., so the programmer has lots of capability to see what the tool path will perform.

The completed program can be viewed on screen (Figure 8-32) as it looks at the control.

Although the steps given here for Mazatrol conversational programming may seem like a lot of effort, the truth is that when you get used to this style of data entry,

Figure 8-32 Finished Mill Program Display
Courtesy SolutionWare Corp.

Figure 8-33 MAZAK Integrex
Courtesy Daco Precision Manufacturers

the programming process can be very quick and simple. Even complex multi-tasking machines like the Mazak Integrex (Figure 8-33) are easily programmed using Mazatrol.

THE FUTURE OF CNC PROGRAMMING

The ultimate goal of any manufacturing is to increase productivity by improving efficiency. We want to minimize wasted and idle time and cut lead times, while maintaining accuracy. Modern technology enables these goals to become reality.

The PC has revolutionized our society; manufacturing has been a benefactor because of the direct effect on controlling CNC machines. Now PCs are used to network machines together in order to manage workloads. These networks also make it possible to communicate between machines and the office, to manage and download programs, to obtain machine status and operation reports in real time, and to monitor other network locations. Machine monitoring includes automatic operation, machine stop and feed hold, set-up, and alarm. Spindle load and spindle speed are recorded to provide reports and information on completed workpiece counts. Some machines utilize both a PC and CNC fused into one, providing bi-directional communication between the PC and the CNC. This communication enhances intelligent CNC control systems, so they can respond to questions, make suggestions, and provide detailed reports about machine operation and production status.

The most advanced RISC CPU (Reduced Instruction Set Computer, Central Processing Unit) technology is used to provide faster processing speeds, which, in turn, help to achieve reduced set-up and cycle times.

Knowledge-based navigation functions let us determine optimum metal cutting conditions prior to actual cutting, dependent upon stored data. Based on the part program, tool data, and workpiece material, the navigation functions suggest the optimum cutting conditions. They show where improvements in cycle time can be achieved through changes in spindle speed, feed rate, and tools.

Advancements in the graphical cutting simulation allow the 3-dimensional solid part model to be displayed. This feature can be used to show part sections for checking

Part 8 Mazatrol Conversational Programming
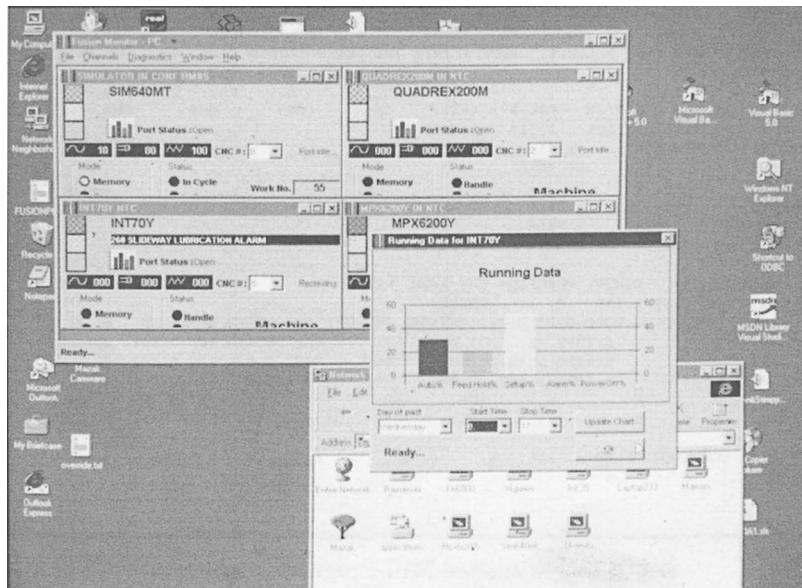
Figure 8-34 Machine Maintenance Window
Courtesy MAZAK Corp.

inside diameters and deep holes. Views can be rotated. These added capabilities aid in program verification of tool paths.

To keep machine utilization high, modern controls enable self-diagnostic functions for service and maintenance. Self-diagnostic menus quickly trouble shoot the cause of an alarm and suggest possible solutions for trouble-shooting. Alarm displays (Figure 8-35) indicate when scheduled maintenance (Figure 8-34) is required and on-line support is available.

As technological advancements continue at an unbelievable pace, the manufacturing industry will undoubt-

Figure 8-35 Machine Diagnostics Window
Courtesy MAZAK Corp.

edly benefit. Innovative new methods for CAD/CAM and Conversational programming of CNC machines will continue to emerge. For the machinist/programmer, this means that new methods and tools used for programming will require a life-long learning approach. This book has been intended to begin that learning process.

Part 8 Study Questions

1. A Turning Center program is made up of four basic parts:

   a. BAR, CPY, CNR, EDG

   b. Program, Shape Check, Tool Path, and Automatic Cycle

   c. Common Data Process, Machining Process, Sequence Data, and End Process

   d. Common Data Unit, Coordinate System, Machining Units, Sequence Data, and an End Unit

2. A Machining Center program is made up of these basic parts:

   a. BAR, CPY, CNR, EDG

   b. Program, Shape Check, Tool Path, and Automatic Cycle

   c. Common Data Process, Machining Process, Sequence Data, and End Process

   d. Common Data Unit, Coordinate System, Machining Units, Sequence Data, and an End Unit

3. If a 1/2-13 THD is to be turned, what is the lead?

4. What type of Machining Unit would be used to mill the contour of a part?

   a. Point Machining

   b. Line Machining

   c. Face Machining

   d. Manual Program

5. Mazatrol programming has a feature that will automatically develop tool sequences, based on the type of Machining Unit.

   T or F

Part 8 Mazatrol Conversational Programming

6. In a turning center program, where do you set the Maximum Spindle RPM Limit?

      a. END Unit

      b. Machining Process

      c. Common Data Process

      d. Sequence Data

7. Graphic simulation capabilities are limited to tool path centerline on current controllers.

      T or F

8. To set multiple Workpiece offsets to machine an array of parts, use the_____.

      a. WPC

      b. Multi 5x2

      c. Offset Type

      d. Manual Program

9. What does the acronym SRV-Z represent?

10. When AUTO SET is pressed from the Menu keys for the APRCH-X and APRCH-Y, why are question marks inserted into the program for the X and Y coordinates?

11. When creating a Turning Center program, the AUTO SET Menu key is pressed for Surface Velocity and Feedrate. On what criteria are these calculations based?

      a. Work Offset and Tool Material

      b. Work Material and Tool Material

      c. Spindle Speed and Feedrate

      d. Stock Removal Amount and Depth of Cut.

