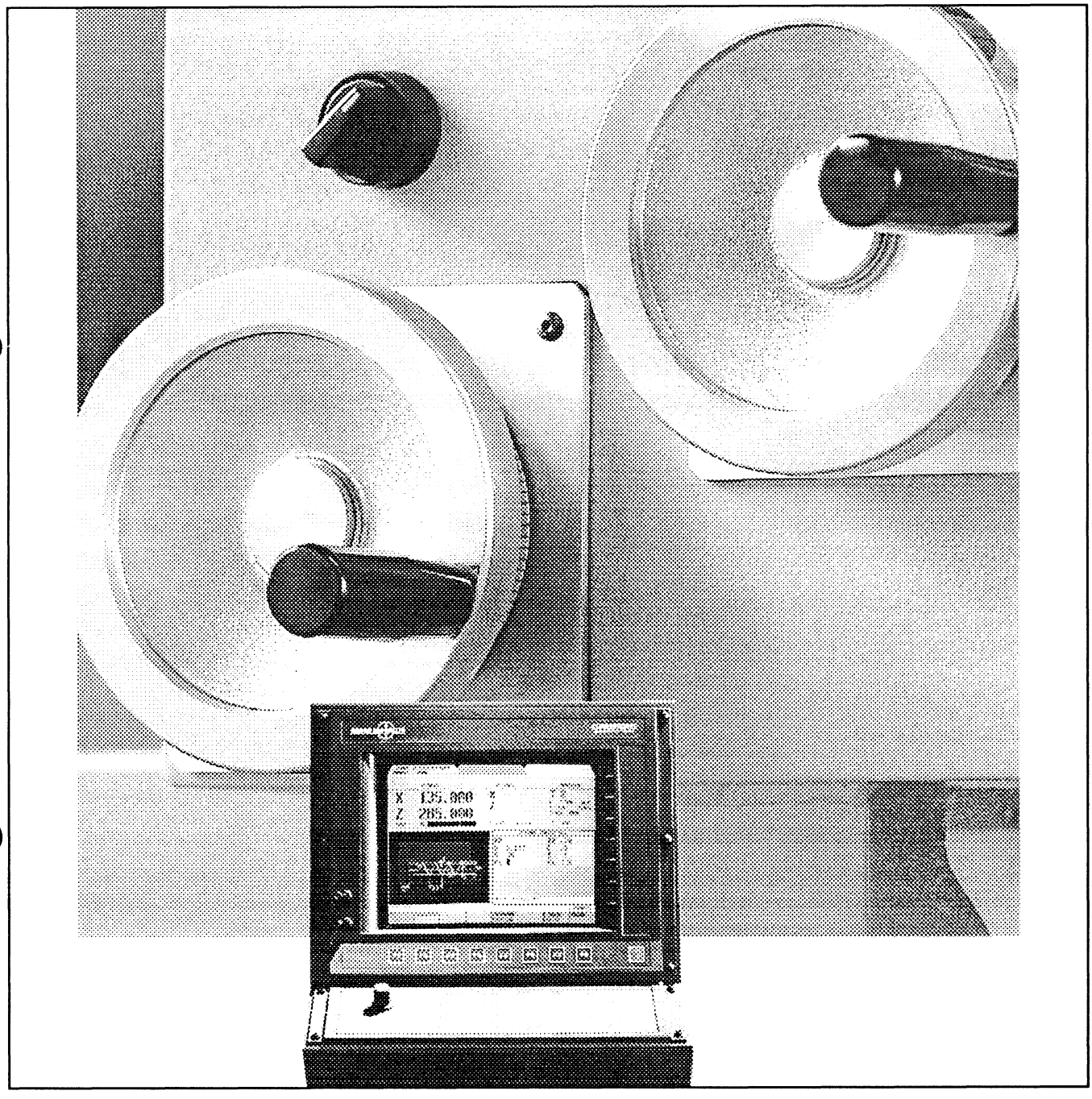
User Manual V260/V350

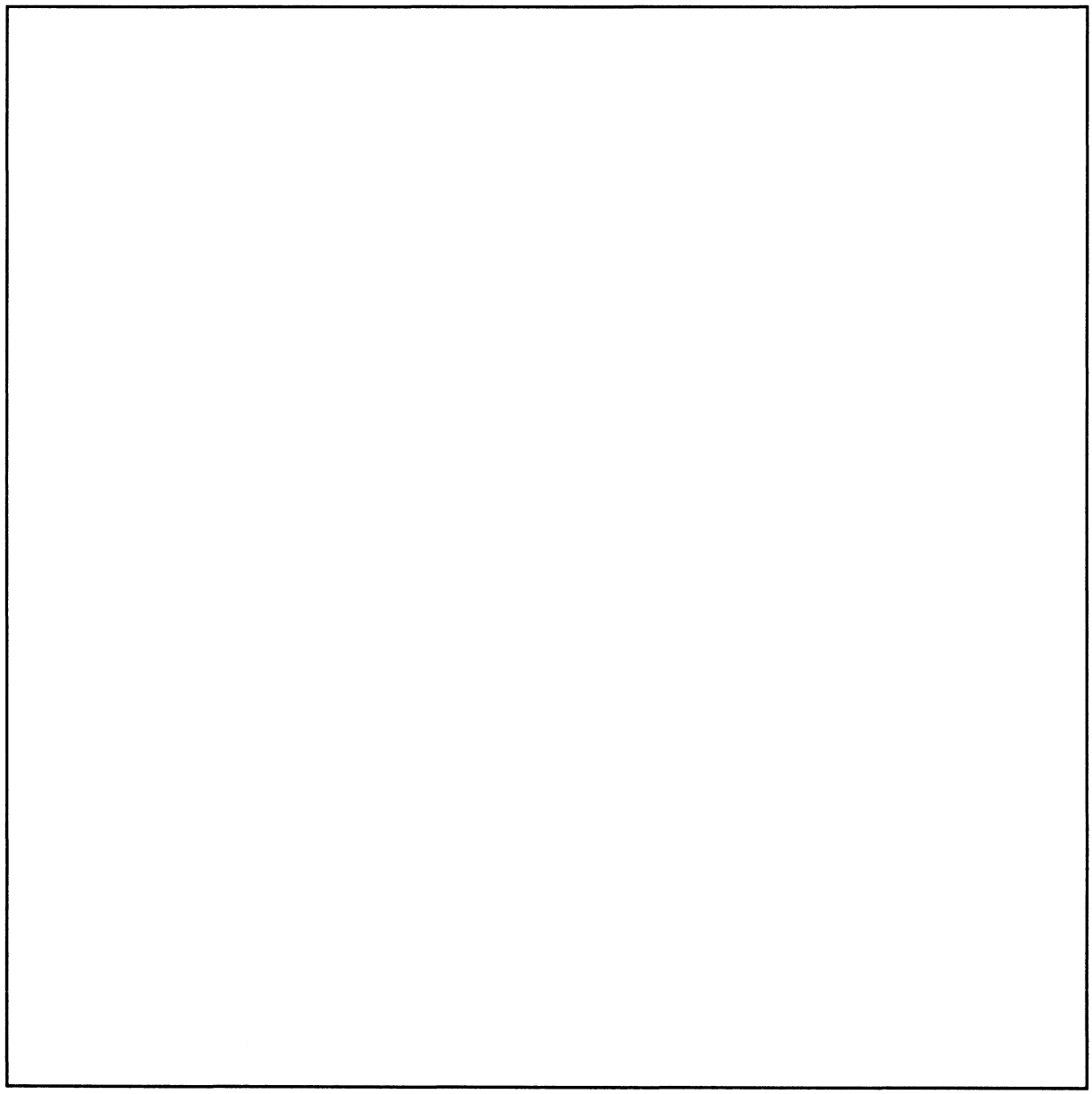


MANUAL PLUS

GRUNDIG numeric

USER MANUAL

MANUAL PLUS User Manual Turning V260/V350



GRUNDIG
numeric

First edition	Software Version V260	940506
Second edition	Software Version V261	940520
Third edition	Software Version V262	940621
Fourth edition	Software Version V263/V350	950301
Fifth edition	Software Version V264/V350	950601

