FAGOR 800T CNC
OPERATING MANUAL

Ref. 9701 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual is addressed to the machine operator.

It includes the necessary information for new users as well as advanced subjects for those who are already familiar with the 800T CNC product.

It may not be necessary to read this whole manual. Consult the list of "New Features and Modifications".

This manual explains all the functions of the 800T CNC family. Consult the Comparison Table for the models in order to find the specific ones offered by your CNC.

Chapters 1, 2, 3 and 4 show how to operate with this CNC.

This CNC permits machining the "Profile of a part", chapter 6, or perform a series of "Automatic Operations" detailed in chapter 5. All these machining operations may be carried out in two ways:

* "Semi-automatically", where the operator controls the movements.
* "Automatically" (Cycle level), where the operator programs the operation and the CNC executes it automatically.

Chapter 7 "Working with Part-programs" shows how to create parts consisting of Profiles and Automatic Operations. The Part-programs are stored in the internal CNC memory and may be sent out to a peripheral device or PC.

Chapter 8 "Programming examples" shows how to edit several part-programs.

There is also an appendix on error codes which indicates some of the probable reasons which could cause each one of them.

Notes: The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

INDEX

Section | Page

INTRODUCTION

Chapter 1. CONCEPTS

Chapter 2. BASIC OPERATIONS

Section | Page

Chapter 3. AUXILIARY FUNCTIONS

Chapter 4. SPINDLE

Section Page

Chapter 5. AUTOMATIC OPERATIONS

Chapter 6. PROFILES

Chapter 7. WORKING WITH PART-PROGRAMS

Chapter 8. PROGRAMMING EXAMPLES

ERROR CODES

COMPARISON TABLE FOR FAGOR 800T CNC MODELS

AVAILABLE 800T CNC MODELS

Compact model with 8" amber CRT.

Modular model with 9" amber Monitor.

Consisting of Central Unit, Monitor and Keyboard.

Modular model with 14" Color Monitor

Consisting of Central Unit, Monitor and Keyboard.

TECHNICAL DESCRIPTION

	800-T	800-TI	800-TG	800-TGI
X, Z axes control	l	l	l	l
Spindle control	l	l	l	l
Spindle in RPM	l	l	l	l
Constant Surface Speed (CSS)	l	l	l	l
Spindle Orientation	l	l	l	l
Tools	32	32	32	32
Tool Compensation	l	l	l	l
Live Tool	l	l	l	l
Electronic Handwheels	2	2	2	2
RS 232C Communications	l	l	l	l
Integrated PLC (PLCI)		l		l
ISO-coded program editing (P99996)	l	l	l	l
Execution of ISO-coded program (P99996)	l	l	l	l
Graphics			l	l

NEW FEATURES AND MODIFICATIONS

Date: **April 1993** ***Software Version:*** **2.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
Rapid jog depending on position of Feedrate Override Switch	Operating Manual	Section 2.3.1
Tool for the finishing pass	Installation Manual Operating Manual	Section 3.5 Section 3.5
Handwheel movement limited to maximum allowed F	Operating Manual	Section 2.3.3
Control of software travel limits when using a handwheel		
Display format for S	Installation Manual	Section 6
Possibility to activate/deactivate outputs O1, O2, O3 after interrupting the program		
Automatic operation "Profile Rounding"	Operating Manual	Section 5.5.3
Profiles	Operating Manual	Chapter 6

Date: **October 1993**	***Software Version:*** **3.1 and newer**	
FEATURE	**AFFECTED MANUAL AND SECTION**	
Spindle acc./dec.	Operating Manual	Chapter 6
RPM Limitation when operating in CSS	Operating Manual	Section 4.3.1
Spindle orientation	Installation Manual Operating Manual	Section 6.4.1 Section 4.8
Live tool	Installation Manual Operating Manual	Section 5.9 Section 2.3
Automatic operation "Simple Drilling"	Operating Manual	Section 5.8
Automatic operation "Multiple Drilling"	Operating Manual	Section 5.9

Date: **December 1993** ***Software Version:*** **3.2 and newer**

FEATURES	**AFFECTED MANUAL AND SECTION**	
Assign a 5-digit number to the part program	Operating Manual	Chapter 7
Save part programs out to a peripheral	Operating Manual	Section 7.7
Automatic operation "Slot milling"	Operating Manual	Section 5.10
Delay before opening the positioning loop	Installation Manual	Section 4.3.2
"Special modes" accessing password	Installation Manual	Section 3.7
Handwheel inactive when Feedrate Override Switch out of handwheel positions	Installation Manual	Section 4.3.2

Date: **July 1994** ***Software Version:*** **4.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
Linear and Bell-shaped spindle acc./dec.	Installation Manual	Section 5.8
Profile with/without corner rounding.	Operating Manual	Section 6.2
Threading operation also with thread exit.	Operating Manual	Section 5.6.2
Rapid jog at 200% or depending on the position of the Feedrate Override Switch.	Installation Manual Operating Manual	Section 4.3.3 Section 2.3.1
Tool inspection	Installation Manual Operating Manual Operating Manual	Section 3.4.3 Section 3.4.3 Section 5.1.3
Execution of program 99996	Installation Manual Operating Manual	Section 3.11 Section 3.10

Date: **January 1995** ***Software version:*** **5.1 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
M3/M44 confirmation by detecting feedback reversal	Installation Manual	Section 6.4
JOG movements also in mm/rev		
Handwheel governed by the PLCI	Installation Manual	Section 4.3.2
Spindle inhibit from PLCI	PLCI Manual	
Clear all arithmetic parameter contents setting them to "0".	Installation Manual Operating Manual	Section 3.10 Section 3.9 & 7.9
Automatic rounding operation (Cycle level) with angle other than 90°.	Operating Manual	Section 5.5.2
Automatic grooving operation on the face of the part and finishing pass.	Operating Manual	Section 5.7
Automatic profile rounding operation by pattern repeat of profile or roughing.	Operating Manual	Section 5.5.3
Approach point in profile rounding operation (modification).	Operating Manual	Section 5.5.3
Automatic Profile execution, Cycle Level, by pattern repeat or roughing.	Operating Manual	Section 6.2
Approach point in automatic Profile execution (modification).	Operating Manual	Section 6.2
Automatic tapping operation.	Operating Manual	Section 5.8
M20 at the end of part-program execution.	Installation Manual	Section 3.8.3.1
Graphic simulation	Operating Manual	Section 5.1.3
Execution / Simulation of program P99996 (ISO-coded user program)	Installation Manual Operating Manual	Section 3.11 Section 3.10
Automatic or Single-block execution of P99996	Operating Manual	Section 3.10
Editing of program P99996	Installation Manual Operating Manual Programming Manual	Section 3.12 Section 3.11
ISO-coded user program P99994 to store subroutines	Programming Manual	Chapter 9
Subroutine associated to the execution of a tool (only when executing program P99996)	Installation Manual Programming Manual	Section 4.3.4
ISO codes of the 800T CNC	Programming Manual	

Date: **March 1995** ***Software version:*** **5.2 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Editing of program P99996 in all models.		
When interrupting the execution, the following keys are enabled: spindle, coolant, O1, O2, O3 and TOOL.	Installation Manual Operating Manual Operating Manual Operating Manual	Section 3.11 Section 3.10 Section 5.1.4 Section 7.5
Incremental JOG movements taking current work units (radius or diameter) into account.	Installation Manual	Section 4.3.3
ISO programming. New functions: G47, G48 (single block treatment).	Programming Manual	Section 6.7
ISO programming. New function: G86 (Longitudinal threadcutting canned cycle).	Programming Manual	Section 8.17
Request from the PLCI for real spindle rpm.	PLCI Manual	

Date: **November 1995** ***Software version:*** **5.5 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Tool offset modification while in execution.	Operating Manual	Section 3.4.4
Operation with a single electronic handwheel.	Installation Manual Installation Manual	Section 4.3.2 Section 7.5
Actual "S" speed reading from the PLCI.	PLCI Manual	

INTRODUCTION

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Protections of the unit itself

Central Unit

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

All the digital inputs and outputs are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Monitor

The type of protection fuse depends on the type of monitor. See the identification label of the unit itself.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

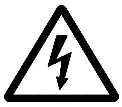
Safety symbols

Symbols which may appear on the manual

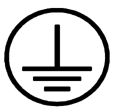
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.



"Electrical Shock" symbol
It indicates that point may be under electrical voltage



"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the CNC, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

    When sending the monitor, especially protect the CRT glass.

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE 800T CNC

800T CNC OEM Manual	Is directed to the machine builder or person in charge of installing and starting up the CNC. It has the Installation manual inside. Sometimes, it may contain an additional manual describing New Software Features recently implemented.
800T CNC USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC. Sometimes, it may contain an additional manual describing New Software Features recently implemented.
DNC 25/30 Software Manual	Is directed to people using the optional DNC communications software.
DNC 25/30 Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the 800 without using the DNC25/30 software..
PLCI Manual	To be used when the CNC has an integrated PLC. Is directed to the machine builder or person in charge of installing and starting up the PLCI.
DNC-PLC Manual	Is directed to people using the optional communications software: DNC-PLC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The operation manual consists of the following sections:

Index

Comparative Table for Fagor 800T CNC models

New Features and modifications

Introduction
- Safety Conditions
- Material returning conditions
- Additional remarks
- Fagor documents for the 800T CNC
- Manual Contents

Chapter 1 Concepts
- Indicates the layout of the keyboard, operator panel and information on the monitor.
- Describes the display units and how to modify them
- Indicates the reference systems to be set.
- How to reference the machine and how to preset coordinates.
- How to operate with absolute and incremental coordinates

Chapter 2 Basic operations
- Simple description of the operating modes available at the CNC
- Screen description
- Description of the display units and how to change them
- Indicates how to select the axis feedrate.
- How to select the working tool and the live tool
- How to jog the machine with jog keys or with the electronic handwheel.
- How to select the starting point (BEGIN) and the end point (END).
- How to position the tool at the BEGIN or END point.
- How to activate and deactivate external devices.

Chapter 3 Auxiliary functions:
- Indicates how to select working units (mm/inches).
- How to select radius or diameter work modes
- How to select feedrate units (mm/min /mm/rev)
- How to set the tool table.
- How to calibrate and inspect tools.
- How to define the finishing pass for the automatic operations.
- How to define the safety distance for automatic operations
- How to select and define the automatic operations:
  - Simple drilling, multiple drilling and slot milling.
- How to operate with peripherals.
- How to lock and unlock the program memory.
- How to edit, execute and simulate program 99996.

Chapter 4 Spindle
- Indicates how to select the spindle operating mode
- How to operate the spindle in rpm or CSS mode
- How to change gears manually and automatically
- How to select the spindle turning direction (clockwise or counter-clockwise)
- How to work with spindle orientation (angular positioning)

Chapter 5 Automatic operations.
- Indicates how to select and program each automatic operation.
- Operating modes: "Semi-automatic and automatic"
- How to select the machining conditions of the automatic operations.
- How to execute and simulate an automatic operation.

Chapter 6 Profiles
- Semi-automatic mode:
  - Shows how to gather points and move from point to point
- Automatic mode: shows how to define the profile and how to execute it.

Chapter 7 Working with parts.

Indicates how to access the part-program directory.
How to select a part-program, edit it, simulate it and execute it.
How to execute an operation previously stored in a part-program.
How to modify a part-program.
How to delete a part-program.
How to operate with peripherals.
How to lock and unlock the part-program memory.

Chapter 8 Programming examples

Error codes.

1. CONCEPTS

After powering the 800T CNC, the monitor shows the CNC model name and the message:

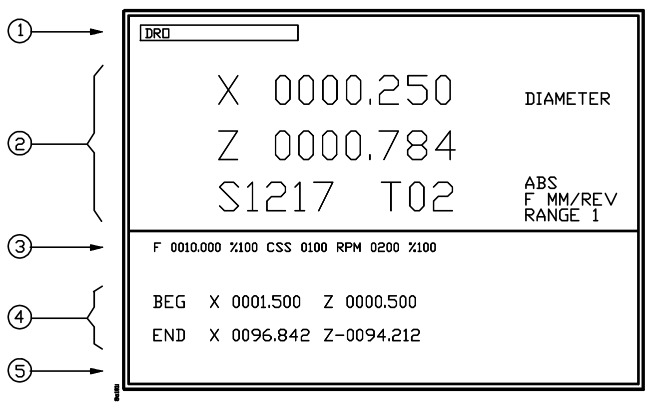
***** GENERAL TEST ***   Passed**

Press any key to access the CNC standard work mode.

If the GENERAL TEST was not successful, the CNC will display the detected errors which must be corrected before operating with the machine.

1.1 CRT DESCRIPTION

The CRT of this model is divided into the following areas or display windows:



1.- This window indicates the selected operating mode: DRO, Turning, threading, etc.

It also indicates the CNC status while executing the automatic operations (in execution, interrupted or in position).

2.- Main window.

This displays shows the current tool position (X and Z coordinates), as well as the spindle speed (S) and the tool (T) currently selected.

It also shows the display units being selected and the active spindle speed range.

3.- This window shows the following information:

* The axis feedrate (F) currently selected and the override percentage (%) currently being applied.

* When the RPM mode is selected, the CNC displays the word: "**RPM**" and the value of the spindle speed currently active in revolutions per minute.

* When the Constant Surface Speed mode (CSS) is selected, the CNC shows the word: "**MAX**" and the value of the maximum speed allowed for the spindle in rpm.

Atention:

The **CSS** value is given in m/min or feet/min. The **MAX** value is also given in rpm as well as the actual spindle speed "S" displayed at the main window.

* The percentage (%) of the programmed spindle speed being applied.

* The direction of the spindle rotation.

* The tool to be used to perform the selected automatic operation.

This data is defined while editing the automatic operations to be stored. This way, every time a previously stored part is executed, the CNC will carry out each one of the automatic operations with the tool and spindle turning direction set in the editing mode.

4.- This window shows the coordinates of the BEGIN and END points.

Also, when selecting an automatic operation, it will show the corresponding parameters and a drawing representing it.

5.- CNC communications and editing window.

1.2 *KEYBOARD DESCRIPTION*

It consists of the following keys:

Numeric keyboard consisting of the following keys: **0,1,2,3,4,5,6 ,7 ,8 ,9,** [•] **,** [+/−] to enter integer and decimal values with or without sign.

[=] To assign values to the machine parameters during CNC installation.

[↑] [↓] To select the previous or next option when so required by the displayed menu as well as to carry out the machine reference zero (home) search.

[←] [→] To move the zoom window at the compact model. To do this at the modular model, use the [0] [•] keys.

[X] To select the X axis for later data entry or modification regarding this axis. Once this value is keyed in, press **[ENTER]**.

[Z] To select the Z axis for later data entry or modification regarding this axis. Once this value is keyed in, press **[ENTER]**.

[F] To select the axis feedrate for later entry or modification of its value. Once this value is keyed in, press **[ENTER]**.

[S] To select the spindle speed (rpm) for later entry or modification of its value. Before pressing this key, select the type of spindle speed (CSS or rpm).

Once this value is keyed in, it is possible to:

* Press . The CNC will assume this value as theoretical spindle speed.
* Press **[ENTER]**. The CNC will store this value but it does not change the current theoretical spindle speed.

  This option is very useful when editing operations to be stored later.

[TOOL] To select the new tool. Once the new tool has been selected, it is possible to:

* Press . The CNC selects the new tool.
* Press **[ENTER]**. The CNC stores this value but it does not select any tool.

  This option is very useful when editing operations to be stored later.

[ENTER] To validate the commands generated at the editing window.

[RECALL] To recover values previously entered into part-programs or CNC tables for later analysis and modification.

Before pressing this key, use the up and down arrow keys to move the cursor and select the operation to be analyzed.

CLEAR To delete the last character entered in the editing window.

RESET To reset the CNC and assume the default values set by machine parameters. Also, this key must be pressed after modifying the machine parameter values in order for the CNC to assume them.

During the execution of an automatic operation it is necessary to previously stop its execution. The CNC will also request confirmation of the command being necessary to press this key again to acknowledge it. To cancel the reset command, press [CLEAR] instead.

If this key is pressed while an automatic operation (turning, facing, etc.) is selected, the CNC will quit that mode and will return to the DRO display mode.

AUX To access the menu for the auxiliary functions of the CNC.

To turn the coolant on or off. When the coolant is on, the lamp of the key will also be on.

01 02 03 With these keys it is possible to activate or deactivate outputs O1, O2 and O3. Their lamps will turn on when the corresponding outputs are on.

CSS To select the type of spindle control to be used: rpm or Constant Surface speed (CSS).

The CNC will highlight the selected option. Besides, the lamp of this key will stay on when the Constant Surface Speed mode is selected.

inc/abs This key is used to access the incremental mode (INC). When this mode is selected, the lamp of this key will stay on. To return to the standard mode, press this key again and its lamp will turn off.

SINGLE It selects the mode in which the automatic operation will be executed.

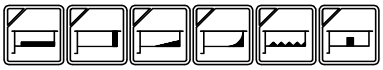
Continuous mode. The key lamp stays off and the selected operation will be carried out from beginning to end without interruptions.

Single mode. the key lamp stays on and the selected operation is executed a single pass at a time. The key must be pressed to run each pass.

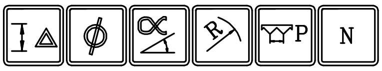
SIMUL It selects the program simulation mode at the compact model. To do this at the modular model, use AUX S

1.2.1 KEYS FOR AUTOMATIC OPERATIONS

It consists of the following keys:



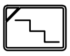
They allow the selection of one of the automatic operations offered by this CNC.



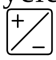
They are used to define the parameters corresponding to the automatic operation that has been selected.

To select the coordinates of the BEGIN point for later modification or to command the machine to move to that point.

To select the coordinates of the END point for later modification or to command the machine to move to that point.



To access the "point-to-point movement" mode.

To select the operating mode in the automatic operations: Semiautomatic, Cycle level 1 and Cycle level 2 **at the compact model**. To do this at the modular model, use  instead.

1.2.2 *SPECIAL KEYSTROKE SEQUENCES*

• = This keystroke sequence blanks the CRT out. Just press any key to recover the normal display,

If an error occurs while the CRT is blank, it will resume the normal display.

[S] With this sequence, it is possible to select the angular spindle orienting position.

[S] The CNC allows this sequence when the Constant Surface Speed (CSS) mode has been selected and it is used to set the maximum spindle speed (**MAX**) in that mode.

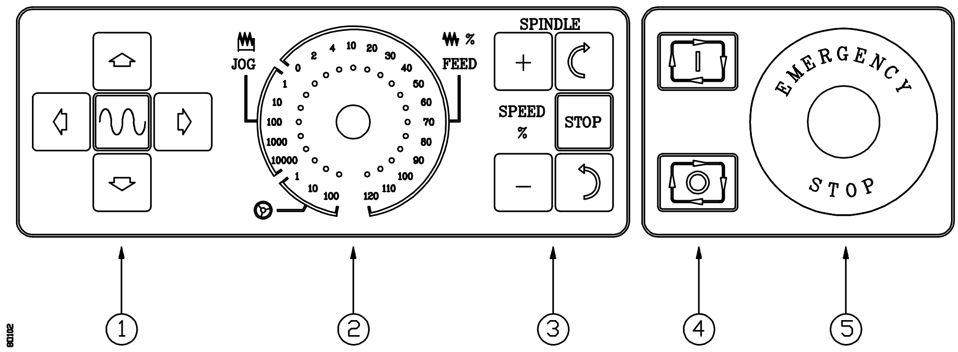
Once the value has been keyed in, press **[ENTER]** so it is assumed by the CNC.

TOOL **[S]** When the machine has a live tool, this sequence permits selecting its turning speed (**TRPM**).

= and = to quit the part-program directory and switch to editing the selected automatic operation.

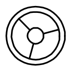
1.2.3 OPERATOR PANEL

Depending on their function, this panel is divided into the following areas:



1.- Keyboard to jog the axes.

2.- Selector switch consisting of the following elements:

 To select the multiplying factor applied by the CNC to the pulses from the electronic handwheel (1, 10, 100).

JOG To select the distance the axis will move (1, 10, 100, 1000 microns or ten-thousandths of an inch) when pressing the corresponding key.

FEED To change the programmed feedrate between 0% and 120%.

3.- Keyboard to control the spindle. It can be started in the desired direction, stopped or change its turning speed between 50% and 120% of the programmed speed with an incremental step of 5%.

4.- Keyboard for [START] and [STOP] of the programmed movements, automatic operations and part programs.

5.- Location for the Emergency-Stop button (E-stop).

1.3 DISPLAY UNITS (mm/inches)

The main window of the 800T CNC shows at all times the X and Z coordinates of the tool position.

With this CNC it is possible to display the position of the axes in either mm or inches.

To change the type of display units, press [AUX] and select the "MM/INCH" option. Every time this option is selected, the CNC will toggle the display units from mm to inches and vice versa showing the axis position in the selected units.

To exit the operating mode for auxiliary functions and return to the standard display mode, press [AUX], [END] or [CLEAR].

1.3.1 X AXIS DISPLAY UNITS (radius/diameter)

The 800T CNC can show the X axis position in radius or diameter.

The main window and next to the X axis position display shows the "DIAMETER" or "RADIUS" message indicating the selected display units.

To change the type of units, press [AUX] and select the "radius/diameter" option. Every time this option is selected, the CNC will toggle the display units from one to the other and it will show the X position in the new selected units.

To exit the operating mode for auxiliary functions and return to the standard display mode, press [AUX], [END] or [CLEAR].

1.4 REFERENCE SYSTEMS

The machine where this CNC is installed needs to have the **Machine Reference Zero** (home) for each axis defined. This point is set by the manufacturer as the origin of the coordinate system of the machine.

It is also possible to establish another origin point for programming the part dimensions, Part Zero. This new origin point is chosen freely by the operator and the position values shown by the CNC are referred to this point.

Bear in mind that to select the Part Zero, the CNC must be working in absolute coordinates, inc/abs lamp off. If it is not off, press its key.

The Part Zero stays selected even when the CNC is off and it will be lost when a new Part Zero is selected or when the Machine Reference Zero (home) is searched.

1.4.1 HOME SEARCH

The Machine Reference Zero search is done one axis at a time by following these steps:

* Press the key corresponding to the axis to be homed [X] [Z] and then the up arrow key.

* The editing window will request confirmation of the command. Press the key for the CNC to perform the home search on that axis.

  If the home search is not desired, press any other key. To cancel the home search once in progress, press [CLEAR].

When doing a home search, the CNC initializes the displays of the axis and it cancels the Part Zero that was selected.

1.4.2 ZERO PRESET

It is possible to preset the desired Part Zero in order to use the coordinates relative to the blue-prints of the workpiece without having to modify the various points of the part.

To preset the Part Zero, follow these steps:

* The CNC must be working in absolute coordinates, inc abs lamp off. If it is not off, press its key.

* Press the key corresponding to the axis to be preset [X] [Z] and then [ENTER].

  The CNC will request confirmation of the command, press [ENTER] again.

* Repeat this operation for the other axis.

Every time this operation is done, the CNC assumes that point as the new coordinate origin.

1.4.3 COORDINATE PRESET

With this feature it is possible to preset the desired coordinate (position) values in order to use the coordinates relative to the blue-prints of the workpiece without having to modify the various points of the part.

Also, this type of preset may be used when it is more convenient to work from coordinate value towards zero instead of doing it from one coordinate value to the next, as usual.

To preset a coordinate value, follow these steps:

* Press the key corresponding to the axis to be preset: [X] [Z].

* Key in the desired position value for that point.

* Press [ENTER] so the CNC assumes that preset value as the new position value (coordinate) for that axis.

  The CNC will request confirmation of the command, press [ENTER] again.

* Repeat this operation for the other axis.

Every time a preset is done, the CNC assumes a new Part Zero which will be located at a distance from the preset point equal to the preset value.

1.5 *OPERATION IN INCREMENTAL MODE*

With this CNC it is possible to set a Floating Zero or Incremental Zero, besides the Part Zero described earlier, in order to use coordinates relative to any point of the part

In order to work in the incremental mode, it is necessary that the coordinate values displayed by the CNC be incremental, inc abs lamp on. If it is not on, press this key.

On the other hand, the CNC shows at all times, to the right of the main window, the type of coordinates being selected (ABS/INC).

Atention:

Every time the incremental mode is selected, the CNC takes as Floating Zero the Part Zero point currently active. Therefore, it will keep showing the same X, Z position values.

To set another Floating Zero, another coordinate or Part Zero must be preset. From that moment on, the coordinates shown by the CNC will be referred to the new Floating Zero established.

The CNC keeps the Part Zero preset when working in absolute mode and it will display the X, Z position referred to it when switching back from incremental to absolute mode.

2. *BASIC OPERATIONS*

2.1 AXIS FEEDRATE SETTING

With this CNC it is possible to set the feedrate (F) for the axes as often as desired in order to move them at the proper speed each time.

Also, the Operator Panel has a multi-position switch which allows modifying the feedrate during those moves by applying the selected percentage (%) override to the feedrate selected in each case. This percentage amount is indicated by the Feedrate Override Switch (FEED) and it has a range from 0% thru 120% of the set feedrate.

Follow these steps to set the axis feedrate (F):

* Press [F]

* Enter the value via keyboard and press [ENTER].

  When working in mm/min., this value must be between 0 and 9999 mm/min.

  When working in inches/min., this value must be between 0 and 393 inches/min

  When working in mm/rev., this value must be between 0 and 500.000 mm/rev.

  When working in inches/rev., this value must be between 0 and 19.685 inches/rev.

The CNC assumes this value and it displays it on the screen. It also displays the percentage of the feedrate override currently selected by the FEED switch. For example: F1200 100%.

<u>Atention:</u>

When the CNC displays a value of "F0000", it applies the maximum feedrate set by machine parameter for each axis.

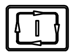
2.2 *WORK TOOL SELECTION*

Regardless of whether the machine has an automatic tool changer or not, the CNC must "know" at all times which tool is being used for the machining operation.

<u>The machine has an automatic tool changer.</u>

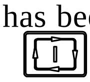
To select a new tool (T), follow these steps:

* Press [**TOOL**]

* Key in the number to the tool to be selected and press 

The CNC will manage the tool change and it will assume all the offset values corresponding to the new selected tool. These offset values (tool lengths and radius) will be taken into account by the CNC when performing any operation from this moment on.

<u>The machine does not have an automatic tool changer.</u>

The CNC must know at all times which tool is being used for machining. To achieve this, every time a new tool is selected, after it has been changed, press [TOOL] followed by the selected tool and, then, press 

The CNC will assume all the offset values corresponding to the new selected tool. These offset values (tool lengths and radius) will be taken into account by the CNC when performing any operation from this moment on.

If a new tool is to be selected when executing a cycle or preprogrammed part, the CNC will display a message indicating the number of the new tool to be selected.

It also interrupts the execution of the program until the tool has been changed and the operator presses the [ENTER] key.

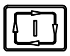
2.2.1 LIVE TOOL

If the machine has a live tool and it is not selected, the CNC shows the following type of information:

F 0100.000 100% RPM 1500 100% T4

Where **T4** indicates the tool currently selected.

To select the live tool, press the following keystroke sequence:

* TOOL **[S]**; After which the CNC shows: "**T RPM =**".
* Enter the rpm value for the live tool.
* Press 

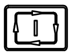
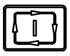
The CNC activates the live tool and displays the following type of information:

F 0100.000 100% RPM 1500 100% TRPM 800 T4

Where **TRPM** indicates the selected live tool speed.

To select a new live tool speed, press: TOOL **[S]** and after keying in the new value, it is possible to press:

* **[ENTER**]. The CNC updates the display information with this new value but the spindle keeps turning (if it was) at the previously selected speed.

  In order to make the live tool turn at the new speed,  must be pressed.
*  . The CNC updates the display information and the live tool starts turning at the new selected speed.

To stop the live tool, a "0" speed must be selected by means of the following keystroke sequence: TOOL **[S] [0] [ENTER]**.

If the manufacturer has set the corresponding machine parameter allowing the live tool speed to be overridden, it will be possible to do so by means of the spindle speed override keys + − located on the operator panel.

2.3 AXES JOG

2.3.1 CONTINUOUS JOG

With this option it is possible to jog the axes of the machine one at a time.

Once the axis feedrate has been preset and the feedrate override (0% thru 120%) has been selected at the operator panel switch (FEED), press the JOG key corresponding to the desired axis and its jogging direction.

Depending on the value assigned to machine parameter P12, this movement will be carried out as follows:

* If P12=Y, the axes will move while the selected JOG key is kept pressed.

* If P12=N, the axes will start moving when the JOG key is pressed and it will stop when either the key or the JOG key for another axis is pressed. In this latter case, the other axis will start moving.

If while an axis is being jogged, the key is pressed, the CNC will behave as follows:

* If machine parameter "P617(6)=0", it will move at % of the JOG feedrate indicated on the table below.

% selected	0	2	4	10	20	30	40	50	60	70	80	90	100	110	120
% applied	0	102	104	110	120	130	140	150	160	170	180	190	200	200	200

  This % override will be maintained as long as this key is maintained pressed and it will recover the previous feedrate percentage (0% thru 120%) when releasing this key.

* If machine parameter "P617(6)=1", it will move at 100% of the value set for rapid traverse (P111, P311).

Atention:

The CNC takes into account parameter P617(6) from version 3.3 on. On previous versions, the CNC acts as if "P617(6)=0"

2.3.2 INCREMENTAL JOG

With this option it is possible to jog the desired axis and in the desired direction a distance selected by the JOG positions of the FEED switch on the operator panel. The feedrate used in these incremental moves is set by the machine manufacturer.

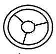
The positions available at this switch are 1, 10, 100, 1000 and 10000 which indicate the number of units to be moved. These units are those used in the display format. Example:

Switch position	Incremental move
1	0.001 mm or 0.0001 inch
10	0.010 mm or 0.0010 inch
100	0.100 mm or 0.0100 inch
1000	1.000 mm or 0.1000 inch
10000	10.000 mm or 1.0000 inch

After selecting the desired switch position, every time the jog key is pressed, the axis will move the indicated amount and in the chosen direction.

2.3.3 AXES JOG VIA ELECTRONIC HANDWHEEL

With this option it is possible to jog the axes of the machine by means of an electronic handwheel.

To do this, first select any of the  positions at the FEED switch of the Operator Panel which correspond to the electronic handwheel.

The available positions are 1, 10 and 100 which indicate the multiplying factor to be applied to the pulses coming from the electronic handwheel.

This way, and after applying this factor, the desired distance units are obtained for axis jog. These units correspond to the display units being used.

Example: Handwheel resolution: 250 pulses per turn

Switch position	Distance jogged per turn
1	0.250 mm or 0.0250 inch
10	2.500 mm or 0.2500 inch
100	25.000 mm or 2.5000 inches

Depending on the setting of machine parameter "P617(5)", the CNC behaves as follows at the rest of the positions of the Feedrate Override Switch of the operator panel.

P617(5)=0 The axes may be jogged with the handwheel as if the MFO switch were at the "x1" position.
P617(5)=1 The axes may **not** be jogged with the handwheel.

When attempting to move an axis faster than the feedrate value set by machine parameters P110 and P310, the CNC will limit it to those values ignoring the rest of the handwheel pulses thus preventing any following errors from being issued.

The machine has one electronic handwheel

Once the desired switch position has been selected, press one of the JOG keys corresponding to the axis to be jogged. The selected axis will be highlighted on the screen.

When using a FAGOR handwheel with axis selector button, the axis can be selected as follows:

* Press the button located on the rear of the handwheel. The CNC will select the first axis and it will show it highlighted on the screen.

* If this button is pressed again, the CNC will select the other axis and when pressed again it will select the previous one again and so forth.

* If this button is maintained pressed for more than 2 seconds, the CNC will de-select the axis.

The machine will move the selected axis as the handwheel is being turned in one direction or the other depending on the turning direction of the handwheel.

The machine has two electronic handwheels

The machine will move each one of the axes as the corresponding handwheel is turned according to the selected switch position and turning direction.

2.4 BEGINNING POINT (BEGIN) AND END POINT (END)

With this CNC it is possible to set a beginning point (BEGIN) and an end point (END to facilitate the machining tasks.

These points may be re-defined as often as desired and may be used to set the ends of the part, the boundaries (limits) of a particular machining area, etc.

The CNC also offers functions which allow moving the tool, automatically and at the programmed speed, to the Begin and End points.

When some of the automatic operations are selected, these movements will be carried out as follows: For example, if the turning operation is selected, the movements to the Begin and End points will be performed only along the Z axis regardless of the X value set for those points.

This way, it is possible to perform semi-automatic machining operations. For example, if the machining operations are limited by means of the BEGIN and END points, the operator can manually control the tool penetration at each pass and command the CNC to control the machining and withdrawal moves.

2.4.1 BEGIN AND END POINT SETTING

Positioning the machine

Move the axes to the desired point by means of the mechanical or electronic handwheel or the JOG keys of the Operator Panel.

Follow these keystroke sequences to set the BEGIN and END points:

* "[BEGIN] [ENTER]"
  "[END] [ENTER]".

  The CNC assumes as the new X and Z coordinates of the BEGIN or END points the values currently being displayed.

* "[BEGIN] [X] [ENTER]"
  "[BEGIN] [Z] [ENTER]"
  "[END] [X] [ENTER]"
  "[END] [Z] [ENTER]".

  Only the coordinate of the selected axis is modified. The value of the other axis does not change.

Without moving the axes. From the keyboard.

When the Begin or End point is set only by keyboard entry without moving the axes, follow these steps:

1.- Press [BEGIN] or [END], depending on the point to be set.

2.- Press [X] to set this value.

3.- Enter the corresponding coordinate value for this axis.

4.- Press [Z] to set this value.

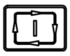
5.- Enter the corresponding coordinate value for this axis.

6.- Press [ENTER].

The CNC will modify the coordinates corresponding to the selected axes. If only one has been set, the one for the other axis will not be changed.

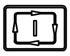
2.4.2 *POSITIONING AT BEGIN OR END POINTS*

To move the tool up to the Begin or End point, do the following:

* Press [BEGIN] to move to this point or [END] to move to this other point.
* Press 

The CNC will move the tool to the selected point automatically and at the programmed feedrate.

When only one axis is to be moved, follow these steps:

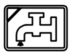
* Press [BEGIN] to move to this point or [END] to move to this other point.
* Press the key corresponding to the axis to be moved [X] or [Z].
* Press 

The CNC will move the tool to the selected point, along that axis, automatically and at the programmed feedrate. The other axis will not move.

2.5 *ACTIVATING/DEACTIVATING EXTERNAL DEVICES*

With this CNC it is possible to activate and deactivate 4 external devices including the coolant. The other devices depend on the type of machine.

These devices may be activated or deactivated at any time unless indicated otherwise by the machine manufacturer.

To do this, the following keys are available: [01] [02] [03] 

Each one of these keys have a lamp to indicate that the device is on (lamp on) or the device is off (lamp off).

Every time one of these keys is pressed, the status of the corresponding device will toggle (activated/deactivated) as well as the corresponding key lamp.

3. *AUXILIARY FUNCTIONS*

Once the standard work mode has been selected, press [AUX] to access this option.

The CNC will then show a menu of options. To select the desired option, press the key for its corresponding number.

However, it is possible to access the "Cycle Finishing Pass and safety distance" option directly by pressing [AUX] while in the "Automatic Operations" mode.

When selecting the "AUXILIARY MODES" option, the CNC requests the access code (password) to be able to use the various tables and modes for the OEM.

Press [END] to quit any of these options and return to the standard display mode.

3.1 MILLIMETERS <—> INCHES

When selecting this option, the CNC changes the display units from mm to inches and vice versa showing the new X and Z coordinates of the axes in the new selected units.

The axis feedrate (F) will also be shown in the new selected units.

The units for the axis feedrate will also be changed and they will appear to the right of the main window.

For example, if the axis display was in mm and the feedrate was given in mm/rev., the new units for the axis position will be inches and for the feedrate in inches/rev.

It must be borne in mind that the values stored for BEGIN and END, the data for special operations and the data corresponding to the "point-to-point movement" mode have no units. **Therefore, the values will remain the same when toggling from mm to inches or vice versa.**

3.2 RADIUS <—> DIAMETER

When selecting this option, the CNC changes the display units for the X axis from radius to diameter or vice-versa and it shows the X axis coordinates in the new selected units.

The text corresponding to these units will also change. These units appear to the right of the X position value (coordinate).

It must be borne in mind that the values stored for BEGIN and END, the data for special operations and the data corresponding to the "point-to-point movement" mode have no units. **Therefore, the values will remain the same when toggling from radius to diameter or vice versa.**

3.3 F MM(INCHES)/MIN <—> F MM(INCHES)/REV

When selecting this option, the CNC changes the units corresponding to the axis feedrate from mm/min to mm/rev (or vice versa) when the display is in mm or from inches/min to inches/rev (or vice versa) when the display is in inches.

These units appear at the right-hand side of the main window.

The value assigned to the axis feedrate "F" does not change.

3.4 TOOL

When selecting this option, the CNC allows access to the tool table and it is possible to calibrate tools.

3.4.1 TOOL TABLE

When selecting this option, the CNC shows the values assigned to each offset; that is, the dimensions of each tool being used to machine the parts.

Once the tool offset table is selected, the operator may move the cursor one line at a time by using the up and down arrow keys.

Each tool offset has several fields which define the dimensions of the tool. These fields are the following:

* Tool length along the X axis.

  It is given in radius and in the units currently selected. Its value range is:

  X ±8388.607 mm or X ±330.2599 inches.

* Tool length along the Z axis.

  It is given in the units currently selected. Its value range is:

  Z ±8388.607 mm or Z ±330.2599 inches.

* Tool radius.

  It is given in the units currently selected. Its maximum value is:

  R 1000.000 mm or R 39.3700 inches.

  The CNC will take this "R" value and the type of tool (location code or shape code "F") when machining the programmed profile. In other words, the CNC will apply tool compensation in all finishing passes.

* Tool type.

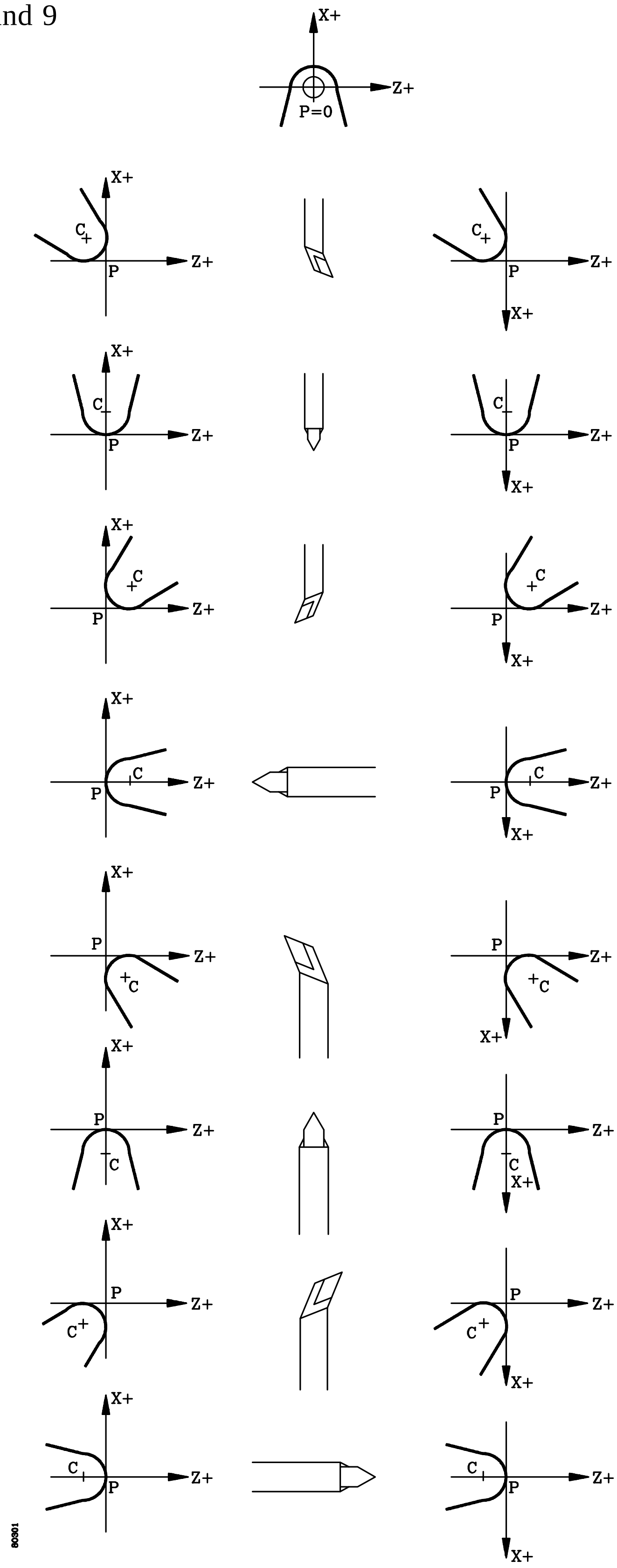
  To indicate the type of tool being used, the CNC offers 10 different location codes (F0 thru F9).

  This factor depends on the shape of the tool being used and on the sides of the cutter used for machining.

  The following drawing shows the tool types commonly used on a lathe indicating in all of them the center (C) of the cutter and its theoretical tip (P).

Codes 0 and 9

Left	Right
Code 1	Code 7
Code 2	Code 6
Code 3	Code 5
Code 4	Code 4
Code 5	Code 3
Code 6	Code 2
Code 7	Code 1
Code 8	Code 8



* Tool length wear along the X axis.

It is given in diameter and in the units currently selected. Its value range is:

I ±32.766 mm or I ±1.2900 inches.

The CNC will add this value to the nominal tool length along the X axis to calculate the real (total) tool length (X+I).

* Tool length wear along the Z axis.

It is given in the units currently selected. Its value range is:

K ±32.766 mm or K ±1.2900 inches.

The CNC will add this value to the nominal tool length along the Z axis to calculate the real (total) tool length (Z+K).

3.4.1.1 MODIFICATION OF TOOL DIMENSIONS

To initialize a table by setting all the fields of each tool to "0", key in the following sequence: [R] [P] [N] [ENTER].

This CNC offers the option for "TOOL CALIBRATION" which is described next. Once they have been calibrated, the CNC assigns to each tool offset the X and Z dimensions of the corresponding tool.

To edit or modify the contents of a tool offset ("R" and "F" values) or to modify its dimensions ("X", "Z", "I", "K" values), select the corresponding offset at the CNC by keying the desired tool number and then pressing [RECALL].

The editing area will show the current values assigned to that tool offset.

To change these values, move the cursor by means of the up and down arrow keys until it is placed over the current value. The new values must be keyed in over the current ones.

Once the new values have been keyed in, press [ENTER] so that they are stored in memory.

To quit this mode, move the cursor to the right until it is out of the editing area and then press [END].

3.4.2 *TOOL CALIBRATION*

With this option it is possible to calibrate and load the tool dimensions onto the tool offset table of the CNC.

The CNC shows a graphic at the lower right-hand side of the monitor used as a user guide during tool calibration highlighting the data being requested at the time.

To calibrate a tool, a work-piece of known dimensions is used and the procedure is as follows:

1.- The CNC requests the work-piece dimension along the X axis.

Key in this value and press [ENTER]. It must be given in the machine work units (radius or diameter).

2.- The CNC requests the work-piece dimension along the Z axis.

Key in this value and press [ENTER].

3.- The CNC requests the number of the tool being calibrated.

Press [TOOL], then, key in the tool number and, then, press for the CNC to select it.

4.- Move the axes by means of either the mechanical handwheels, the electronic handwheel or the JOG keys on the Operator Panel until the tool touches the work-piece along the X axis.

Then, press the keystroke sequence: [X], [ENTER].

The CNC will show the work-piece coordinate along the X axis and the tool will be calibrated along this axis.

5.- Move the axes by means of either the mechanical handwheels, the electronic handwheel or the JOG keys on the Operator Panel until the tool touches the work-piece along the Z axis.

Then, press the keystroke sequence: [Z], [ENTER].

The CNC will show the work-piece coordinate along the Z axis and the tool will be calibrated along this axis.

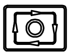
The CNC will request a new tool to be calibrated. Repeat steps 3, 4 and 5 as needed

Press [END] To quit this mode and return to the standard display mode.

3.4.3 TOOL INSPECTION

With this option it is possible to interrupt the execution of program P99996 and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press  to interrupt the program.

b) Press [**TOOL**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the program was interrupted.

the screen will display the following message:

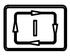
RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the program was interrupted.

e) Jog the axes to the program interruption position by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

f) Press  to resume the execution of program P99996.

3.4.4 *TOOL OFFSET MODIFICATION*

With this option, it is possible to modify the tool wear compensations (I,K) to correct the tool dimensions.

This option can only be accessed during the execution of a part or the user program "P99996" both while in execution and while interrupted. This option is typically used after Tool Inspection.

To select this option, press

The CNC displays the letter "T".

Key in the tool number whose offset is to be modified and press [**RECALL**].

The CNC displays the current tool offset values highlighting the "I" value, Example:

T01	X59.800	Z144.195	F3	R0.4
	I 0.200	K0.340		
I				

Key in the new value for "I" and press

The CNC now highlights the "K" value. For example:

T01	X59.800	Z144.195	F3	R0.4
	I 0.200	K0.340		
I 0.2	K			

Key in the new "K" value and press [**ENTER**].

The CNC <u>adds</u> the new values to the previous ones and displays their resulting values. For example, I0.2 K0.1:

T01	X59.800	Z144.195	F3	R0.4
	I 0.400	K0.440		

To change the offset of another tool, press [**TOOL**] and repeat the steps described above.

To quit this option and return to the execution mode, press [**END**].

3.5 CYCLE FINISHING PASS AND SAFETY DISTANCE

With this option it is possible to define the parameters: "finishing pass, finishing feedrate, finishing tool and safety distances along X and Z" that will be used in the automatic operations.

When storing a machining operation as part of a part-program, the CNC stores these parameters with the values of the machining operation.

It is possible to access these parameters directly by pressing [AUX] while in the "Automatic Operations" mode.

The parameters are:

% $\triangle$ Finishing pass = % of the roughing pass

It indicates the percentage (%) of the programmed roughing pass used as finishing pass.

It is given by an integer. If a value of 0 or 100 is assigned, all the machining passes (roughing and finishing) will be identical.

% F Finishing feedrate = % of the roughing feedrate

It indicates the percentage (%) of the programmed roughing feedrate used as finishing feedrate.

It is given by an integer. If a value of 0 is assigned, the feedrate of the finishing pass will be the same as the one used for the roughing passes.

T Tool to be used for the finishing pass

It is possible to perform the roughing operation with one tool (the one selected in the operation itself) and use the one indicated by this parameter to do the finishing operation (integer between 0 and 32).

If a value of 0 is assigned, the finishing operation will be carried out with the same tool used for the roughing operations

The finishing tool is selected once the roughing operation is completed. If the machine has an automatic tool changer, the tool will be selected automatically; but, when the tool change is to be carried out by the operator, the CNC will indicate the tool to be selected.

When the tool change is done manually, the CNC will request this change being necessary to press [I] once the tool change is completed in order to resume part-program execution.

Safety distance along the X axis for automatic operations.

It indicates the distance, along the X axis and with respect to the "BEGIN" point, where the tool will be positioned in the approach move.

Safety distance along the Z axis for automatic operations.

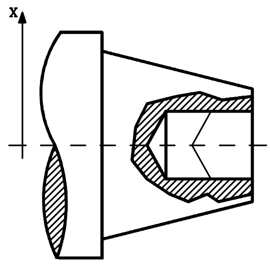
It indicates the distance, along the Z axis and with respect to the "BEGIN" point, where the tool will be positioned in the approach move.

Every time one of these options is selected, the CNC will highlight it. Also, at the bottom of the screen, the CNC will request the new value to be assigned to that parameter.

After defining the new value, press [ENTER] in order for the CNC to assume it.

3.6 *OTHER AUTOMATIC OPERATIONS*

When pressing [AUX] and selecting the option **[6]** corresponding to "OTHER CYCLES", or in DRO mode, [+/–] or [LEVEL +] (at the compact model) is pressed, the CNC will show the following machining cycles:

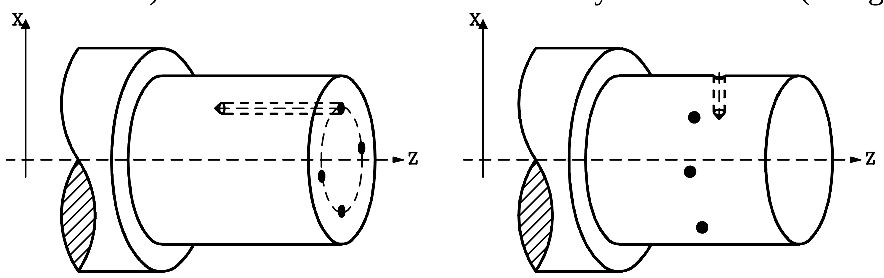


SIMPLE DRILLING. TAPPING

Consisting in drilling the face of the part only along its turning axis (center line).

MULTIPLE DRILLING.

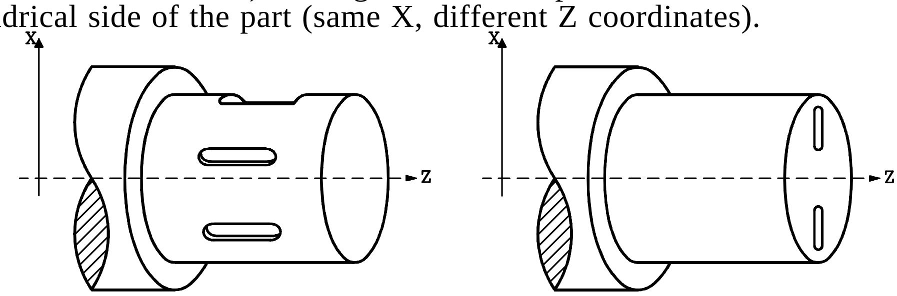
With this cycle it is possible to drill concentric holes on the face of the part (along the Z axis) as well as radial holes on its cylindrical side (along the X axis).



This feature requires spindle orientation and live tool. If the CNC lacks both features, it will not display this canned cycle.

SLOT MILLING.

With this cycle it is possible to mill radial slots on the face of a part (same Z, different X coordinates) or longitudinal slots parallel to the center line on the cylindrical side of the part (same X, different Z coordinates).



This feature requires spindle orientation and live tool. If the CNC lacks both features, it will not display this canned cycle.

A full description of these cycles can be found in the chapter on "Automatic Operations" of this manual.

To quit the editing or execution of these cycles press any other operation key or:

* Press [+/–] or [LEVEL +] to return to the "Other automatic operations" menu.
* and then press [+/–] or [LEVEL +] again to return to the DRO mode.

3.7 AUXILIARY MODES

When selecting this option, the CNC shows the following menu:

1 - SPECIAL MODES
2 - PERIPHERALS
3 - LOCK / UNLOCK
4 - PROGRAM 99996 EXECUTION
5 - EDITING PROGRAM 99996

When selecting the "Special Modes" option, the CNC requests the password or manufacturer's access code indicated in the installation manual.

After accessing one of these modes and operate with it, press **[END]** to quit. At this point, the CNC will show this menu again. Press **[END]** once more to return to the standard display mode.

3.8 PERIPHERALS

With this CNC it is possible to communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer in order to transfer programs from and to one another. This communication may be managed either from the CNC when in the "**Peripheral mode**" or from the computer by means of FAGOR's DNC protocol in which case the CNC may be in any of its operating mode.

3.8.1 PERIPHERAL MODE

In this mode, the CNC may communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer having a standard off-the-shelf communications program.

To access this mode, press **[AUX]** and, after selecting the "**Auxiliary modes**", press the key corresponding to the "**Peripherals**" option.

Once this option is selected, the upper left-hand side of the CNC screen will show the following menu:

0 - **RECEIVE FROM** (Fagor) **FLOPPY DISK UNIT**
1 - **SEND TO** (Fagor) **FLOPPY DISK UNIT**
2 - **RECEIVE FROM GENERAL DEVICE**
3 - **SEND TO GENERAL DEVICE**
4 - (Fagor) **FLOPPY DISK UNIT DIRECTORY**
5 - (Fagor) **DELETE FLOPPY DISK UNIT PROGRAM**
6 - **DNC ON/OFF**

In order to use any of these options, the DNC mode must be **inactive.** If it is active (the upper right-hand side of the screen shows: **DNC**), press **[6]** (DNC ON/OFF) to deactivate it (the **DNC** letters disappear).

With options 0, 1, 2 and 3 it is possible to transfer machine parameters, the decoded M function table and the leadscrew error compensation table to a peripheral device.

The lower right-hand side of the CNC screen will show a directory of up to 7 part-programs of the 10 that may be stored. To see the rest of them, use ⇧ ⇩

To do this, key in the desired number when the CNC requests the number of the program to be transferred and press [**ENTER**].

P00000 to P99990	Corresponding to part-programs
P99994 and P99996	Special user program in ISO code
P99997	**Is for internal use and CANNOT be transmitted back and forth**
P99998	Used to associate texts to PLC messages
P99999	Machine parameters and tables

Atention:

The part-programs cannot be edited at the peripheral device or computer.

The CRT will show the message: "**RECEIVING**" or "**SENDING**" during the program transfer and the message: "**PROGRAM NUM. P23256** (for example) **RECEIVED**" or "**SENT**" when the transmission is completed.

When the transmission is not correct, it will display the message: "Transmission error" and when the data received by the CNC is not recognized (different format) by the CNC, it will issue the message: "Incorrect data received".

The CNC memory must be unlocked in order to perform any data transmission; if not so, the CNC will return to the menu of the peripheral mode.

When transmitting from a peripheral device other than a FAGOR Floppy Disk Unit, the following aspects must be considered:

* The program must begin with a "NULL" character (ASCII 00) followed by "%" "program number" (for example %23256) and a "LINE FEED" character (LF).

* Blank spaces, the carriage-return key and the "+" sign are ignored.

* The program must end with either 20 "NULL" characters (ASCII 00) or with one "ESCAPE" character or with one "EOT" character.

* Press [**CL**] to cancel the transmission. The CNC will issue the message: PROCESS ABORTED".

FLOPPY DISK UNIT DIRECTORY

This option displays the programs stored on the disk inserted in the FAGOR Floppy Disk Unit and the number of characters (size) of each one of them.

It also shows the number of free characters available (free memory space) on the tape.

DELETE FLOPPY DISK UNIT PROGRAM

With this option it is possible to delete a program contained at the FAGOR Floppy Disk Unit.

The CNC requests the number of the program to be deleted. After keying in the desired number, press [**ENTER**].

Once the program has been deleted, the CNC will display the message: "PROGRAM NUM: P____ DELETED".

It also shows the number of free characters on the disk (free memory space).

3.8.2 DNC COMMUNICATIONS

To be able to use this feature, the DNC communication must be active (the upper right-hand side of the screen shows: DNC). To do this the corresponding parameters [P605(5,6,7,8); P606(8)] must be set accordingly and option [**6**] of the "**Peripherals**" mode selected if it was not active.

Once active and by using the **FAGORDNC** application software supplied, upon request, in floppy disks it is possible to perform the following operations from the computer:

. Obtain the CNC's part-program directory.
. Transfer part-programs and tables from and to the CNC.
. Delete part-programs at the CNC.
. Certain remote control of the machine.

Atention:

At the CNC any operating mode may be selected.

3.9 *LOCK/UNLOCK*

With this option it is possible to lock/unlock the part-program memory.

To access this option, press AUX and, after selecting the "**Auxiliary modes**", press the key corresponding to the "**LOCK/UNLOCK**" option.

The codes used to do this are:

N0000 [ENTER] Unlocks the part-program memory.

N1111 [ENTER] Locks the part-program memory.

PF000 [ENTER] Erases all the arithmetic parameters (data for automatic operations) setting them to "0".

3.10 *EXECUTION / SIMULATION OF PROGRAM P99996*

To select this option, press [AUX] and after selecting "Auxiliary Modes", press the key corresponding to "EXECUTION OF PROGRAM P99996".

Program P99996 is a special user program in ISO code. It may be edited (written) at the CNC or at a PC and, then, be sent to the CNC via the Peripherals option.

Once this option has been selected, it is possible to execute or simulate this program.

To simulate program P99996, press [SIMUL] at the compact CNC model and [AUX] at the modular CNC model.

The way to operate in either case is described next.

3.10.1 EXECUTION OF PROGRAM P99996

When selecting the option: "Execution of program P99996", the CNC displays the following information:

```
AUTOMATIC          P99996   N0000
N00  G90
N10  G94
N20  T1.1
N30  F2000

      COMMAND        ACTUAL        TO  GO

    X  0000.000    X  0000.000    X  0000.000
    Z  0000.000    Z  0000.000    Z  0000.000
    S  0000        S  0000        (RPM)

F0000.000  %100   S0000  %100   T00.00
G05  01  95
M41
```

The top line shows the message "AUTOMATIC", the program number (P99996) and the number of the first block of the program or that of the block being in execution.

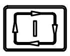
Then, the CRT shows the contents of the first program blocks. If the program is being executed, the first block of the list will be the one being executed at the time.

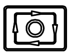
The position values along X and Z indicate the programmed values (COMMAND), the current position (ACTUAL) and the distance remaining (TO GO) for the axes to reach the "command" position.

It also shows the selected spindle speed, programmed value multiplied by the active %S override (COMMAND), and the real spindle speed (ACTUAL).

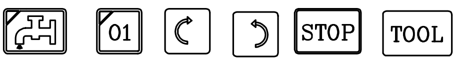
The bottom of the screen shows the machining conditions currently selected. The programmed feedrate F, the % F override, the programmed spindle speed S, the %S override, the programmed Tool as well as the active G and M functions.

To execute program P99996, proceed as follows:

* Select, if so desired, the first block to be executed indicated at the upper right-hand corner (by default: N0000), by keying in N**** [RECALL] and...
* press 

To interrupt the program, press 

Once interrupted, the following keys are enabled:

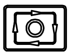


To resume execution, press 

3.10.1.1 TOOL INSPECTION

With this option it is possible to interrupt the execution of program P99996 and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press  to interrupt the program.

b) Press [**TOOL**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the program was interrupted.

The screen will display the following message:

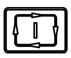
RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the program was interrupted.

e) Jog the axes to the program interruption position by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

f) Press  to resume the execution of program P99996.

3.10.1.2 EXECUTION MODES

With this CNC it is possible to execute program P99996 from beginning to end or one block at a time by pressing [SINGLE]

The top line of the screen shows the operating mode currently selected either "Automatic" or "Single Block".

To switch from one mode to the other, press [SINGLE] again

Once the execution mode has been selected, press

3.10.1.3 CNC RESET

With this option it is possible to reset the CNC setting it to the initial conditions established by the machine parameters. When quitting this operating mode, the CNC displays the DRO mode.

To reset the CNC, simply interrupt the program execution, if running, and press [RESET]

The CNC will request confirmation of this function by blinking the message: **"RESET?"**.

To go ahead with reset, press [RESET] again; but to cancel it, press [CLEAR]

3.10.1.4 DISPLAYING PROGRAM BLOCKS

To display the previous or following blocks to those appearing on the screen, press:

Displays the previous blocks

Displays the following blocks

Atention:

Bear in mind that P99996 always starts executing from the currently selected starting block, regardless of the blocks currently displayed on the screen. By default, this starting block is N000.

To select another starting block, press N (block number) **[RECALL]**. For example: N110 RECALL.

3.10.1.5 DISPLAY MODES

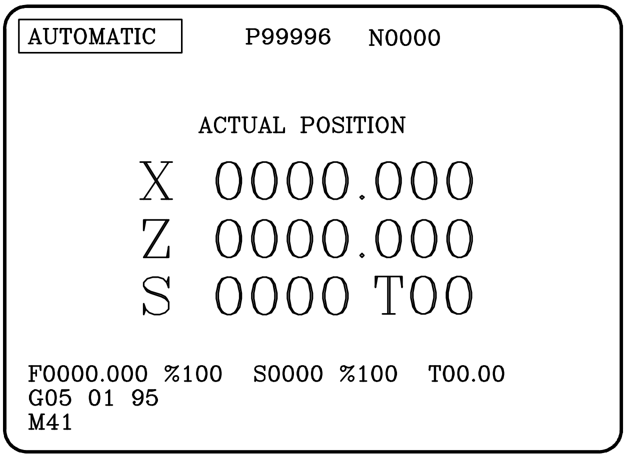
There are 4 display modes which can be selected by means of the following keys:

[0] STANDARD
[1] ACTUAL POSITION
[2] FOLLOWING ERROR
[3] ARITHMETIC PARAMETER

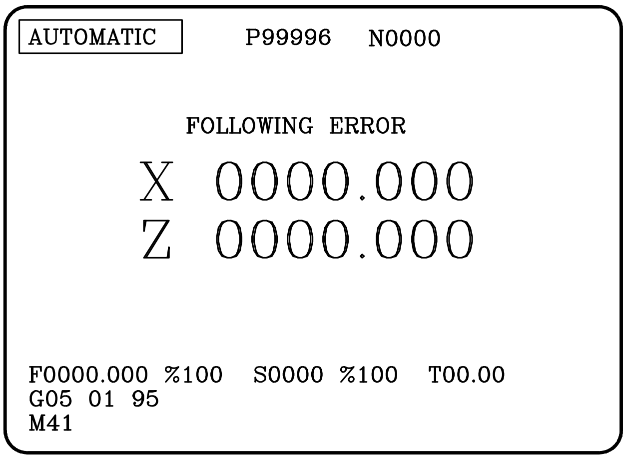
STANDARD display mode

It is the mode described before. When accessing the "Execution of program P99996" option, the CNC selects this display mode.

ACTUAL POSITION display mode



FOLLOWING ERROR display mode



ARITHMETIC PARAMETERS display mode

```
AUTOMATIC          P99996    N0000
P000:  0.0000000     P001:  0.0000000
P002:  0.0000000     P003:  0.0000000
P004:  0.0000000     P005:  0.0000000
P006:  0.0000000     P007:  0.0000000

     COMMAND       ACTUAL        TO  GO

   X  0000.000    X  0000.000    X  0000.000
   Z  0000.000    Z  0000.000    Z  0000.000
   S  0000        S  0000        (RPM)

F0000.000  %100   S0000  %100   T00.00
G05  01  95
M41
```

This mode shows a group of 8 arithmetic parameters. To view the previous and following ones, use these keys:

Displays the previous parameters

Displays the following ones

The value of each parameter may be expressed in one of the following formats:

P46 = -1724.9281	Decimal notation
P47 = -.10842021 E-2	Scientific notation

Where "E-2" means 10^{-2} (1/100). Therefore, the two types of notation for the same parameter below have the same value:

P47= -0.001234	P47= -0.1234 E-2
P48= 1234.5678	P48= 1.2345678 E3

3.10.2 SIMULATION OF PROGRAM P99996

With this CNC it is possible to check program P99996 in dry-run before executing it.

To do this, press SIMUL at the compact model and AUX at the modular model.

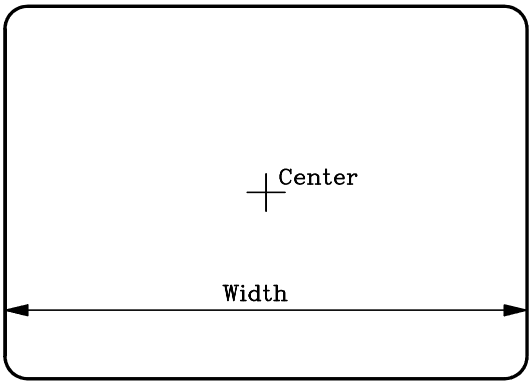
The screen shows a graphic page.

The lower left-hand side of the screen shows the axes of the plane.

To define the display area proceed as follows:

* Press SIMUL at the compact model and AUX at the modular model.
* Indicate the XZ coordinates of the position to be displayed at the center of the screen.
* Set the width of the display area.

After keying in each value, press **[ENTER]**.



To check the part, press . This will start the corresponding graphic simulation.

Press **[CLEAR]** to clear the screen, and **[END]** to quit the simulation mode.

3.10.2.1 *ZOOM FUNCTION*

With this function, it is possible to enlarge or reduce the whole graphic-representation or part of it. To do this, the simulation of the program must be either interrupted or finished.

Press **[Z]**. The screen will show a rectangle over the original drawing. This rectangle represents the new display area to be enlarged or reduced.

To change the dimensions of the rectangle, use these keys:

[+/−] Reduces the size of the rectangle (zoom in).

[=] Increases the size of the rectangle (zoom out).

Use the following keys to move the zoom window around:

At the compact model [←] [→] [↑] [↓]
At the modular model [0] [•] [↑] [↓]

To set the area selected with the zoom window as he new display area, press **[ENTER]**.

To see he selected area enlarged or reduced while keeping the previous display area values, press [SIMUL] at the compact model and [AUX] at the modular model.

The area contained in the zoom window will now fill the whole screen.

To return to the previous display area (prior to the zoom), press **[END]**.

To use the zoom again, just press **[Z]** and proceed as before.

To quit the ZOOM function and return to the graphic representation, press **[END]**.

3.11 *EDITING PROGRAM 99996*

Program 99996 is a special ISO-coded user program. It can be edited either in this operating mode or at a PC and then sent out to this CNC.

To select this option, press **[AUX]** and after selecting "Auxiliary Modes", press the key corresponding to "**EDITING PROGRAM P99996**".

The CNC displays the editing page for this program.

If the program is currently being edited, the CNC shows a group of program blocks (lines).

Use the to display the display the previous and following lines.

To edit a new line, follow this procedure:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Key in all the pertinent data for that line and press **[ENTER]**.

The programming format to be used is described in the programming manual. The keys on the front panel may be used: **[X]**, **[Z]**, **[S]**, **[F]**, **[N]** as well as: **[TOOL]** for T, for P, for R and for A.

However, since some function keys are missing (G, M, I, K), an assisted editor is also available.

To access it, press **[AUX]**. After analyzing the syntax of what has been edited so far, the CNC will display, one by one, all the functions which can be edited at the time.

Press **[CL]** to delete characters.

To modify a previously edited line, proceed as follows:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Press **[RECALL]**. The bottom of the screen of the CNC, editing area, will show the contents of that line.

3.- Use one of these methods to modify the contents:

a) Use the **[CL]** key to delete characters and edit it as described above.

b) Use the keys to position the cursor over the section to be modified and use the **[CL]** key to delete characters or **[INC/ABS]** to insert data.

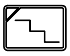
While in the data inserting mode, the characters behind the cursor appear blinking. It is not possible to use assisted programming (the **[AUX]** key).

Key in all the desired data and press **[INC/ABS]**. If the syntax of the new line is correct, the CNC will display it without blinking and, if not, it will show it blinking until it is edited correctly.

4.- Once the line has been modified, press **[ENTER]**. The CNC will assume it replacing the previous one.

To delete a program line, proceed as follows:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Press  and the CNC will delete it from memory.

4. SPINDLE

4.1 SPINDLE OPERATING MODE SELECTION

With this CNC it is possible to work with the spindle in revolutions per minute (RPM) or at Constant Surface Speed (CSS).

Press [CSS] to select the desired work mode. The CNC will show the selected option highlighted, besides, the lamp corresponding to this key will be on when working at Constant Surface Speed and off when working in rpm.

In order for the CNC to change the work mode it is necessary to execute the following sequence: Press [CSS], select the new spindle speed and press  . If this sequence is not followed, the CNC will maintain the previous work mode.

The spindle speed value remains active until either a new one is preset, [RESET] is pressed or the CNC is turned off.

4.2 *SPINDLE IN REVOLUTIONS PER MINUTE (RPM)*

Once this work mode has been selected, the CNC shows the following type of information:

F 0100.000 100% RPM 1500 100% T2

Where the selected RPM are indicated followed by the percentage of spindle speed override being applied.

If the CNC does not display this information line, it means that it is not in this mode. To access it, press **[CSS]**.

To select another spindle speed, press **[S]**, and after keying in the desired value, press the key .

A value between S0 and S9999 rev./min. may be programmed; however, the maximum programmable speed is set by the OEM (consult the instruction book of the machine). The CNC will apply this value whenever a value equal to or greater than this one is programmed.

Once the new speed has been selected, the CNC will act as follows:

* When the spindle is turning, the CNC will output the analog voltage corresponding to the new selected spindle speed.

  If the new rpm correspond to another range, the CNC will either generate or request a range change before outputting the corresponding analog voltage for the new selected speed.

* When the spindle is stopped, the CNC will store the new selected value in order to output, later, the corresponding new analog voltage when the spindle is turned on.

  If the new selected rpm correspond to another range, the CNC will either generate or request a range change.

The programmed spindle speed may be overridden between 50% and 120% with an incremental step of 5% by using the + – keys on the Operator Panel.

4.3 CONSTANT SURFACE SPEED

In order to work in this mode, it is necessary to have an encoder mounted on the spindle.

Before entering this mode (CSS), the right spindle speed range must be selected. If not so, the CNC will select the first range (lowest).

Also, the CSS mode must be selected (**[CSS]** lamp on). If not so, press this key to select it.

Once this work mode (CSS) has been selected, the CRT will display the following type of information:

F 0100.000 100% CSS 180 100% MAX 1500 T2

Which indicates:

* The selected Constant Surface speed (CSS).

  This value is given in m/min or feet/min. depending on the work units currently active.

  To select another value, press [S] and after keying in the desired value, press

* The percentage (**%**) of spindle speed override being applied.

  The programmed Constant Surface Speed may be overridden between 50% and 120% with an incremental step of 5% by using the + – keys on the operator panel.

* The maximum spindle speed permitted (**MAX**).

  To select another maximum speed, press **[S]** and after keying in the new value, press **[ENTER]**.

4.3.1 CONSTANT SURFACE SPEED LIMIT

When working at constant surface speed, it may be interesting to limit the spindle speed (rpm). To do this, the CNC shows the **MAX ????** value corresponding to the maximum speed the spindle may reach.

To set this speed, press **[S]**

Key in the value and press **[ENTER]**

The entered value will appear to the right of **MAX** and, from then on, the spindle speed (rpm) will be limited to this value.

4.4 SPINDLE SPEED RANGE CHANGE

With this CNC, the machine can have a gear box in order to adapt the speeds and torques of the spindle motor to the various machining requirements.

When the new spindle speed "S" requires a range change, the CNC will act as follows:

* If the machine has an automatic range changer, the CNC will select the corresponding range.

* When the machine does not have an automatic range changer, the CNC interrupts the execution of the program and indicates to the operator which range corresponds to the selected "S" speed so he can change it.

4.4.1 MANUAL SPINDLE RANGE CHANGE

When not having an automatic range changer and the new selected spindle speed "S" requires a range change, the CNC acts as follows:

1.- Once the need for a range change has been detected, the CNC will show at the last line of the editing window, the range to be selected.

2.- After making the range change, press [ENTER].

3.- The CNC will then consider the range change completed and it will output the analog voltage corresponding to the new spindle speed selected.

If the new selected spindle speed "S" requires a range change and it is to be ignored, press [CLEAR] in step "2" instead of [ENTER]. The CNC will cancel the range change operation and it will recover the spindle speed set before.

4.4.2 AUTOMATIC SPINDLE RANGE CHANGE

When having an automatic range changer, the CNC will manage the electrical cabinet to perform that change, thus not requiring operator intervention.

4.5 CLOCKWISE SPINDLE ROTATION

To turn the spindle clockwise once the spindle speed has been selected, press

When the spindle is turning, a new speed may be selected or change the current speed by means of the following keys:

+ Every time this key is pressed, the CNC increases the spindle speed by 5%. The maximum being 120% of the programmed speed.

It must be borne in mind that the maximum speed is limited by the value assigned to the range currently selected.

− Every time this key is pressed, the CNC decreases the spindle speed by 5%. The minimum being 50% of the programmed speed.

4.6 COUNTER-CLOCKWISE SPINDLE ROTATION

To turn the spindle counter-clockwise once the spindle speed has been selected, press

When the spindle is turning, a new speed may be selected or change the current speed by means of the following keys:

+ Every time this key is pressed, the CNC increases the spindle speed by 5%. The maximum being 120% of the programmed speed.

It must be borne in mind that the maximum speed is limited by the value assigned to the range currently selected.

− Every time this key is pressed, the CNC decreases the spindle speed by 5%. The minimum being 50% of the programmed speed.

4.7 SPINDLE STOP

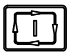
To stop the spindle rotation, press STOP

The CNC stores the "S" speed which was selected before stopping and the spindle resumes this speed when pressing the or key.

4.8 SPINDLE ORIENTATION

If the manufacturer offers this feature on the machine (standard on this CNC), the operator may orient the spindle to the desired angular position.

To do this, key in the following keystroke sequence:

* **[S]** , the bottom of the screen will display: **"S POS ="**.
* Key in the desired orientation angle value. For example: S20 or S35.006
* Press  .

Every time the spindle is being oriented after working in open loop (in regular rpm mode), the CNC will slow down the spindle speed below the value indicated by parameter P706 (if it was turning); it will home it (search for the marker pulse of the spindle encoder) and it will finally position (orient) it to the specified angle (S POS=).

This spindle position will be displayed in whole degrees and in large characters as:
S320 °

The spindle will only be homed prior to being oriented whenever it is switched from operating in open loop to doing it in closed loop (orientation).

When changing back from closed loop to open loop, the display will show the spindle rpm by replacing the "°" symbol with **"RPM"** characters.

The spindle will switch to open loop when pressing  or  ; after an emergency or on power-up.

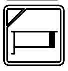
5. AUTOMATIC OPERATIONS

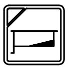
5.1 INTRODUCTION

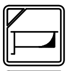
This CNC has a series of keys to access each of the basic operations offered for a lathe. All these keys have a lamp which stays on while the corresponding function is selected. To de-select it, just press the key again.

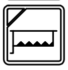
The basic operations to choose from by pressing each one of these keys are:

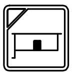
 Turning

 Facing

 Taper turning

 Rounding

 Threading

 Grooving

Every time one of these operations is selected, the CNC shows at the bottom of the screen the data corresponding to the selected operation and a help graphic.

All the basic operations, except grooving, can be executed in different ways either in "SEMI-AUTOMATIC" or "AUTOMATIC" mode ("CYCLE").

5.1.1 AUTOMATIC OPERATIONS IN SEMI-AUTOMATIC MODE

When selecting the "SEMI-AUTOMATIC" mode, the operator controls the movements of the machine by means of either mechanical handwheels, electronic handwheels or JOG keys.

However, in order to make it easier on the operator, it is possible to set the initial and final points (BEGIN and END), the slope of a chamfer, the rounding radius, etc.

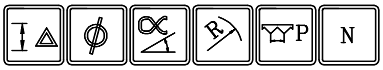
Before starting to execute an automatic operation, it is necessary to define the machining conditions (spindle speed, axes feedrate, tool, etc). However, it is possible to select other values while executing automatic operations. To do this, their execution must be interrupted first.

5.1.2 AUTOMATIC OPERATIONS IN AUTOMATIC MODE (CYCLE)

When selecting the automatic mode "CYCLE", it is possible to define an operation and execute it automatically.

In each operation it is necessary to define the data associated with that operation besides the "BEGIN" and "END" points.

The keys available for selecting these data are:



To define a data which has no selection key (%, H, TW), press another data's key (for example: **[BEGIN]**) and use the up and down arrow keys to select the desired data.

Each machining operation also has the following parameters associated with it: "finishing pass, finishing feedrate, finishing tool and safety distances along "X" and "Z". To define them, press **[AUX]** and operate as described in the section on "Cycle finishing pass and safety distance" of the chapter on "Auxiliary functions" of this manual.

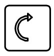
It must be borne in mind that, in order to obtain a proper part finish, the CNC applies tool radius compensation on the finishing pass. Therefore, it is necessary to indicate, at the tool offset table, the value of the tool tip radius and the location code of the tool (tool shape) to be used in this operation.

5.1.2.1 MACHINING CONDITIONS

The information shown at the main window when selecting the automatic mode (CYCLE) is the following:

When operating in Constant Surface Speed:

F % CSS % MAX T

When **<u>not</u>** operating in Constant Surface Speed: 

F % CSS % T

The meaning of each one of these fields is: 

F Currently selected axis feedrate.

% Currently applied % override to the programmed axis feedrate "F".

CSS Constant Surface Speed to execute the cycle.

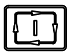
RPM Spindle speed to execute the cycle.

Both values are the ones currently selected to execute the cycle and the "S" value shown at the main window corresponds to the actual (real) spindle speed.

To set the spindle speed, follow one of these procedures:

* Press **[S]**, key in the desired value and press **[ENTER]**.

  This value is taken as the one to be used in the operation being edited. Therefore, it does not modify the actual spindle speed nor the "S" value displayed at the main window.

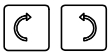
* Press **[S]**, key in the desired value and press 

  The CNC modifies the actual (real) spindle speed updating the "S" value displayed at the main window.

  This new values is also taken as the one to be used when executing the automatic operation being edited.

% Percentage of the programmed spindle speed "S" currently applied.

MAX Maximum spindle speed, in rpm, when operating in CSS.

 Spindle turning direction to be used when executing the cycle.

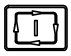
To change the turning direction to be used when executing the cycle, press **[3]**. The CNC will show the new selected direction but it will not modify the actual status of the spindle.

T The tool to be used when executing the cycle.

To select the number of the tool to be used during the cycle, use one of the following methods:

* Press **[TOOL]** and, after keying in the desired number, press **[ENTER]**.

  The CNC stores the new selected tool but it will maintain the previous one active.

* Press **[TOOL]** and, after keying in the desired number, press 

  The CNC selects the new tool and assumes it for the automatic operation being edited.

  It must be borne in mind that the CNC uses this tool for roughing and that it is possible to select another tool for the finishing pass.

Atention:

When storing an automatic operation, the CNC stores all these machining conditions together with the values and parameters defining the cycle. This way, when executing a part previously stored, the CNC will execute each one of the automatic operations with the tool, spindle turning direction, spindle speed, finishing pass, finishing tool and safety distances defined while editing it.

5.1.3 SIMULATION

When selecting the Automatic mode (Cycle), it is possible to check an automatic operation in dry-run before executing it.

To do this, press SIMUL at the compact model and AUX at the modular model.

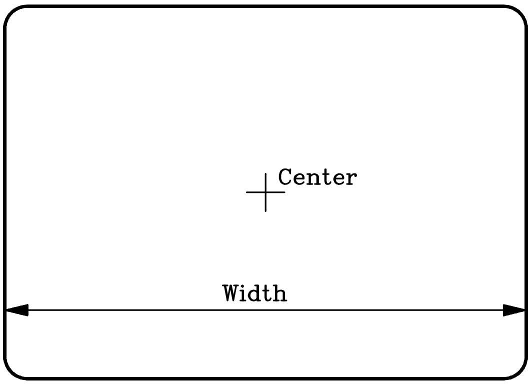
The screen shows a graphic page.

The lower left-hand side of the screen shows the axes of the plane.

To define the display area proceed as follows:

* Press SIMUL at the compact model and AUX at the modular model.
* Indicate the XZ coordinates of the position to be displayed at the center of the screen.
* Set the width of the display area.

After keying in each value, press **[ENTER]**.



To check the part, press . This will start the corresponding graphic simulation.

Press **[CLEAR]** to clear the screen, and **[END]** to quit the simulation mode.

5.1.3.1 ZOOM FUNCTION

With this function, it is possible to enlarge or reduce the whole graphic-representation or part of it. To do this, the simulation of the program must be either interrupted or finished.

Press **[Z]**. The screen will show a rectangle over the original drawing. This rectangle represents the new display area to be enlarged or reduced.

To change the dimensions of the rectangle, use these keys:

[+/−] Reduces the size of the rectangle (zoom in).

[=] Increases the size of the rectangle (zoom out).

Use the following keys to move the zoom window around:

At the compact model [←] [→] [↑] [↓]
At the modular model [0] [•] [↑] [↓]

To set the area selected with the zoom window as he new display area, press **[ENTER]**.

To see he selected area enlarged or reduced while keeping the previous display area values, press [SIMUL] at the compact model and [AUX] at the modular model.

The area contained in the zoom window will now fill the whole screen.

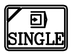
To return to the previous display area (prior to the zoom), press **[END]**.

To use the zoom again, just press **[Z]** and proceed as before.

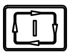
To quit the ZOOM function and return to the graphic representation, press **[END]**.

5.1.4 EXECUTION

When selecting the automatic mode (Cycle), it is possible to execute an automatic operation from beginning to end or one pass at a time by pressing 

To de-select this mode and return to the standard execution mode, press 

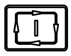
again.

Once the desired execution mode has been selected, press 

The CNC assumes the selected machining values F, S, T as well as the spindle turning direction and it executes the automatic operation following these steps:

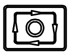
1.- The CNC starts the spindle with the set spindle speed S and turning direction.

2.- If the execution of the automatic operation has been programmed with a new tool T, the CNC will move the axes to the tool change position if the machine so requires.

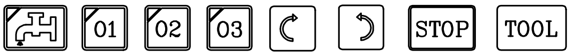
When the tool is changed manually; once it has been changed, press  to resume the execution of the operation.

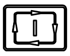
3.- The CNC carries out the machining defined in the automatic operation.

4.- Once the automatic operation has concluded, the spindle stops and the axes return to their position prior to pressing 

To interrupt the program, press 

Once interrupted, the following keys are enabled:

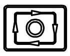


To resume execution, press 

5.1.4.1 TOOL INSPECTION

With this option it is possible to interrupt the execution of an automatic operation and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press  to interrupt the execution of the automatic operation.

b) Press [**TOOL**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the execution was interrupted.

the screen will display the following message:

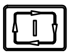
RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the execution was interrupted.

e) Jog the axes to the point of interruption by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

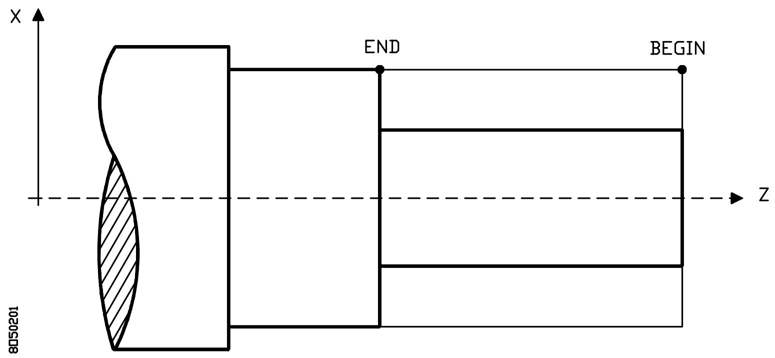
f) Press  to resume the execution of the automatic operation.

5.2 TURNING

This option is selected by pressing [key] and it makes it possible to turn the programmed section.