Appendices

Appendices

Appendix A-1: English Drill Sizes

Drill	Decimal	Drill	Decimal	Drill	Decimal	Drill	Decimal
80	0.0135	40	0.098	2	0.2210	33/64	0.5156
79	0.0145	39	0.0995	1	0.2280	17/32	0.5312
1/64	0.0156	38	0.1015	A	0.2340	35/64	0.5469
78	0.0160	37	0.104	15/64	0.2344	**9/16**	0.5625
77	0.0180	36	0.1065	B	0.2380	37/64	0.5781
76	0.0200	7/64	0.1094	C	0.2420	19/32	0.5938
75	0.0210	35	0.1100	D	0.2460	39/64	0.6094
74	0.0225	34	0.1110	E	0.2500	**5/8**	**0.625**
73	0.0240	33	0.1130	**1/4**	**0.2500**	41/64	0.6406
72	0.0250	32	0.1160	F	0.2570	21/32	0.6562
71	0.0260	31	0.1200	G	0.2610	43/64	0.6719
70	0.0280	**1/8**	0.1250	17/64	0.2656	**11/16**	0.6875
69	0.0292	30	0.1285	H	0.2660	45/64	0.7031
68	0.0310	29	0.1360	I	0.2720	23/32	0.7188
1/32	0.0312	28	0.1405	J	0.2770	47/64	0.7344
67	0.0330	9/64	0.1406	K	0.2810	**3/4**	**0.7500**
66	0.0350	27	0.1440	9/32	0.2812	49/64	0.7656
65	0.0350	26	0.1470	L	0.2900	25/32	0.7812
64	0.0360	25	0.1495	M	295	51/64	0.7969
63	0.0370	24	0.1520	19/64	0.2969	13/16	0.8125
62	0.0380	23	0.1540	N	0.3020	53/64	0.828
61	0.0390	5/32	0.1562	**5/16**	0.3125	27/32	0.8438
60	0.0400	22	0.1570	O	0.3160	55/64	0.8594
59	0.0410	21	0.1590	P	0.3230	**7/8**	0.8750
58	0.0420	20	0.1610	21/64	0.3281	57/64	0.8906
57	0.0430	19	0.1660	Q	0.3320	29/32	0.9062
56	0.0465	18	0.1695	R	0.3390	59/64	0.9219
3/64	0.0469	11/64	0.1719	11/32	0.3438	**15/16**	**0.9375**
55	0.0520	17	0.1730	S	0.3480	61/64	0.9531
54	0.0550	16	0.1770	T	0.3580	31/32	0.9688
53	0.0595	15	0.1800	23/64	0.3594	63/64	0.9844
1/16	**0.0625**	14	0.1820	U	0.3680	1	1.0000
52	0.0635	13	0.1850	**3/8**	**0.3750**		
51	0.0670	**3/16**	0.1875	V	0.3770	plus 1/64	
50	0.0700	12	0.1890	W	0.3860	increments	
49	0.0730	11	0.1910	25/64	0.3906	up to 1-7/8"	
48	0.0760	10	0.1935	X	0.3970		
5/64	0780	9	0.1960	Y	0.4040	plus 1/32	
47	0.0785	8	0.1990	13/32	0.4062	increments	
46	0.0810	7	0.2010	Z	0.4130	up to 2-1/4"	
45	0.0820	13/64	0.2031	27/64	0.4219		
44	0.0860	6	0.2040	**7/16**	**0.4375**	plus 1/16	
43	0.0890	5	0.2055	29/64	0.4531	increments	
42	0.0935	4	0.2090	15/32	0.4688	up to 4-1/4"	
3/32	0.0938	3	0.2130	31/64	0.4844		
41	0.0960	7/32	0.2188	**1/2**	**0.5000**		

Appendices

Appendix A-2: Metric Drill Sizes

Drill	Decimal	Drill	Decimal	Drill	Decimal	Drill	Decimal
0.35	.0138	2.50	.0984	6.20	.2441	**10.00**	**.3937**
0.38	.0150	2.55	.1004	6.25	.2461	10.20	.4016
0.40	.0157	2.60	.1024	6.30	.2480	10.50	.4134
0.42	.0165	2.65	.1043	6.40	.2520	10.80	.4252
0.45	.0177	2.70	.1063	6.50	.2559	**11.00**	**.4331**
0.48	.0189	2.75	.1083	6.60	.2598	11.20	.4409
0.50	.0197	2.80	.1102	6.70	.2638	11.50	.4528
0.55	.0217	2.90	.1142	6.75	.2657	11.80	.4646
0.60	.0236	**3.00**	.1181	6.80	.2667	**12.00**	**.4724**
0.65	.0256	3.10	.1220	6.90	.2717	12.20	.4803
0.70	.0276	3.20	.1260	**7.00**	**.2756**	12.50	.4921
0.75	.0295	3.25	.1280	7.10	.2795	**13.00**	**.5118**
0.80	.0315	3.30	.1299	7.20	.2835	13.50	.5315
0.85	.0335	3.40	.1339	7.25	.2854	**14.00**	**.5512**
0.90	.0354	3.50	.1378	7.30	.2874	14.50	.5709
0.95	.0374	3.60	.1417	7.40	.2913	**15.00**	**.5906**
1.00	.0394	3.70	.1457	7.50	.2953	15.50	.6102
1.05	.0413	3.75	.1476	7.60	.2992	**16.00**	**.6299**
1.10	.0433	3.80	.1496	7.70	.3031	16.50	.6496
1.15	.0453	3.90	.1564	7.75	.3051	**17.00**	**.6693**
1.20	.0472	**4.00**	**.1575**	7.80	.3071	17.50	.6890
1.25	.0492	4.10	.1614	7.90	.3110	**18.00**	**.7087**
1.30	.0512	4.20	.1654	**8.00**	**.3150**	18.50	.7283
1.35	.0531	4.25	.1693	8.10	.3189	**19.00**	**.7480**
1.40	.0551	4.30	.1732	8.20	.3228	19.50	.7677
1.45	.0571	4.40	.1772	8.25	.3248	**20.00**	**.7874**
1.50	.0591	4.50	.1811	8.30	.3268	20.50	.8071
1.55	.0610	4.60	.1850	8.40	.3307	**21.00**	**.8268**
1.60	.0630	4.70	.1870	8.50	.3346	21.50	.8465
1.65	.0650	4.75	.1890	8.60	.3386	**22.00**	**.8661**
1.70	.0669	4.80	.1920	8.70	.3425	22.50	.8858
1.75	.0689	4.90	.1929	8.75	.3445	**23.00**	**.9055**
1.80	.0700	**5.00**	**.1969**	8.80	.3465	23.50	.9252
1.85	.0728	5.10	.2008	8.90	.3504	**24.00**	**.9449**
1.90	.0748	5.20	.2047	**9.00**	**.3543**	24.50	.9646
1.95	.0768	5.25	.2067	9.10	.3583	**25.00**	**.9843**
2.00	**.0787**	5.30	.2087	9.20	.3622		
2.05	.0807	5.40	.2126	9.25	.3642	plus 1 mm	
2.10	.0827	5.50	.2165	9.30	.3661	increments	
2.15	.0846	5.60	.2205	9.40	.3701	up to 48 mm	
2.20	.0866	5.70	.2244	9.50	.3740		
2.25	.0886	5.75	.2264	9.60	.3780	plus 5 mm	
2.30	.0906	5.80	.2283	9.70	.3819	increments	
2.35	.0925	5.90	.2323	9.75	.3839	up to 105 mm	
2.40	.0945	**6.00**	**.2362**	9.80	.3858		
2.45	.0965	6.10	.2402	9.90	.3898		