9418 859 06181

DER6-32.8-1414_C01

1. INTRODUCTION

The MANUAL PLUS has been designed for a new generation of manually operated machine tools.

Manually operated machine tools have not changed since the development of the gear box linking the spindle with the axes feed drives. The MANUAL PLUS breaks away from the limitations of gear driven axes by using modern control technology and closed loop servo systems to synchronise and position spindle and axes movements. By breaking this fixed mechanical link between spindle and axes the MANUAL PLUS extends the range of machining operations possible on a manual machine tool.

In its basic operating mode the MANUAL PLUS performs and is operated exactly the same as a traditional high performance manual machine tool. Using manually operated handwheels to move axes and to machine parts. The Teach-in and Program execution modes extend the operation of the manual machine tool into a semi-automatic machine. Capable of undertaking the machining of complex machining operations and profiles, automatically, from a list of machine cycles stored in its memory.

Using the MANUAL PLUS on a manually operated machine has several other advantages for the operator one of which is safety. Operators can work safely behind full machine guards, controlling the machining operation remotely.

1.1 The MANUAL PLUS

The MANUAL PLUS has two control areas: the Control Station and the Machine Control Panel. Figure 1.1 below shows the MANUAL PLUS control. The controls located on the panels and their operation are described in Chapter 2 MANUAL PLUS CONTROLS.

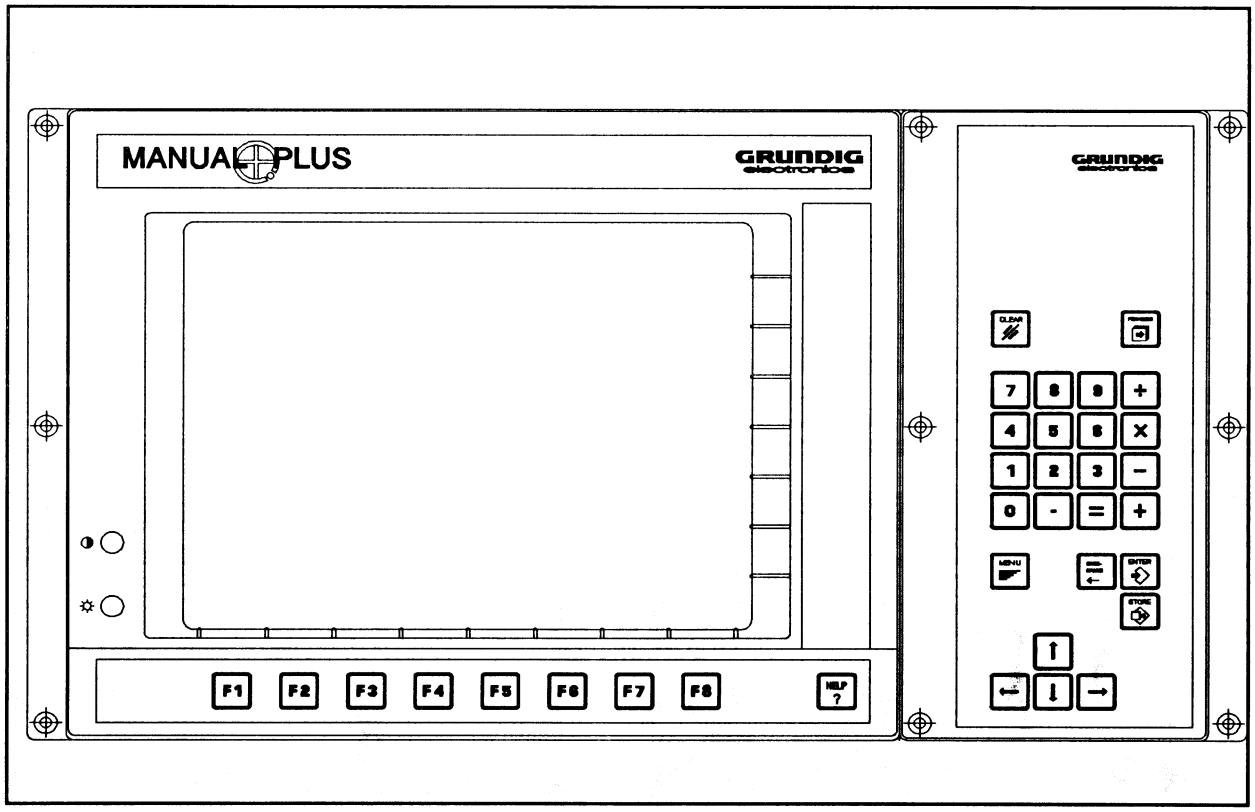


Figure 1.1: The MANUAL PLUS Control Station

1.2 MANUAL PLUS OVERVIEW

Figure 1.2 below gives a summary of operating and machining capabilities of the MANUAL PLUS.

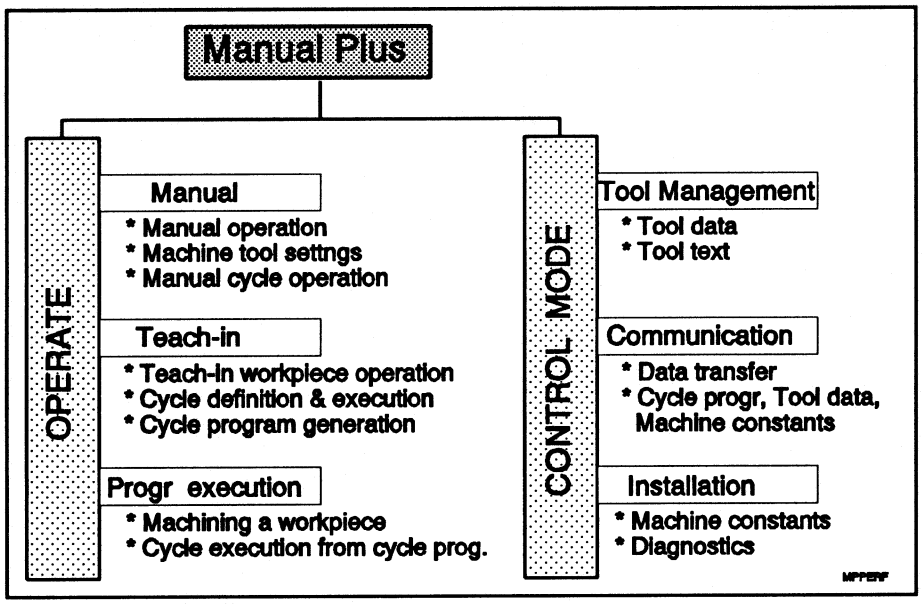


Figure 1.2: Capabilities

OPERATE

Manual Operation	Conventional operation, allows manual axes movements, entry of machine tool settings and entry and execution of single cycles in manual cycle operations.
Teach-in	Preparation mode for semi-automatic machining. Allows entry and execution of cycles stored in a cycle program for execution in the Program Execution mode.
Program Execution	Automatic machining, executes the cycles stored in a cycle program.