To select either the "Semiautomatic" or the "Automatic" (Cycle) mode, press [+/–] or [LEVEL +] (at the compact model)

5.2.1 "SEMI-AUTOMATIC" TURNING



It uses the "BEGIN" and "END" values. The CNC displays the selected values and allows to select new ones if so desired.

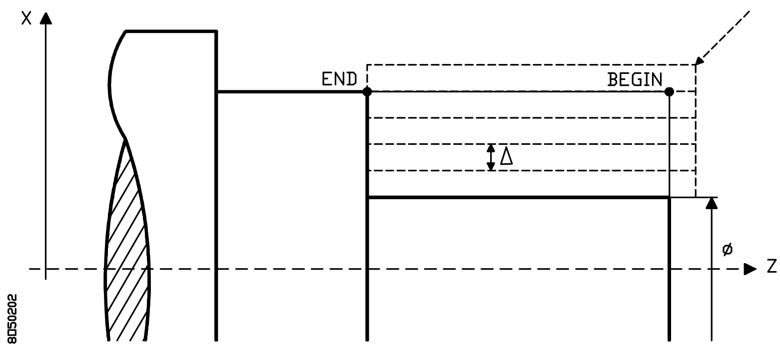
The X axis movements will be made by means of either a mechanical handwheel, the electronic handwheel or the JOG keys on the operator panel.

The Z axis movements will be carried out by using the following keystroke sequences:

[BEGIN] [cycle start] Moves the Z axis only up to the "BEGIN" point.

[END] [cycle start] Moves the Z axis only up to the "END" point.

5.2.2 AUTOMATIC TURNING (CYCLE mode)



The CNC displays the selected values and lets select new ones if so desired.

The following data can also be defined:

f Indicates the final diameter to be obtained in the turning operation.

Δ Indicates the turning pass and it is given by a positive radius value.

If "0" is programmed, the CNC will use the "N" value.

N Defines the number of roughing passes. When completed, the CNC will run the finishing pass. This parameter will be taken into account when Δ=0.

If both the Δ and "N" are programmed with a "0" value, the CNC will issue the corresponding error message.

If a finishing pass has been set (%Δ other than "0"), the CNC behaves as follows:

Example 1: To remove 20 mm with a pass of Δ = 2.

With %Δ=50 The CNC runs ten 1.9mm passes plus a 1mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 The CNC runs ten 2mm passes, the last of which is carried out with the finishing tool T at the selected %F.

If %F = 100, the finishing pass is run at the same feedrate as the roughing passes.

Example 2: To remove 1mm, setting N=1.

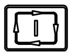
With %Δ=40 The CNC runs one 0.6mm pass plus one 0.4mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 and %F=50 The CNC runs one 1mm pass with the finishing tool T at 50% of the programmed F.

With %Δ=100 and %F=100 The CNC runs one 1mm pass with the finishing tool T at the programmed F.

Basic operation:

The turning operation may be executed from beginning to end without interruptions or a pass at a time by pressing 

Once the proper data has been entered, press  so the CNC executes the turning operation.

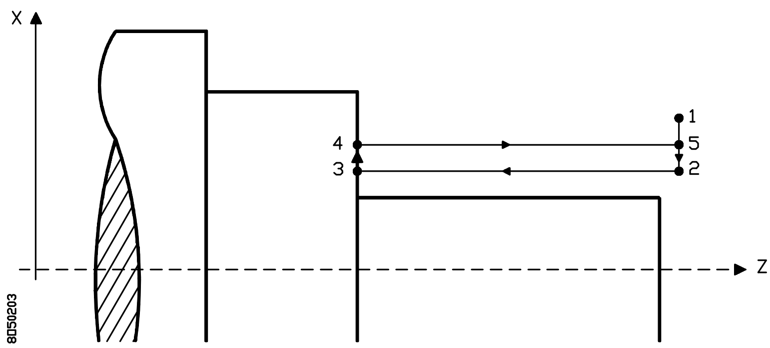
Before starting to execute the part, the CNC will calculate the real pass to be used along the X axis (all the passes will be identical) and the corresponding finishing pass.

The machining steps will be the following:

1.- The spindle will start at the selected speed and turning direction.

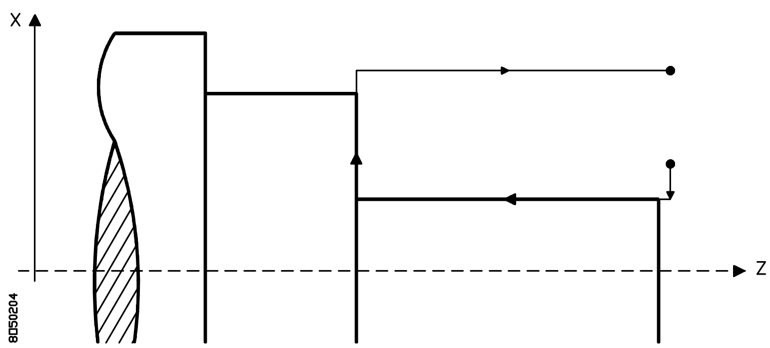
2.- If the cycle has been programmed to be executed with another tool, the CNC will perform a tool change moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the Begin point keeping the selected safety distance from it along the X and Z axes.



4.- Every roughing pass is performed as indicated in the illustration, starting from point 1, going through points 2, 3 and 4 and ending at point 5.

5.- Once the roughing operation has concluded, the CNC runs the finishing pass as indicated below and the turning operation will end at the cycle calling point.



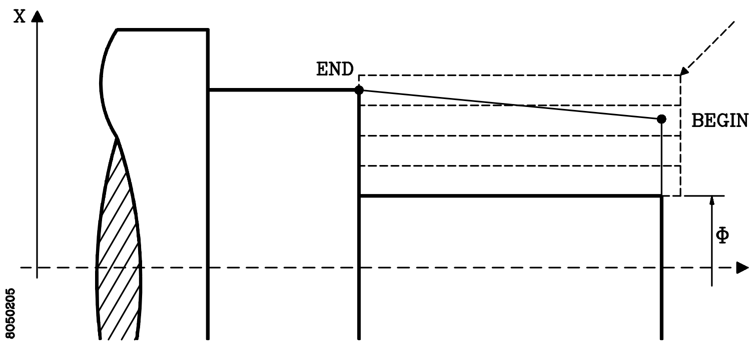
The feedrate for the finishing pass is defined by the % of the current programmed feedrate.

If the cycle does not have a finishing pass, the tool will move to the cycle calling point after running the last roughing pass.

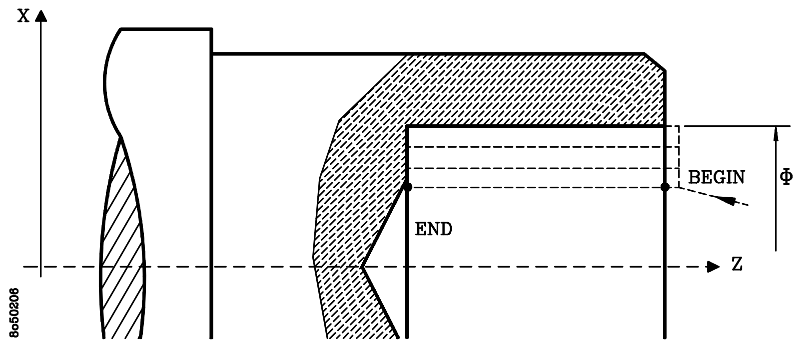
6.- The CNC stops the spindle.

Considerations:

When the surface to be machined is not completely cylindrical, the CNC analyzes the X coordinates of the "BEGIN" and "END" points and it takes as beginning point the one with the outmost X coordinate value.



An inside turning operation is defined in the same way as an outside one. Therefore, the CNC analyzes the programmed end diameter and the BEGIN coordinate in order to tell the type of turning operation to carry out.

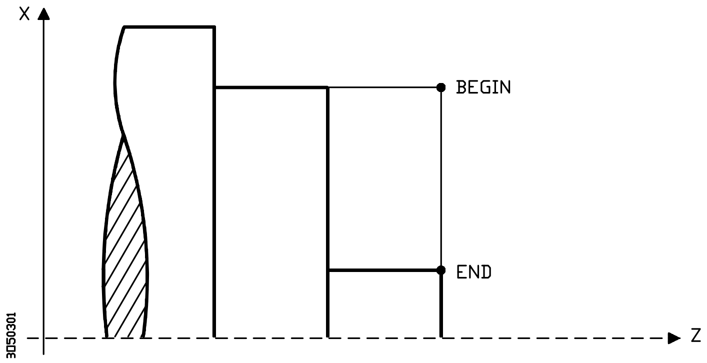


5.3 FACING

This option is selected by pressing and it makes it possible to face the programmed section.

To select either the "Semiautomatic" or the "Automatic" (Cycle) mode, press or LEVEL + (at the compact model)

5.3.1 "SEMI-AUTOMATIC" FACING



It uses the "BEGIN" and "END" values. The CNC displays the selected values and allows to select new ones if so desired.

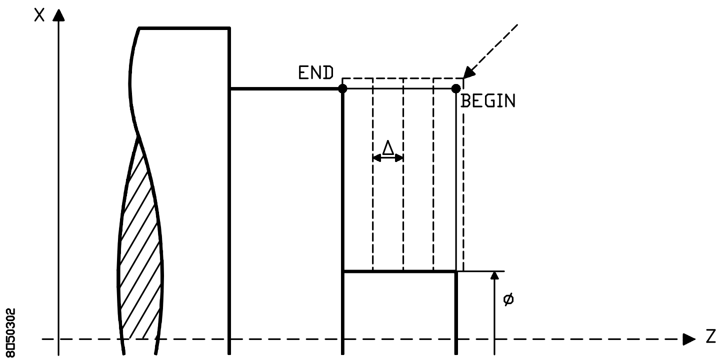
The X axis movements will be made by means of either a mechanical handwheel, the electronic handwheel or the JOG keys on the operator panel.

The Z axis movements will be carried out by using the following keystroke sequences:

BEGIN Moves the Z axis only up to the "BEGIN" point.

END Moves the Z axis only up to the "END" point.

5.3.2 AUTOMATIC FACING (CYCLE mode)



The CNC displays the selected values and allows to select new ones if so desired.

The following data can also be defined:

ϕ Indicates the final diameter to be obtained in the facing operation.

Δ Indicates the facing pass and it is given by a positive radius value.

If "0" is programmed, the CNC will use the "N" value.

N Defines the number of roughing passes. When completed, the CNC will run the finishing pass. This parameter will be taken into account when Δ=0.

If both the Δ and "N" are programmed with a "0" value, the CNC will issue the corresponding error message.

If a finishing pass has been set (%Δ other than "0"), the CNC behaves as follows:

Example 1: To remove 20 mm with a pass of Δ = 2.

With %Δ=50 The CNC runs ten 1.9mm passes plus a 1mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 The CNC runs ten 2mm passes, the last of which is carried out with the finishing tool T at the selected %F.

If %F = 100, the finishing pass is run at the same feedrate as the roughing passes.

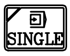
Example 2: To remove 1mm, setting N=1.

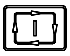
With %Δ=40 The CNC runs one 0.6mm pass plus one 0.4mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 and %F=50 The CNC runs one 1mm pass with the finishing tool T at 50% of the programmed F.

With %Δ=100 and %F=100 The CNC runs one 1mm pass with the finishing tool T at the programmed F.

Basic operation:

The facing operation may be executed from beginning to end without interruptions or a pass at a time by pressing 

Once the proper data has been entered, press  so the CNC executes the facing operation.

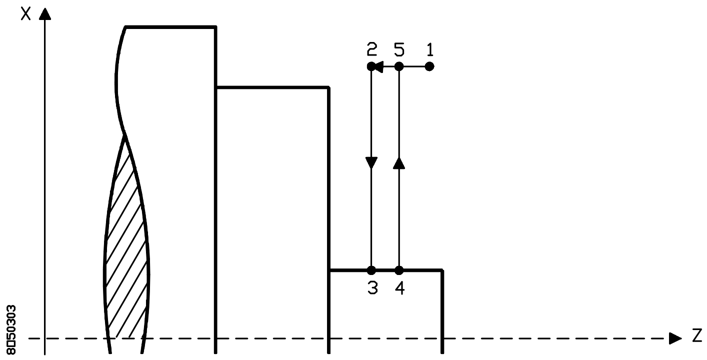
Before starting to execute the part, the CNC will calculate the real pass to be used along the Z axis to be used (all the passes will be identical) and the corresponding finishing pass.

The machining steps will be the following:

1.- The spindle will start at the selected speed and facing direction.

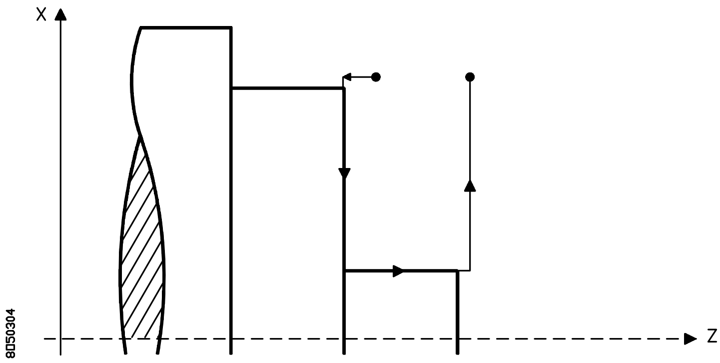
2.- If the cycle has been programmed to be executed with another tool, the CNC will perform a tool change moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the Begin point keeping the selected safety distance from it along the X and Z axes.



4.- Every roughing pass is performed as indicated in the illustration, starting from point 1, going through points 2, 3 and 4 and ending at point 5.

5.- Once the roughing operation has concluded, the CNC runs the finishing pass as indicated below and the facing operation will end at the cycle calling point.



The feedrate for the finishing pass is defined by the % of the current programmed feedrate.
If the cycle does not have a finishing pass, the tool will move to the cycle calling point after running the last roughing pass.

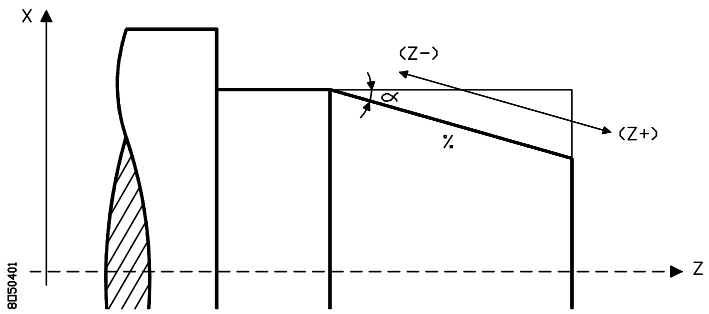
6.- The CNC stops the spindle.

5.4 TAPER TURNING

Press to select this option. It permits the simultaneous machining along the X and Z axes in order to obtain chamfers and tapered surfaces.

To select either the "Semiautomatic" or the "Automatic" (Cycle) mode, press or LEVEL + (at the compact model)

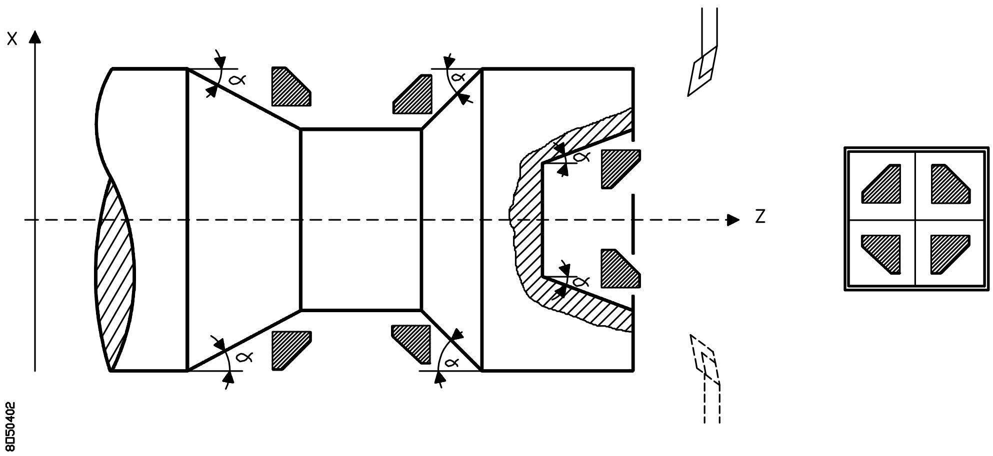
5.4.1 "SEMI-AUTOMATIC" TAPER TURNING



It is possible to perform chamfers being necessary to define the angle "α" or the slope "%" of the chamfer.

Since there is no key to select it, press the one corresponding to another data (for example: and then press the down arrow key to select the "%" data.

It is also necessary to select the type of profile to be machined by means of the up and down arrow keys as indicated below:

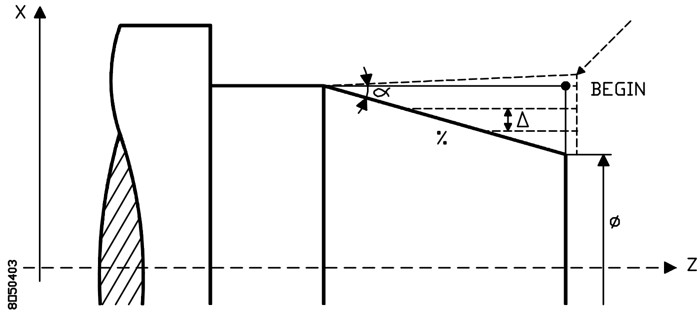


The operator will move the machine to the beginning point by means of either the mechanical handwheel, electronic handwheel or the JOG keys on the operator panel.

To carry out the chamfer with the indicated inclination, use the JOG keys corresponding to the Z axis (Z+ and Z-) according to the direction of movement.

The tool will move with the indicated slope and in the chosen direction until is pressed.

5.4.2 "AUTOMATIC" TAPER TURNING (CYCLE mode)



The CNC displays the selected values and allows to select new ones if so desired.

The following data can also be defined:

ϕ Indicates the final diameter to be obtained in the turning operation.

Δ Indicates the turning pass and it is given by a positive radius value.

If "0" is programmed, the CNC will use the "N" value.

N Defines the number of roughing passes. When completed, the CNC will run the finishing pass. This parameter will be taken into account when=0.

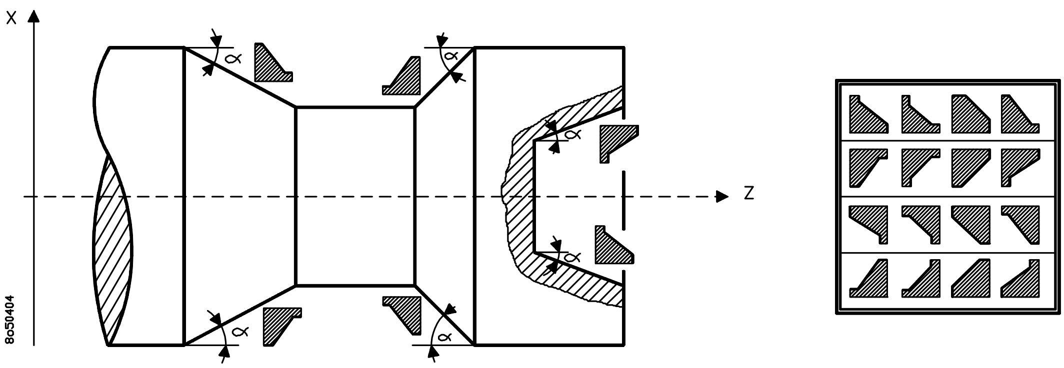
α Defines the angle of the chamfer with respect to the Z axis.

% Defines the slope of the chamfer.

Since there is no key to select it, press the one corresponding to another data (for example:) and then press the down arrow key to select the "%" data.

When the operator defines α or "%", the CNC will update both values.

The CNC, in order to obtain a good part finish, applies tool compensation on the last machining pass and to do so it needs to know the type of profile to be machined. the bottom of the screen shows the currently selected type of profile, use the up and down arrow keys to select another one if so desired.



If a finishing pass has been set (%Δ other than "0"), the CNC behaves as follows:

Example 1: To remove 20 mm with a pass of Δ = 2.

With %Δ=50 The CNC runs ten 1.9mm passes plus a 1mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 The CNC runs ten 2mm passes, the last of which is carried out with the finishing tool T at the selected %F.

If %F = 100, the finishing pass is run at the same feedrate as the roughing passes.

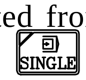
Example 2: To remove 1mm, setting N=1.

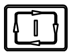
With %Δ=40 The CNC runs one 0.6mm pass plus one 0.4mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 and %F=50 The CNC runs one 1mm pass with the finishing tool T at 50% of the programmed F.

With %Δ=100 and %F=100 The CNC runs one 1mm pass with the finishing tool T at the programmed F.

Basic operation:

The taper turning operation may be executed from beginning to end without interruptions or a pass at a time by pressing 

Once the proper data has been entered, press  so the CNC executes the taper turning operation.

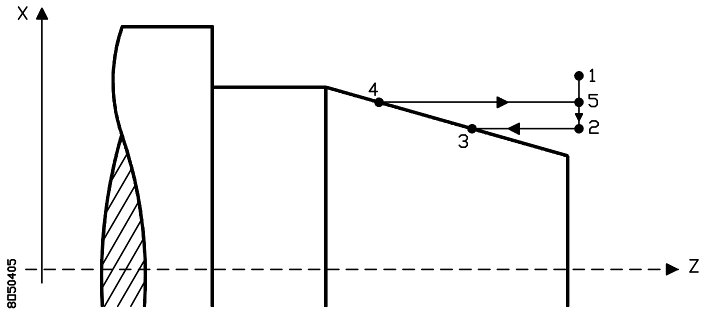
Before starting to execute the part, the CNC will calculate the real pass to be used along the X axis (all the passes will be identical) and the corresponding finishing pass.

The machining steps will be the following:

1.- The spindle will start at the selected speed and turning direction.

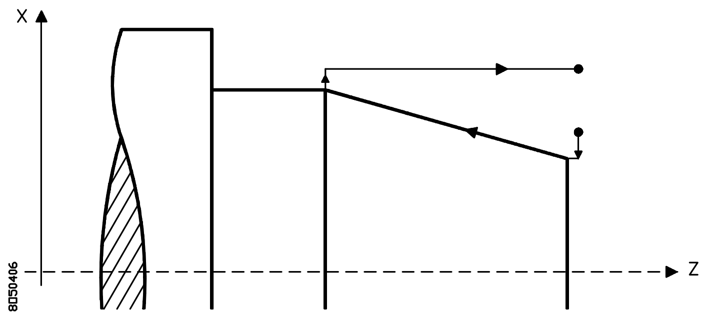
2.- If the cycle has been programmed to be executed with another tool, the CNC will perform a tool change moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the Begin point keeping the selected safety distance from it along the X and Z axes.



4.- Every roughing pass is performed as indicated in the illustration, starting from point 1, going through points 2, 3 and 4 and ending at point 5.

5.- Once the roughing operation has concluded, the CNC runs the finishing pass as indicated below and the turning operation will end at the cycle calling point.



The feedrate for the finishing pass is defined by the % of the current programmed feedrate.

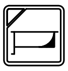
If the cycle does not have a finishing pass, the tool will move to the cycle calling point after running the last roughing pass.

6.- The CNC stops the spindle.

Atention:

It must be borne in mind that the CNC applies tool radius compensation on the last pass or finishing pass in order to obtain the proper part finish. It is necessary to indicate, on the tool offset table, the cutter tip radius value as well as the location code (shape code) of the tool to machine with.

5.5 ROUNDING

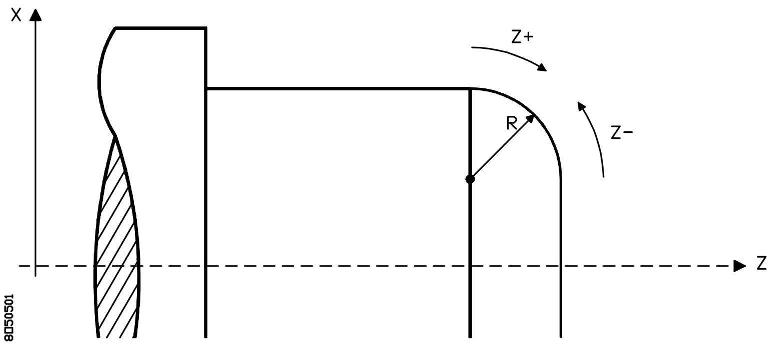
To select this option press 

It can be executed in three different ways: in "SEMI-AUTOMATIC" mode, "AUTOMATIC ROUNDING" mode (CYCLE) and "AUTOMATIC PROFILE ROUNDING" mode (CYCLE).

To change modes, just press  or 

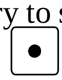
(at the compact model) .

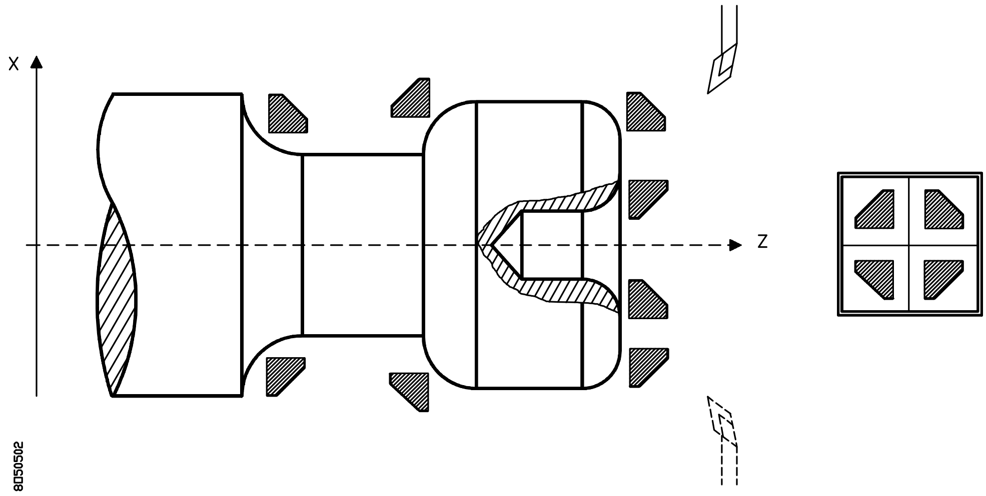
5.5.1 "SEMI-AUTOMATIC" ROUNDING



The following data must be defined:

R Indicates the rounding radius.

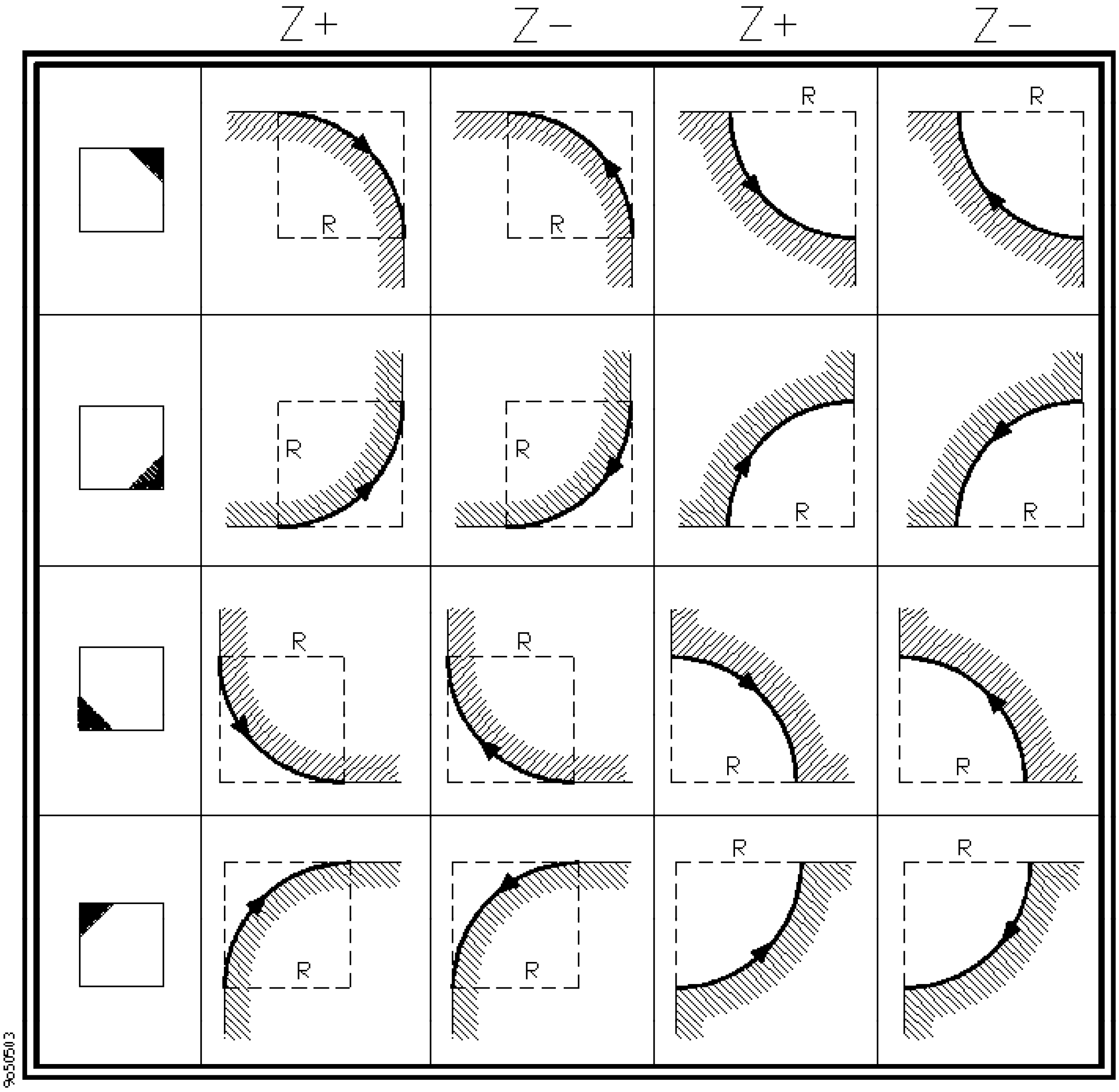
It is also necessary to select the type of rounding to be performed (concave or convex) by means of the  key and the type of profile corresponding to the corner to be rounded by means of the up and down arrow keys as shown below:



Before performing the rounding operation, position the tool at the beginning point by means of either the mechanical handwheel, electronic handwheel or the JOG keys on the operator panel.

Then use the JOG keys for the Z axis (Z+ and Z-) according to the direction of the movement.

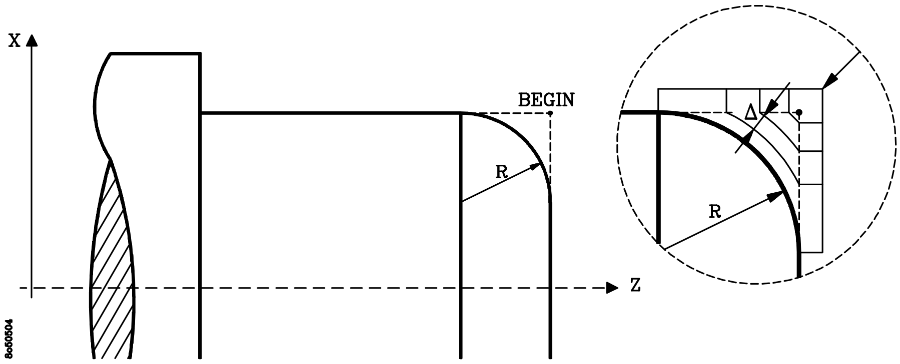
The CNC will perform the corresponding 90° rounding. See figure.



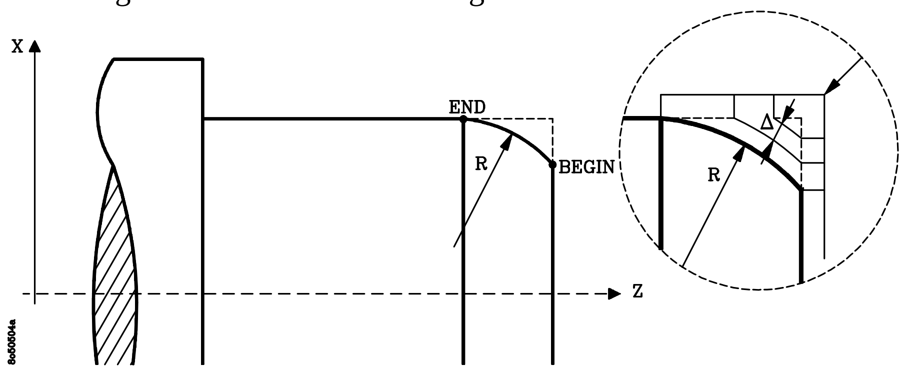
5.5.2 *AUTOMATIC ROUNDING (CYCLE mode)*

The automatic rounding may be defined in two ways:

a) By indicating the theoretical corner to be rounded and its radius.



b) By indicating both ends of the rounding and its radius.



In the a) case, it is necessary to define the "BEGIN" point and in the b) case both the "BEGIN" and "END" points must be defined.

The following data must also be defined:

R Defines the rounding radius.

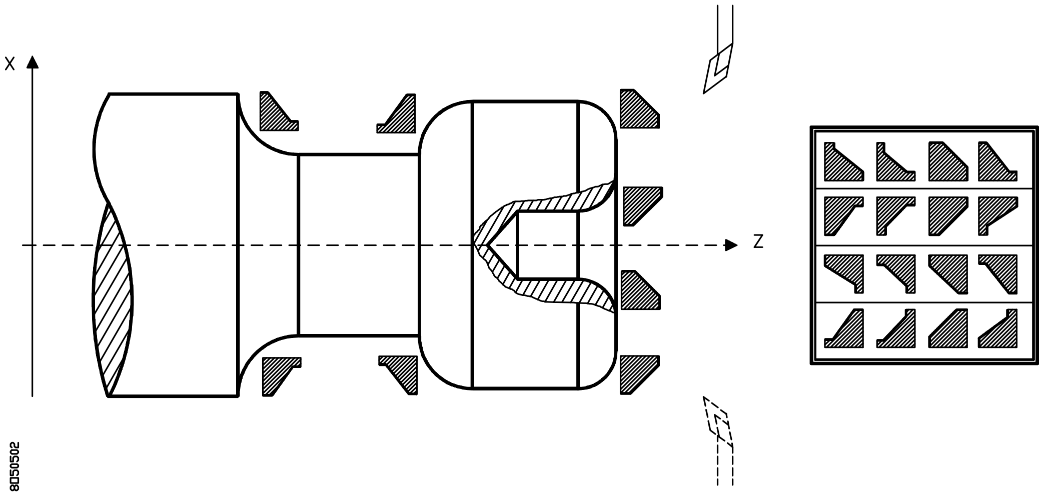
Δ Defines the distance between two consecutive rounding passes.

If programmed with a "0" value, the CNC will take the "N" value into account.

N Defines the number of rounding passes in the roughing stage. When completed, the CNC will run the finishing pass. This parameter will be taken into account when $\Delta = 0$.

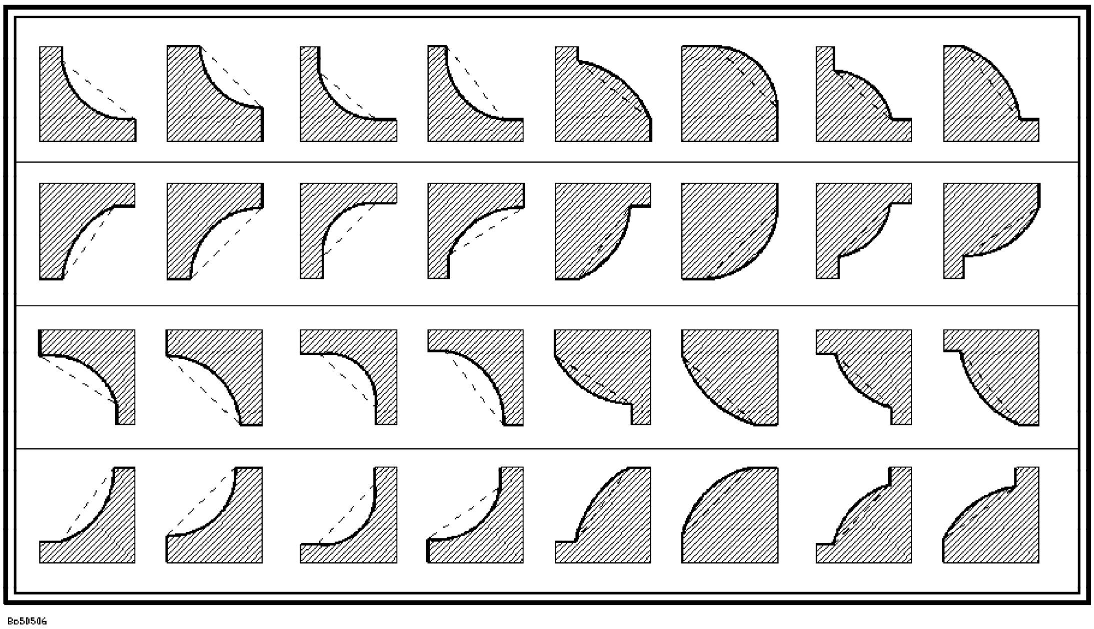
If both "Δ" and "N" are programmed with a "0" value, the CNC will issue the corresponding error message.

The CNC, in order to obtain a good part finish, applies tool compensation on the last machining pass and to do so, it needs to know the type of profile to be machined. The bottom of the screen shows the type of profile currently selected and another may be chosen by means of the up and down arrow keys.



It is also necessary to select, by means of the ● key the type of rounding to carry out (concave or convex).

The various possible profiles are:



If a finishing pass has been set (%Δ other than "0"), the CNC behaves as follows:

Example 1: To remove 20 mm with a pass of Δ = 2.

With %Δ=50 The CNC runs ten 1.9mm passes plus a 1mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 The CNC runs ten 2mm passes, the last of which is carried out with the finishing tool T at the selected %F.

If %F = 100, the finishing pass is run at the same feedrate as the roughing passes.

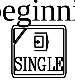
Example 2: To remove 1mm, setting N=1.

With %Δ=40 The CNC runs one 0.6mm pass plus one 0.4mm finishing pass with the finishing tool T at the selected %F.

With %Δ=100 and %F=50 The CNC runs one 1mm pass with the finishing tool T at 50% of the programmed F.

With %Δ=100 and %F=100 The CNC runs one 1mm pass with the finishing tool T at the programmed F.

Basic operation:

The rounding operation may be executed from beginning to end without interruptions or a pass at a time by pressing 

Once the proper data has been entered, press  so the CNC executes the rounding operation.

Before starting to execute the part, the CNC will calculate the actual pass to be used (all the passes will be identical) and the corresponding finishing pass.

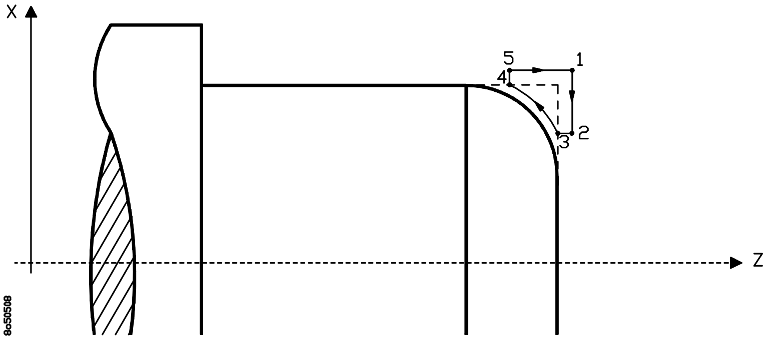
The machining steps will be the following:

1.- The spindle will start at the selected speed and turning direction.

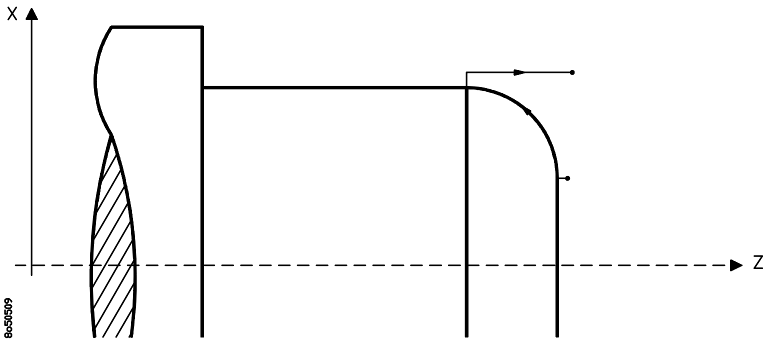
2.- If the cycle has been programmed to be executed with another tool, the CNC will perform a tool change moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the theoretical corner keeping the selected safety distance from it along the X and Z axes.

4.- Every rounding pass is carried out as indicated by the illustration starting at point "1", going through points "2", "3", "4" and "5" and ending at point "1".



5.- Once the roughing operation has concluded, the CNC runs the finishing pass as indicated below and the rounding operation will end at the cycle calling point.



The feedrate for the finishing pass is defined by the current % of the programmed feedrate.

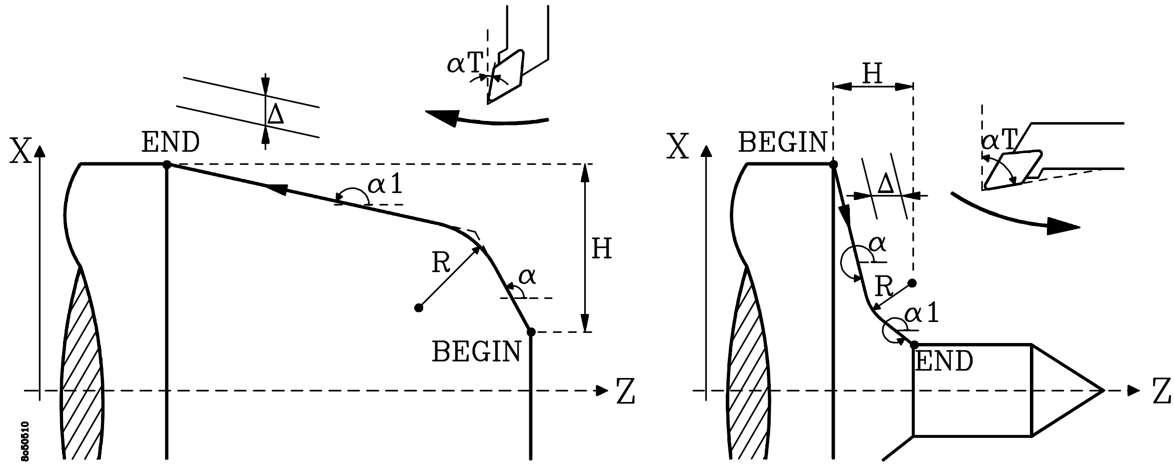
If the cycle does not have a finishing pass, the tool will move to the cycle calling point after running the last roughing pass.

6.- The CNC stops the spindle.

Atention:

It must be borne in mind that the CNC applies tool radius compensation on the last pass or finishing pass in order to obtain the proper part finish. It is necessary to indicate, on the tool offset table, the cutter tip radius value as well as the location code (tool shape) of the tool to machine with.

5.5.3 "AUTOMATIC PROFILE ROUNDING"



The CNC will show the current BEGIN and END values being possible to select new ones if so desired.

The following data must also be defined:

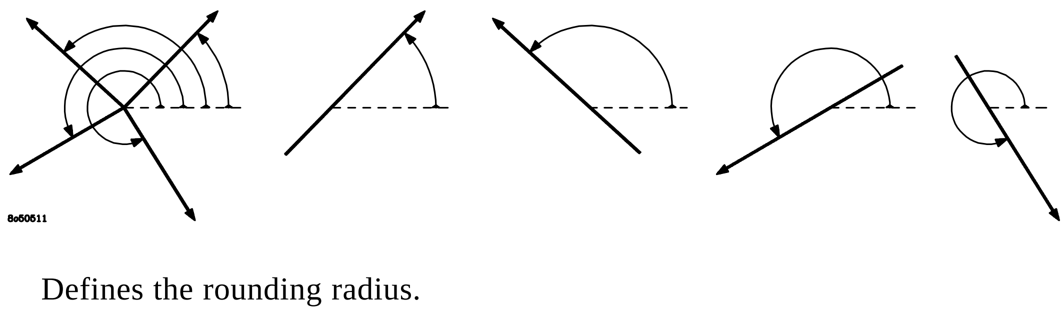
α Defines the angle of the first rounding section (which goes from the BEGIN point) with respect to the Z axis.

$\alpha 1$ Defines the angle of the second rounding section (which ends at the END point) with respect to the Z axis.

Since it has no key for its selection, proceed in either one of the following ways:

* Press the [α] key several times.
* Select another data and then use the up and down arrow keys to select this parameter: $\alpha 1$

When defining the α and $\alpha 1$ angles, indicate the number of degrees between the machining path and the Z axis taking always into account the direction of movement.



R Defines the rounding radius.

H It defines the amount of material to be removed from the original piece. It must be programmed in radius and with a positive value. If programmed with a value of "0", the CNC will issue the corresponding error.

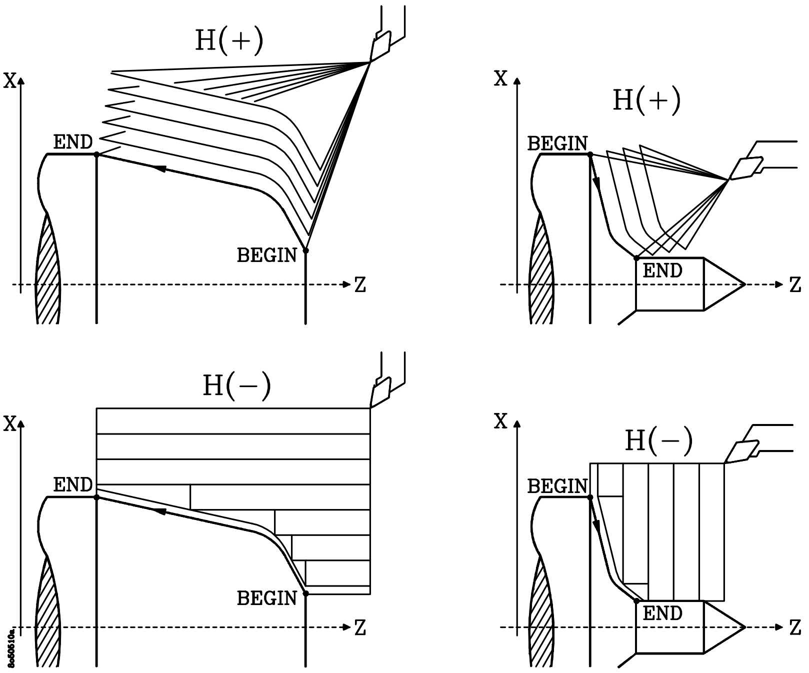
Since there is no key to select it, proceed as follows:

* Select another data and use the up and down arrow keys to select this parameter "H".

Depending on the sign assigned to "H", the roughing of the part will be performed as follows:

"H(+)" It will carry out several passes. All of them parallel to the programmed profile.

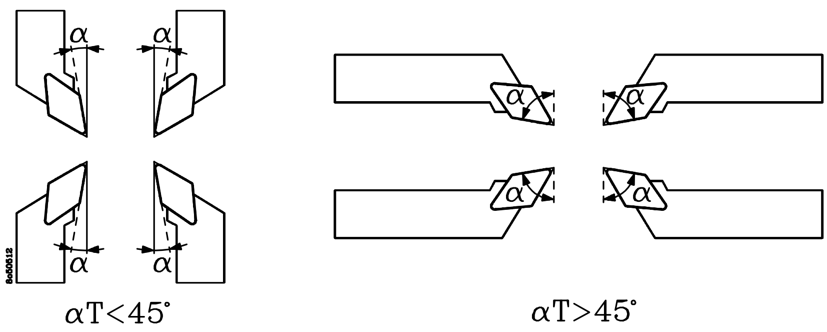
"H(-)" It will first rough out the part, turning or facing, and it will run a final roughing pass to leave the finishing stock.



Δ Defines the distance between two consecutive rounding passes.

If programmed with a value of "0", the CNC will issue the corresponding error.

αT Defines the cutter angle indicating the angle between the cutter and the X axis as shown below:



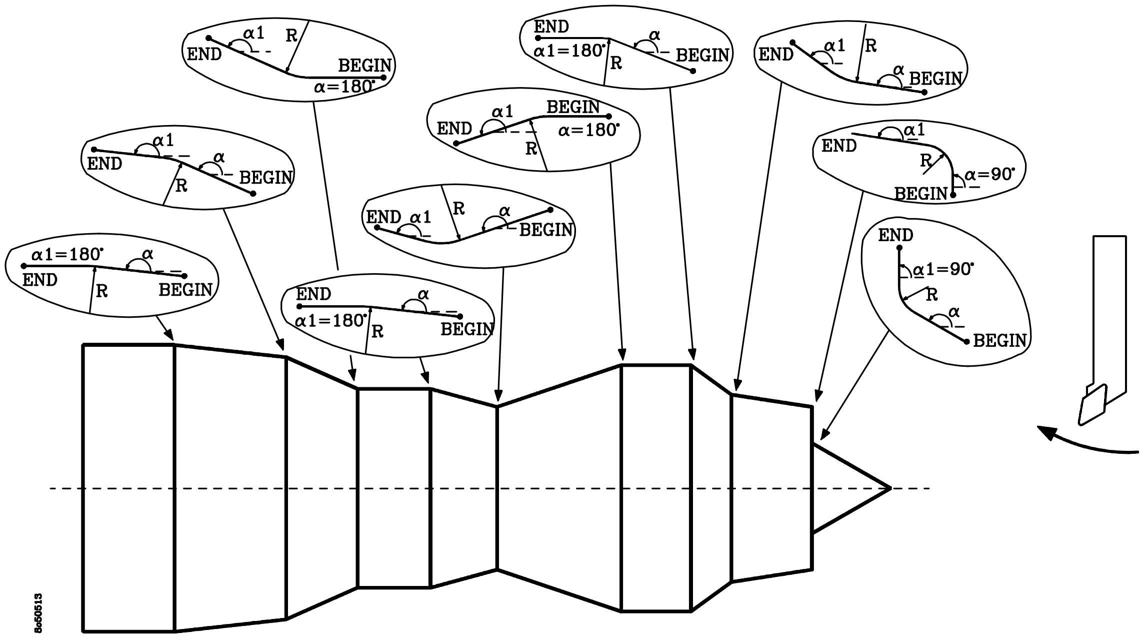
Since there is no key to select it, proceed in one of the following ways:

* Press [α] several times

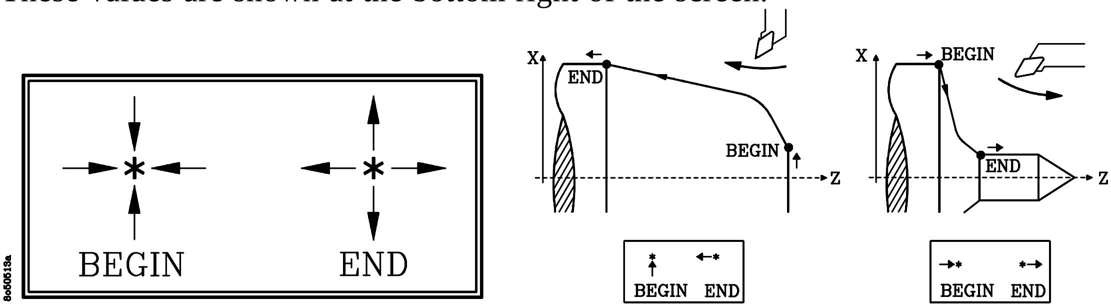
* Select another data and use the up and down arrow keys to select this parameter "**αT**"

If the cutter angle is less or equal to 45°, the "H" and "Δ" values indicate the excess material and the step along X. On the other hand, if the cutter angle is greater than 45°, the "H" and "Δ" values indicate the excess material and the step along Z.

The following figure shows a part indicating several examples of how the "AUTOMATIC PROFILE ROUNDING" function may be used.

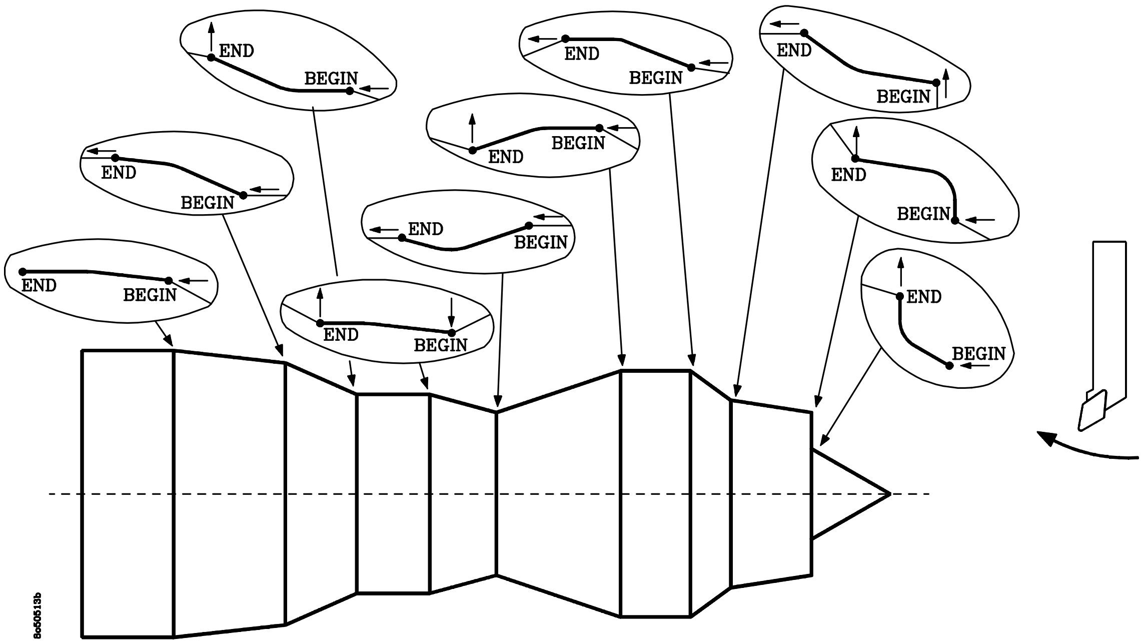


The CNC, in order to obtain a good part finish, applies tool compensation on the last machining pass. To do so, it needs to know how the tool enters and exits the profile. These values are shown at the bottom right of the screen.



To select the way the tool enters the profile (BEGIN point) use the up arrow key and to select the way the tool exits the profile (END point), use the down arrow key.

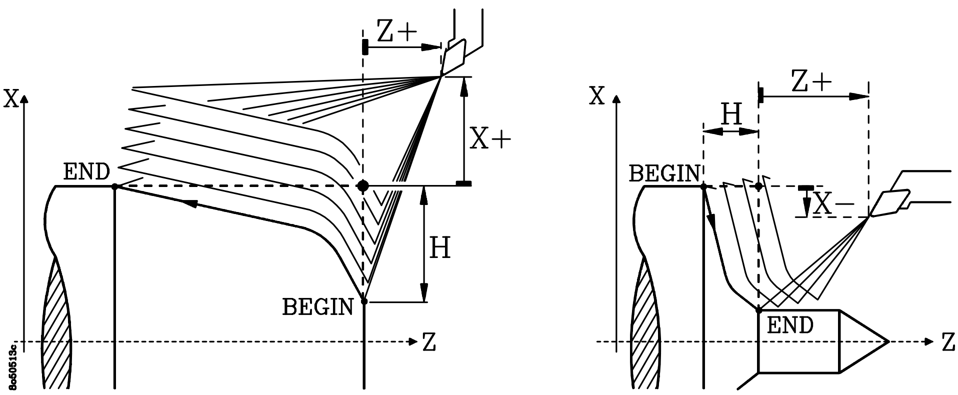
The next figure shows a part indicating with several examples how to define the tool entry and exit.



Before starting the programmed operation, the CNC positions the tool over the "BEGIN" point at an "H" distance from it. This distance is taken along the X axis when the tool angle (φT) is less than or equal to 45° and along the Z axis when is greater than 45°.

However, it is possible to select any other point, referred to it, by means of the safety distances "X" and "Z".

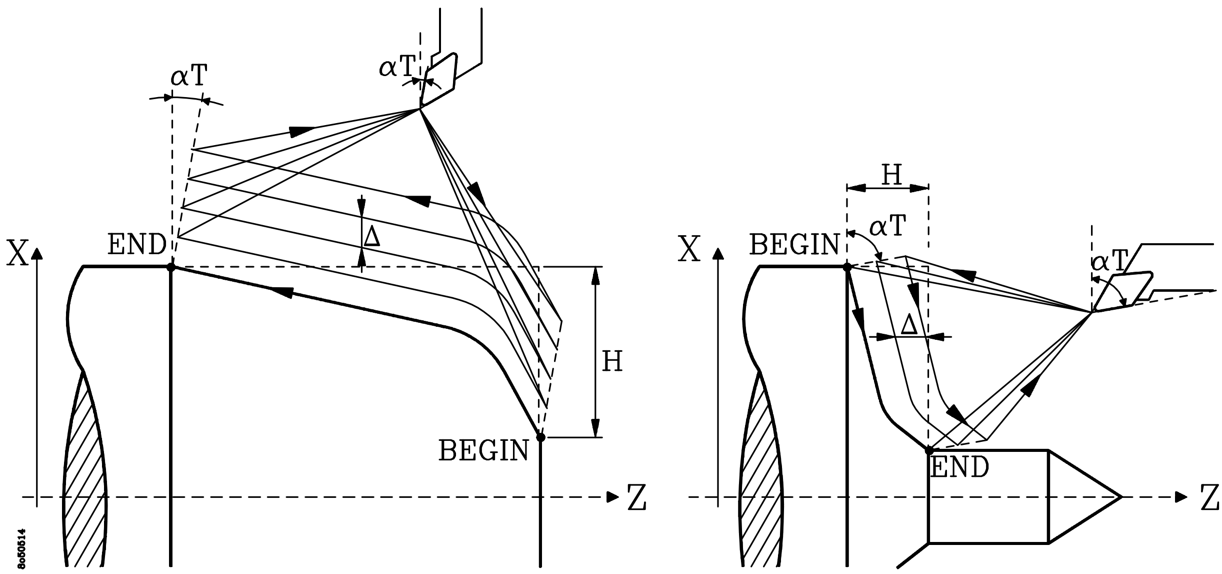
Both parameters must be defined with the corresponding sign as shown below:

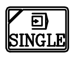


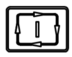
To define the safety distances in "X" and "Z", press [AUX] and proceed as indicated in the section on "Cycle finishing pass and safety distance" of the chapter on "Auxiliary functions" in this manual.

When storing a "profile rounding" as part of part-program, the CNC stores, as it does in the other machining operations, the parameters: Finishing pass, finishing feedrate, finishing tool and safety distances along "X" and "Z" together with the values defined by the operation.

Basic operation:



It is possible to perform the rounding operation from beginning to end without interruption or a pass at a time by pressing 

Once the proper data has been entered, press  for the CNC to execute the rounding operation.

Before starting the part execution, the CNC calculates the actual pass used during the rounding operation (all the passes will be identical) and the corresponding finishing pass.

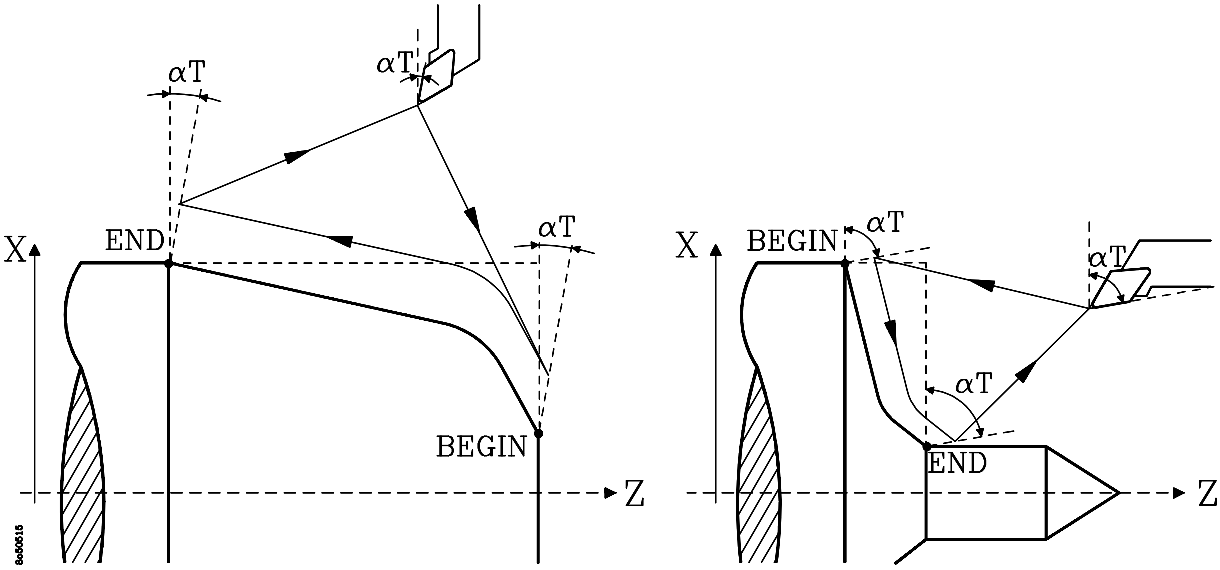
The machining steps will be the following:

1.- The spindle will start at the selected speed and turning direction.

2.- If the cycle has been programmed to be executed with another tool, the CNC will perform a tool change moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the BEGIN point keeping the selected safety distance from it along the X and Z axes.

4.- Successive rounding passes will be run, all of them parallel to the programmed profile and maintaining the tool angle (αT) at the beginning of the profile (BEGIN point) as well as at its exit (END point)



5.- Once the roughing operation has concluded, the CNC runs the finishing pass as indicated below and the rounding operation will end at the cycle calling point.

The feedrate for the finishing pass is defined by the current % of the programmed feedrate.

If the cycle does not have a finishing pass, the tool will move to the cycle calling point after running the last roughing pass.

6.- The CNC stops the spindle.

Atention:

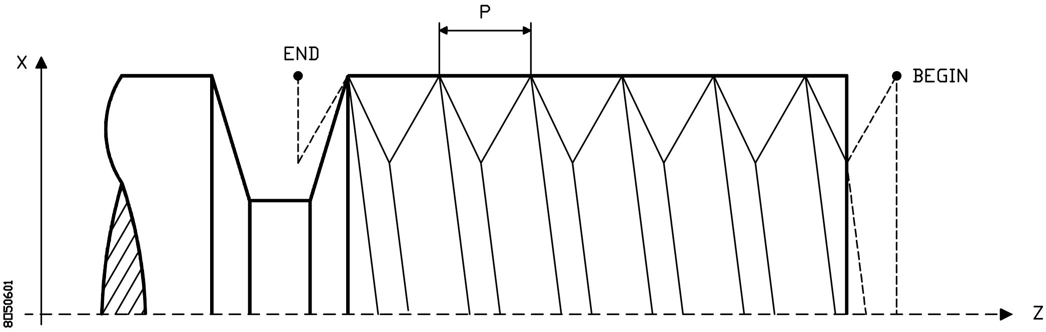
It must be borne in mind that the CNC applies tool radius compensation on the last pass or finishing pass in order to obtain the proper part finish. It is necessary to indicate, on the tool offset table, the cutter tip radius value as well as the location code (tool shape) of the tool to machine with.

When only a single pass is desired, program Δ=H and parameters: %Δ=0 and %F=0.

5.6 THREADING

With this option it is possible to make a thread along the Z axis. Press to select this option.

5.6.1 "SEMI-AUTOMATIC" THREADING



It uses the "BEGIN" and "END" values. The CNC displays the currently selected values and new ones may be set if so desired.

It is also necessary to define "P" corresponding to the pitch of the thread. The right-hand or left-hand threads will be made by previously selecting the turning direction of the spindle.

The X axis movements are carried out by means of either the mechanical handwheel, electronic handwheel or the JOG keys on the operator panel.

The Z axis movements are carried out by the following keystroke sequences:

BEGIN Moves only the Z axis to the "BEGIN" point.

END Moves only the Z axis to the "END" point.

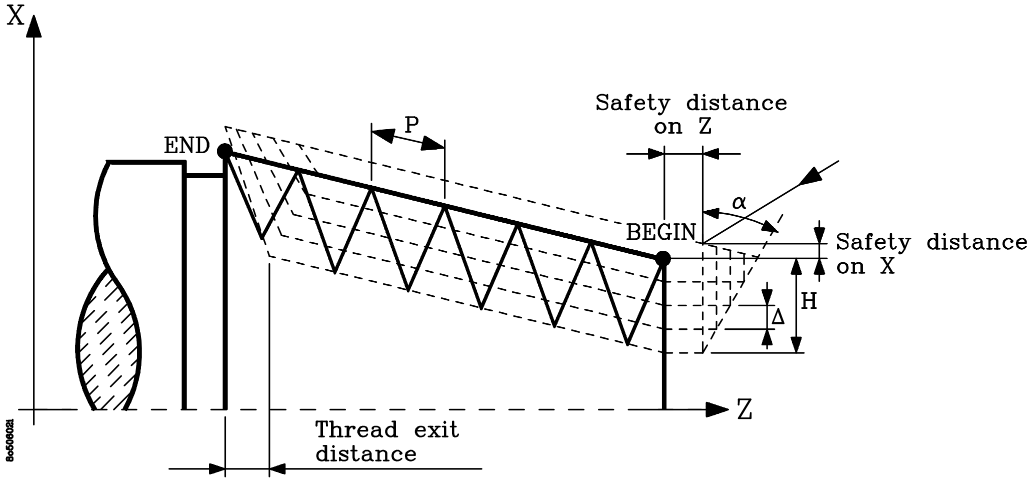
This movement is always synchronized with the spindle. Therefore, this sequence must be followed to perform the threading operation and the sequence BEGIN to withdraw (once the cutter has been withdrawn).

Considerations:

It is a good idea to define the "BEGIN" point away from the work-piece in order for the whole thread to be executed at the same feedrate.

It is also convenient, if the shape of the work-piece allows it, to define the "END" point out of the threading area in order to avoid unwanted machining at that point.

5.6.2 *AUTOMATIC THREADING (CYCLE mode)*



The CNC shows the "BEGIN" and "END" values currently selected and they can be changed if so desired.

If the "BEGIN" and "END" coordinates are aligned along the X axis, the CNC will perform a cylindrical thread and a taper thread if otherwise maintaining the inclination set by these two points.

The threading operation has the "End of thread distance" parameter associated to it besides the ones for "finishing pass, feedrate and safety distances along X and Z".

This parameter indicates the distance at the end of the thread where it begins the exit from it. The CNC keeps threading during this exit movement.

To set it, press **[AUX]** and select the "Thread Exit Distance" option. If set to "0", the cutter will retract straight out, without any Thread Exit Distance.

The following data must also be defined:

P Thread pitch. The right-hand or left-hand threads are done by previously selecting the spindle turning direction.

If a pitch of "0" is programmed, the CNC will issue the corresponding error message.

H Defines the total depth of the thread with a positive radius value.

If programmed with a value of "0", the CNC will issue the corresponding error message.

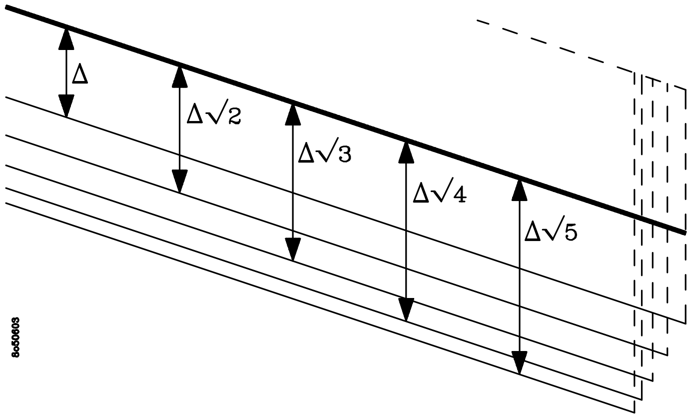
Since there is no key to select it, proceed as follows:

* Press **[P]**, followed by the down arrow key.

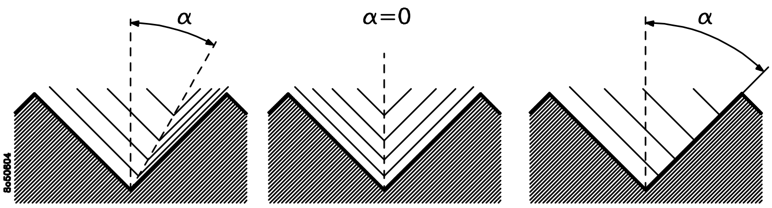
* Select the "P" data and assign its corresponding value, The CNC will then request the "H" value.

Δ Defines the depth of the threading passes. If programmed with a value of "0", the CNC will issue the corresponding error message.

It is given in radius and the depth of each pass is a function of the corresponding pass number ($\Delta\sqrt{n}$) as shown in the next figure:



α Defines the tool penetration angle. It is referred to the X axis.



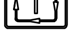
If "0" is programmed, the thread will be done with radial penetration.

If a value equal to half the tool angle is assigned, the penetration will be done along the side of the thread.

Inside and outside threads are possible. The bottom right-hand side of the screen shows the currently selected thread type. Another one may be selected by means of the up and down arrow keys.

Basic operation:

It is possible to execute the thread from beginning to end without interruptions or a pass at a time by pressing 

Once the proper data has been entered, press  for the CNC to perform the thread.

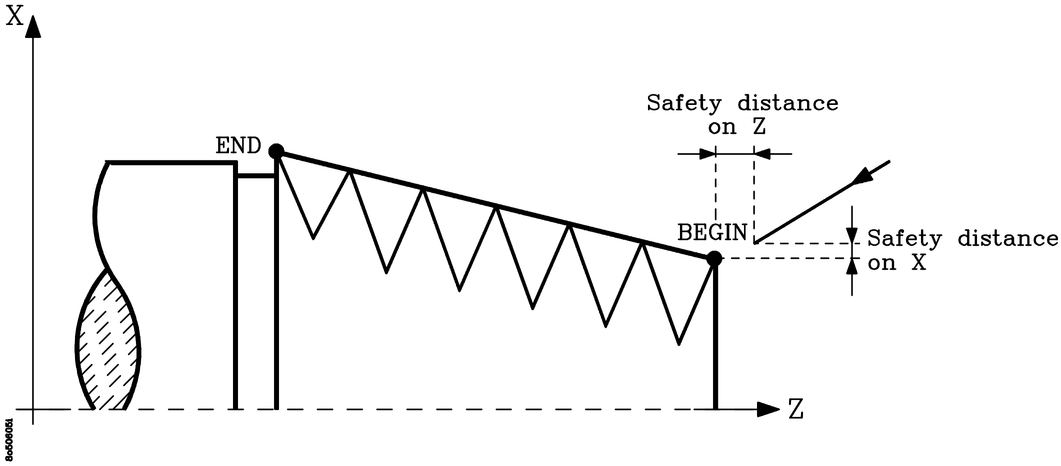
Before starting to execute, the CNC will calculate the real penetration for each threading pass and the depth used at the finishing pass.

The machining steps will be the following:

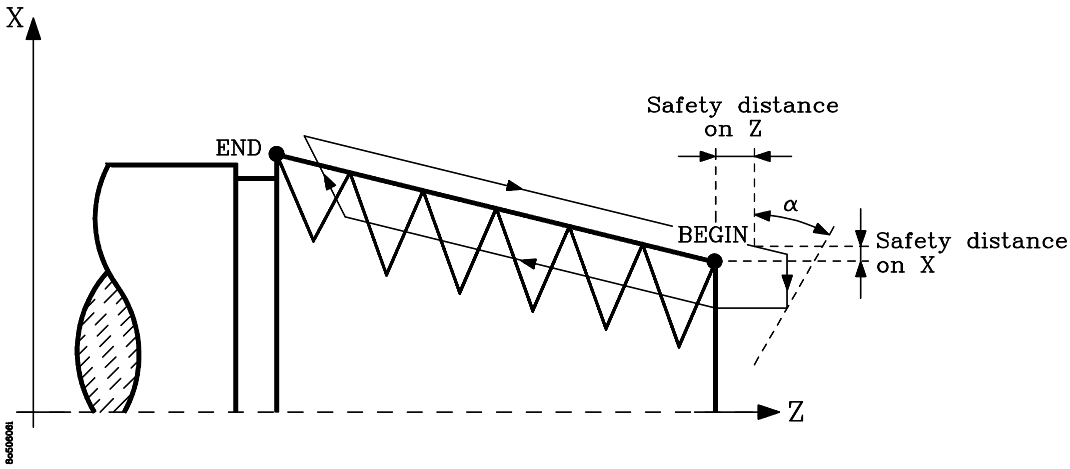
1.- The spindle will start with the selected speed and turning direction.

2.- If the cycle has been programmed to be executed with another tool, the CNC will perform a tool change moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the Begin point keeping the selected safety distance from it along the X and Z axes.



4.- Every threading pass is performed as indicated in the illustration. The Z axis movement is synchronized with the spindle in such a way that all the threading passes always start at the same point.

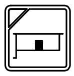


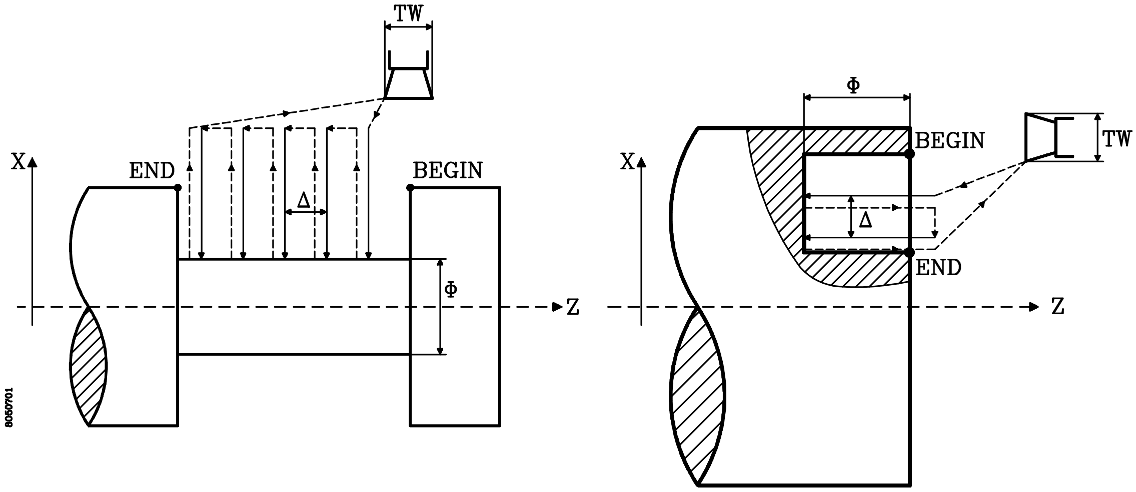
5.- Once the roughing operation of the thread has concluded, if α is other than "0", the last roughing pass is repeated.

6.- The tool returns to the cycle calling point.

7.- The CNC stops the spindle.

5.7 GROOVING

This operation is selected by pressing 



The CNC shows the "BEGIN" and "END" values currently selected and they may be changed if so desired.

The calibrated tool end must be considered when setting the "BEGIN" and "END" points, since the Z axis movements will finish when the calibrated end of the tool reaches that point.

The following data must also be defined:

Φ Indicates the final diameter or final depth to be obtained with the grooving operation.

Δ Defines the grooving step. If programmed with a value of "0", the CNC will take the "N" value into account.

N Defines the number of grooving passes. This parameter will be taken into account when Δ=0.

If both the Δ and "N" are programmed with a "0" value, the CNC will issue the corresponding error message.

TW Indicates the tool width.

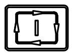
Since there is no key to select it, proceed as follows:

* Press **[TOOL]** followed by the **[Z]** key.
* Press **[N]** followed by the down arrow key.

To obtain a finishing pass, press **[AUX]** and set %Δ to other than "0"

Basic operation:

It is possible to perform a grooving operation from beginning to end without interruptions or a pass at a time by pressing 

Once the proper data has been entered, press  for the CNC to execute the grooving operation.

Before starting to execute, the CNC will calculate the real pass along the Z axis required for the grooving operation. All the passes will be identical.

The machining steps will be the following:

1.- The spindle will start at the selected speed and turning direction.

2.- If the execution of the cycle has been programmed with another tool, the CNC will make a tool change by moving the axes to the tool change position if so required by the machine.

3.- The tool will approach the Begin point maintaining the selected safety distances along the X and Z axes.

4.- Every grooving pass is carried out as follows:

a/ Position the cutter facing the area to be grooved.

The first time it will be positioned at a "TW" (tool width) distance from the "BEGIN" point.

The rest of the times, it will move the grooving pass.

b/ Grooving of the section.

If there is no finishing pass, %Δ = 0, to the bottom of the groove.

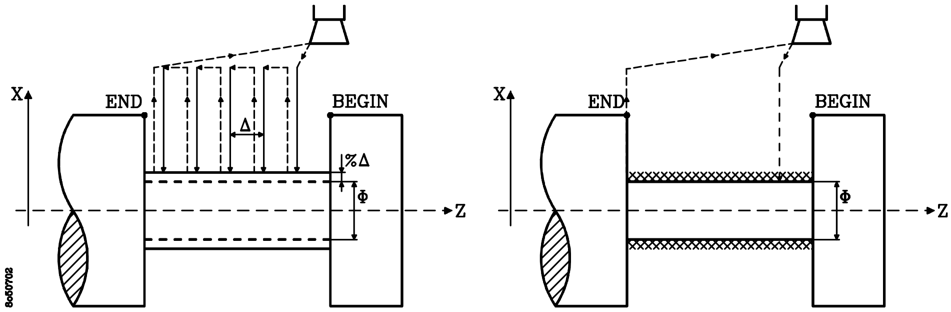
If there is finishing pass, it leaves the amount set by %Δ for the finishing pass.

c/ Dwell at the bottom.

In order to achieve a good part finish, the cutter will stay at the bottom of the groove for another 2 spindle turns.

d/ Withdrawal on X.

5.- If set for a finishing pass, after the last grooving pass, the CNC will run a final finishing pass to the whole groove.



6.- The tool positions at the cycle calling pass.

7.- The CNC stops the spindle.

Considerations:

If the "BEGIN" and "END" coordinates match neither along X nor along Z, the CNC assumes it to be a cylindrical grooving and it takes as beginning point along X the one with the outmost X coordinate value.

5.8. *SIMPLE DRILLING. TAPPING*

By means of this cycle, it is possible to drill or tap a hole along the center line of the part (X0). During this operation the spindle will work in the RPM mode and **NOT** at Constant Surface Speed.

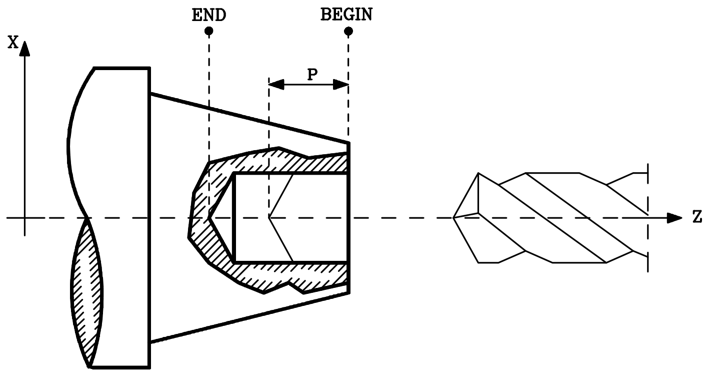
To access this operation, proceed as follows:

* Press [AUX] , select option **[6]** (Other cycles) and, then, "Simple Drilling. Tapping".
* When in DRO mode, press [+/–] or [LEVEL +] and select the "Simple Drilling. Tapping" option.

Once the new cycle has been selected, The CNC will display the following data:

F 1000 %100 RPM 1500 %100 T3

Machining feedrate (m/min or m/rev)
Percentage of Feedrate Override
Spindle RPM. No CSS.
Percentage of spindle speed override
Drilling tool



The CNC will show the "BEGIN" and "END" values currently available. New ones may be selected if so desired.

The following data must also be defined:

P Maximum penetration at each drilling peck. Set P=0 when tapping.

To set the safety distances along X and Z, press [AUX] , the CNC will display a new menu. After setting both values, press [+/–] or [LEVEL +]

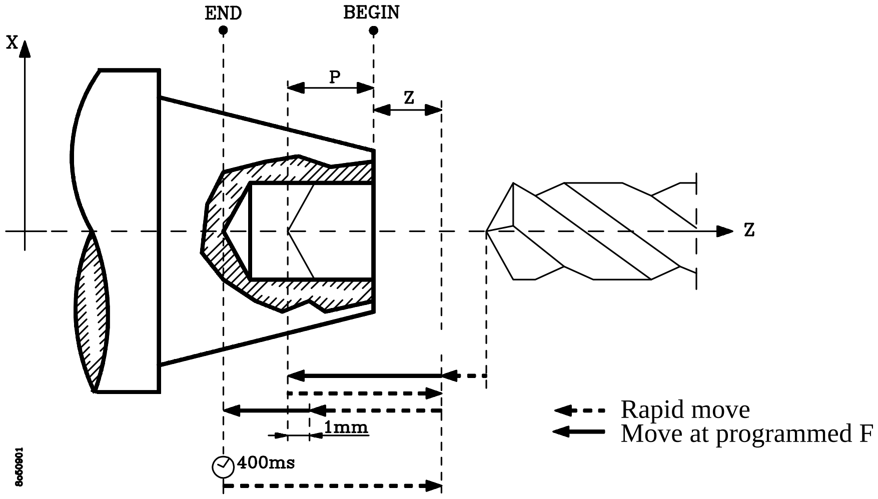
To quit this operation, proceed as follows:

* Press [+/–] or [LEVEL +] to return to the "Other automatic operations" menu (Other cycles).
* Press [+/–] or [LEVEL +] again to return to the DRO mode.

Atention:

When using two spindles with slow direction reversal, it is recommended to set machine parameter "P617(7)=1" (M3/M4 confirmation by detecting spindle feedback reversal).

Basic simple drilling operation:



1.- Starts the spindle at the indicated RPM and turning direction.

2.- Changes the tool if necessary.

3.- Positions the tool in rapid over the BEGIN point (X=0) at the safety distance from it as indicated for the Z axis.

4.- Performs the drilling operation in the following steps:

4.1- Movement at programmed F drilling a distance P.

4.2- Rapid withdrawal to "BEGIN + safety distance" in order to remove drilled-out material.

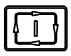
4.3.- Rapid Approach to "P-1mm"; thus positioned at 1 mm from the material.