Appendix A-3: English Threads

Thread Size*	Minor Diameter	Tap Drill 75% Thread	Decimal Equivalent	Thread Size*	Minor Diameter	Tap Drill 75% Thread	Decimal Equivalent
# 0-80	.0447	3/64	.0469	3/8-24	.3239	Q	.3320
# 2-56	.0641	#50	.0700	7/16-14	.3499	U	.3680
# 2-64	.0668	#50	.0700	7/16-20	.3762	25/64	.3906
# 4-40	.0813	#43	.0890	1/2-13	.4056	27/64	.4219
# 4-48	.0864	#42	.0935	1/2-20	.4387	29/64	.4531
# 5-40	.0943	#38	.1015	9/16-12	.4603	31/64	.4844
# 5-44	.0971	#37	.1040	9/16-18	.4943	17/32	.5312
# 6-32	.0997	#36	.1065	5/8-11	.5135	9/16	.5265
# 6-40	.1073	#33	.1130	5/8-18	.5568	37/64	.5781
# 8-32	.1257	#29	.1360	3/4-10	.6273	21/32	.6563
# 8-36	.1299	#29	.1360	3/4-16	.6733	11/16	.6875
# 10-24	.1389	#25	.1495	7/8-9	.7387	49/64	.7656
# 10-32	.1517	#21	.1590	7/8-14	.7874	53/64	.8281
1/4-20	.1887	#7	.2010	1-8	.8466	7/8	.8750
1/4-28	.2062	#3	.2130	1-12	.8987	15/16	.9375
5/16-18	.2443	F	.2570	1 1/8-7	.9497	63/64	.9843
5/16-24	.2614	I	.2720	1 1/8-12	1.0228	1 3/64	1.0469
3/8-16	.2983	5/16	.3125				
Coarse series threads shown in **BOLD**							
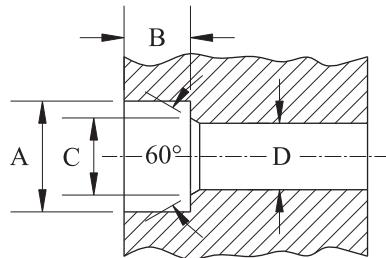
Appendix A-4: Metric Threads

Basic Thread Size	Tap Drill 75% Thread	Tap Drill Inch	Basic Thread Size	Tap Drill 75% Thread	Tap Drill Inch
M1.6x0.35	1.25mm	#55	M14x2	12.00mm	15/32
M2x0.4	1.60mm	#52	M14x1.5	12.50mm	1/2
M2.5x0.45	2.05mm	#46	M16x2	14.00mm	35/64
M3x0.5	2.50mm	#39	M16x1.5	14.50mm	37/64
M3.5x0.6	2.90mm	#32	M18x2.5	15.50mm	39/64
M4x0.7	3.30mm	#30	M18x1.5	16.50mm	21/32
M5x0.8	4.20mm	#19	M20x2.5	17.50mm	11/16
M6x1	5.00mm	#8	M20x1.5	18.50mm	47/64
M8x1.25	6.80mm	H	M22x2.5	19.50mm	49/64
M8X1.00	7.00mm	J	M22x1.5	20.50mm	13/16
M10x1.5	8.50mm	R	M24x3	21.00mm	53/64
M10x1.25	8.80mm	11/32	M24x2	22.00mm	7/8
M12x1.75	10.20mm	13/32	M27x3	24.00mm	15/16
M12x1.25	10.80mm	27/64	M27x2	25.00mm	1.00

Figure A-1: Counterbored Holes
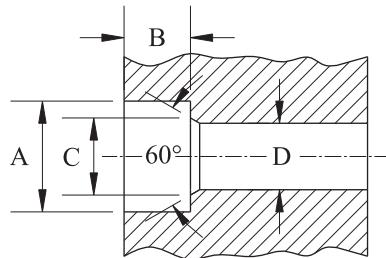
Appendix A-5: U.S. Socket Head Cap Screws					
SCREW DIA	A COUNTERBORE DIA	B COUNTERBORE DEPTH	C COUNTERSINK DIA	D CLEARANCE DIA	
				NORMAL FIT	CLOSE FIT
#0	1/8	.060	.074	#49	#51
#2	3/16	.086	.102	#36	3/32
#4	7/32	.112	.130	#29	1/8
#5	1/4	.125	.145	#23	9/64
#6	9/32	.138	.158	#18	#23
#8	5/16	.164	.188	#10	#15
#10	3/8	.190	.218	#2	#5
1/4	7/16	.250	.278	9/32	17/64
5/16	17/32	.312	.346	11/32	21/64
3/8	5/8	.375	.415	13/32	25/64
7/16	23/32	.438	.483	15/32	29/64
1/2	13/16	.500	.552	17/32	33/64
5/8	1	.625	.689	21/32	41/64
3/4	1-3/16	.750	.828	25/32	49/64
7/8	1-3/8	.875	.963	29/32	57/64
1	1-5/8	1.000	1.100	1-1/32	1-1/64
1-1/4	2	1.250	1.370	1-5/16	1-9/32
1-1/2	2-3/8	1.500	1.640	1-9/16	1-17/32
1-3/4	2-3/4	1.750	1.910	1-13/16	1-25/32
2	3-1/8	2.000	2.180	2-1/16	2-1/32
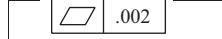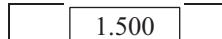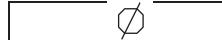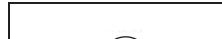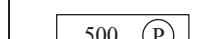
Figure A-1: Counterbored Holes

Appendix A-6: Metric Socket Head Cap Screws				
SCREW DIA	A COUNTERBORE DIA	B COUNTERBORE DEPTH	C COUNTERSINK DIA	D CLEARANCE DIA
				NORMAL FIT	CLOSE FIT
M1.6	3.50mm	1.6mm	2.0mm	1.95mm	1.80mm
M2	4.40mm	2mm	2.6mm	2.40mm	2.20mm
M2.5	5.40mm	2.5mm	3.1mm	3.00mm	2.70mm
M3	6.50mm	3mm	3.6mm	3.70mm	3.40mm
M4	8.25mm	4mm	4.7mm	4.80mm	4.40mm
M5	9.75mm	5mm	5.7mm	5.80mm	5.40mm
M6	11.20mm	6mm	6.8mm	6.80mm	6.40mm
M8	14.50mm	8mm	9.2mm	8.80mm	8.40mm
M10	17.50mm	10mm	11.2mm	10.80mm	10.50mm
M12	19.50mm	12mm	14.2mm	13.00mm	12.50mm
M14	22.50mm	14mm	16.2mm	15.00mm	14.50mm
M16	25.50mm	16mm	18.2mm	17.00mm	16.50mm
M20	31.50mm	20mm	22.4mm	21.00mm	20.50mm
M24	37.50mm	24mm	26.4mm	25.00mm	24.50mm
M30	47.50mm	30mm	33.4mm	31.50mm	31.00mm
M36	56.50mm	36mm	39.4mm	37.50mm	37.00mm
M42	66.00mm	42mm	45.6mm	44.00mm	43.00mm
M48	75.00mm	48mm	52.6mm	50.00mm	49.00mm
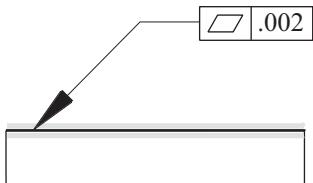
Appendix A-7: Geometric Symbols and Definitions

Refer to *Interpretation of Geometric Dimensioning and Tolerancing 3rd Edition*,
Based on ASME Y14.5-2009, ISBN978-0-8311-3421-1 for complete details.