CONTROL

Tool Management	Entry and storage of information relating to the tools used during machining processes.
Communication	Transfer of cycle programs, tool data and machine constants to and from the MANUAL PLUS.
Installation	Allows access to the machine constants and diagnostic capabilities.

1.3 This Manual

This manual describes the operation of the MANUAL PLUS machine tool control system. It is arranged to help you become familiar with the look, location of controls and operation of the MANUAL PLUS as quickly as possible. Extensive use is made of pictures and flow diagrams to simplify the operation and understanding of the MANUAL PLUS.

1.3.1 Chapters within the manual

Chapter 1 INTRODUCTION
This gives a brief overview of the MANUAL PLUS, where it fits into today's machine shops and its capabilities.

Chapter 2 MANUAL PLUS CONTROLS
This chapter identifies the individual control panels. All controls and switches are located and identified. The screen displays, menus and how to enter and select menu choices are introduced here.

Chapter 3 BASIC OPERATING PRINCIPLES
This chapter is intended for anyone new to machine tools which use coordinates systems. It describes how a coordinate positioning system operates and some of the terms used. The various machine and work piece reference points used by the MANUAL PLUS are identified. Also the concept of measuring tool lengths and the effect of tool nose radius are introduced.

Chapter 4 GETTING STARTED
This chapter deals with turning on the MANUAL PLUS and the start up screens expected. Also covered is the X and Z reference procedure. This must be performed before the MANUAL PLUS can be operated.

Chapter 5 MACHINE TOOL SETTING
This chapter covers the machine tool settings. These are standard machine tool settings, spindle speeds and feed rates. The MANUAL PLUS introduces new possibilities, such as constant cutting speed in m/min, which is not possible on manually operated machine tools. The concept of using tool numbers is also introduced in this chapter.

Chapter 6 TOOL MEASUREMENT
This chapter describes the reasons for measuring tools, a method to measure tools and what happens with the information. The effect of the tool orientation when measuring a tool is given. Also the Tool table and Tool type tables are displayed and their contents described.

Chapter 7 MANUAL OPERATION
Describes using the MANUAL PLUS as a manually operated machine tool. Understanding how to use the handwheels and jog controls to operate the axes and the requirements to program single cycles to machine a work piece is described. The operation of the semi-automatic functions, cycle start and cycle stop are introduced.

Chapter 8 TEACH_IN
This chapter covers the steps and actions necessary to make a cycle program. It describes what a cycle program is, suggests a work method and goes through all the different steps required to produce a cycle program.

Chapter 9 CYCLE PROGRAM EXECUTION
Once a cycle program has been made it must be executed. Chapter 9 describes how to select a cycle program, the set-up stages before execution and executing a cycle program.

Chapter 10 COMMUNICATION
The MANUAL PLUS has powerful communication capabilities built in. This chapter describes how to use these capabilities to communicate with other devices.

Chapter 11 EXAMPLE
This chapter contains a complete description how to program an example work piece. Each step is given along with the necessary data which must be entered. Working through the example shows you how cycle programs are built up and how cycles are programmed.

Appendix
The Appendices covers subjects in detail and which may not be necessary for every day use of the MANUAL PLUS. Such as error messages and Operator Machine Constants. A complete overview of all the cycles and their parameters is also contained in Appendix Available Cycles. Routine diagnostics are described in appendix Diagnostics.

2. MANUAL PLUS CONTROLS

2.1 Introduction

This chapter introduces you to the controls and screen displays found on the MANUAL PLUS. Each of the MANUAL PLUS parts are identified along with the controls. Read through this Chapter to familiarize yourself with the location of the controls and the display formats. The operation of each control is introduced and explained fully in Chapters and sections relating to its operation.

2.2 The CONTROL Station

The Control Station consists of a display unit with eight function keys, one help key and the entry panel. The display shows machine status information and function key possibilities. Machine control data such as; spindle speed, feed rate, tool numbers and axes positions, are entered via the entry panel (Figure 2.2: The Data Entry Panel).