4.4.- Drilling a distance P at programmed F. Thus travelling P+1mm.

4.5.- Repeat steps: 4.2, 4.3 and 4.4 until the END position is reached.

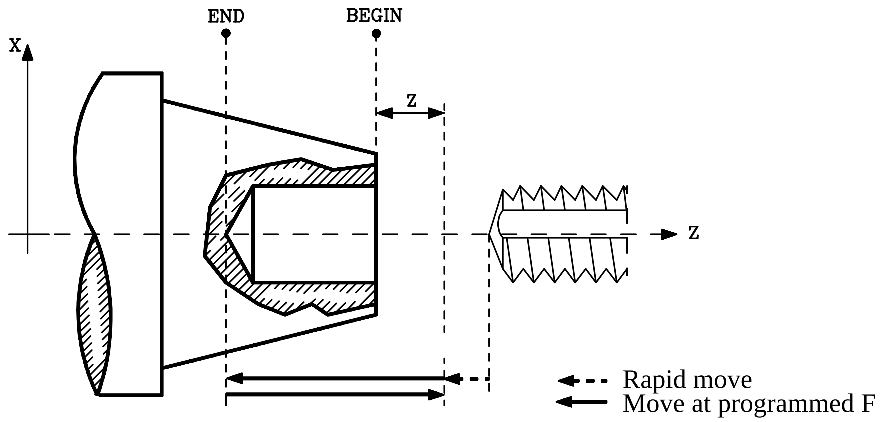
5.- 400msec. dwell at the bottom of the hole in order to improve its finish.

6.- Rapid withdrawal to "BEGIN + safety distance" for drilled-out material removal.

7.- Rapid positioning to where  was pressed.

8.- The CNC stops the spindle.

Basic tapping operation:



1.- Starts the spindle at the indicated RPM and turning direction.

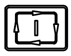
2.- If the cycle is to be executed with another tool, the CNC will carry out a tool change by moving the axes to the tool change position if so required by the machine.

3.- Positions the tool at point X0, part center, at the set Z safety distance from the "BEGIN" point.

4.- Tap-in motion to the "END" point at the programmed feedrate.

5.- Spindle turning reversal.

6.- Tap-out movement (withdrawal) to "BEGIN + safety distance Z".

7.- Rapid traverse (positioning) to where  was pressed.

8.- The CNC stops the spindle.

5.8.1 PROGRAMMING EXAMPLES

Drilling:

A 20 mm hole is to be drilled on the face of the part along its center line using T05 and the spindle turning at 2000 rpm.

The drilling feedrate is 1mm/min. with 5mm pecks in order to remove the drilled out material.

F0001.000	100%	RPM 2000	100%	T05
BEGIN Z	0000.000			
END Z	-0020.000			
P	0005.00			

Safety distance Z = 3.000
Safety distance X = 10.000

Tapping:

The hole in the previous example is to be tapped at a feedrate of 1mm/mm. with the spindle turning at 1000 rpm. The tapping tool is T06.

F0001.000	100%	RPM 1000	100%	T06
BEGIN Z	0000.000			
END Z	-0020.000			
P	0			

Safety distance Z = 3.000
Safety distance X = 10.000

5.9 MULTIPLE DRILLING

With this operation it is possible to drill concentric holes on the face of the part (along the Z axis) as well as on its cylindric side (along the X axis).

To access this operation, proceed as follows:

* Press [AUX] , select option **[6]** (Other cycles) and, then, "Multiple Drilling".
* When in DRO mode, press [+/–] or [LEVEL +] and select the "Multiple Drilling" option.

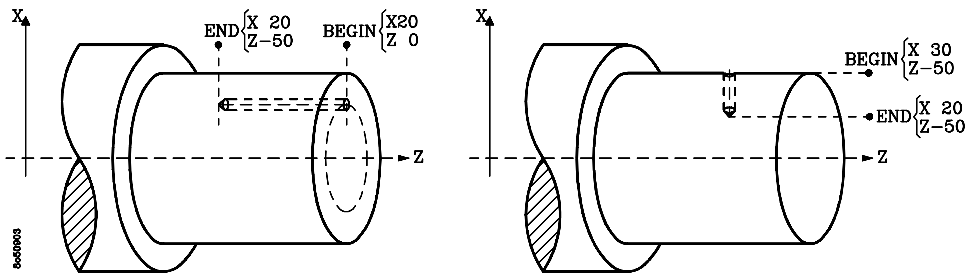
Once the new cycle has been selected, The CNC will display the following data:

The CNC shows this cycle when having spindle orientation and live tool.

Once selected, the CNC will show the following type of information:

F 1000 %100 TRPM 1500 %100 T3

Machining feedrate (m/min)
Percentage of Feedrate Override
Live tool RPM
Live tool speed override
if P609(8)=1.
Drilling tool



The CNC will show the "BEGIN" and "END" values currently available. New ones may be selected if so desired.

The following data must also be defined:

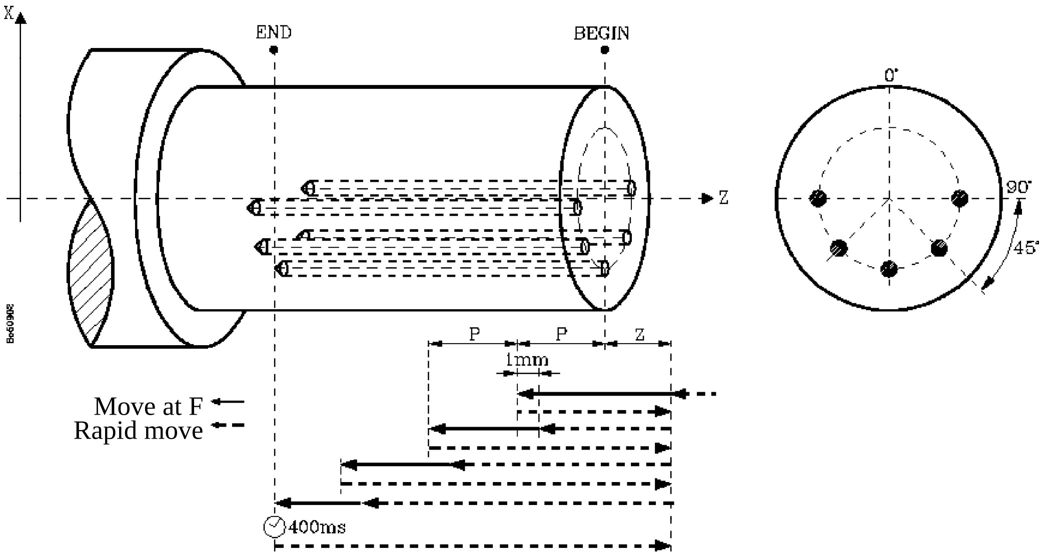
P Maximum penetration per peck drilled-out material removal
α Angular position of first hole
Δ Angular increment between holes
N Number of holes to be drilled

To set the safety distances along X and Z, press [AUX] , the CNC will display a new menu. After setting both values, press [+/–] or [LEVEL +]

To quit this operation, proceed as follows:

* Press [+/–] or [LEVEL +] to return to the "Other automatic operations" menu.
* Press [+/–] or [LEVEL +] again to return to the DRO mode.

Basic operation:



1.- Does a home search on the spindle if it was in the rpm mode before.

2.- Makes a tool change if necessary.

3.- Starts the live tool at the indicated rpm (TRPM).

4.- Orients the spindle to the indicated angular position (α)

5.- Positions the axes over the BEGIN point keeping the safety distances.

6.- Performs the drilling operations in the following steps:

6.1- Drill a distance P at the programmed F.
6.2- Rapid withdrawal to "BEGIN + Safety Distance" for drilled-out material removal.
6.3.- Rapid approach to "P-1mm".
6.4.- Drill the next P increment at the programmed F. Thus travelling "P+1mm."
6.5.- Repeat steps 6.2, 6.3 and 6.4 until the END is reached.

7.- Stays at the bottom of the hole for 400msec. in order to improve hole finish.

8.- Rapid withdrawal to "BEGIN + Safety Distance" for drilled-out material removal.

9.- Depending on the N value (number of holes):

9.1.- Orients the spindle to current angular position + Δ

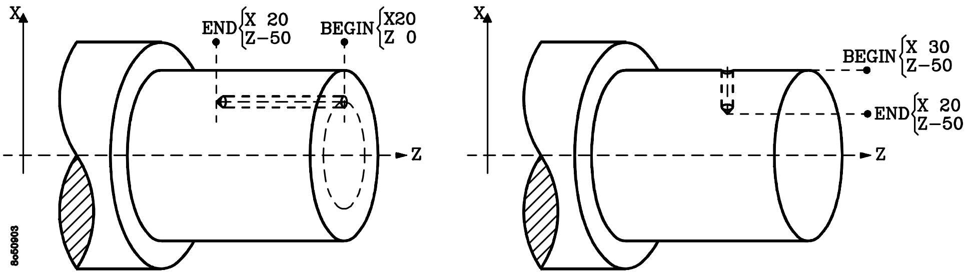
9.2.- Repeats drilling steps 6, 7 and 8.

10.- Returns to where was pressed.

11.- The CNC stops the live tool.

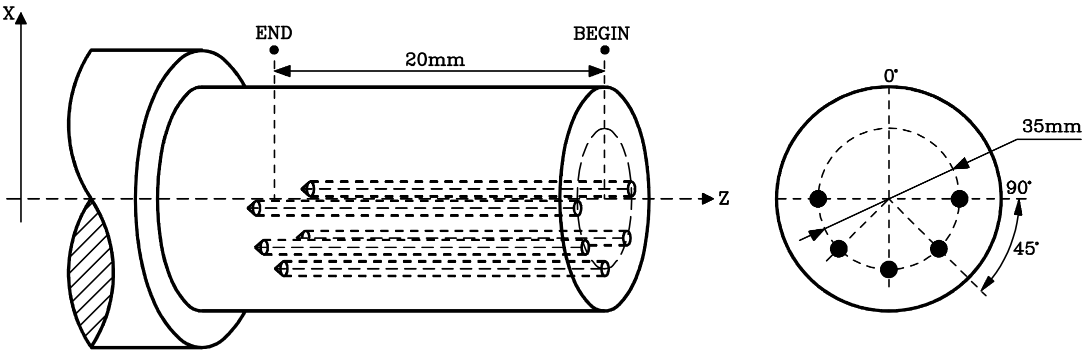
Atention:

Depending on the coordinates given to BEGIN and END, the holes will be drilled along the X axis (on the cylindrical surface of the part) or along the Z axis (on the face of the part).



In order for this cycle to run, either the X coordinates of both BEGIN and END points **or** their Z coordinates **must be the same**. Otherwise, this CNC will issue an error.

Example: Five 20mm-deep holes are to be drilled. The first one is located at 90° and the other four at 45° from each other. They are also located at a diameter of 35mm. A live tool turning at 2000 rpm is being used.



F1000 % 100 TRPM 2000 %100 T3

BEGIN X35.000 Z0
END X35.000 Z-20.000
TRPM 2000
P 5
α 90
Δ 45
N 5
Safety DistanceX =... 0.000
Safety Distance Z = . 3.000

5.10 SLOT MILLING

With this operation it is possible to mill radial slots on the face of the part (same Z coordinate, different X coordinate) as well as parallel to the part's center line on its cylindric side (same X coordinate, different Z coordinate).

To access this operation, proceed as follows:

* Press [AUX] , select option **[6]** (Other cycles) and, then, "Slot Milling".
* When in DRO mode, press [+/–] or [LEVEL +] and select the "Slot Milling" option.

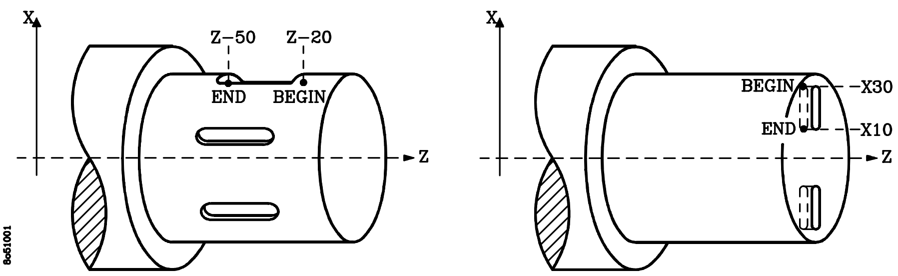
Once the new cycle has been selected, The CNC will display the following data:

The CNC shows this cycle when having spindle orientation and live tool.

Once selected, the CNC will show the following type of information:

F 1000 %100 TRPM 1500 %100 T3

Display	Meaning
F 1000	Machining feedrate (G94 m/min)
%100	Percentage of Feedrate Override
TRPM 1500	Live tool RPM
%100	Live tool speed override if P609(8) = 1
T3	Slot milling tool



The CNC will show the "BEGIN" and "END" values currently available. New ones may be selected if so desired.

The following data must also be defined:

α Angular position of first slot
Δ Angular increment between slots
N Number of slots to be milled

To set the safety distances along X and Z, press [AUX] , the CNC will display a new menu. After setting both values, press [+/–] or [LEVEL +]

To quit this operation, proceed as follows:

* Press [+/–] or [LEVEL +] to return to the "Other automatic operations" menu.
* Press [+/–] or [LEVEL +] again to return to the DRO mode.

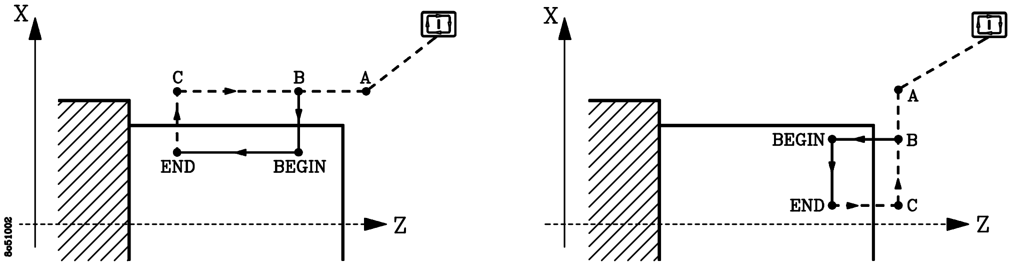
Basic operation:

1.- Does a home search on the spindle if it was in the rpm mode before.

2.- Makes a tool change if necessary.

3.- Starts the love tool at the indicated rpm (TRPM).

4.- Orients the spindle to the indicated angular position (a)



5.- Positions the axes in rapid to the approach point "A" (according to the safety distances).

6.- If the approach point is not lined up with the BEGIN point, the tool will move to point "B"..

7.- It mills the slot in the following steps:

7.1- The tool penetrates to the BEGIN point at the programmed feedrate.

7.2- It mills the slot by moving either the X axis or the Z axis accordingly up to the END point at the programmed feedrate.

7.3.- Returns in rapid to point "C".

7.4.- Returns in rápido to point "B"

8.- Depending on the value of N (Number of slots):

8.1.- Orients the spindle to the current angular position + Δ
8.2.- Repeats the movements of paragraph 7.

9.- Returns in rapid to where was pressed.

10.- The CNC stops the live tool.

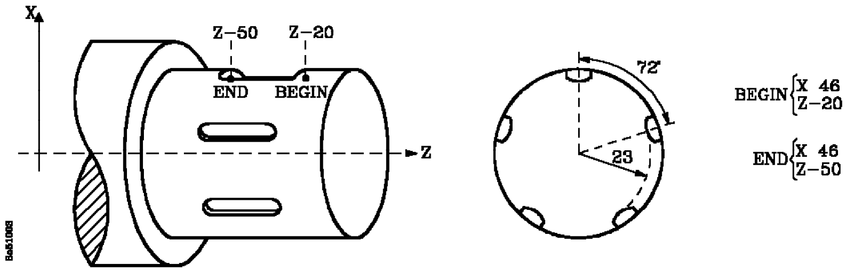
Atention:

Depending on the orientation defined with the BEGIN and END coordinates, the slots will be milled along the Z axis (on the cylindric surface of the part) or along the X axis (on the face of the part).

In order for this cycle to run, either the X coordinates of both BEGIN and END points or their Z coordinates **must be the same**. Otherwise, the CNC will issue an error.

Example in diameters: We want to mill 5 slots at 72º from eachother starting at 0º, they will be 30mm long and will be milled at a diameter of 46mm.

The live tool will turn at 1000 rpm and the milling feedrate will be F1000 (1m/min)



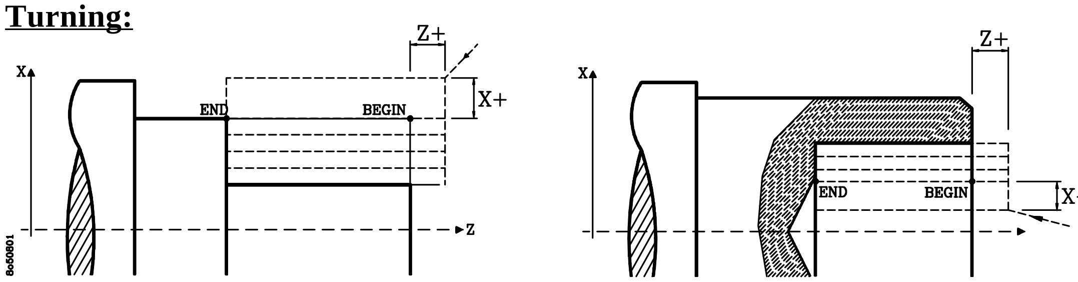
Once selected, the CNC will show the following type of information:

F 1000 %100 TRPM 1500 %100 T3		
BEGIN	X46.000	Z-20.000
END	X46.000	Z-50.000
α	0	
Δ	72	
N	5	
Safety Distance X = 5.000		
Safety Distance Z = 5.000		

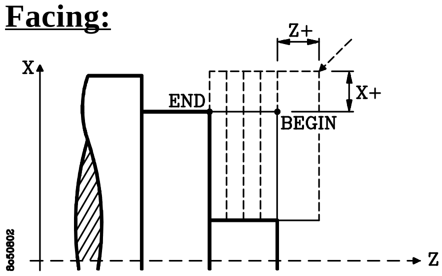
5.11 USAGE OF THE SAFETY DISTANCE

Whenever an automatic operation is executed in the automatic mode (CYCLE mode), the CNC applies the values defined as safety distance along X and Z.

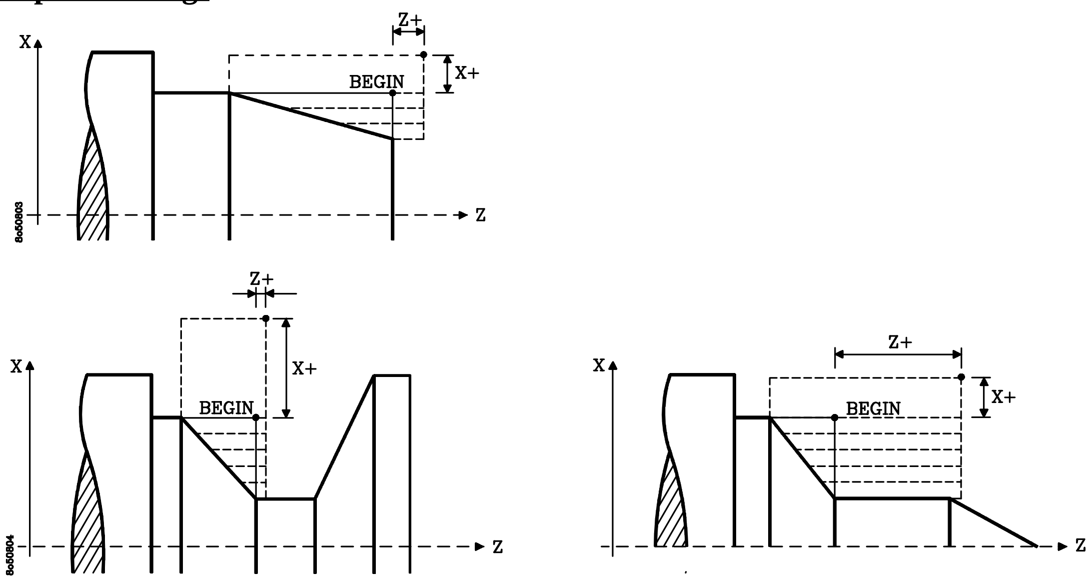
Turning:



Facing:



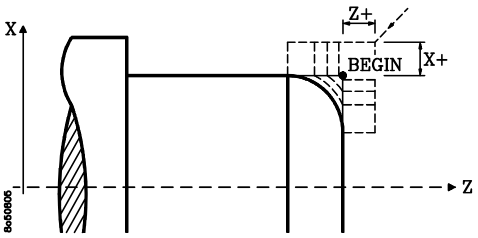
Taper turning:

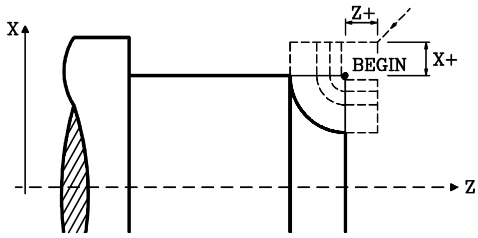


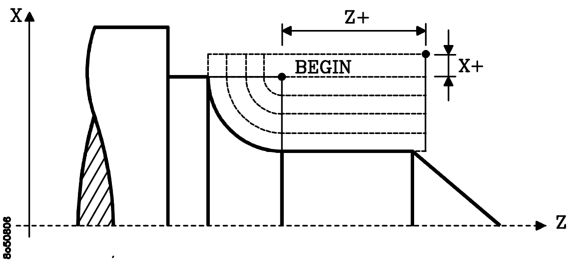
The figure on the left shows how the safety distance along X may be used so the BEGIN point of the operation is located outside the part thus preventing collisions between the tool and the part.

The figure on the right shows how the safety distance along Z may be used to carry out two machining operations in one. "Taper turning" + "Regular turning".

Rounding:

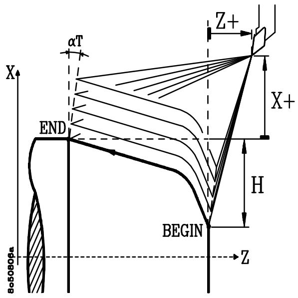


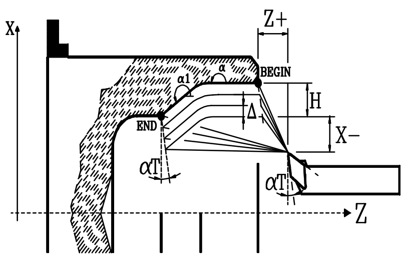




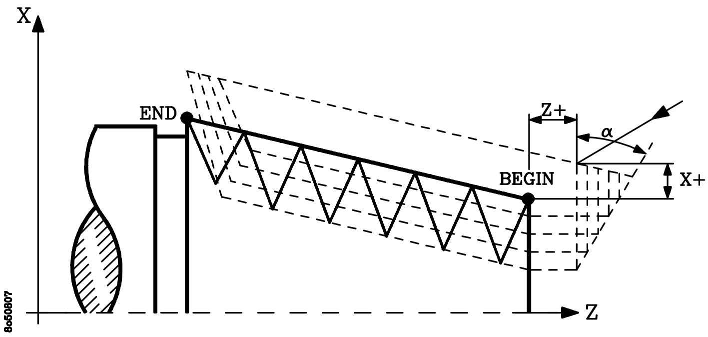
The last figure shows how the safety distance along Z may be used to carry out two machining operations in one. "Rounding" + "Turning".

Profile rounding:

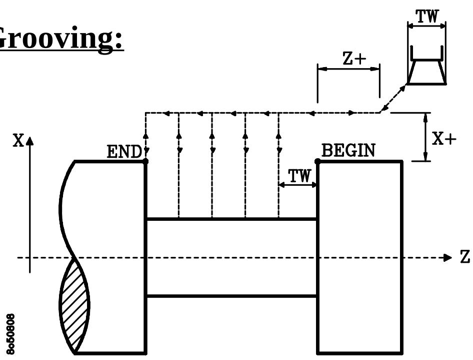




Threadcutting:



Grooving:



6. *PROFILES*

With this option it is possible to store up to 12 points (X, Z) in order to later machine that profile or make movements to any of those points.

In order to work in this mode, the lamp must be on. If not on, press this key.

When accessing this mode, the bottom of the CNC screen shows the position values (coordinates) of the first three points of the profile.

To edit or modify any of them, press **[P]**. The CNC will show the highlighted coordinates of the first point (P1).

To select another point, use the up and down arrow keys and press **[CL]** to quit the editing mode.

If while in the editing mode and being any point (for example: P10) currently selected the **[P]** key is pressed, the CNC will select the first profile point "P1" again.

It is possible to edit and execute profiles in "SEMI-AUTOMATIC" or "AUTOMATIC" mode. Press to select the desired mode.

With this CNC, it is possible to edit and execute profiles in "Semi-automatic" or in "Automatic" mode. To select the desired mode, press or LEVEL + (at the compact model).

6.1 PROFILE IN "SEMI-AUTOMATIC" MODE

In this mode it is possible to define up to 12 points and later move the axes from point to point.

The movements are carried out in a straight line between consecutive points; that is from P1 to P2, from P2 to P3 and so on.

6.1.1 POINT STORAGE

To delete the current information on all the points, press the keystroke sequence: **[CLEAR] [ENTER]**. The CNC will set all the points to X0, Z0.

The points must be set one at a time and it is recommended to start out from "P1" and continue sequentially.

Use the up and down arrow keys to select a point which will appear highlighted on the screen.

Then, to assign the desired value to it, use one of the following options:

a/ Move the axes to the desired position by means of either mechanical handwheels, electronic handwheels or the JOG keys on the operator panel.

Press **[ENTER]**.

The CNC will assign the displayed position value to this point and it will select the next point.

b/ Press **[X]**, key in the desired X value for that point and press **[ENTER]**.

The CNC will assign this value to the X axis without modifying the Z axis value.

By pressing: **[X] [ENTER]**, the CNC assigns the value of X0.

Repeat this operation for the Z axis, if so desired.

6.1.2 POINT-TO-POINT MOVEMENT

The CNC moves both axes at the same time (interpolating) from one point to the next in a straight line.

It is possible to carry out all the movements either without interruption or one movement at a time (point to point) be pressing the SINGLE key.

Before making a move, use the up and down arrow keys to select the first destination point (position), (for example P5).

Then, press and the axes will move to the indicated point.

Once this position is reached, the CNC will select the next point (for example P6) and it will be ready to move to this new point

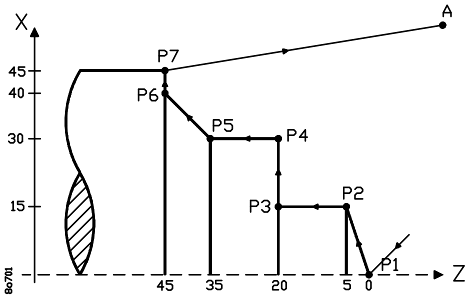
If the “SINGLE” mode is selected, the CNC will wait for the key to be pressed in order to move the axes to this new position. But, if the “SINGLE” mode is not selected, the CNC will continue executing the programmed movements until position “P12” is reached.

Atention:

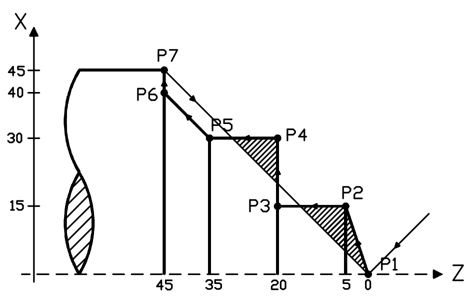
To execute the movements in a continuous fashion when not all the points are being used, it is recommended to define all the unused points with the value corresponding to the withdrawal position of the part.

Example: When using points P1 thru P7, it is recommended to set points P8 thru P12 to the value of point A. This example is programmed in diameter.

P1=X0 Z0 P2=X30 Z5 P3=X30 Z20 P4=X60 Z20
P5=X60 Z35 P6=X80 Z45 P7=X90 Z45



Points P8 thru P12 = A



Points P8 thru P12 = X0 Z0

6.1.3 *SPECIAL FEATURES*

This work mode shows, next to the coordinates corresponding to the 12 points that may define the profile, the "BEGIN" and "END" coordinates selected at the time.

It is possible to select new coordinates for the "BEGIN" and "END" points by pressing the key associated to each one of those points and assigning them the desired value. For example:

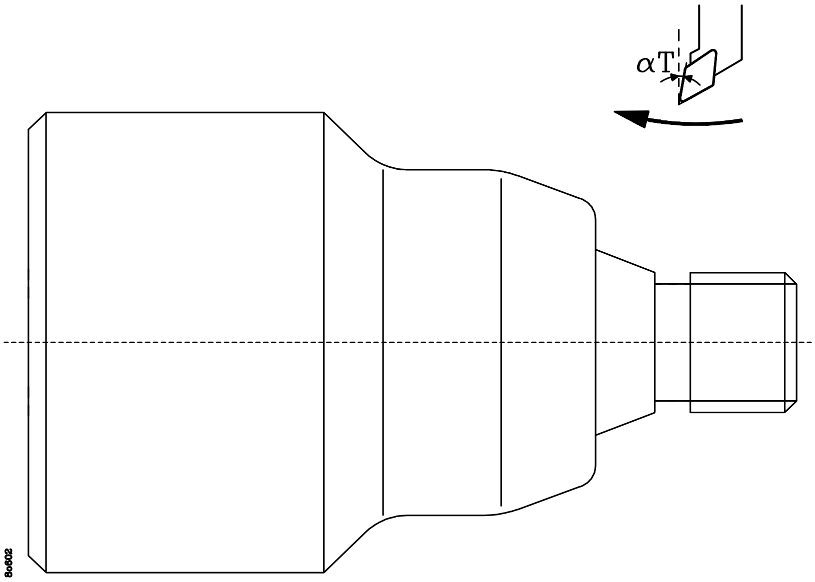
[BEGIN] [1] [0] [.] [5] [ENTER]

It is also possible to assign to the "BEGIN" and "END" points the coordinates of any point of the profile. To do this, proceed as follows:

* Select in reverse order the point whose coordinates are to be assigned to the "BEGIN" or "END" point. To do this, use the **[P]** key as well as the up and down arrow keys.
* Press **[BEGIN]** or **[END]** accordingly
* Press **[ENTER]**.

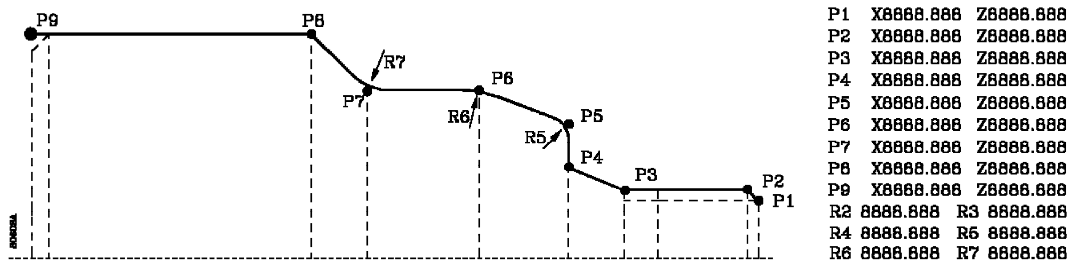
6.2 *PROFILE IN AUTOMATIC (CYCLE) MODE*

When selecting the AUTOMATIC MODE (CYCLE), it is possible to define a profile and execute it automatically.

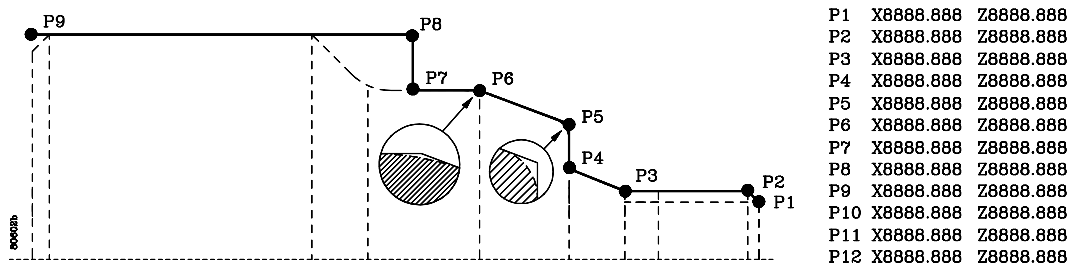


With this feature it is possible to rough parts in a single operation by defining either the profile or a path close ot it formed by straight sections.

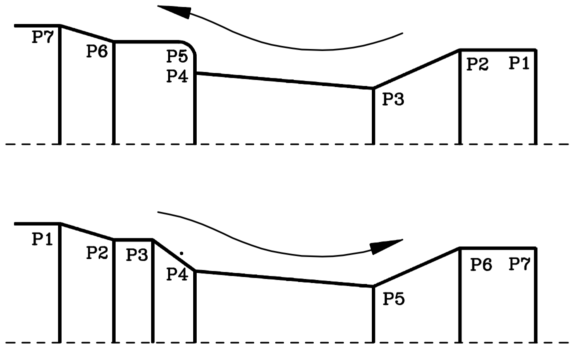
If machine parameter "P617(8)=1", up to 9 points may be defined as well as rounding associated with points: P2, P3, P4, P5, P6 and P7.



If machine parameter "P617(8)=0", the profile must consist of straight sections only and no rounding may be defined. The profile may be defined by 12 points.



Point P1 is the beginning point of the profile and it depends on the machining direction. The rest of the points must be correlative

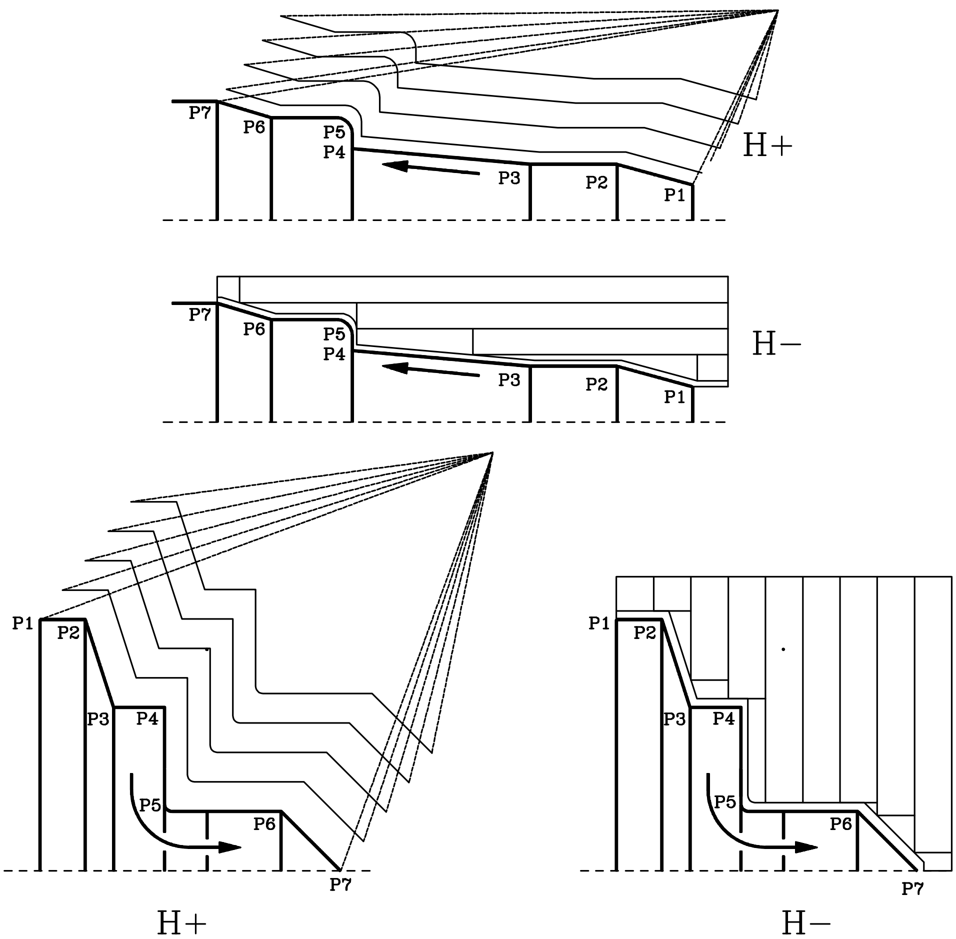


When not all the definition points are used, the unused points must be set with the value corresponding to the last point of the profile. In the example of the illustration, P12 must be = P11 = P10 = P9 = P8 = P7.

Depending on the sign assigned to parameter "H", the roughing operation will be carried out as follows:

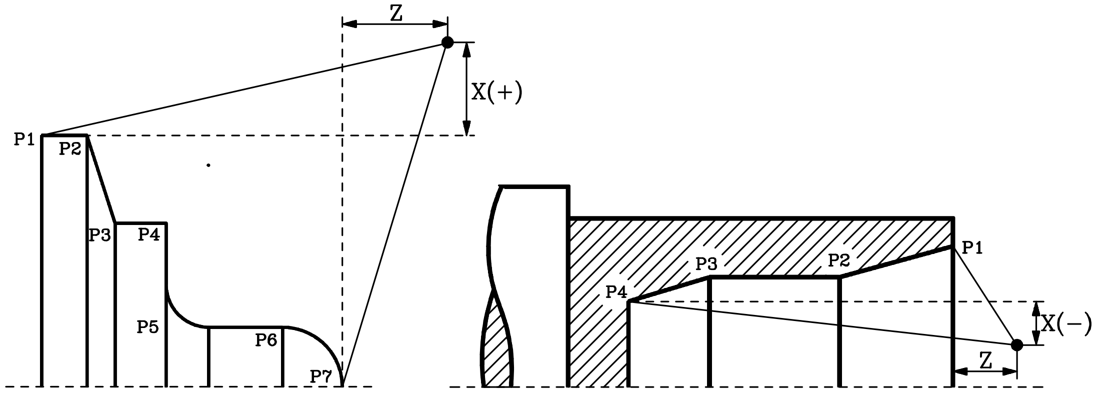
"H(+)" The successive passes will be carried out parallel to the programmed profile.

"H(-)" An initial roughing operation will be carried out, turning or facing, and a final roughing pass to maintain the finishing stock (excess material).



The profile in automatic mode has the following parameters associated to it: finishing pass, finishing feedrate, finishing tool and safety distances along X and Z. To define them, press **[AUX]** and operate as described in the section on "Cycle finishing pass and safety distance" of the chapter on "Auxiliary Functions" in this manual.

For outside machining, the safety distance along X must be positive (X+) and negative (X-) for inside machining.



It must be borne in mind that in order to obtain a proper part finish, the CNC applies the tool radius compensation while machining the profile. Therefore, it is necessary to indicate on the tool offset table the cutter radius and the location code (shape) of the tool to be used for this operation.

The information shown at the main window when selecting the automatic mode (CYCLE) is the following:

When operating in Constant Surface Speed:

F % CSS % MAX T

When **<u>not</u>** operating in Constant Surface Speed:

F % CSS % T

To change operating modes, press **[CSS]**.

The meaning of each one of these fields is:

F Currently selected axis feedrate.

% Currently applied % override to the programmed axis feedrate "F".

CSS Constant Surface Speed to execute the cycle.

RPM Spindle speed to execute the cycle.

In either operating mode (RPM or CSS), both values are the ones currently selected to execute the cycle and the "S" value shown at the main window corresponds to the actual (real) spindle speed.

To set the spindle speed, follow one of these procedures:

* Press **[S]**, key in the desired value and press **[ENTER]**.

  This value is taken as the one to be used in the operation being edited. Therefore, it does not modify the actual spindle speed nor the "S" value displayed at the main window.

* Press **[S]**, key in the desired value and press

  The CNC modifies the actual (real) spindle speed updating the "S" value displayed at the main window.

  This new values is also taken as the one to be used when executing the automatic operation being edited.

% Percentage of the programmed spindle speed "S" currently applied.

MAX Maximum spindle speed, in rpm, when operating in CSS.

Spindle turning direction to be used when executing the cycle.

To change the turning direction to be used when executing the cycle, press **[3]**. The CNC will show the new selected direction but it will not modify the actual status of the spindle.

T The tool to be used when executing the cycle.

To select the number of the tool to be used during the cycle, use one of the following methods:

* Press **[TOOL]** and, after keying in the desired number, press **[ENTER]**.

  The CNC stores the new selected tool but it will maintain the previous one active.

* Press **[TOOL]** and, after keying in the desired number, press

  The CNC selects the new tool and assumes it for the automatic operation being edited.

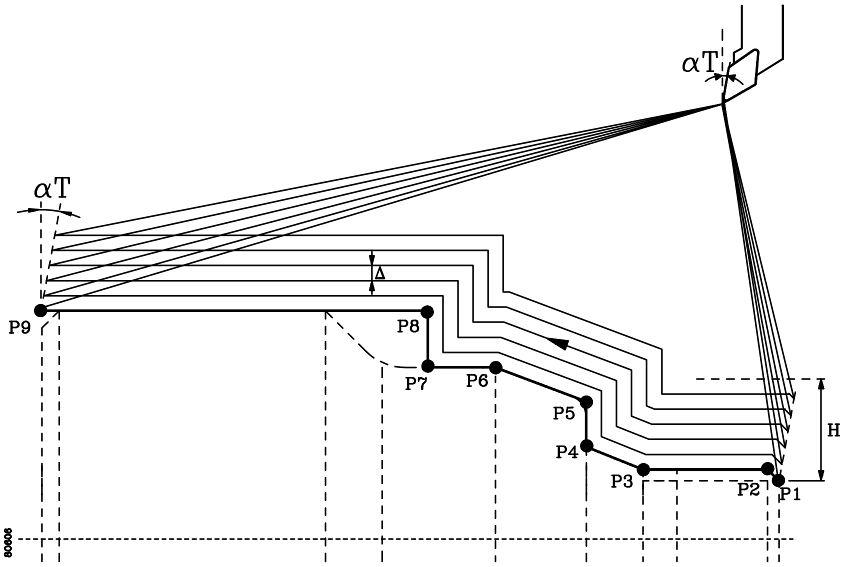
  It must be borne in mind that the CNC uses this tool for roughing and that it is possible to select another tool for the finishing pass.

Atention:

When storing an automatic operation, the CNC stores all these machining conditions together with the values and parameters defining the cycle. This way, when executing a part previously stored, the CNC will execute each one of the automatic operations with the tool, spindle turning direction, spindle speed, finishing pass, finishing tool and safety distances defined while editing it.

6.2.1 PROFILE DEFINITION

When selecting the AUTOMATIC MODE (CYCLE), the CNC show the values corresponding to the profile points currently available; being possible to select new values if so desired.



The following data must also be defined:

H Defines the amount of material to be removed from the original part. It is programmed in radius.

- If programmed with a positive value, the CNC will perform roughing passes parallel to the programmed profile.

- If programmed with a "0" value, the CNC will perform one single finishing pass (no roughing passes).

- If programmed with a negative value, the roughing pass will be a turning or facing pass. Also, if a profile is defined with "valleys", the CNC will issue the corresponding error.

Since there is no key to select it, do the following:

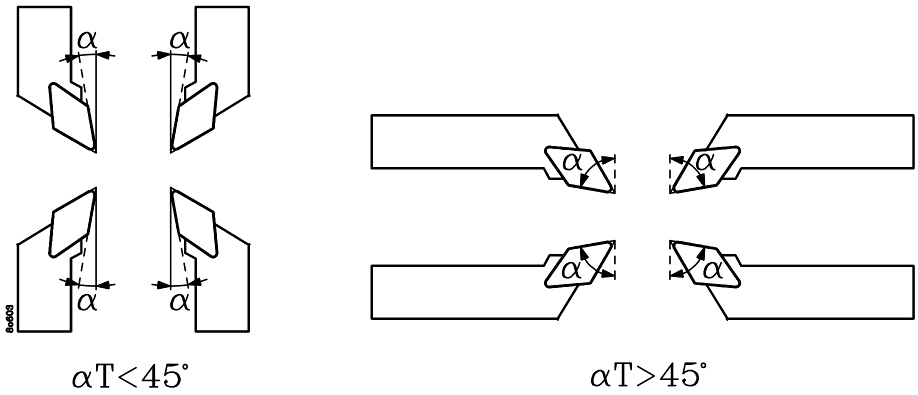
* Select another data and then use the up and down arrow keys to select parameter "H".

Δ Defines the distance between two consecutive passes.

If programmed with a value of 0, the CNC will issue the corresponding error message.

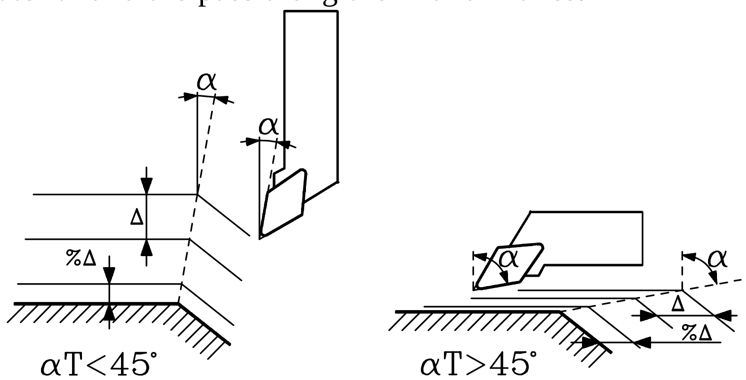
αT Defines the cutter angle indicating the angle between its edge with respect to the X axis as shown below:

To select it, press [α]

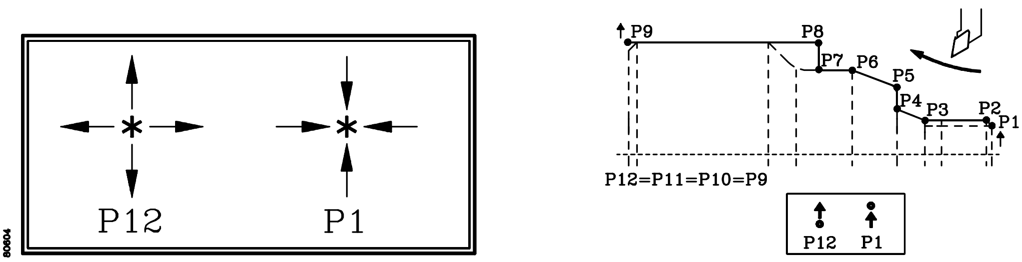


If the angle of the cutter is less or equal to 45°, the "**H**" and "**Δ**" values indicate the excess material and the pass along X. On the other hand, if the angle is greater than 45°, the "**H**" and "**Δ**" values indicate the excess material and the pass along the Z axis

Depending on the values of "**H**", "Δ" and "**αT**", the CNC always calculates the excess material and the pass along the X and Z axes.



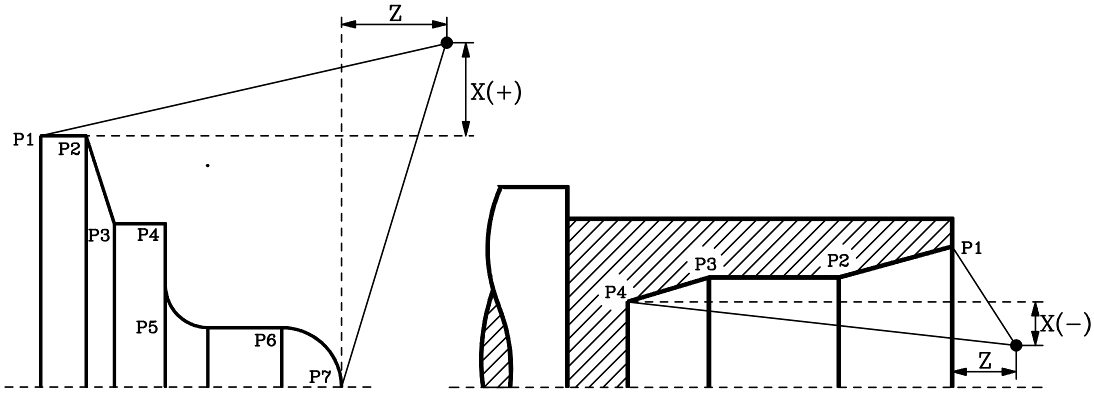
In order to obtain a good part finish, the CNC applies tool compensation. To do that, it must know how the tool enters the profile and how it exits. Those values are shown at the bottom right of the screen.



To select the way the tool enters the profile (point P1) press the up arrow key and to select the way the tool exits the profile (point P12) press the down arrow key.

When the "part program" window is being displayed, press "=" to switch to tool entry/ exit selection mode (into/from the profile).

Before starting the programmed operation, the CNC positions the tool at a safety distance X, Z from the beginning and end points.



To define the "X" and "Z" distances, press the **[AUX]** key and operate as described in the section on "Cycle finishing pass and safety distance" of the chapter on "Auxiliary function" of this manual.

When storing the "Profile" as part of a part-program, the CNC stores, as in the other machining operations, the following parameters: Finishing pass, finishing feedrate, finishing tool and the safety distances along X and Z together with the values defined by the operation.

Basic operation:

With this CNC it is possible to execute the profile from beginning to end or one pass at a time by pressing SINGLE

Once the proper data has been entered, press for the CNC to execute the profile.

Before starting to execute, the CNC will calculate the actual pass (all the passes will be the same) and the corresponding finishing pass.

The machining steps will be the following:

1.- The spindle will be started at the selected speed and in the indicated turning direction.

2.- If the cycle execution has been programmed with another tool, the CNC will make the tool change by moving to the change position if so required by the machine.

3.- The tool will be positioned at the beginning point, defined by means of the safety distances "X" and "Z".

4.- Depending on the value assigned to "H", the profile roughing operation will be performed as follows:

- If programmed with a positive value, the CNC will carry out successive roughing passes parallel to the programmed profile and maintaining the tool angle (**αT**) at the beginning point of the profile as well as when exiting it.

- If programmed with a "0" value, the CNC will perform a single finishing pass (no roughing passes).

- If programmed with a negative value, the roughing operation will be a turning or facing operation. Also, if a profile with "valleys" is defined, the CNC will issue the corresponding error.

5.- Once the roughing operation is completed, the CNC will run the finishing pass and it will end at the cycle calling point.

The feedrate for the finishing pass is set by the % of the programmed feedrate selected at the time.

6.- The CNC will stop the spindle.

Atention:

It must be borne in mind that the CNC applies tool radius compensation while machining the profile in order to obtain the proper part finish. It is necessary to indicate, on the tool offset table, the cutter tip radius value as well as the location code (tool shape) of the tool to machine with.

When only a single pass is desired, use one of these two methods:

> Program H=0

> Program Δ=H and parameters: %Δ=0 and %F=0.

7. *WORKING WITH PART-PROGRAMS*

The 800T CNC can store up to 10 part-programs.

Each of these programs may have up to 20 basic operations.
Each of these operations will have been edited by the operator in the "AUTOMATIC" mode (CYCLE) and in the manner described in the chapter about "AUTOMATIC OPERATIONS".

7.1 ACCESS TO THE PART-PROGRAM TABLE

To access this table, press [**RECALL**].

The upper right-hand side of the screen will show a directory of up to 7 part-programs of the 10 that may be stored, always numbered with 5 digits and comprised between "00000" and "99995". To see the rest, use

The dashes indicate that there is no part-program. The symbols displayed to the right of the corresponding part number mean the following:

PART	
01435	**[*]**
47632	**[*]**
32540	**[*]**
-----	**[ ]**
-----	**[ ]**
-----	**[ ]**
-----	**[ ]**
EXIT	

[*] Indicates that the part-program has been previously edited and it contains data.

[ ] Indicates that the part-program contains no data.

To assign a number to the desired part, proceed as follows:

. Position the cursor over it using the keys and knowing that the selection rolls over.

. Press [**P**]. The selected line will appear in reverse video blinking the number "**00000**".

. Press the digits of the number to be assigned and then press [**ENTER**]. If after pressing this key, this number keeps blinking, it means that this number has already been assigned to another part.

. If [CLEAR] is pressed, the selected line will reutrn to its previous number if it had one.

To quit the part-program table, position the cursor over "EXIT" and press [ENTER].

7.2 PART-PROGRAM SELECTION

To analyze the contents of a part-program in order to edit it or modify it, select it at the part-program table and press [**RECALL**].

Each part may consist of up to 20 basic operations. However, the upper right-hand side of the screen will show a set of 7 operations.

PART 01346
1 -TURNING
2 -FACING
3
THREADING
4 -
5 -
6 - ?
7 - ?
EXIT

Whenever a part-program is accessed, the cursor appears at its first free position.

The free positions are indicated by means of the "?" character and the occupied positions indicate the type of operation that has been edited in them. Although the profiles are treated as a single operations, they occupy two positions.

Each one of these operations will have been previously edited by the operator in the "AUTOMATIC" mode (CYCLE) and in the manner described in the chapter about "AUTOMATIC OPERATIONS".

To select one of these operations, position the cursor over it by using the [↑] [↓] keys.

To quit the part-program option, position the cursor over "EXIT" and press **[ENTER**].

To return to the part-program directory (previous menu), press the up arrow key until the cursor is positioned over "PART 01346" and, then, press the [↑] key once more.

7.3 PART-PROGRAM EDITIG

A part-program consists of various operations. Therefore, to edit it, we must edit its different operations.

Each operation will be edited as any normal operation and in the manner described in the section about "AUTOMATIC OPERATIONS".

However, when editing a taper turning, rounding or threading operation, the [=] [↑] or [=] [↓] keys must be used to select the corner to be machined, the type of rounding (concave or convex), the type of thread (inside or outside) or the type of profile entry/exit to be performed.

All the operations of a part-program must be edited in the "AUTOMATIC" mode (CYCLE).

Once the part-program has been selected and the operation has been defined, position the cursor over the operation number to be assigned to it and press [**ENTER**] to store it in memory.

Also, if so desired, it is possible to execute the operation before entering it in memory and, this way, check that it works properly.

When pressing [ENTER], the CNC requests confirmation of the command. The following cases are possible:

* The selected operation number was free.

  Once the command to store in memory is confirmed, the CNC will include the new operation in the indicated position and the operation listing will be updated.

* The selected operation number was occupied.

  When the CNC requests confirmation of this command, it asks whether it is desired to:

  Replace by pressing [ENTER].

  The new operation will occupy the selected position and the previous operation will disappear. The rest of the operations will keep their original positions.

  Insert by pressing [1].

  The new operation will occupy the selected position and the one which was there before as well as the following ones (including the free ones) will be shifted back one position.

  If position number 20 is already occupied, the CNC will display a message indicating that this command cannot be executed.

  Ignore (do nothing) by pressing [CLEAR].

<u>**Notes:**</u> When editing several operations of a part-program, it is recommended to start from operation "1" and use consecutive positions.

When executing a part-program, the CNC always starts from operation "1" and ends the execution when a free position is found, even when the program has other operations.

Each one of the operations of the part-program is stored in memory with all the data which was edited with.

* Data specific to the operation: BEGIN, END, angles, increment, etc.
* Machining conditions: F, S, T, spindle turning direction, etc.
* Finishing pass, finishing feedrate, finishing tool and safety distances along "X" and "Z".

7.4 PART-PROGRAM SIMULATION

With this CNC it is possible to simulate or check a part-program in dry-run before executing it.

PART 01346
1 -TURNING
2 -FACING
3 THREADING
4 - ?
5 - ?
6 - ?
7 - ?

When simulating a part, the CNC always begins with operation "1" and ends when it detects an empty position even if the part has other operations later on.

To do this, select the corresponding part (PART 01346), press [SIMUL] at the compact model or the key sequence:

[AUX] [S] at the modular model.

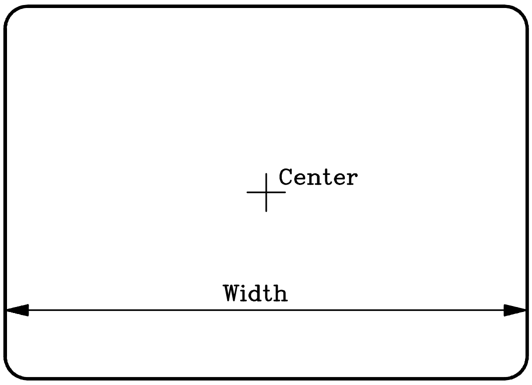
The screen shows a graphic page.

The lower left-hand side of the screen shows the axes of the plane.

To define the display area proceed as follows:

* Press [SIMUL] at the compact model and [AUX] at the modular model.
* Indicate the XZ coordinates of the position to be displayed at the center of the screen.
* Set the width of the display area.

After keying in each value, press **[ENTER]**.



To check the part, press . This will start the corresponding graphic simulation.

Press **[CLEAR]** to clear the screen, and **[END]** to quit the simulation mode.

7.4.1 ZOOM FUNCTION

With this function, it is possible to enlarge or reduce the whole graphic-representation or part of it. To do this, the simulation of the program must be either interrupted or finished.

Press **[Z]**. The screen will show a rectangle over the original drawing. This rectangle represents the new display area to be enlarged or reduced.

To change the dimensions of the rectangle, use these keys:

[+/−] Reduces the size of the rectangle (zoom in).

[=] Increases the size of the rectangle (zoom out).

Use the following keys to move the zoom window around:

At the compact model [◄] [►] [▲] [▼]
At the modular model [0] [•] [▲] [▼]

To set the area selected with the zoom window as he new display area, press **[ENTER]**.

To see he selected area enlarged or reduced while keeping the previous display area values, press [SIMUL] at the compact model and [AUX] at the modular model.

The area contained in the zoom window will now fill the whole screen.

To return to the previous display area (prior to the zoom), press **[END]**.

To use the zoom again, just press **[Z]** and proceed as before.

To quit the ZOOM function and return to the graphic representation, press **[END]**.

7.5 *PART-PROGRAM EXECUTION*

When executing a part-program, the CNC always starts from operation "1" and ends when a free position is found, even when the program has other operations.

```
PART    01346
1 -TURNING
2 -FACING
3
THREADING
4 -      ?
5 -      ?
6 -      ?
7 -      ?
```

To execute a part-program, select it by positioning the cursor over its corresponding header (PART 01346) and press

Once the part-program is selected, it is executed operation after operation starting from the first one.

Every time the CNC selects an operation, it will show it highlighted and it will make a copy into the editing area (bottom of the screen) displaying the selected operation with all its parameters.

When ending an operation, the tool is positioned at the BEGIN point of this operation maintaining the safety distances. This point is also the beginning point for the next operation.

Once the execution of each operation has finished, the tool is positioned at the BEGIN point of the executed operation maintaining the safety distances.

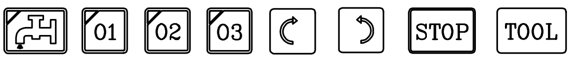
When defining the various operations, it must be borne in mind that the tool moves in a straight line from one operation to the next and from the BEGIN point of the last operation to the BEGIN point of the next one.

The part-program ends when a free position is found even if there are more operations defined in later blocks.

Once all the operations have been executed, the tool will return to the point where the execution of the part-program began.

To interrupt the program, press

Once interrupted, the following keys are enabled:



To resume execution, press

Note: It must be borne in mind that the CNC always executes what is defined in the editing area, bottom of the screen. Therefore, **the cursor must be positioned over the part's header (PART 01346) before pressing**

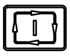
If when pressing , the cursor is positioned over one of the automatic operations, the CNC executes the operation appearing on the general screen.

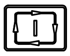
7.5.1 *EXECUTION OF A PREVIOUSLY STORED OPERATION*

To do this, select the corresponding part-program, position the cursor over the desired operation and press **[RECALL**].

The CNC recovers all the values that the operation was stored with and it shows them at the bottom of the screen.

- * Data specific to the operation: BEGIN, END, angles, increment, etc.
- * Machining conditions: F, S, T, spindle turning direction, etc.
- * Finishing pass, finishing feedrate, finishing tool and safety distances along "X" and "Z".

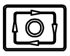
Then, press  to execute the selected operation.

It is possible to modify any data before pressing  if so desired.

7.5.2 *TOOL INSPECTION*

With this option it is possible to interrupt the execution of program P99996 and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press  to interrupt the program.

b) Press [**TOOL**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the program was interrupted.

The screen will display the following message:

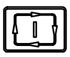
RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the program was interrupted.

e) Jog the axes to the program interruption position by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

f) Press  to resume the execution of program P99996.

7.6 PART-PROGRAM MODIFICATION

To **modify an operation,** select the corresponding part-program, position the cursor over the desired operation and press [RECALL].

The CNC recovers all the values stored with that operation and it displays them at the bottom of the screen.

From this moment, the operation may be modified as any normal operation and in the manner described in the section about "AUTOMATIC OPERATIONS".

Once all the modifications have been made, press [ENTER] to store it again in memory and verify that it works properly.

Once [ENTER] has been pressed, the CNC requests the confirmation of the command. Press [ENTER] again to confirm it (replace option).

To **delete an operation**, select the corresponding part-program, position the cursor over the desired operation and press [CLEAR].

The CNC will request confirmation of the command.

When deleting an operation, the CNC compresses the part-program shifting all the following operations one position forward.

To **insert a new operation**, follow the same procedure as for editing a part-program.

Once the operation has been defined, position the cursor over the operation number to be assigned to it and press [ENTER] to store it in memory.

The CNC requests confirmation of the command. Press [1] to insert this new one or [ENTER] to replace the current (old) one.

To **copy an already existing operation** into another position, move the cursor over the operation to be copied and press [RECALL].

The CNC recovers all the values stored with that operation and it displays them at the bottom of the screen.

Then, select the operation number where it is to be copied and press [ENTER]. The CNC will request confirmation of the command.

7.7 *PART-PROGRAM DELETION*

To delete a part-program, choose one of the following methods:

Select the desired part-program on the part-program directory and press [CLEAR] or select the desired part-program, position the cursor over its header (PART01435) and press [**CLEAR**]. In either case, the CNC will request confirmation of the command.

```
  PART
01435  [*]
47632  [*]
32540  [*]
-----  [  ]
-----  [  ]
-----  [  ]
-----  [  ]
  EXIT
```

```
 PART   01435
1 - TURNING
2 - FACING
3 -
THREADING
4 -     ?
5 -     ?
6 -     ?
7 -     ?
```

7.8 PERIPHERALS

With this CNC it is possible to communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer in order to transfer programs from and to one another. This communication may be managed either from the CNC when in the "**Peripheral mode**" or from the computer by means of FAGOR's DNC protocol in which case the CNC may be in any of its operating mode.

7.8.1 PERIPHERAL MODE

In this mode, the CNC may communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer having a standard off-the-shelf communications program.

To access this mode, select the "**Peripherals**" option of the "**Auxiliary modes**" after pressing [**AUX**].

Once this option is selected, the upper left-hand side of the CNC screen will show the following menu:

0 - RECEIVE FROM (Fagor) **FLOPPY DISK UNIT**
1 - SEND TO (Fagor) **FLOPPY DISK UNIT**
2 - RECEIVE FROM GENERAL DEVICE
3 - SEND TO GENERAL DEVICE
4 - (Fagor) **FLOPPY DISK UNIT DIRECTORY**
5 - (Fagor) **DELETE FLOPPY DISK UNIT PROGRAM**
6 - DNC ON/OFF

In order to use any of these options, the DNC mode must be **inactive.** If it is active (the upper right-hand side of the screen shows: **DNC**), press [**6**] (DNC ON/OFF) to deactivate it (the **DNC** letters disappear).

With options 0, 1, 2 and 3 it is possible to transfer machine parameters, the decoded M function table and the leadscrew error compensation table to a peripheral device.

The lower right-hand side of the CNC screen will show a directory of up to 7 part-programs of the 10 that may be stored. To see the rest of them, use

To do this, key in the desired number when the CNC requests the number of the program to be transferred and press [**ENTER**].

P00000 to P99990	Corresponding to part-programs
P99994 to P99996	Special ISO-coded user program
P99997	**For internal use and CANNOT be transmitted back and forth**
P99998	Used to associate texts to PLC messages
P99999	Machine parameters and tables

Important note: The part-programs cannot be edited at the peripheral device or computer.

The CRT will show the message: "**RECEIVING**" or "**SENDING**" during the program transfer and the message: "**PROGRAM NUM. P23256** (for example) **RECEIVED**" or "**SENT**" when the transmission is completed.

When the transmission is not correct, it will display the message: "Transmission error" and when the data received by the CNC is not recognized (different format) by the CNC, it will issue the message: "Incorrect data received".

The CNC memory must be unlocked in order to perform any data transmission; if not so, the CNC will return to the menu of the peripheral mode.

When transmitting from a peripheral device other than a FAGOR Floppy Disk Unit, the following aspects must be considered:

* The program must begin with a "NULL" character (ASCII 00) followed by "%" "program number" (for example %23256) and a "LINE FEED" character (LF).

* Blank spaces, the carriage-return key and the "+" sign are ignored.

* The program must end with either 20 "NULL" characters (ASCII 00) or with one "ESCAPE" character or with one "EOT" character.

* Press [**CL**] to cancel the transmission. The CNC will issue the message: PROCESS ABORTED".

FLOPPY DISK UNIT DIRECTORY

This option displays the programs stored on the disk inserted in the FAGOR Floppy Disk Unit and the number of characters (size) of each one of them.

It also shows the number of free characters available (free memory space) on the tape.

DELETE FLOPPY DISK UNIT PROGRAM

With this option it is possible to delete a program contained at the FAGOR Floppy Disk Unit.

The CNC requests the number of the program to be deleted. After keying in the desired number, press [**ENTER**].

Once the program has been deleted, the CNC will display the message: "PROGRAM NUM: P____ DELETED".

It also shows the number of free characters on the disk (free memory space).

7.8.2 DNC COMMUNICATIONS

To be able to use this feature, the DNC communication must be active (the upper right-hand side of the screen shows: DNC). To do this the corresponding parameters must be set accordingly by the manufacturer and option [**6**] of the "**Peripherals**" mode selected if it was not active.

Once active and by using the **FAGORDNC** application software supplied, upon request, in floppy disks it is possible to perform the following operations from the computer:

. Obtain the CNC's part-program directory.
. Transfer part-programs and tables from and to the CNC.
. Delete part-programs at the CNC.
. Certain remote control of the machine.

Note: At the CNC any operating mode may be selected.

7.9 LOCK/UNLOCK

With this option it is possible to lock/unlock the part-program memory in order to protect them against accidental manipulation.

To select this option, press **[AUX]** and, after selecting "Auxiliary Modes", press the key corresponding to "LOCK/UNLOCK".

The codes used to do this are:

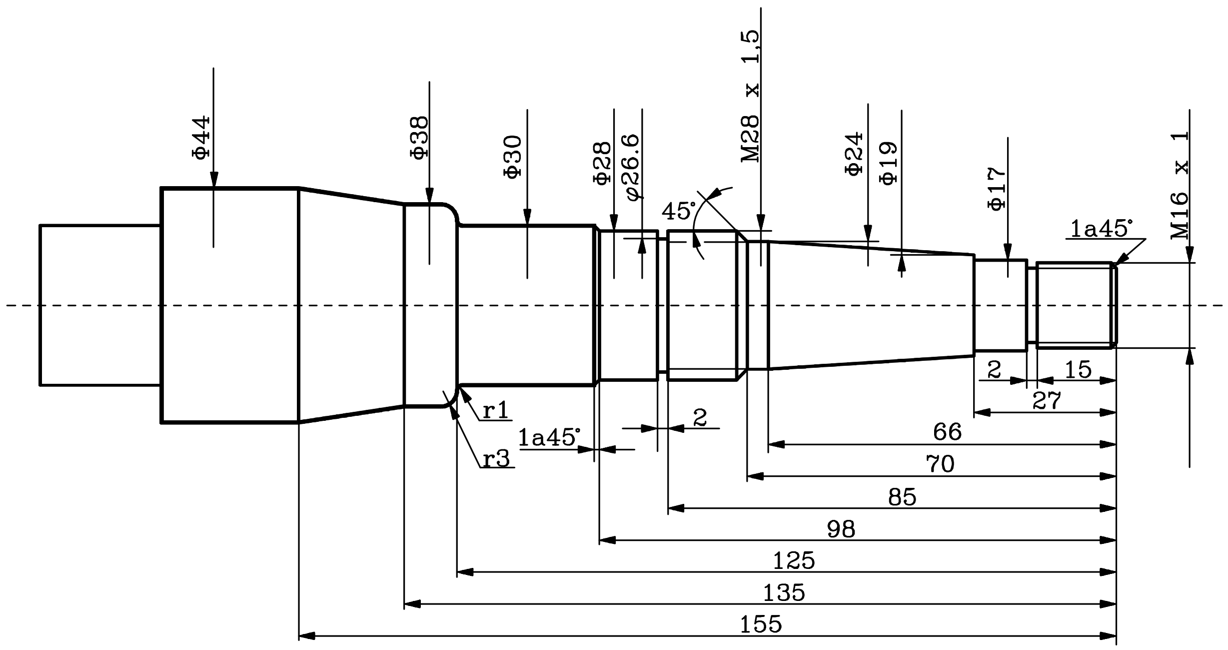
N0000 [ENTER] Unlocks part-program memory.

N1111 [ENTER] Locks part-program memory.

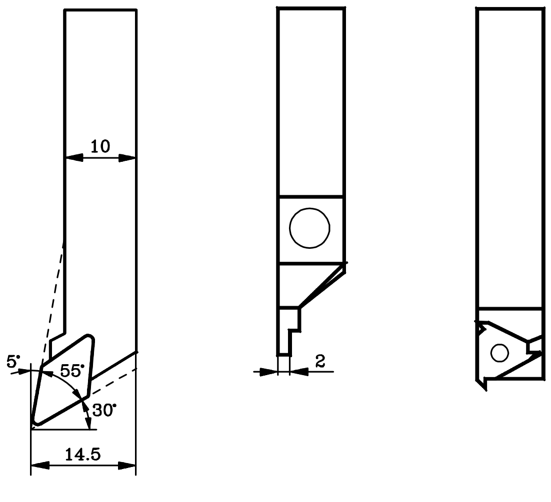
PF000 [ENTER] Erases the contents of all arithmetic parameters (for automatic operations) setting them to "0".

8. *PROGRAMMING EXAMPLES*

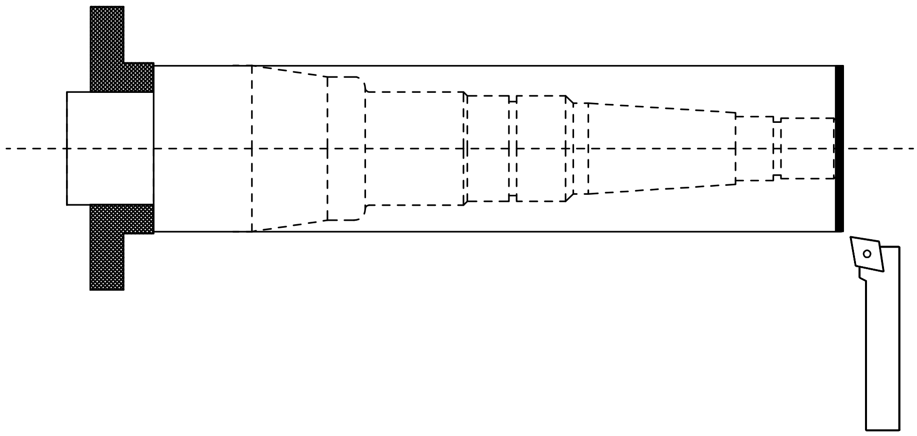
EXAMPLE 1

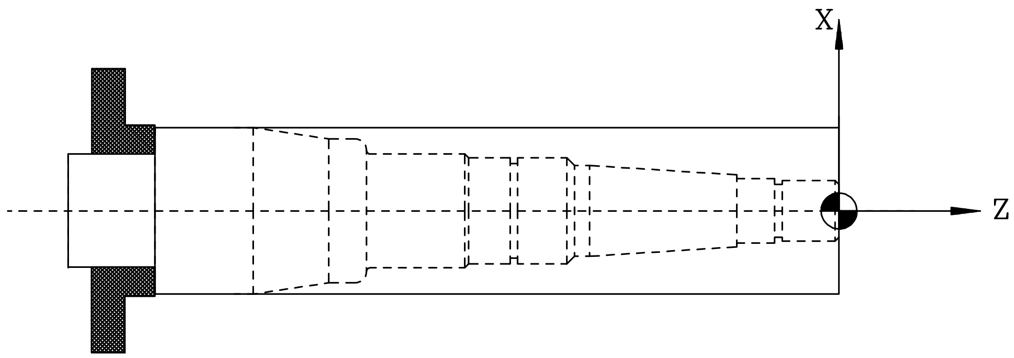


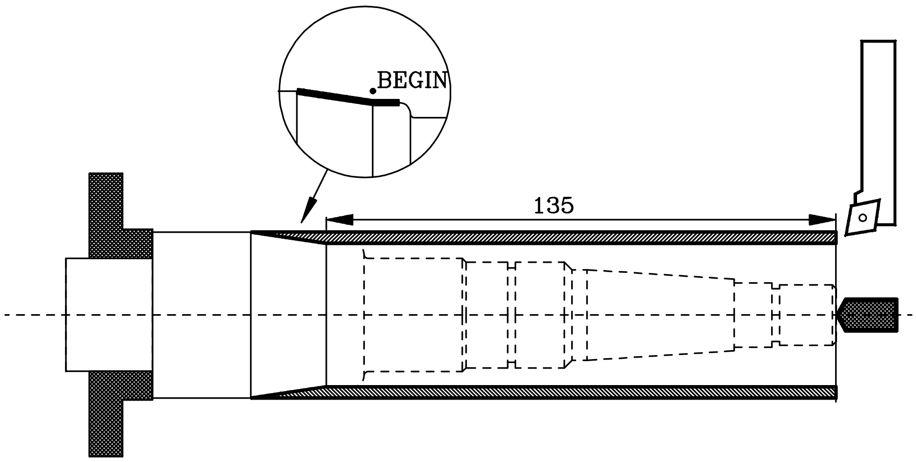
Φ44
Φ38
Φ30
Φ28
φ26.6
45°
M28 x 1,5
Φ24
Φ19
Φ17
1a45°
M16 x 1
r1
r3
1a45°
2
2
15
27
66
70
85
98
125
135
155



10
5°
55°
30°
14.5
2
T1
T2
T3



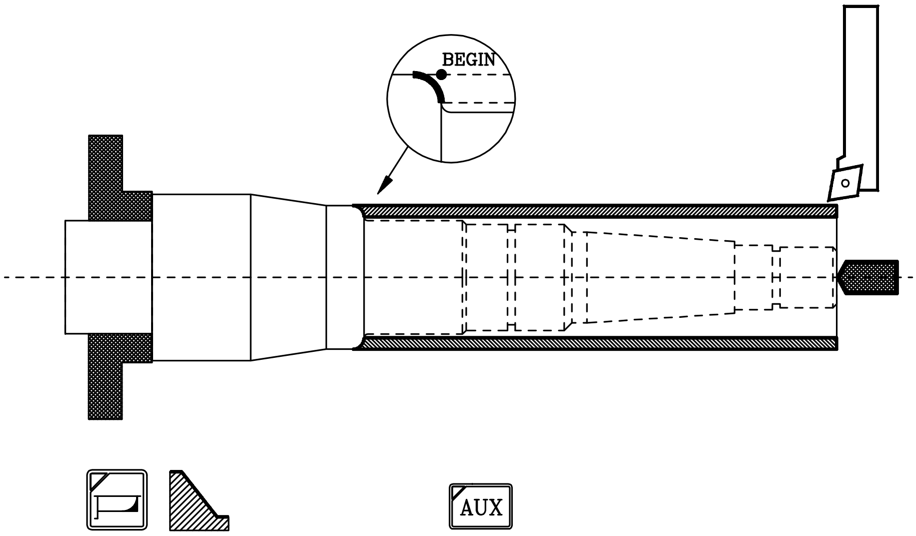




BEGIN: X44 Z−135
Φ 38
Δ
N
% 15
T 1
F
CSS

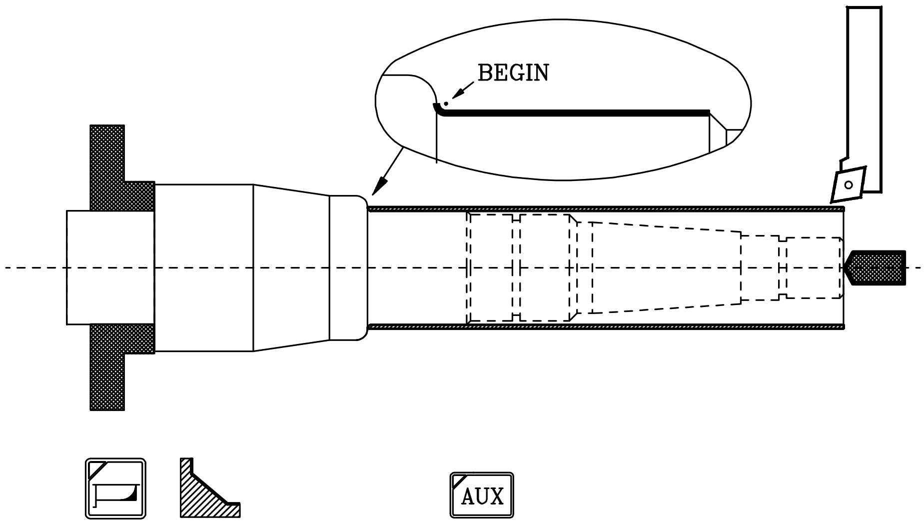
%Δ
%F
X 1
Z 136 (135+1)
T

$$\% = \frac{(44-38)/2}{155-135}$$



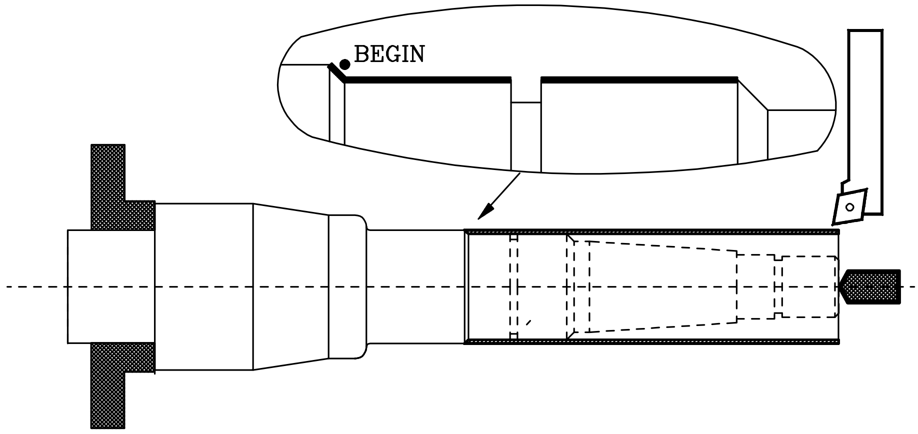
BEGIN: X38 Z−125
R 3
Δ
N
T 1
F
CSS

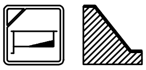
%Δ
%F
X 1
Z 126
T



BEGIN: X32 Z−124
R 1
Δ
N
T 1
F
CSS

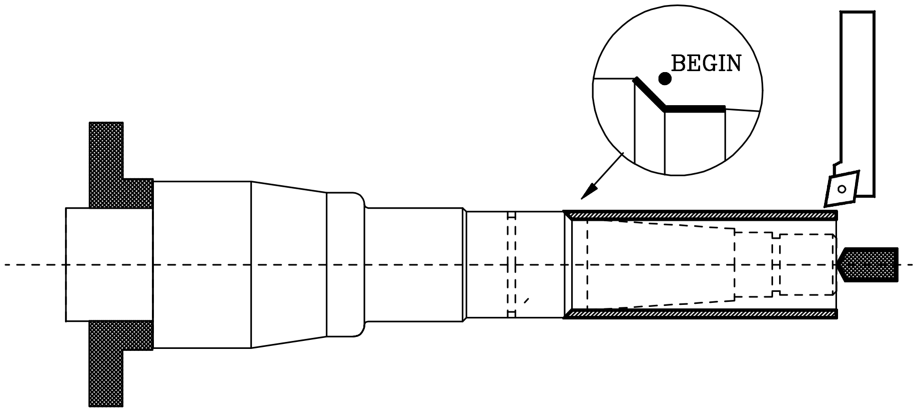
%Δ
%F
X 1
Z 126
T

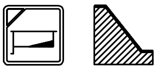




BEGIN: X30 Z−98
Φ 28
Δ
N
α 45°
T 1
F
CSS

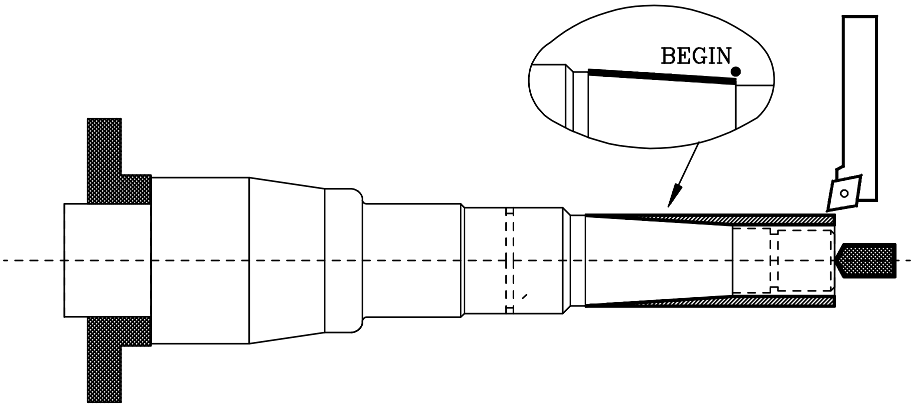
%Δ
%F
X 1
Z 99
T

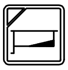




BEGIN: X28 Z−70
Φ 24
Δ
N
α 45°
T 1
F
CSS

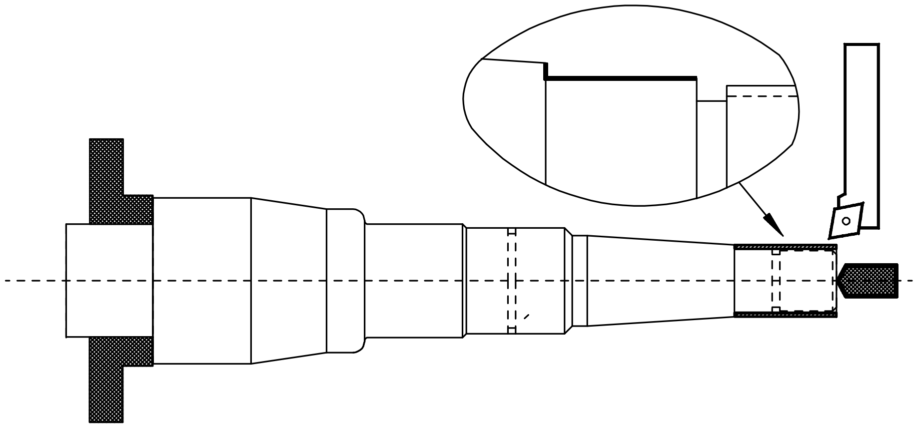
%Δ
%F
X 1
Z 71
T

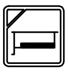


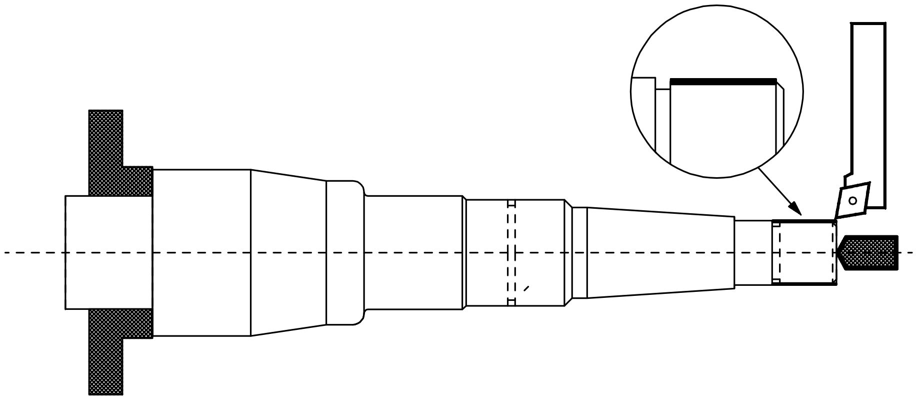
BEGIN:	X24 Z−27	%Δ	
Φ	19	%F	
Δ		X	1
N		Z	28
%	6.41	T	
T	1		
F			
CSS			

$$\% = \frac{(24-19)/2}{66-27}$$

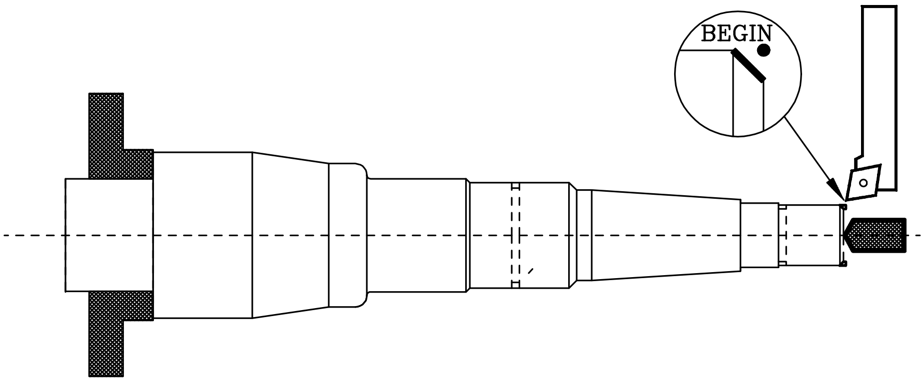


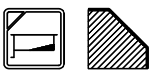
 

BEGIN:	X19 Z0	%Δ	
END:	X19 Z−27	%F	
Φ	17	X	1
Δ		Z	1
N		T	
T	1		
F			
CSS			

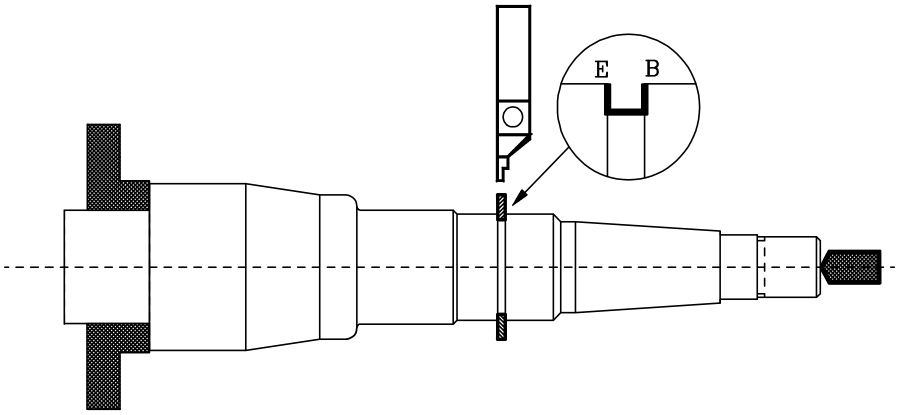


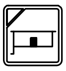
BEGIN:	X17 Z0	%Δ	
END:	X17 Z−17	%F	
Φ	16	X	1
Δ		Z	1
N			
T	1		
F			
CSS			