Symbol	Definition
▱ .002 ⊕ ⌀ .005 Ⓜ A	**Feature-Control Frame** A specification box that shows a particular geometric characteristic (flatness, straightness, etc.) applied to a part feature and states the allowable tolerance. The feature's tolerance may be individual, or related to one or more datums. Any datum references and tolerance modifiers are also shown.
A	**Datum Feature** A flag which designates a physical feature of the part to be used as a reference to measure geometric characteristics of other part features.
A1 A2 A3	**Datum Targets** Callouts occasionally needed to designate specific points, lines, or areas on an actual part to be used to establish a theoretical datum feature.
1.500	**Basic Dimension** A box around any drawing dimension makes it a "basic" dimension, a theoretically exact value used as a reference for measuring geometric characteristics and tolerances of other part features.
⌀	**Cylindrical Tolerance Zone** This symbol, commonly used to indicate a diameter dimension, also specifies a cylindrically shaped tolerance zone in a feature-control frame.
Ⓜ	**Maximum Material Condition** Abbreviation: MMC. A tolerance modifier that applies the stated tight tolerance zone only while the part theoretically contains the maximum amount of material permitted within its dimensional limits (e.g. minimum hole diameters and maximum shaft diameters), allowing more variation under normal conditions.
Ⓛ	**Least Material Condition** Abbreviation: LMC. A tolerance modifier that applies the stated tight tolerance zone only while the part theoretically contains the minimum amount of material permitted within its dimensional limits (e.g. maximum hole diameters and minimum shaft diameters), allowing more variation under normal conditions.
Ⓢ	**Regardless of Feature Size** Abbreviation: RFS. A tolerance modifier that applies the stated tight tolerance zone under all size conditions. RFS is generally assumed if neither MMC nor LMC are stated.
.500 Ⓟ	**Projected Tolerance Zone** An additional specification box attached underneath a feature-control frame. It extends the feature's tolerance zone beyond the part's surface by the stated distance, ensuring perpendicularity for proper alignment of mating parts.

Appendix A-8: Geometric Characteristics

◻ .002
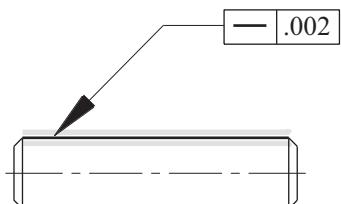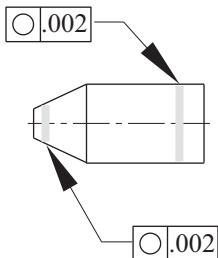
▱ **Flatness**

All points on the indicated surface must lie in a single plane, within the specified tolerance zone.

— .002
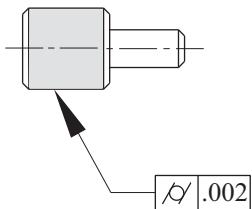
— **Straightness**

All points on the indicated surface or axis must lie in a straight line in the direction shown, within the specified tolerance zone

◯.002

◯ .002

◯ **Circularity (Roundness)**

If the indicated surface were sliced by any plane perpendicular to its axis, the resulting outline must be a perfect circle, within the specified tolerance zone.
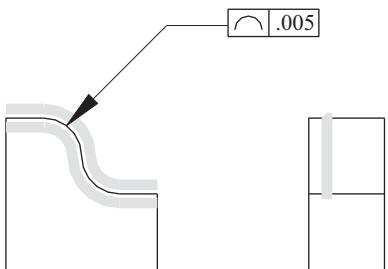
⌭ **Cylindricity**

All points on the indicated surface must lie in a perfect cylinder around a center axis, within the specified tolerance zone.

⌭ .002

Refer to *Interpretation of Geometric Dimensioning and Tolerancing, 3rd Edition,* Based on ASME Y14.5-2009, ISBN978-0-8311-3421-1 for complete details.

Appendix A-8: Geometric Characteristics *(continued)*

⌒ .005

⌒ **Linear Profile**

All points on any full slice of the indicated surface must lie on its theoretical two-dimensional profile, as defined by basic dimensions, within the specified tolerance zone. The profile may or may not be oriented with respect to datums.

⌒ .005 A

–A–

⌓ .005
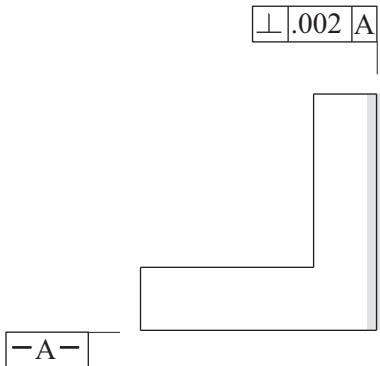
⌓ **Surface Profile**

All points on the indicated surface must lie on its theoretical three-dimensional profile, as defined by basic dimensions, within the specified tolerance zone. The profile may or may not be oriented with respect to datums.

⌓ .005 A

–A–

Refer to *Interpretation of Geometric Dimensioning and Tolerancing, 3rd Edition,* Based on ASME Y14.5-2009, ISBN978-0-8311-3421-1 for complete details.

Appendix A-8: Geometric Characteristics *(continued)*

⊥ | .002 | A
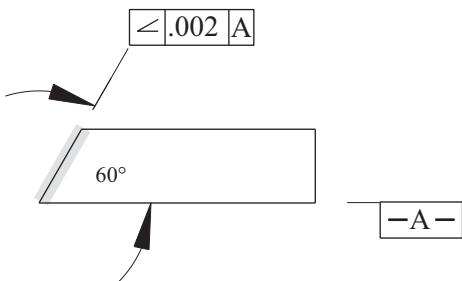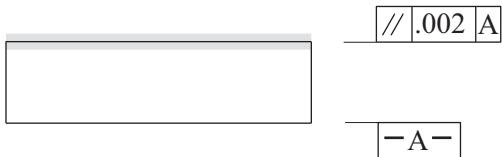
⊥ **Perpendicularity (Squareness)**

All points on the indicated surface, axis, or line must lie in a single plane exactly 90° from the designated datum plane or axis, within the specified tolerance zone.

−A−

∠ | .002 | A

∠ **Angularity**

60°

−A−

All points on the indicated surface or axis must lie in a single plane at exactly the specified angle from the designated datum plane or axis, within the specified tolerance zone.

// | .002 | A

// **Parallelism**

−A−
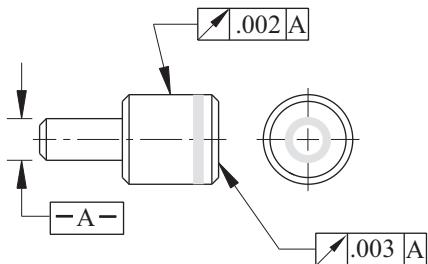
All points on the indicated surface or axis must lie in a single plane parallel to the designated datum plane or axis, within the specified tolerance zone.