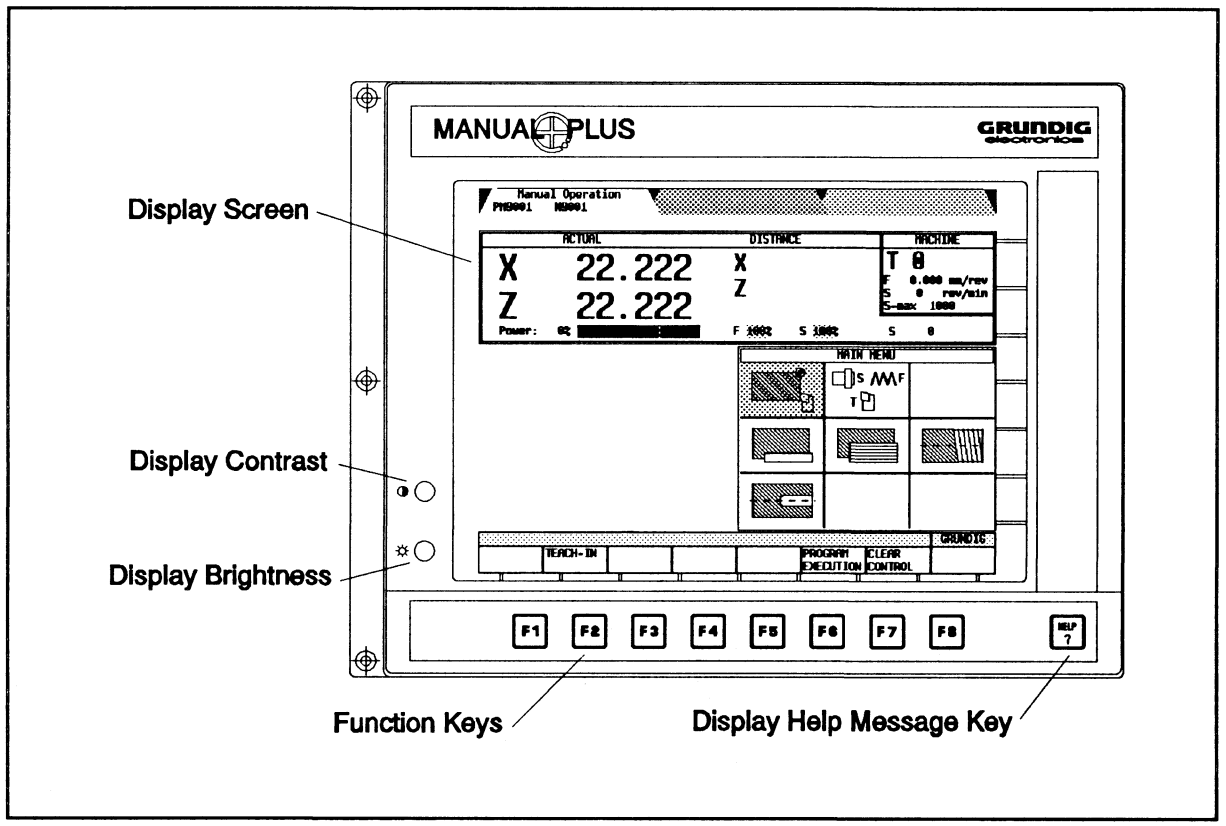


Figure 2.1: The Display Unit

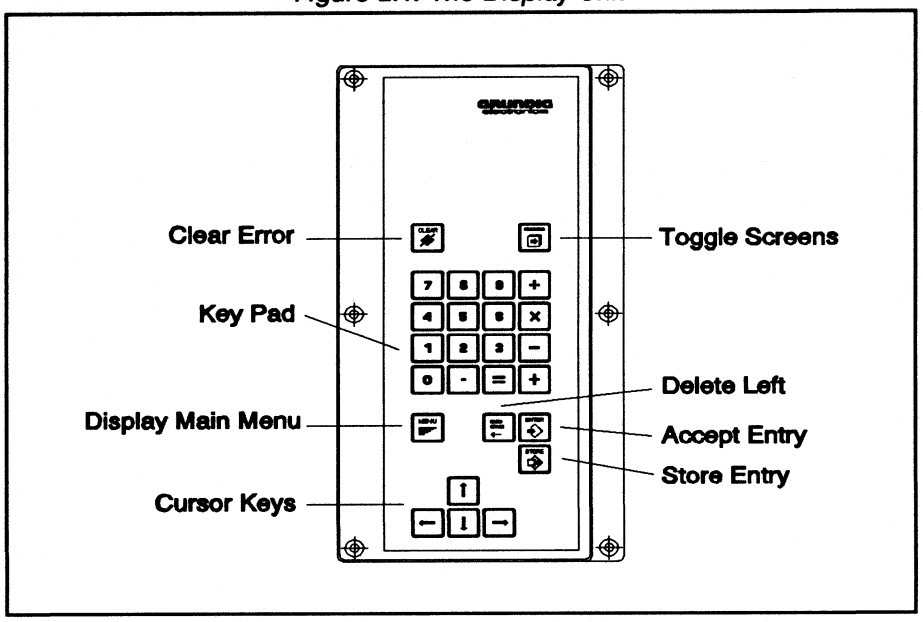


Figure 2.2: The Data Entry Panel

2.2.1 The Machine Control Panel

The machine control panel is designed and supplied by the machine tool builder. Therefore the position and styles of the controls on this panel may differ from machine to machine. The minimum functions required on the machine tool control panel are given below. Some manufactures may place additional options and functions on the machine control panel. Read the machine tool builder's documentation in conjunction with this information.

Emergency stop	Standard push down/twist to release emergency stop push button.
X Handwheel	Rotary handwheel for positioning the X axis.
Z Handwheel	Rotary handwheel for positioning the Z axis.
Handwheel Resolution	Sets the handwheel resolution to either 1 increment per handwheel division, 10 increments per handwheel division, or a 100 increments per handwheel division.
Axes Jog Control	Axis jog actuator, moves the axis at feed or rapid rate depending upon the rapid/feed selector.
Rapid/Feed selector	Selects the axes jog speed.
Spindle Start CW	Starts spindle rotation in the clockwise direction.
Spindle Start CCW	Starts spindle rotation in the counter clockwise direction.
Spindle STOP	Stops spindle rotation and any programmed feed.
Feed Rate Override	(Optional position). Adjusts feed rate between a minimum and maximum value. Located on either the Data Entry panel or the Machine Tool control panel.
Spindle Speed Override	(Optional position). Adjusts spindle speed between a minimum and maximum value. Located on either the Data Entry panel or Machine Tool control panel.

Cycle Start	Starts machine instructions, a single cycle or a cycle program in program execution mode.
Cycle Stop	Stops the execution of a cycle. Spindle rotation continues. Pressing CYCLE START completes the cycle. Cycle Stop must be pressed before a cycle can be cancelled.
Coolant ON/OFF	Turns the coolant supply on and off.
Tool Change Request/Compete	An illuminated push button. Lights when a tool change is requested. The push button confirms that the requested tool change is complete.

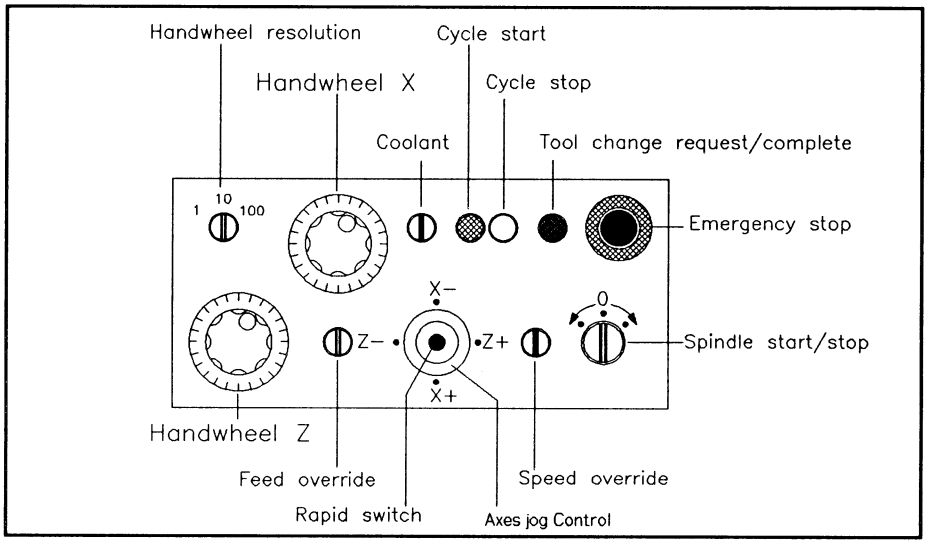


Figure 2.3: Example Machine Control Panel

2.3 The OPERATE Screen Display

The MANUAL PLUS use the latest display techniques to present information to the operator. Information is grouped and separated into two sperate screens. Switch between the screens by pressing the PROCESS push button. This section introduces the screens and identifies the different areas on the display.