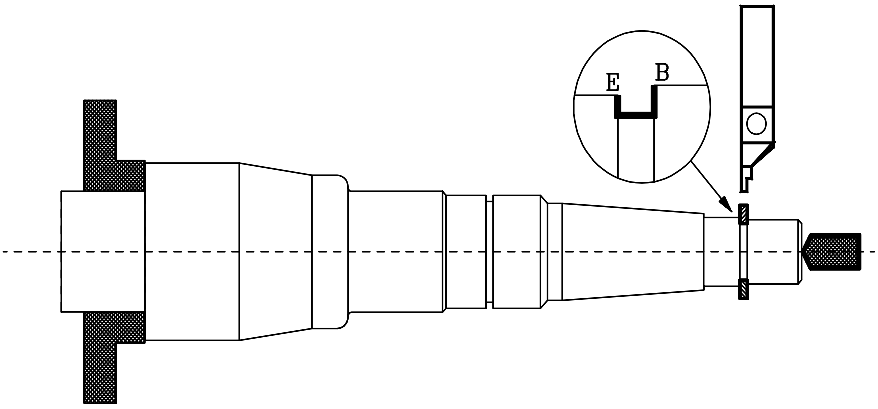
BEGIN:	X16 Z0	%Δ	
Φ	14	%F	
Δ		X	13
N		Z	1
α	45°	T	
T	1		
F			
CSS			





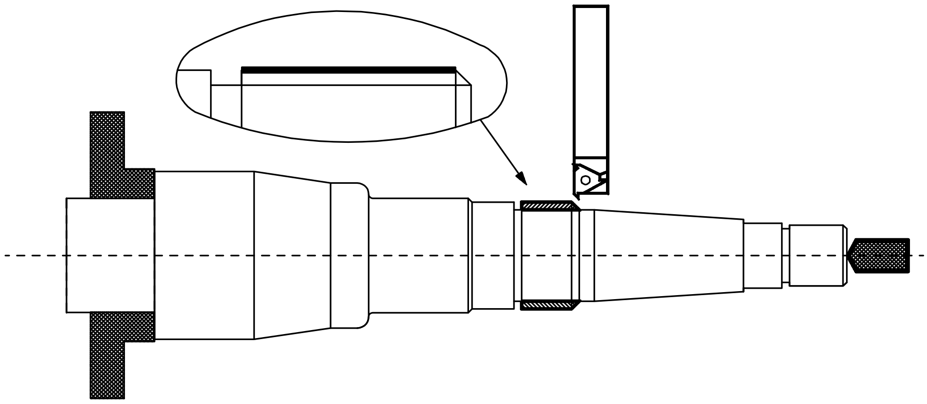
BEGIN: X28 Z-85
END: X28 Z-87
Φ 26.5
N 1
TW 2
T 2
F
CSS

%Δ
%F
X 8
Z 0
T



BEGIN: X16 Z-15
END: X17 Z-17
Φ 14
N 1
TW 2
T 2
F
CSS

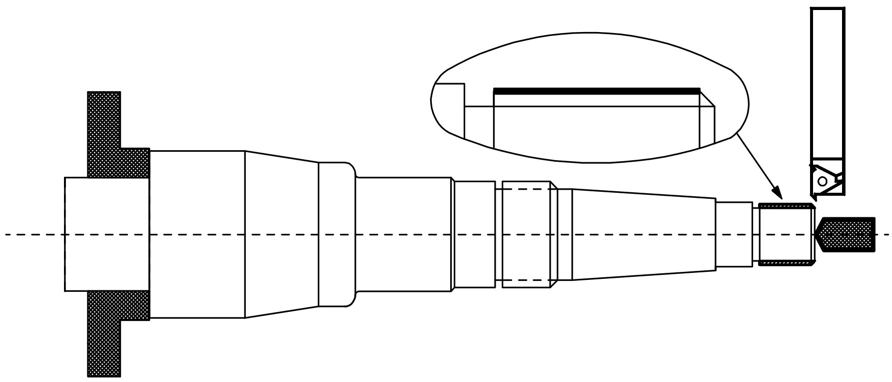
%Δ
%F
X 13
Z 0
T



AUX

BEGIN: X28 Z−70
END: X28 Z−86
P 1.5
H 1.5x0.613=0.919
Δ
α
T 3
F
CSS

%Δ
%F
X 8
Z 1

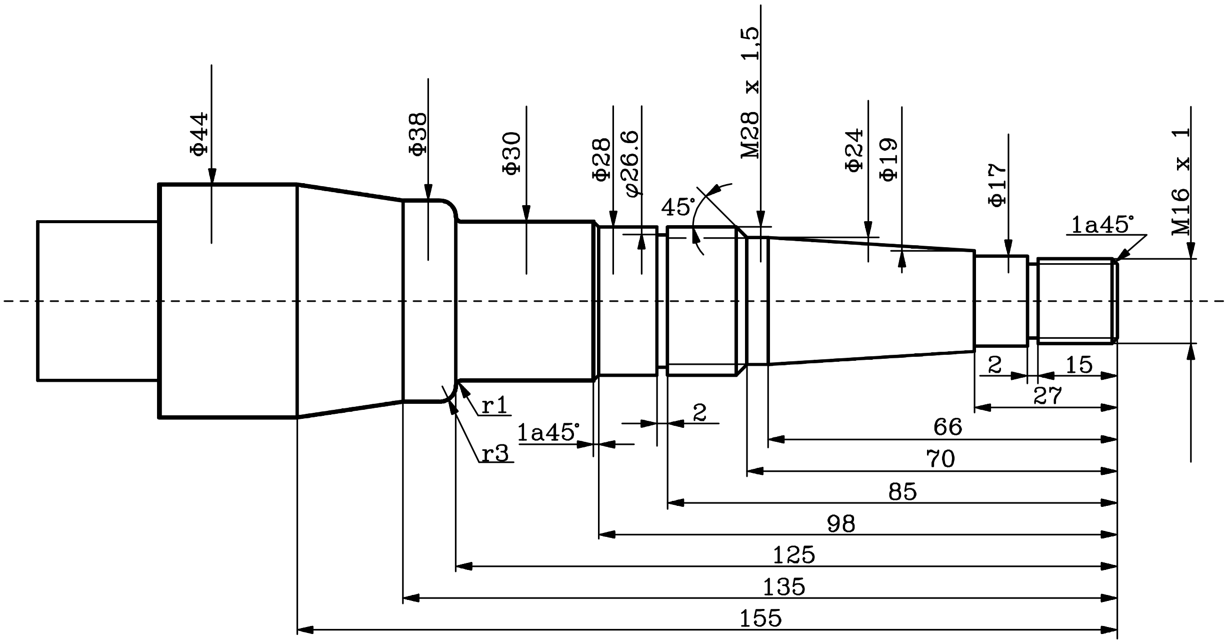


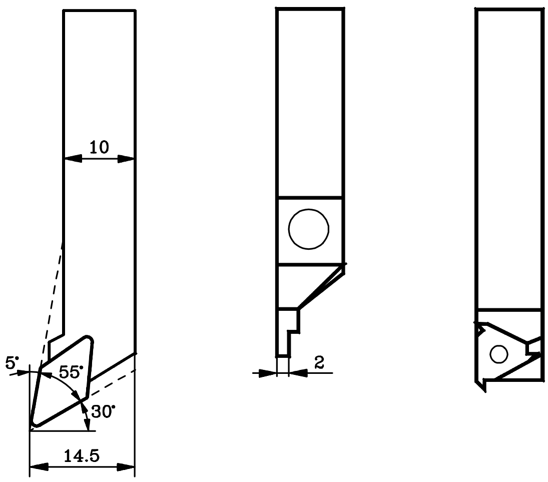
AUX

BEGIN: X16 Z0
END: X16 Z−16
P 1
H 0.613
Δ
α
T 3
F
CSS

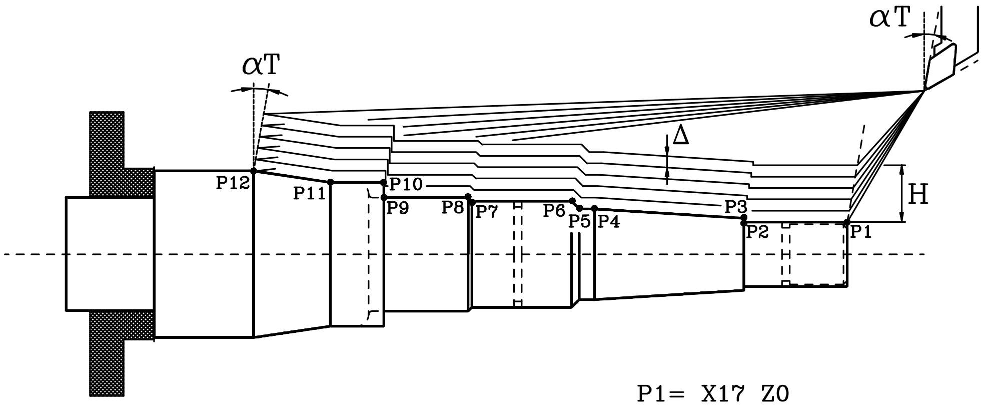
%Δ
%F
X 15
Z 0

EXAMPLE 2

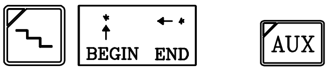




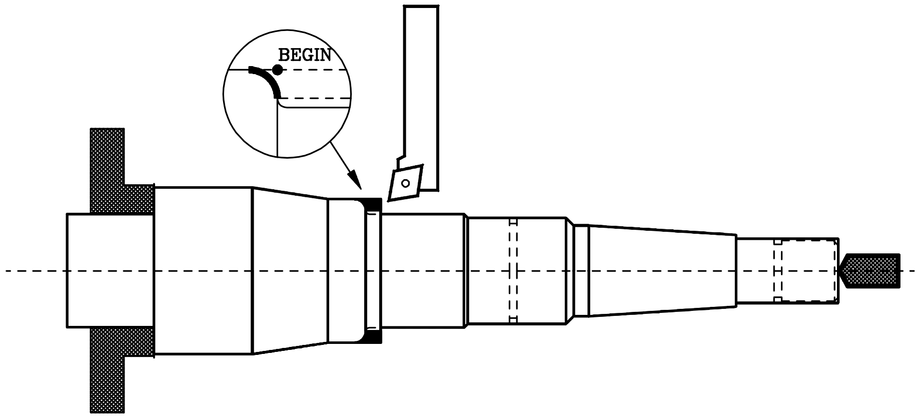
T1 T2 T3



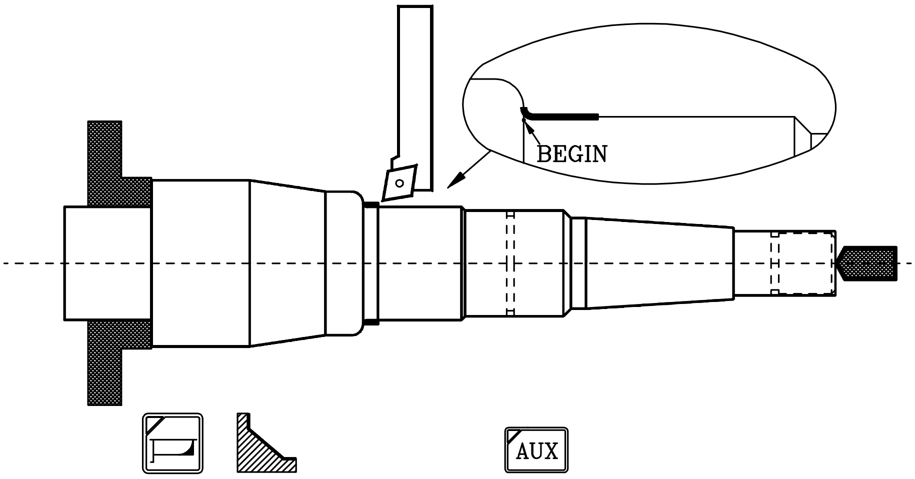
P1= X17 Z0
P2= X17 Z−27
P3= X19 Z−27
P4= X24 Z−66
P5= X24 Z−70
P6= X28 Z−72
P7= X28 Z−98
P8= X30 Z−99
P9= X30 Z−115
P10=X38 Z−115
P11=X38 Z−135
P12=X44 Z−155



H	15	%Δ	
Δ		%F	
αT	5°	X	40
T	1	Z	20
F		T	
CSS			

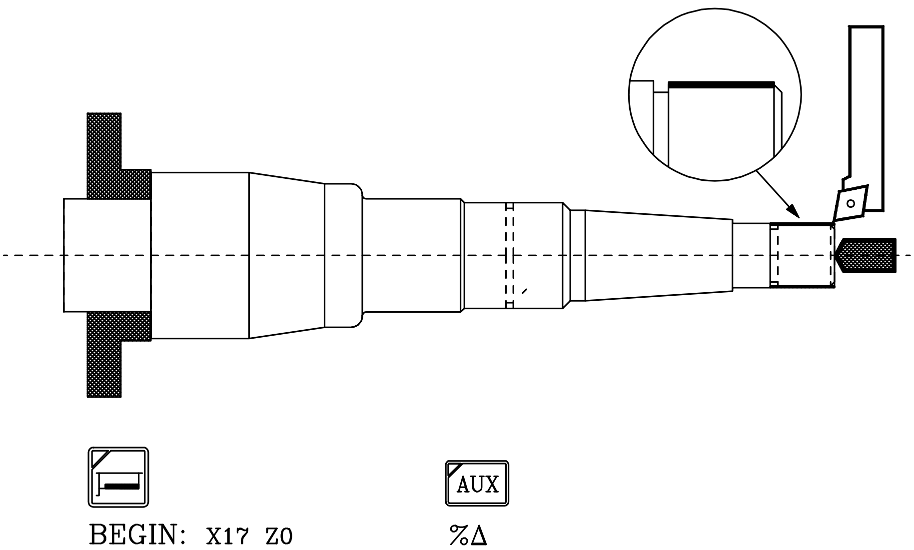


BEGIN:	X38 Z−125	%Δ	
R	3	%F	
Δ		X	1
N		Z	11
T	1	T	
F			
CSS			



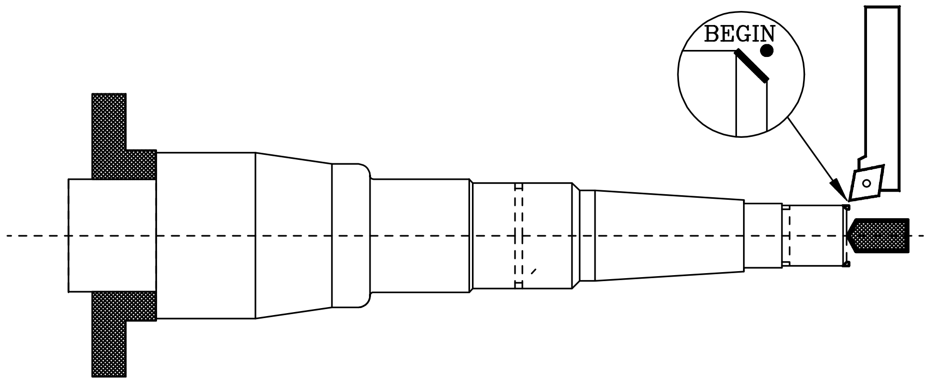
BEGIN: X30 Z−125
R 1
Δ
N
T 1
F
CSS

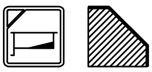
%Δ
%F
X 1
Z 11
T



BEGIN: X17 Z0
END: X17 Z−17
Φ 16
Δ
N
T 1
F
CSS

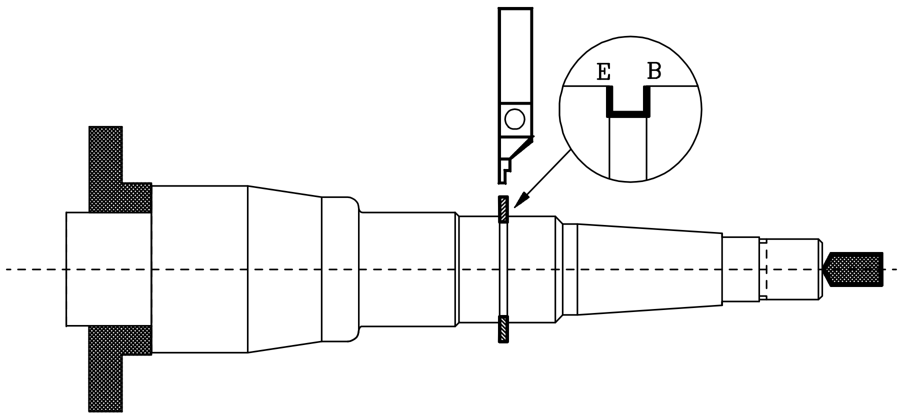
%Δ
%F
X 1
Z 1





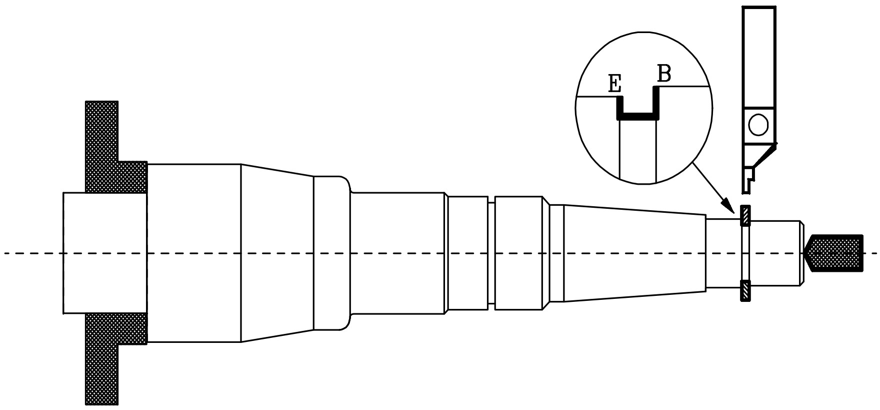
BEGIN: X16 Z0
Φ 14
Δ
N
α 45°
T 1
F
CSS

%Δ
%F
X 13
Z 1
T

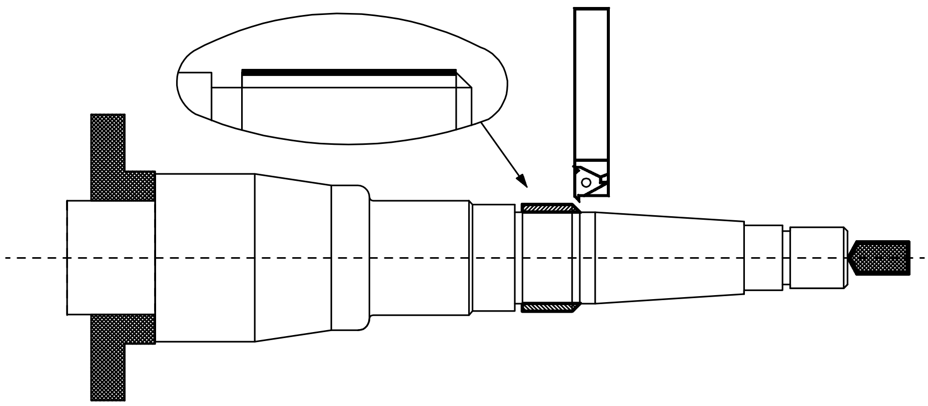


BEGIN: X28 Z-85
END: X28 Z-87
Φ 26.5
N 1
TW 2
T 2
F
CSS

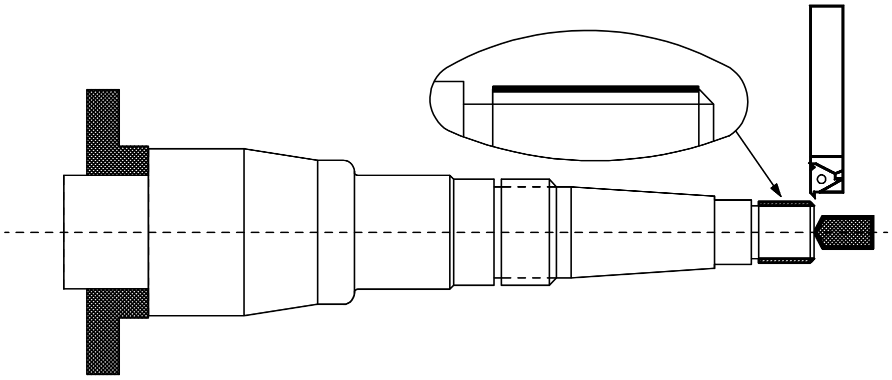
%Δ
%F
X 8
Z 0
T



BEGIN:	X16 Z−15	%Δ	
END:	X17 Z−17	%F	
Φ	14	X	13
N	1	Z	0
TW	2	T	
T	2		
F			
CSS			

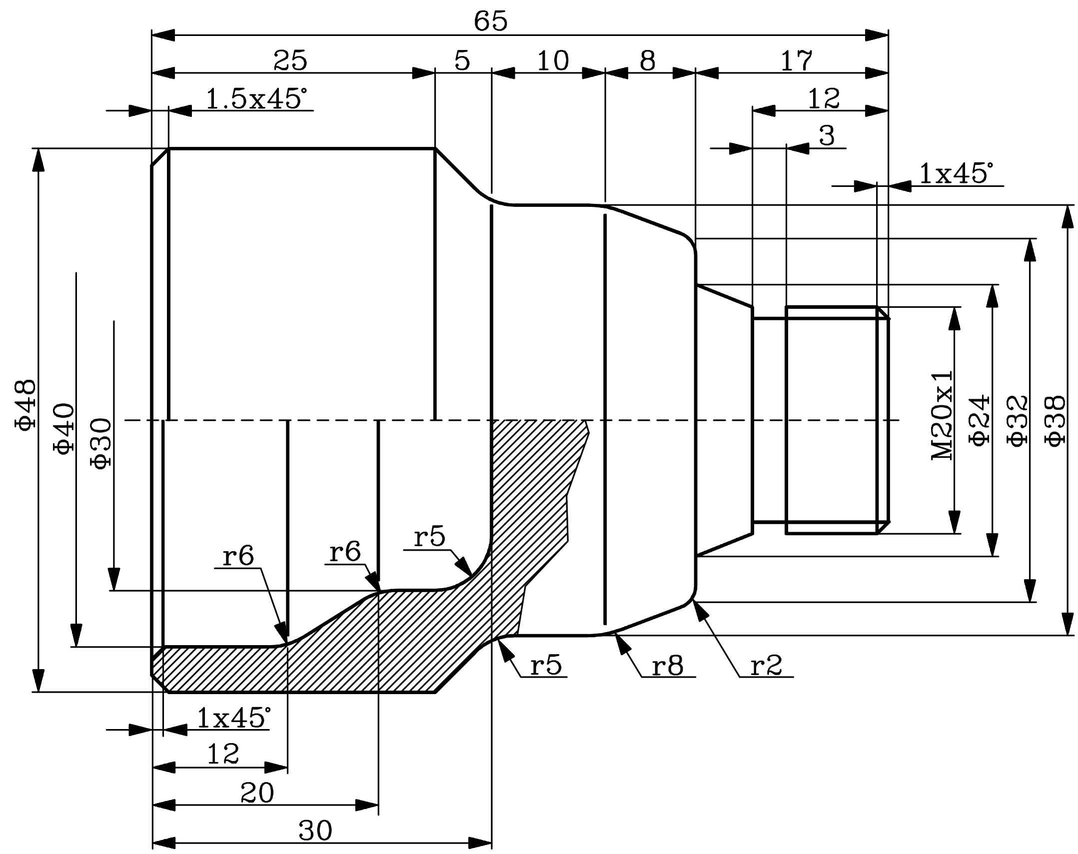


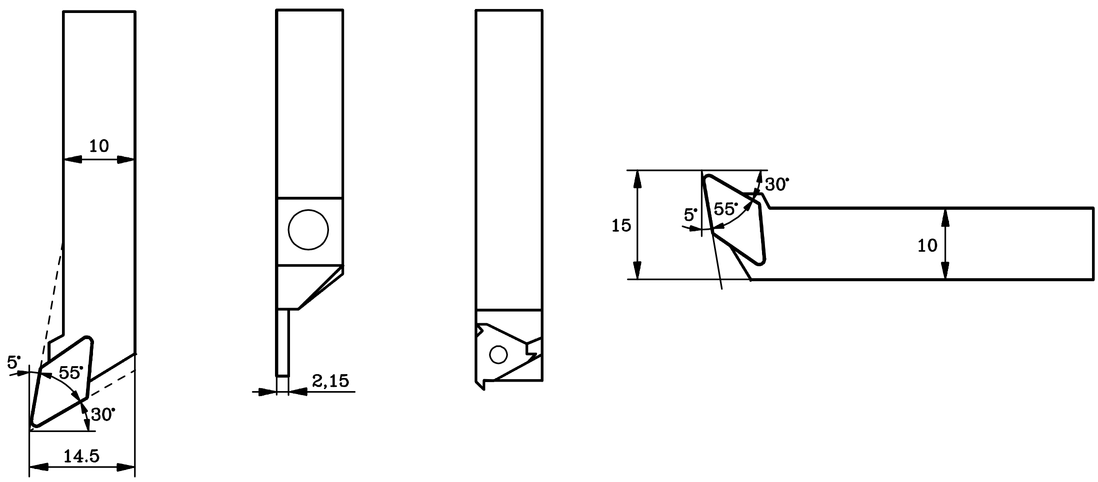
BEGIN:	X28 Z−70	%Δ	
END:	X28 Z−86	%F	
P	1.5	X	8
H	1.5x0.613=0.919	Z	1
Δ			
α			
T	3		
F			
CSS			



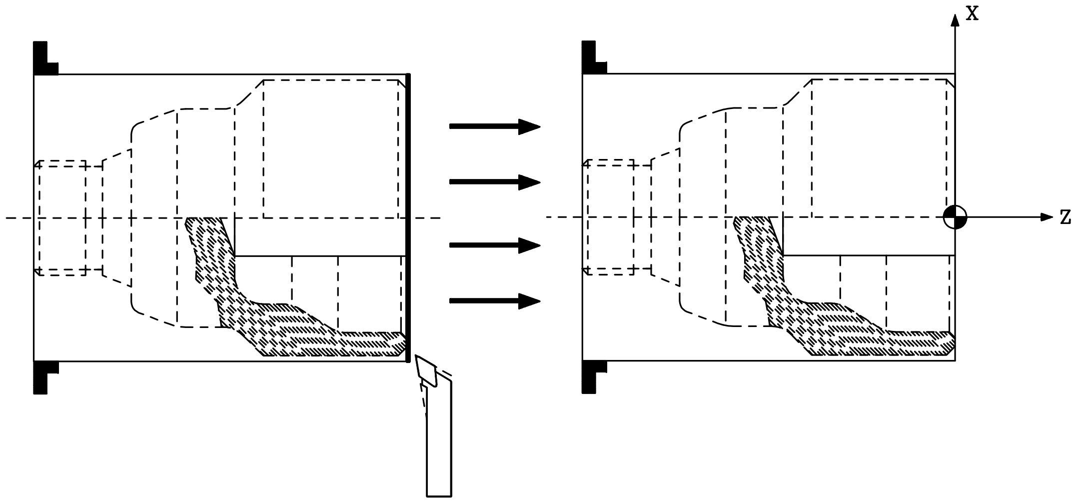
BEGIN:	X16 Z0	%Δ	
END:	X16 Z−16	%F	
P	1	X	15
H	0.613	Z	0
Δ			
α			
T	3		
F			
CSS			

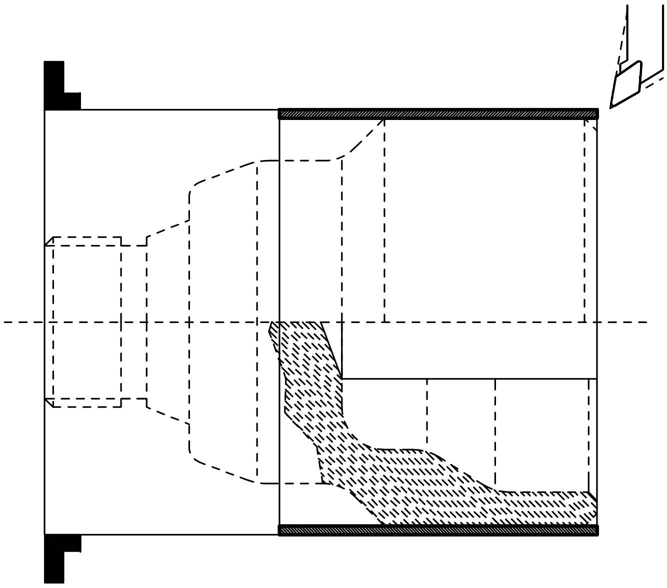
EXAMPLE 3

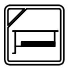




T1 T2 T3 T4

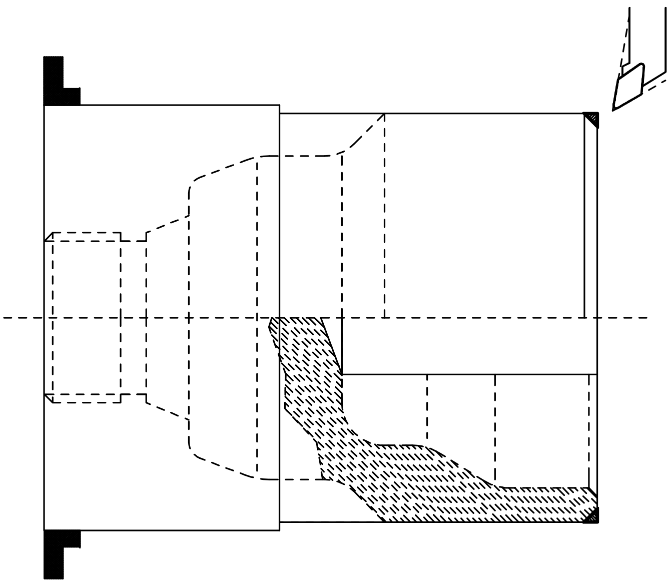


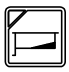


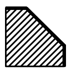


BEGIN: X50 Z0
END: X50 Z−38
Φ 48
Δ
N
T 1
F
CSS

%Δ
%F
X 1
Z 1
T

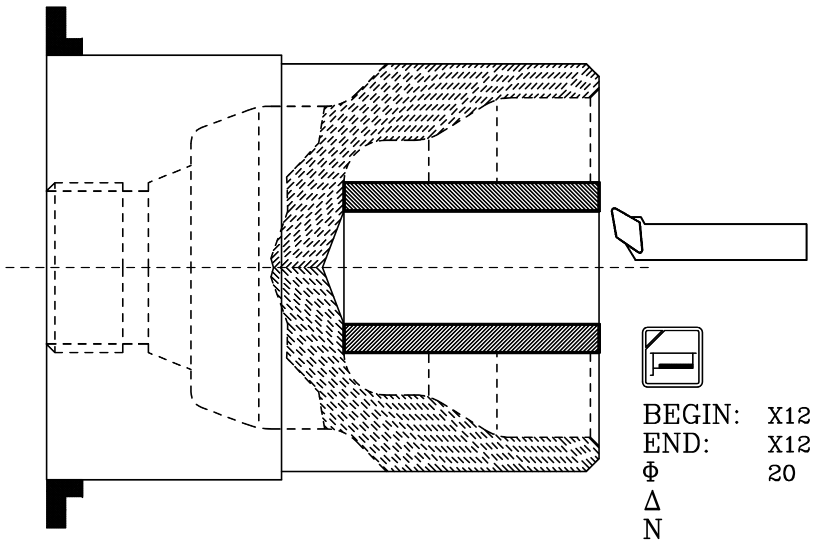






BEGIN: X48 Z0
Φ 46
Δ
N
α 45°
T 1
F
CSS

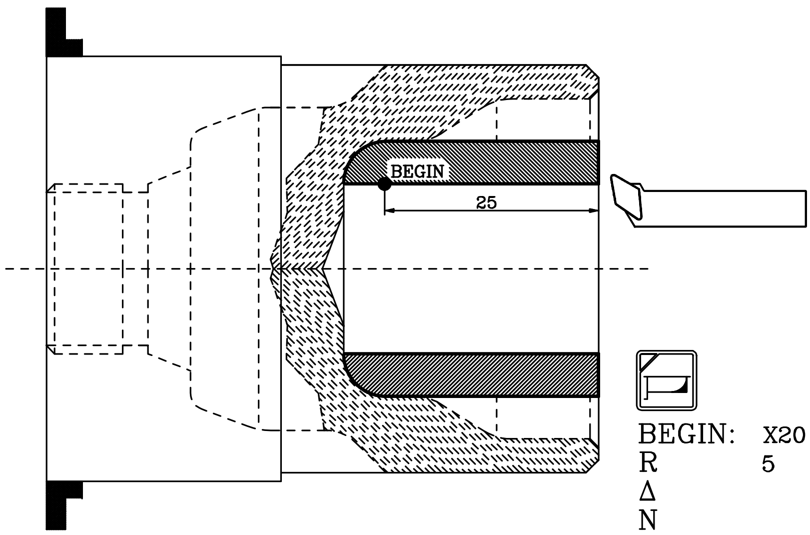
%Δ
%F
X 1
Z 1
T



BEGIN: X12 Z0
END: X12 Z−30
Φ 20
Δ
N
T 4
F
CSS

AUX

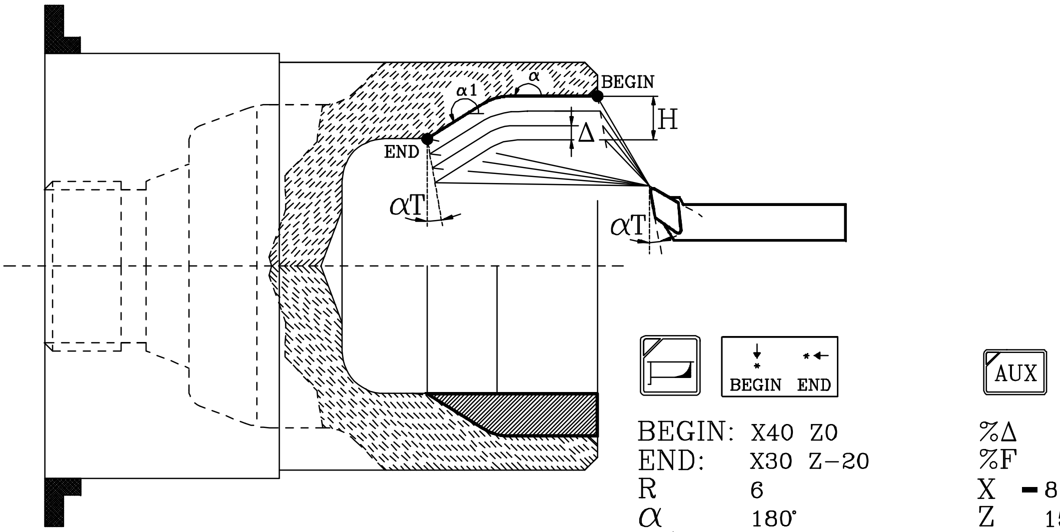
%Δ
%F
X 1
Z 1
T



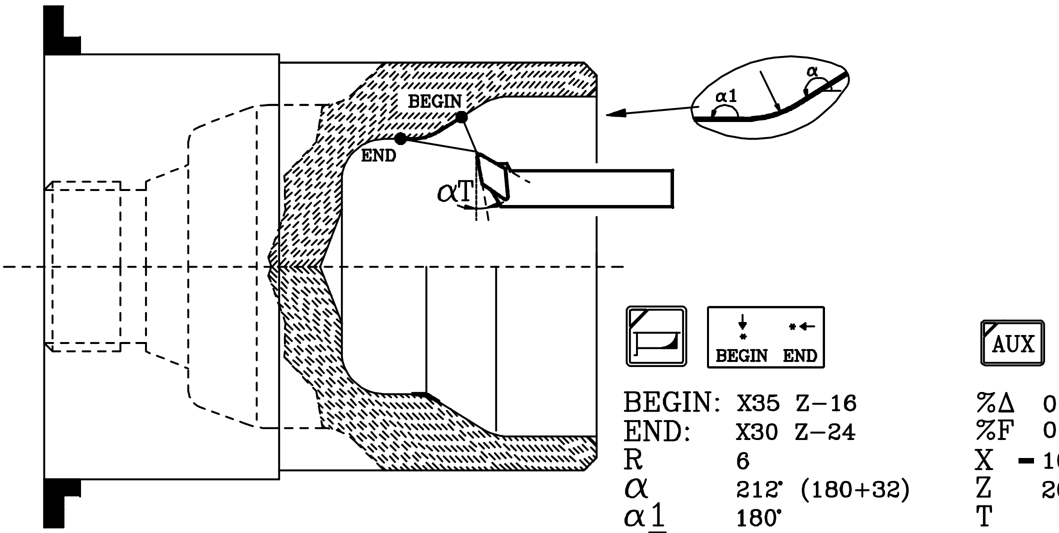
BEGIN: X20 Z−25
R 5
Δ
N
T 4
F
CSS

AUX

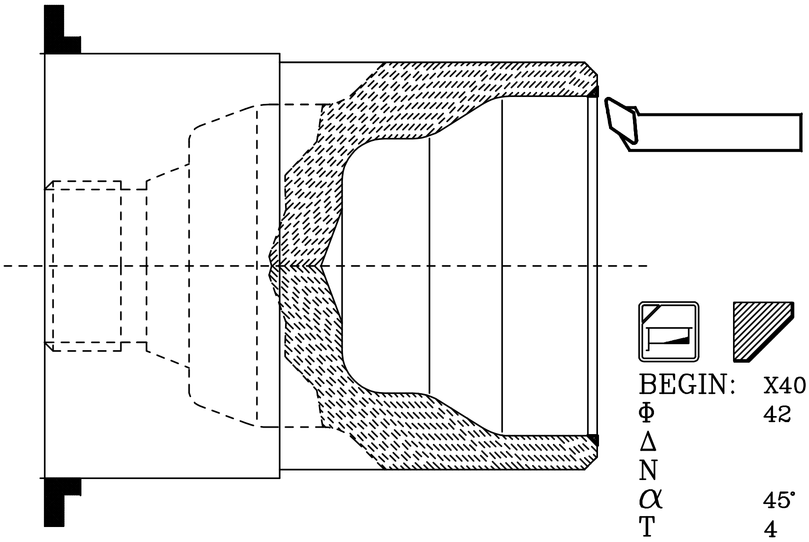
%Δ
%F
X 1
Z 26 (25+1)
T



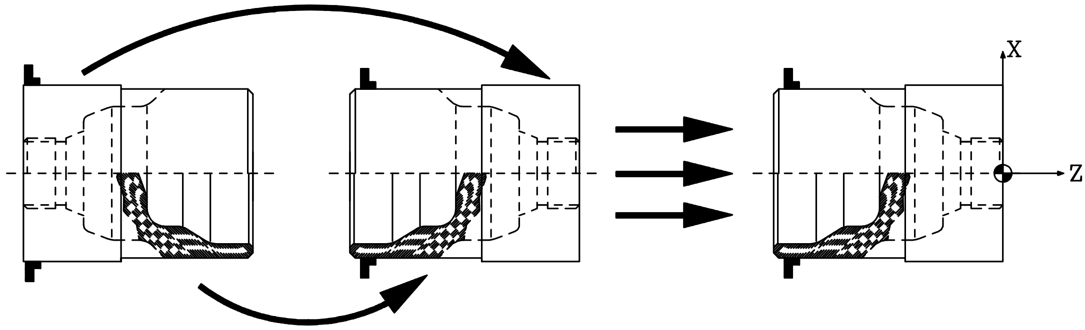
BEGIN:	X40 Z0	%Δ	
END:	X30 Z–20	%F	
R	6	X	– 8
α	180°	Z	15
α1	212° (180+32)	T	
αT	5°		
Δ			
H	5.5		
T	4		
F			
CSS			

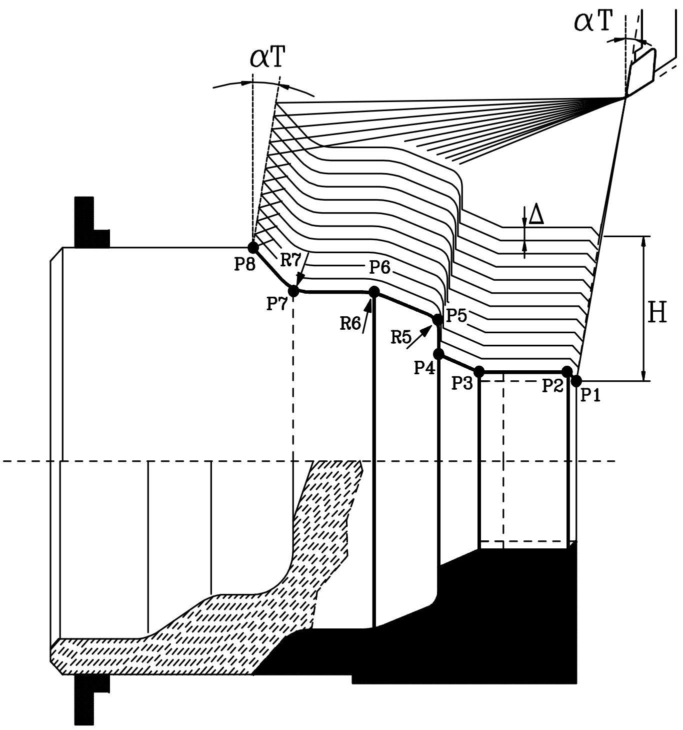


BEGIN:	X35 Z–16	%Δ	0
END:	X30 Z–24	%F	0
R	6	X	– 10
α	212° (180+32)	Z	20
α1	180°	T	
αT	5°		
Δ	0.5		
H	0.5		
T	4		
F			
CSS			



BEGIN:	X40 Z0	%Δ	
Φ	42	%F	
Δ		X	1
N		Z	1
α	45°	T	
T	4		
F			
CSS			

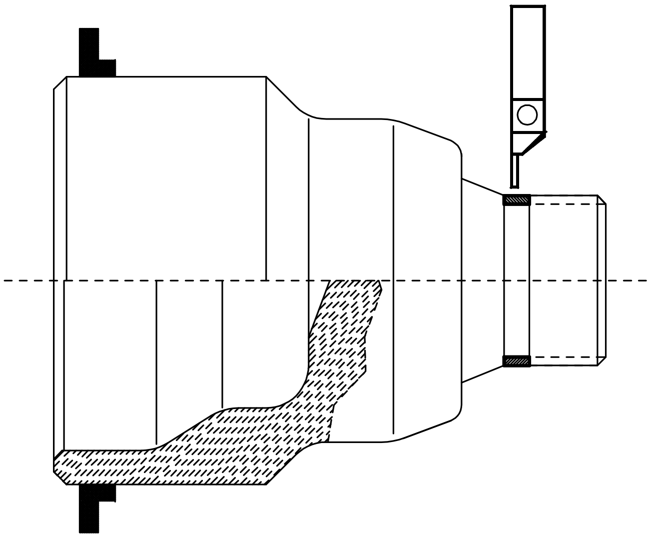




P1 = X18 Z0
P2 = X20 Z−1
P3 = X20 Z−12
P4 = X24 Z−17
P5 = X32 Z−17 R5 = 2
P6 = X38 Z−25 R6 = 8
P7 = X38 Z−35 R7 = 5
P8 = X48 Z−40

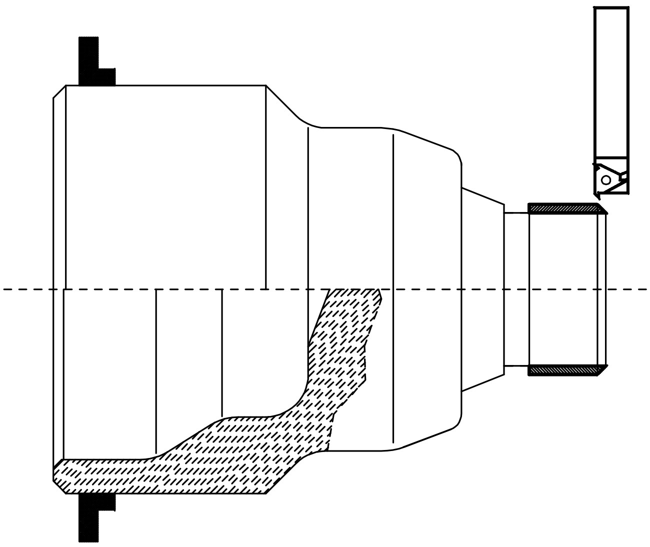
P1 P12

H	18	%Δ	
Δ		%F	
αT	5°	X	15
T	1	Z	7
F		T	
CSS			



AUX

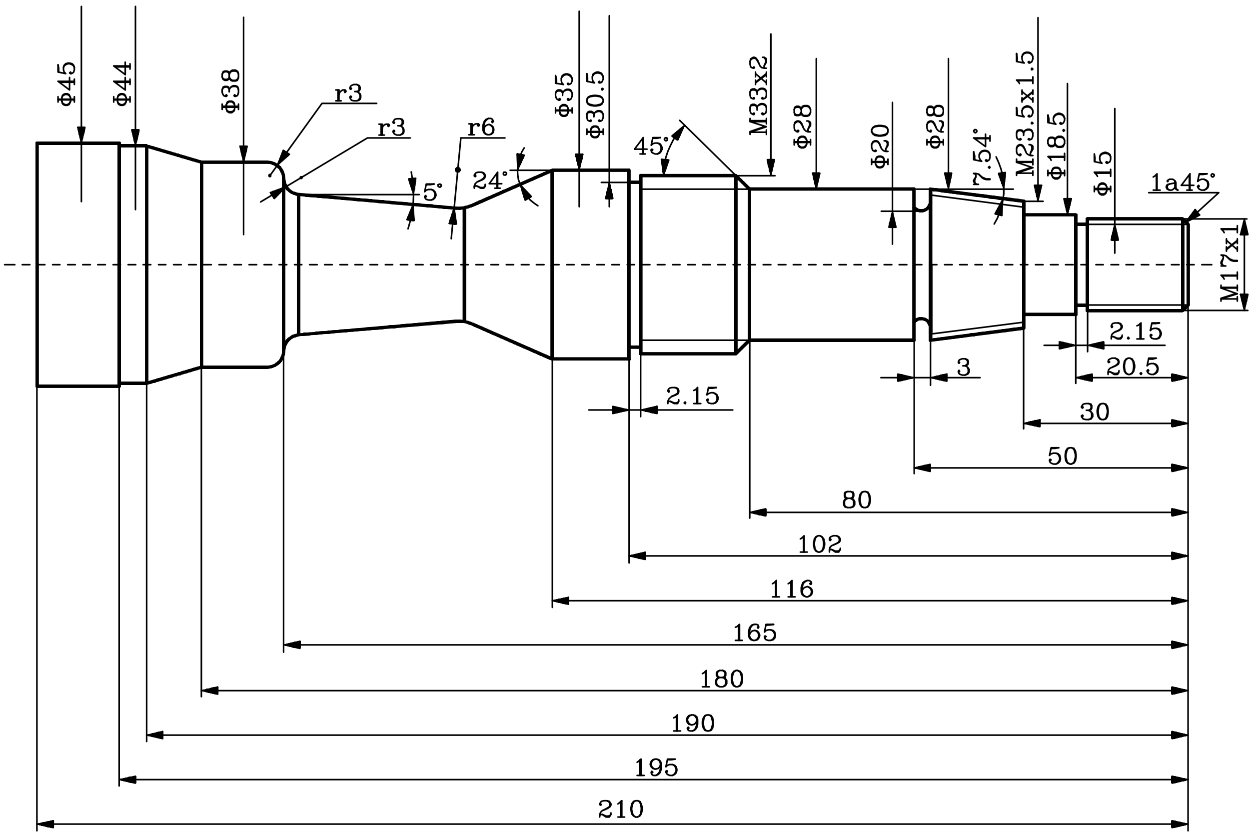
BEGIN:	X20 Z−9	%Δ	
END:	X20 Z−12	%F	
Φ	18	X	15
N	2	Z	1
TW	2.15	T	
T	2		
F			
CSS			



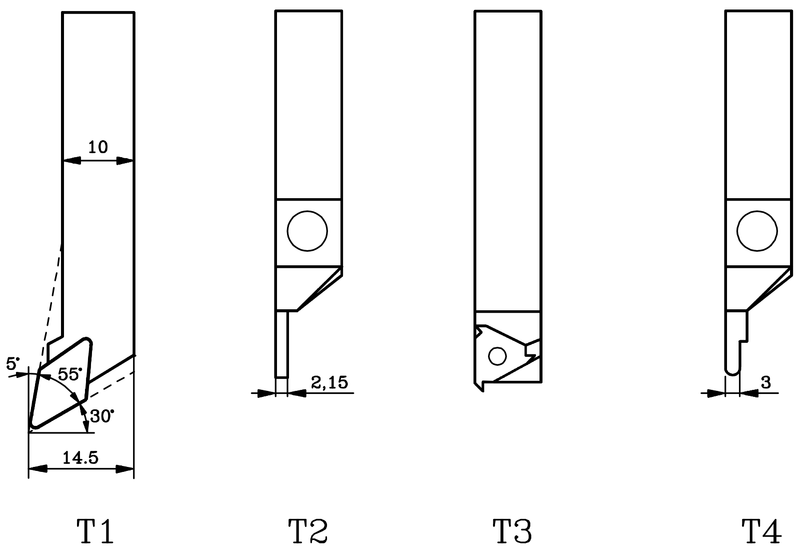
AUX

BEGIN:	X20 Z0	%Δ	
END:	X20 Z−10	%F	
P	1	X	15
H	0.613	Z	0
Δ		T	
α			
T	3		
F			
CSS			

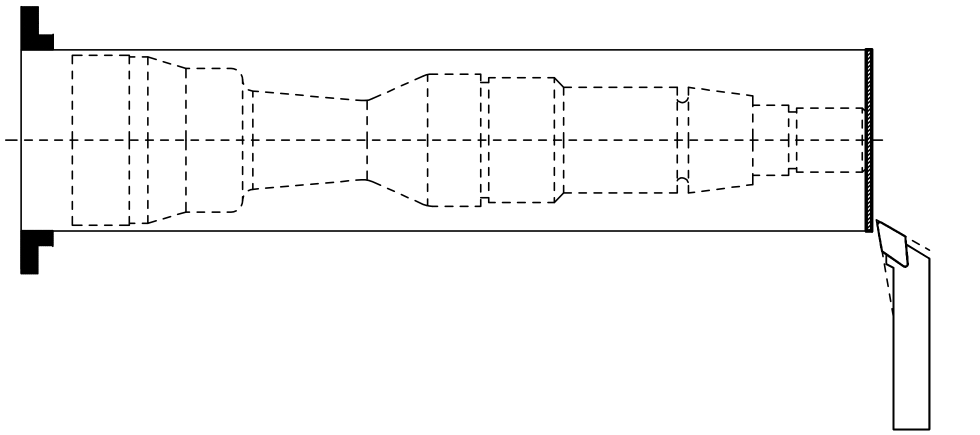
EXAMPLE 4

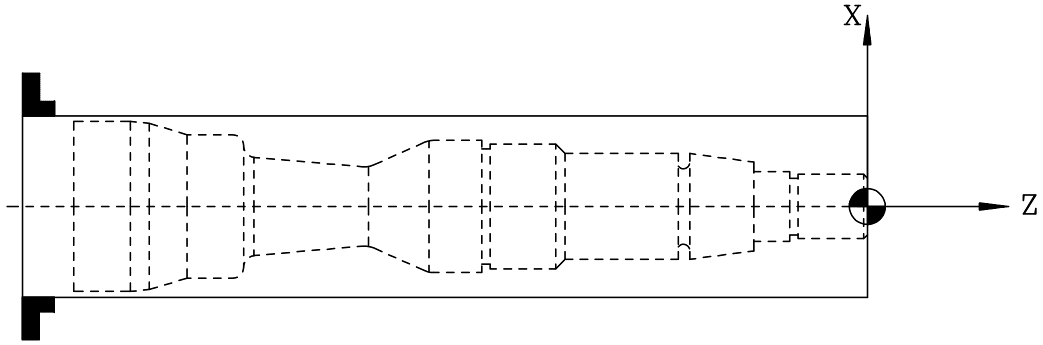


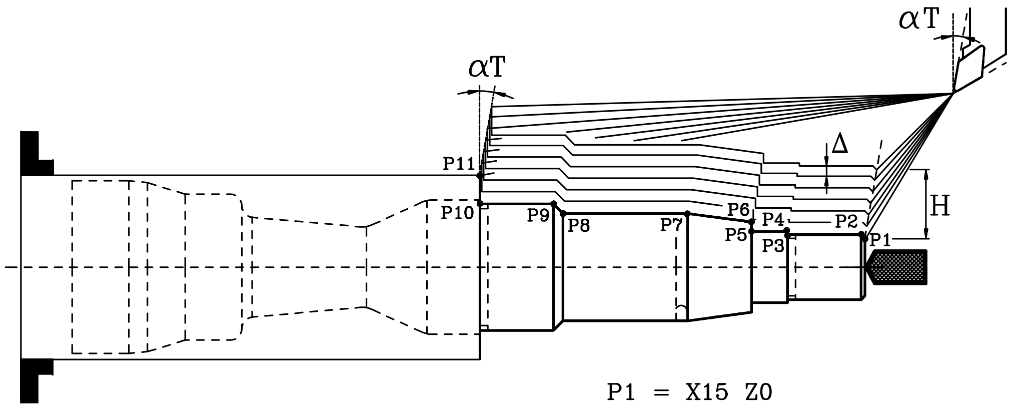
Φ45
Φ44
Φ38
r3
r3
r6
5°
24°
Φ35
Φ30.5
45°
M33x2
Φ28
Φ20
Φ28
7.54°
M23.5x1.5
Φ18.5
Φ15
1a45°
M17x1
2.15
20.5
3
2.15
30
50
80
102
116
165
180
190
195
210



10
5°
55°
30°
14.5
2,15
3
T1
T2
T3
T4





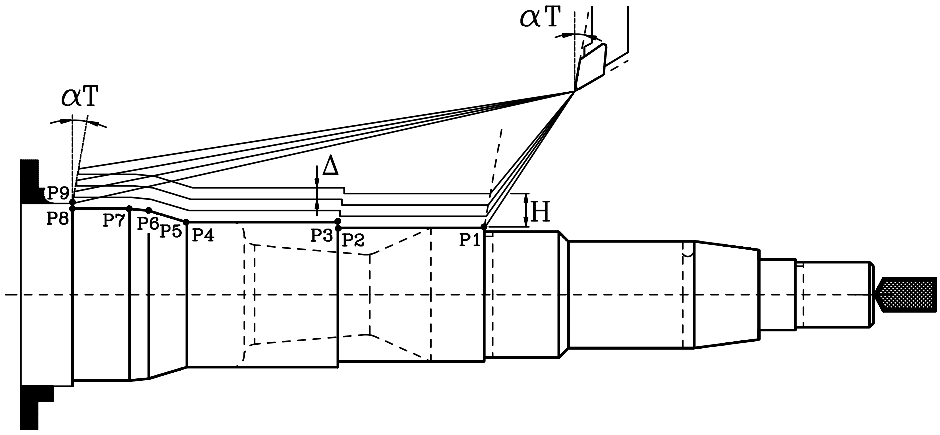


P1 P12

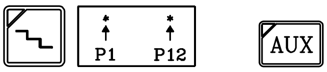
AUX

H	17	%Δ	
Δ		%F	
αT	5°	X	20
T	1	Z	23
F		T	
CSS			

P1 = X15 Z0
P2 = X17 Z−1
P3 = X17 Z−20.5
P4 = X18.5 Z−20.5
P5 = X18.5 Z−30
P6 = X23.5 Z−30
P7 = X28 Z−47
P8 = X28 Z−80
P9 = X33 Z−82.5
P10= X33 Z−102
P11= X48 Z−102
P12= X48 Z−102

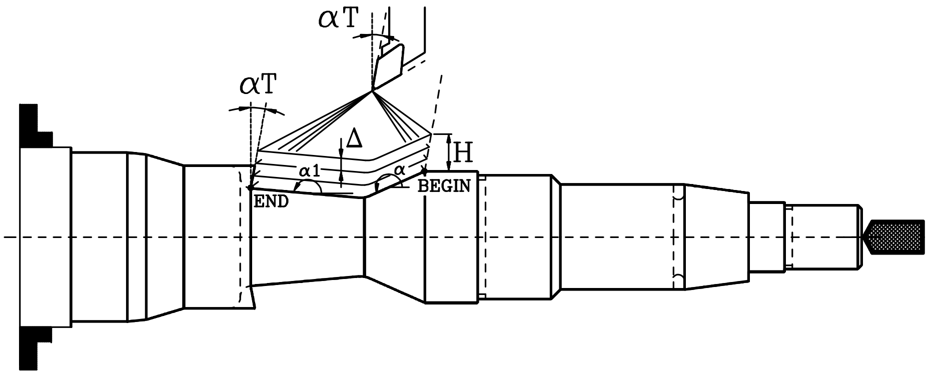


P1 =	X35 Z−102
P2 =	X35 Z−140
P3 =	X38 Z−140
P4 =	X38 Z−180
P5 =	X44 Z−190
P6 =	X44 Z−195
P7 =	X45 Z−195
P8 =	X45 Z−210
P9 =	X48 Z−210
P10=	X48 Z−210
P11=	X48 Z−210
P12=	X48 Z−210



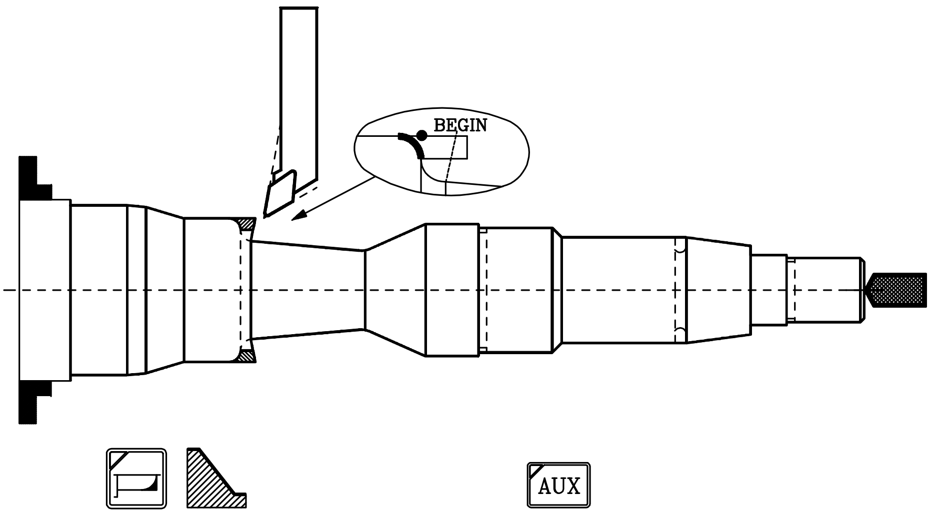
H	7
Δ	
αT	5°
T	1
F	
CSS	

%Δ	
%F	
X	28
Z	24
T	



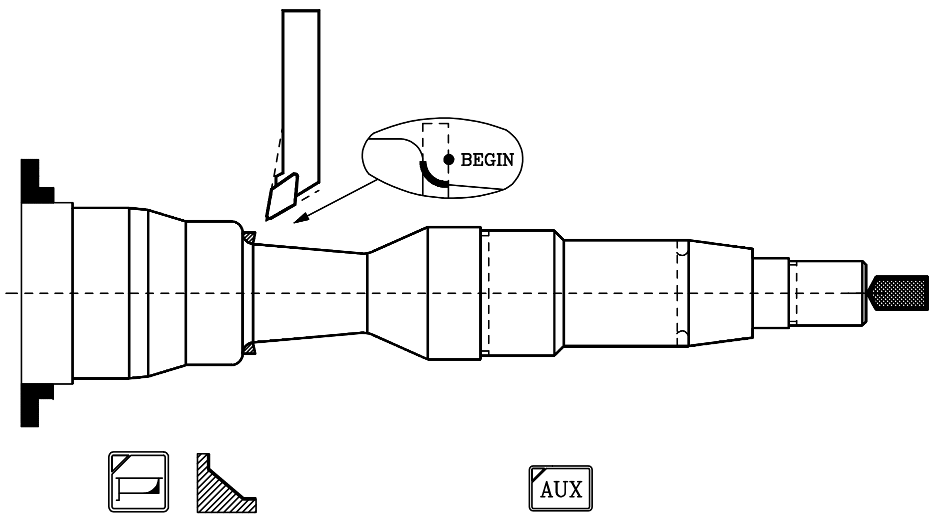
BEGIN:	X35 Z−116	
END:	X26 Z−162	
R	6	
α	204°	(24+180)
α1	175°	(180−5)
αT	5°	
Δ		
H	10	
T	1	
F		
CSS		

%Δ	
%F	
X	12
Z	−14
T	



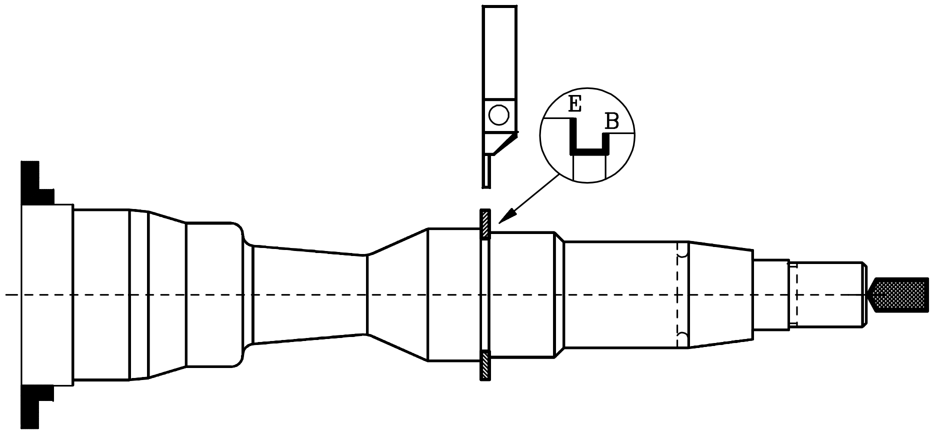
BEGIN: X38 Z−165
R 3
Δ
N
T 1
F
CSS

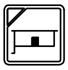
%Δ
%F
X 1
Z 6
T



BEGIN: X32 Z−162
R 3
Δ
N
T 1
F
CSS

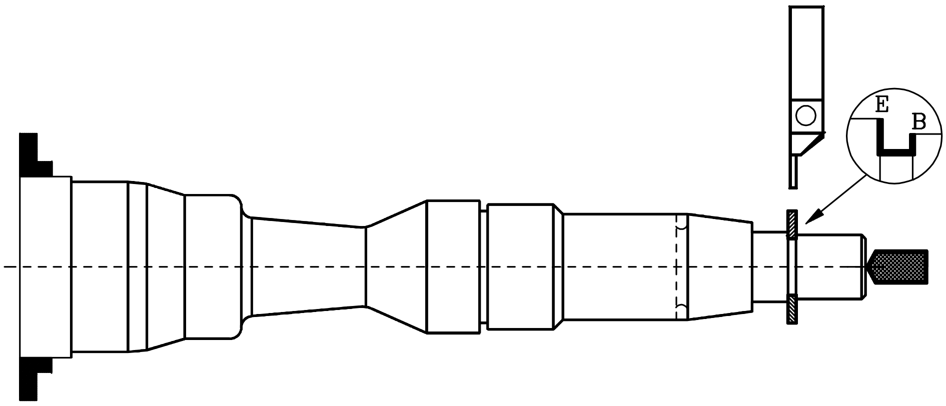
%Δ
%F
X 5
Z 1
T

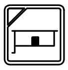




BEGIN: X33 Z−99.85
END: X35 Z−102
Φ 30.5
Δ
N
TW 2.15
T 2
F
CSS

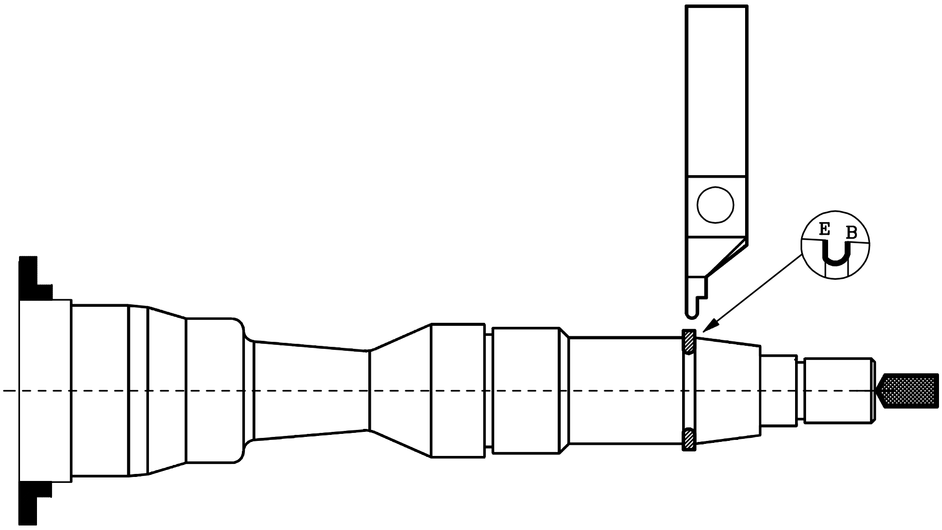
%Δ
%F
X 5
Z 1
T

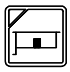




BEGIN: X17 Z−18.35
END: X18.5 Z−20.5
Φ 15
Δ
N
TW 2.15
T 2
F
CSS

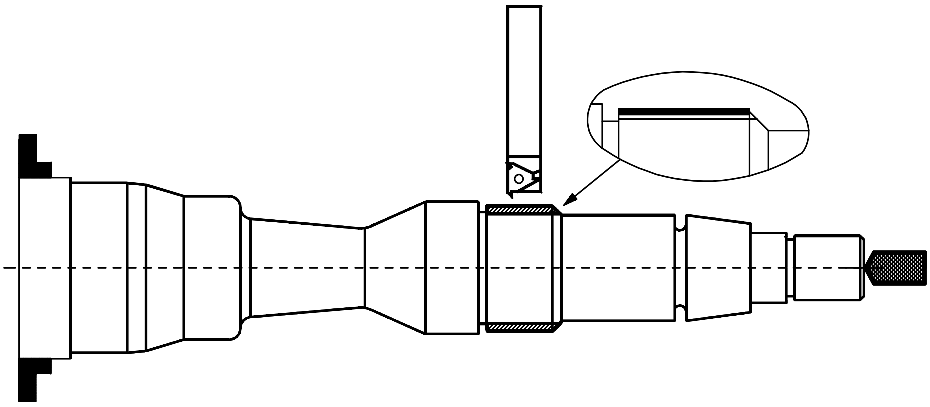
%Δ
%F
X 11
Z 1
T





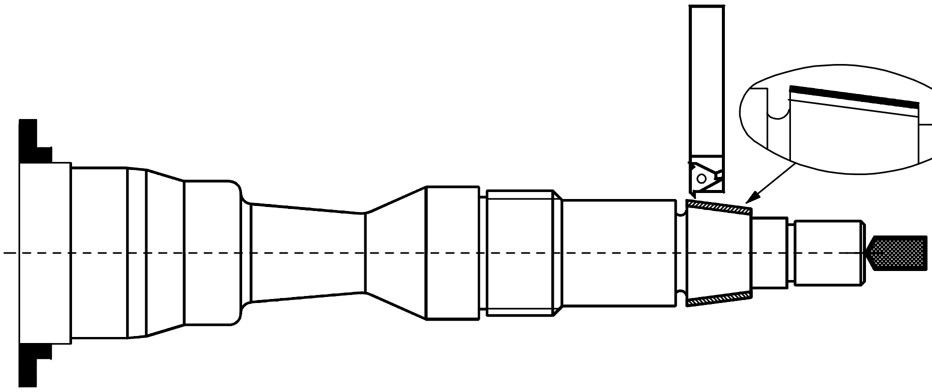
BEGIN:	X28 Z−47
END:	X28 Z−50
Φ	20
Δ	
N	
TW	3
T	4
F	
CSS	

%Δ	
%F	
X	10
Z	1
T	



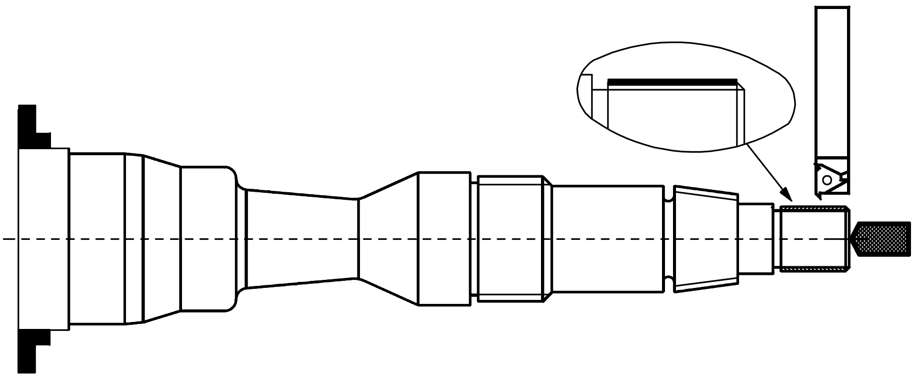
BEGIN:	X33 Z−80
END:	X33 Z−101
P	2
H	2x0.613=1.226
Δ	
α	
T	3
F	
CSS	

%Δ	
%F	
X	5
Z	1
T	



BEGIN:	X23.5 Z−30
END:	X28 Z−48
P	1.5
H	1.5x0.613=0.919
Δ	
α	
T	3
F	
CSS	

%Δ	
%F	
X	10
Z	1
T	



BEGIN:	X17 Z0
END:	X17 Z−19.5
P	1
H	0.613
Δ	
α	
T	3
F	
CSS	

%Δ	
%F	
X	15
Z	1
T	

ERROR CODES

001 This error occurs when the first character of the block to be executed is not an "N".

002 Too many digits when defining a function in general.

003 A negative value has been assigned to a function which does not accept the (-) sign or an incorrect value has been given to a canned cycle parameter.

004 A canned cycle has been defined while function G02, G03 or G33 was active.

005 Parametric block programmed wrong.

006 There are more than 10 parameters affected in a block.

007 Division by zero.

008 Square root of a negative number.

009 Parameter value too large

010 * The range or the Constant Surface Speed has not been programmed

011 More than 7 "M" functions in a block.

012 This error occurs in the following cases:

> Function G50 is programmed wrong

> Tool dimension values too large.

> Zero offset values (G53/G59) too large.

013 Canned cycle profile defined incorrectly.

014 A block has been programmed which is incorrect either by itself or in relation with the program history up to that instant.

015 Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G53, G54, G55, G56, G57, G58, G59, G72, G73, G74, G92 and G93 must be programmed alone in a block.

016 The called subroutine or block does not exist or the block searched by means of special function F17 does not exist.

017 Negative or too large thread pitch value.

018 Error in blocks where the points are defined by means of angle-angle or angle-coordinate.

019 This error is issued in the following cases:

> After defining G20, G21, G22 or G23, the number of the subroutine it refers to is missing.

> The "N" character has not been programmed after function G25, G26, G27, G28 or G29.

> Too many nesting levels.

020 More than one spindle range have been defined in the same block.

021 This error will be issued in the following cases:

> There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.
> The corresponding axis has not been defined in the addressed block

022 An axis is repeated when programming G74.

023 K has not been programmed after G04.

025 Error in a definition block or subroutine call, or when defining either conditional or unconditional jumps.

026 This error is issued in the following cases:

> Memory overflow.

> Not enough free tape or CNC memory to store the part-program.

027 I//K has not been defined for a circular interpolation or thread.

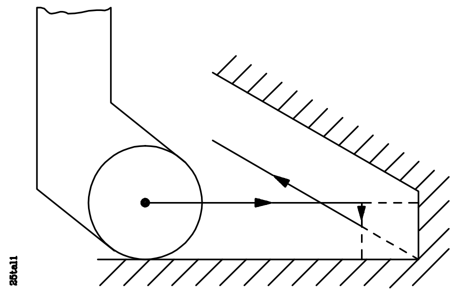
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

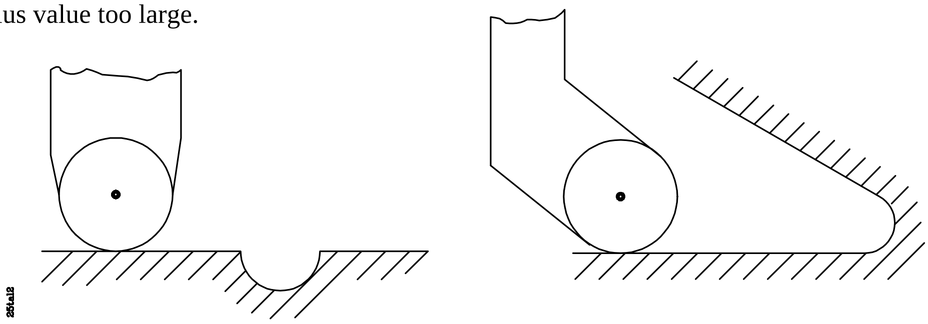
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

030 The programmed G function does not exist.

031 Tool radius value too large.



032 Tool radius value too large.



033 A movement of over 8388 mm or 330.26 inches has been programmed.

Example: Being the Z axis position Z-5000, if we want to move it to point Z5000, the CNC will issue error 33 when programming the block N10 Z5000 since the programmed move will be:
Z5000 - Z-5000 = 10000 mm.

In order to make this move without issuing this error, it must be carried out in two stages as indicated below:

```
N10 Z0        ; 5000 mm move
N10 Z5000     ; 5000 mm move
```

034 S or F value too large.

035 Not enough information for corner rounding, chamfering or compensation.

036 Repeated subroutine.

037 Function M19 programmed incorrectly.

038 Function G72 programmed incorrectly.

It must be borne in mind that if G72 is applied only to one axis, this axis must be positioned at part zero (0 value) at the time the scaling factor is applied.

039 This error occurs in the following cases:

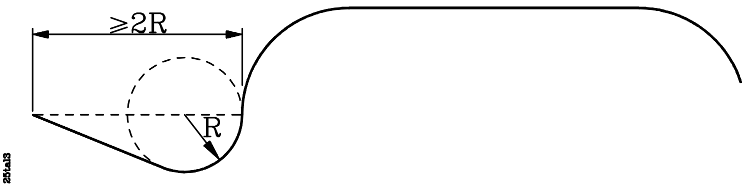
> More than 15 nesting levels when calling subroutines.

> A block has been programmed which contains a jump to itself. Example: N120 G25 N120.

040 The programmed arc does not go through the defined end point (tolerance 0.01mm) or there is no arc that goes through the points defined by G08 or G09.

041 This error is issued when programming a tangential entry as in the following cases:

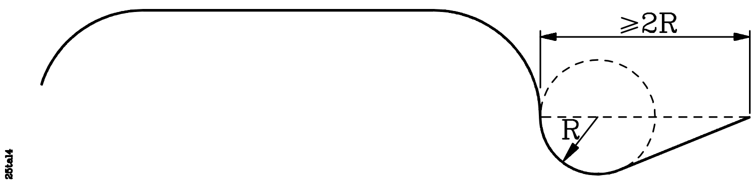
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> If the tangential entry is to be applied to an arc (G02, G03), The tangential entry must be defined in a linear block.

042 This error is issued when programming a tangential exit as in the following cases:

> There is no room to perform the tangential exit. A clearance of twice the rounding radius or greater is required.



> If the tangential exit is to be applied to an arc (G02, G03), The tangential exit must be defined in a linear block.

043 Polar origin coordinates (G93) defined incorrectly.

044 Function M45 S programmed wrong (speed of the live tool).

045 Function G36, G37, G38 or G39 programmed incorrectly.

046 Polar coordinates defined incorrectly.

047 A zero movement has been programmed during radius compensation or corner rounding.

048 Start or cancel tool radius compensation while in G02 or G03.

049 Chamfer programmed incorrectly.

050 Constant Surface Speed has been selected while the machine uses the BCD coded spindle speed output.

054 There is no tape in the cassette reader or the reader head cover is open or there is no disk in the FAGOR Floppy Disk Unit.

055 Parity error when reading or writing a cassette or a disk.

057 Write-protected tape or disk.

058 Sluggish tape or disk movement.

059 CNC communication error with the cassette reader or FAGOR Floppy Disk Unit.

060 Internal CNC hardware error. Consult with the Technical Service Department.

061 Battery error.

The memory contents will be kept for 10 more days (with the CNC off) from the moment this error occurs. The whole battery module located on the back must be replaced. Consult with the Technical Service Department.

Atention:

Due to danger of explosion or combustion: do not try to recharge the battery, do not expose it to temperatures higher than 100°C (232°F) and do not short the battery leads.

064 * External emergency input (pin 14 of connector I/O1) is activated.

065 * This error comes up if, while probing (G75), the programmed position is reached without receiving the probe signal.

066 * X axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

068 * Z axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

070 ** X axis following error.

072 ** Z axis following error.

074 ** "S" value (spindle speed) too large.

075 ** Feedback error at connector A1.

076 ** Feedback error at connector A2.

077 ** Feedback error at connector A3.

078 ** Feedback error at connector A4.

079 ** Feedback error at connector A5.

087 ** Internal CNC hardware error. Consult with the Technical Service Department.

088 ** Internal CNC hardware error. Consult with the Technical Service Department.

089 * All the axes have not been homed.

This error comes up when it is mandatory to search home on all axes after power-up. This requirement is set by machine parameter.

090 ** Internal CNC hardware error. Consult with the Technical Service Department.

091 ** Internal CNC hardware error. Consult with the Technical Service Department.

092 ** Internal CNC hardware error. Consult with the Technical Service Department.

093 ** Internal CNC hardware error. Consult with the Technical Service Department.

094 Parity error in tool table or zero offset table G53-G59.

095 ** Parity error in general parameters.

096 ** Parity error in Z axis parameters.

098 ** Parity error in X axis parameters.

099 ** Parity error in M table.

100 ** Internal CNC hardware error. Consult with the Technical Service Department.

101 ** Internal CNC hardware error. Consult with the Technical Service Department.

105 This error comes up in the following cases:

> A comment has more than 43 characters.

> A program has been defined with more than 5 characters.

> A block number has more than 4 characters.

> Strange characters in memory.

106 ** Inside temperature limit exceeded.