Refer to *Interpretation of Geometric Dimensioning and Tolerancing, 3rd Edition,* Based on ASME Y14.5-2009, ISBN978-0-8311-3421-1 for complete details.
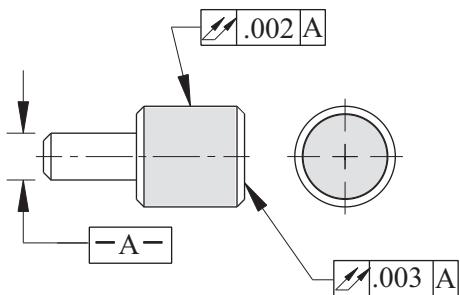
Appendix A-8: Geometric Characteristics *(continued)*
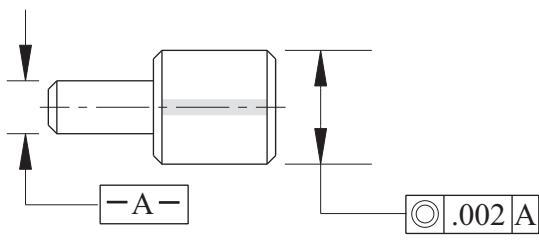
Circular Runout

Each circular element of the indicated surface is allowed to deviate only the specified amount from its theoretical form and orientation during 360° rotation about the designate datum axis.

.002 A

.003 A

−A−
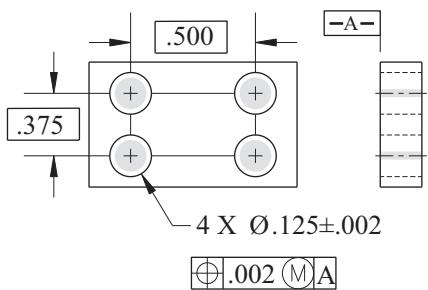
Total Runout

The entire indicated surface is allowed to deviate only the specified amount from its theoretical form and orientation during 360° rotation about the designated datum axis.

.002 A

.003 A

−A−

Concentricity

If the indicated surface were sliced by any plane perpendicular to the designated datum axis, every slice's center of area must lie on the datum axis. within the specified cylindrical tolerance zone (controls rotational balance).

−A−

.002 A
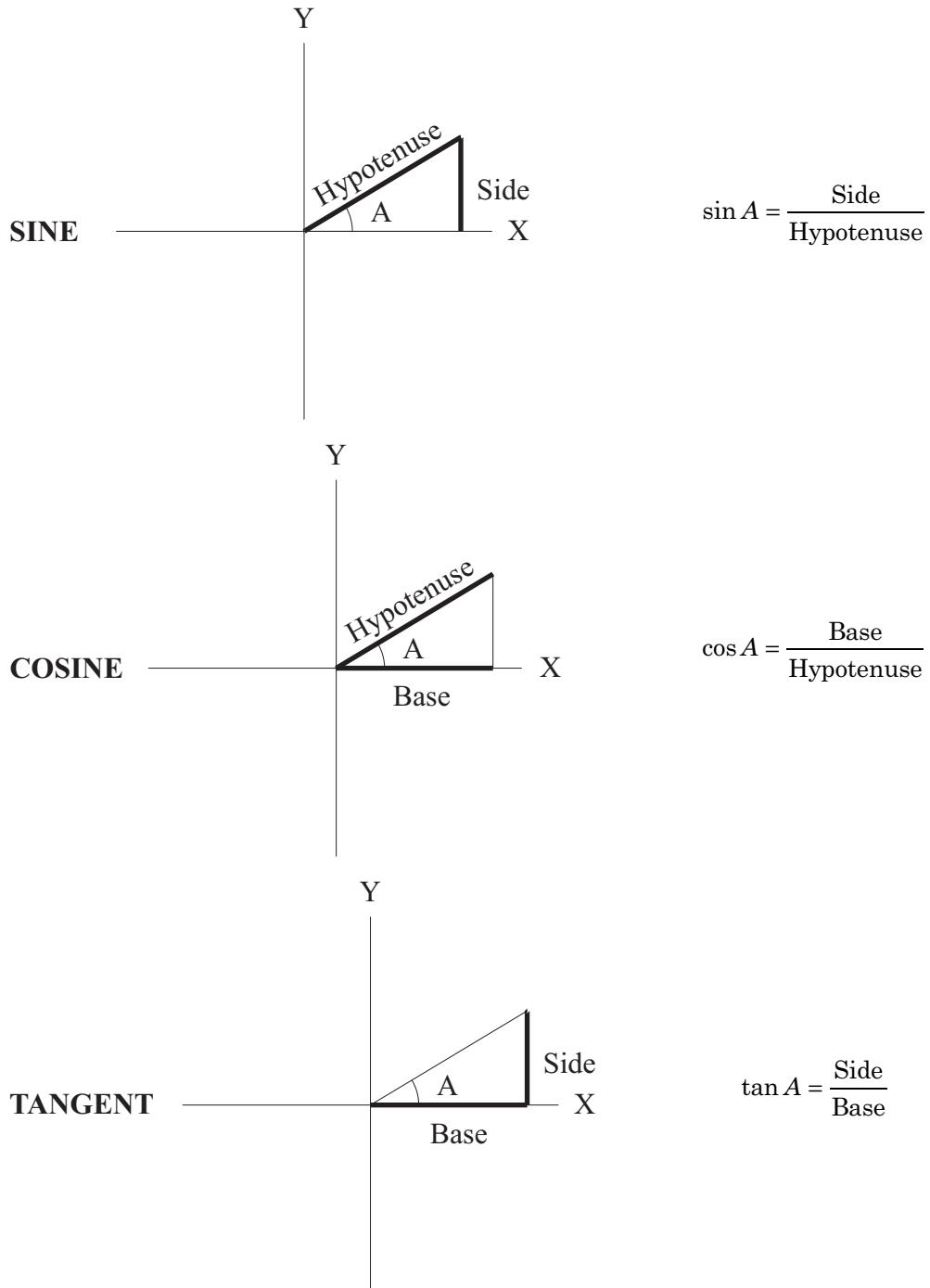
Position (Replaces Symetry)

The indicated feature's axis must be located within the specified tolerance zone from its true theoretical position, correctly oriented relative to the designated datum plane or axis.

.500

.375

−A−

4 X Ø.125±.002

.002 M A
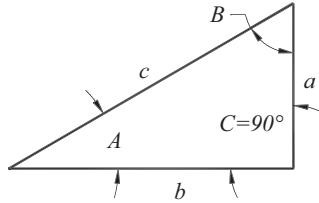
Refer to *Interpretation of Geometric Dimensioning and Tolerancing, 3rd Edition,* Based on ASME Y14.5-2009, ISBN978-0-8311-3421-1 for complete details.