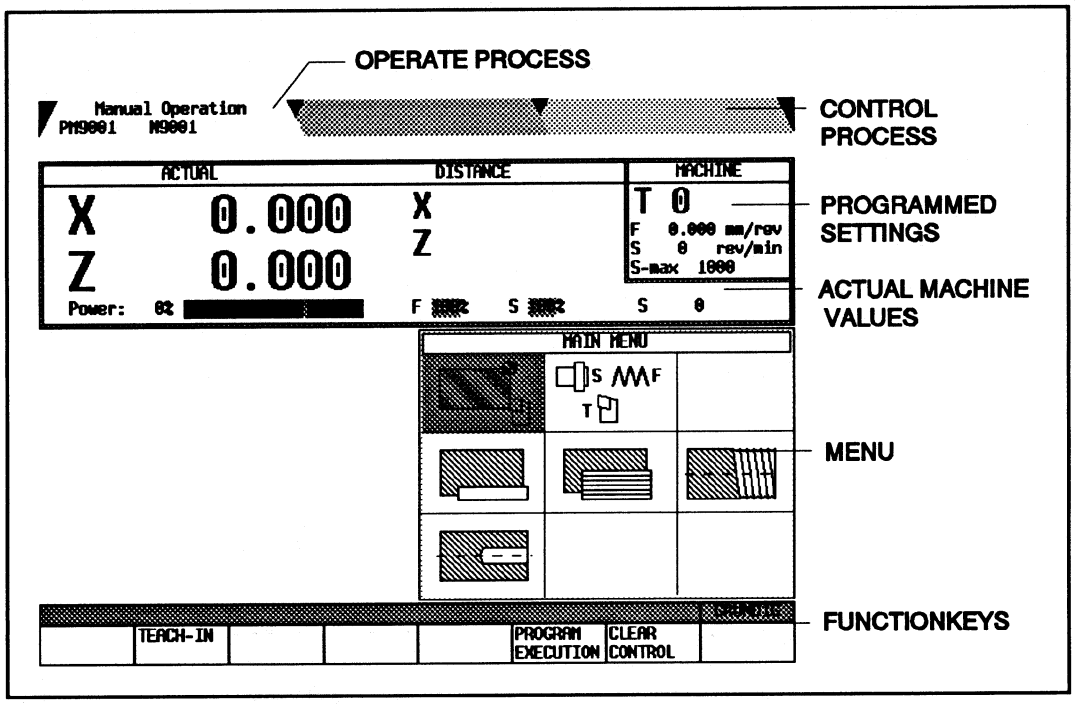


Figure 2.4: Manual Operation Screen

2.4 The CONTROL screen display

This screen contains items about the MANUAL PLUS set up and operation. For example, tool management and communication parameters.

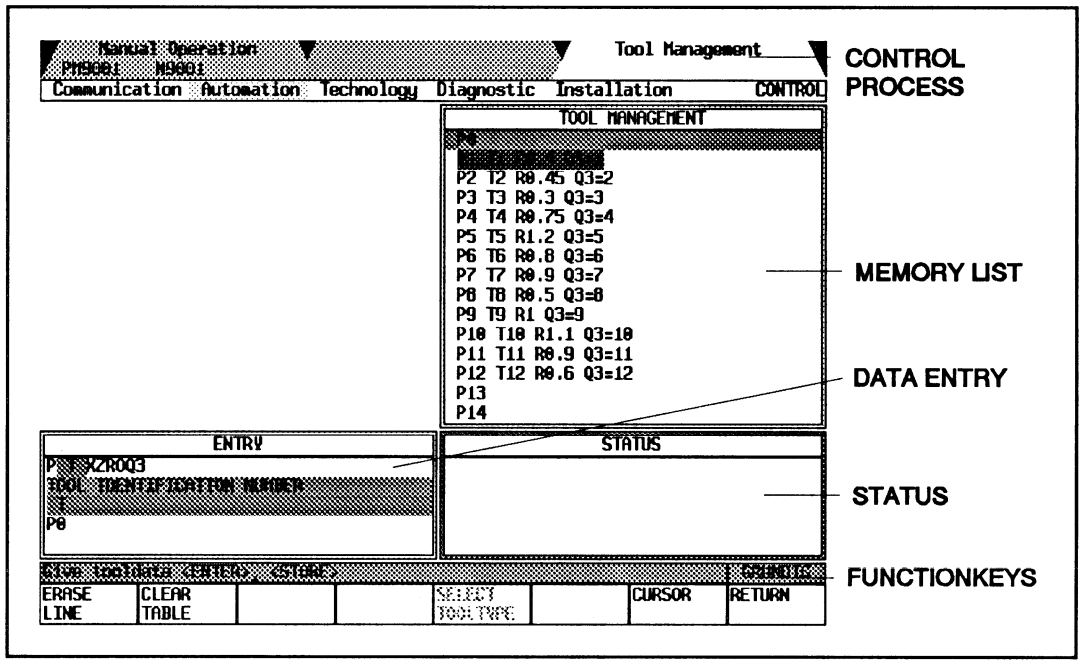


Figure 2.5: Diagram CONTROL Display Screen

Note:

It is only possible to switch to the CONTROL screen when the main menu of the OPERATE screen is displayed (that is from Manual Operation).

2.5 Selecting Menus

2.5.1 The OPERATE screen

The OPERATE screen uses graphical menus to command machine functions and operations. The MANUAL menu is displayed whenever you press the MENU push button.

To select a function from the graphical OPERATE menus, use the cursor keys to position the highlight in one of the graphical symbols, then press ENTER. Alternatively, use the numeric key pad to select and enter a menu number. This is the short cut method. The position of the numbers on the key pad relate to the position of the graphic symbol on the menu grid. See Figure 2.7 below.

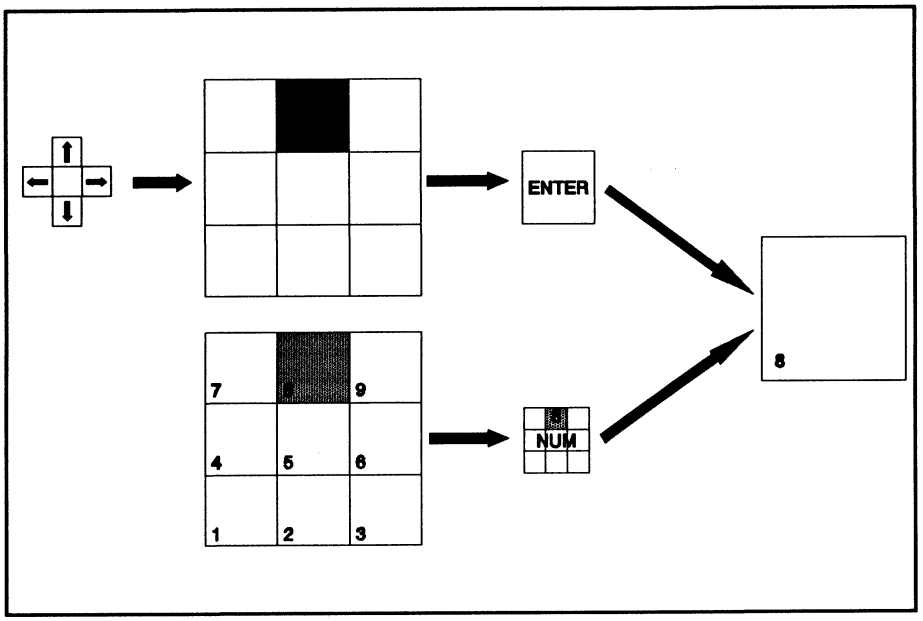


Figure 2.7: Operate Menu Selection

2.5.2 The CONTROL screen

The CONTROL screen uses drop down menus to select the different modes. To select a menu in the CONTROL screen, press the MENU push button. Use the LEFT and RIGHT cursor keys to display other drop down menus. Use the UP and DOWN cursor keys to highlight possibilities within the drop down menus. Press ENTER to select and enter the chosen function.