108 ** Error in Z axis leadscrew error compensation parameters.

110 ** Error in X axis leadscrew error compensation parameters.

111 * FAGOR LAN line error. Hardware installed incorrectly.

112 * FAGOR LAN error. It comes up in the following instances:

> When the configuration of the LAN nodes is incorrect.

> The LAN configuration has been changed. One of the nodes is no longer present (active).

When this error occurs, access the LAN mode, editing or monitoring, before executing a program block.

113 * FAGOR LAN error. A node is not ready to work in the LAN. For example:

> The PLC64 program is not compiled.

>A G52 type block has been sent to an 82CNC while it was in execution.

114 * FAGOR LAN error. An incorrect command has been sent out to a node.

115 * Watch-dog error in the periodic module.

This error occurs when the periodic module takes longer than 5 milliseconds.

116 * Watch-dog error in the main module.

This error occurs when the main module takes longer than half the time indicated in machine parameter "P729".

117 * The internal CNC information requested by activating marks M1901 thru M1949 is not available.

118 * An attempt has been made to modify an unavailable internal CNC variable by means of marks M1950 thru M1964.

119 Error when writing machine parameters, the decoded M function table and the leadscrew error compensation tables into the EEPROM memory.

This error may occur when after locking the machine parameters, the decoded M function table and the leadscrew error compensation tables, one tries to save this information into the EEPROM memory.

120 Checksum error when recovering (restoring) the machine parameters, the decoded M function table and leadscrew error compensation tables from the EEPROM memory.

Atention:

The **ERRORS** indicated with "*" behave as follows:

They stop the axis feed and the spindle rotation by cancelling the Enable signals and the analog outputs of the CNC.

They interrupt the execution of the part-program of the CNC if it was being executed.

The **ERRORS** indicated with "**" besides behaving as those with an "*", they activate the INTERNAL EMERGENCY OUTPUT.

FAGOR 800T CNC

PROGRAMMING MANUAL

Ref. 9701 (ing)

ABOUT THE INFORMATION IN THIS MANUAL

This manual should be used when elaborating an ISO-coded program.

This CNC can store 2 user-defined ISO-coded programs:

P99994	Special ISO-coded user program to store subroutines.
P99996	ISO-coded user Part-program.

Both programs may be written at a PC and sent out to the CNC. The Peripherals section of the Operating Manual describes how to transfer information between the CNC and a PC.

P99996 may be edited at the CNC; but P99994 cannot be accessed from the CNC. It must be edited at a PC or peripheral device.

This manual describes all the information on the ISO code used by the 800T CNC.

Notes: The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

INDEX

Section | Page

Section Page

Chapter 4 OTHER FUNCTIONS

Chapter 5 PATH CONTROL

Chapter 6 ADDITIONAL PREPARATORY FUNCTIONS

Chapter 7 TOOL COMPENSATION

Section Page

Chapter 8 MACHINING CANNED CYCLES

Chapter 9 SUBROUTINES

Chapter 10 PARAMETRIC PROGRAMMING

PROGRAMMING EXAMPLE

ERROR CODES

COMPARISON TABLE FOR FAGOR 800T CNC MODELS

AVAILABLE 800T CNC MODELS

Compact model with 8" amber CRT.

Modular model with 9" amber Monitor.

Consisting of Central Unit, Monitor and Keyboard.

Modular model with 14" Color Monitor

Consisting of Central Unit, Monitor and Keyboard.

TECHNICAL DESCRIPTION

	800-T	800-TI	800-TG	800-TGI
X, Z axes control	l	l	l	l
Spindle control	l	l	l	l
Spindle in RPM	l	l	l	l
Constant Surface Speed (CSS)	l	l	l	l
Spindle Orientation	l	l	l	l
Tools	32	32	32	32
Tool Compensation	l	l	l	l
Live Tool	l	l	l	l
Electronic Handwheels	2	2	2	2
RS 232C Communications	l	l	l	l
Integrated PLC (PLCI)		l		l
ISO-coded program editing (P99996)	l	l	l	l
Execution of ISO-coded program (P99996)	l	l	l	l
Graphics			l	l

NEW FEATURES AND MODIFICATIONS

Date: **April 1993** ***Software Version:*** **2.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
Rapid jog depending on position of Feedrate Override Switch	Operating Manual	Section 2.3.1
Tool for the finishing pass	Installation Manual Operating Manual	Section 3.5 Section 3.5
Handwheel movement limited to maximum allowed F	Operating Manual	Section 2.3.3
Control of software travel limits when using a handwheel		
Display format for S	Installation Manual	Section 6
Possibility to activate/deactivate outputs O1, O2, O3 after interrupting the program		
Automatic operation "Profile Rounding"	Operating Manual	Section 5.5.3
Profiles	Operating Manual	Chapter 6

Date: **October 1993**	***Software Version:*** **3.1 and newer**	
FEATURE	**AFFECTED MANUAL AND SECTION**	
Spindle acc./dec.	Operating Manual	Chapter 6
RPM Limitation when operating in CSS	Operating Manual	Section 4.3.1
Spindle orientation	Installation Manual Operating Manual	Section 6.4.1 Section 4.8
Live tool	Installation Manual Operating Manual	Section 5.9 Section 2.3
Automatic operation "Simple Drilling"	Operating Manual	Section 5.8
Automatic operation "Multiple Drilling"	Operating Manual	Section 5.9

Date: **December 1993** ***Software Version:*** **3.2 and newer**

FEATURES	**AFFECTED MANUAL AND SECTION**	
Assign a 5-digit number to the part program	Operating Manual	Chapter 7
Save part programs out to a peripheral	Operating Manual	Section 7.7
Automatic operation "Slot milling"	Operating Manual	Section 5.10
Delay before opening the positioning loop	Installation Manual	Section 4.3.2
"Special modes" accessing password	Installation Manual	Section 3.7
Handwheel inactive when Feedrate Override Switch out of handwheel positions	Installation Manual	Section 4.3.2

Date: **July 1994** ***Software Version:*** **4.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
Linear and Bell-shaped spindle acc./dec.	Installation Manual	Section 5.8
Profile with/without corner rounding.	Operating Manual	Section 6.2
Threading operation also with thread exit.	Operating Manual	Section 5.6.2
Rapid jog at 200% or depending on the position of the Feedrate Override Switch.	Installation Manual Operating Manual	Section 4.3.3 Section 2.3.1
Tool inspection	Installation Manual Operating Manual Operating Manual	Section 3.4.3 Section 3.4.3 Section 5.1.3
Execution of program 99996	Installation Manual Operating Manual	Section 3.11 Section 3.10

Date: **January 1995** ***Software version:*** **5.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
M3/M44 confirmation by detecting feedback reversal	Installation Manual	Section 6.4
JOG movements also in mm/rev		
Handwheel governed by the PLCI	Installation Manual	Section 4.3.2
Spindle inhibit from PLCI	PLCI Manual	
Clear all arithmetic parameter contents setting them to "0".	Installation Manual Operating Manual	Section 3.10 Section 3.9 & 7.9
Automatic rounding operation (Cycle level) with angle other than 90°.	Operating Manual	Section 5.5.2
Automatic grooving operation on the face of the part and finishing pass.	Operating Manual	Section 5.7
Automatic profile rounding operation by pattern repeat of profile or roughing.	Operating Manual	Section 5.5.3
Approach point in profile rounding operation (modification).	Operating Manual	Section 5.5.3
Automatic Profile execution, Cycle Level, by pattern repeat or roughing.	Operating Manual	Section 6.2
Approach point in automatic Profile execution (modification).	Operating Manual	Section 6.2
Automatic tapping operation.	Operating Manual	Section 5.8
M20 at the end of part-program execution.	Installation Manual	Section 3.8.3.1
Graphic simulation	Operating Manual	Section 5.1.3
Execution / Simulation of program P99996 (ISO-coded user program)	Installation Manual Operating Manual	Section 3.11 Section 3.10
Automatic or Single-block execution of P99996	Operating Manual	Section 3.10
Editing of program P99996	Installation Manual Operating Manual Programming Manual	Section 3.12 Section 3.11
ISO-coded user program P99994 to store subroutines	Programming Manual	Chapter 9
Subroutine associated to the execution of a tool (only when executing program P99996)	Installation Manual Programming Manual	Section 4.3.4
ISO codes of the 800T CNC	Programming Manual	

Date: **March 1995** ***Software version:*** **5.2 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Editing of program P99996 in all models.		
When interrupting the execution, the following keys are enabled: spindle, coolant, O1, O2, O3 and TOOL.	Installation Manual Operating Manual Operating Manual Operating Manual	Section 3.11 Section 3.10 Section 5.1.4 Section 7.5
Incremental JOG movements taking current work units (radius or diameter) into account.	Installation Manual	Section 4.3.3
ISO programming. New functions: G47, G48 (single block treatment).	Programming Manual	Section 6.7
ISO programming. New function: G86 (Longitudinal threadcutting canned cycle).	Programming Manual	Section 8.17
Request from the PLCI for real spindle rpm.	PLCI Manual	

Date: **November 1995** ***Software version:*** **5.5 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Tool offset modification while in execution.	Operating Manual	Section 3.4.4
Operation with a single electronic handwheel.	Installation Manual Installation Manual	Section 4.3.2 Section 7.5
Actual "S" speed reading from the PLCI.	PLCI Manual	

INTRODUCTION

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Protections of the unit itself

Central Unit

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

All the digital inputs and outputs are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Monitor

The type of protection fuse depends on the type of monitor. See the identification label of the unit itself.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

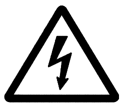
Safety symbols

Symbols which may appear on the manual

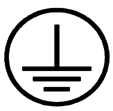
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.



"Electrical Shock" symbol
It indicates that point may be under electrical voltage



"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the CNC, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

When sending the monitor, especially protect the CRT glass.

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION
FOR THE 800T CNC

800T CNC OEM Manual	Is directed to the machine builder or person in charge of installing and starting up the CNC. It has the Installation manual inside. Sometimes, it may contain an additional manual describing New Software Features recently implemented.
800T CNC USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC. Sometimes, it may contain an additional manual describing New Software Features recently implemented.
DNC 25/30 Software Manual	Is directed to people using the optional DNC communications software.
DNC 25/30 Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the 800 without using the DNC25/30 software..
PLCI Manual	To be used when the CNC has an integrated PLC. Is directed to the machine builder or person in charge of installing and starting up the PLCI.
DNC-PLC Manual	Is directed to people using the optional communications software: DNC-PLC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The programming manual consists of the following sections:

Index

Comparative Table for Fagor 800T CNC models

New Features and modifications

Introduction
- Summary of safety conditions
- Material returning conditions
- Fagor documents for the 800T CNC
- Manual Contents

Chapter 1 Program writing
- Structure of the part-program and of all its blocks
- It shows the available preparatory G functions.

Chapter 2 Coordinate programming
- It shows how to select the work planes, work units, type of programming (absolute / incremental)
- It describes the coordinate systems for coordinate programming (Cartesian, polar, cylindrical, by means of angles, by means of an angle and a Cartesian coordinate).

Chapter 3 Reference systems
- It shows how to program machine reference search and coordinate preset as well as the zero offsets and polar origin preset.
- It indicates how to save the current coordinate origin and retrieve it later on.

Chapter 4 Other functions:
- It shows how to program the preparatory functions regarding axis feedrate and spindle speed.
- How to program the spindle turning speed. (rpm, CSS).
- How to program spindle orientation.
- How to program the tool and how to modify the table values via user program.
- How to program the auxiliary "M" functions.

Chapter 5 Path control
- It describes how to program the part in square and round corner.
- How to program fast positioning, linear and circular interpolations.
- How to program tangential entries and exits as well as corner rounding and chamfering.
- How to program threading and movement against hardstop

Chapter 6 Additional Preparatory Functions
- It shows how to program a dwell.
- How to apply mirror image functions.
- How to display an error code.
- How to work with jumps and unconditional jumps.
- How to program electronic threading
- How to apply the scaling factor.
- How to work with a probe
- How to program the single-block treatment.

Chapter 7 Tool compensation
- It shows how to program tool radius and length compensation.

Chapter 8 Machining canned cycles
- It shows how to program the different machining canned cycles.

Chapter 9 Subroutines.

Special user program for subroutines: P99994
It shows how to identify a standard and a parametric subroutine.
How to program a call to a standard or to a parametric subroutine.
It shows the subroutine nesting levels.

Chapter 10 Parametric programming

It shows how to use parametric programming (assignments, operators, jump functions, etc.)

Programming Example
Error codes

1. PROGRAM WRITING

A CNC program consists of a series of blocks or instructions.

These blocks or instructions consist of words formed by capital letters, signs and numbers

The signs and numbers for this CNC are:

. + -
0 1 2 3 4 5 6 7 8 9

It is possible to program without a number when the value is zero and without the sign when it is positive.

When programming, the numbers of a word may be replaced with an arithmetic parameter. Later on, during basic execution, the CNC will replace the arithmetic parameter with its value. For example:

If XP3 has been programmed, the CNC will replace P3 with its numeric value when executing this instruction obtaining a result such as X20, X20.567, X-0.003, etc.

1.1 CNC PROGRAM STRUCTURE

All program blocks (lines) of the program will have the following structure:

Block number + Block contents

1.2 BLOCK NUMBER

The block number serves to identify each one of the program blocks.

It is expressed by the letter "N" followed by up to 4 digits (0-9999).

These program blocks must be in numerical order (N10, N15, N37, N46, etc.). It is recommended not to assign them sequential numbers (N10, N11, N12, N13, etc.) in order to be able to insert future blocks between them if so required.

Atention:

All throughout this manual, we will indicate an "N4" format when referring to a block number meaning that the "N" letter must be followed by up to 4-digits (without decimals),

1.2.1 CONDITIONAL BLOCK (BLOCK SKIP)

There are two types of conditional blocks:

a) REGULAR BLOCK SKIP: N4.

If the block number N4 is followed by a period (.), the block is set as a regular block skip. This means that the CNC will execute it only if the corresponding conditional (block skip) input is activated.

When executing any program, the CNC reads four blocks ahead (look ahead) of the one currently in execution.

In order for the conditional block to be executed, the conditional input must be activated 4 blocks ahead of the conditional block.

b) SPECIAL CONDITIONAL BLOCK (BLOCK SKIP): N4..

If the block number N4 is followed by two periods (..), the block is set as a special block skip. This means that the CNC will execute it only if the corresponding conditional (block skip) input is activated.

In this case, the conditional input may be activated while executing the block before the conditional one.

The special conditional block N4.., cancels tool radius compensation G41 and G42.

1.3 BLOCK CONTENTS

It must be written with commands in ISO language, especially designed to control the movements of the axes, since it provides information and positioning conditions as well as feedrate data.

Each program block may contain the following functions:

G Preparatory functions
X, Z Coordinates (position values) of the axes
F Feedrate
S Spindle turning speed
T Tool number
M Auxiliary (miscellaneous) functions

This order must be maintained in the block although all these functions need not be in each block.

Depending on the selected work units (mm or inches), the following programming format must be used:

Millimeter format: N4 G2 X±4.3 Z±4.3 F4 S4 T2.2 M2
Inch format: N4 G2 X±3.4 Z±3.4 F4 S4 T2.2 M2

Throughout this manual the following formats will be mentioned and their meanings are:

"N4" Block (program line) number indicating that the letter "N" must be followed by up to 4 digits (N0 through N9999).

"G2" Referring to a preparatory function indicating that the letter "G" must be followed by up to 2 digits (G0 through G99).

± 4.3 Meaning that a positive or negative number may follow the letter (X or Z) by up to 4 integers and up to 3 decimals.

± 3.4 Meaning that a positive or negative number may follow the letter (X or Z) by up to 3 integers and up to 4 decimals.

"F4" Referring to axis feedrate, indicating that the letter "F" must be followed by up to 4 digits (F0 through F9999) when working in mm/min or inches/min.

When working in mm/rev. the format is F3.4 and when in inches/rev. F2.4

"S4" Referring to spindle speed, indicating that the letter "S" must be followed by up to 4 digits (S0 through S9999).

T2.2 Referring to the work tool, indicating that the letter "T" must be followed by up to 2 integers and two decimals.

"M2" Referring to the miscellaneous functions, indicating that the letter "M" must be followed by up to 2 digits (M0 through M99).

1.4 PREPARATORY FUNCTIONS (G)

These functions are programmed by means of the letter "G" followed by two digits (G2).

They are always programmed at the beginning of the block and they are used to set the geometry and operating conditions of the CNC.

1.4.1 "G" FUNCTION TABLE FOR THIS CNC

Function	M	D	Meaning	Section
G00	*		Rapid positioning	5.3
G01	*	*	Linear interpolation	5.4
G02	*		Clockwise circular interpolation	5.5
G03	*		Counter-clockwise circular interpolation	5.5
G04			Dwell	6.1
G05	*		Round corner	5.1
G06			Circular interpolation with absolute center coordinates	5.5
G07	*		Square corner	5.2
G08			Arc tangent to previous path	5.6
G09			Arc defined by three points	5.7
G20			Call to a standard subroutine	9.2
G21			Call to a parametric subroutine	9.4
G22			Identification of a standard subroutine	9.1
G23			Identification of a parametric subroutine	9.3
G24			End of subroutine	9.
G25			Unconditional jump/call	6.3
G26			Jump/Call if equal to 0	10.6
G27			Jump/Call if not equal to 0	10.6
G28			Jump/Call if smaller than	10.6
G29			Jump/Call if equal to or greater than	10.6
G30			Display error code	6.2
G31			Store coordinate origin	3.5
G32			Recover coordinate origin previously stored with G31	3.5
G33	*		Threadcutting	6.4
G36			Automatic radius blend	5.10
G37			Tangential entry	5.8
G38			Tangential exit	5.9
G39			Chamfer	5.11
G40	*	*	Cancelation of tool radius compensation	7.4
G41	*		Left-hand tool radius compensation	7.1
G42	*		Right-hand tool radius compensation	7.1
G47	*		Single-Block treatment: ON	6.7
G48	*	*	Single-Block treatment: OFF	6.7
G49	*		Programmable feedrate override %	4.1
G50			Load tool dimensions into tool offset table	4.3

Function	M	D	Meaning	Section
G51			Correct tool dimensions	4.3
G53/G59	*		Zero offsets	3.4
G66			Pattern repeat (roughing canned cycle following part's shape)	8.10
G67 N0			Turning canned cycle	8.1
G67 N1			Facing canned cycle	8.2
G67 N2			Taper turning canned cycle	8.3
G67 N3			Threading canned cycle	8.4
G67 N4			Rounding canned cycle	8.5
G67 N5			Grooving canned cycle	8.6
G67 N6			Simple drilling canned cycle	8.7
G67 N7			Multiple drilling canned cycle	8.8
G67 N8			Slot milling canned cycle	8.9
G68			Roughing canned cycle along the X axis	8.11
G69			Roughing canned cycle along the Z axis	8.12
G70	*		Inch programming	2.1
G71	*		Metric programming (in mm)	2.1
G72	*		Scaling factor	6.5
G74			Machine Reference (Home) search	3.1
G75			Probing	6.6
G81			Turning canned cycle for straight sections	8.13
G82			Facing canned cycle for straight sections	8.14
G84			Turning canned cycle for circular sections	8.15
G85			Facing canned cycle for circular sections	8.16
G86			Longitudinal threadcutting canned cycle	8.17
G90	*	*	Programming in absolute coordinates	2.2
G91	*		Programming in incremental coordinates	2.2
G92			Coordinate preset	3.2
			Maximum S value limit setting	4.2
G93			Polar origin preset	3.3
G94	*		Axis feedrate F in mm/min. (0.1 inch/min.)	4.1
G95	*	*	Axis feedrate F in mm/rev. (0.1 inch/rev.)	4.1
G96	*		S speed in m/min. (ft/min.) (Constant Surface Speed)	4.2
G97	*	*	S speed in rev./min.	4.2

"**M**" means MODAL. In other words, that once the "G" function has been executed, it remains active until another incompatible "G" function, M02, M30, EMERGENCY, or RESET is executed or the CNC is turned off and back on.

"**D**" means BY DEFAULT. That is, that they will be assumed by the CNC on power-up, after executing an M02, M30 or after an EMERGENCY or RESET.

One block may contain all the desired "G" functions and in any order **except**: G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G51, G53/G59, G72, G74 and G92 which must be programmed alone in a block for being special. If incompatible "G" functions are programmed in the same block, the CNC assumes the last one.

2. *COORDINATE PROGRAMMING*

2.1 MEASURING UNITS. MILLIMETERS (G71) OR INCHES (G70)

The CNC has machine parameter "P13" to set the measuring units to be used.

However, these units may be changed along the program by means of the following "G" functions:

G70 Programming in inches
G71 Programming in millimeters

Depending on whether G70 or G71 has been programmed, the CNC assumes those units for all the following blocks.

Functions G70/G71 are modal and incompatible with each other.

On power-up, after an M02, M30, RESET or EMERGENCY, the CNC assumes the measuring units set by machine parameter "P13".

2.2 *ABSOLUTE (G90) / INCREMENTAL (G91) PROGRAMMING*

The coordinates of a point may be programmed either in absolute, G90, or in incremental, G91.

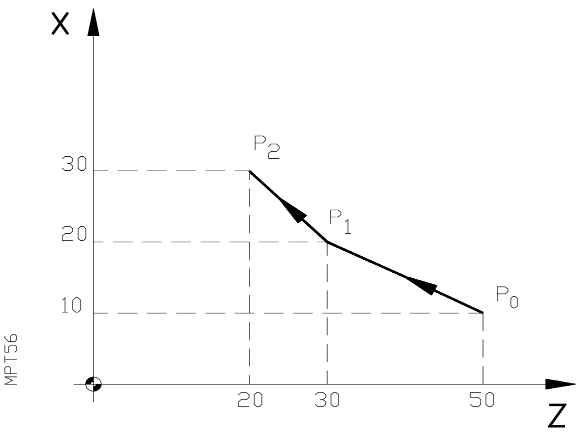
When working in G90, the coordinates of the programmed point are referred to the coordinate origin point.

When working in G91, the coordinates of the programmed point are referred to the end point (target point) of the previous block.

On power-up, after executing an M02, M30, EMERGENCY or RESET, the CNC assumes function G90.

Functions G90 and G91 are incompatible with each other.

Examples when the X axis is programmed in diameter.



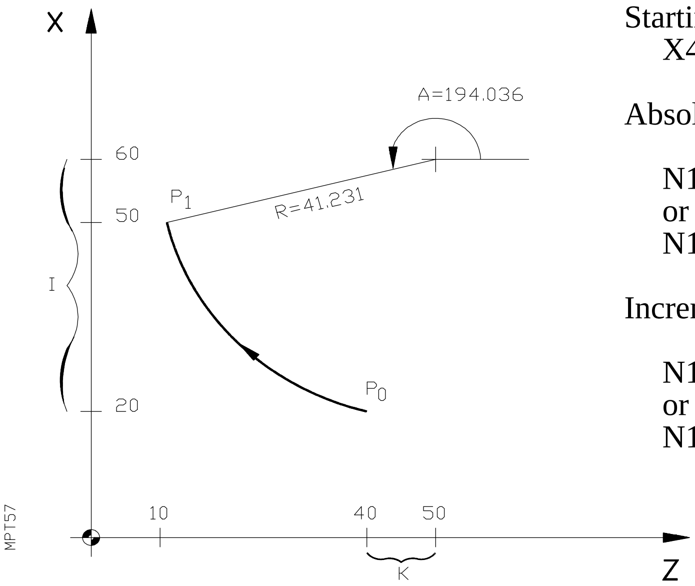
Starting point (P0)
X20 Z50.

Absolute programming G90

N100 G90 G01 X40 Z30 P0 —> P1
N110 X60 Z20 P1 —> P2

Incremental programming G91

N100 G91 G01 X20 Z-20 P0 —> P1
N110 X20 Z-10 P1 —> P2



Starting point (P0)
X40 Z40

Absolute programming G90

N100 G90 G02 X100 Z10 I40 K10
or
N100 G90 G02 X100 Z10 R41.231

Incremental programming G91

N100 G91 G02 X60 Z-30 I40 K10
or
N100 G91 G02 X60 Z-30 R41.231

2.3 COORDINATE PROGRAMMING

With this CNC, it is possible to program the axis coordinates in the following formats:

- Cartesian coordinates
- Polar coordinates
- Programming by two angles
- Programming by one angle and one cartesian coordinate

2.3.1 CARTESIAN COORDINATES

Their programming format is:

In mm: X±4.3 Z±4.3
In inches: X±3.4 Z±3.4

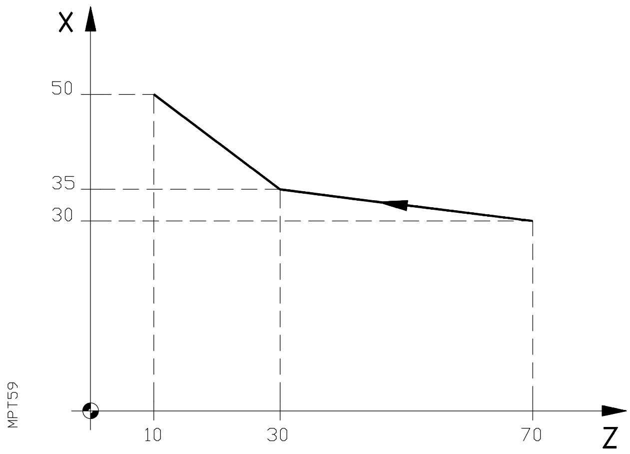
The values of the programmed coordinates will be either absolute or incremental depending on whether G90 or G91 has been programmed.

Positive coordinate values do not require the "+" sign. Leading zeros as well as trailing decimal zeros may also be omitted (0010.100 = 10.1).

Example: X axis programmed in diameter being "X60 Z70" the starting point.

Absolute coordinates:
N100 G90 X70 Z30
N110 X100 Z10

Incremental coordinates:
N100 G91 X10 Z-40
N110 X30 Z-20



2.3.2 POLAR COORDINATES

The format to define a particular point is:

In mm: R±4.3 A±3.3
In inches: R±3.4 A±3.3

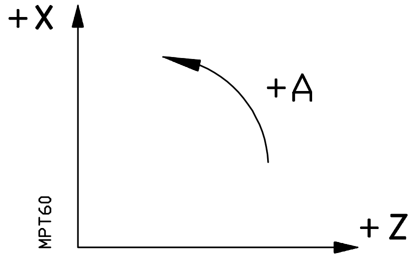
Where "R" is the radius value and "A" is the angle with respect to the polar origin. "A" is always given in degrees. On power-up, after an M02, M30, EMERGENCY or RESET, the CNC assumes point X0 Z0 as polar origin. This polar origin may be changed by programming function "G93".

The "R" and "A" values will be either absolute or incremental depending on whether G90 or G91 has been programmed.

Positive values do not require the "+" sign. Leading zeros as well as trailing decimal zeros may also be omitted (0010.100 = 10.1).

When programming rapid moves (G00) or linear interpolations, "R" and "A" must be programmed.

When programming circular interpolations (G02 or G03), the A±3.3 of the arc's end point and the coordinates of the arc center with respect to the starting point must be programmed.

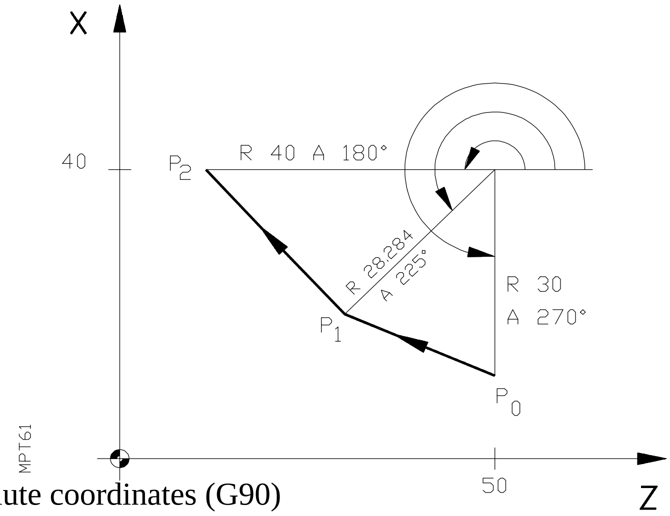


In the case of a circular interpolation (G02 or G03) when working in polar coordinates, the center of the arc must be defined by means of "I, K", like when using cartesian coordinates.

When programming a circular interpolation with G02, G03, the CNC assumes the center of the arc as the new polar origin.

Programming examples in millimeters and X axis in diameter.

Example 1



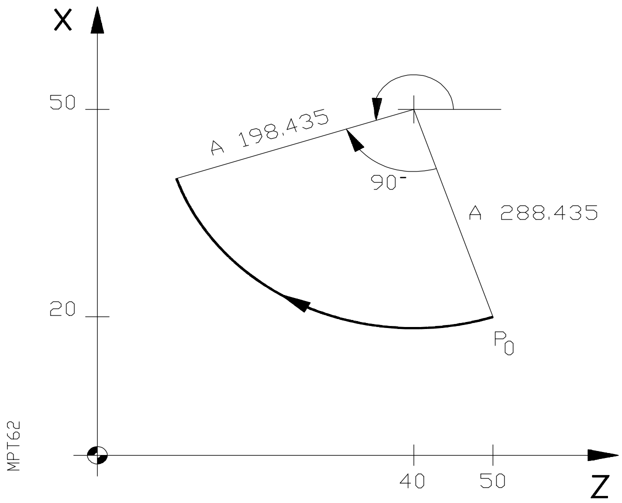
In absolute coordinates (G90)

N100 G93 I80 K50 .... (Polar origin preset)
N110 G01 G90 R30 A270 .... P0
N120 R28.284 A225 .... P1
N130 R40 A180 .... P2

In incremental coordinates (G91)

N100 G93 I80 K50 .... (Polar origin preset)
N110 G01 G90 R30 A270 .... P0
N120 G91 R-1.716 A-45 .... P1
N130 R11.716 A-45....................... P2

Example 2, assuming P0 (X40 Z50) as the starting point:



In absolute coordinates (G90)

N100 G90 G02 A198.435 I30 K-10

or N100 G93 I100 K40
N110 G90 G02 A198.435

In incremental coordinates (G91)

N100 G91 G02 A-90 I30 K-10

or N100 G93 I100 K40
N110 G91 G02 A-90

2.3.3 PROGRAMMING BY TWO ANGLES (A1, A2)

An intermediate point of a path may also be defined by:
A1 A2 (X, Z). Where:

A1 is the exit angle from the starting point of the path (P0).

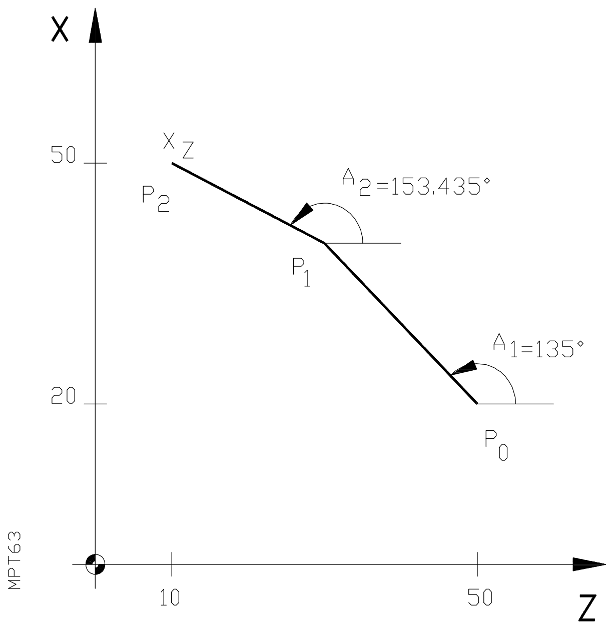
A2 is the exit angle from the intermediate point (P1).

(X, Z) are the coordinates of the end point (P2).

The CNC automatically calculates the P1 coordinates.

Programming example, where P0 (X40 Z50) is the starting point and the X axis is programmed in diameter.

N100 A135 A153.435
N110 X100 Z10



2.3.4 BY ONE ANGLE AND ONE CARTESIAN COORDINATE

It is also possible to define a point by means of the exit angle of the path at the previous point and one cartesian coordinate of the point to be defined.

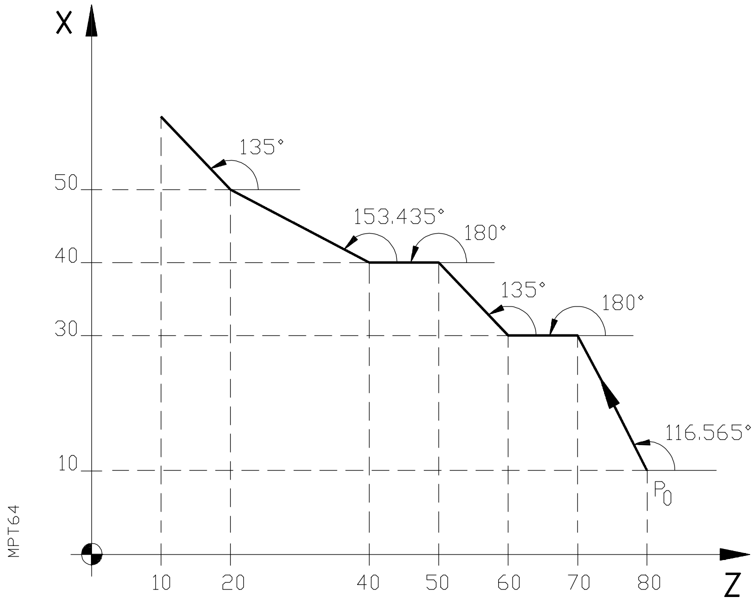
Programming example where P0 (X20 Z80) is the starting point and the X axis is programmed in diameter.

In absolute coordinates

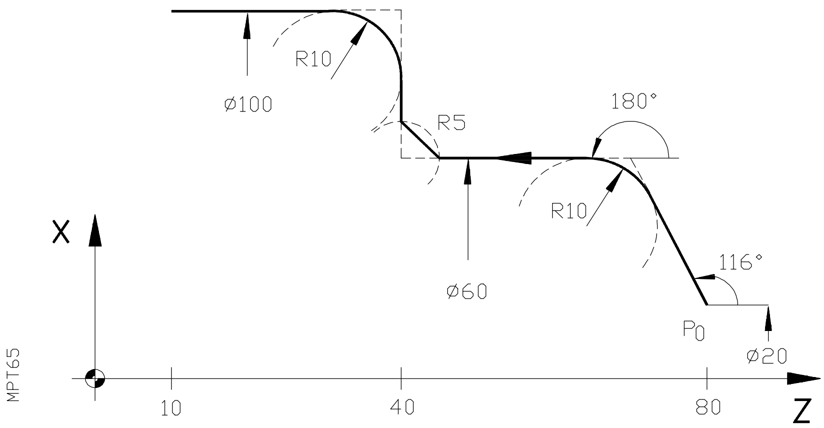
N100 G90
N110 A116.565 X60
N120 A180 Z60
N130 A135 X80
N140 A180 Z40
N150 A153.435 X100
N160 A135 Z10

In incremental coordinates

N100 G91
N110 A116.565 X40
N120 A180 Z-10
N130 A135 X20
N140 A180 Z-10
N150 A153.435 X20
N160 A135 Z-10



When defining points by two angles or by one angle and one coordinate, it is possible to insert roundings, chamfers, tangential entries and exits.



Starting point: P0 (X20 Z80)

N100 G01 G36 R10 A116 A180
N110 G39 R5 X60 Z40
N120 G36 R10 A90 X100
N130 A180 Z10

3. *REFERENCE SYSTEMS*

3.1 MACHINE REFERENCE (HOME) SEARCH (G74)

When programming G74 in a block, the CNC moves the axes to the machine reference point (home).

Two cases may occur:

a) Search on both axes.

If the block only contains G74, the CNC moves the X axis first and then the Z axis.

b) Search on one axis only or on both axes in a particular order.

To home only one axis, indicate the desired axis after function G74.

To home both axes; but in a particular order, other than the one described in a), program G74 followed by the axes in the desired order.

A block containing G74 cannot have any other function.

When the homing axis reaches home, the screen displays the distance from that point to the last programmed part zero minus the tool length along that axis (X or Z).

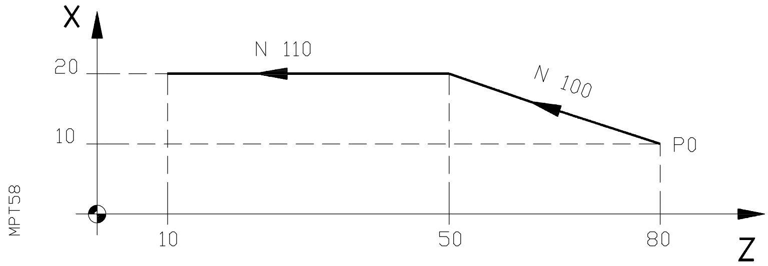
3.2 COORDINATE PRESET (G92)

By means of function G92, it is possible to preset any value for the CNC axes which translates into being able to apply any zero offset.

When programming function G92, the axes do not move and the CNC assumes the axis values programmed after G92 as the new coordinates for those axes.

Programming format: N4 G92 X Z .

Example: X axis programmed in diameter and starting point: P0 (X20 Z80)



Without using function G92:

N100 G01 G90 X40 Z50
N110 Z10

Using function G92:

N90 G92 X20 Z0 (Point P0 now becomes point X20 Z0)
N100 G90 X40 Z-30
N110 Z-70

The block containing G92 cannot have any other function.

The coordinate preset by G92 always refers to the current theoretical position of the axes.

3.3 *POLAR ORIGIN PRESET (G93)*

By means of function G93, it is possible to preset any point as the origin of polar coordinates.

There are two ways to preset a polar origin.

a) By setting the coordinates of the polar origin.

Format N4 G93 I+/-4.3 K+/-4.3 in mm
N4 G93 I+/-3.4 K+/-3.4 in inches

N	Block number
G93	Polar origin presetting code
I	Abscissa value of the polar origin, X value (always an absolute value).
K	Ordinate value of the polar origin, Z value (always an absolute value).

When using this polar origin presetting method, the CNC does not admit any other information in the same block.

b) By assuming the current point as the new polar origin.

If in any block, a G93 is also programmed, it will mean that the new origin point will be the current position **before** moving the axes to the position programmed in that block.

Atention:

When programming a circular interpolation with G02, G03, the CNC assumes the center of the arc as the new polar origin.

On power-up or after an M02, M30, EMERGENCY or RESET, the CNC assumes the X0,Z0 point as the polar origin.

3.4 *ZERO OFFSETS (G53...G59)*

By means of functions G53, G54, G55, G56, G57, G58 and G59, it is possible to operate with 7 different zero offsets. These offset values are stored in the CNC memory and are referred to machine reference zero (home).

To access the zero offset table (G53-G59), press [AUX] [3] [1]

Once the table is displayed, it is possible to clear all the offsets by pressing:
[N] [ENTER]

Functions G53-G59 must be used to load a zero offset into the table or to apply one of them on the running program.

Loading a zero offset into the table.

Absolute value loading. To load the values set by X, Z into the desired table address (G53-G59).

Format: N4 G53-G59 X+/-4.3 Z+/-4.3 in mm,
N4 G53-G59 X+/-3.4 Z+/-3.4 in inches.

N Block number
G Zero offset code (G53 through G59)
X Zero offset value, referred to home, for the X axis
Z Zero offset value, referred to home, for the Z axis

Incremental value loading. To increment the existing zero offset values (G53/G59) by a set amount (I, K).

Format: N4 G53-G59 I+/-4.3 K+/-4.3 in mm,
N4 G53-G59 I+/-3.4 K+/-3.4 in inches.

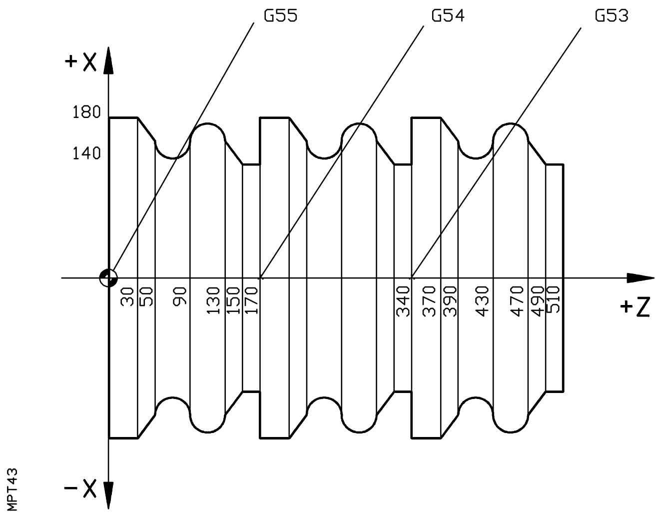
N Block number
G Zero offset code (G53 through G59)
I Amount to be added to the previously stored X axis zero offset value
K Amount to be added to the previously stored Z axis zero offset value

Apply a particular pre-defined zero offset onto the running program.

Format: N4 G53-G59

It applies a zero offset, according to the values stored at the indicated table address (G53-G59) onto the program currently in execution.

Example:



Assuming that the tool is positioned at X200 Z530, the X axis is programmed in radius and home is at X0 Z0, the path will be programmed as follows:

```
N10   G53 X0 Z340              Load zero offset into the table
N20   G54 X0 Z170              Load zero offset into the table
N30   G55 X0 Z0                Load zero offset into the table
N40   G90 G01 F250
N50   G53                      Apply this particular zero offset
N60   X140 Z170
N70   Z150
N80   X160 Z130
N90   G03 X160 Z90 I0 K-20
N100  G08 X160 Z50
N110  G01 X180 Z30
N120  Z0
N130  X140
N140  G54                      Apply this particular zero offset
N150  G25 N60.130.1
N160  G55                      Apply this particular zero offset
N170  G25 N60.120.1
N180  G00 X200 Z530
N190  M30
```

3.5 STORE AND RETRIEVE CURRENT PART ZERO (G31, G32)

G31 : Store current Part Zero.
G32 : Retrieve Part Zero previously stored by means of G31.

With G31, it is possible to store the current part zero at any time and retrieve it later on by using function G32.

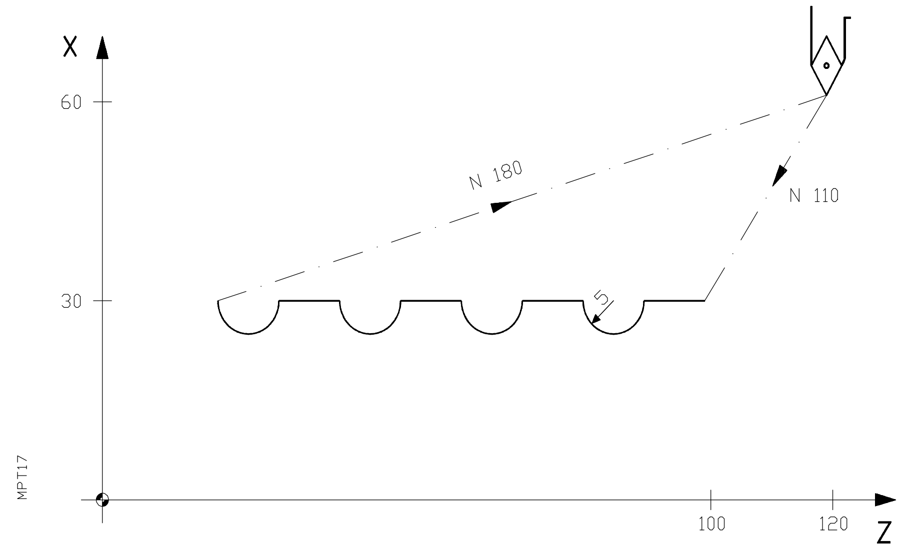
This feature may prove useful whenever it is necessary to utilize several part zeros in the same program since it allows to dimension part of the program with respect to one zero, store it with G31, change the part zero with G92 or G53-G59, dimension it with respect to the new part zero and, finally, recover the initial part zero with G32.
Functions G31 and G32 must be programmed alone in the block. Their format is:

N4 G31
N4 G32

Where:

N4	Block number
G31	Store current Part Zero
G32	Retrieve Part Zero previously stored by means of G31.

Example:



X axis programmed in diameter. Starting point X120 Z120.

N110 X60 Z100	(Approach the part)
N120 G31	(Store part zero)
N130 G92 X0 Z0	(Zero offset)
N140 G01 X0 Z-10	(Machining operation)
N150 G02 X0 Z-20 R5	(Machining operation)
N160 G25 N130.150.3	(Machining operation)
N170 G32	(Retrieve original part zero)
N180 G00 X120 Z120	(Return to starting point)

4. OTHER FUNCTIONS

4.1 FEEDRATE PROGRAMMING (F)

The meaning of "F" (programmable feedrate) varies depending on whether the machine operates in G94 or in G95 and also depending on whether mm or inches are being used. The table below shows the differences.

Millimeters

	Format	Programming units	Minimum values	Maximum values
G94	F4	F1 (1mm/min)	F1 (1mm/min)	F9999 (9999mm/min)
G95	F3.4	F1 (1mm/rev)	F0.0001 (0.0001mm/rev)	F500 (500mm/rev)

Inches

	Format	Programming units	Minimum values	Maximum values
G94	F4	F1 (0.1inch/min)	F1 (0.1inch/min)	F3937 (393.7inch/min)
G95	F2.4	F1 (1inch/rev)	F0.0001 (0.0001inch/rev)	F19.685 (19.685inch/rev)

The maximum actual feedrate of the machine may be limited by a lower value (see the instruction manual of the machine).

The maximum machining value may be programmed directly by using the F0 code.

Example: On a machine whose maximum programmable feedrate is 10,000 mm/min, it may be programmed as F10000 or F0.

The programmed "F" value becomes effective when performing a linear interpolation G01 or a circular one G02/G03.

When the "F" function is not programmed, the CNC will assume "F0".

When positioning in rapid, G00, the axes will move at its maximum speed regardless of the programmed "F" value.

This maximum "rapid positioning" feedrate is set for each axis during machine set-up and its maximum value cannot exceed 65.535m/min or 2580 inches/min. (see the instruction manual of the machine).

The programmed feedrate may be varied either between 0% and 120% or between 0% and 100%, depending on the setting of machine parameter P600(3), by means of the Feedrate Override Switch on the operator panel of the CNC as long as **neither** a threading operation (G33, G86, G87) **nor** a probing movement **is** taking place (G57).

4.1.1 FEEDRATE "F" IN mm/min or tenths-of-an-inch/min (G94)

From the moment G94 is programmed, the CNC assumes the programmed feedrate values (F4) to be in mm/min or 0.1inch/min units (F10 = 1 inch/min).

Function G94 is modal which means that once programmed, it remains active until G95 is programmed.

4.1.2 FEEDRATE IN mm/rev or inches/rev (G95)

From the moment G95 is programmed, the CNC assumes that all the programmed feedrate values are in mm/rev (3.4) or in inches/rev units (F2.4).

The maximum programmable values are 500mm/rev (F500) and 19.685 inches/rev (F19.685).

Function G95 is modal which means that once programmed, it remains active until G94 is programmed.

4.1.3 PROGRAMMABLE FEEDRATE OVERRIDE (G49)

By means of function G49, it is possible to indicate the override % to be applied on to the programmed feedrate.

When G49 is active, the Manual Feedrate Override switch of the operator panel has no effect. The programming format is: G49 K (1/120)

"G49 K" is followed by the % amount to be applied as feedrate override which must be an integer between 1 and 120.

Function G49 is modal; that is, once the % has been programmed, it is maintained until another % value is programmed or this function is cancelled.

To cancel this function, program either "G49 K" or G49 alone.

This function is also cancelled when executing an M02, M30, RESET or EMERGENCY.

Function G49 K must be programmed alone in the block.

4.2 SPINDLE SPEED AND SPINDLE ORIENTATION (S)

The "S" code has two meanings:

a) Spindle speed

The spindle speed is programmed directly in rpm or mm/min (ft/min) by means of the "S4" code. It is programmed in m/min (ft/min) when operating at Constant Surface Speed.

Any integer value (non-decimal) between S0 and S9999 may be programmed. This maximum value is limited by the actual maximum value set for each particular machine by the corresponding machine parameter.

Consult the instructions manual for your particular machine.

The programmed spindle speed may be varied between 50% and 120% by the corresponding keys of the CNC front panel as long as no threading operation is taking place (G33, G86 or G87).

When operating in G96, at Constant Surface Speed, the possible "S" values are:

S0-S3047 (0 m/min., 3047 m/min.).
S0-S9999 (0 ft/min., 9999 ft/min.).

b) Spindle orientation

When programming S4.3 after M19, the S4.3 code means the spindle orientation in degrees as counted from the spindle reference pulse (home). The CNC will output an analog voltage set by machine parameters P606(2) and P702 until the spindle reaches the angular position indicated by S4.3.

4.2.1 SPINDLE "S" IN rev./min (G97)

From the moment G97 is programmed, the CNC assumes the programmed values (S4) to be in rpm.

If when programming G97 no spindle speed is programmed, the CNC assumes the current spindle speed as the programmed value. Function G97 is modal and is cancelled by G96.

On power-up, the CNC assumes function G97.

4.2.2 CONSTANT SURFACE SPEED "S" IN m/min or feet/min (G96)

From the moment G96 is programmed, the CNC assumes the spindle speed (S4) to be in m/min (feet/min) and the lathe goes into Constant Surface Speed mode of operation.

The CNC assumes the currently selected spindle speed range (gear) as the active one. If none is selected, the desired one must be programmed in the same block (M41, M42, M43, M44).

If no range is either selected or programmed in the block containing G96, the CNC will issue error 10. This error will not be issued when the machine only uses a single spindle range (gear). In that case the CNC executes function M41 to select it.

It is recommended to program both G96 and the spindle speed (S4) in the same block.

If only G96 is programmed, the CNC assumes as the CSS speed the last CSS speed used in that operating mode. If there is no previous CSS value, the CNC issues error 10.

If the first movement after a G96 is made in rapid (G00), the CNC calculates the spindle rpm using the final diameter of that movement as part diameter.

If the first movement after a G96 is made in G01, G02 or G03, the CNC calculates the spindle rpm using the diameter value when G96 was executed.

G96 is modal and, thus, will remain active until G97 is programmed.

4.2.3 SPINDLE SPEED "S" LIMIT FOR CONSTANT SURFACE SPEED (G92)

By using G92, it is possible to limit the maximum spindle speed when operating at Constant Surface Speed (in G96).

By programming a "N4 G92 S4" type block, the spindle **rpm** is limited to the indicated "S4" value.

The CNC calculates, at all times, the required spindle rpm for the programmed CSS value (m/min or inches/min).

If the calculated spindle rpm exceed the "G92 S4" limit, the CNC will set the actual spindle speed to this maximum value.

4.3 TOOL PROGRAMMING (T)

The tool to be used is programmed by means of code: T2.2

The two digits to the left of the decimal point indicate the tool number to be selected.

The two decimals indicate the tool offset number to be applied.

Up to 32 tools (T1 through T32) and 32 tool offsets (Txx.1 through Txx.32) may be programmed.

The "T" function may be programmed in the following ways:

T2.2 The CNC selects the indicated tool and assumes the values of the indicated tool offset.

T2 The CNC selects the indicated tool and assumes the values of the same offset number as the indicated tool number. For example, "T19.19" may also be programmed as "T19".

T.2 The CNC does not change the tool but assumes the values of the indicated tool offset number.

The CNC always applies the tool length values (X, Z, I, K) stored in the tool offset table.

When programming G41 or G42, the CNC applies the corresponding "R" value stored in the tool offset table.

If no "T" has been programmed, the CNC applies the "T00" code corresponding to a tool with zero dimensions.

Each address of the tool offset table (01 through 32) contains the following fields:

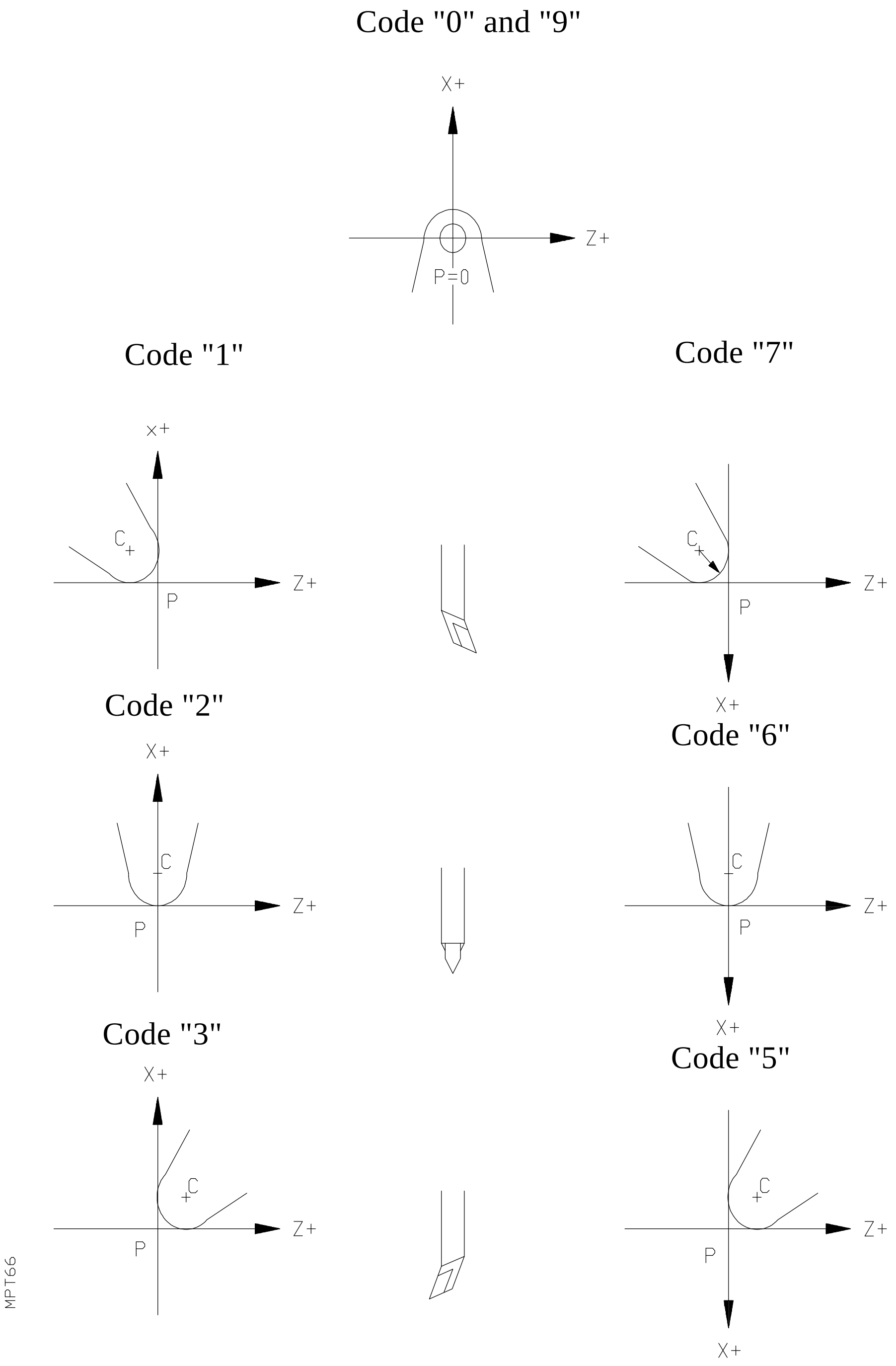
X Tool length along the X axis .... ±8388.607 mm (±330.2599 inches)
Z Tool length along the Z axis .... ±8388.607 mm (±330.2599 inches)
F Location code (tool shape) ....F0 thru F9 (see figure)
R Tool radius .... 1000.000 mm (39.3700 inches)
I Tool wear along the X axis (**always in diameter**). ±32.766 mm (±1.2900")
K Tool length wear along the Z axis................ ±32.766 mm (±1.2900 inches)

Atention:

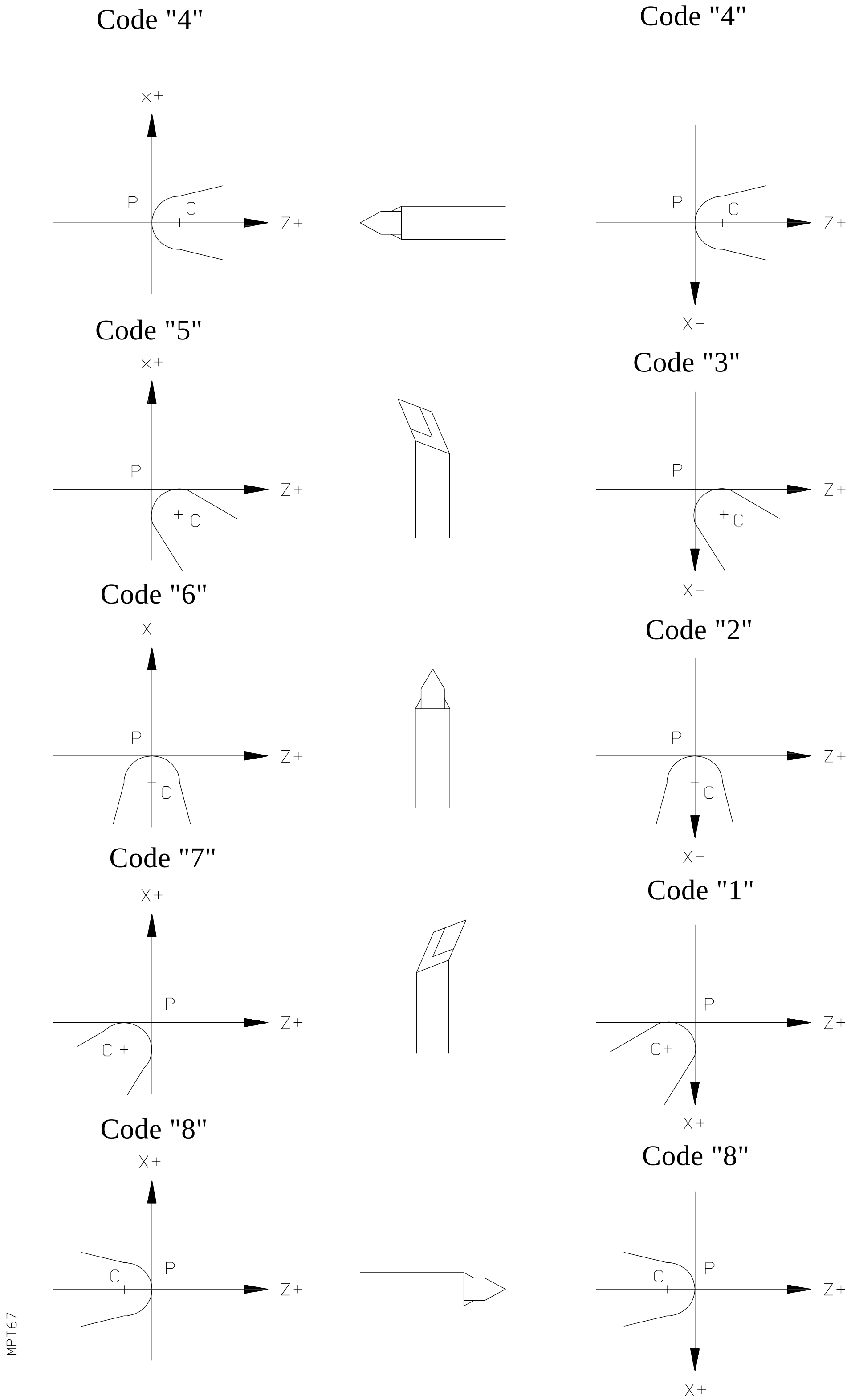
If the machine manufacturer has associated a subroutine with the T function, nothing must be programmed after the "T" function. Otherwise, the CNC will issue the corresponding error message.

If the tool change has no subroutine associated with it (manufacturer), the CNC outputs the code for the new tool, displays the message: "TOOL CHANGE" (in English for all language versions) and it interrupts program execution.

LOCATION CODES (TOOL SHAPE)



P: Tool Tip
C: Tool Center



P: Tool Tip
C: Tool Center

4.3.1 LOADING TOOL DIMENSIONS INTO TOOL TABLE (G50)

By means of function G50, it is possible to enter or edit the dimensions of the various tools in the tool offset table.

G50 may be programmed in several ways:

a) Loading all the dimensions of a tool

By programming N4 G50 T2 X±4.3 Z±4.3 F1 R4.3 I±2.3 K±2.3

The values set by X, Z, F, R, I, K are loaded into the table address indicated by T2.

N4	Block number
G50	Tool dimensions loading code
T	Tool table address (T01-T32)
X	Tool length along the X axis
Z	Tool length along the Z axis
F1	Location code (tool shape - F0 through F9)
R	Tool radius
I	Tool wear along the X axis (always in diameter)
K	Tool wear along the Z axis

The new X, Z, F, R, I, K values replace the previous ones at position T2.

b) Changing one or more table values.

To do this, just program those values following "G52 T2", the rest of the values remain unchanged.

When programming this way, bear in mind the following considerations:

* If "X" or "Z" or both are programmed without "I" or "K;" the previous tool length values (X, Z) are replaced by the new ones and their corresponding wear values (I or K or both) are set to zero.

* If either "I±2.3" or "I±2.3 K±2.3" is programmed after "G50 T2", these values are added to or subtracted from their previous values.

Atention:

The block containing function G50 cannot have any other type of information.

4.3.2 CORRECTING TOOL DIMENSIONS (G51)

By means of function G51, it is possible to alter the "I, K" values of the active tool (currently in use) without changing the values stored in the tool offset table.

Format: N4 G51 I±4.3 K±4.3 in mm
N4 G51 I±3.4 K±3.4 in inches

N4 Block number
G51 Modifying code
I Value to be added to or subtracted from the "I" value currently applied to the current tool length.
K Value to be added to or subtracted from the "K" value currently applied to the current tool length.

These values do not alter the tool table which means that the next time the same tool is used, the CNC will assume, again, the "I, K" values of the table ignoring the correction values programmed by G51.

A block containing function G51 cannot have any other type of information.

4.4 *MISCELLANEOUS FUNCTIONS (M)*

The auxiliary functions are programmed by means of the "M2" code.

Up to 96 different auxiliary functions (M00 through M99) may be programmed except for M41, M42, M43, M44 implicit with the "S" code if machine parameter "P601(1)=1". If this parameter is **not** set to "1", M41, M42, M43, M44 **must be** programmed. The auxiliary functions are output in BCD code.

The CNC also has **15** decoded outputs for miscellaneous functions. These outputs will be assigned to the required functions during the final adjustment of the CNC to the machine.

The miscellaneous functions which are not assigned a decoded output are always performed at the beginning of the block in which they are programmed.

In assigning a decoded output to a miscellaneous functions, a decision is also made as to whether it is to be performed at the beginning or at the end of the block in which it is programmed. Up to a maximum of seven miscellaneous functions may be programmed in one block.

When more than one miscellaneous function is programmed in a block, the CNC executes them consecutively in the order in which they are programmed.

Some of these miscellaneous functions have an internal meaning assigned to them in the CNC.

M00. PROGRAM STOP

When the CNC reads code **M00** in a block, it halts the program. Press the **Cycle Start** key to continue.

It is recommended that this function be set in the table of decoded **M** functions so that it is executed at the end of the block in which it is programmed (see Installation and Start up Manual).

M01. CONDITIONAL PROGRAM STOP

Same as **M00** except that the CNC only takes it into account if the conditional stop input (block skip) is activated.

M02. END OF PROGRAM

This code indicates end of program and performs a general reset function of the CNC (reset to initial conditions). It also acts as an **M05**.

As in the case of **M00**, it is recommended that this function be set so that it is executed at the end of the block in which it is programmed.

M30. END OF PROGRAM WITH RETURN TO THE BEGINNING

Same as **M02** except that the CNC goes back to the first block at the beginning of the program. It also acts as an **M05**.

M03. CLOCKWISE START OF THE SPINDLE

It is recommended that this function be set so that it is executed at the beginning of the block in which it is programmed.

M04. COUNTER-CLOCKWISE START OF THE SPINDLE

Same as **M03** except that the spindle rotates in the opposite direction.

M05. SPINDLE STOP

It is recommended that this be set so that the CNC executes it at the end of the block in which it is programmed.

M10, M11. ASSOCIATED TO EXTERNAL DEVICE O1.
M12, M13. ASSOCIATED TO EXTERNAL DEVICE O2.
M14, M15. ASSOCIATED TO EXTERNAL DEVICE O3.

Codes associated to the keys corresponding to the external devices "O1", "O2" and "O3". Codes M10, M12 and M14 activate their corresponding outputs and codes M11, M13 and M15 deactivate them.

M19. SPINDLE ORIENTATION

If M19 S4.3 is programmed, the spindle will rotate at the speed and direction set by machine-parameters P606(2) and P706 and will stop at the point identified by S4.3, in degrees, referred to the machine reference zero (home).

When the spindle is positioned within the dead band (P707), the CNC sends out the spindle locking signal (decoded M15) and the spindle is kept in closed loop; applying the gain as per P708, min. and an analog voltage as per P709.

When programming in a block M19 S4.3, more information is not allowed in that block.

M20. END OF PART-PROGRAM EXECUTION

This code indicates that the part execution has ended. For example, on a machine with a bar feeder, the PLC could control the machining of several parts in a row by using this function.

M41,M42,M43,M44 SPINDLE RANGE SELECTION

If "P601(1) = 1", these codes are automatically generated by the CNC, when an **S** function is programmed. If this parameter is set to **"0", M41,M42,M43** and **M44** must be programmed.

When operating at constant surface speed (G96) these functions must necessarily be programmed even if P601(1) is set to 1.

M45. SELECTION OF ROTATION SPEED OF THE LIVE TOOL

Programming format: **N4 M45 S±4**

S±4 defines the direction and rotation speed of the live tool.

The ± sign defines the direction of turn. With **S+4**, it will turn in one direction and with **S-4** will turn in the opposite direction.

The programmable value range is between **S0** and **S9999** (0-9999 rpm.).

5. *PATH CONTROL*

5.1 ROUND CORNER (G05)

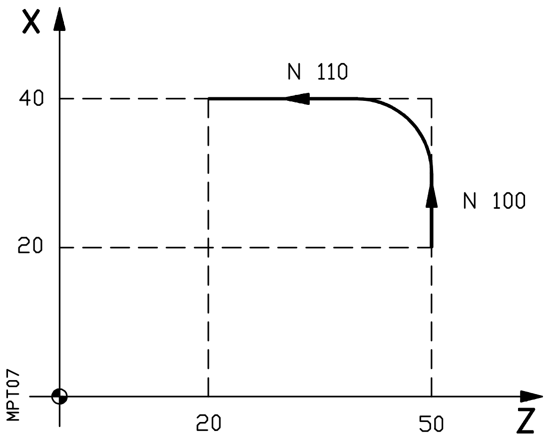
When operating in G05, the CNC starts executing the next block of the program as soon as the axes start decelerating to reach the target position programmed in the previous block.

Therefore, the movements programmed in the next block are executed **before** the axes reach their target position programmed in the previous block.

Example:

X programmed in diameter
Starting point: X40 Z50.

N100 G90 G01 G05 X80
N110 Z20



As seen in the example, the corners appear rounded.

The difference between the theoretical path and the real one depends on the actual axis feedrate.

The greater the feedrate, the greater the difference is between the theoretical and real paths (greater radius).

Function G05 is modal and incompatible with G07. Function G05 may also be programmed as G5.

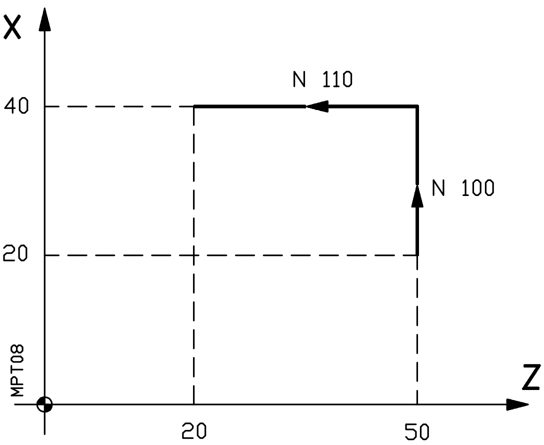
5.2 SQUARE CORNER (G07)

When operating in G07, the CNC starts executing the next block of the program once the axes have reached their target position programmed in the previous block.

Example:

X axis programmed in diameter.
Starting point: X40 Z50.

N100 G90 G01 G07 X80
N110 Z20



The theoretical and real path are the same.

Function G07 is modal and incompatible with G05. Function G07 may also be programmed as G7.

5.3 RAPID POSITIONING (G00)

Movements programmed after G00 are carried out at the maximum speed set by the corresponding machine parameters.

When two axes move simultaneously, the resulting path is a straight line from the beginning point to the end point at the feedrate of the slower axis.

When programming G00, the last programmed F is not canceled. Therefore, when programming G01, G02 or G03 afterwards, the CNC recovers that F and applies it.

Machine parameter P4 determines whether the Manual Feedrate Override (MFO) is ignored being fixed at 100% or not, thus being able to apply between 0% and 100% override.

The G00 code "freezes" the current tool radius compensation (G41, G42 without effect) until a G01, G02 or G03 is programmed again.

"G00" is modal and incompatible with G01, G02, G03 and G33.

"G00" may also be programmed as "G" or "G0".

5.4 LINEAR INTERPOLATION (G01)

When executing a G01, the programmed axes move in a straight line from the starting point to the end point at the programmed Feedrate "F".

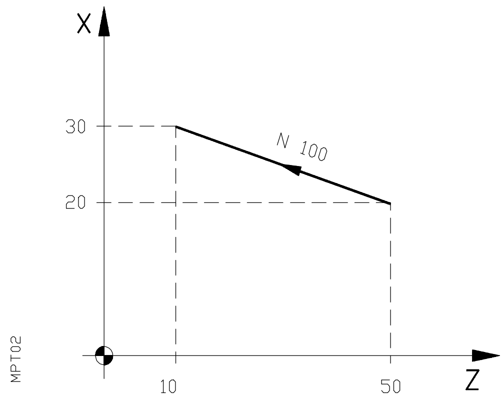
The CNC calculates the feedrate for each axis so the resulting straight path is carried out at the programmed "F".

Example:

X axis programmed in diameter.

Starting point: X40 Z50

N100 G90 G01 X60 Z10 F300



By means of the "MFO" on the CNC keyboard, it is possible to override the programmed "F" between 0% and 120% or between 0% and 100% depending on the setting of machine parameter P600(3).

If machine parameter "P600(3)=0", while keeping the rapid jog key: pressed, the CNC applies 200% of the programmed "F" to a G01 move.

"G01" is modal and incompatible with G00, G02, G03 and G33.

"G01" may also be programmed as "G1".

On CNC power-up, after executing an M02/M30, after an EMERGENCY or RESET, the CNC assumes "G01".

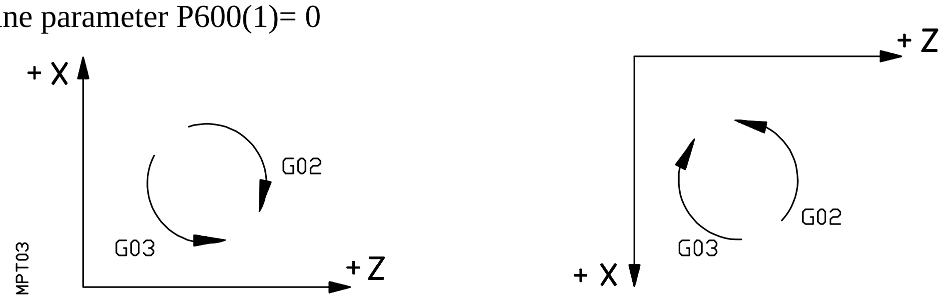
5.5 CIRCULAR INTERPOLATION (G02, G03)

G02: Clockwise circular interpolation.
G03: Counter-clockwise circular interpolation.

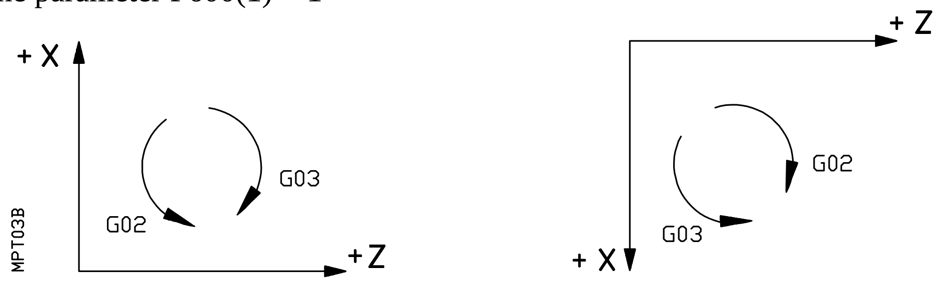
Movements programmed after a G02 or G03 follow a circular path at the programmed feedrate.

The clockwise (G02) and counter-clockwise (G03) directions have been defined according to the following criteria:

a) Machine parameter P600(1)= 0



b) Machine parameter P600(1) = 1



"G02" and "G03" are modal and incompatible with each other as well as with G00, G01 and G33.

Functions G74, G75, or any canned cycle cancel G02 and G03.

"G02" and "G03" may also be programmed as "G2" and "G3".

The cartesian coordinate format for a block containing a circular interpolation is:

N4 G02 (G03) X±4.3 Z±4.3 I±4.3 K±4.3

N4 : Block number
G02 (G03) : Interpolation code.
X±4.3 : X coordinate of the end point of the arc.
Z±4.3 : Z coordinate of the end point of the arc.
I±4.3 : Distance from the starting point to the center of the arc along the X axis.
K±4.3 : Distance from the starting point to the center of the arc along the Z axis.

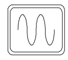
The "I" and "K" values are programmed with a sign and they must always be programmed even when their values are "0".

The polar coordinate format for a block containing a circular interpolation is:

N4 G02 (G03) A±3.3 I±4.3 K±4.3

N4 : Block number.
G02 (G03) : Interpolation code.
A±3.3 : Angle with respect to the polar center of the arc's end point.
I±4.3 : Distance from the starting point to the center of the arc along the X axis.
K±4.3 : Distance from the starting point to the center of the arc along the Z axis.

When a circular interpolation is programmed with G02 or G03, the CNC assumes the arc center as the new polar origin. In this case, even when the X axis is programmed in diameter, **the "I" value must always be given in radius.**

If machine parameter "P600(3)=0", while keeping the rapid jog key:  pressed, the CNC applies 200% of the programmed "F" to a G02/G03 move.

5.5.1 CIRCULAR INTERPOLATION BY PROGRAMMING THE ARC RADIUS

Format In mm: G02 (G03) X±4.3 Z±4.3 R±4.3
In inches: G02 (G03) X±3.4 Z±3.4 R±3.4

Where: G02(G03) Circular interpolation code
X X coordinate of the end point
Z Z coordinate of the end point
R Radius of the arc

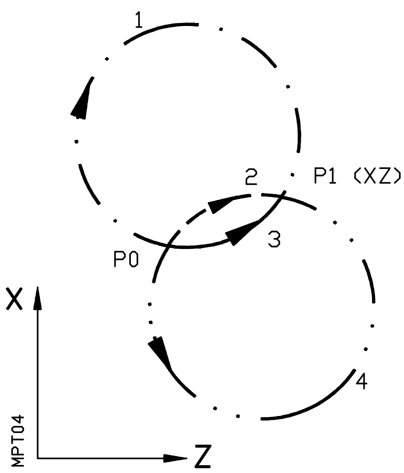
This means that the circular interpolation may be programmed by means of the end point and the arc radius instead of the arc center coordinates (I, K).

When the arc is smaller than 180º, the radius must be programmed with a positive sign and with negative sign if otherwise.

Considering P0 as the starting point of the arc and P1 as the end point, four different arcs may be defined with the same radius.

Depending on the direction of the circular interpolation G02 or G03 and the sign of the radius, the resulting arc programming formats are:

Arc 1 G02 X Z R -
Arc 2 G02 X Z R +
Arc 3 G03 X Z R +
Arc 4 G03 X Z R -



Atention:

If a complete circle is programmed using any of these four formats, the CNC will issue error 47 indicating that there are infinite solutions.

5.5.2 CIRCULAR INTERPOLATION WITH ABSOLUTE CENTER COORDINATES (G06)

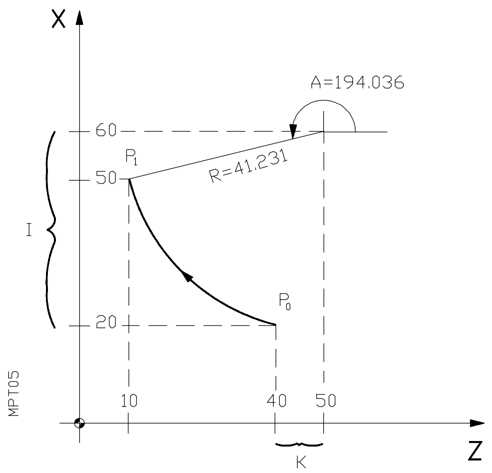
By adding function G06 to a block containing a circular interpolation, it is possible to use absolute coordinates for the arc center (I, K); that is, with respect to the part zero and not referred to the arc's starting point.

"G06" is NOT MODAL. Therefore, it must be programmed every time the arc center coordinates are to be absolute.

When using "G06", the "I" value must be programmed either in radius or diameter depending on the setting of machine parameter "P11".

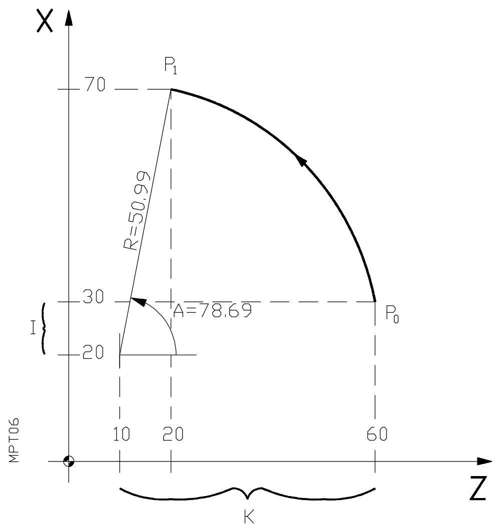
5.5.3 PROGRAMMING EXAMPLES

Assuming absolute coordinate programming (G90) and the X axis in diameter.



Starting Point P0 (X40 Z40)

Cartesian coordinates
N4 G02 X100 Z10 I40 K10
Polar coordinates
N4 G02 A194.036 I40 K10
Radius programming
N4 G02 X100 Z10 R41.231
Programming with G06
N4 G02 G06 X100 Z10 I120 K50



Starting point P0 (X60 Z60)

Cartesian coordinates
N4 G03 X140 Z20 I-10 K-50
Polar coordinates
N4 G03 A78.69 I-10 K-50
Radius programming
N4 G03 X140 Z20 R50.99
Programming with G06
N4 G03 G06 X140 Z20 I40 K10

5.6 ARC TANGENT TO PREVIOUS PATH (G08)

By means of function G08, it is possible to program an arc tangent to the previous path without having to program its center coordinates (I, K).

The cartesian coordinate format is:

N4 G08 X±4.3 Z±4.3 in mm
N4 G08 X±3.4 Z±3.4 in inches

N4	Block number.
G08	Code for the arc tangent to previous path.
X	X coordinate of the end point of the arc.
Z	Z coordinate of the end point of the arc.

The polar coordinate format is:

N4 G08 R±4.3 A±4.3 in mm
N4 G08 R±3.4 A±4.3 in inches

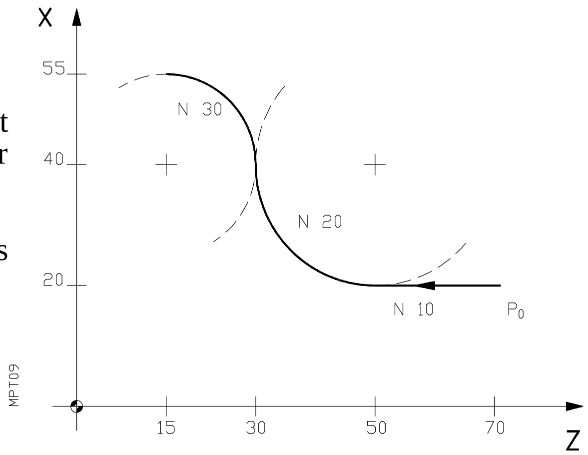
N4	Block number
G08	Code for the arc tangent to previous path.
R	Radius (with respect to the polar origin) of the arc's end point.
A	Angle (with respect to the polar origin) of the arc's end point.

Example:

X axis programmed in diameter.

Being "P0 (X40 Z70)" the starting point, a straight line followed by an arc tangent to it and another arc tangent to the previous arc would be programmed as follows:

```
N110 G90 G01 Z50
N120 G08 X80 Z30
N130 G08 X110 Z15
```



Since both arcs are tangential, it is not necessary to program their center coordinates (I, K).

Without using G08, the program looks like this:

```
N110 G90 G01 Z50
N120 G02 X80 Z30 I20 K0
N130 G03 X110 Z15 I0 K-15
```

"G08" is not modal, it only replaces G02 and G03 in the block containing it. It may be programmed whenever a tangential arc is desired. The previous path may be straight line or another arc.

Atention:

G08 cannot be used to draw a complete circle since there are infinite solutions. The CNC will issue error 47 when attempting to do so.

5.7 *ARC DEFINED BY THREE POINTS (G09)*

By means of function G09, it is possible to program an arc by defining the end point and an intermediate point (the starting point is the end point of the previous block or the current position of the axes).

In other words, any intermediate point may be programmed instead of the center coordinates (I, K).

The cartesian coordinate format is:

N4 G09 X±4.3 Z±4.3 I±4.3 K±4.3

N4	Block number
G09	Code for arc defined by three points
X	X coordinate of the arc's end point
Z	Z coordinate of the arc's end point
I	X coordinate of an intermediate point of the arc
K	Z coordinate of an intermediate point of the arc

The polar coordinate format is:

N4 G09 R±4.3 A±4.3 I±4.3 K±4.3

N4	Block number
G09	Code for arc defined by three points
R	Radius of the arc's end point (with respect to the polar origin).
A	Angle of the arc's end point (with respect to the polar origin).
I	X coordinate of an intermediate point of the arc
K	Z coordinate of an intermediate point of the arc

As can be observed here, the intermediate point must always be expressed in cartesian coordinates.