Appendix A-9: Trigonometry Functions

SINE

$$\sin A = \frac{\text{Side}}{\text{Hypotenuse}}$$

COSINE

$$\cos A = \frac{\text{Base}}{\text{Hypotenuse}}$$

TANGENT

$$\tan A = \frac{\text{Side}}{\text{Base}}$$
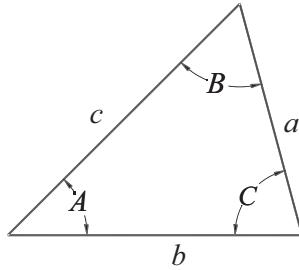
Appendix A-10: Right Triangles

Known Sides and Angles	Unknown Sides and Angles			Area
a and b	$c = \sqrt{a^2 + b^2}$	$A = \arctan \dfrac{a}{b}$	$B = \arctan \dfrac{b}{a}$	$\dfrac{a \times b}{2}$
a and c	$b = \sqrt{c^2 - a^2}$	$A = \arcsin \dfrac{a}{c}$	$A = \arccos \dfrac{a}{c}$	$\dfrac{a \times \sqrt{c^2 - a^2}}{2}$
b and c	$a = \sqrt{c^2 - b^2}$	$A = \arccos \dfrac{b}{c}$	$B = \arcsin \dfrac{b}{c}$	$\dfrac{b \times \sqrt{c^2 - b^2}}{2}$
a and $\angle A$	$b = \dfrac{a}{\tan A}$	$c = \dfrac{a}{\sin A}$	$B = 90° - A$	$\dfrac{a^2}{2 \times \tan A}$
a and $\angle B$	$b = a \times \tan B$	$c = \dfrac{a}{\cos B}$	$A = 90° - B$	$\dfrac{a^2 \times \tan B}{2}$
b and $\angle A$	$a = b \times \tan A$	$c = \dfrac{b}{\cos A}$	$B = 90° - A$	$\dfrac{b^2 \times \tan A}{2}$
b and $\angle B$	$a = \dfrac{b}{\tan B}$	$c = \dfrac{b}{\sin B}$	$A = 90° - B$	$\dfrac{b^2}{2 \times \tan B}$
c and $\angle A$	$a = c \times \sin A$	$b = c \times \cos A$	$B = 90° - A$	$c^2 \times \sin A \times \cos$
c and $\angle B$	$a = c \times \cos B$	$b = c \times \sin B$	$A = 90° - B$	$c^2 \times \sin B \times \cos$

Appendix A-11: Oblique Triangles

Known Sides and Angles	Unknown Sides and Angles			Area
All three sides a, b, c	$A = \arccos\dfrac{b^2 + c^2 - a^2}{2bc}$	$B = \arcsin\dfrac{b \times \sin A}{a}$	$C = 180° - A - B$	$\dfrac{a \times b \times \sin C}{2}$
Two sides and the angle between them $a, b, \angle C$	$c = \sqrt{a^2 + b^2 - (2ab \times \cos C)}$	$A = \arctan\dfrac{a \times \sin C}{b - (a \times \cos C)}$	$B = 180° - A - C$	$\dfrac{a \times b \times \sin C}{2}$
Two sides and the angle opposite one of the sides $a, b, \angle A$ ($\angle B$ less than 90°)	$B = \arcsin\dfrac{b \times \sin A}{a}$	$C = 180° - A - B$	$c = \dfrac{a \times \sin C}{\sin A}$	$\dfrac{a \times b \times \sin C}{2}$
Two sides and the angle opposite one of the sides $a, b, \angle A$ ($\angle B$ greater than 90°)	$B = 180° - \arcsin\dfrac{b \times \sin A}{a}$	$C = 180° - A - B$	$c = \dfrac{a \times \sin C}{\sin A}$	$\dfrac{a \times b \times \sin C}{2}$
One side and two angles $a, \angle A, \angle B$	$b = \dfrac{a \times \sin B}{\sin A}$	$C = 180° - A - B$	$c = \dfrac{a \times \sin C}{\sin A}$	$\dfrac{a \times b \times \sin C}{2}$

Appendix A-12: Popular Acronyms

ANSI	American National Standards Institute
AOS	Algebraic Order System
APT	Automatically Programmed Tools
ASCII	American Standard Code for Information Interchange
ASME	American Society of Mechanical Engineers
ATC	Automatic Tool Changer
AVI	Audio Video Interleave
BCD	Binary Coded Decimal
CAD	Computer-Aided Design
CAD/CAM	Computer-Aided Design and Computer-Aided Manufacturing
CAM	Computer-Aided Manufacturing
CCW	Counterclockwise
CD-ROM	Compact Disc – Read Only Memory
CD-RW	Compact Disc – Re-Writable
CDC	Cutter Diameter Compensation
CIM	Computer-Integrated Manufacturing
CMM	Coordinate Measuring Machine
CNC	Computer Numerical Control
CPL	Coordinate Position Locator
CPU	Central Processing Unit
CRC	Cutter Radius Compensation
CRT	Cathode Ray Tube
CSS	Constant Surface Speed
CW	Clockwise
DNC	Direct Numerical Control
DVD	Digital Video Disc
DXF	Drawing Exchange Format
EDM	Electronic Discharge Machine
EIA	Electronics Industries Association
EOB	End of Block
FBM	Feature-Based Machining
FMS	Flexible Manufacturing System
FPT	Feed per Tooth
GB	Gigabit
G-Code	Preparatory Functions (commands)

Appendices

GD&T	Geometric Dimensioning & Tolerancing
GHz	Gigahertz
GUI	Graphical User Interface
HMI	Human Machine Interface
HP	Horsepower
HSS	High Speed Steel
HSK	"Hohl Schaft Kegel" Hollow Taper Shank
ID	Inside Diameter
IGES	Initial Graphics Exchange Specification
IPM	Inches per Minute (also in/min)
IPR	Inches per Revolution (also in/rev)
IPW	In-Process Workpiece
ISO	International Standards Organization
KW	Kilowatt
LAN	Local Area Network
LCD	Liquid Crystal Display
LED	Light Emitting Diode
MB	Megabit
MB1, 2, or 3	Mouse Button 1 = left, 2 = Scroll or 3 = right
M-Codes	Miscellaneous Functions
MCS	Machine Coordinate System
MCT	Machine Center Tool (Pocket)
MCU	Machine Control Unit
MDI	Manual Data Input
MGi	Manual Guide i
MHz	Megahertz
MKE	Machining Knowledge Editor
MKL	Machining Knowledge Library
Mm	Millimeters
MPG	Manual Pulse Generator
MRU	Most Recently Used
NC	Numerical Control
OD	Outside Diameter
PC	Personal Computer
PCD	Polycrystalline Diamond
PCMCI	Portable Computer Memory Card Interface
PH	Acidity or Alkalinity of Coolant

Appendices

PLC	Programmable Logic Controller
PMI	Product Manufacturing Information
PSI	Pounds per Square Inch
PVD	Physical Vapor Deposition
PWE	Parameter Write Enable
R-8	Taper Designation
RAM	Random Access Memory
RFID	Radio Frequency Identification
RISC	Reduced Instruction Set Computer
ROM	Read Only Memory
RPM	Revolutions per Minute (also rev/min or r/min)
RS232	Industry Standard Cabling Interface
SFM	Surface Feet/Minute
SME	Society of Manufacturing Engineering
STL	Sterolithography
TiCN	Titanium Carbon Nitride
TiN	Titanium Nitride
TLO	Tool Length Offset
TNRC	Tool Nose Radius Compensation
TPM	Total Productive Maintenance
UDF	User Defined Feature
UI	User Interface
USB	Uniform Serial Bus
WCS	Work Coordinate System

Please also see Part 8 of this text for acronyms specific to Mazatrol Conversational Programming.