Alternatively, use the numeric short cut keys. Each menu possibility is assigned a unique two digit number which is displayed, as the functions are highlighted, at the right hand end of the menu bar. The first digit is the Drop Down menu number, the second digit is the menu function number. For example, 11 is the first function in the first Drop Down menu.

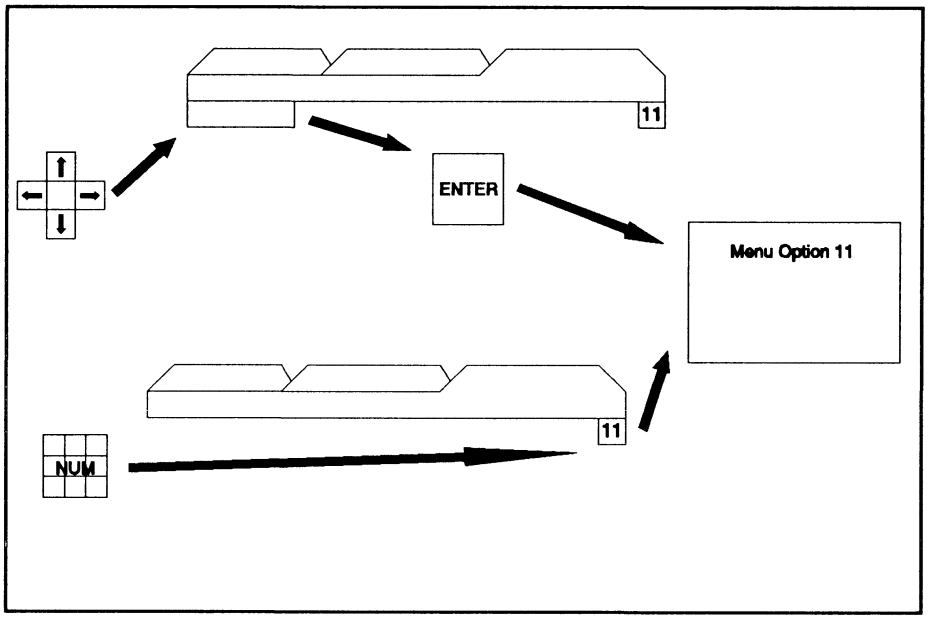


Figure 2.8: Control Menu selection

2.6 Function Keys

Function keys are the eight numbered F keys along the bottom of the screen on the display unit. These function keys align with eight boxes displayed along the bottom part of the screen. The texts in these boxes change depending upon the current operating mode, giving only those possibilities which are available for any given operating mode.

Some of the Function keys are toggle actions; pressing once activates the function, pressing again de-activates the function. Functions are active when highlighted.

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

Figure 2.6: Function Keys and Text Boxes

2.7 The HELP Key

The HELP key is located with the function keys along the bottom of the display unit. Pressing this key displays a new line with function keys on the bottom of the operate screen:

F1 TOOLLIST — This function key displays a box containing the list of available tools.

F5 INCH = > METRIC — Recalculates the contents of the data entry item where the cursor is positioned from inch to metric (i.e. multiply it with 25.4). This function can be used to program an inch workpiece on a metric machine.

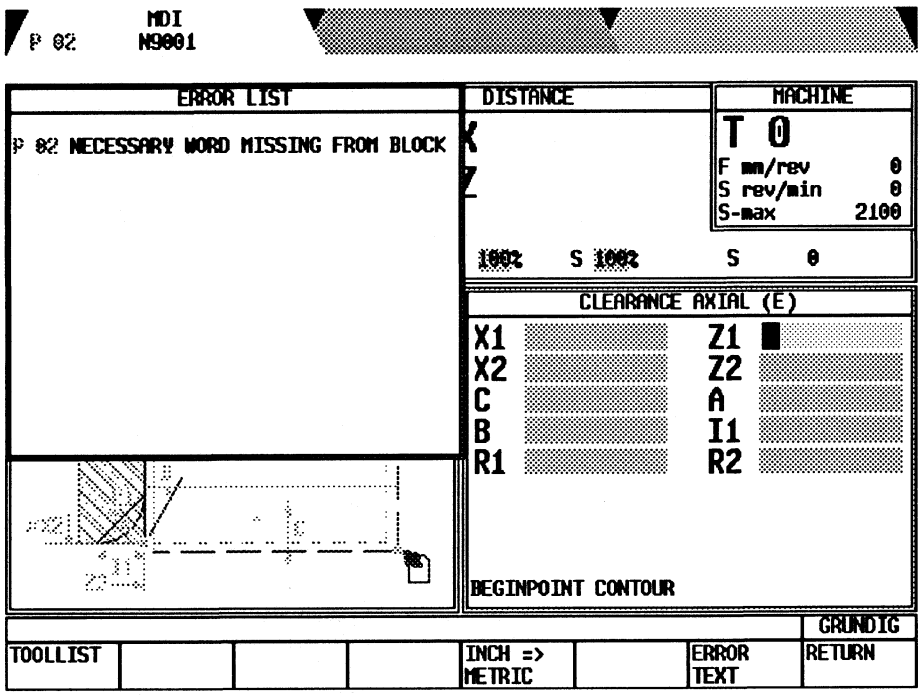


Figure 2.7: Operate screen with "HELP" Function key line and a example of an error-text

2.8 Data entry and Graphics Support Windows

Associated with every machining cycle in the MANUAL PLUS is a data entry and graphic support window. These windows display and request information necessary to complete the chosen cycle. The general lay out for these windows is given below.