Example:

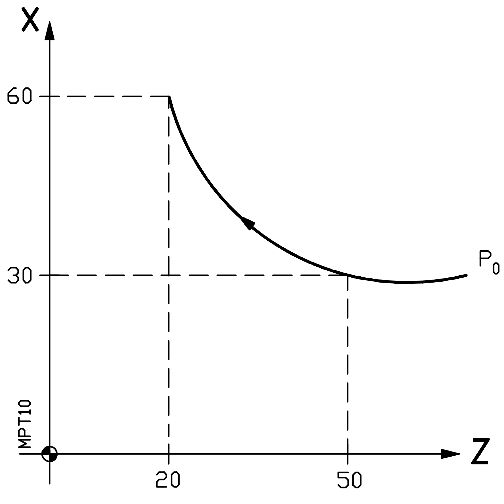
X axis programmed in diameter.

Starting point: P0 (X60 Z70) and end point of the arc: (X120 Z20), the block will look like this:

N4 G09 X120 Z20 I60 K50

"G09" is not modal. The direction of the arc (G02 or G03) is not required when programming G09.

"G09" only replaces "G02" and "G03" in the block containing it.



Atention:

A complete circle cannot be drawn by using "G09" since all three points must be different. The CNC will issue error 40 when attempting to do so.

5.8 TANGENTIAL ENTRY (G37)

By means of function "G37", it is possible to link two paths tangentially without having to calculate the intersection points.

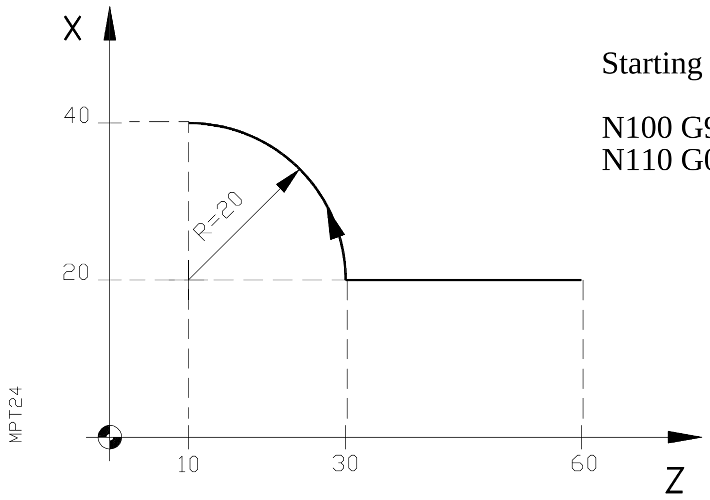
"G37" is not modal and, therefore, must be programmed every time the tangential link between two paths is desired. These paths may be: **straight**-to-straight or **straight**-to-curve. "G37" must be followed by the radius of the entry arc (R4.3 in mm or R3.4 in inches).

The radius value must always be positive.

"G37 R" must be programmed in the **linear** movement to be affected by it (either a G00 or a G01).

If "G37 R" is programmed in a circular interpolation (G02 or G03), the CNC will issue error 41.

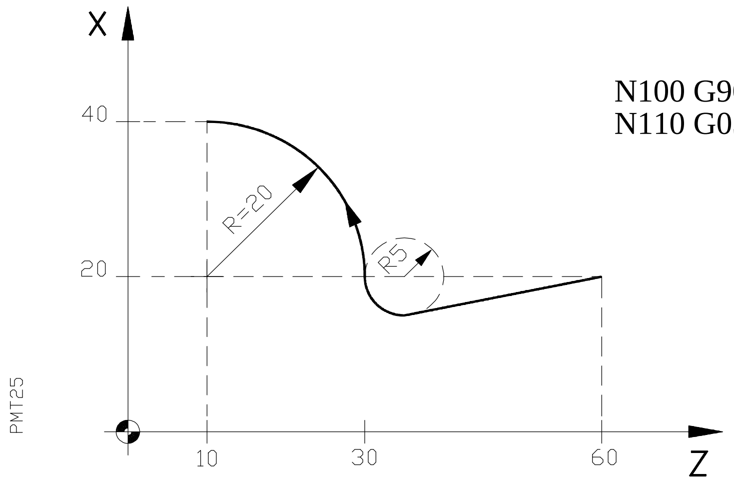
Example: X axis programmed in radius.



Starting point: X20 Z60

N100 G90 G01 X20 Z30
N110 G03 X40 Z10 R20

Same example applying a tangential entry with a 5mm radius:



N100 G90 G01 G37 R5 X20 Z30
N110 G03 X40 Z10 R20

5.9 *TANGENTIAL EXIT (G38)*

By means of function "G38", it is possible to link two paths tangentially without having to calculate the intersection points.

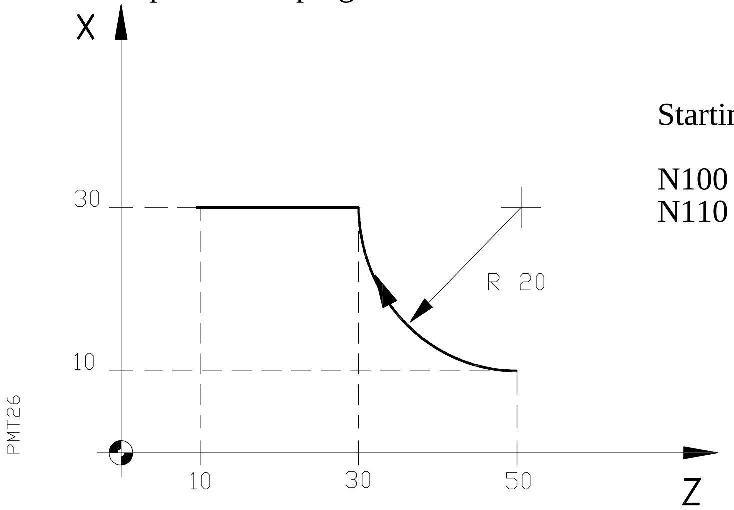
"G38" is not modal and, therefore, must be programmed every time the tangential link between two paths is desired. These paths may be: straight-to-**straight** or curve-to-**straight**. "G38" must be followed by the radius of the entry arc (R4.3 in mm or R3.4 in inches).

The radius value must always be positive.

"G38 R" must be programmed in the **block prior to the linear one** (either a G00 or a G01) being affected by it .

If the next programmed path, after the one containing "G38 R", is an arc (G02 or G03), the CNC will issue error 42.

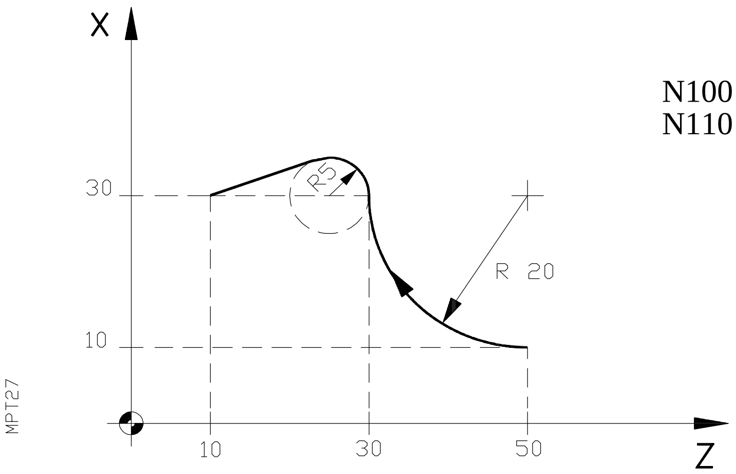
Example: X axis programmed in radius.



Starting point: X10 Z50

N100 G90 G02 X30 Z30 R20
N110 G01 X30 Z10

Same example applying a tangential exit with a 5mm radius:



N100 G90 G38 R5 G02 X30 Z30 R20
N110 G01 X30 Z10

5.10 *AUTOMATIC RADIUS BLEND (G36)*

On turning operations, it is possible to round corners (blend paths) with a particular radius without the need to calculate the center of the arc or its starting and end points, simply by using function G36.

"G36" is not modal and, therefore, must be programmed every time this feature is desired.

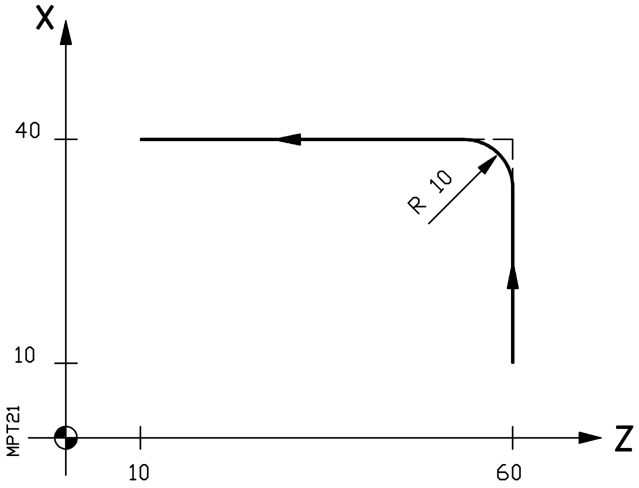
The blending radius is programmed by R4.3 in mm or R3.4 in inches and it must be a positive value.

Examples: The X axis is programmed in diameter.

1st radius blend. Straight-to-straight

Starting point: X20 Z60

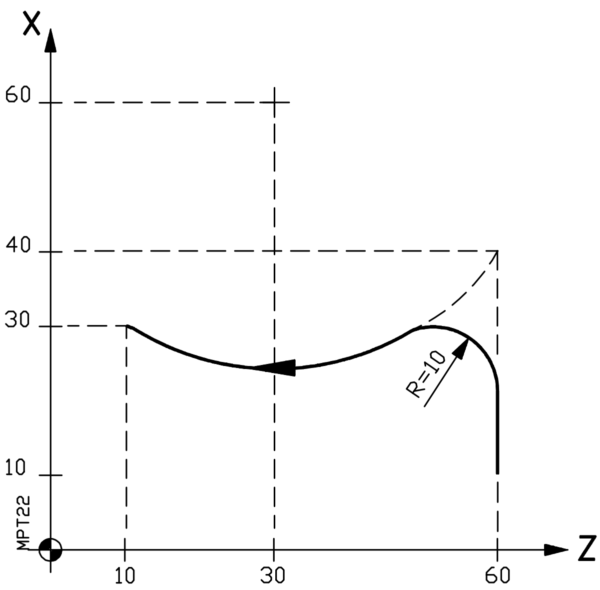
N100 G90 G01 G36 R10 X80
N110 Z10



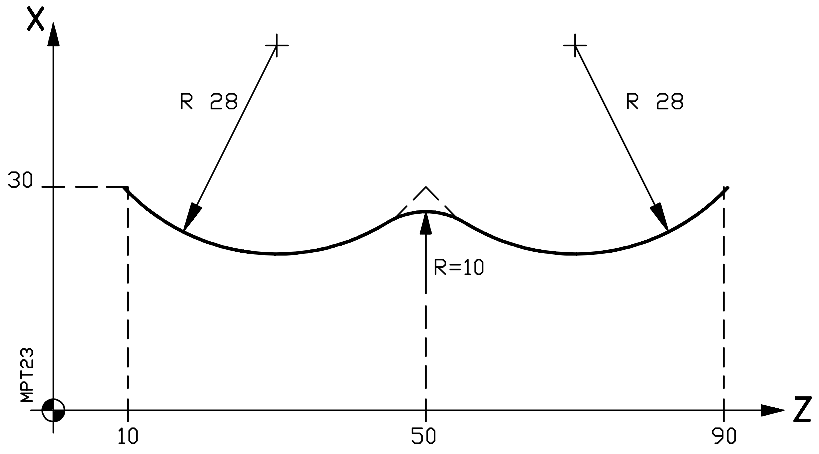
2nd radius blend. Straight-to-curve

Starting point: X20 Z60

N100 G90 G01 G36 R10 X80
N110 G02 X60 Z10 I20 K-30



3rd radius blend. Curve-curve



Starting point: X60 Z90

N100 G90 G02 G36 R10 X60 Z50 R28
N110 X60 Z10 R28

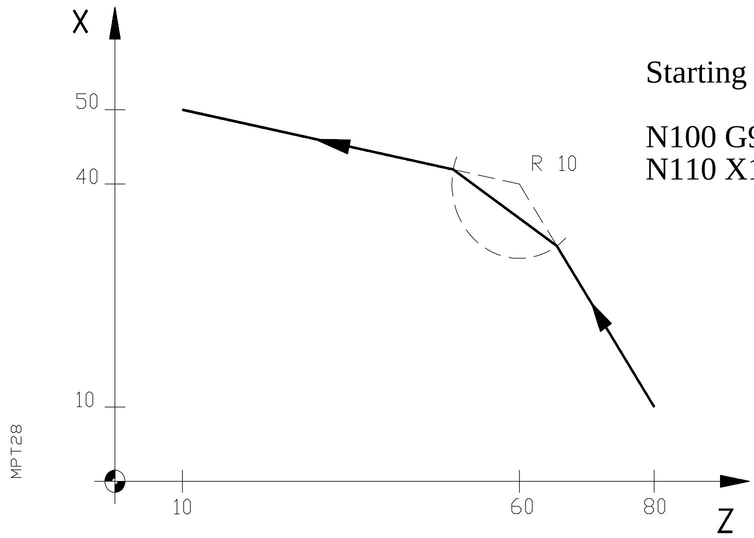
5.11 CHAMFER (G39)

On turning operations, it is possible to chamfer the intersection of two straight lines by using function G39 without having to calculate the intersection point.

"G39" is not modal and, therefore, must be programmed every time a chamfer is desired. This function must be programmed in the **block whose end** is to be chamfered.

A positive value of R4.3 in mm or R3.4 in inches, sets the chamfer distance from the intersection point.

Example: X axis programmed in diameter.



Starting point: X20 Z80

N100 G90 G01 G39 R10 X80 Z60
N110 X100 Z10

6. *ADDITIONAL PREPARATORY FUNCTIONS*

6.1 DWELL (G04)

By means of function "G04", it is possible to program a dwell (delay).

The amount of dwell time is indicated by the letter "K".

Example: G04 K0.05 => 0.05 second dwell
G04 K2.5 => 2.5 second dwell

When "K" is followed by a figure, it can be a value between 0 and 99.99 seconds whereas when followed by an arithmetic parameter (KP3) this parameter value may be between 0 and 655.35 seconds.

The dwell is executed at the beginning of the block containing it. "G04" may also be programmed as "G4".

6.2 DISPLAY ERROR CODE (G30)

As soon as the CNC reads a block containing the "G30" code, interrupts the running program and displays the indicated error.

Programming format: N4 G30 K2

N4	Block number
G30	Error programming code
K2(0-99)	Error code number to be displayed

The error code may also be programmed by an arithmetic parameter between P0 and P255. For example: N4 G30 KP123.

This code, in combination with G26, G27, G28 and G29, permits interrupting the program and detecting possible measuring errors, etc.

A block containing "G30" cannot have any other information.

Atention:

When wishing the CNC's own error code comments NOT to appear on the screen, the error code number to be displayed by "G30" must be greater than those used by the CNC.

Remember that the operator may write comments in the program and the CNC will display them when executing the corresponding block.

6.3 UNCONDITIONAL JUMP / CALL (G25)

Function "G25" may be used to jump from one block to another within the same program. The block containing "G25" may not contain any other information. There are two programming formats.

Format a) N4 G25 N4

N4 Block number
G25 Unconditional jump code
N4 Block number jumping to

When the CNC reads this block, the program "jumps" to the indicated block and continues its execution from there on.

Example:
```
N0 G00 X100
N5 Z50
N10 G25 N50
N15 X50
N20 Z70
N50 G01 X20
```

When reaching block N10, the CNC skips blocks N15 and N20 continuing from N50 to the end of the program.

Format b) N4 G25 N4.4.2

N4 Block number
G25 Unconditional jump code
N4.4.2
- Number of repetitions
- Number of the last block to be executed
- Number of the block jumping to

When the CNC reads a block of this type, jumps to the block defined between "N" and the first period.

Executes the program portion between this block and the one defined between the two periods as many times as indicated by the last figure.

This last figure may be between 0 and 99. However, if an arithmetic parameter is used to define it, its value may be between 0 and 255.

When writing only N4.4, the CNC will execute this program section only once (same as for: N4.4.1)

Once the CNC has completed the last repetition of the section, it returns to the block following the one containing "G25 N4.4.2 "

Example:

```
N0 G00 X10
N5 Z20
N10 G01 X50 M3
N15 G00 Z0
N20 X0
N25 G25 N0.20.8
N30 M30
```

When reaching "N25", the CNC jumps to "N0" and executes the section "N0 through N20" eight times. It, then, returns to "N30".

The preparatory functions corresponding to the conditional jumps/calls (G26, G27, G28, G29 and G30) will be described in the section regarding PARAMETRIC PROGRAMMING, OPERATIONS WITH PARAMETERS later on in this manual.

6.4 ELECTRONIC THREADING (G33)

By using function G33, it is possible to program and perform longitudinal, facial and taper threading. In order to be able to use this feature, a rotary encoder must be mounted on the spindle. "G33" is modal and, therefore, will remain active until canceled by G00, G01, G02, G03, M02, M30, EMERGENCY or RESET.

Longitudinal threading Its programming format is: N4 G33 Z±4.3 K3.4, where:

N4	Block number
G33	Threading code
Z	Final Z coordinate of the thread
K	Thread pitch along the Z axis

The Z coordinate will be either absolute or incremental depending on whether in "G90" or "G91". The axis feedrate cannot be overridden while in "G33" by means of the MFO switch and the CNC will apply 100% of the programmed "F". The spindle rpm cannot be overridden either by the spindle override keys.

Facial threading (Spiral) Its programming format is: N4 G33 X±4.3 I3.4, where:

N4	Block number
G33	Threading code
X	Final X coordinate of the thread
I	Thread pitch along the X axis

The X coordinate will be either absolute or incremental depending on whether in "G90" or "G91".

Taper threading Its programming format is: N4 G33 X±4.3 Z±4.3 I3.4 K3.4, where:

N4	Block number
G33	Threading code
X	Final X coordinate of the thread
Z	Final Z coordinate of the thread
K	Thread pitch along the Z axis
I	Thread pitch along the X axis

The X and Z coordinate will be either absolute or incremental depending on whether in "G90" or "G91".

On taper threading, it suffices to program the threading pith along one axis since the CNC calculates the pitch along the other axis. In other words, it may be programmed like this:

N4 G33 X±4.3 Z±4.3 I3.4 or like this: N4 G33 X±4.3 Z±4.3 K3.4

Nevertheless, both pitches (I, K) may be programmed in order to obtain taper threads with multiple pitches different from the ones the CNC would calculate.

Atention:

The threading operation must begin away from the part (with the tool "cutting air") in order to avoid possible problems originated when the axes start moving.

6.4.1 EXAMPLES

a) Longitudinal threading

2mm-deep cylindrical thread with a 5mm-pitch.

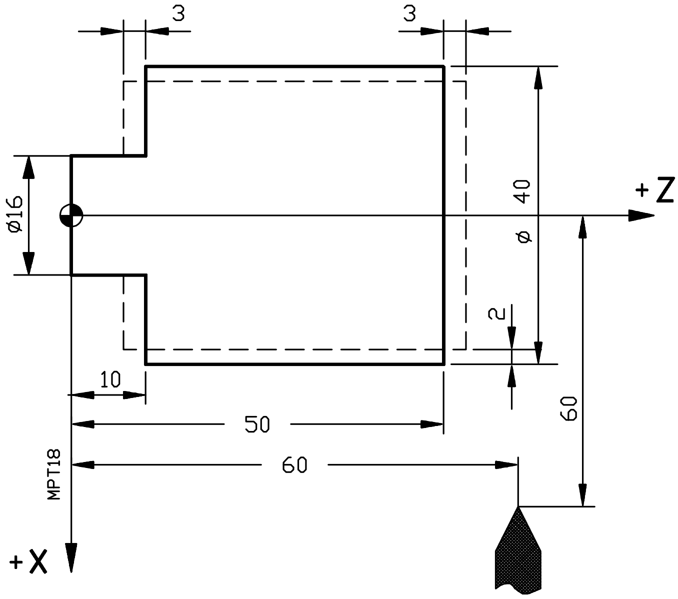
The current tool position is: X60 Z60 (X in radius)

Absolute coordinates

N0 G00 G90 X18 Z53
N5 G33 Z7 K5
N10 G00 X60
N15 Z60

Incremental coordinates

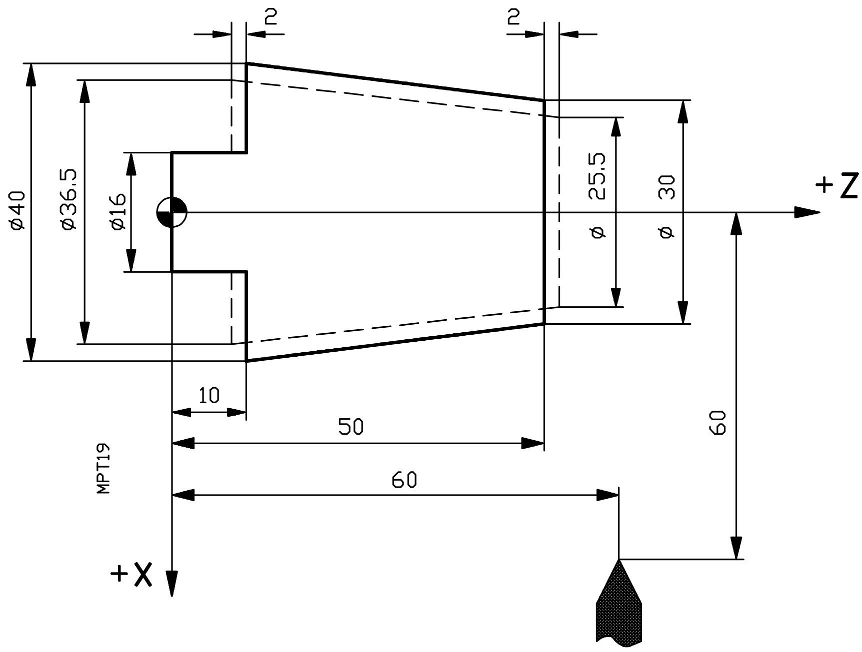
N0 G00 G91 X-42 Z-7
N5 G33 Z-46 K5
N10 G00 X42
N15 Z53



b) Taper threading

2mm-deep taper thread along the Z axis with a 5mm-pitch.

The current tool position is: X60 Z60 (X in radius)



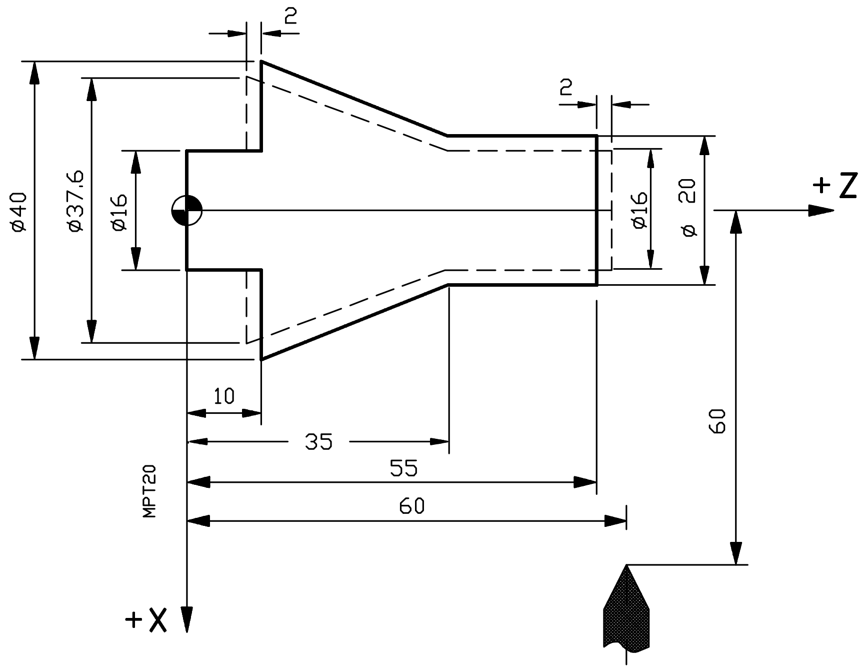
Absolute coordinates	Incremental coordinates
N0 G00 G90 X12.75 Z52	N0 G00 G91 X-47.25 Z-8
N5 G33 X18.25 Z8 K5	N5 G33 X5.5 Z-44 K5
N10 G00 X60	N10 G00 X41.75
N15 Z60	N15 Z52

c) Joining multiple threads

When operating in "round corner" (G05), different threads may be joined together on the same part.

Joining a longitudinal thread and a tapered one along the Z axis. Both 2 mm deep and with 5mm pitch.

The current tool position is: X60 Z60 (X in radius)



Absolute coordinates

N0 G00 G90 X8 Z57
N5 G33 G05 Z35 K5
N10 X18.8 Z8 K5
N15 G00 X60
N20 Z60

6.5 SCALING FACTOR (G72)

By using functions "G72", it is possible to enlarge or reduce programmed parts.

This way, it is possible to program several parts having the same shape but different sizes in a single program. "G72" must be programmed alone in the block.

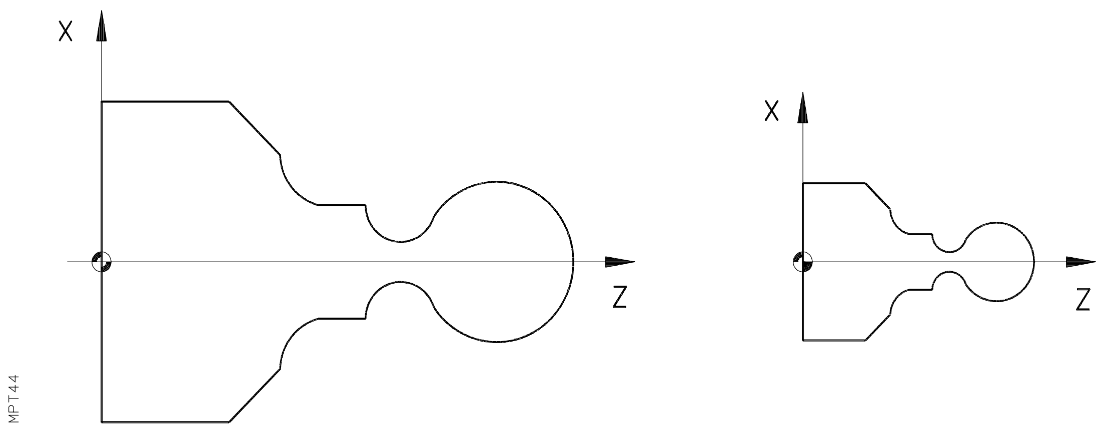
The programming format: N4 G72 K2.4

N	Block number
G72	Scaling factor code
K	Value of the scaling factor

Minimum value	K0.0001	(Multiplied by 0.0001).
Maximum value	K99.9999	(Multiplied by 99.9999).

After "G72", all the programmed coordinates will be multiplied by the indicated K value until another scaling factor is programmed or it is canceled.

To cancel the scaling factor, just program another one with a "K1" value. It is also canceled when executing an M2, M30, RESET or EMERGENCY.



6.6 PROBING (G75)

By means of this function, it is possible to use a touch probe connected to the CNC.

The programming format is:

N4 G75 X±4.3 Z±4.3 in millimeters.
N4 G75 X±3.4 Z±3.4 in inches.

The axes will move until the probe signal is received. Once it is received, the CNC will consider that the block has been executed (in position) and it will assume the current position of the axes (at the instant the probe signal was received) as their theoretical position. The probing feedrate cannot be overridden by the MFO switch and it will be set at 100% of the programmed value.

If the axes reach the programmed position before the CNC receives the probe signal, the CNC will issue error 65.

Once this block has been executed, the current position values of the axes may be assigned to arithmetic parameters. This, combined with the possibility to perform mathematic operations with the arithmetic parameters, permits editing special tool calibration and part measuring programs.

Function "G75" implies functions "G01" and "G40" which means that once "G75" is executed, the CNC assumes "G01" (linear interpolation) and "G40" (cancelation of tool radius compensation).

6.7 *SINGLE-BLOCK TREATMENT. ON (G47), OFF (G48)*

This CNC considers a "single block" the section of a program contained between a G47 and a G48.

After executing function G47, the CNC executes all the following blocks in a row until a G48 is detected.

When pressing the key while executing a "Single-Block" block, either in Automatic or block by block, the CNC goes on executing all the following blocks until a G48 is executed.

While function G47 is active, the Manual Feedrate Override switch as well as the spindle speed override keys are ignored, thus that section of the program is executed at 100% of the programmed "F" and "S".

Functions G47 and G48 are modal and incompatible with each other.

On CNC power-up, after executing an M02/M30, after an EMERGENCY or RESET, the CNC assumes G48.

7. *TOOL COMPENSATION*

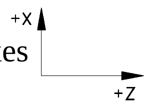
Usually, machining operations demand the calculation of the actual tool path taking its length and radius dimensions into account so the part ends up with the desired dimensions.

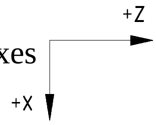
Using tool length and radius compensation makes it possible to program the part contour directly without worrying about tool dimensions since the CNC makes the necessary calculations based on the part contour and on the tool dimensions stored in the tool offset table.

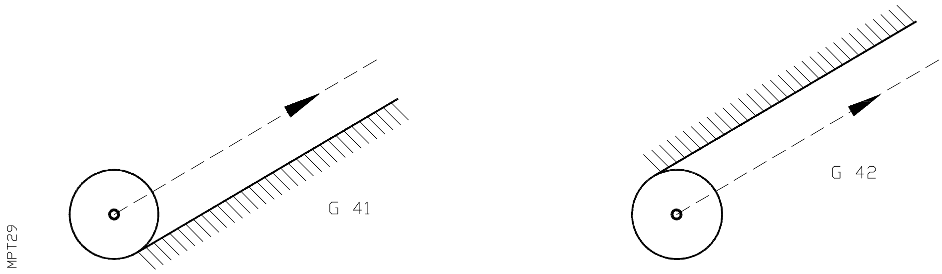
Every time a tool (T2) is selected, the CNC automatically applies its corresponding tool length compensation stored in the table (X, Z, I, K) without having to program any "G" code.

There are three "G" codes for tool radius compensation:

G40	Cancels tool radius compensation
G41	Applies left-hand tool radius compensation
G42	Applies right-hand tool radius compensation

Machine parameter P600(1) = 0 and axes 

or P600(1) = 1 and axes 

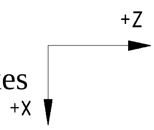


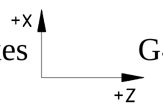
G41:

The tool stays to the left of the part in the machining direction.

G42:

The tool stays to the right of the part in the machining direction.

With P600(1) = 0 and axes 

or P600(1) = 1 and axes 

G41 acts like a G42 and vice versa.

The CNC offers a table of up to 32 tools for length (L) and radius (R) compensation as well as location codes (tool shape - F) for each tool.

To set these tool compensation values, use the "Tool Table" option of the "Auxiliary function" menu.

The tool table may also be set via part-program using function "G50".

The maximum values for each tool table field are:

X, Z (Tool length) ±8388.607 mm (±330.2599 inches)

I, K (Tool length wear) ±32.766 mm (±1.2900 inches)

R (Tool radius) 1000.000 mm (39.3700 inches)

F (Location or tool shape code). It must be defined for using tool radius compensation. Possible codes: F0 through F9 (see illustration on the next pages).

Tool radius compensation becomes active by programming "G41" or "G42" assuming the table value by means of the "T" function (T01 through T32). If no "T" has been programmed, the CNC assumes the "T00" value which corresponds to a tool with **no** dimensions (0).

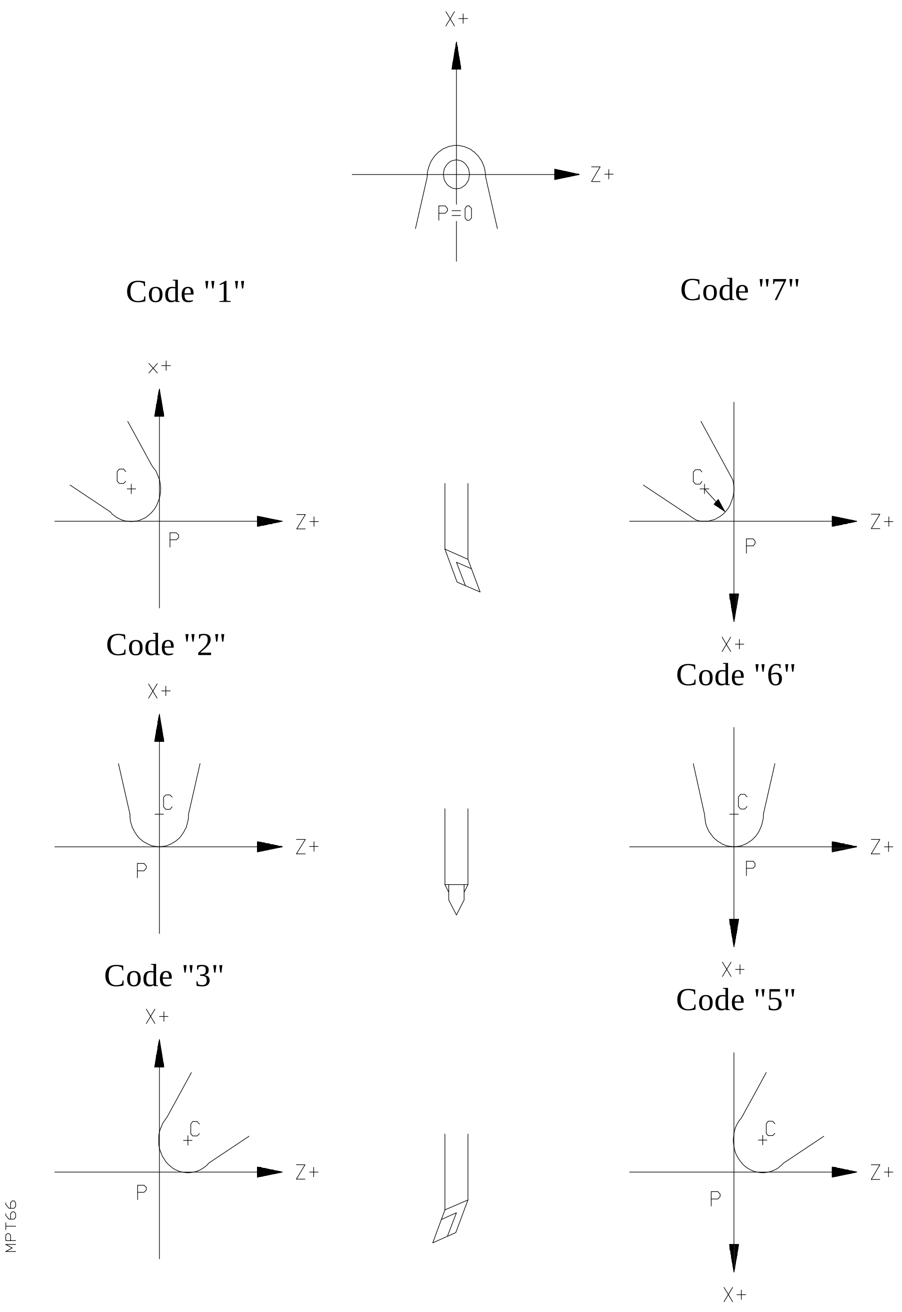
"G41" and "G42" are modal (maintained) and are canceled by G40, M02, M30, an EMERGENCY or a general RESET.

Atention:

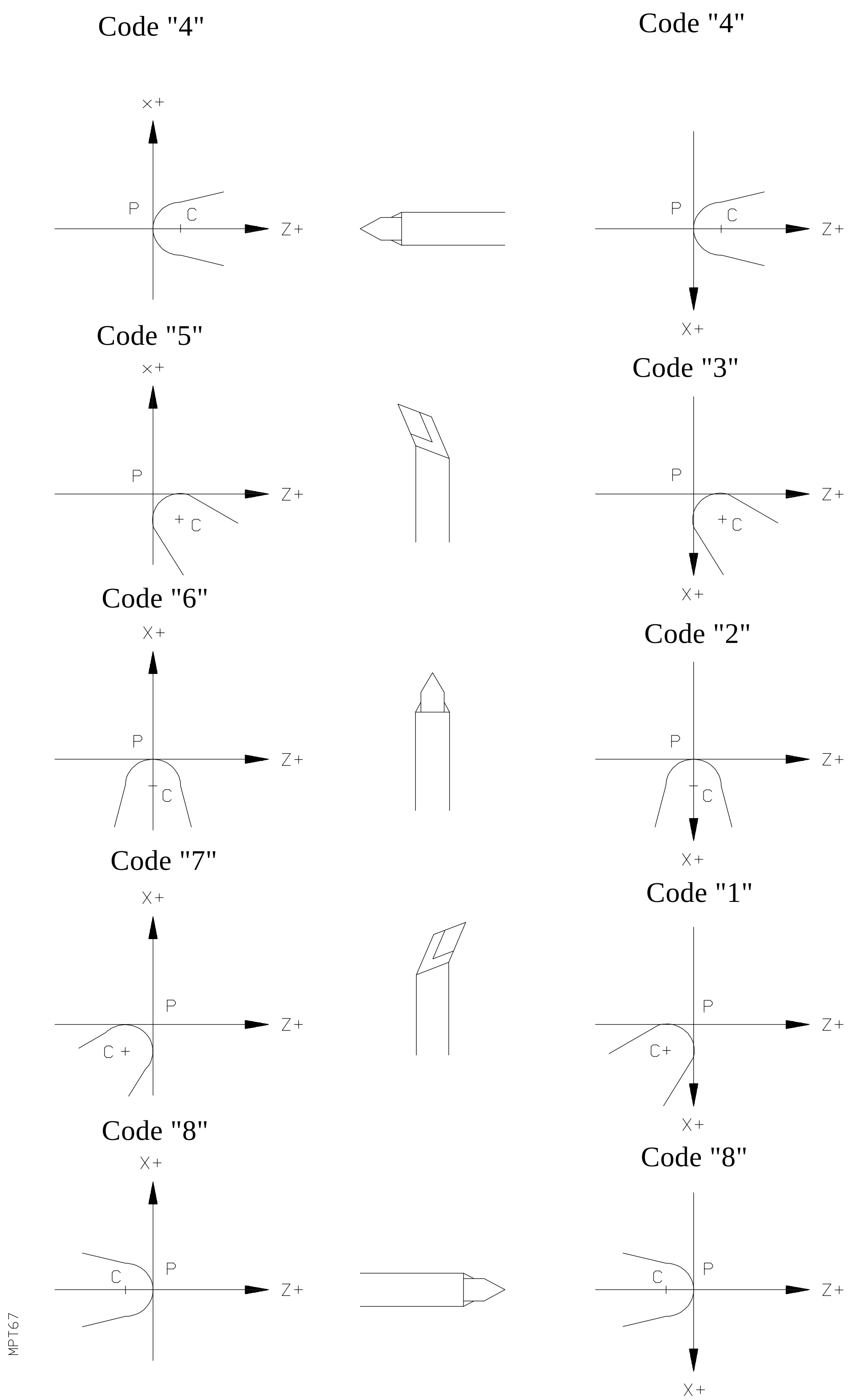
The "I" values for tool radius wear must always be programmed in diameter.

LOCATION CODES (TOOL SHAPE)

Code "0" and "9"



P: Tool tip
C: Tool center



P: Tool tip
C: Tool center

7.1 SELECT AND INITIATE TOOL RADIUS COMPENSATION (G41, G42)

To activate it, codes "G41" and "G42" must be used.

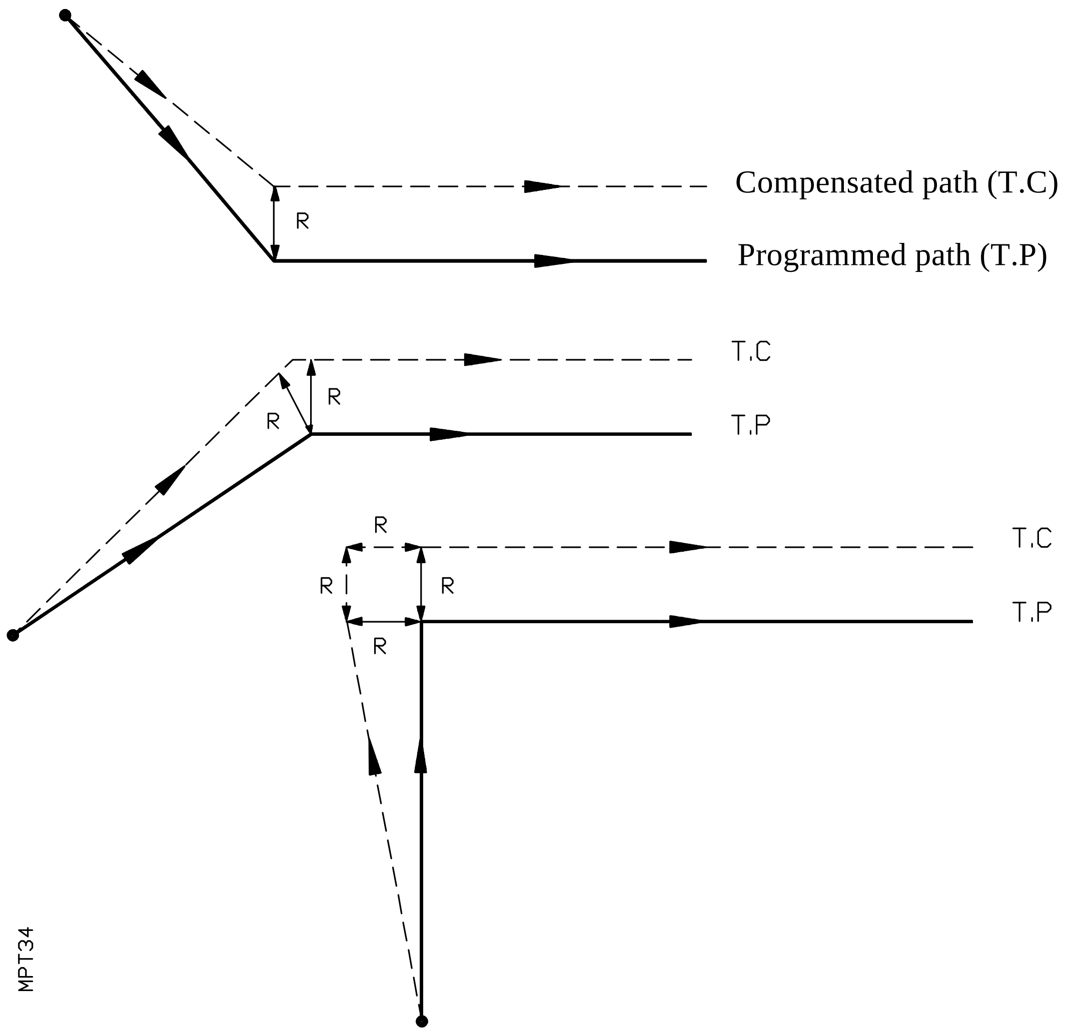
The block containing "G41" or "G42" or the previous one must contain the "T" function (T01 through T32) selecting the tool offset to be applied. If no tool has been selected, the CNC assumes "T00" with tool offset values equal to "0" (no dimensions).

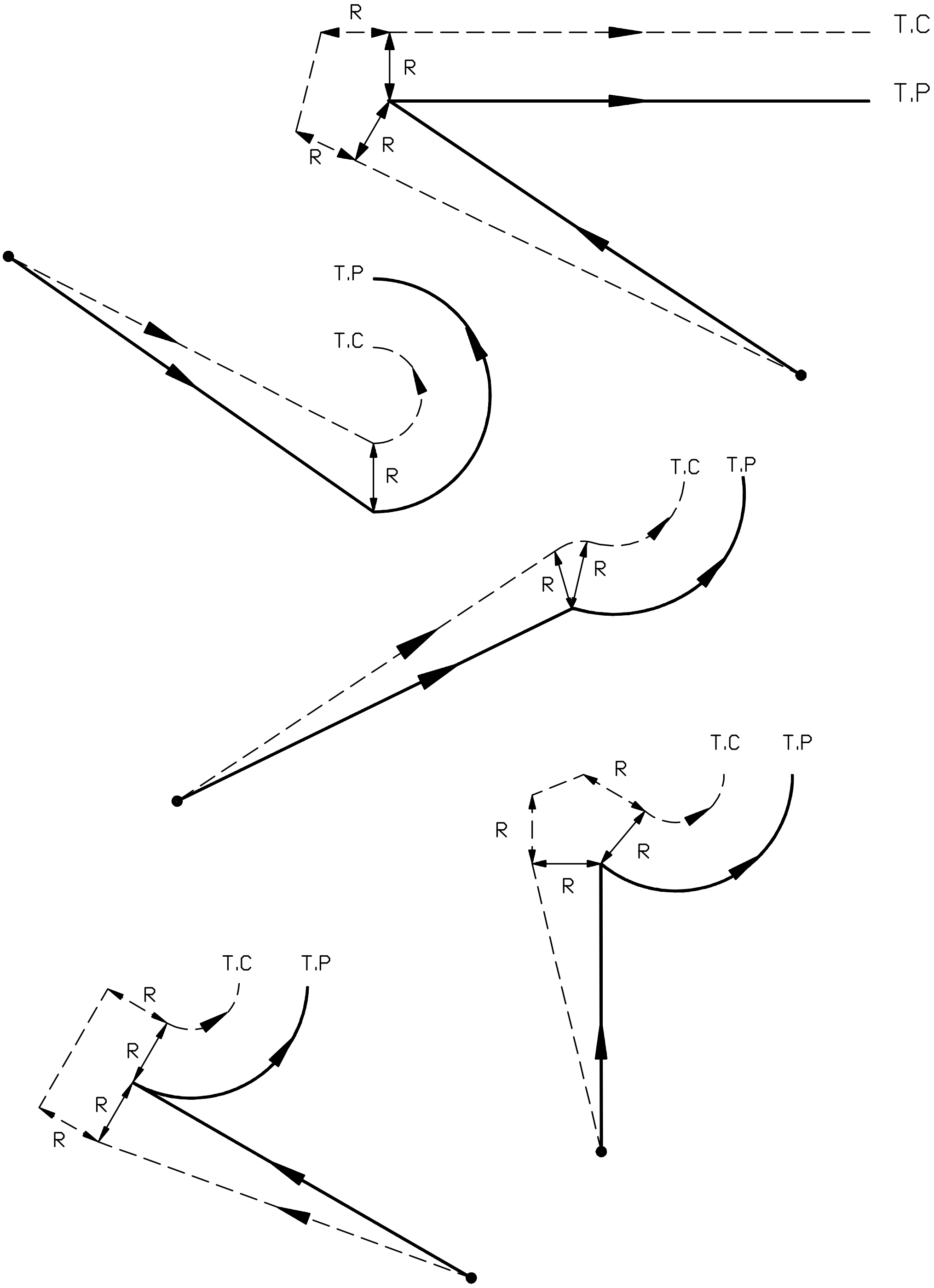
Atention:

"G41" or "G42" can only be selected on linear movements ("G00" or "G01" active).

If this type of compensation is first mentioned on a circular movement ("G02" or "G03"), the CNC will issue error 48.

The next pages show the various cases where tool radius compensation may be initiated.



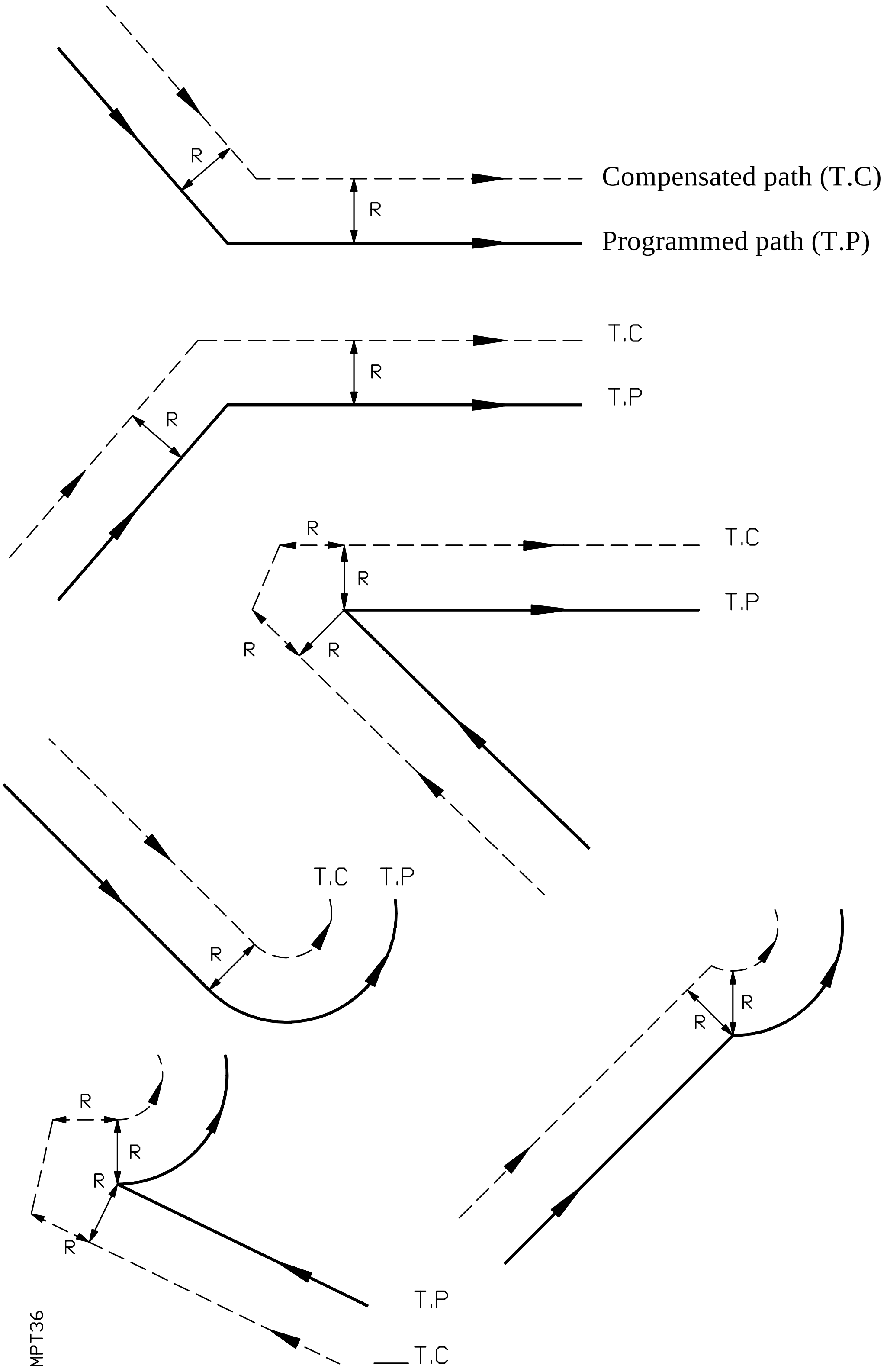


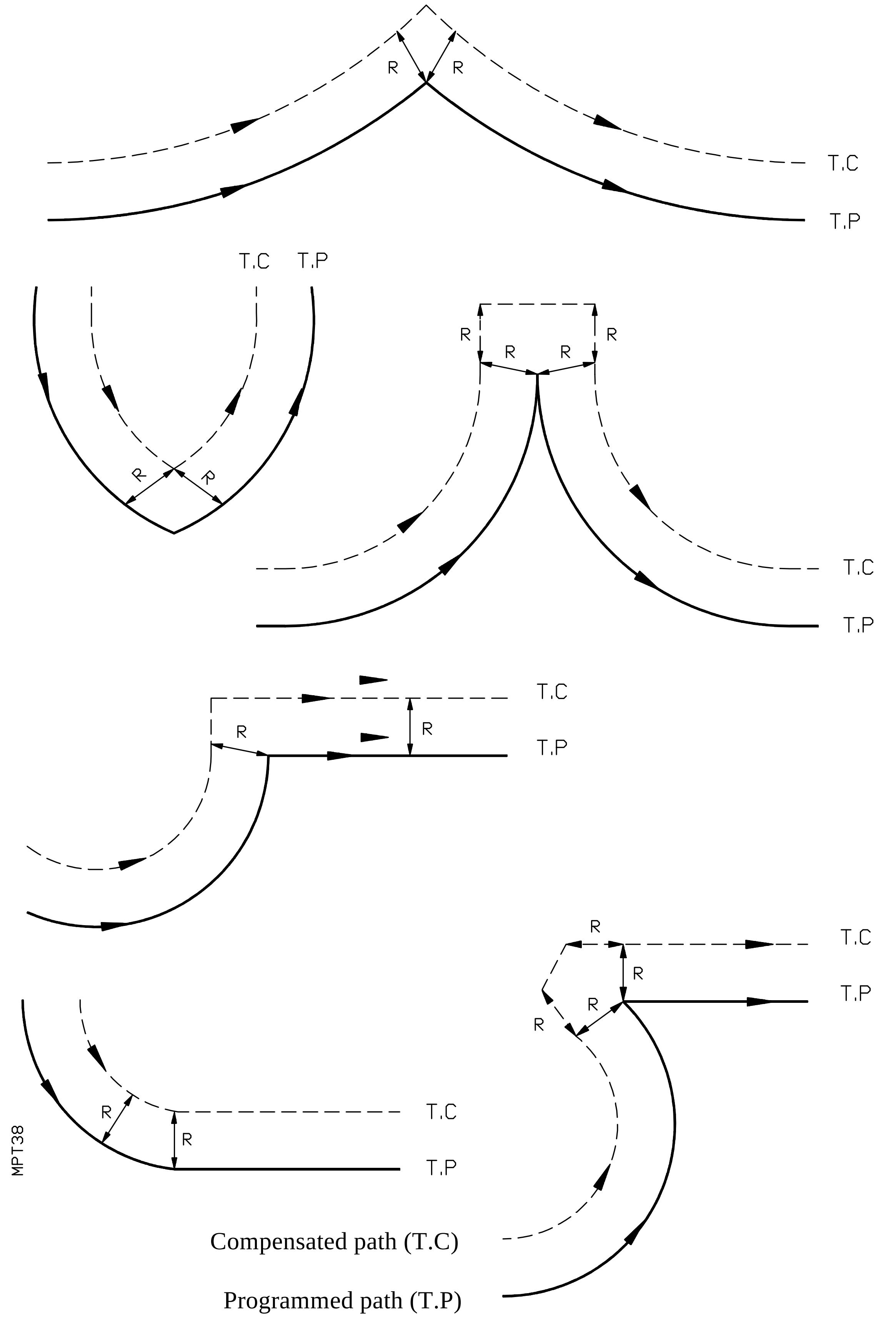
T.P = Programmed path

T.C. = Compensated path

7.2 OPERATION WITH TOOL RADIUS COMPENSATION

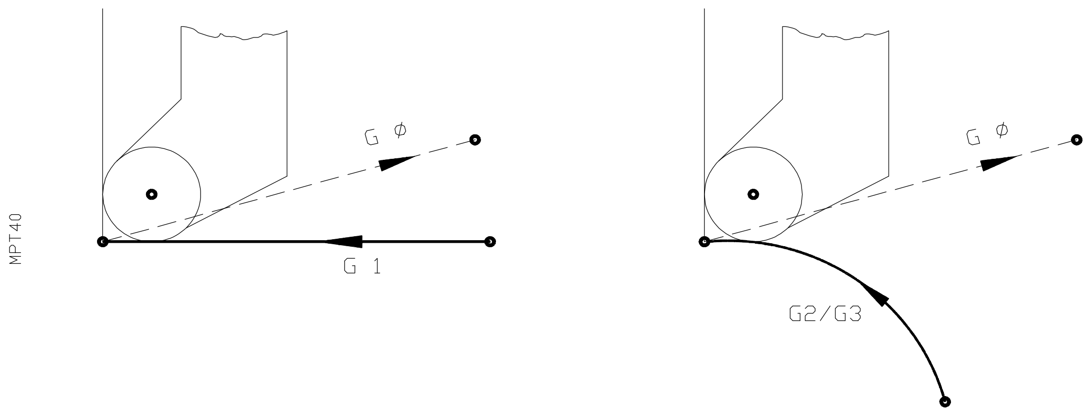
Next, some drawing are shown describing the various tool paths with and without tool radius compensation.




R
R
T.C
T.P
T.C
T.P
R
R
R
R
R
R
T.C
T.P
T.C
R
R
T.P
R
R
T.C
R
R
T.P
R
R
T.C
T.P
MPT38
Compensated path (T.C)
Programmed path (T.P)

7.3 TEMPORARY TOOL RADIUS CANCELLATION WITH G00

When switching from "G01", "G02", or "G03" to "G00", the tool stays tangent to the perpendicular at the end of the movement programmed in the block containing "G01", "G02" or "G03".



The same thing happens when programming a block containing "G40" but without movement data. The following movements in "G00" are carried out without radius compensation.

When switching **from "G00" to** "G01", "G02" or "G03", tool radius compensation is reactivated.

Special case: If the CNC does not have enough data for compensating; but the movement is in G00, it will execute it without radius compensation.

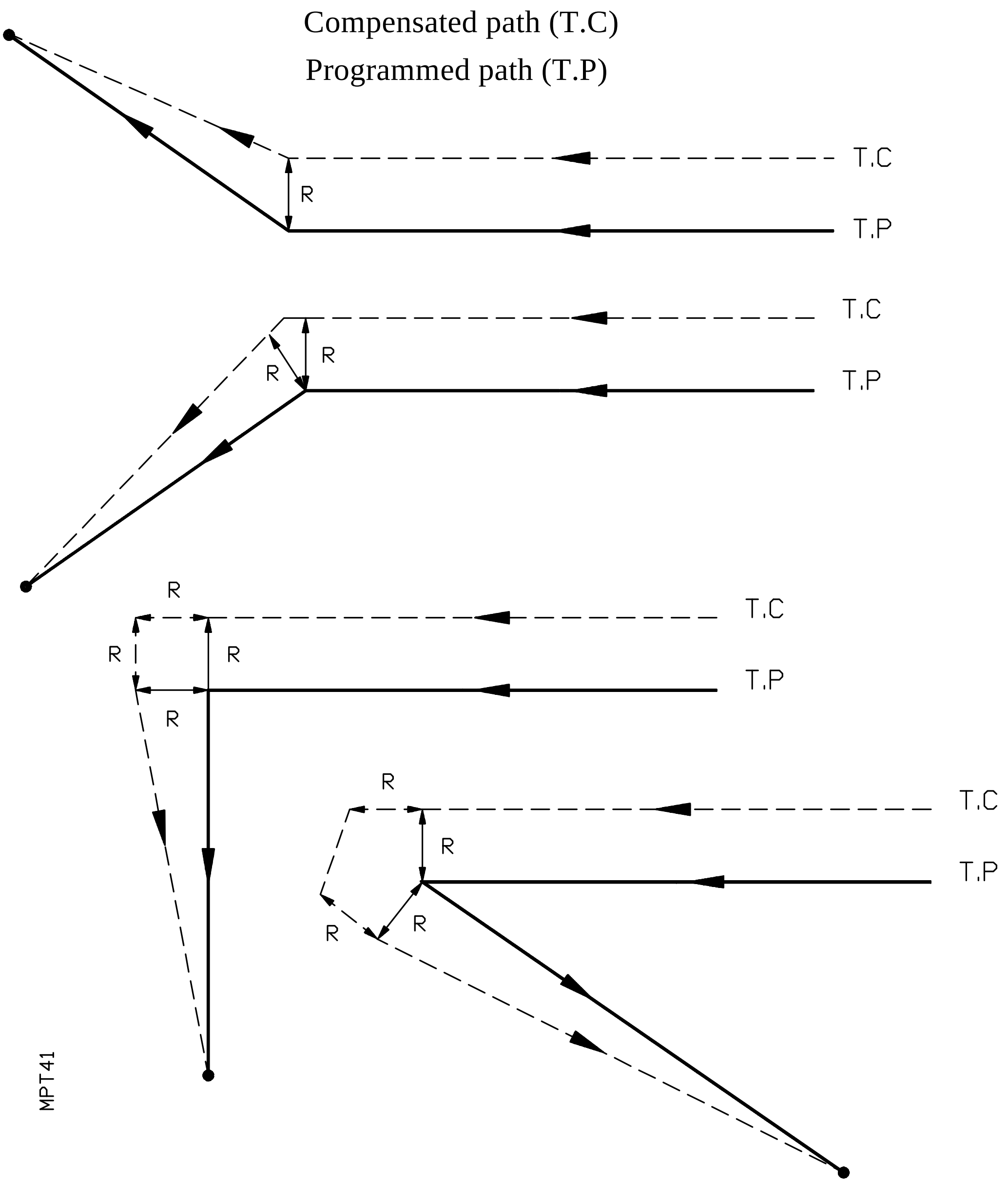
7.4 CANCELLATION OF TOOL RADIUS COMPENSATION (G40)

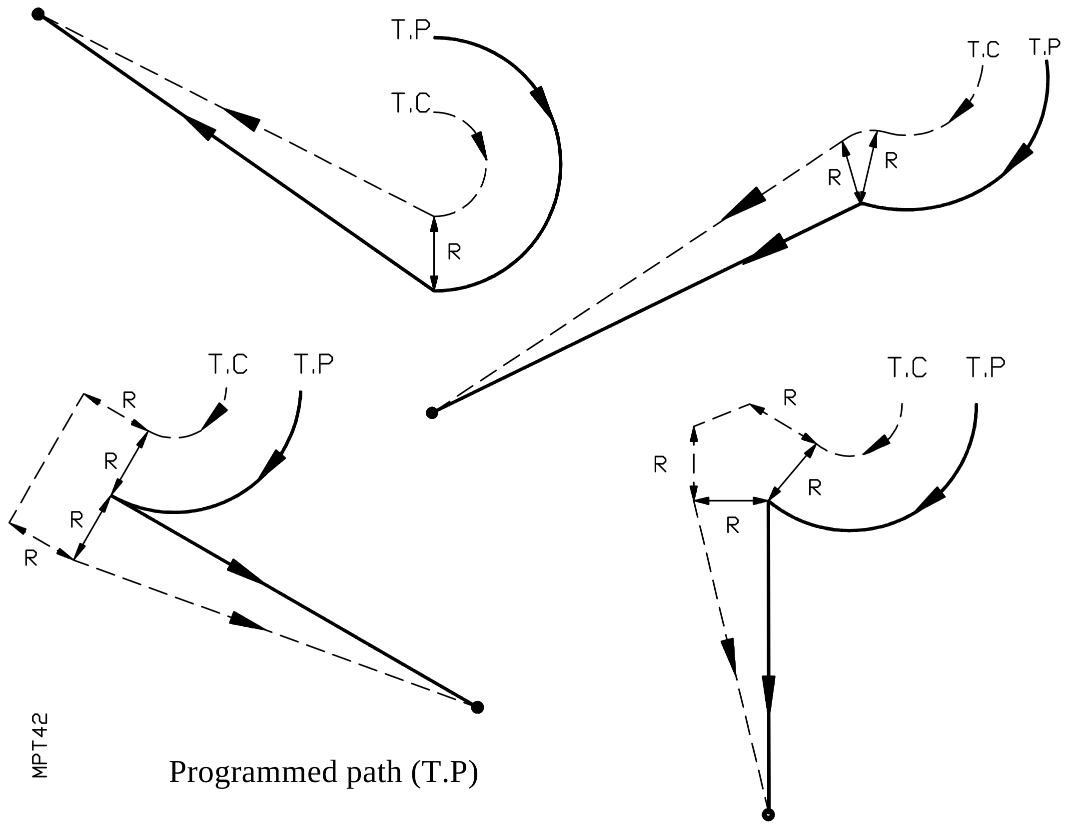
Tool radius compensation is canceled by function "G40".

Bear in mind that "G40" can only be programmed in a linear motion block (G00 or G01).

If "G40" is programmed in a circular motion block (G02 or G03), the CNC will issue error 48.

The next drawings show the different cancellation cases.





Programmed path (T.P)

Compensated path (T.C)

8. *MACHINING CANNED CYCLES*

This CNC offers the following machining canned cycles:

G67 N0	Turning.
G67 N1	Facing.
G67 N2	Taper turning
G67 N3	Threadcutting
G67 N4	Rounding
G67 N5	Grooving
G67 N6	Multiple drilling
G67 N7	Simple drilling/tapping
G67 N8	Slot milling along X or Z
G66	Pattern repeat (not accessible from the keyboard)
G68	Stock removal along X (not accessible from the keyboard)
G69	Stock removal along Z (not accessible from the keyboard)
G81	Turning of straight sections (not accessible from the keyboard)
G82	Facing of straight sections (not accessible from the keyboard)
G84	Turning of curved sections (not accessible from the keyboard)
G85	Facing of curved sections (not accessible from the keyboard)
G86	Longitudinal treadcutting canned cycle (not accessible from the keyboard)

Parameters related to the canned cycles:

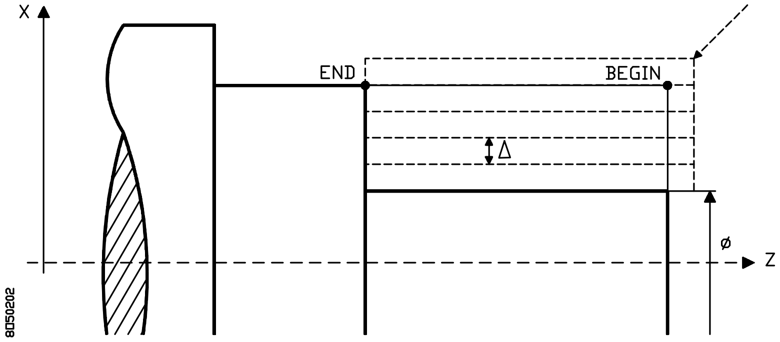
The canned cycles may alter the values of parameters P0 through P99.

On power-up, after a Reset and every time the execution mode of P99996 is quit, the CNC updates (resets) the following arithmetic parameters:

P190	X coordinate of the tool change position set by the machine manufacturer.
P191	Z coordinate of the tool change position set by the machine manufacturer.
P201	Work units (0 = mm, 1 = inches).
P24	Work units (0 = Radus, 1 = Diameter)

When programming the canned cycles, if the value being assigned to a parameter is a constant, write "K" after the "=" sign. For example: N4 G66 P0 = K25 ......

8.1 *TURNING CANNED CYCLE (G67 N0)*



Basic parameters defining a cycle:

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Turning pass "D". It must have a positive radius value.

If programmed with a "0" value, the CNC will assume parameter P5 (number of roughing passes). If both P4 and P5 are equal to "0", the CNC will issue the corresponding message.

P6 Final diameter (f) to be obtained in the turning operation.

P19 Safety distance along the X axis (always in radius).

P20 Safety distance along the Z axis.

Parameters related to the finishing pass:

If no finishing pass is desired, set: P22=K0 and P23=K0

P22 Percentage (%) of programmed roughing pass used as finishing pass.

P23 Percentage (%) of programmed roughing feedrate used as finishing feedrate.

Parameters related to the finishing tool:

When using one roughing tool and a different finishing tool, set the following parameters.

When using the same tool for roughing and finishing, set P26=K0.

P26 Indicates the finishing tool to be used.

If the manufacturer has set a tool change position, arithmetic parameters P190 and P191, the axes will move to that position to change the tool.

If the manufacturer has **not** set a tool change position, arithmetic parameters P134 and P135 **must be set** for tool change position.

P134 X coordinate of the tool withdrawal point at the end of the cycle. It is set in the current work units (in radius or diameter).

P135 Z coordinate of the tool withdrawal point at the end of the cycle.

The CNC considers P134 and P135 if the manufacturer **has not** set a tool change position.

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

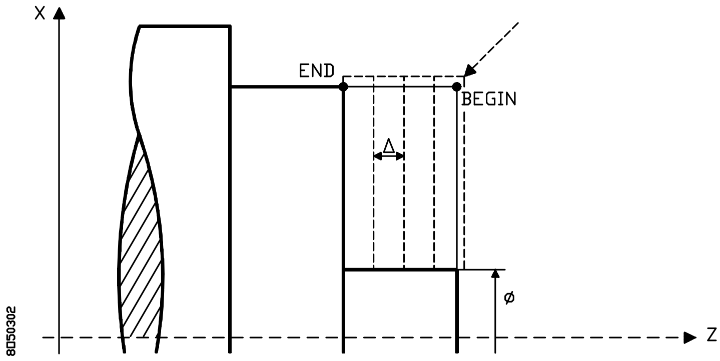
```
N2 F10 S1000 M03
N4 P100=K P101=K P102=K P103=K P4=K P19=K P20=K ...
N6 G67 N0
```

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.2 *FACING CANNED CYCLE (G67 N1)*



Basic parameters defining a cycle:

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Turning pass "D". It must have a positive radius value.

If programmed with a "0" value, the CNC will assume parameter P5 (number of roughing passes). If both P4 and P5 are equal to "0", the CNC will issue the corresponding message.

P6 Final diameter (f) to be obtained in the turning operation.

P19 Safety distance along the X axis (always in radius).

P20 Safety distance along the Z axis.

Parameters related to the finishing pass:

If no finishing pass is desired, set: P22=K0 and P23=K0

P22 Percentage (%) of programmed roughing pass used as finishing pass.

P23 Percentage (%) of programmed roughing feedrate used as finishing feedrate.

Parameters related to the finishing tool:

When using one roughing tool and a different finishing tool, set the following parameters.

When using the same tool for roughing and finishing, set P26=K0.

P26 Indicates the finishing tool to be used.

If the manufacturer has set a tool change position, arithmetic parameters P190 and P191, the axes will move to that position to change the tool.

If the manufacturer has **not** set a tool change position, arithmetic parameters P134 and P135 **must be set** for tool change position.

P134 X coordinate of the tool withdrawal point at the end of the cycle. It is set in the current work units (in radius or diameter).

P135 Z coordinate of the tool withdrawal point at the end of the cycle.

The CNC considers P134 and P135 if the manufacturer **has not** set a tool change position.

<u>General concepts:</u>

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

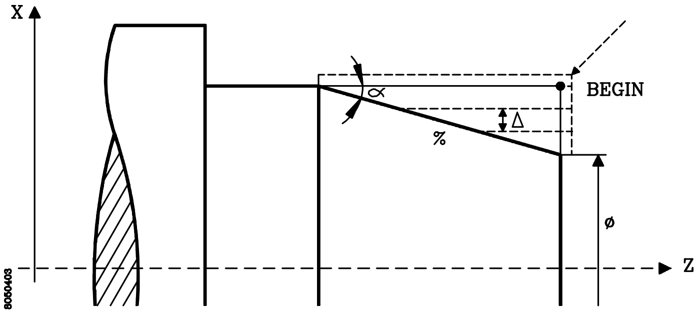
    N2 F10 S1000 M03
    N4 P100=K P101=K P102=K P103=K P4=K P19=K P20=K ...
    N6 G67 N1

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.3 TAPER TURNING CANNED CYCLE (G67 N2)



<u>Basic parameters defining a cycle:</u>

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

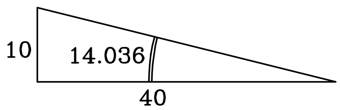
P101 Z coordinate of the "BEGIN" point.

P4 Turning pass "D". It must have a positive radius value.

If programmed with a "0" value, the CNC will assume parameter P5 (number of roughing passes). If both P4 and P5 are equal to "0", the CNC will issue the corresponding message.

P6 Final diameter (f) to be obtained in the turning operation.

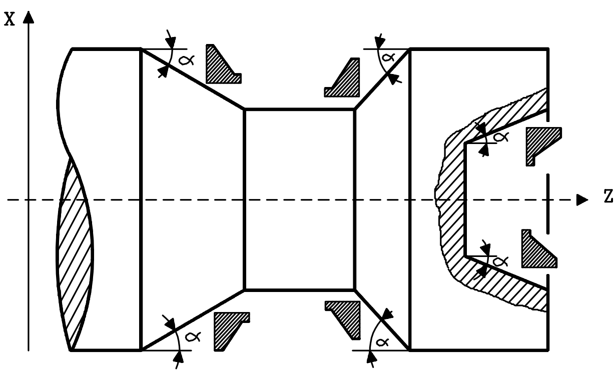
P7 Slope of the taper (%):

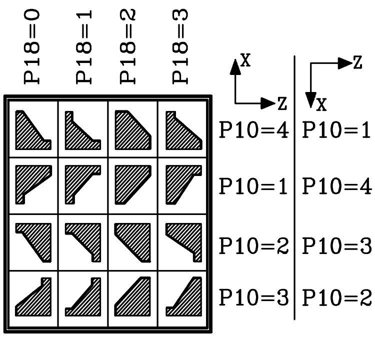


P7 = 100 x (10/40) = 25
P7 = 100 x tangent 14,036 =25

P10 Quadrant to be machined. Together with P18, it defines the type of corner.

P18 Indicates the corner geometry. Together with P10, it defines the type of corner.





P19 Safety distance along the X axis (always in radius).

P20 Safety distance along the Z axis.

Parameters related to the finishing pass:

If no finishing pass is desired, set: P22=K0 and P23=K0

P22 Percentage (%) of programmed roughing pass used as finishing pass.

P23 Percentage (%) of programmed roughing feedrate used as finishing feedrate.

Parameters related to the finishing tool:

When using one roughing tool and a different finishing tool, set the following parameters.

When using the same tool for roughing and finishing, set P26=K0.

P26 Indicates the finishing tool to be used.

If the manufacturer has set a tool change position, arithmetic parameters P190 and P191, the axes will move to that position to change the tool.

If the manufacturer has **not** set a tool change position, arithmetic parameters P134 and P135 **must be set** for tool change position.

P134 X coordinate of the tool withdrawal point at the end of the cycle. It is set in the current work units (in radius or diameter).

P135 Z coordinate of the tool withdrawal point at the end of the cycle.

The CNC considers P134 and P135 if the manufacturer **has not** set a tool change position.

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

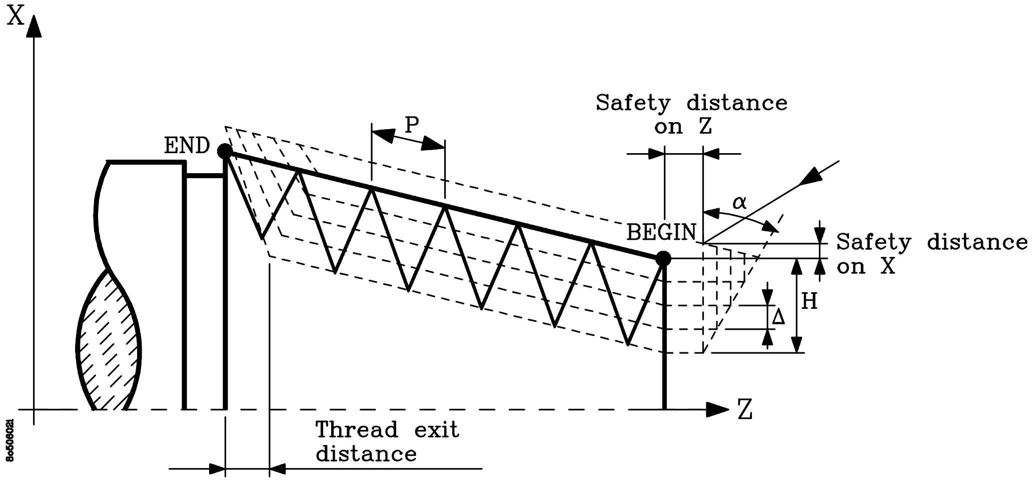
N2 F10 S1000 M03
N4 P100=K P101=K P102=K P4=K P6=K P7=K P10=K P18=K P19=K P20=K
N6 G67 N2

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.4. THREADCUTTING CANNED CYCLE (G67 N3)



<u>Basic parameters defining a cycle:</u>

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

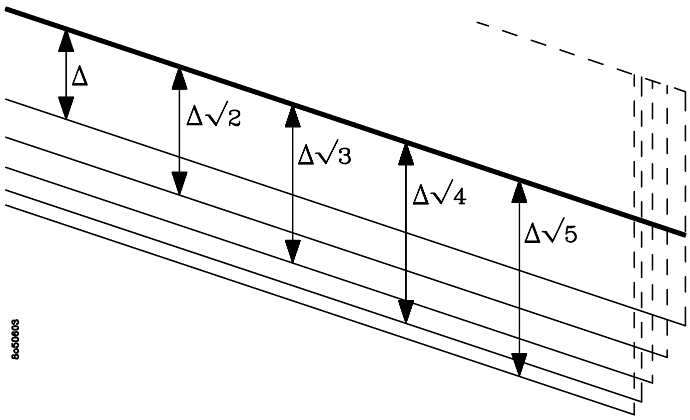
P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Threading pass "D". It must have a positive radius value.

If programmed with a "0" value, the CNC will issue the corresponding error.

The depth of each pass is a function of the corresponding pass number ($D \sqrt{n}$), as shown below:

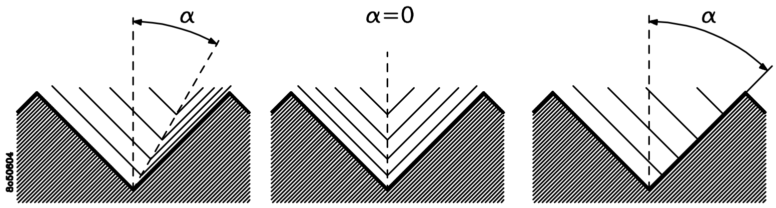


P8 Sets the threading pitch "P". P28 determines the spindle turning direction for right-hand or left-hand threads

P11 Indicates the type of thread. P11=0 for outside thread and P11=1 for inside..

P14 Sets the depth of the thread "H". It must have a positive radius value. If programmed with a "0" value, the CNC will issue the corresponding error.

P16 Defines the penetration angle of the tool with respect to the X axis.



If equal to "0", the thread penetration will be radial. If equal to half the tool angle, the tool will penetrate following the flank of the thread.

P19 Safety distance along the X axis (in radius)

P20 Safety distance along the Z axis

P128 Thread exit distance. It sets the distance from the end of the thread when it starts leaving it. It does so making a taper thread maintaining a Z pitch equal to P10.

If P128=K0, there is no thread exit. If P128 is negative, the CNC issues error 3.

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

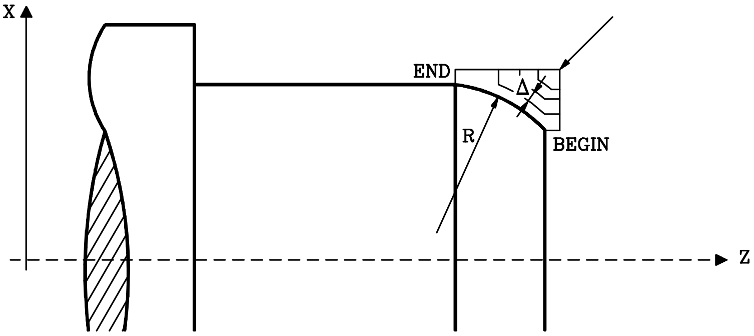
N2 F10 S1000 M03
N4 P100=K P101=K P102=K P103=K P4=K P8=K P11=K P14=K P16=K ...
N6 G67 N3

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

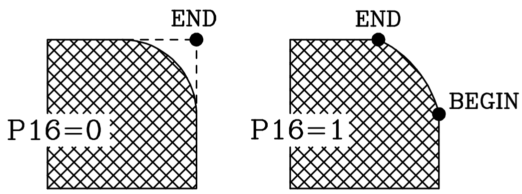
5.- The exit conditions are G00 and G90.

8.5 ROUNDING CANNED CYCLE (G67 N4)



Basic parameters defining a cycle:

P16 Type of cycle.



P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Rounding pass "D". It must have a positive radius value.

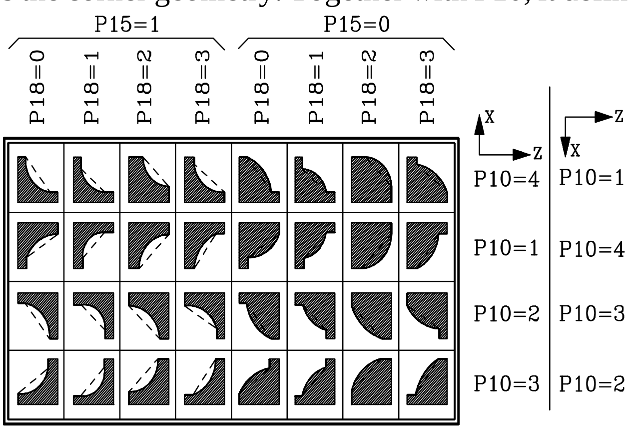
If programmed with a "0" value, the CNC will assume parameter P5 (number of rounding passes). If both P4 and P5 are equal to "0", the CNC will issue the corresponding message.

P9 Defines the rounding radius "R".

P10 Indicates the quadrant to be machined. Together with P15 and P18 sets the type of corner.

P15 Type of corner. P15=0 for convex rounding and P15=1 for concave rounding..

P18 Indicates the corner geometry. Together with P10, it defines the type of corner.



P19 Safety distance along the X axis (in radius)

P20 Safety distance along the Z axis

<u>Parameters related to the finishing pass:</u>

If no finishing pass is desired, set: P22=K0 and P23=K0

P22 Percentage (%) of programmed roughing pass used as finishing pass.

P23 Percentage (%) of programmed roughing feedrate used as finishing feedrate.

<u>Parameters related to the finishing tool:</u>

When using one roughing tool and a different finishing tool, set the following parameters.

When using the same tool for roughing and finishing, set P26=K0.

P26 Indicates the finishing tool to be used.

If the manufacturer has set a tool change position, arithmetic parameters P190 and P191, the axes will move to that position to change the tool.

If the manufacturer has **not** set a tool change position, arithmetic parameters P134 and P135 **must be set** for tool change position.

P134 X coordinate of the tool withdrawal point at the end of the cycle. It is set in the current work units (in radius or diameter).

P135 Z coordinate of the tool withdrawal point at the end of the cycle.

The CNC considers P134 and P135 if the manufacturer **has not** set a tool change position.

<u>General concepts:</u>

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

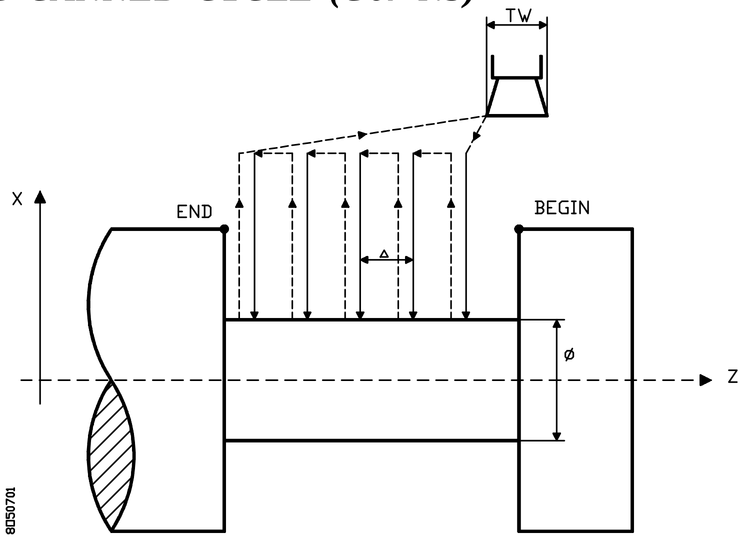
N2 F10 S1000 M03
N4 P16=K P100=K P101=K P102=K P103=K P4=K P9=K P10=K P15=K ...
N6 G67 N4

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.6 *GROOVING CANNED CYCLE (G67 N5)*



Basic parameters defining a cycle:

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Grooving pass "D". It must have a positive radius value.