Glossary

A-Axis

The *A*-axis is an auxiliary rotary axis that rotates about the *X*-axis. Angular movements are specified in decimal degrees. Positive angular values refer to counter-clockwise rotation and negative angular values indicate clockwise rotation.

Absolute Dimension

All numerical values (dimensional measurements) are derived from a fixed origin or datum in the coordinate system.

Address

Commonly referred to as letter address because, in programming, each program word is preceded by a letter in order to identify what function is to be executed. Examples of letter address are: S for spindle speed designation, T for tool identification, M for miscellaneous functions, and G for preparatory functions.

Auxiliary axis

An auxiliary axis is any axis that is in addition to the primary axes of X, Y, and Z. These axes can be rotary (A, B, or C) or linear (U, V, or W). They are also called secondary axes.

Axis

The axis is the primary identifier of the cutting tool direction of movement in relationship to the machine type and orientation. The three linear axes for a machining center are X, Y, and Z; they are perpendicular to each other. The rotary axes are; A, B, and C.

B-Axis

The *B*-axis is an auxiliary rotary axis that rotates about the *Y*-axis. Angular movements are specified in decimal degrees. Positive angular values refer to counter-clockwise rotation and negative angular values indicate clockwise rotation.

Block

A single line of CNC code consisting of program words that identify the activities the machine is to execute. Generally, each block is preceded by a block or sequence number (N) and is followed by an "End of Block" (EOB) character, represented by the semicolon (;).

Glossary

Block Skip

Block Skp is sometimes called Block Delete or Optional Block Skip. When the Block Delete or Optional Block Skip button or switch is on, the controller skips execution of the program blocks that are preceded with "/"and that end with the end of block (;) character. If the button or switch is off, the machine will execute the programmed blocks and disregards the "/" symbol.

C-Axis

The *C*-axis is an auxiliary rotary axis that rotates about the *Z*-axis. Angular movements are specified in decimal degrees. Positive angular values refer to counterclockwise rotation and negative angular values clockwise.

Canned Cycle

The function of a given cycle is defined as a set of operations assigned to one block and performed automatically without any possibility of interruption. Examples of canned cycles are: Cycle G81, which will perform a simple drill cycle ,and G84, which will perform tapping. Canned cycles require additional information such as coordinate locations, reference plane values, and peck amounts. Canned cycles simplify the part program by decreasing programming time. Another name for Canned cycles is fixed cycles.

Cartesian Coordinates

A coordinate system that consists of three axes (X, Y, and Z) that are perpendicular to each other. A grid is formed consisting of numerical graduations, representing the distances from the intersection of the three axes (called the origin).

Chamfer

A beveled cut of 45 degrees on the edge of a part design.

Circular Interpolation

A programming feature that enables programming of two axes simultaneously to create arcs and circles. Information generally needed includes the location of the arc center, the arc radius, the starting and ending points of the arc, and the direction of cutting motion.

Coordinates

Numerical values that define the positional location of points from a predetermined zero point or origin from within the Cartesian Coordinate System.

Datum

A datum is an exact point, axis, or plane. A datum is the origin from which the dimensional location and/or the characteristics of features of a part are established.

Dry Run

Sometimes the CNC part program is executed with *no part mounted*, to verify the programmed path of the tool under automatic operation. The typical form of dry run is

set by activating the DRY RUN function on the control during automatic cycle, where all of the rapid and work feeds are changed to the rapid traverse feed set in the parameters instead of the programmed feed. DRY RUN is also used to check a new program on the machine without any work actually being performed by the tool. This is particularly useful on programs with long cycle times so the operator can progress through the program more quickly.

Dwell

Dwell is determined by the preparatory function G04 and by using the letter address P or X, which corresponds to the time duration of dwell (also U for lathes). When used, dwell causes a pause in the machining operation for the length of time indicated in seconds (X or U), milliseconds (P), or revolutions (depending on parameter setting).

End of Block Character

A special character represented by the semicolon (;) that identifies the end of a program block. Known by the acronym EOB or E-O-B.

End of program

A miscellaneous function (M30) is placed in the last line of a program to indicate the end of the part program. At this command, the spindle, coolant, and feed are stopped and the program is returned to its start.

F-Word

The F-Word is utilized to determine the work feed ,rates (cutting feedrates). This program word is used to establish feed rate values and precedes a numeric input for the feed amount in Inches per Minute (IPM), or Inches per Revolution (IPR), and Meters per Minute (m/min), or Millimeters per Revolution (mm/rev) for metric programs. The value that is set by this command stays effective until changed by reentering a new value.

Feedrate Override

Feedrate Override allows control of the traverse feedrate by adjustment of a rotary dial. This function allows the control of the cutting feedrates defined by the F-word in the program by increasing or decreasing the percentage of the value entered in the program. It can also be used to control feedrates during jog mode function.

Fillet

A rounding of an interior (concave) or exterior edge (convex) of a part design, sometimes called rounds.

Fixed Cycle

See Canned Cycle

G-Codes

Preparatory functions (G-Codes) are programmed with an address G, typically followed by two digits, to establish the mode of operation in which the tool moves.

Glossary

Gnomen

A gnomon is a graphical representation of three perpendicular axes connected at the origin; it allows the programmer the ability to manipulate model transformations dynamically.

Incremental Dimension

An incremental dimension is a position within the coordinate system in which each numerical value is taken from the previous point.

Jog

Activating the JOG feed mode allows the selection of manual feeds along a single *X, Y,* or *Z* axis (rotary axes may be jogged as well). With the mode activated, use the Axis/Direction buttons and the Speed/Multiply buttons to move the desired axis at the chosen feed rate (in/min or mm/min) and amount.

Linear Interpolation

This function allows programming of one, two, or three axes simultaneously; it enables movement either along a straight-line path or at an angle in plane or space.

Machine Home

A reference position located within the machine tool working envelope determined by the manufacturer, in order to establish a measurement system for the machine.

Machine Lock

An operation panel control, usually a button or a toggle switch that allows the operator to lock all of the axes movements in order to check long programs for errors. If an error is encountered during this process, an alarm will be displayed on the control.

Manual Data Input (MDI)

The MDI mode enables the automatic control of the machine, using information entered in the form of blocks through the control panel without interfering with the basic program.

Miscellaneous Function

Miscellaneous functions are used to command various auxiliary operations such as activating coolant flow (M08) or starting clockwise spindle rotation (M03). The code consists of the letter M, typically followed by two digits. The M-Code is normally the last entry in a block.

Modal Commands

Modal commands remain in effect until they are replaced by another command from the same group. The F-Word is modal as are many G-Codes.

Glossary

Origin

A starting point for the coordinate system used to machine parts; a fixed point on a blueprint from which dimensions are taken.

Polar Coordinate System

A rotational coordinate system that locates points within a plane with respect to their distance from a fixed point of origin or pole by angular and radial values.

Preparatory Function (G-Word)

See G-Codes

Quadrant

A quadrant is one fourth of a 2-dimensional grid in a plane of the rectangular coordinate system for measurement. It also represents an arc of 90° that is one fourth of a circle. Quadrant 1 is located in the upper right corner and quadrants 2 through 4 proceed in the counterclockwise direction.