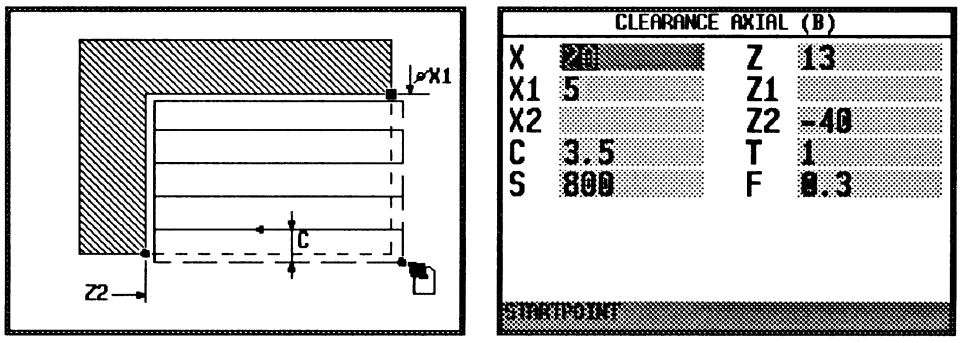
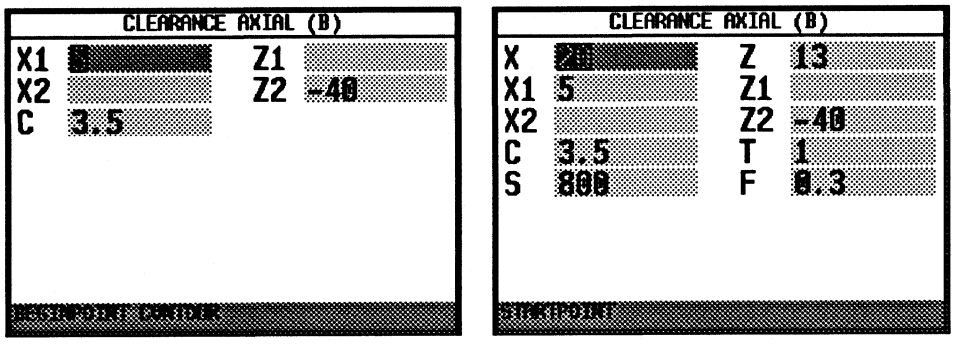


Figure 2.9: Graphic and Dialogue Windows

2.8.1 Data Entry Window

The data entry window displays items required to complete the requested machining operation. Geometric items are displayed on a graphic picture.
In the Manual operation mode the position of the tool (X and Z) and the technology items F, S and T are not displayed. The cycle is in that case executed with the actual values of these items.
In the Teach-In operation mode these items are displayed and can be programmed seperatly for each cycle.

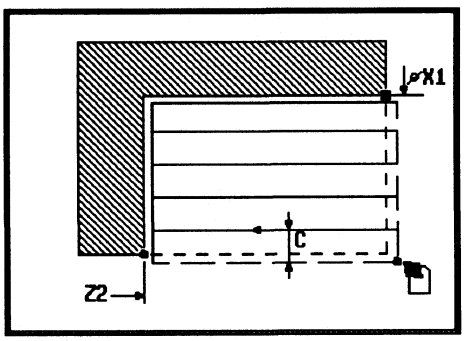


2.8.2 The Graphics Support Window

The graphic window displays a graphic representation of the chosen cycle. All items requiring an input value are positioned on the graphic.

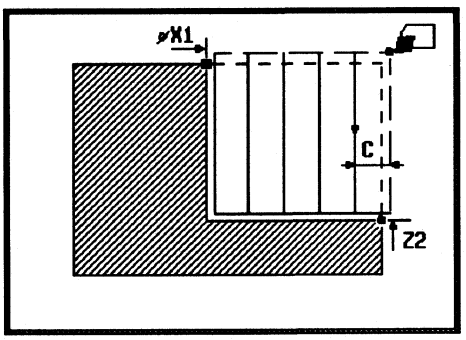
2.8.2.1 Standard Graphic Support

The graphics is displayed with a horizontal Z-axis and a vertical X-axis.



2.8.2.2 Graphic Support for vertical lathes (option)

An option of the Manual Plus is the Vertical Graphic Support.
With this feature the static support pictures and the Graphic Cycle Check are displayed for vertical machines.
The graphics is displayed with a horizontal X-axis and a vertical Z-axis.



2.8.3 Data Entry

Data is entered through the Data Entry window. Use the cursor UP and DOWN keys to highlight the parameter boxes in the Data Entry window. As each parameter box is highlighted a short description of the parameter is given along the bottom of the window. Figure 2.10 below shows the data entry sequence to program a cycle.

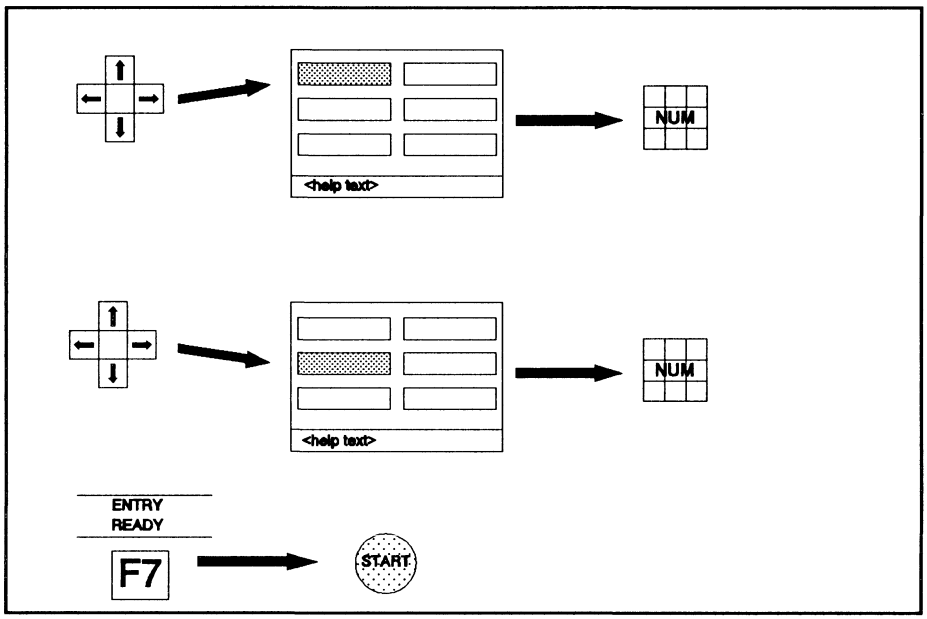


Figure 2.10 Data Entry Sequence

The following push buttons and actions aid data entry:

- LEFT and RIGHT cursor keys select a character in a parameter box. The character can then be changed.
- The BACKSPACE push button deletes the character to the left of the cursor.
- Returning to a parameter box and pressing a character key, clears the parameter box and displays only the new character.
- Pressing ENTER has the same function as cursor down, moving to the next item in sequence in the list.

Note:

In some cycles it may not be possible to select an item with the cursor. In this case the item is not necessary for that particular cycle. These items are only used for the symmetry (X1,Z1 or X2,Z2 on one row) in the data entry window. Some items maybe optional. This is indicated in the explanation text in the window.

2.8.4 Inch Initialisation

The MANUAL PLUS can be initialised with an INCH dimensional unit system. In an INCH dimensional unit system the smallest units for display and programming are:

- Linear dimensions: 0.0001 inch
- Feed rate: 0.0001 inch/rev
- Pitch of thread: 0.0001 inch/rev
- Constant cutting speed: 1 feet/min

Retract distances in cycles of 0.1 mm correspond with a distance of 0.01 inch.

The initialisation mode should not be changed during normal use. The result would be that information in user memories (program, tool memory) is divided or multiplied by a factor 10.

Note:

In a MANUAL PLUS with an INCH dimensional unit system, the values in the machine constants are still displayed as metric values. They are however internally in the CNC recalculated and used as inch values.