If programmed with a "0" value, the CNC will assume parameter P5 (number of grooving passes). If both P4 and P5 are equal to "0", the CNC will issue the corresponding message.

P6 Final diameter (f) to be obtained in the grooving operation.

P12 Indicates the tool width (TW).

P13 Sets the time the cutter will remain at the bottom of each grooving pass. It is given in seconds. Therefore, P13=K1.5 means 1.5 seconds.

P19 Safety distance along the X axis (in radius)

P20 Safety distance along the Z axis

Parameters related to the finishing pass:

If no finishing pass is desired, set: P22=K0 and P23=K0

P22 Percentage (%) of programmed roughing pass used as finishing pass.

P23 Percentage (%) of programmed roughing feedrate used as finishing feedrate.

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

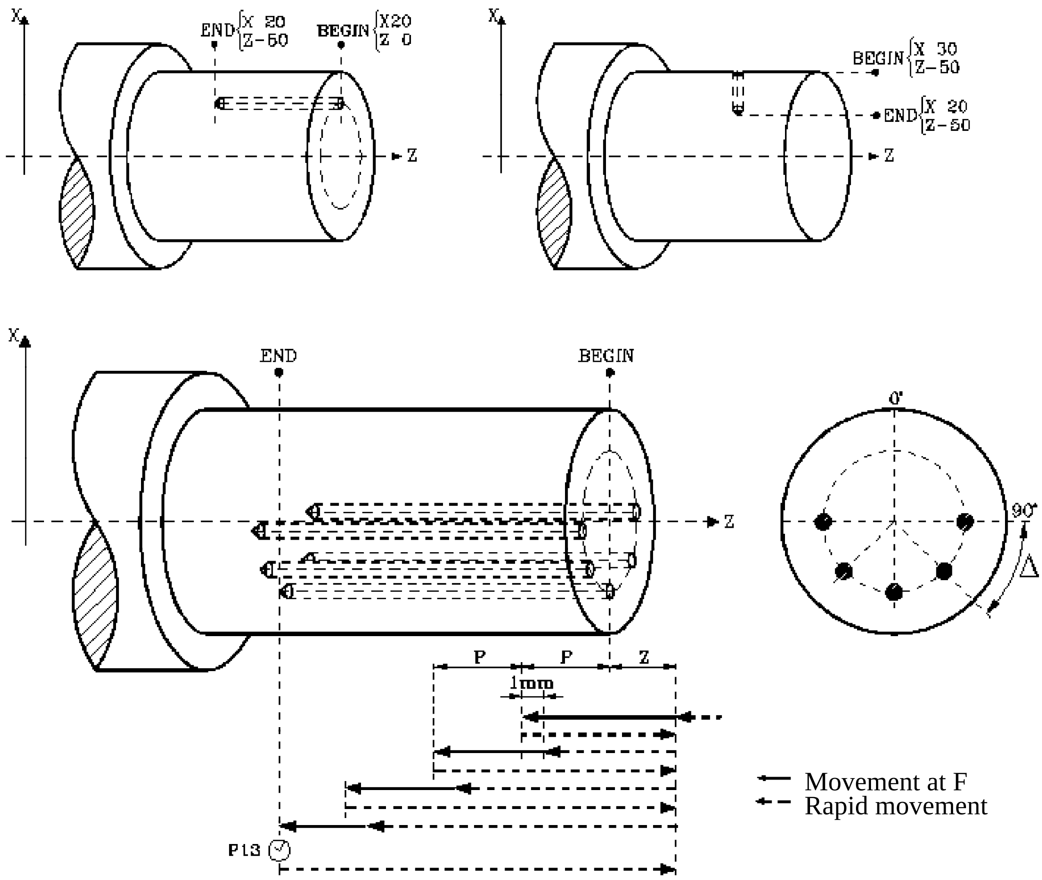
    N2 F10 S1000 M03
    N4 P100=K P101=K P102=K P103=K P4=K P6=K P12=K P19=K ...
    N6 G67 N5

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.7 *MULTIPLE DRILLING CANNED CYCLE (G67 N6)*



Basic parameters defining a cycle:

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Angular increment "D" between holes. It must be given in degrees.

P5 Number of holes to be drilled (N).

P8 Maximum pecking depth (chip removal) "P". When referred to the X axis, it must be expressed in radius.

P13 Sets the time the drill bit will remain at the bottom of the hole. It is given in seconds. Therefore, P13=K1.5 means 1.5 seconds.

P16 Indicates the angular position (a) of the first hole.

P19 Safety distance along the X axis (in radius)

P20 Safety distance along the Z axis

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

N2 F10 S1000 M03
N4 P100=K P101=K P102=K P103=K P4=K P5=K P8=K P13=K P16=K..
N6 G67 N6

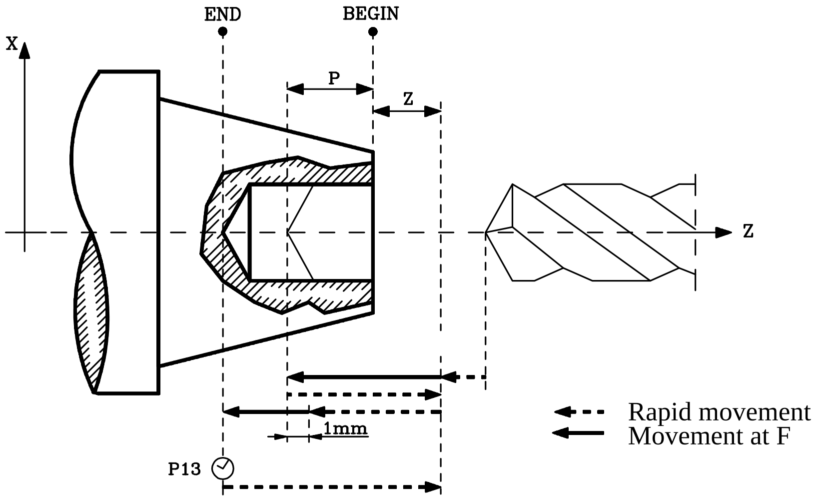
3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.8 *SIMPLE DRILLING CANNED CYCLE (G67 N7)*
TAPPING CANNED CYCLE (G67 N7)

These two cycles are defined in a similar way. Their only difference is parameter "P8" pitch or maximum depth (P). "P" must be set to "0" for tapping and to any other value for simple drilling.



<u>Basic parameters defining a cycle:</u>

P101 Z coordinate of the "BEGIN" point.

P103 Z coordinate of the "END" point.

P8 Indicates the pitch or maximum pecking distance (P) chip removal.

"P8" must be set to "0" for tapping and to any other value for simple drilling.

P13 Sets the time the drill bit will remain at the bottom of the hole. It is given in seconds. Therefore, P13=K1.5 means 1.5 seconds.

P19 Safety distance along the X axis (in radius)

P20 Safety distance along the Z axis

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

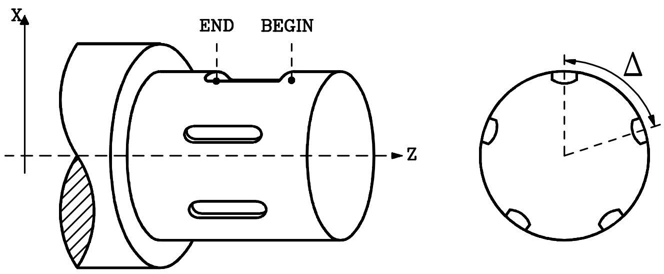
N2 F10 S1000 M03
N4 P101=K P103=K P8=K P13=K P19=K P20=K
N6 G67 N7

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.9 SLOT MILLING CANNED CYCLE (G67 N8)



Basic parameters defining a cycle:

P100 X coordinate of the "BEGIN" point in the selected units (in radius or diameter).

P101 Z coordinate of the "BEGIN" point.

P102 X coordinate of the "END" point in the selected units (in radius or diameter).

P103 Z coordinate of the "END" point.

P4 Angular increment "Δ" between slots. It must be given in degrees.

P5 Number of slots to be milled.

P16 Indicates the angular position (a) of the first slot.

P19 Safety distance along the X axis (in radius)

P20 Safety distance along the Z axis

P21 Milling feedrate.

General concepts:

1. The machining conditions (feedrate spindle speed, etc.) must be programmed **before** calling the cycle.

2.- These parameters must be programmed in the block **before** the one calling the cycle.

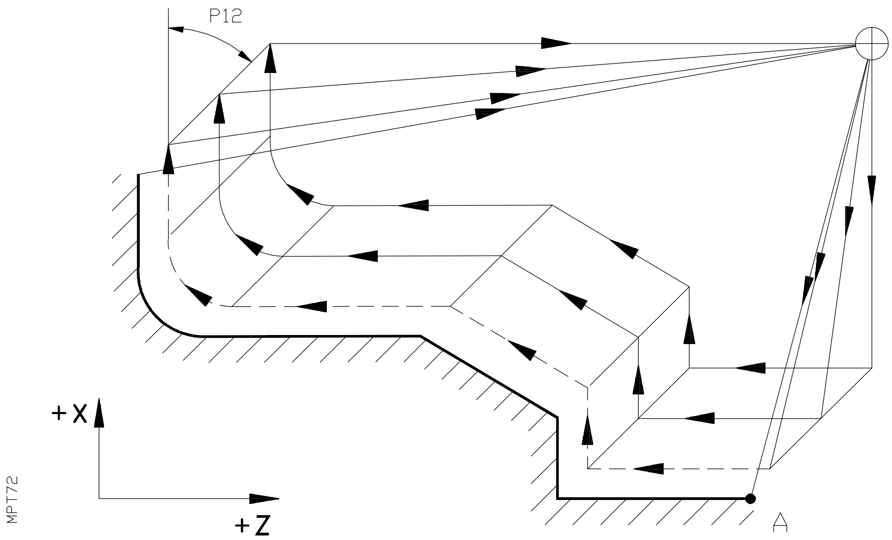
    N2 F10 S1000 M03
    N4 P100=K P101=K P102=K P103=K P4=K P5=K P16=K P19=K P20=K
    N6 G67 N8

3.- The canned cycle **does not** alter the call parameters P22, P23 and P26 which may be used in later cycles. However, **it does** modify the values of the basic parameters defining the cycle.

4.- The basic operation is described in detail in the Operating manual.

5.- The exit conditions are G00 and G90.

8.10 PATTERN REPEATING CANNED CYCLE (G66)



Format:

N4 G66 P0=K P1=K P4=K P5=K P7=K P8=K P9=K P12=K P13=K P14=K

Meaning of the parameters:

P0: X coordinate value of the initial point A, in radius or diameter.

P1: Z coordinate value of the initial point A.

P4: Total amount of stock to be removed. It must be equal to or greater than 0 and equal to or greater than the finishing stock allowance or error 3 will be issued. Depending on P12 it will be identified as residual in X or Z.

P5: Max. step. It must be greater than zero or error 3 will be issued. Depending on P12 it will be identified as step along X or Z axis. The real step calculated by the CNC will be equal or smaller than the max. step.

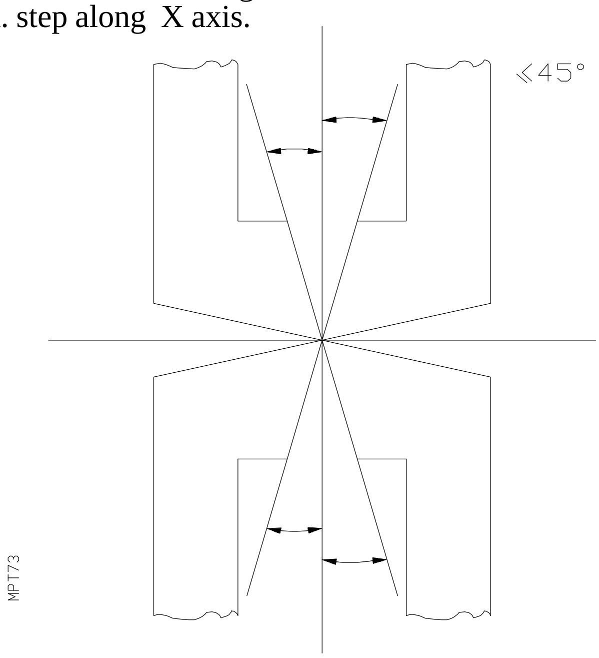
P7: Finishing stock allowance on the X axis. It must be equal to or greater than 0, or error 3 will be issued.

P8: Finishing stock allowance on the Z axis. It must be equal to or greater than 0, or error 3 will be issued.

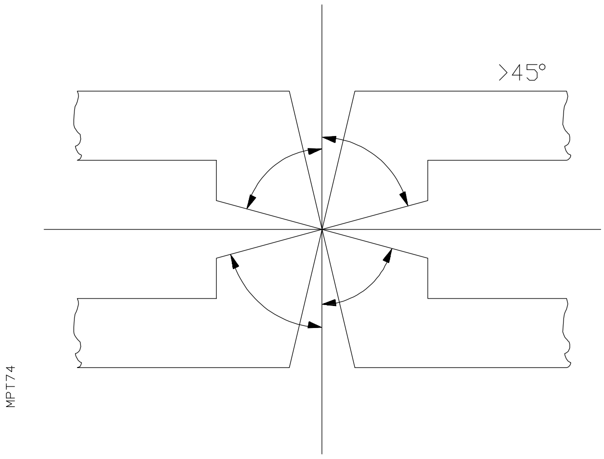
P9: Feedrate for the finishing pass. If it is = 0, there is no finishing pass. If it is negative, error 3 will be issued.

P12: Cutter angle. Its value must be comprised between 0 degrees and 90 degrees or error 3 will be issued.

If it is equal or smaller than 45 degree P4 will be taken as residual stock on the X axis and P5 as max. step along X axis.



If it is greater than 45 degree, P4 will be taken as residual stock on the Z axis and P5 as max. step along Z axis.



P13: Number of the first block to define the pattern.

P14: Number of the last block to define the pattern.

The following points should be borne in mind when programming this canned cycle:

1. The definition of the pattern must not include point A because it is identified by P0 and P1.

2. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.

3. The parameters must be entered either in the cycle calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

4. The exit conditions of the cycle are G00 and G90.

5. The pattern can be made up by straight lines, arcs, roundings, tangential entries, tangential exits and chamfers.

6. Absolute or incremental programming can be used.

7. In the definition of the pattern no "T" function can exist.

8. The approaching and withdrawing movements are carried out in rapid and the rest at the programmed feedrate.

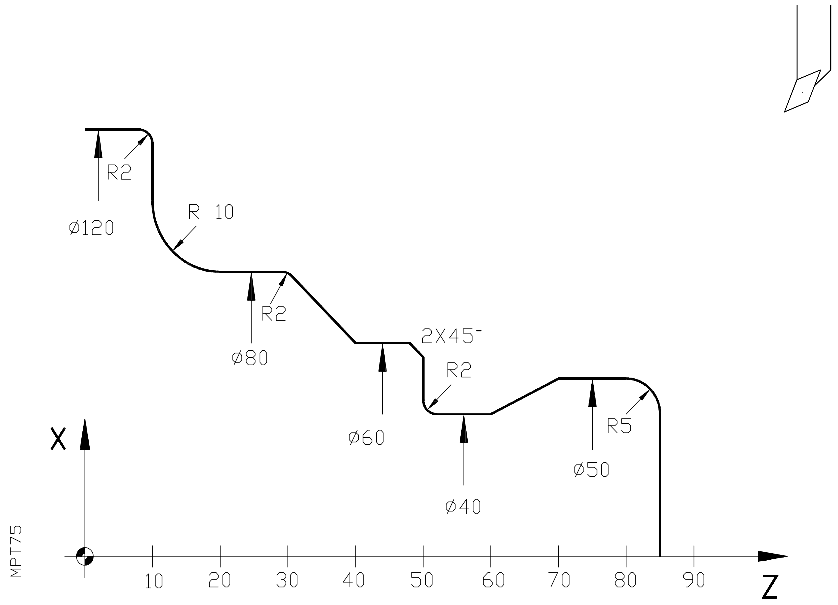
9. The cycle is completed on the starting position of the tool.

10. Tool radius compensation (G41,G42) can be used.

11. The coordinates values (X,Z) of the point from where the cycle is called must be different from P0 and P1 respectively. Otherwise error 4 will be generated.

12. The machining movements will be made at the programmed feedrate.

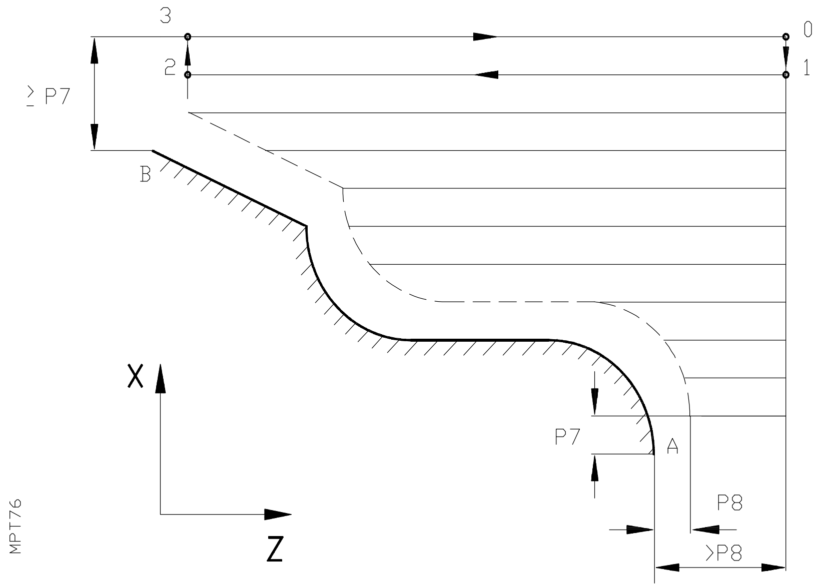
Example G66: X in diameter.



N100 —
N110 G90 G00 G42 X150 Z115
N120 G66 P0=K0 P1=K85 P4=K20 P5=K5 P7=K1 P8=K1
P9=K100 P12=K40 P13=K200 P14=K290
N130 G40 X160 Z135
N140 M30

N200 G36 R5 X50 Z85 .... Profile (pattern) definition
N210 X50 Z70
N220 X40 Z60
N230 G36 R2 X40 Z50
N240 G39 R2 X60 Z50
N250 X60 Z40
N260 G36 R2 X80 Z30
N270 G36 R10 X80 Z10
N280 G36 R2 X120 Z10
N290 X120 Z0

8.11 STOCK REMOVAL CANNED CYCLE ALONG X (G68)



Format:

N4 G68 P0=K P1=K P5=K P7=K P8=K P9=K P13=K P14=K

Meaning of the parameters:

P0: Absolute X coordinate value of the starting point (A) in radius or diameter.

P1: Absolute Z coordinate value of the starting point A.

P5: Max. depth of cut per pass (radius). It must be greater than zero or error 3 will be issued. The real step calculated by the CNC will be equal or smaller than the max. step.

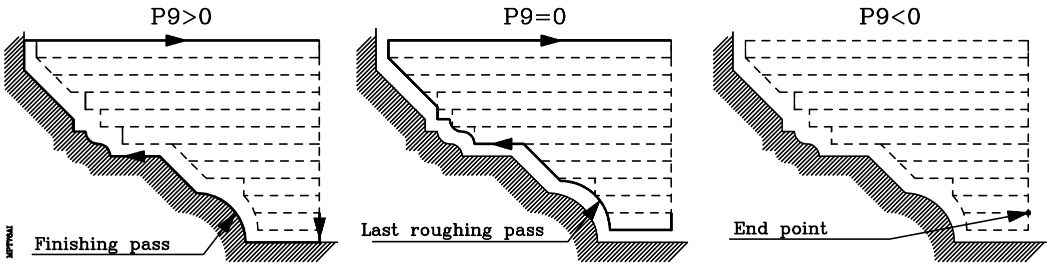
P7: Finishing stock allowance along X axis (radius). It must be greater or equal to zero, or error 3 will be issued.

P8: Finishing stock allowance along Z axis. It must be greater or equal to zero or error 3 will be issued.

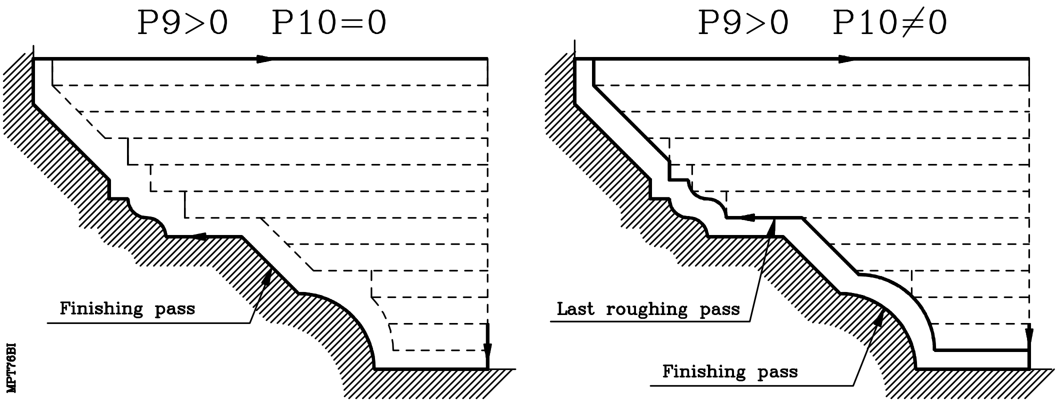
P9: Feedrate of the finishing pass.

If P9=0, there will be no finishing pass; but there will be a final roughing pass maintaining the excess material indicated by P7 and P8.

If it has a negative value, neither a final roughing pass nor a finishing pass will be carried out.



P10: This parameter must be assigned a value other than "0" in order for the CNC to carry out a final roughing pass prior to the finishing pass.



P13: Number of the first block to define the pattern.

P14: Number of the last block to define the pattern. It must be greater than P13. Otherwise error 13 is generated.

When programming this canned cycle the following should be borne in mind:

1. The distance between the starting point 0 and final point (B) along X must be equal to or greater than P7.

   To avoid error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P7+NP5, N being an integer number (any multiple of P5).

2. The distance from 0 to A along the Z axis should be higher than P8.

3. The definition of the pattern must not include point A because it is identified by P0 and P1.

4. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.

   The parameters can be programmed in the calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

   The movements of the elementary work cycle (see drawing) are carried out as follows: At the programmed feedrate from point "1" to point "2" and from "2" to "3"; but in rapid from point "0" to point "1" and from "3" to "0".

   The exit conditions are G00 and G90.

5. The pattern can be made up of straight lines and arcs.

   All the blocks of pattern definition will be programmed with cartesian coordinates being mandatory to program the two axes in absolute, otherwise, the CRT will display error 21.

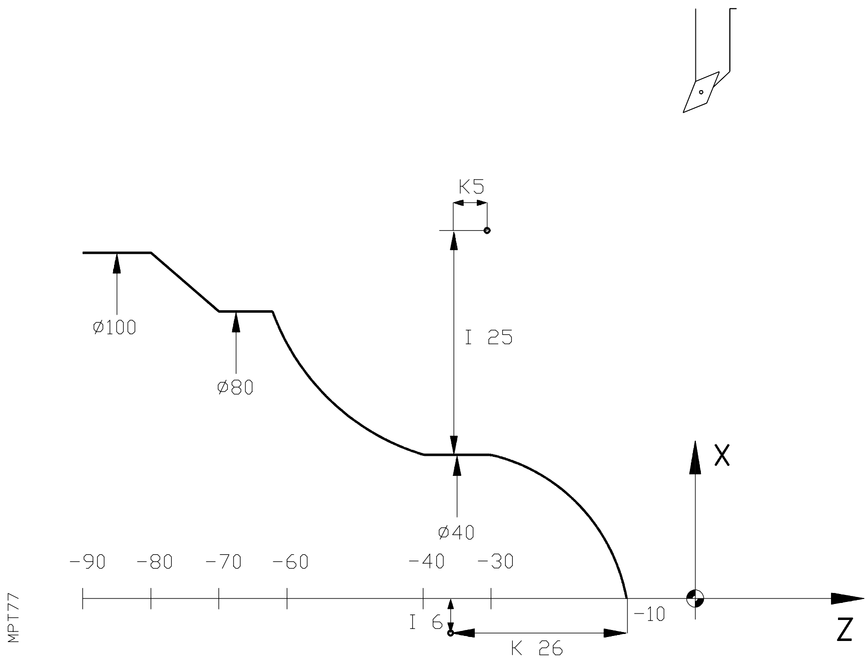
   If arcs are included in the definition, they must be programmed with the center I,K coordinates, referred to the arc's starting point and with the relevant sign.

   If functions F, S, T or M are programmed in the definition, they will be ignored except for the finishing pass. No polar definitions can be used.

6. The cycle is completed on the starting position of the tool 0.

7. If the last movement prior to calling the canned cycle (G68) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be issued.

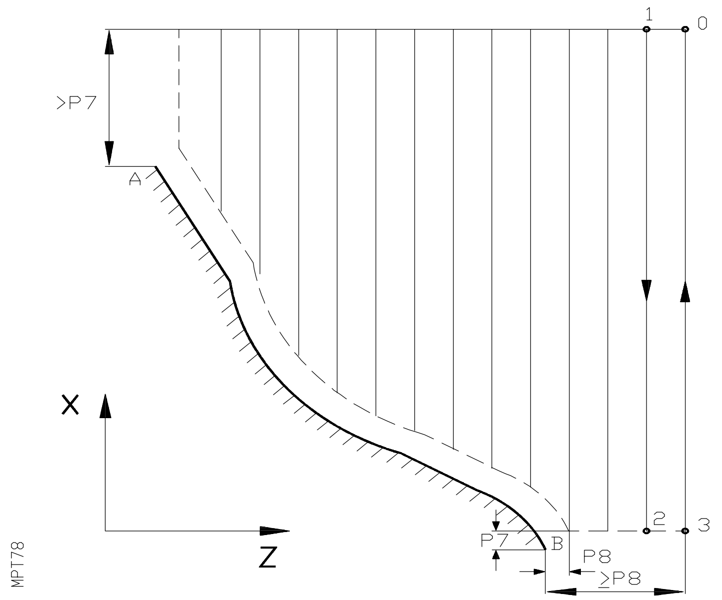
Example G68.



N100 —
N110 G42 G00 X120 Z0
N120 G68 P0=K0 P1=K-10 P5=K2 P7=K0.8 P8=K0.8
P9=K100 P13=K200 P14=K250
N130 G40 X130 Z10
N140 M30

N200 G03 X40 Z-30 I-6 K-26 .... Profile definition
N210 G01 X40 Z-40
N220 G02 X80 Z-60 I25 K5
N230 G01 X80 Z-70
N240 X100 Z-80
N250 X100 Z-90

8.12 STOCK REMOVAL CANNED CYCLE ALONG Z (G69)



Format:

N4 G69 P0=K P1=K P5=K P7=K P8=K P9=K P13=K P14=K

Parameter meaning:

P0: X coordinate of the starting point (A) in radius or diameter.

P1: Z coordinate of the starting point (A).

P5: Max. step. It must be greater than zero or error 3 will be issued. The real step calculated by the CNC will be smaller than or equal to the max. step.

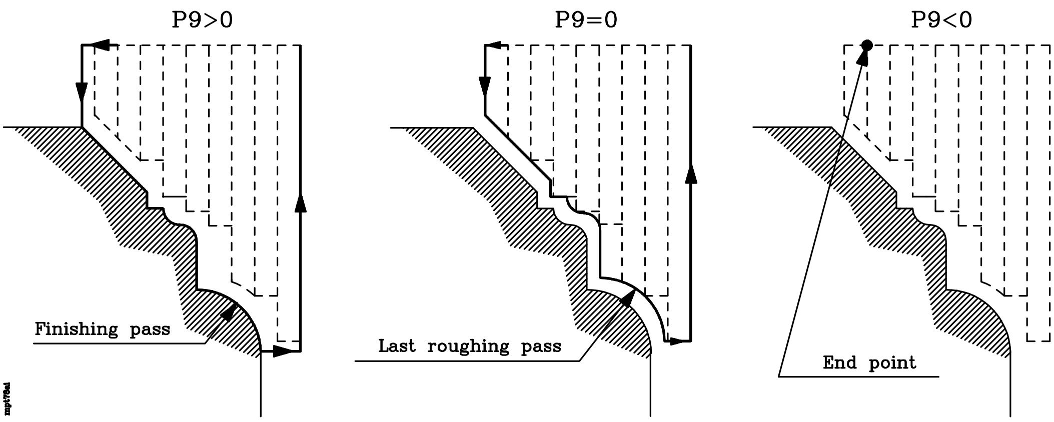
P7: Finishing stock allowance along X axis. It must be greater or equal to zero or error 3 will be issued.

P8: Finishing stock allowance along Z axis. It must be greater or equal to zero or error 3 will be issued.

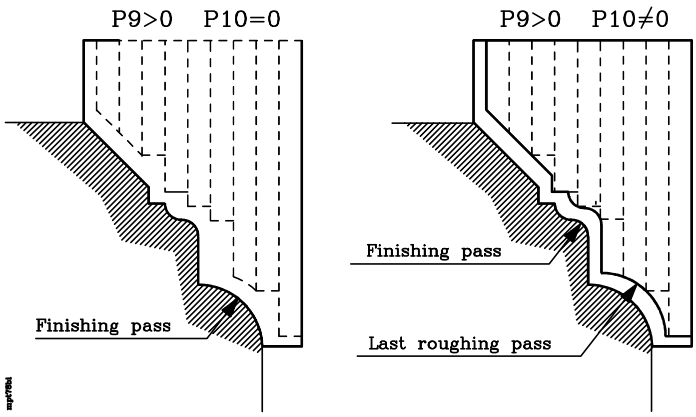
P9: Feedrate of the finishing pass.

If P9=0, there will be no finishing pass; but there will be a final roughing pass maintaining the excess material indicated by P7 and P8.

If it has a negative value, neither a final roughing pass nor a finishing pass will be carried out.



P10: This parameter must be assigned a value other than "0" in order for the CNC to carry out a final roughing pass prior to the finishing pass.



P13: Number of the first block to define the profile.

P14: Number of the last block to define the pattern. It must be higher than P13 or error 13 will be issued.

When programming this canned cycle, the following should be borne in mind:

1. The distance between the starting point "0" and point "B" along the Z axis must be equal or greater than P8.

   To avoid passes that are too thin or generating error P31 when operating with tool compensation the value of this distance (from 0 to B) should be equal to P8+NP5, N being an entire number (any multiple of P5).

2. The distance from 0 to A along. The axis should be higher than P7.

3. The definition of the pattern must not include point A because it is identified by P0 and P1.

4. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.

   The parameters can be programmed in the calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

   The movements of the elementary work cycle (see drawing) are carried out as follows: At the programmed feedrate from point "1" to point "2" and from "2" to "3"; but in rapid from point "0" to point "1" and from "3" to "0".

   The exit conditions are G00 and G90.

5. The pattern can be made up of straight lines and arcs.

   All the blocks of pattern definition will be programmed with cartesian coordinates being mandatory to program the two axes in absolute, otherwise, the CRT will display error 21.

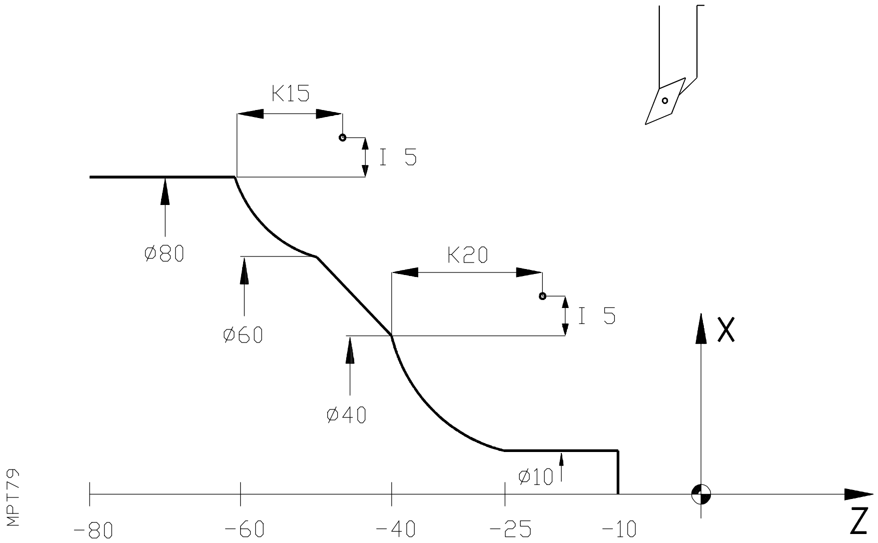
   If arcs are included in the definition, they must be programmed with the center’s I,K coordinates, referred to the arc’s starting point and with the relevant sign.

   If functions F,S,T or M are programmed in the definition, they will be ignored except for the finishing pass. No polar definitions can be programmed.

6. The cycle is completed on the starting position of the tool (0).

7. If the last movement prior to calling the canned cycle (G69) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be issued.

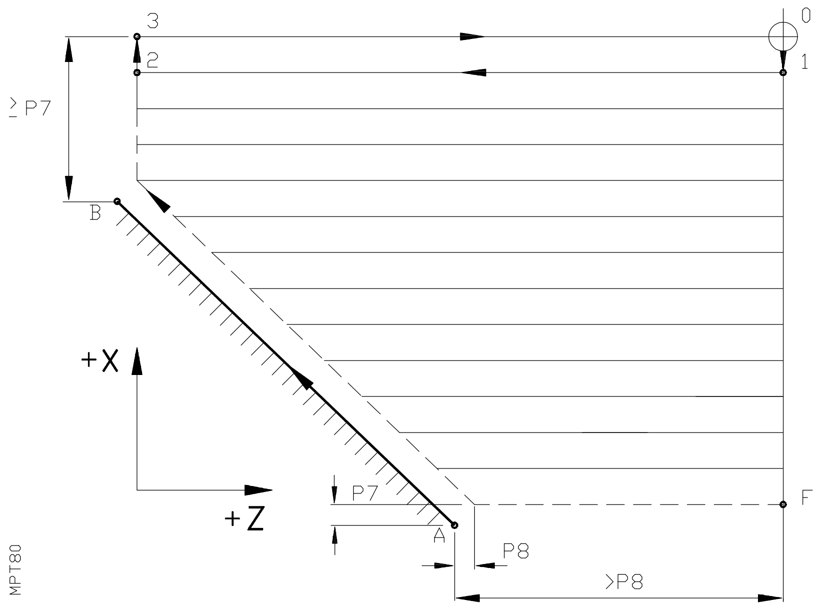
Example G69.



N190 —
N200 G41 G0 X90 Z-5
N210 G69 P0=K80 P1=K-80 P5=K2 P7=K0.8 P8=K0.8 P9=K100 P13=K300 P14=K340
N220 G40 X100 Z0
N230 M30

N300 G01 X80 Z-60 .... Profile definition
N310 G03 X60 Z-50 I5 K15
N320 G01 X40 Z-40
N330 G03 X10 Z-25 I5 K20
N340 G01 X10 Z-10

8.13 *TURNING CANNED CYCLE WITH STRAIGHT SECTIONS (G81)*



Format:

N4 G81 P0=K P1=K P2=K P3=K P5=K P7=K P8=K P9=K

Parameter meaning:

P0: X coordinate value of point A (radius or diameter).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameter).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error 3 will be issued. The real value calculated by the CNC will be smaller than or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error 3 will be issued.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error 3 will be issued.

P9: Feedrate of the finishing pass. If it is zero, there will be no finishing pass. If it is negative, error 3 will be issued.

Programming example (in diameter): Starting point: "0" (X134 Z47) and profile points: "A" (X0 Z0) and "B" (X90 Z-45).

N90 G00 X134 Z47 .... (Tool positioning at point "0").
N100 G81 P0=K0 P1=K0 P2=K90 P3=K-45 P5=K5 P7=K3 P8=K4 P9=K100

When programming this canned cycle, the following should be borne in mind:

1. The distance between the starting point 0 and final point (B) along the X axis must be equal or greater than P7.

   To avoid passes that are too thin or generating error 31 when operting with tool compensation the value of this distance (from 0 to B) should be equal to P7+NP5, N being an entire number.

2. The distance from 0 to A along Z axis should be higher than P8.

3. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.

   The parameters can be programmed in the calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

   The movements of the elementary work cycle (see drawing) are carried out as follows: At the programmed feedrate from point "1" to point "2" and from "2" to "3"; but in rapid from point "0" to point "1" and from "3" to "0".

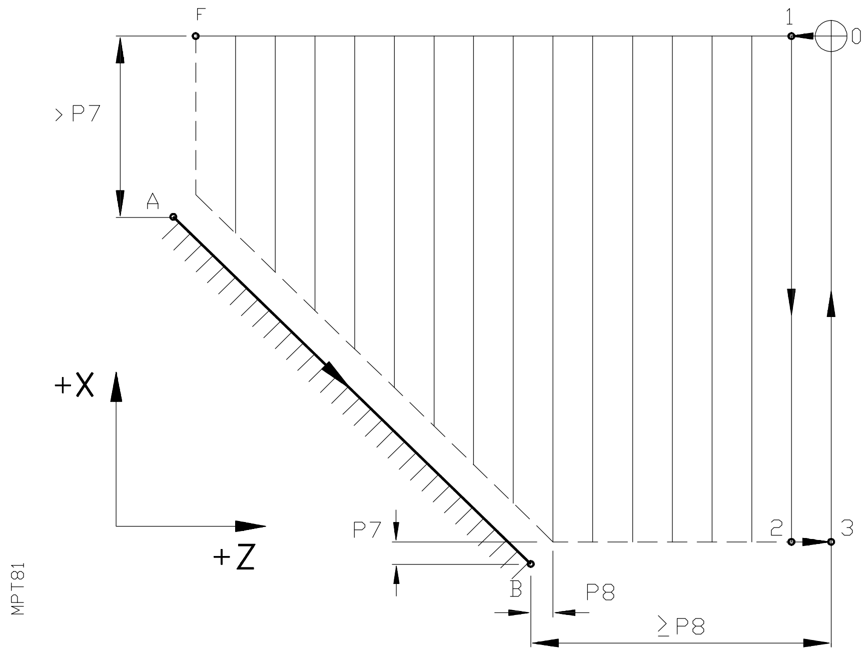
   The exit conditions of the cycle are G00 and G90.

4. If the position of the tool is not correct to execute the cycle, error 4 will be issued. If it is correct a prior horizontal turning will be carried out if necessary

5. If there is a finishing pass the cycle will be completed on the starting position of the tool (0). Otherwise the cycle will end on point F.

6. If the last movement prior to calling the canned cycle (G81) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be issued.

8.14 *FACING CANNED CYCLE WITH STRAIGHT SECTIONS (G82)*



Format:

N4 G82 P0=K P1=K P2=K P3=K P5=K P7=K P8=K P9=K

Parameter meaning:

P0: X coordinate value of point A (radius or diameter).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameter).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error 3 will be one displayed. The real step calculated by the CNC will be smaller than or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error 3 will be issued.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error 3 will be issued.

P9: Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error 3 will be issued.

Programming example (in diameter):
Starting point "0" (X136 Z39). Profile points: "A" (X90 Z-45) and "B" (X0 Z0).

N90 G00 X136 Z39 .... (Tool positioning at point "0").
N100 G82 P0=K90 P1=K-45 P2=K0 P3=K0 P5=K5 P7=K3 P8=K4 P9=K100

When programming this canned cycle, the following points must be borne in mind:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8.

   To avoid passes that are too thin or generating error 31 when operting with tool compensation the value of this distance (from 0 to B) should be equal to P8+NP5, N being an entire number.

2. The distance from 0 to A along the X axis should be higher than P7.

3. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.

   The parameters can be programmed in the calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

   The movements of the elementary work cycle (see drawing) are carried out as follows: At the programmed feedrate from point "1" to point "2" and from "2" to "3"; but in rapid from point "0" to point "1" and from "3" to "0".

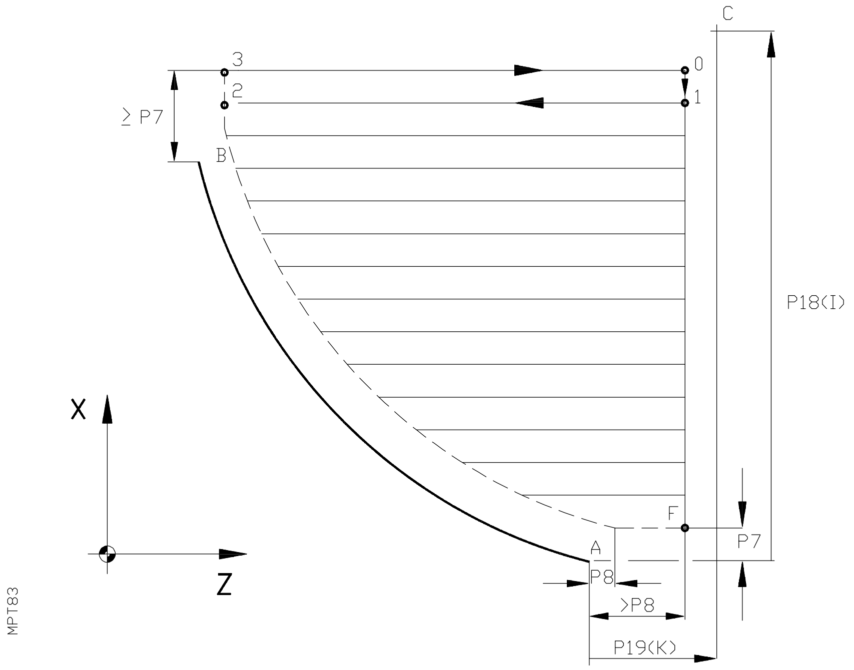
   The exit conditions of the cycle are G00 and G90.

4. If the position of the tool is not correct to execute the cycle, error 4 will be issued. If it is correct a previous vertical facing will be carried out if necessary

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end at point F.

6. If the last movement prior to calling the canned cycle (G82) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise, error 35 will be issued.

8.15 TURNING CANNED CYCLE WITH CURVED SECTIONS (G84)



Format:

N4 G84 P0=K P1=K P2=K P3=K P5=K P7=K P8=K P9=K P18=K P19=K

Parameter meaning:

P0: X coordinate value of point A (radius or diameter).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameter).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error 3 will be issued. The real step calculated by the CNC will be smaller than or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error 3 will be issued.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error 3 will be issued.

P9: Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error 3 will be issued.

P18: Distance **I** between the point **A** and the arc's center along the X axis. Although the X axis is programmed in diameter, the values of I are always programmed in radius.

P19: **K** distance between the point **A** and the arc's center along the Z axis.

Programming example (in diameter):
Starting point: "0" (X149 Z86). Profile points: "A" (X0 Z71), B(X120 Z11). Arc center: "C" (X160 Z91).

N90 G00 X149 Z86 .... (Tool positioning at point "0")
N100 G84 P0=K0 P1=K71 P2=K120 P3=K11 P5=K5 P7=K4 P8=K4 P9=K100
        P18=K80 P19=K20

When programming this canned cycle, take into account the following aspects:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8.

   To avoid passes that are too thin or generating error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P8+N P5, N being an entire number.

2. The distance between the starting point 0 and the point (A), along the X axis, should be higher than P7.

3. The machining conditions (feedrate, spindle rotation ...) must be programmed before calling the cycle.

   The parameters can be programmed either in the cycle calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

   The movements of the elemental work cycle (see drawing) are carried out as follows: At the programmed feedrate from point "1" to point "2" and from "2" to "3"; but in rapid from point "0" to point "1" and from "3" to "0".

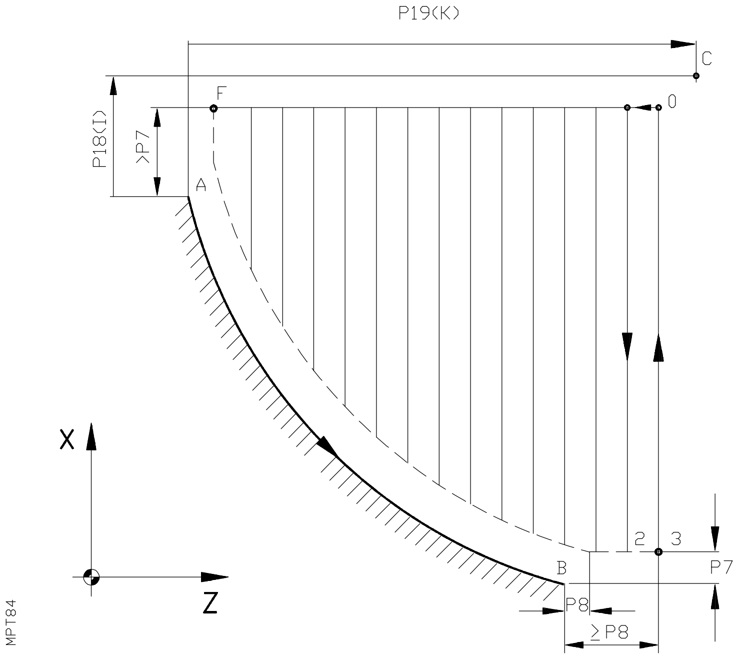
   The exit conditions are G00 and G90.

4. If the position of the tool is not correct to execute the cycle, error 4 will be issued. If it is correct a prior horizontal turning will be carried out if necessary.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end at point F.

6. If the last movement prior to calling the canned cycle has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be issued.

8.16 *FACING CANNED CYCLE WITH CURVED SECTIONS (G85)*



Format:

N4 G85 P0=K P1=K P2=K P3=K P5=K P7=K P8=K P9=K P18=K P19=K

Parameter meaning:

P0: X coordinate value of point A (radius or diameter).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameter).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error 3 will be issued. The real step calculated by the CNC will be smaller than or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error 3 will be issued.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error 3 will be issued.

P9: Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error 3 will be issued.

P18: Distance **I** between the point **A** and the arc's center along the X axis. Although the X axis is programmed in diameter, the values of **I** are always programmed in radius.

P19: **K** distance between the point **A** and the arc's center along the Z axis.

Programming example (in diameter):
Starting point "0" (X150 Z85). Profile points: "A" (X118 Z11), "B" (X0 Z70). Arc center: "C" (X160 Z91)

N90 G00 X150 Z85 .... (Tool positioning at point "0")
N100 G85 P0=K118 P1=K11 P2=K0 P3=K70 P5=K5 P7=K4 P8=K4 P9=K100
P18=K21 P19=K80

When programming this canned cycle, take into account the following aspects:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8.

   To avoid passes that are too thin or generating error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P8+N P5, N being an entire number.

2. The distance between the starting point 0 and the point (A), along the X axis, should be higher than P7.

3. The machining conditions (feedrate, spindle rotation ...) must be programmed before calling the cycle.

   The parameters can be programmed either in the cycle calling block or in previous blocks.

   This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

   The movements of the elementary work cycle (see drawing) are carried out as follows: At the programmed feedrate from point "1" to point "2" and from "2" to "3"; but in rapid from point "0" to point "1" and from "3" to "0".

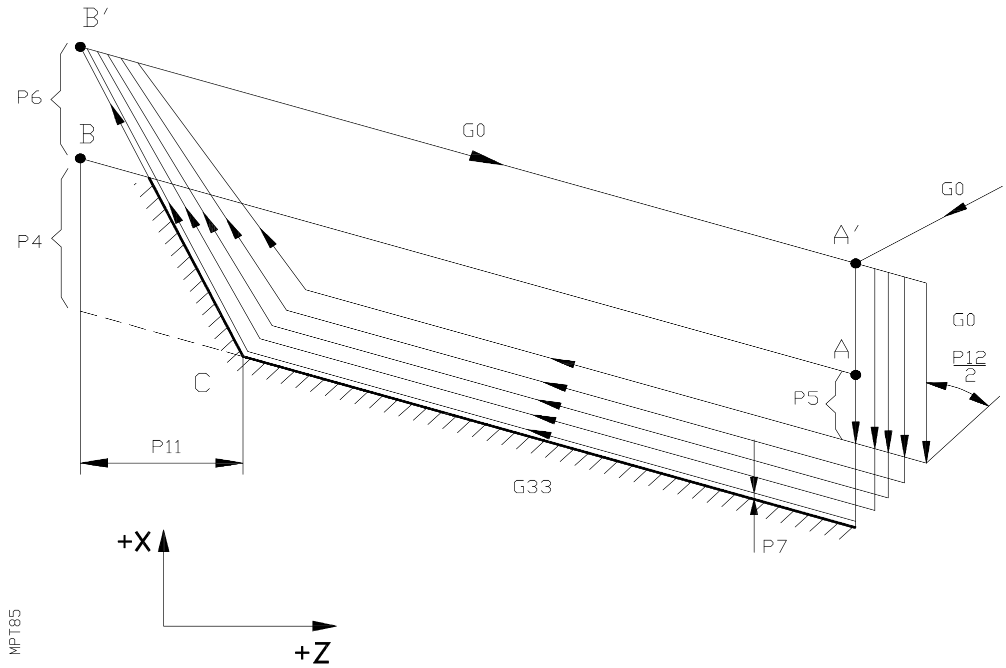
   The exit conditions are G00 and G90.

4. If the position of the tool is not correct to execute the cycle, error 4 will be issued. If it is correct a prior vertical facing will be carried out if necessary.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end a point F.

6. If the last movement prior to calling the canned cycle has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be issued.

8.17 *LONGITUDINAL THREADCUTTING CYCLE (G86)*



Format:

N4 G86 P0=K P1=K P2=K P3=K P4=K P5=K P6=K P7=K P10=K P11=K P12=K

Meaning of the parameters:

P0: Absolute X value of the starting point of the thread (A) in radius or diameters.

P1: Absolute Z value of the starting point of the thread (A).

P2: Absolute X value of the final point of the thread (B) in radius or diameters.

P3: Absolute Z value of the final point of the thread (B).

P4: Depth of the thread (in radius). If will have a positive value in external threads and a negative one in internal threads. If it is zero error code 3 will be displayed.

P5: Initial pass (in radius). It defines the depth of the first cutting pass. The subsequent passes will depend on the sign given to the parameter.

- If the sign is **positive**, the depth of the second pass will be P5 $\sqrt{2}$ and the depth of the 11th will be P5 $\sqrt{n}$, until the finishing depth is reached.

- If the sign is **negative**, the penetration increment will be constant and of a value equal to the absolute value of the parameter.

- If the value is 0, error 3 will be issued.

P6: Safety distance in radius. It indicates at what distance from the surface of the thread it starts returning to point A'.

- If the value is positive, this movement will be done in G05 (rounded corner). The 0 value is considered positive.

- If the value is negative, this movement will be done in G07 (square corner).

P7: Finishing pass (in radius):

- If it is 0, the previous pass is repeated.

- If the value is positive, the finishing pass will be carried out maintaining a P12/2 angle with the X axis.

- If the value is negative, the finishing pass will be done with radial entry.

P10: Thread pitch along the Z axis.

P11: Thread exit. It defines the distance from the end of the thread to the point where the exit starts. If it is negative, error code 3 will be displayed. If other than zero, the CB' section is a tapered thread whose pitch along Z axis is P10. If it is zero, the CB' section is executed in G00.

P12: Angle of the tool's nose. It makes the starting points of the successive passes to be at a P12/2 angle with X axis.

When programming this canned cycle, the following must be borne in mind.

1. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.

2. The parameters can be programmed in the calling block or in previous blocks.

3. This canned cycle **does not** alter the call parameters which may be used in later cycles. However, **it does** modify the values of parameters P70 through P99.

4. The exit conditions are G00,G07,G40, G90 and G97.

5. The cycle starts with a G00 approach to point A' and ends at A' as well. When executing the block, the F feedrate speed cannot be altered by turning the **FEEDRATE** knob whose value will be "set" at 100%.

9. SUBROUTINES

A subroutine is a section of a program which, properly identified, may be called upon to be executed from any position (block) of the program.

A subroutine may be called upon several times from different program positions or from different programs.

With a single call, the subroutine may be executed up to 255 times.

A subroutine may be included within the user-defined program P99996 or may be stored in the special program P99994 for user-defined subroutines.

Standard and parametric subroutines are basically identical. Their only difference being that the calling block for a parametric subroutine (G21 N2.2) may contain up to 15 definition parameters.

In the case of a standard subroutine, the parameters may not be defined in the calling block.

The maximum number of parameters of a standard or parametric subroutine is 255 (P0 through P254).

9.1 SPECIAL PROGRAM P99994 FOR USER SUBROUTINES

Program P99994 must be edited at a PC and sent out to the CNC. It cannot be modified at the CNC.

It must only contain the user-defined subroutines edited in ISO code.

When program P99996 calls upon a subroutine, the CNC looks for it in P99996 first and, then, if not found, in P99994.

Using P99994 is recommended when several user-defined programs P99996 are being utilized. This way, if P99994 contains all the usual subroutines, they will not have to be repeated in each P99996 program.

When having a subroutine associated to a tool, machine parameter "P730", it is recommended to store that subroutine in program P99994.

9.2 *IDENTIFICATION OF A STANDARD SUBROUTINE (G22)*

A standard subroutine (not parametric), always starts with a block containing function "G22". The structure of this block is:

N4 G22 N2	N4	Block number
	G22	Defines the beginning of the subroutine
	N2	Identifies the subroutine (number between "N0" and "N99").

This block cannot contain any additional information.

Program all the desired blocks after this first subroutine block and remember that a standard subroutine may also contain parametric blocks.

A subroutine must always end with a block of the type: N4 G24. This "G24" code indicates the end of the subroutine. This block cannot contain any other information.

Programming example:	N0	G22 N25
	N10	X20
	N15	P0=P0 F1 P1
	N20	G24

<u>Atention:</u>

Subroutines "N91" through "N99" may not be defined because they are utilized by the CNC.

The CNC memory may not contain two standard subroutines with identical ID numbers even if they belong to different programs. However, it is possible to use the same number to identify a standard subroutine and a parametric one.

9.3 *CALLING A STANDARD SUBROUTINE (G20)*

A standard subroutine may be called upon from any program or other subroutine (standard or parametric). To do this, "G20" must be used. The structure of a calling block is:

N4 G20 N2.2	N4	Block number
	G20	Calling a subroutine
	N2.2	The two digits to the left of the period, identify the subroutine being called upon (00 through 99). The two digits to the right of the period, indicate the number of times that subroutine is to be executed (00 through 99).

The number of times may also be programmed by an arithmetic parameter between P0 and P255. For example: N4 G20 N10.P123

If the number of times is not programmed, the CNC will execute it only once.

The block calling the standard subroutine may not contain any other information.

9.4 IDENTIFICATION OF A PARAMETRIC SUBROUTINE (G23)

A parametric subroutine always starts with a block containing a "G23".

The structure of this first block is:

N4 G23 N2	N4	Block number
	G23	Defines the beginning of a parametric subroutine
	N2	Identifies the subroutine (number between "N0" and "N99").

This block cannot contain any additional information.

Program all the desired blocks after this first subroutine block and remember that a parametric subroutine may also contain parametric blocks.

A subroutine must always end with a block of the type: N4 G24. This "G24" code indicates the end of the subroutine. This block cannot contain any other information.

Atention:

Subroutines "N91" through "N99" may not be defined because they are utilized by the CNC.

The CNC memory may not contain two parametric subroutines with identical ID numbers even if they belong to different programs. However, it is possible to use the same number to identify a standard subroutine and a parametric one.

9.5 CALLING A PARAMETRIC SUBROUTINE (G21)

A parametric subroutine may be called upon from any program or other subroutine (standard or parametric). To do this, "G21" must be used.

The structure of a calling block is:

N4 G21 N2.2 P3=K±5.5 P3=K±5.5

N4	Block number
G21	Calling a parametric subroutine
N2.2	The two digits to the left of the period, identify the subroutine being called upon (00 through 99). The two digits to the right of the period, indicate the number of times that subroutine is to be executed (00 through 99).

The number of times may also be programmed by an arithmetic parameter between P0 and P255. For example: N4 G21 N10.P123

If the number of times is not programmed, the CNC will execute it only once.

P3	Number of the arithmetic parameter (P00-P254).
K	Value assigned to the arithmetic parameter.

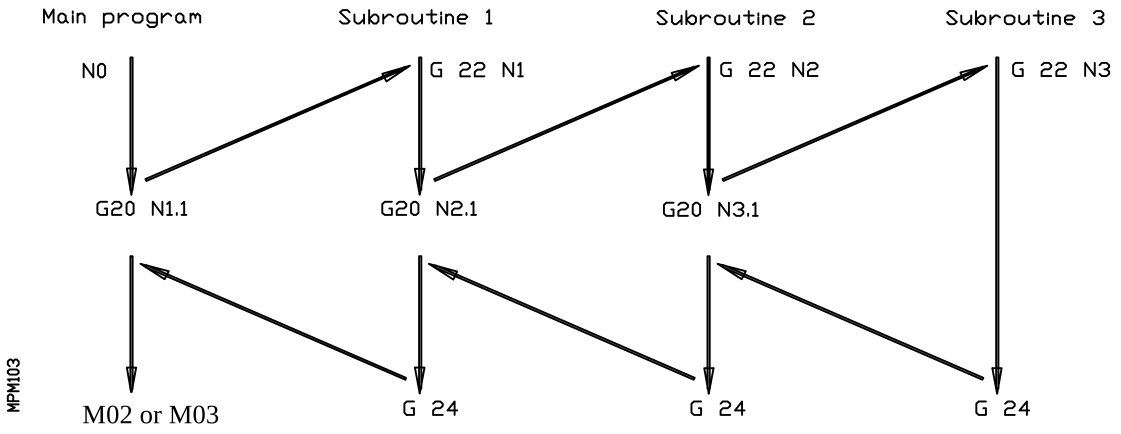
Atention:

When concluding the execution of the parametric subroutine (G24), the parameters recover the values assigned to them in the calling block even when they have been set to other values throughout the program.

9.6 NESTING LEVELS

It is possible to call upon a subroutine from the main program or from another subroutine (standard or parametric) and upon another one from this one and so on up to 15 levels. Each one of these levels may be repeated 255 times.

Subroutine nesting diagram



10. PARAMETRIC PROGRAMMING

This CNC offers 255 arithmetic parameters (P0 through P254) which can be used to edit parametric blocks and perform math operations and jumps within a program. These parametric blocks may be written anywhere in the program.

The various operations to be performed with these parameters are:

F1	Addition
F2	Subtraction
F3	Multiplication
F4	Division
F5	Square root
F6	Square root of the sum of squares (Pythagorean formula)
F7	Sine
F8	Cosine
F9	Tangent
F10	Arc tangent
F11	Comparison
F12	Entire part (integer without decimals)
F13	Entire part plus one
F14	Entire part minus one
F15	Absolute value (without sign)
F16	Complement (invert sign)
F17	Memory address (location) of the indicated block
F18	X coordinate contained in the block located at the indicated address
F19	Z coordinate contained in the block located at the indicated address
F20	Memory address (location) of the block previous to the indicated one
F21	I coordinate contained in the block located at the indicated address
F22	K coordinate contained in the block located at the indicated address
F23	Selected tool offset number
F24	X value of the indicated tool
F25	Z value of the indicated tool
F26	F value of the indicated tool
F27	R value of the indicated tool
F28	I value of the indicated tool
F29	K value of the indicated tool
F30	Logic AND function
F31	Logic OR function
F32	Logic XOR function
F33	Logic NOR function
F34	*Not being used at this time*
F35	*Not being used at this time*
F36	Selected tool number

10.1 *ASSIGNMENTS*

Any value may be assigned to any parameter.

a) N4 P1 = P2

P1 takes the value of P2 while P2 keeps its original value

b) N4 P1 = K1.5

P1 takes the value of 1.5

"K" indicates that it is a constant. Its value range is: ±99999.99999.

c) N4 P1= H (HEXADECIMAL value)

P1 takes the HEXADECIMAL value indicated after "H".
Possible "H" values: 0/FFFFFFFF.

d) N4 P1 = X

P1 takes the current theoretical coordinate value of the X axis.

e) N4 P1 = Z

P1 takes the current theoretical coordinate value of the Z axis.

h) N4 P1 = R

P1 takes the value of "1" if machine parameter P11 (radius /diameter) is set for radius and the value of "2" if set for diameter.

i) N4 P1 = T

This CNC has an internal clock which keeps track of the program execution (running) time.

P1 takes the current value of this clock (always in hundredths of a second).

In order to find out the execution time required by some parts or operations, include this type of blocks at the beginning and end of the program section to be timed and then subtract the obtained values.

j) N4 P1= 0X

P1 takes the current theoretical X axis coordinate referred to Machine Reference Zero (home).

l) N4 P1= 0Z

P1 takes the current theoretical Z axis coordinate referred to Machine Reference Zero (home).

10.2 OPERATORS "F1" through "F16"

F1 Addition

Example: N4 P1 = P2 F1 P3

P1 takes the value resulting from adding P2 and P3. That is, P1 = P2 + P3.

It may also be programmed as: N4 P1 = P2 F1 K2 which means that P1 takes the value of P2 + 2. The letter "K" indicates that it is a constant.

When the same parameter appears as an addend and a result, N4 P1 = P1 F1 K2 , it indicates that, from this moment on, it increments its value by "2" (P1 = P1 + 2).

F2 Subtraction

N4 P10 = P2 F2 P3 .... P10 = P2 - P3
N4 P10 = P2 F2 K3 .... P10 = P2 - 3
N4 P10 = P10 F2 K1 .... P10 = P10 - 1

F3 Multiplication

N4 P17 = P2 F3 P30 .... P17 = P2 x P30
N4 P17 = P2 F3 K4 .... P17 = P2 x 4
N4 P17 = P17 F3 K8 .... P17 = P17x 8

F4 Division

N4 P8 = P7 F4 P35 .... P8 = P7 : P35
N4 P8 = P2 F4 K5 .... P8 = P2 : 5
N4 P8 = P8 F4 K2 .... P8 = P8 : 2

F5 Square root

N4 P15 = F5 P23 .... P15 = $\sqrt{P23}$
N4 P14 = F5 K9.................... P14 = $\sqrt{9}$
N4 P18 = F5 P18 .... P18 = $\sqrt{P18}$

F6 Square root of the sum of squares

N4 P60 = P2 F6 P3 .... P60 = $\sqrt{P2^2 + P3^2}$
N4 P50 = P40 F6 K5 .... P50 = $\sqrt{P40^2 + 5^2}$
N4 P1 = P1 F6 K4 .... P1 = $\sqrt{P1^2 + 4^2}$

F7 Sine

N4 P1 = F7 K5...................... P1 = Sine 5 degrees
N4 P1 = F7 P2 .... P1 = Sine P2

The angle (P2) must be given in degrees.

F8 Cosine

N4 P1 = F8 P2 .... P1 = Cosine P2
N4 P1 = F8 K75.................... P1 = Cosine 75 degrees

F9 Tangent

N4 P1 = F9 P2 .... P1 = tg P2
N5 P1 = F9 K30.................. P1 = tg 30 degrees

F10 Arc tangent

N4 P1 = F10 P2 .... P1 = arc. tg P2 (result in degrees)
N4 P1 = F10 K0.5............... P1 = arc. tg 0.5

F11 Comparison

It compares one parameter with another one or with a constant and it activates the conditional jump flags (its usefulness will be described in the section on conditional jumps G26, G27, G28, G29).

N4 P1 = F11 P2

If P1 = P2, the "if zero" flag is activated.
If P1 $\geq$ P2, the "if equal to or greater than" flag is activated.
If P1 < P2, the "if smaller than" flag is activated.

It may also be programmed as: N4 P1 = F11 K6

F12 Entire Part (Integer without decimals)

N4 P1=F12 P2 .... P1 takes the integer value of P2.
N4 P1=F12 K5.4................. P1 = 5

F13 Entire part plus one

N4 P1 = F13 P2 .... P1 takes the integer value of P2 plus 1.
N4 P1 = F13 K5.4............... P1 = 5 + 1 = 6

F14 Entire part minus one

N4 P1 = F14 P27 .... P1 takes the integer value of P27 minus 1.
N4 P5 = F14 K5.4............... P5 = 5 - 1 = 4

F15 Absolute value

N4 P1 = F15 P2 .... P1 takes the absolute value of P2.
N4 P1 = F15 K-8 .... P1 = 8

F16 Complement

N4 P7 = F16 P20 .... P7 takes the value of P20 with the opposite sign. P7 = -P20
N4 P7 = F16 K10................ P7 = -10

10.3 OPERATORS "F17" through "F29"

These functions do not affect the jump flags.

F17 N4 P1 = F17 P2

P1 takes the memory address value (location) of the block number indicated by P2.

Example N4 P1 = F17 K12. P1 takes the value of the memory address where block N12 is located.

F18 N4 P1=F18 P2

P1 takes the X coordinate value contained in the block whose address is (located at) P2.

F18 does not admit a constant operand. Example : **P1 = F18 K2 IS WRONG.**

F19 N4 P1=F19 P2

P1 takes the Z coordinate value contained in the block located at P2.

F19 does not admit a constant operand. Example: **P1 = F19 K3 IS WRONG**

F20 N4 P1 = F20 P2

P1 takes the memory address value (location) of the block previous to the one located at P2.

F20 does not admit a constant operand. Example: **P1 = F20 K4. IS WRONG**

F21 N4 P1=F21 P2

P1 takes the "I" coordinate value appearing at the block located in P2.

F21 does not admit a constant operand. Example: **P1 = F21 K2. IS WRONG.**

F22 N4 P1=F22 P2

P1 takes the "K" coordinate value appearing at the block located in P2.

F22 does not admit a constant operand. Example: **P1 = F22 K3. IS WRONG.**

F23 N4 P1 = F23

P1 takes the number of the current active tool.

F24 This function may be programmed in two different ways:

N4 P9=F24 K2	P9 takes the X value appearing in position 2 of the tool table.
N4 P8=F24 P12	P8 takes the X value appearing in the tool table position indicated by the value of P12.

F25 This function may be programmed in two different ways:

N4 P15=F25 K16	P15 takes the Z value appearing in position 16 of the tool table.
N4 P13=F25 P34	P13 takes the Z value appearing in the tool table position indicated by the value of P34.

F26 This function may be programmed in two different ways:

N4 P6=F26 K32	P6 takes the F value appearing in position 32 of the tool table.
N4 P14=F26 P15	P14 takes the F value appearing in the tool table position indicated by the value of P15.

F27 This function may be programmed in two different ways:

N4 P90=F27 K13	P90 takes the R value appearing in position 13 of the tool table.
N4 P28=F27 P5	P28 takes the R value appearing in the tool table position indicated by the value of P5.

F28 This function may be programmed in two different ways:

N4 P17=F28 K10	P17 takes the I value appearing in position 10 of the tool table.
N4 P19=F28 P63	P19 takes the I value appearing in the tool table position indicated by the value of P63.

F29 This function may be programmed in two different ways:

N4 P15=F29 K27	P15 takes the K value appearing in position 27 of the tool table.
N4 P13=F29 P25	P13 takes the K value appearing in the tool table position indicated by the value of P25.

One block may contain all the desired assignments and operations as long as they do not modify more than 15 parameters.

10.4 BINARY OPERATORS "F30" THROUGH "F33"

The available binary operations are:

F30	Logic AND function
F31	Logic OR function
F32	Logic XOR function
F33	Logic NOR function

These BINARY operations also activate the internal flags depending on the results of their operations for later use when programming CONDITIONAL JUMPS/CALLS (G26, G27, G28, G29).

Binary operations may be performed between:

Parameters	P1 = P2 F30 P3
Parameters and constants	P11 = P25 F31 H(8)
Constants	P19 = K2 F32 K5

The "H" value must be a positive hexadecimal number of up to 8 characters. In other words, it must be between "0" and "FFFFFFFF" and may not be the first operand.

F30 Logic AND function

Example: N4 P1= P2 F30 P3	Value of P2	Value of P3	Value of P1
	A5C631F	C883D	C001D

F31 Logic OR function

Example: N4 P11= P25 F31 H35AF9D01

Value of P25	Value of H	Value of P11
48BE6	35AF9D01	35AF9FE7

F32 Logic XOR function

Example: N4 P19= P72 F32 H91C6EF

Value of P72	Value of H	Value of P19
AB456	91C6EF	9B72B9

F33 Logic NOT function

Example: N4 P154= F33 P88 P154 takes the "ones" complement of (inverts its bits) P88

Value of P88	Value of P154
4A52D63F	B5AD29C0

10.5 OPERATOR "F36"

This function does not affect the jump flags.

F36 N4 P1 = F36

P1 takes the current tool number

10.6 CONDITIONAL JUMP FUNCTIONS (G26, G27, G28, G29)

They are similar to function G25 (unconditional jump) described in the chapter on "Additional Preparatory Functions" in this manual.

Functions G26, G27, G28 and G29, before jumping to the indicated program block, verify that the required condition has been met.

G26 Jump if zero.	It requires the "Zero" condition to be met.
G27 Jump if not zero.	It requires the "Zero" condition NOT to be met.
G28 Jump if less than zero.	It requires the "Less than" condition to be met.
G29 Jump if equal/greater than "0".	It requires the "Less than" condition NOT to be met.

The "Zero" condition also referred to as "Equal to" is activated in the following cases:

* When the result of an operation is equal to zero.
  Example: N001 P1 P3 F2 K5   If P3 = 5

* If both sides of a comparison are identical
  Example: N002 P1 F11 K8   If P1 = 8

The "Less than" condition is also referred to as "Negative" and it is activated in the following cases:

* When the result of an operation is less than zero (negative).
  Example: N001 P1 = P3 F2 K5   If P3 < 5

* When the first operand in a comparison is smaller than second one.
  Example: N002 P1 F11 K8   If P1 < 8

Atention:

Assignments and non-parametric functions do not alter the status of the condition flags.

Programming example:

```
N060  P2    F11 K22
N065  G01   X10
N070        Y20
N071  G26   N100
N072  G28   N200
N073  G29   N300
```

Block N060 does a comparison
Blocks N65 and N70 do not change the condition flags.
Thus, If P2 = 22, the program continues at block N100
If P2 < 22, the program continues at block N200
If P2 > 22, the program continues at block N300

Care must be taken when programming functions G26 and G29. If, in the above example, the following were programmed instead:

```
N071 G28   N200
N072 G29   N300
N073 G26   N100
```

The program would not execute N073. With P2 < 22, it would jump to N200 and, with P2 ≥ 22, it would jump to N300. Thus, skipping block N073.

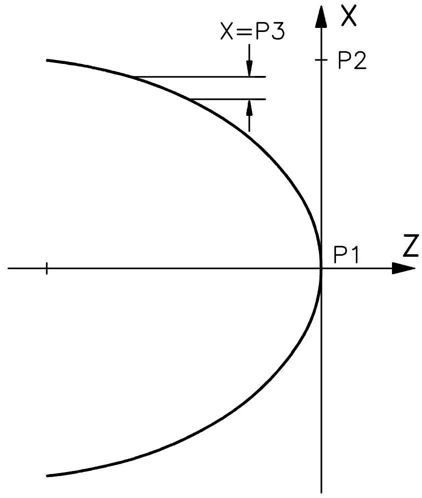
Parametric programming example to calculate the coordinates of the various points forming a parabola. The X axis is programmed in diameter.

The formula for a parabola:

$Z = - K X^2$

Where:

P0 = K
P4 = X
P5 = Z



The call parameters are:

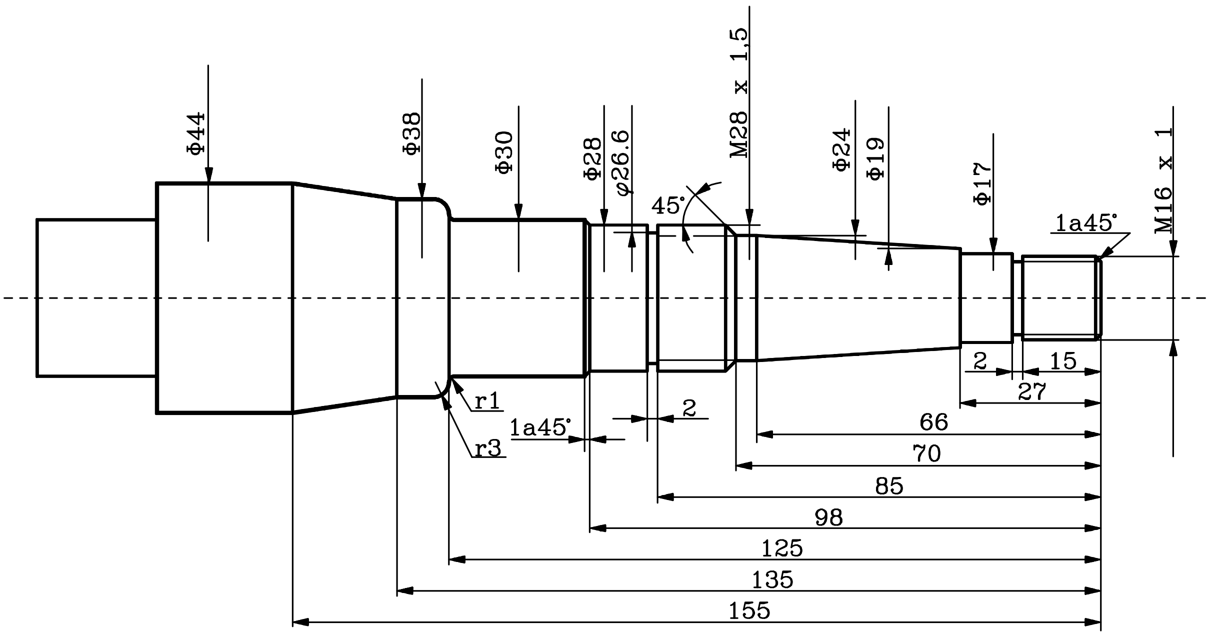
P0	=	K
P1	=	Starting X coordinate
P2	=	Final X coordinate
P3	=	Increment along X.