Rectangular Coordinate System

See Cartesian Coordinates

Right-Hand Rule

Using the right hand with palm up, the thumb will be pointing in the positive linear axis direction for *X,* then the little and ring fingers of the hand are folded over to touch the palm, the middle finger is allowed to point upwards (positive Z-axis) and the index finger is pointing in the *Y*-axis positive direction (vertical machine orientation).

Sequence Number (N-Word)

Sequence numbers (also called the block numbers) are identified by the letter N and are followed by one to five digits. A block number provides easier access to information contained in the program. The arrangement of block numbers in a given program can be random, but typically is sequenced in increments of one, two, five, or ten. The most common step increments are five or ten. Block or sequence numbers can be omitted from a block (except in special cases, such as in some lathe cycles). The logical location for sequence numbers is at tool changes enabling the restart of that tool.

Single Block

The execution of a single block of information in the program is initiated by activating a switch or button on the control panel. While in this mode, each time the cycle start button is pressed, only one block of information will be executed.

Sub-Program

The subprogram is a subordinate program to the main program. It is registered in the controller memory with the letter O, followed by a four- or five-digit number, same as the main program. In the main program, a subprogram is called by using the M98 function with the P-address to identify the sub-program number. M99 (called for in the

Glossary

subprogram) is then the function that ends a subprogram. Subprograms greatly simplify programming and decrease the amount of data that must be placed into the controller memory.

Tool Changer

A mechanical apparatus used to automatically change cutting tools by program control on CNC machines. The tool changer may be a magazine-type, with random access or, a carousel. The magazine-type uses a device similar to a robot arm that transfers tools from the magazine to the spindle and vice versa.

Tool Length Offset (TLO)

Tool Length Offsets, or TLO, are called in the milling program by the H word. The measured values representing the difference between the spindle face gage line and the tools tip are input into corresponding offset registers and are needed for proper positioning of the tool along the Z-axis.

U-Axis

An additional linear axis parallel to the *X*-axis.

V-Axis

An additional linear axis parallel to the *Y*-axis.

W-Axis

An additional linear axis parallel to the *Z*-axis.

Work envelope

The working envelope is the maximum area the machine travels for two axes in all four directions.

Workpiece Zero

A starting point, workpiece zero is the point from which dimensions on the workpiece are established. Sometimes referred to as part zero.

X-Axis

The axis of motion that is always horizontal and parallel to the machine tool table. For a vertical milling machine, this axis moves left or right.

Y-Axis

The axis of motion that is perpendicular to both *X*- and *Z*-axes in relation to the machine tool table. For a vertical milling machine, this axis moves forward and backward.

Z-Axis

The machine tool axis of motion that is always parallel to the primary spindle. For a vertical milling machine, this axis moves up and down.

Programming of CNC Machines 4th Edition
Answers to Study Questions

Part 1, **1.** T, **2.** c, **3.** b, **4.** c, **5.** d, **6.** a, **7.** F, **8.** 3, **9.** T, **10.** T, **11.** X0Y0, X0Y5.0, X2.5Y5.0, X4.0Y3.5, X4.0Y0, X0Y0 **12.** X0Y0, Y5.0, X2.5, X1.5Y–1.5, Y–3.5, X–4.0 **13.** Daily

Part 2, 1. T, **2.** Position (All) or Program (Check)**, 3.** Active commands**, 4.** Pressing the Input key enters a whole number where +Input enters an incremental amount**, 5.** c, **6.** d,**7.** T, **8.** b, **9.** a, **10.** Manual Data Input**, 11.** Offset**, 12.** T, **13.** Jog

Part 3, **1.** c, **2.** T, **3.** The first two digits refer to the geometry offset and the second two are to the wear offset. **4.** b, **5.** d, **6.** T, **7.** b, **8.** T, **9.** a, **10.** b, **11.** c, **12.** b, **13.** b, **14.** a negative sign, **15.** T, **16.** d, **17.** b, **18.** d, **19.** b **20.** 3; 6; 8; 6, **21.** c, **22.** Contour profile points may be used

Part 4, 1. b, **2.** d, **3.** c, **4.** d, **5.** b, **6.** d, **7.** c, **8.** T, **9.** d, **10.** c, **11.** d, **12.** a, **13.** T, **14.** F, **15.** b, **16.** b, **17.** a, **18.** c, **19.** d, **20.** a, **21.** a. M00, b. M01, c. M30, d. M03, e. M04, f. M05, g. M08, h. M09, i. M19, j. M98, k. M99, l. M06, **22.** a. G01, b. G02, c. G00, d. G04, e. G90, f. G91, g. G80, h. G83, i. G81, j. G41, k. G42, l. G40, m. G28, n. G20, o. G21, p. G84, q. G92, r. G54, s.G43, **23.** a. O, b. N, c. G, d. M, e. X, f. Y, g. Z, h. R, i. F, j. S, k. L, l. T, m. H, n. D o. A, p. B, q. C, r. P, s. Q, t. I, u. J, v. K, **24.** c, **25.** T, **26.** T **27.** 25–140 SFM, .001–.004 FPT

Part 5, 1. T, **2.** T, **3.** d, **4.** A fillet is a convex machined radius where two surfaces meet at an angle. **5.** via the Menu Bar by pressing Create and then Fillet or the fillet button on the Sketcher Toolbar. **6.** A chamfer is a beveled edge, normally machined at 45º to the adjacent surfaces. This operation is commonly used to remove burrs and eliminate sharp corners around holes or part edges. **7.** via the Menu Bar by pressing Create and then Chamfer or the Chamfer button on the Sketcher Toolbar. **8.** c, **9.** c, **10.** a, **11.** T, **12.** d, **13.** To aid in the selection of specific characteristics of entities by snapping to them when activated and picked with the mouse. **14.** The software compensates the tool path by the radial amount based on the tool selected. **15.** a or d, **16.** T, **17.** Angle, **18.** T **19.** To use G41 or G42 Tool Compensation **20. 21.** Dynamic XForm

Part 6, 1. FBM is the automated process of defining machining operations from modeled features based on best practices that are established in a Machining Knowledge Library, **2.** Repetitive tasks, **3.** IGES, STEP, STL, **4.** Specify the Part and the raw material Blank, Use Feature Finder to Find Parametric Features, Create Feature Processes and Operations, Generate Tool Paths, Review and Edit as needed, Verify Toolpaths, Find Features again for any UDFs, Repeat above steps, Post Process, Complete Setup Documentation, Release to Production, **5.** T, **6.** c, **7.** T, **8.** T**, 9.** d, **10.** F

Answers to Study Questions

Part 7, 1. Parameter Write Enable, **2.** NC Guide i is the PC version and Manual Guide I is installed on the controller. They are virtually identical in function**, 3.** T, **4.** To identify if the tool can cut the geometry, **5.** To reuse repeating lines of code and shorten data entry, **6.** To force the input of variable data, **7.** ALTER, **8.** F, **9.** Tool Offset, **10.** T, **11.** T, **12.** POS/SIZE, **13.** ALTER

Part 8, 1. c, **2. d, 3.** .0769, **4.** b, **5.** T, **6.** c, **7.** F, **8.** b, **9.** The amount of stock removal in Z, **10.** At this point, the tool to be used has not been selected. The values will be input when the tool path is checked after the unit is completed. **11.** b

Index

Index

Index

Index

Index

Index

Index

Index

Index