Program:

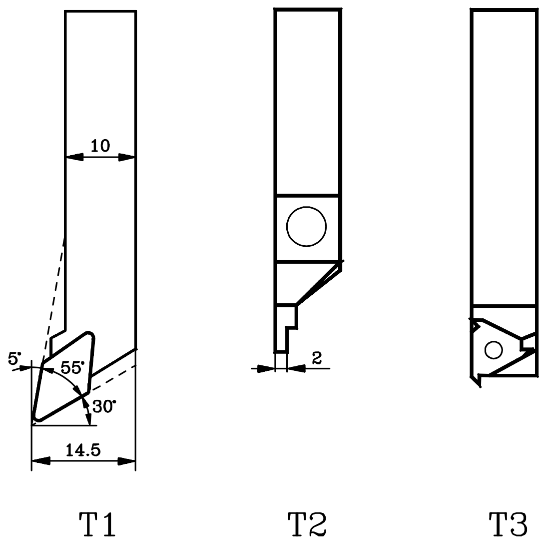
```
N0080  G21 N56.1 P0=K0.01 P1=K00 P2=K100 P3=K1...... Call a subroutine
N0090  M30 ............................................................ End of program


N0110  G23 N56 ........................................ Definition of a subroutine
N0120  P4=P1 ........................................... X= starting X
N0130  P4=P4 F1 P3 P4=F11 P2 ............. Compare new X with final X
N0140  G28 N160 ...................................... If smaller, continue at N160
N0150  P4=P2 ........................................... If greater, new X = final X
N0160  P5=P4 F3 P4 P5=P5 F3 P0
       P5=F16 P5 ..................................... Calculate new Z coordinate
N0170  G01 XP4 ZP5 ................................ Move to new point (X, Z)
N0180  P4=F11 P2 ..................................... Compare new X wih final X
N0190  G27 N130 ...................................... If smaller, calculate next point, go to N130
N0200  G24 ................................................ End of subroutine
```

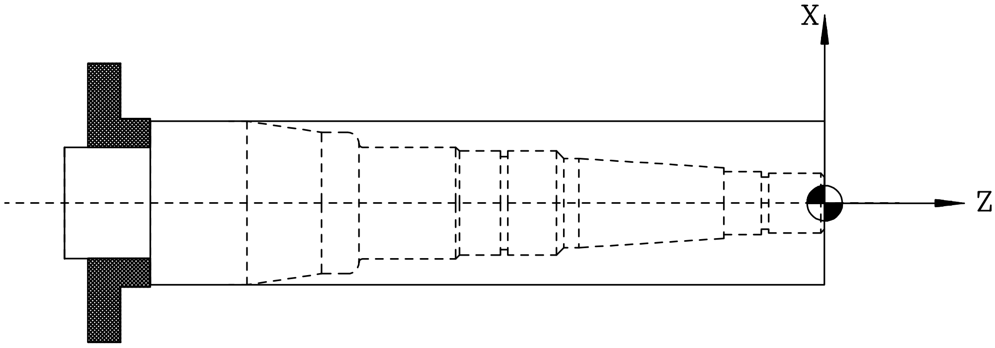
EXAMPLE



Φ44
Φ38
Φ30
Φ28
φ26.6
M28 x 1.5
45°
Φ24
Φ19
Φ17
1a45°
M16 x 1
r1
r3
1a45°
2
2
15
27
66
70
85
98
125
135
155



10
5°
55°
30°
14.5
2
T1
T2
T3

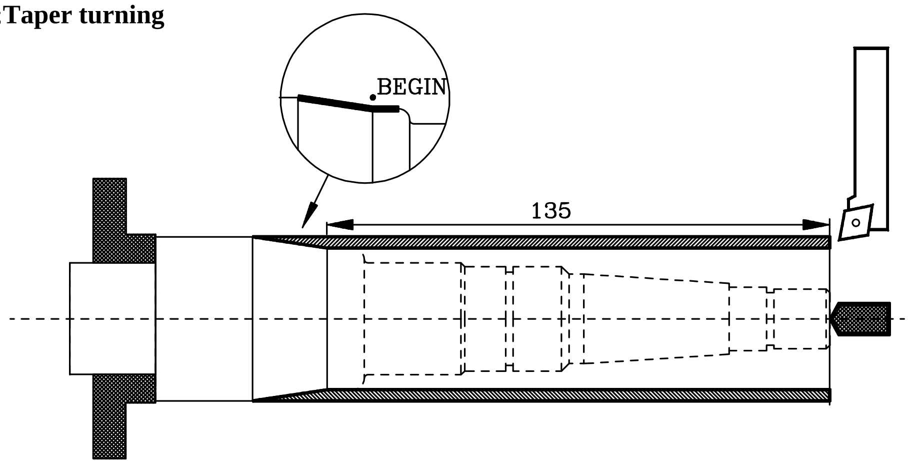


N00 G95 F10 S1000 M03 ;Machining conditions

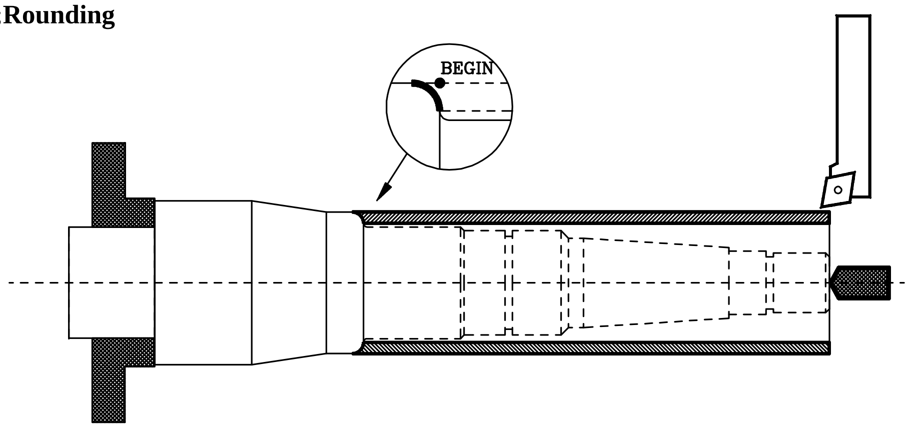
N05 G0 X80 Z50 ;Goes to tool change position
N06 T1.1 ;Selects tool "1"

N10 P22=K0 P23=K0 P26=0 ;There is no finishing pass

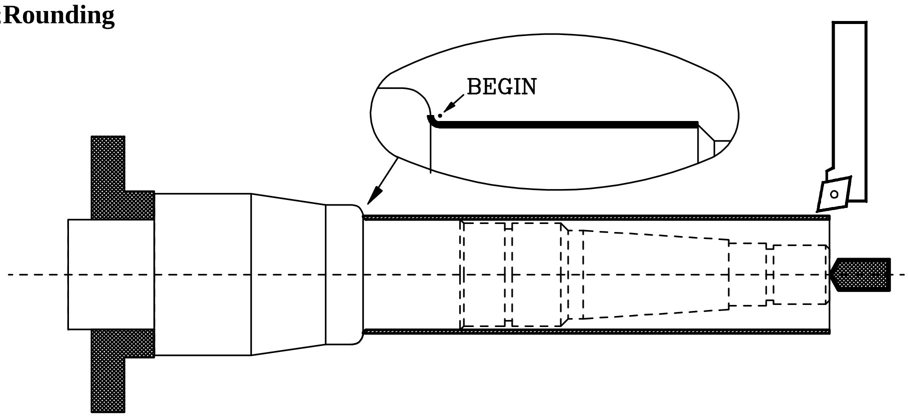
N20 P100=K44 P101=K-135 P4=K1 P6=K38 P7=K15 P10=K4 P18=K3 P19=K1 P20=K136
N25 G67 N2 ;Taper turning



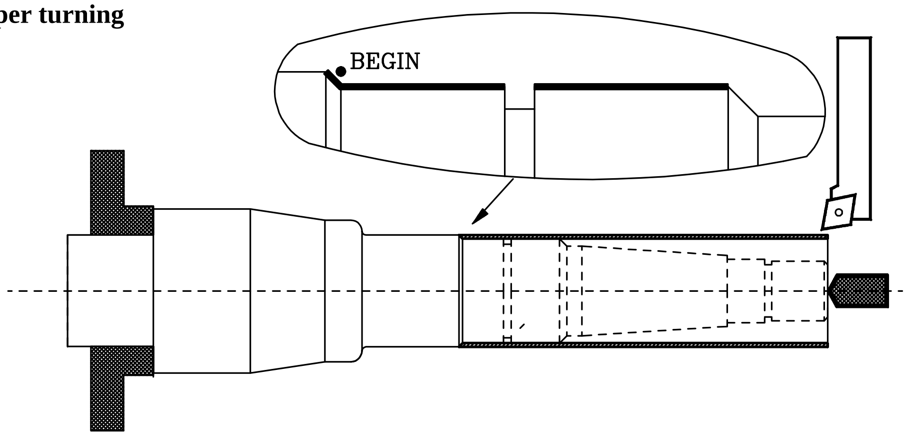
N30 P16=K0 P100=K38 P101=K-125 P4=K1 P9=K3 P10=K4 P15=K0 P18=K3 P19=K1 P20=K126
N35 G67 N4 ;Rounding



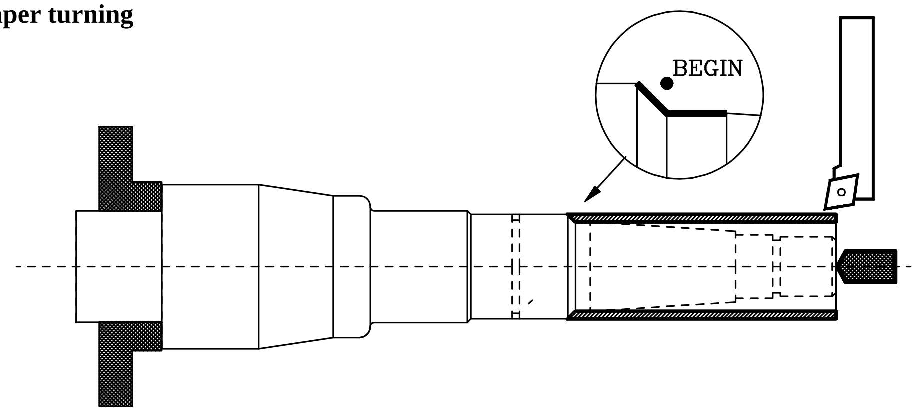
N40 P16=K0 P100=K32 P101=K-124 P4=K1 P9=K1 P10=K4 P15=K1 P18=K3 P19=K1 P20=K126
N45 G67 N4 ;Rounding



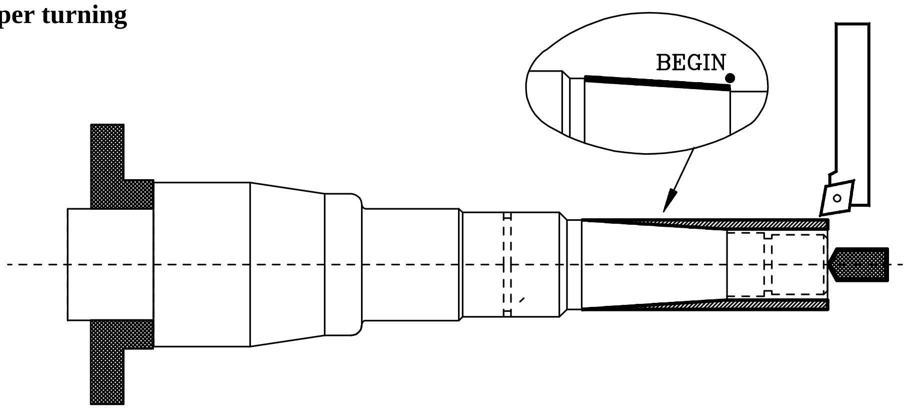
N40 P100=K30 P101=K-98 P4=K1 P6=K28 P7=K100 P10=K4 P18=K3 P19=K1 P20=K99
N45 G67 N2 ;Taper turning



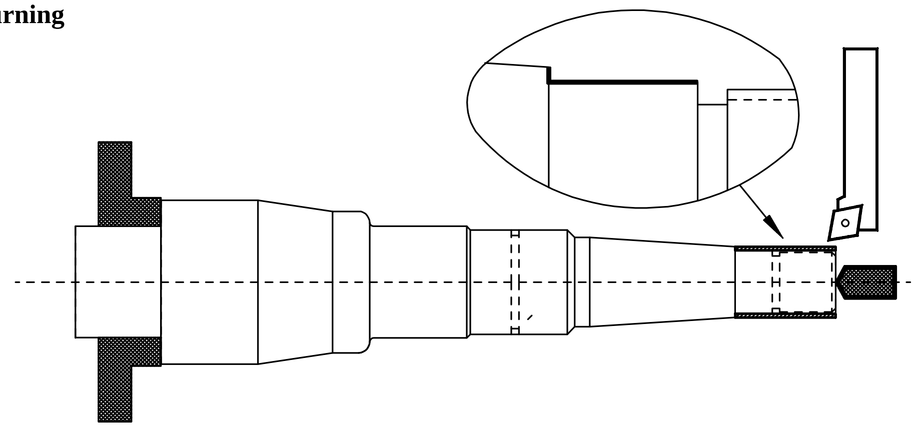
N50 P100=K28 P101=K-70 P4=K1 P6=K24 P7=K100 P10=K4 P18=K3 P19=K1 P20=K71
N55 G67 N2 ;Taper turning



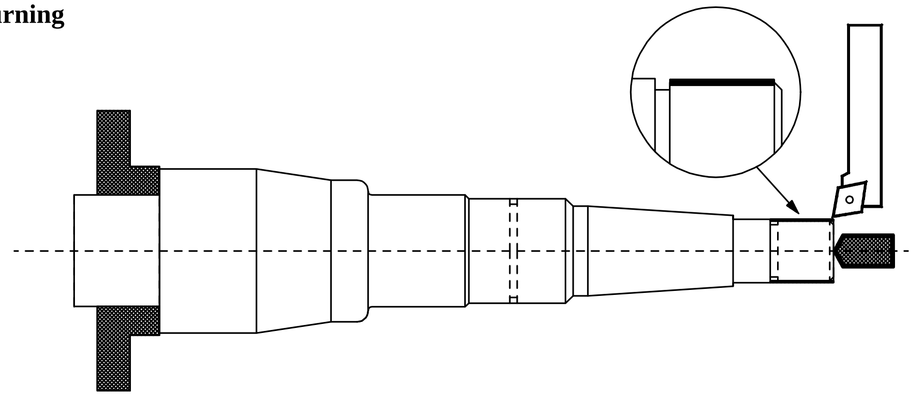
N60 P100=K24 P101=K-27 P4=K1 P6=K19 P7=K6.41 P10=K4 P18=K3 P19=K1 P20=K28
N65 G67 N2 ;Taper turning



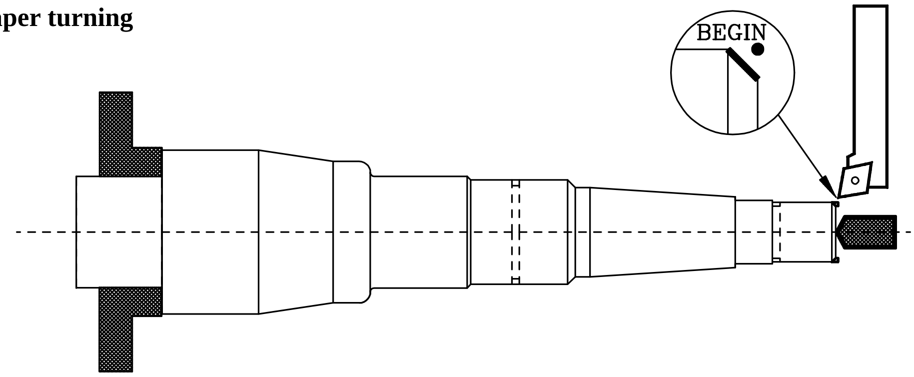
N70 P100=K19 P101=K0 P102=K19 P103=K-27 P4=K1 P6=K17 P19=K1 P20=K1
N75 G67 N0 ;Turning



N80 P100=K17 P101=K0 P102=K17 P103=K-17 P4=K1 P6=K16 P19=K1 P20=K1
N85 G67 N0   ;Turning



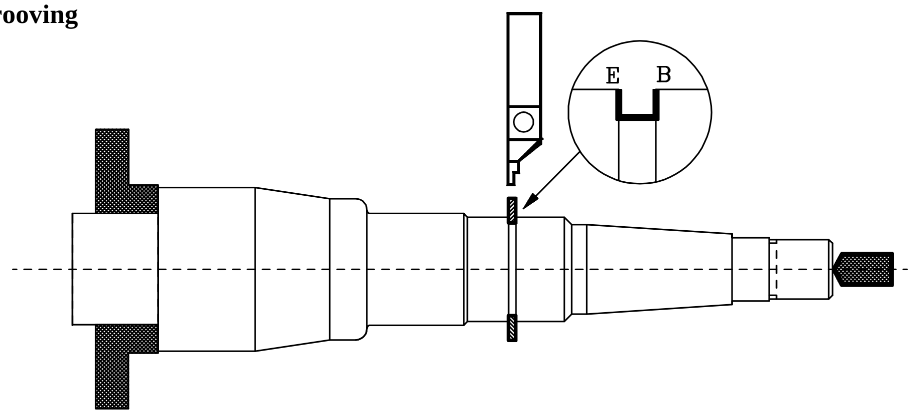
N90 P100=K16 P101=K0 P4=K1 P6=K14 P7=K100 P10=K4 P18=K2 P19=K13 P20=K1
N95 G67 N2   ;Taper turning



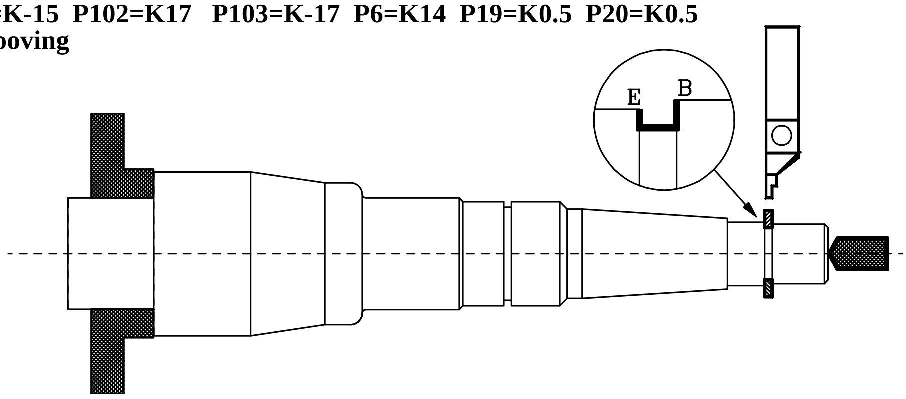
N100 G0 X80 Z50   ;Goes to tool change position
N101 T2.2   ;Selects tool "2"

N105 P12=K2 P13=K0.5   ;Width and dwell

N110 P100=K28 P101=K-85 P102=K28 P103=K-87 P4=K1.7 P6=K26.5 P19=K8 P20=K0
N115 G67 N5   ;Grooving

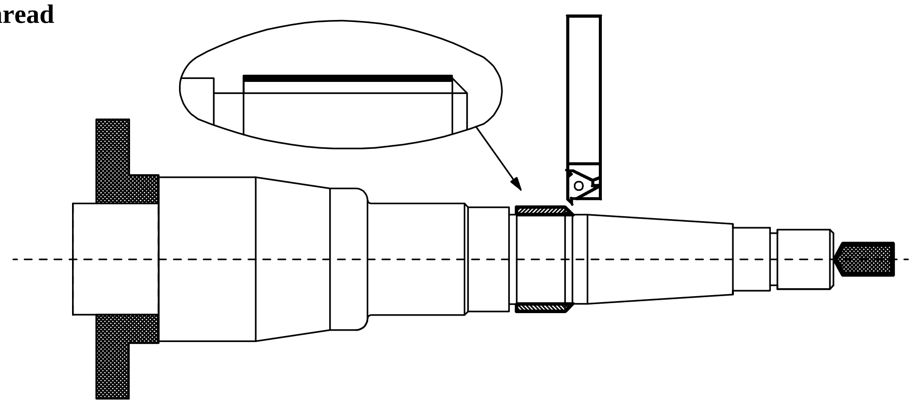


N120 P100=K16 P101=K-15 P102=K17   P103=K-17 P6=K14 P19=K0.5 P20=K0.5
N125 G67 N5   ;Grooving

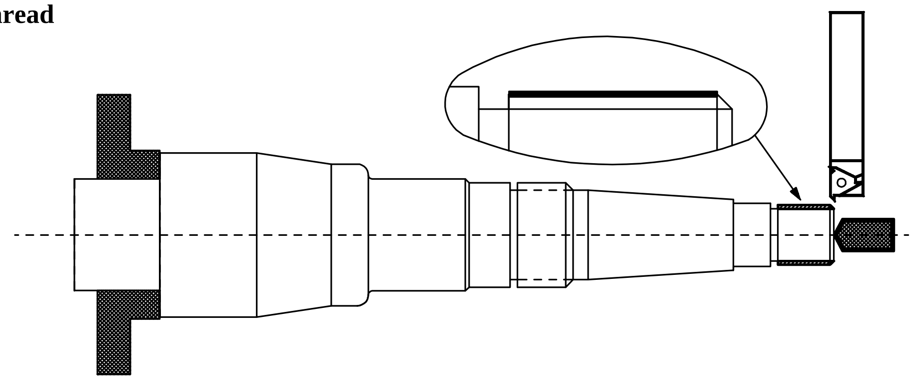


```
N130  G0  X80  Z50                                  ;Goes to tool change position
N131  T3.3                                          ;Selects tool "3"

N140  P4=K0.7  P11=K0  P16=K0  P128=K0    ;Pitch, outside thread, angle
N141  P100=K28  P101=K-70  P102=K28  P103=K-86  P8=K1.5  P14=K0.919  P19=K8  P20=K1
N145  G67 N3          ;Thread
```



```
N150  P4=K0.7  P11=K0  P16=K0  P128=K0    ;Pitch, outside thread, angle
N151  P100=K16  P101=K0  P102=K16  P103=K-16  P8=K1  P14=K0.613  P19=K15  P20=K1
N155  G67 N3          ;Thread
```



```
N160  G0  X80  Z50                                  ;Goest to withdrawal point
N170  M30                                           End of program
```

ERROR CODES

001 This error occurs when the first character of the block to be executed is not an "N".

002 Too many digits when defining a function in general.

003 A negative value has been assigned to a function which does not accept the (-) sign or an incorrect value has been given to a canned cycle parameter.

004 A canned cycle has been defined while function G02, G03 or G33 was active.

005 Parametric block programmed wrong.

006 There are more than 10 parameters affected in a block.

007 Division by zero.

008 Square root of a negative number.

009 Parameter value too large

010 * The range or the Constant Surface Speed has not been programmed

011 More than 7 "M" functions in a block.

012 This error occurs in the following cases:

> Function G50 is programmed wrong

> Tool dimension values too large.

> Zero offset values (G53/G59) too large.

013 Canned cycle profile defined incorrectly.

014 A block has been programmed which is incorrect either by itself or in relation with the program history up to that instant.

015 Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G53, G54, G55, G56, G57, G58, G59, G72, G73, G74, G92 and G93 must be programmed alone in a block.

016 The called subroutine or block does not exist or the block searched by means of special function F17 does not exist.

017 Negative or too large thread pitch value.

018 Error in blocks where the points are defined by means of angle-angle or angle-coordinate.

019 This error is issued in the following cases:

> After defining G20, G21, G22 or G23, the number of the subroutine it refers to is missing.

> The "N" character has not been programmed after function G25, G26, G27, G28 or G29.

> Too many nesting levels.

020 More than one spindle range have been defined in the same block.

021 This error will be issued in the following cases:

> There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.
> The corresponding axis has not been defined in the addressed block

022 An axis is repeated when programming G74.

023 K has not been programmed after G04.

025 Error in a definition block or subroutine call, or when defining either conditional or unconditional jumps.

026 This error is issued in the following cases:

> Memory overflow.

> Not enough free tape or CNC memory to store the part-program.

027 I//K has not been defined for a circular interpolation or thread.

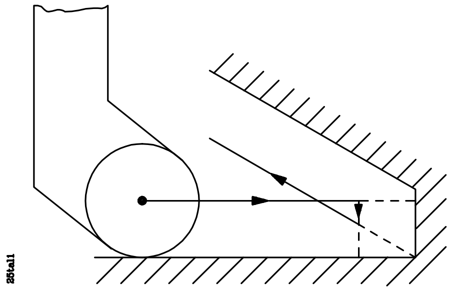
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

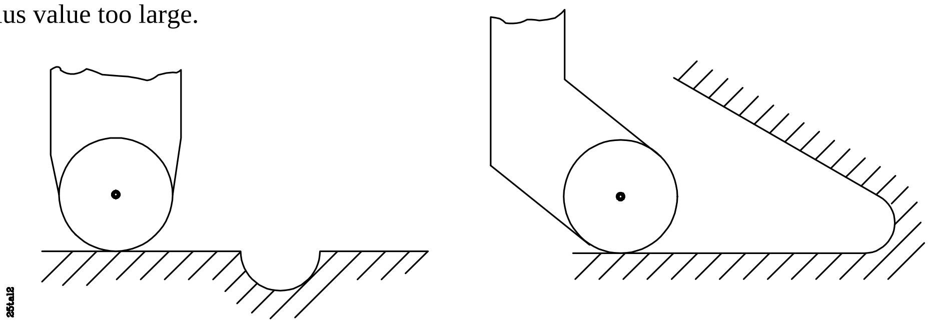
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

030 The programmed G function does not exist.

031 Tool radius value too large.



032 Tool radius value too large.



033 A movement of over 8388 mm or 330.26 inches has been programmed.

Example: Being the Z axis position Z-5000, if we want to move it to point Z5000, the CNC will issue error 33 when programming the block N10 Z5000 since the programmed move will be:
Z5000 - Z-5000 = 10000 mm.

In order to make this move without issuing this error, it must be carried out in two stages as indicated below:

```
N10 Z0          ; 5000 mm move
N10 Z5000       ; 5000 mm move
```

034 S or F value too large.

035 Not enough information for corner rounding, chamfering or compensation.

036 Repeated subroutine.

037 Function M19 programmed incorrectly.

038 Function G72 programmed incorrectly.

It must be borne in mind that if G72 is applied only to one axis, this axis must be positioned at part zero (0 value) at the time the scaling factor is applied.

039 This error occurs in the following cases:

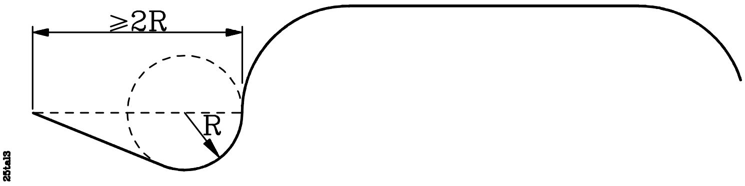
> More than 15 nesting levels when calling subroutines.

> A block has been programmed which contains a jump to itself. Example: N120 G25 N120.

040 The programmed arc does not go through the defined end point (tolerance 0.01mm) or there is no arc that goes through the points defined by G08 or G09.

041 This error is issued when programming a tangential entry as in the following cases:

> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> If the tangential entry is to be applied to an arc (G02, G03), The tangential entry must be defined in a linear block.

042 This error is issued when programming a tangential exit as in the following cases:

> There is no room to perform the tangential exit. A clearance of twice the rounding radius or greater is required.

> If the tangential exit is to be applied to an arc (G02, G03), The tangential exit must be defined in a linear block.

043 Polar origin coordinates (G93) defined incorrectly.

044 Function M45 S programmed wrong (speed of the live tool).

045 Function G36, G37, G38 or G39 programmed incorrectly.

046 Polar coordinates defined incorrectly.

047 A zero movement has been programmed during radius compensation or corner rounding.

048 Start or cancel tool radius compensation while in G02 or G03.

049 Chamfer programmed incorrectly.

050 Constant Surface Speed has been selected while the machine uses the BCD coded spindle speed output.

054 There is no tape in the cassette reader or the reader head cover is open or there is no disk in the FAGOR Floppy Disk Unit.

055 Parity error when reading or writing a cassette or a disk.

057 Write-protected tape or disk.

058 Sluggish tape or disk movement.

059 CNC communication error with the cassette reader or FAGOR Floppy Disk Unit.

060 Internal CNC hardware error. Consult with the Technical Service Department.

061 Battery error.

The memory contents will be kept for 10 more days (with the CNC off) from the moment this error occurs. The whole battery module located on the back must be replaced. Consult with the Technical Service Department.

Atention:

Due to danger of explosion or combustion: do not try to recharge the battery, do not expose it to temperatures higher than 100°C (232°F) and do not short the battery leads.

064 * External emergency input (pin 14 of connector I/O1) is activated.

065 * This error comes up if, while probing (G75), the programmed position is reached without receiving the probe signal.

066 * X axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

068 * Z axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

070 ** X axis following error.

072 ** Z axis following error.

074 ** "S" value (spindle speed) too large.

075 ** Feedback error at connector A1.

076 ** Feedback error at connector A2.

077 ** Feedback error at connector A3.

078 ** Feedback error at connector A4.

079 ** Feedback error at connector A5.

087 ** Internal CNC hardware error. Consult with the Technical Service Department.

088 ** Internal CNC hardware error. Consult with the Technical Service Department.

089 * All the axes have not been homed.

This error comes up when it is mandatory to search home on all axes after power-up. This requirement is set by machine parameter.

090 ** Internal CNC hardware error. Consult with the Technical Service Department.

091 ** Internal CNC hardware error. Consult with the Technical Service Department.

092 ** Internal CNC hardware error. Consult with the Technical Service Department.

093 ** Internal CNC hardware error. Consult with the Technical Service Department.

094 Parity error in tool table or zero offset table G53-G59.

095 ** Parity error in general parameters.

096 ** Parity error in Z axis parameters.

098 ** Parity error in X axis parameters.

099 ** Parity error in M table.

100 ** Internal CNC hardware error. Consult with the Technical Service Department.

101 ** Internal CNC hardware error. Consult with the Technical Service Department.

105 This error comes up in the following cases:

> A comment has more than 43 characters.

> A program has been defined with more than 5 characters.

> A block number has more than 4 characters.

> Strange characters in memory.

106 ** Inside temperature limit exceeded.

108 ** Error in Z axis leadscrew error compensation parameters.

110 ** Error in X axis leadscrew error compensation parameters.

111 * FAGOR LAN line error. Hardware installed incorrectly.

112 * FAGOR LAN error. It comes up in the following instances:

> When the configuration of the LAN nodes is incorrect.

> The LAN configuration has been changed. One of the nodes is no longer present (active).

When this error occurs, access the LAN mode, editing or monitoring, before executing a program block.

113 * FAGOR LAN error. A node is not ready to work in the LAN. For example:

> The PLC64 program is not compiled.

>A G52 type block has been sent to an 82CNC while it was in execution.

114 * FAGOR LAN error. An incorrect command has been sent out to a node.

115 * Watch-dog error in the periodic module.

This error occurs when the periodic module takes longer than 5 milliseconds.

116 * Watch-dog error in the main module.

This error occurs when the main module takes longer than half the time indicated in machine parameter "P729".

117 * The internal CNC information requested by activating marks M1901 thru M1949 is not available.

118 * An attempt has been made to modify an unavailable internal CNC variable by means of marks M1950 thru M1964.

119 Error when writing machine parameters, the decoded M function table and the leadscrew error compensation tables into the EEPROM memory.

This error may occur when after locking the machine parameters, the decoded M function table and the leadscrew error compensation tables, one tries to save this information into the EEPROM memory.

120 Checksum error when recovering (restoring) the machine parameters, the decoded M function table and leadscrew error compensation tables from the EEPROM memory.

Atention:

The **ERRORS** indicated with "*" behave as follows:

They stop the axis feed and the spindle rotation by cancelling the Enable signals and the analog outputs of the CNC.

They interrupt the execution of the part-program of the CNC if it was being executed.

The **ERRORS** indicated with "**" besides behaving as those with an "*", they activate the INTERNAL EMERGENCY OUTPUT.

Version 5.2 (March 1995)

1. P621(4). DIVIDING FACTOR FOR ELECTRONIC HANDWHEEL FEEDBACK SIGNALS

Machine parameter P621(4) is used with P602(4) and P621(5) which indicate the multiplying factor for the electronic handwheel feedback signals for the 1st and 2nd axis respectively.

Machine parameter P627(1) indicates whether all handwheel feedback signals are to be divided or not.
P621(4)=0 They are not divided.
P621(4)=1 All handwheel feedback signals are divided by two.

Examples for the X axis so the CNC assumes 100 pulses/turn with 25, 50 and 100 line handwheels:

25 line Fagor handwheel:	P602(4)=0 and P621(4)=0	25 x 4 / 1 = 100 lines
50 line Fagor handwheel:	P602(4)=1 and P621(4)=0	50 x 2 / 1 = 100 lines
100 line Fagor handwheel:	P602(4)=1 and P621(4)=1	100 x 2 / 2 = 100 lines

Version 5.6 (June 1996)

1. JOG WITH MASTER HANDWHEEL

With this feature it is possible to jog the machine with the Master Handwheel once the path has been defined.

Requirements:

The control of the "Jog with Master Handwheel" is carried out with the Second Handwheel. Therefore, the machine must have two electronic handwheels and none mechanical.

Parameter setting:

Machine parameter "P622(6)" indicates whether this feature is being used or not.

P622(6) = 0 "Jog with Master handwheel" **is not** available.
P622(6) = 1 "Jog with Master handwheel" **is** available.

As stated above, the control of the "Jog with Master Handwheel" is carried out with the Second Handwheel. Therefore, the machine must have two electronic handwheels and none mechanical. This means that:

P621(7)=1 The machine does not use mechanical handwheels.
P622(3)=0 It uses two electronic handwheels.
P609(1)=0 The first handwheel is not a FAGOR 100P model.

The Master handwheel is connected via connector "A4". It admits both sine-wave and square-wave differential signals. This implies setting the following machine parameters as follows:

P621(6) Counting direction of the "Master Handwheel".
P621(3) Feedback units of the "Master Handwheel".
P621(1,2) Feedback resolution of the "Master Handwheel".
P621(5) Feedback multiplying factor for the "Master Handwheel".

Selection:

a) CNC Models: 800TI and 800TGI. From the PLCI.

Once all machine parameters have been set, PLCI output O39 must be used to enable or disable the "Jog with Master Handwheel" feature.

Parameter P622(6)	PLCI output O39	"Jog with Master Handwheel"
P622(6) = 0	-----	Feature not available
P622(6) = 1	O39 = 0	Feature disabled
P622(6) = 1	O39 = 1	Feature enabled

b) CNC Models: 800T and 800TG. Using pin 11 of connector "I/O 1".

Once all machine parameters have been set, the "Jog with Master Handwheel" input (pin 11 of I/O 1) must be used to enable or disable the "Jog with Master Handwheel" feature.

Parameter P622(6)	Pin 11 I/O1	"Jog with Master Handwheel"
P622(6) = 0	-----	Feature not available
P622(6) = 1	Pin 11 at 0Vdc	Feature disabled
P622(6) = 1	Pin 11 at 24Vdc	Feature enabled

Basic Operation. (P622(6)=1, O39=1)

a) *When the machine is stopped.*

Only the first handwheel is enabled, the second one (Master) is disabled.
Therefore, only the X axis may be jogged with the handwheels.

b) *When the machine is running (CNC in Execution).*

The axes do not start moving until the Master Handwheel is turned.

The axis feedrate depends on the turning speed of the Master Handwheel. If it stops, the axes also stop.

If the Master Handwheel is turned in the opposite direction, the axes also invert their moving direction (Return Function for one block only).

c) *The "Jog with Master Handwheel" feature may be used with any type of execution, be it a cycle, an ISO-coded program, a Chamfer, etc.*

Usually, with the CNC in execution, the first handwheel is disabled, except for the semi-automatic mode of the automatic operations: "Taper Turning" and "Rounding".

On both Semi-automatic operations, the Master Handwheel controls the feedrate of the tool path and the First Handwheel will move the X axis.

"Jog with Master Handwheel" feature disabled. (P622(6)=1, O39=0)

When this feature is disabled, PLC output O39 is set to "0" and the handwheels operate like until now (as on previous versions).

2. DYNAMIC GRAPHICS WHILE IN EXECUTION

Until now, with the 800T CNC, a part program could be simulated (verified) graphically before running it.

From now on, it is also possible to display dynamic graphics of the machining path while in execution.

Requirements:

This application requires a 800TG or 800TGI CNC model (G for graphics).

Operation:

When running an Automatic Operation, a Part program, the ISO-coded program in Automatic or Single Block mode, it is now possible to display the machining path dynamically in the execution stage.

To do this, once the execution has started, the following keys may be pressed:

- **[4]** The CNC displays the graphics screen.
- **[3]** The CNC shows the "Command, Actual and To-go" coordinates of the axes and, at the top of the screen, the values of the Arithmetic parameters.
- **[2]** The CNC displays the Following Error (axis lag) in large characters.
- **[1]** The CNC displays the actual axis position in large characters.
- **[0]** The CNC returns to the standard display.

3. WORK ZONE / EXCLUSION ZONE

With this feature it is possible to select a predefined zone as work zone or exclusion zone from the PLCI.

Requirements:

This application requires an 800TI or 800TGI CNC model since one must use outputs O46 and O47 of the PLCI to set the zone as work zone or exclusion zone.

Parameter setting:

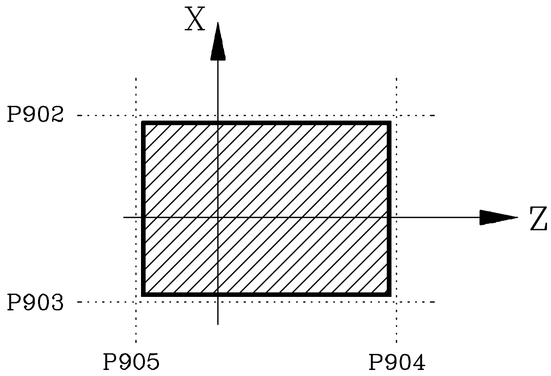
Machine parameter "P622(5)" indicates whether the CNC allows setting a work zone or an exclusion zone.

P622(5) = 0 This feature **is not** available.
P622(5) = 1 This feature **is** available.

When using this feature "P622(5)=1" the following machine parameters must also be set to define the zone to be considered either as work zone or exclusion zone.

P902 Most positive X coordinate
P903 Least positive X coordinate
P904 Most positive Z coordinate
P905 Least positive Z coordinate

The CNC must be turned off and back on in order for the new parameter values to be assumed.



Selection:

Once all machine parameters have been set, PLCI outputs O46 and O47 must be used to select the predefined zone either as a work zone or as an exclusion zone.

PLCI Output O46	PLCI Output O47	"Work Zone or Exclusion Zone" Feature
O46 = 0	O47 = 0	Feature disabled
O46 = 0	O47 = 1	Zone enabled as Work Zone (No-exit zone)
O46 = 1	O47 = 0	Zone enabled as Exclusion Zone (No-entry zone)
O46 = 1	O47 = 1	Feature disabled

Basic operation. "P622(5)=1"

On power-up, the CNC assumes the zone set by machine parameters "P902, P903, P904 and P905".

Nevertheless, the zone boundaries may be changed via part-program by allocating the new values to the following arithmetic parameters:

P206 Most positive X coordinate
P207 Least positive X coordinate
P208 Most positive Z coordinate
P209 Least positive Z coordinate

The CNC will then assume these new values; but it will not modify the actual settings of machine parameters "P902, P903, P904 and P905".

On the other hand, it must be kept in mind that, on power-up, the CNC will reset these zone boundaries to the values set by machine parameters.

As described earlier, this predefined zone may be enabled either as a work zone (no-exit) or as an exclusion zone (no-entry) from the PLCI by means of outputs O46 and O47.

When set as a work zone, the CNC acts as follows:

· The axes cannot be jogged **out of** this zone by using the jog keys or the handwheels.
· If attempted to do so during execution, the CNC will issue error 67: «X, Z Limit Error»

When set as an exclusion zone, the CNC acts as follows:

· The axes cannot be jogged **into** this zone by using the jog keys or the handwheels.
· If attempted to do so during execution, the CNC will issue error 67: «X, Z Limit Error»

4. MANUAL SPINDLE GEAR CHANGERS

Operation on previous versions

To manually change the spindle speed range (gear), machine parameter "P601(1)" had to be set to "0".

When the new selected spindle speed "S" involved a gear change, the CNC displayed a message indicating which range had to be selected.

The operator had to proceed as follows:

1st- Stop the spindle
2nd- Manually change gears
3rd- Restore spindle rotation
4th- Press **[ENTER]**

The CNC resumed program execution.

Operation on current and future versions

To manually change the spindle speed range (gear), machine parameter "P601(1)" must set to "0".

When the new selected spindle speed "S" involves a gear change, the CNC displays a message indicating which range has to be selected.

The operator must proceed as follows:

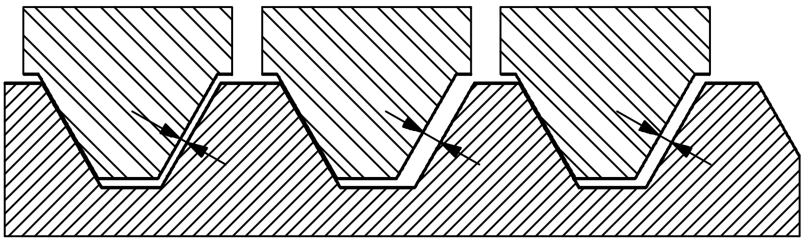
1st- Manually change gears
2nd- Press **[ENTER]**

The CNC restores spindle rotation and resumes program execution.

5. VARIABLE BACKLASH COMPENSATION

Until now, the 800T CNC allowed for a single leadscrew backlash compensation.

From now on, it is also possible to compensate for motion-reversal backlash depending on the particular backlash areas of the axes.



Requirements:

The leadscrew error compensation tables are now used for leadscrew error compensation and for this "Variable Backlash Compensation" (at the same time).

Parameter setting:

Machine parameters "P622(7)" and "P622(8)" indicate whether this feature is available or not.

P622(7)	= 0	Not available for the Z axis.
P622(7)	= 1	Available for the Z axis.
P622(8)	= 0	Not available for the X axis.
P622(8)	= 1	Available for the X axis.

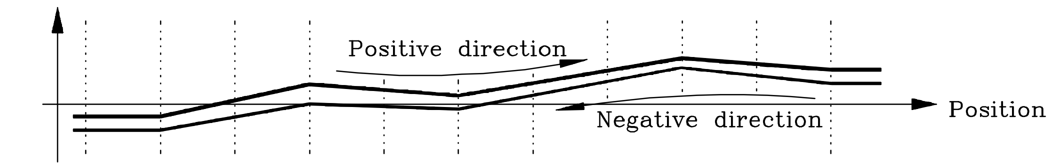
In order to use "Variable Backlash Compensation", regular leadscrew compensation must also be activated.

P605(2) = 0	X axis Leadscrew error compensation (0= No, 1= Yes)
P605(1) = 0	Z axis Leadscrew error compensation (0= No, 1= Yes)

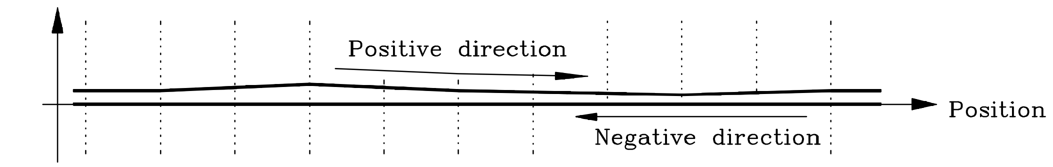
Operation:

The first 15 points of the table are for the positive direction and the other 15 for the negative direction.

When compensating for leadscrew error, the amount of backlash is the difference between both graphs.



When leadscrew error compensation is not to be used, all the values of one of the tables must be set to "0"; thus, the other graph will correspond to the leadscrew backlash.

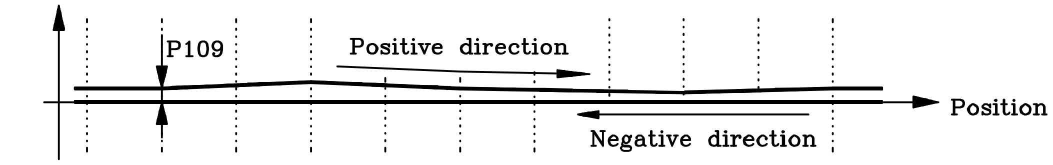


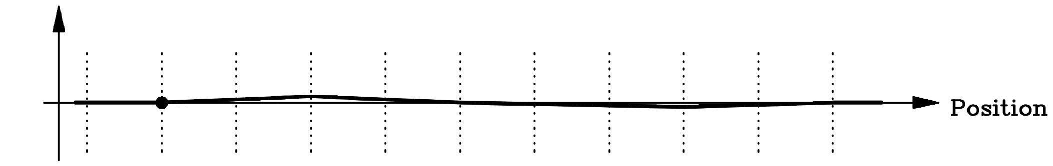
Notes: Both graphs must meet all the requirements of the leadscrew error compensation tables.

One of these requirements states that the Machine Reference Zero (home) must be assigned an error of "0".

If the leadscrew has some backlash at this Machine Reference Zero point, that amount of backlash must be allocated to machine parameter P109 or P309 (Backlash for the X axis or Z axis) and all the remaining points of the table must be offset by that amount.

Example:





Operation:

When using "Variable Backlash Compensation", the CNC operates with leadscrew error compensation and, therefore, it applies at all times the backlash compensation set in the table for that point and in the machining direction.

When the axis movement is reversed, the CNC swaps graphs restoring or applying the amount of backlash compensation and direction corresponding to that point.

Version 5.7 (July 1996)

1. WORK ZONE / EXCLUSION ZONE DETECTION

When using this feature, the CNC prevents the axes from exiting or entering this zone while jogging them with either the jog keys or the electronic handwheel.

The operator might suspect a malfunction since the CNC does not issue any message. From this version on, in these cases, the CNC will behave as follows:

* When the zone has been set as a Work Zone, the CNC will set PLCI input I46 high when trying to exit the selected zone.

* When the zone has been set as a Work Zone, the CNC will set PLCI input I46 high when trying to enter the selected zone.

2. RESUME EXECUTION AT MID-PROGRAM

If while a part, the program is interrupted (due to a power failure, etc.), it is now possible to resume execution from the interrupted program on. This way, there is no longer need to repeat the whole program, thus saving considerable amount of time.

To resume program execution, follow these steps:

1st Select the DRO mode, the one appearing on CNC power-up after the "General Test Passed". In this mode, no cycle appears selected.

2nd Press **[RECALL]** to open the part-programs window.

3rd Select the part that was running. Use the up and down arrow keys to position over the desired part program and press **[RECALL]**.

4th Use the up and down arrow keys to select the operation being interrupted and press

The CNC will executed the selected operation and it will resume the part-program running it to the end.

Version 6.1 (January 1997)

1. NEW LANGUAGES (Taiwanese and Portuguese)

Machine parameter P99 P99 = 5 Portuguese P99 = 6 Taiwanese

2. MODIFICATIONS ON THE OPERATION WITH A MASTER HANDWHEEL

The operation with the master handwheel is now as follows:

a) *When the machine is stopped.*

Only the first handwheel is enabled, the second one (master) does not work.
Therefore, only the X axis can be jogged with a handwheel.

b) *When the machine is running (CNC in Execution).*

Only the Master handwheel is enabled, the first handwheel does not work.

The axes start moving when turning the Master Handwheel.

The feedrate of the axes depend on the turning speed of the Master Handwheel
When the handwheel stops, the machine also stops.

When the Master Handwheel is turned in the opposite direction, the CNC also reverses the moving direction (Retrace Function of a single block).

c) *Semiautomatic Rounding Operation*

The Semiautomatic Rounding operation starts when turning the Master Handwheel. From that moment on, the first handwheel is inoperative

When stopping the Master Handwheel, the execution is interrupted.
When turning the Master handwheel again, execution is resumed. The turning direction of the handwheel cannot be changed.

When the operation is over, the CNC ignores the turning of the Master Handwheel for 1.4 seconds. Thus preventing another operation from being started.
After this time, when the Master Handwheel is turned, the CNC starts executing a new operation in the indicated direction.

d) *Semiautomatic Taper Turning Operation*

The Semiautomatic Taper Turning Operation starts when turning the Master Handwheel. From that moment on, the first handwheel is inoperative.

When stopping the Master Handwheel, the execution is interrupted.
When turning the Master handwheel again, execution is resumed.

When turning the Master Handwheel in the opposite direction, the operation is over. A new turn of the Master Handwheel in any direction implies the execution of a new operation in the indicated direction.

3. SOFTWARE VERSION OF THE CNC

From this version on, when accessing the EPROM checksum screen,

[Auxiliary Modes] [Special Modes] [8]

The CNC will show the checksum of each EPROM and the Software version of the CNC. For example: Version 6.1

Version 6.4 (May 1997)

1. TOOL CHANGE INDICATOR FOR THE PLC (I97)

On machines with a manual tool changer, when the CNC detects that the tool must be changed, it interrupts the execution and it displays a message for the operator to proceed with the tool change.

Certain precautions must be taken sometimes when changing tools. Those conditions must be handled by the PLC.

Therefore, from this version on, when the CNC displays the tool change warning message, it also activates the PLC input I97 and it cancels it when the message is removed.

Version 6.6 (November 1997)

1. HANDLING FEEDBACK SYSTEMS WITH CODED Io (semi-absolute)

Machine parameters

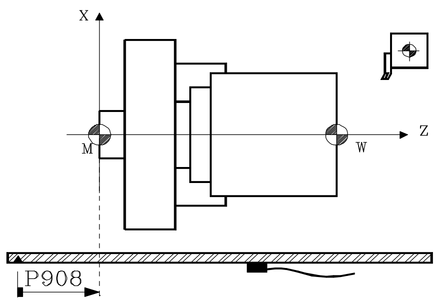
P608(5), P608(8) Type of Home marker signal of the feedback system. X, Z axes. (0 = normal "Io", 1 = coded "Io")
P608(3), P608(6) Period of the coded Io signal. X, Z signal. (0 = 20mm Period, 1 = 100 mm Period)
P608(4), P608(7) Increasing Io sequence with positive or negative count. X, Z axes
(0 = Increasing Io with positive count, 1 = increasing Io with negative count)

Scale	P608 (5)	P608(3)	P608(4)
COS	1	0	1
COC	1	0	0
COX	1	0	0
COVS	1	0	1
COVC	1	0	0
COVX	1	0	0

Scale	P608 (5)	P608(3)	P608(4)
MOVS	1	0	0
MOVC	1	0	0
MOVX	1	0	0
FOT	1	1	0
FOS	1	1	0
FOC	1	1	0

P908, P909 Scale offset or Home position (Machine Reference: M) with respect to the Scale "zero" point. X, Z axes.

Linear transducers (scales) with coded Io have a graduated scale with their own Scale "zero" point. Therefore, a 20 mm or 100 mm move is enough to know the axis position with respect to the Scale "zero" point.



Reference point.

When the feedback system has a coded Io, this point is only used when leadscrew error compensation is needed. The leadscrew error on this point must be "0".

Scale offset setting

The scale offset must be adjusted on one axis at a time. The following procedure is recommended:

* Indicate by parameters "P600(7) & P600(6)" the up or down flank of the marker pulse (Io) of the feedback system.
* Indicate by parameters "P618(8) & P618(7)" the home searching direction for the axes.
* Set parameters "P807 and P808" with the home searching feedrate for the axes.
* Set parameters " P908 and P909" to "0" (scale offset).

* Position the axis at the right position and execute the home search command for that axis..

  [X] or [Z], [up arrow] keys and

  When done with the home search, the CNC will display the axis position with respect to the scale's Zero point.

* After moving the axis to Machine Reference Zero point (home) or up to a known position with respect to home, Jot down the position reading of the CNC for that known position.

  The value to allocate to the machine parameter setting the scale's offset must be calculated with the following formula:

  Value = CNC reading at that point - Home coordinate of that point

  Example for the X axis: If the known position is physically located at 230 mm from Machine Reference Zero (home) and the CNC shows that its position is 423.5 mm, the scale offset will be:

  Machine parameter P908 = 423.5 - 230 = 193.5 mm.

* After setting the machine parameter with this value, press the [RESET] key so that value is assumed by the CNC.

* A new home search must be carried out for the proper values to be assumed for that axis.

2. THREADING WITH CONSTANT PENETRATION PASSES

From this version on, the penetration of each pass will depend on the sign assigned to parameter **D**

When **D** positive, the penetration of each pass depends on the corresponding pass (**D** n)
With **D** negative, the penetration passes remain constant with the absolute value of parameter **D**

3. GENERATING AN ISO-CODED PROGRAM

With this CNC, the ISO code (low level) for an operation or a part-program may be generated.

To use this feature, machine parameter "P623(2)" must be set to "1".

This ISO program always has the number: 99996 and can be stored either at the CNC or at a PC.

Program 99996 is a special user program in ISO code and can be:
Generated from an operation or a part-program.
Edited at the CNC itself via menu option: "Auxiliary Modes - Edit program 99996"
Loaded into the CNC after being generated at a PC.

Generating the ISO program (99996) at the CNC.

This CNC has 7 K of memory space to store program 99996. If the generated program is larger than that, the CNC will issue the relevant error message.

To generate program 99996, proceed as follows:

* If it is an operation, select or define the desired operation.

* If it is a part-program, select the desired one in the part-program directory and place the cursor on its header ("PART 01435". A listing of the operations it consists of must appear).

* Press the keystroke sequence: [AUX] [7]. The CNC will show the graphic simulation screen.

* Press . The CNC starts simulating the part and generating its ISO-coded program 99996.

* When done with the simulation, program 99996 stored in CNC memory will contain all simulated blocks in ISO code.

Generating the ISO program (99996) at a PC

Usually, the 99996 program generated from a part-program exceeds the available memory space of the CNC.

By using "DNC30", this program may be generated at a PC.

To do this, proceed as follows:

* Activate DNC communications and execute the DNC30 program at the PC.

* Select at the PC the menu option: "Program Management - Receive Digitizing".

* At the CNC, select the operation or place the cursor on the part-program header ("PART 01435"). A listing of the operations it consists of must appear).

* Press [AUX] [8]. The CNC will display the graphic simulation screen.

* Press . The CNC starts simulating the part and generating program 99996.

* When done with the simulation, the 99996 program generated at the PC will contain all the blocks simulated by the CNC in ISO code.

  This program can be executed at the CNC through the menu option: "Execute infinite program" of the DNC30.

4. MACHINE SAFETY REGULATION

This CNC offers the following features to comply with machine safety regulations.

Enabling of the CYCLE START key from the PLC

This feature is available when machine parameter "P619(7)=1"

PLC output O25 indicates whether the CYCLE START key is enabled (=1) or not (=0)

Axes movements affected by Feed-Hold. (It was already available)

Feed-Hold input, pin 15 of connector I/O 1, must be normally high.

If while moving the axes, the Feed-Hold input is brought low, the CNC keeps the spindle turning and stops the axes with 0V or velocity command (analog signal) and keeping their enables ON.

When this signal is brought back up, the CNC will resume the movement of the axes.

Axes jogging feedrate limited by PLC.

This feature is available when machine parameter "P619(7)=1"

When activating PLC output O26, the CNC assumes the feedrate set by machine parameter "P812"

Handwheel managed by the PLC.

Machine parameter "P623(3)" indicates whether the axes movements with handwheels are affected by Feed-Hold (=1) or not (=0)

Machine parameter "P622(1)" indicates whether the multiplying factor indicated by the MFO switch position is applied (=0) or the one indicated by the PLC outputs O44 and O45 (=1) (already available)

Spindle control from the PLC.

This feature is available when "P619(7)=1"

Output O27 =1 "tells" the CNC to apply the spindle analog voltage set by the PLC. The value of this analog signal is set at register R156 and sent to the CNC by mark M1956.

R156= 0000 1111 1111 1111 => + 10V.	R156= 0001 1111 1111 1111 => - 10V.
R156= 0000 0111 1111 1111 => + 5V.	R156= 0001 0111 1111 1111 => - 5V.
R156= 0000 0011 1111 1111 => + 2,5V.	R156= 0001 0011 1111 1111 => - 2,5V.
R156= 0000 0000 0000 0000 => + 0V.	R156= 0001 0000 0000 0000 => - 0V.

Also, PLC output O43, lets you control the rotation of the spindle. (Already available).
It must be normally low.
If it is brought up, the CNC stops the spindle.
When it is brought back up, the CNC restarts the spindle.

Information for the PLC on the status of the machine reference (home) search

I88 Home search in progress.
I100 X axis home search done.
I101 Z axis home search done.

R120 The Lower half of this register indicates the code pressed.
This value is maintained for 200 milliseconds unless another key is pressed before then.
This register may be canceled from the PLC after being processed.

R121 bit 1 Indicates that the Turning operation is selected (=1)
bit 2 Indicates that the Facing operation is selected (=1)
bit 3 Indicates that the Taper Turning operation is selected (=1)
bit 4 Indicates that the Rounding operation is selected (=1)
bit 5 Indicates that the Threading operation is selected (=1)
bit 6 Indicates that the Grooving operation is selected (=1)
bit 7 Indicates that the Profiling operation is selected (=1)
bit 8 Indicates that the Auxiliary Modes option is selected (=1)
bit 9 Indicates that the Tool Calibration option is selected (=1)
bit 10 Indicates that the Multiple Drilling operation is selected (=1)
bit 11 Indicates that the Simple Drilling / Tapping operation is selected (=1)
bit 12 Indicates that the Slot milling (keyway) operation is selected (=1)
bit 13 Indicates that the Tool Inspection mode is selected (=1)
bit 14 Indicates that the Graphic Simulation mode is selected (=1)
bit 16 Indicates that the mode for the following cycle parameters: "Finishing pass, finishing feedrate, finishing tool and safety distances on X and Z " is selected (=1)

Version 6.8 (March 1998)

1. NEW LANGUAGES (SWEDISH AND NORWEGIAN)

The languages that can be selected with machine parameter P99 are:

Spanish ...... (P99=0)	German ...... (P99=1)	English .... (P99=2)	French .... (P99=3)	Italian .... (P99=4)
Portuguese .. (P99=5)	Taiwanese .. (P99=6)	Swedish ..... (P99=7)	Norwegian . (P99=8)	

2. 1000 LINE ENCODER AS 1250 LINE AS ENCODER

This feature permits the CNC adapt a 1000 line encoder to be used as 1250 line encoder.

P623(7) Adapts the X axis feedback encoder (0=No, 1=Yes)
P623(8) Adapts the Z axis feedback encoder (0=No, 1=Yes)

A typical case: Having a 1000 line for a 5 mm pitch ballscrew .

The calculations necessary to set the axis resolution will be made with the selected pulses (1000 or 1250)

3. CROSS COMPENSATION

Cross compensation is used for compensating the measuring error suffered by the X axis when moving the Z axis.

P623(6) Cross compensation applied on to the X axis (0=No, 1=Yes)

When using cross compensation, no leadscrew compensation may be applied on the X axis (only on to the Z axis) since its corresponding table is being used for cross compensation with the following values:

P00 = X: ?????.??? P01 = DX: ????.???

in order to properly apply cross compensation, set: P605(2)=1 and P623(6)=1.

Note: The cross compensation table must meet the same requirements as those for the leadscrew error compensation. See section 3.8.4 of the installation manual.

4. PLCI. INPUT I104

When the Feedrate Override Switch on the operator panel is set on one of the handwheel positions (x1, x10, x100), input I104 is set to "1".

Fagor Automation S. Coop.
Mondragón (Spain)

Fagor Automation Catalunya
Barcelona (Spain)

Fagor Industriecommerz GmbH
Göppingen (Germany)

Fagor Italia S.R.L.
Cassina de Pecchi (Milano - Italy)

Fagor Automation Systemes
Clermont Ferrand (France)

Fagor Automation Suisse S. à.r.l.
Renan (Belgium)

Fagor Automation Ltda.
Leça da Palmeira (Potugal)

Fagor Automation (Asia) Ltd.
Hong Kong (China P.R.)

Fagor Automation (Asia) Ltd.
Taichung (Taiwan)

Fagor Automation (S) Pte. Ltd.
Singapore

Beijing Fagor Automation Equipment, Co. Ltd.
Beijing (China P.R.)

Fagor Automation, Shanghai Rep. Office
Shanghai (China P.R.)

Fagor Automation do Brasil Com. Imp. Exp. Ltda.
São Paulo (Brazil)

Fagor Automation Corp.
Elk Grove Village (Chicago - USA)

Fagor Automation West Coast
Newport Beach (California - USA)

Fagor Automation East Coast
New Jersey (USA)

Fagor Automation Montreal Office
Montreal (Canada)

Fagor Automation Canada
Mississauga (Canada)

FAGOR

CNC 800 T

New Features (Ref.9902in)

Version 5.2 (March 1995)

1. P621(4). DIVIDING FACTOR FOR ELECTRONIC HANDWHEEL FEEDBACK SIGNALS

Machine parameter P621(4) is used with P602(4) and P621(5) which indicate the multiplying factor for the electronic handwheel feedback signals for the 1st and 2nd axis respectively.

Machine parameter P627(1) indicates whether all handwheel feedback signals are to be divided or not.
P621(4)=0 They are not divided.
P621(4)=1 All handwheel feedback signals are divided by two.

Examples for the X axis so the CNC assumes 100 pulses/turn with 25, 50 and 100 line handwheels:

25 line Fagor handwheel:	P602(4)=0 and P621(4)=0	25 x 4 / 1 = 100 lines
50 line Fagor handwheel:	P602(4)=1 and P621(4)=0	50 x 2 / 1 = 100 lines
100 line Fagor handwheel:	P602(4)=1 and P621(4)=1	100 x 2 / 2 = 100 lines

Version 5.6 (June 1996)

1. JOG WITH MASTER HANDWHEEL

With this feature it is possible to jog the machine with the Master Handwheel once the path has been defined.

Requirements:

The control of the "Jog with Master Handwheel" is carried out with the Second Handwheel. Therefore, the machine must have two electronic handwheels and none mechanical.

Parameter setting:

Machine parameter "P622(6)" indicates whether this feature is being used or not.

P622(6) = 0 "Jog with Master handwheel" **is not** available.
P622(6) = 1 "Jog with Master handwheel" **is** available.

As stated above, the control of the "Jog with Master Handwheel" is carried out with the Second Handwheel. Therefore, the machine must have two electronic handwheels and none mechanical. This means that:

P621(7)=1 The machine does not use mechanical handwheels.
P622(3)=0 It uses two electronic handwheels.
P609(1)=0 The first handwheel is not a FAGOR 100P model.

The Master handwheel is connected via connector "A4". It admits both sine-wave and square-wave differential signals. This implies setting the following machine parameters as follows:

P621(6) Counting direction of the "Master Handwheel".
P621(3) Feedback units of the "Master Handwheel".
P621(1,2) Feedback resolution of the "Master Handwheel".
P621(5) Feedback multiplying factor for the "Master Handwheel".

Selection:

a) CNC Models: 800TI and 800TGI. From the PLCI.
Once all machine parameters have been set, PLCI output O39 must be used to enable or disable the "Jog with Master Handwheel" feature.

Parameter P622(6)	PLCI output O39	"Jog with Master Handwheel"
P622(6) = 0	-----	Feature not available
P622(6) = 1	O39 = 0	Feature disabled
P622(6) = 1	O39 = 1	Feature enabled

b) CNC Models: 800T and 800TG. Using pin 11 of connector "I/O 1".

Once all machine parameters have been set, the "Jog with Master Handwheel" input (pin 11 of I/O 1) must be used to enable or disable the "Jog with Master Handwheel" feature.

Parameter P622(6)	Pin 11 I/O1	"Jog with Master Handwheel"
P622(6) = 0	-----	Feature not available
P622(6) = 1	Pin 11 at 0Vdc	Feature disabled
P622(6) = 1	Pin 11 at 24Vdc	Feature enabled

Basic Operation. (P622(6)=1, O39=1)

a) *When the machine is stopped.*

Only the first handwheel is enabled, the second one (Master) is disabled.
Therefore, only the X axis may be jogged with the handwheels.

b) *When the machine is running (CNC in Execution).*

The axes do not start moving until the Master Handwheel is turned.

The axis feedrate depends on the turning speed of the Master Handwheel. If it stops, the axes also stop.

If the Master Handwheel is turned in the opposite direction, the axes also invert their moving direction (Return Function for one block only).

c) *The "Jog with Master Handwheel" feature may be used with any type of execution, be it a cycle, an ISO-coded program, a Chamfer, etc.*

Usually, with the CNC in execution, the first handwheel is disabled, except for the semi-automatic mode of the automatic operations: "Taper Turning" and "Rounding".

On both Semi-automatic operations, the Master Handwheel controls the feedrate of the tool path and the First Handwheel will move the X axis.

"Jog with Master Handwheel" feature disabled. (P622(6)=1, O39=0)

When this feature is disabled, PLC output O39 is set to "0" and the handwheels operate like until now (as on previous versions).

2. DYNAMIC GRAPHICS WHILE IN EXECUTION

Until now, with the 800T CNC, a part program could be simulated (verified) graphically before running it.

From now on, it is also possible to display dynamic graphics of the machining path while in execution.

Requirements:

This application requires a 800TG or 800TGI CNC model (G for graphics).

Operation:

When running an Automatic Operation, a Part program, the ISO-coded program in Automatic or Single Block mode, it is now possible to display the machining path dynamically in the execution stage.

To do this, once the execution has started, the following keys may be pressed:

[4] The CNC displays the graphics screen.
[3] The CNC shows the "Command, Actual and To-go" coordinates of the axes and, at the top of the screen, the values of the Arithmetic parameters.
[2] The CNC displays the Following Error (axis lag) in large characters.
[1] The CNC displays the actual axis position in large characters.
[0] The CNC returns to the standard display.

3. WORK ZONE / EXCLUSION ZONE

With this feature it is possible to select a predefined zone as work zone or exclusion zone from the PLCI.

Requirements:

This application requires an 800TI or 800TGI CNC model since one must use outputs O46 and O47 of the PLCI to set the zone as work zone or exclusion zone.

Parameter setting:

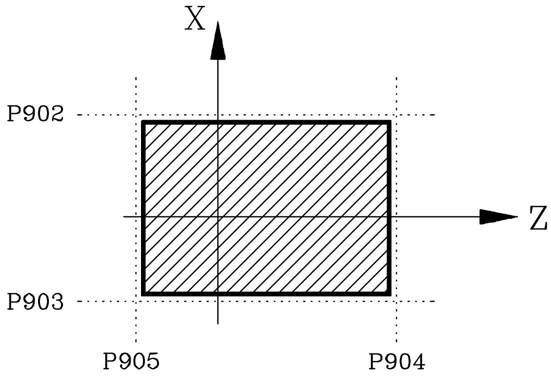
Machine parameter "P622(5)" indicates whether the CNC allows setting a work zone or an exclusion zone.

P622(5) = 0 This feature **is not** available.
P622(5) = 1 This feature **is** available.

When using this feature "P622(5)=1" the following machine parameters must also be set to define the zone to be considered either as work zone or exclusion zone.

P902 Most positive X coordinate
P903 Least positive X coordinate
P904 Most positive Z coordinate
P905 Least positive Z coordinate

The CNC must be turned off and back on in order for the new parameter values to be assumed.



Selection:

Once all machine parameters have been set, PLCI outputs O46 and O47 must be used to select the predefined zone either as a work zone or as an exclusion zone.

PLCI Output O46	PLCI Output O47	"Work Zone or Exclusion Zone" Feature
O46 = 0	O47 = 0	Feature disabled
O46 = 0	O47 = 1	Zone enabled as Work Zone (No-exit zone)
O46 = 1	O47 = 0	Zone enabled as Exclusion Zone (No-entry zone)
O46 = 1	O47 = 1	Feature disabled

Basic operation. "P622(5)=1"

On power-up, the CNC assumes the zone set by machine parameters "P902, P903, P904 and P905".

Nevertheless, the zone boundaries may be changed via part-program by allocating the new values to the following arithmetic parameters:

P206 Most positive X coordinate
P207 Least positive X coordinate
P208 Most positive Z coordinate
P209 Least positive Z coordinate

The CNC will then assume these new values; but it will not modify the actual settings of machine parameters "P902, P903, P904 and P905".

On the other hand, it must be kept in mind that, on power-up, the CNC will reset these zone boundaries to the values set by machine parameters.

As described earlier, this predefined zone may be enabled either as a work zone (no-exit) or as an exclusion zone (no-entry) from the PLCI by means of outputs O46 and O47.

When set as a work zone, the CNC acts as follows:

· The axes cannot be jogged **out of** this zone by using the jog keys or the handwheels.
· If attempted to do so during execution, the CNC will issue error 67: «X, Z Limit Error»

When set as an exclusion zone, the CNC acts as follows:

· The axes cannot be jogged **into** this zone by using the jog keys or the handwheels.
· If attempted to do so during execution, the CNC will issue error 67: «X, Z Limit Error»

4. MANUAL SPINDLE GEAR CHANGERS

Operation on previous versions

To manually change the spindle speed range (gear), machine parameter "P601(1)" had to be set to "0".

When the new selected spindle speed "S" involved a gear change, the CNC displayed a message indicating which range had to be selected.

The operator had to proceed as follows:

1st- Stop the spindle
2nd- Manually change gears
3rd- Restore spindle rotation
4th- Press **[ENTER]**

The CNC resumed program execution.

Operation on current and future versions

To manually change the spindle speed range (gear), machine parameter "P601(1)" must set to "0".

When the new selected spindle speed "S" involves a gear change, the CNC displays a message indicating which range has to be selected.

The operator must proceed as follows:

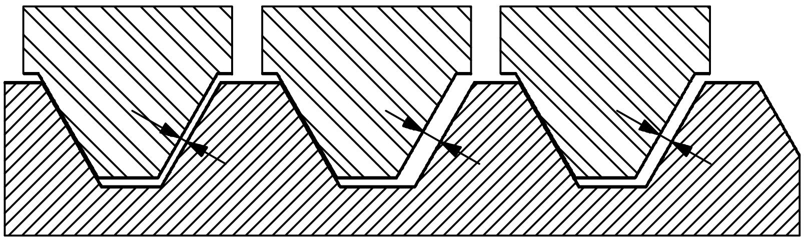
1st- Manually change gears
2nd- Press **[ENTER]**

The CNC restores spindle rotation and resumes program execution.

5. VARIABLE BACKLASH COMPENSATION

Until now, the 800T CNC allowed for a single leadscrew backlash compensation.

From now on, it is also possible to compensate for motion-reversal backlash depending on the particular backlash areas of the axes.



Requirements:

The leadscrew error compensation tables are now used for leadscrew error compensation and for this "Variable Backlash Compensation" (at the same time).

Parameter setting:

Machine parameters "P622(7)" and "P622(8)" indicate whether this feature is available or not.

P622(7) = 0 Not available for the Z axis.
P622(7) = 1 Available for the Z axis.
P622(8) = 0 Not available for the X axis.
P622(8) = 1 Available for the X axis.

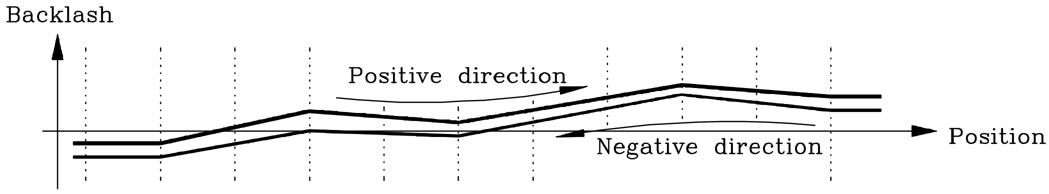
In order to use "Variable Backlash Compensation", regular leadscrew compensation must also be activated.

P605(2) = 0 X axis Leadscrew error compensation (0= No, 1= Yes)
P605(1) = 0 Z axis Leadscrew error compensation (0= No, 1= Yes)

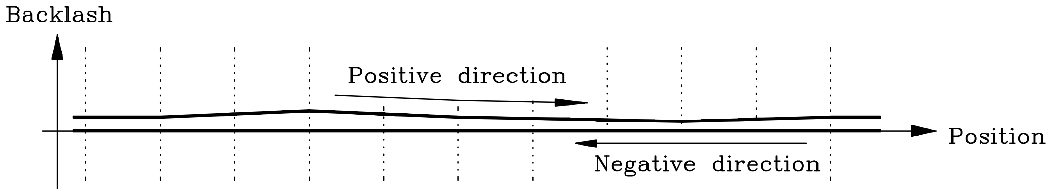
Operation:

The first 15 points of the table are for the positive direction and the other 15 for the negative direction.

When compensating for leadscrew error, the amount of backlash is the difference between both graphs.



When leadscrew error compensation is not to be used, all the values of one of the tables must be set to "0"; thus, the other graph will correspond to the leadscrew backlash.

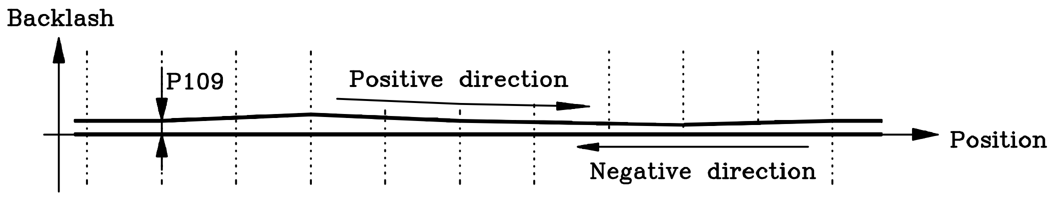


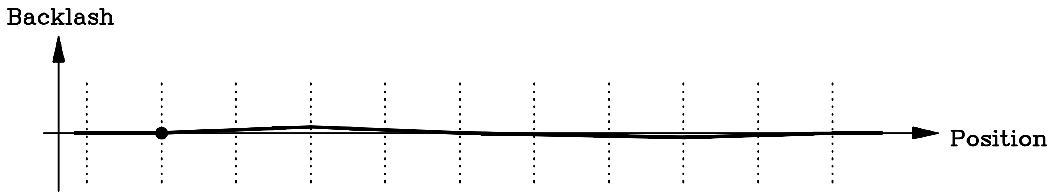
Notes: Both graphs must meet all the requirements of the leadscrew error compensation tables.

One of these requirements states that the Machine Reference Zero (home) must be assigned an error of "0".

If the leadscrew has some backlash at this Machine Reference Zero point, that amount of backlash must be allocated to machine parameter P109 or P309 (Backlash for the X axis or Z axis) and all the remaining points of the table must be offset by that amount.

Example:





Operation:

When using "Variable Backlash Compensation", the CNC operates with leadscrew error compensation and, therefore, it applies at all times the backlash compensation set in the table for that point and in the machining direction.

When the axis movement is reversed, the CNC swaps graphs restoring or applying the amount of backlash compensation and direction corresponding to that point.

Version 5.7 (July 1996)

1. WORK ZONE / EXCLUSION ZONE DETECTION

When using this feature, the CNC prevents the axes from exiting or entering this zone while jogging them with either the jog keys or the electronic handwheel.

The operator might suspect a malfunction since the CNC does not issue any message. From this version on, in these cases, the CNC will behave as follows:

- When the zone has been set as a Work Zone, the CNC will set PLCI input I46 high when trying to exit the selected zone.
- When the zone has been set as a Work Zone, the CNC will set PLCI input I46 high when trying to enter the selected zone.

2. RESUME EXECUTION AT MID-PROGRAM

If while a part, the program is interrupted (due to a power failure, etc.), it is now possible to resume execution from the interrupted program on. This way, there is no longer need to repeat the whole program, thus saving considerable amount of time.

To resume program execution, follow these steps:

1st Select the DRO mode, the one appearing on CNC power-up after the "General Test Passed".
In this mode, no cycle appears selected.

2nd Press **[RECALL]** to open the part-programs window.

3rd Select the part that was running. Use the up and down arrow keys to position over the desired part program and press **[RECALL]**.

4th Use the up and down arrow keys to select the operation being interrupted and press

The CNC will executed the selected operation and it will resume the part-program running it to the end.

Version 6.1 (January 1997)

1. NEW LANGUAGES (Taiwanese and Portuguese)

Machine parameter P99 P99 = 5 Portuguese P99 = 6 Taiwanese

2. MODIFICATIONS ON THE OPERATION WITH A MASTER HANDWHEEL

The operation with the master handwheel is now as follows:

a) When the machine is stopped.

Only the first handwheel is enabled, the second one (master) does not work.
Therefore, only the X axis can be jogged with a handwheel.

b) When the machine is running (CNC in Execution).

Only the Master handwheel is enabled, the first handwheel does not work.

The axes start moving when turning the Master Handwheel.

The feedrate of the axes depend on the turning speed of the Master Handwheel
When the handwheel stops, the machine also stops.

When the Master Handwheel is turned in the opposite direction, the CNC also reverses the moving direction (Retrace Function of a single block).

c) Semiautomatic Rounding Operation

The Semiautomatic Rounding operation starts when turning the Master Handwheel.

When stopping the Master Handwheel, the execution is interrupted.
When turning the Master handwheel again, execution is resumed. The turning direction of the handwheel cannot be changed.

When the operation is over, the CNC ignores the turning of the Master Handwheel for 1.4 seconds. Thus preventing another operation from being started.
After this time, when the Master Handwheel is turned, the CNC starts executing a new operation in the indicated direction.

d) Semiautomatic Taper Turning Operation

The Semiautomatic Taper Turning Operation starts when turning the Master Handwheel.

When stopping the Master Handwheel, the execution is interrupted.
When turning the Master handwheel again, execution is resumed.

When turning the Master Handwheel in the opposite direction, the operation is over. A new turn of the Master Handwheel in any direction implies the execution of a new operation in the indicated direction.

3. SOFTWARE VERSION OF THE CNC

From this version on, when accessing the EPROM checksum screen,

[Auxiliary Modes] [Special Modes] [8]

The CNC will show the checksum of each EPROM and the Software version of the CNC. For example: Version 6.1

Version 6.4 (May 1997)

1. TOOL CHANGE INDICATOR FOR THE PLC (I97)

On machines with a manual tool changer, when the CNC detects that the tool must be changed, it interrupts the execution and it displays a message for the operator to proceed with the tool change.

Certain precautions must be taken sometimes when changing tools. Those conditions must be handled by the PLC.

Therefore, from this version on, when the CNC displays the tool change warning message, it also activates the PLC input I97 and it cancels it when the message is removed.

Version 6.6 (November 1997)

1. HANDLING FEEDBACK SYSTEMS WITH CODED Io (semi-absolute)

Machine parameters

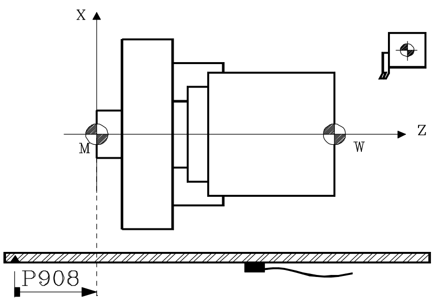
P608(5), P608(8) Type of Home marker signal of the feedback system. X, Z axes. (0 = normal "Io", 1 = coded "Io")
P608(3), P608(6) Period of the coded Io signal. X, Z signal. (0 = 20mm Period, 1 = 100 mm Period)
P608(4), P608(7) Increasing Io sequence with positive or negative count. X, Z axes
(0 = Increasing Io with positive count, 1 = increasing Io with negative count)

Regla	P608 (5)	Scale 8(3)	P608(4)
COS COC COX	1 1 1	0 0 0	1 0 0
COVS COVC COVX	1 1 1	0 0 0	1 0 0

Regla	P608 (5)	Scale 8(3)	P608(4)
MOVS MOVC MOVX	1 1 1	0 0 0	0 0 0
FOT FOS FOC	1 1 1	1 1 1	0 0 0

P908, P909 Scale offset or Home position (Machine Reference: M) with respect to the Scale "zero" point. X, Z axes.

Linear transducers (scales) with coded Io have a graduated scale with their own Scale "zero" point. Therefore, a 20 mm or 100 mm move is enough to know the axis position with respect to the Scale "zero" point.



Reference point.

When the feedback system has a coded Io, this point is only used when leadscrew error compensation is needed. The leadscrew error on this point must be "0".

Scale offset setting

The scale offset must be adjusted on one axis at a time. The following procedure is recommended:

* Indicate by parameters "P600(7) & P600(6)" the up or down flank of the marker pulse (Io) of the feedback system.
* Indicate by parameters "P618(8) & P618(7)" the home searching direction for the axes.
* Set parameters "P807 and P808" with the home searching feedrate for the axes.
* Set parameters " P908 and P909" to "0" (scale offset).

* Position the axis at the right position and execute the home search command for that axis..

  [X] or [Z], [up arrow] keys and

  When done with the home search, the CNC will display the axis position with respect to the scale's Zero point.

* After moving the axis to Machine Reference Zero point (home) or up to a known position with respect to home, Jot down the position reading of the CNC for that known position.

  The value to allocate to the machine parameter setting the scale's offset must be calculated with the following formula:

  Value = CNC reading at that point - Home coordinate of that point

  Example for the X axis: If the known position is physically located at 230 mm from Machine Reference Zero (home) and the CNC shows that its position is 423.5 mm, the scale offset will be:

  Machine parameter P908 = 423.5 - 230 = 193.5 mm.

* After setting the machine parameter with this value, press the [RESET] key so that value is assumed by the CNC.

* A new home search must be carried out for the proper values to be assumed for that axis.

2. THREADING WITH CONSTANT PENETRATION PASSES

From this version on, the penetration of each pass will depend on the sign assigned to parameter **D**

When **D** positive, the penetration of each pass depends on the corresponding pass (**D** n)
With **D** negative, the penetration passes remain constant with the absolute value of parameter **D**

3. GENERATING AN ISO-CODED PROGRAM

With this CNC, the ISO code (low level) for an operation or a part-program may be generated.

To use this feature, machine parameter "P623(2)" must be set to "1".

This ISO program always has the number: 99996 and can be stored either at the CNC or at a PC.

Program 99996 is a special user program in ISO code and can be:
Generated from an operation or a part-program.
Edited at the CNC itself via menu option: "Auxiliary Modes - Edit program 99996"
Loaded into the CNC after being generated at a PC.

Generating the ISO program (99996) at the CNC.

This CNC has 7 K of memory space to store program 99996. If the generated program is larger than that, the CNC will issue the relevant error message.

To generate program 99996, proceed as follows:

* If it is an operation, select or define the desired operation.

* If it is a part-program, select the desired one in the part-program directory and place the cursor on its header ("PART 01435". A listing of the operations it consists of must appear).

* Press the keystroke sequence: [AUX] [7]. The CNC will show the graphic simulation screen.

* Press . The CNC starts simulating the part and generating its ISO-coded program 99996.
* When done with the simulation, program 99996 stored in CNC memory will contain all simulated blocks in ISO code.

Generating the ISO program (99996) at a PC

Usually, the 99996 program generated from a part-program exceeds the available memory space of the CNC.

By using "DNC30", this program may be generated at a PC.

To do this, proceed as follows:

* Activate DNC communications and execute the DNC30 program at the PC.

* Select at the PC the menu option: "Program Management - Receive Digitizing".

* At the CNC, select the operation or place the cursor on the part-program header ("PART 01435"). A listing of the operations it consists of must appear).

* Press [AUX] [8]. The CNC will display the graphic simulation screen.

* Press . The CNC starts simulating the part and generating program 99996.

* When done with the simulation, the 99996 program generated at the PC will contain all the blocks simulated by the CNC in ISO code.

  This program can be executed at the CNC through the menu option: "Execute infinite program" of the DNC30.

4. MACHINE SAFETY REGULATION

This CNC offers the following features to comply with machine safety regulations.

Enabling of the CYCLE START key from the PLC

This feature is available when machine parameter "P619(7)=1"

PLC output O25 indicates whether the CYCLE START key is enabled (=1) or not (=0)

Axes movements affected by Feed-Hold. (It was already available)

Feed-Hold input, pin 15 of connector I/O 1, must be normally high.

If while moving the axes, the Feed-Hold input is brought low, the CNC keeps the spindle turning and stops the axes with 0V or velocity command (analog signal) and keeping their enables ON.

When this signal is brought back up, the CNC will resume the movement of the axes.

Axes jogging feedrate limited by PLC.

This feature is available when machine parameter "P619(7)=1"

When activating PLC output O26, the CNC assumes the feedrate set by machine parameter "P812"

Handwheel managed by the PLC.

Machine parameter "P623(3)" indicates whether the axes movements with handwheels are affected by Feed-Hold (=1) or not (=0)

Machine parameter "P622(1)" indicates whether the multiplying factor indicated by the MFO switch position is applied (=0) or the one indicated by the PLC outputs O44 and O45 (=1) (already available)

Spindle control from the PLC.

This feature is available when "P619(7)=1"

Output O27 =1 "tells" the CNC to apply the spindle analog voltage set by the PLC. The value of this analog signal is set at register R156 and sent to the CNC by mark M1956.

R156= 0000 1111 1111 1111 => + 10V.	R156= 0001 1111 1111 1111 => - 10V.
R156= 0000 0111 1111 1111 => + 5V.	R156= 0001 0111 1111 1111 => - 5V.
R156= 0000 0011 1111 1111 => + 2,5V.	R156= 0001 0011 1111 1111 => - 2,5V.
R156= 0000 0000 0000 0000 => + 0V.	R156= 0001 0000 0000 0000 => - 0V.

Also, PLC output O43, lets you control the rotation of the spindle. (Already available).
It must be normally low.
If it is brought up, the CNC stops the spindle.
When it is brought back up, the CNC restarts the spindle.

Information for the PLC on the status of the machine reference (home) search

I88 Home search in progress.
I100 X axis home search done.
I101 Z axis home search done.

R120 The Lower half of this register indicates the code pressed.
This value is maintained for 200 milliseconds unless another key is pressed before then.
This register may be canceled from the PLC after being processed.

R121 bit 1 Indicates that the Turning operation is selected (=1)
bit 2 Indicates that the Facing operation is selected (=1)
bit 3 Indicates that the Taper Turning operation is selected (=1)
bit 4 Indicates that the Rounding operation is selected (=1)
bit 5 Indicates that the Threading operation is selected (=1)
bit 6 Indicates that the Grooving operation is selected (=1)
bit 7 Indicates that the Profiling operation is selected (=1)
bit 8 Indicates that the Auxiliary Modes option is selected (=1)
bit 9 Indicates that the Tool Calibration option is selected (=1)
bit 10 Indicates that the Multiple Drilling operation is selected (=1)
bit 11 Indicates that the Simple Drilling / Tapping operation is selected (=1)
bit 12 Indicates that the Slot milling (keyway) operation is selected (=1)
bit 13 Indicates that the Tool Inspection mode is selected (=1)
bit 14 Indicates that the Graphic Simulation mode is selected (=1)
bit 16 Indicates that the mode for the following cycle parameters: "Finishing pass, finishing feedrate, finishing tool and safety distances on X and Z " is selected (=1)

Version 6.8 (March 1998)

1. NEW LANGUAGES (SWEDISH AND NORWEGIAN)

The languages that can be selected with machine parameter P99 are:

Spanish ...... (P99=0)	German .... (P99=1)	English ...... (P99=2)	French .... (P99=3)	Italian .... (P99=4)
Portuguese . (P99=5)	Taiwanese ... (P99=6)	Swedish ..... (P99=7)	Norwegian .. (P99=8)	

2. 1000 LINE ENCODER AS 1250 LINE AS ENCODER

This feature permits the CNC adapt a 1000 line encoder to be used as 1250 line encoder.

P623(7) Adapts the X axis feedback encoder (0=No, 1=Yes)
P623(8) Adapts the Z axis feedback encoder (0=No, 1=Yes)

A typical case: Having a 1000 line for a 5 mm pitch ballscrew .

The calculations necessary to set the axis resolution will be made with the selected pulses (1000 or 1250)

3. CROSS COMPENSATION

Cross compensation is used for compensating the measuring error suffered by the X axis when moving the Z axis.

P623(6) Cross compensation applied on to the X axis (0=No, 1=Yes)

When using cross compensation, no leadscrew compensation may be applied on the X axis (only on to the Z axis) since its corresponding table is being used for cross compensation with the following values:

P00 = X: ?????.??? P01 = DX: ????.???

in order to properly apply cross compensation, set: P605(2)=1 and P623(6)=1.

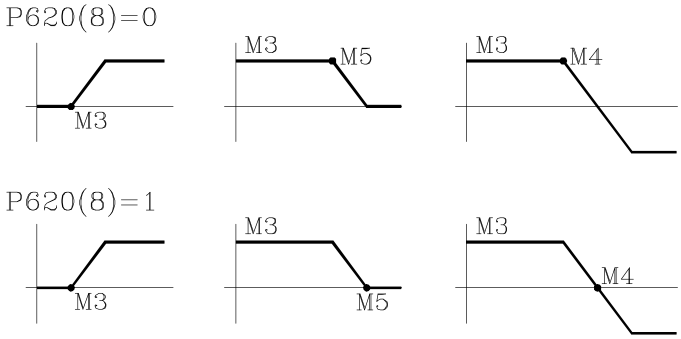
Note: The cross compensation table must meet the same requirements as those for the leadscrew error compensation. See section 3.8.4 of the installation manual.

4. PLCI. INPUT I104

When the Feedrate Override Switch on the operator panel is set on one of the handwheel positions (x1, x10, x100), input I104 is set to "1".

1. NEW MACHINE PARAMETER ASSOCIATED WITH M FUNCTIONS

Machine parameter "P620(8)" indicates when the M functions M3, M4 and M5 are to be output during the acceleration and deceleration of the spindle.



2. CANCEL TOOL OFFSET DURING TOOL CHANGE

From this version on, it is possible to execute, within a subroutine associated with the tool, a "T.0" type block to cancel the tool offset. This allows to move to a specific position without having to do cumbersome calculations.

It is only possible to cancel (T.0) or change (T.xx) the tool offset. The tool cannot be changed (Txx.xx) within the subroutine associated with the tool.

3. X1 FACTOR FOR FEEDBACK PULSES

Machine parameters P620(5) and P620(6) are used together with P602(6) and P602(5) which indicate the multiplying factor of the feedback pulses for the X and Z axes respectively.

They indicate whether x1 factor is applied to the feedback pulses (=1) or not (=0).

P620(5)=0 and P620(6)=0	the x1 factor is NOT applied
P620(5)=1 and P620(6)=1	the x1 factor is applied

Example: We would like to obtain a 0.01 mm resolution with a square signal encoder mounted on the X axis having a leadscrew pitch of 5mm/turn.

Number of Encoder pulses = leadscrew pitch / (multiplying factor x Resolution)

Con P602(6)=0 and P620(5)=0	x4 multiplying factor	Number of pulses = 125
Con P602(6)=1 and P620(5)=0	x2 multiplying factor	Number of pulses = 250
Con P602(6)=0 and P620(5)=1	x2 multiplying factor	Number of pulses = 250
Con P602(6)=1 and P620(5)=1	x1 multiplying factor	Number of pulses = 500