3. BASIC OPERATING PRINCIPLES

3.1 Introduction

This chapter is intended for the operator new to machine tools which use coordinate systems to position axes. It describes the fundamentals and operations required to use the MANUAL PLUS successfully. Subjects covered are naming and orientation of axes. How to position axes in a coordinate system, the machine zero point, machine reference point and work piece zero point. It also describes why we must measure and know the lengths of tools and the method of positioning the tool.

If you are familiar with CNC machine tools or coordinate programmed axes systems you may wish to skip this chapter.

3.2 Axes Identification and Orientation

The cross slide and saddle on a manual lathe are called AXES in automated machine tools.

- The cross slide is assigned the letter X and when represented on diagrams and graphic displays is the vertical (up and down axis). In the MANUAL PLUS all displayed and entered X positions are given as diameters.

- The saddle is assigned the letter Z. This is represented as the horizontal (left to right) axis in diagrams and graphical displays.

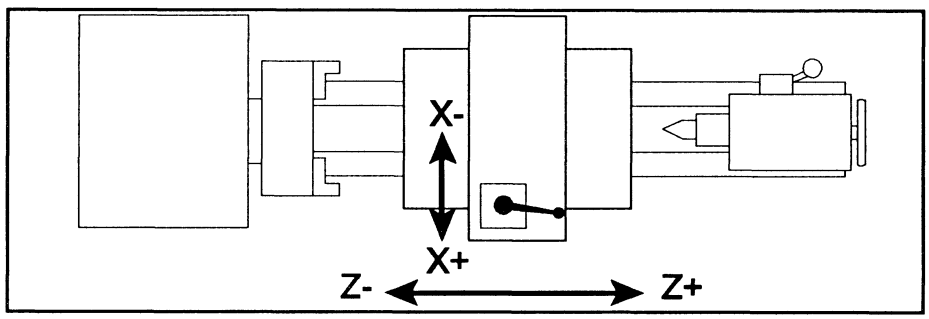


Figure 3.1: Lathe Cross Slide/Saddle Axes

When moving the axes:

- plus movements are classed as a movement away from the work piece.

- minus movements are classed as a movement into the work piece.

How the axes are positioned is covered in the following section Coordinate Systems

3.3 Coordinate System

As mentioned in the previous section, Axes Identification and Direction, the cross slide and saddle are assigned the letters X and Z respectively. These letters are used to represent the two axes on a two dimensional coordinate system. A coordinate system can be pictured as a large sheet of graph paper laid on the bed of the machine tool.

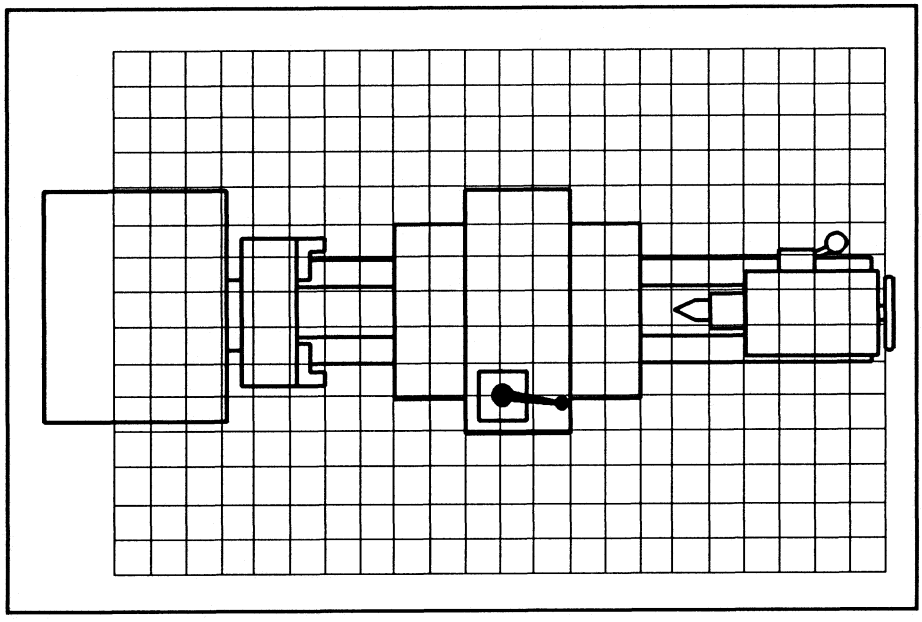


Figure 3.2: Machine Coordinate System

The tool tip can be positioned at each cross point within this sheet of graph paper by defining a position along the X axis and a position along the Z axis. This gives a unique point in the squared area. See Figure 3.3 below.

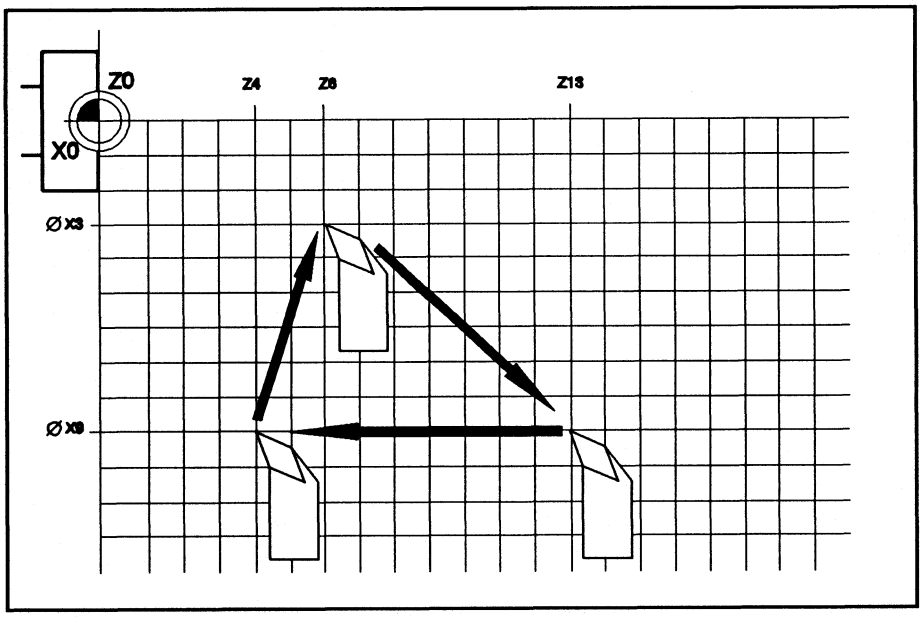


Figure 3.3: Defining a Coordinate Point

In this way a complete profile of the part can be defined by programming the tool tip position for all the start and end coordinates of the individual shapes and features on the part.

The control will assume straight lines between start and end coordinates. Unless special menu functions are selected which specify circular movements between coordinates. As can be imagined the smaller the squares on the sheet of graph paper the more accurate the part can be defined and machined. This is called the resolution of the system. The resolution (the size of the squares) in the MANUAL PLUS is 1 micron (0.001mm). This gives a very high degree of accuracy and the ability to define very small details.

The MANUAL PLUS uses absolute coordinates to define axes positions. This means all axes movements are measured from known points. These are described in the following sections.

3.4 Machine Zero Point

For a coordinate system to work there must be some known point where the X and Z axes meet and both X and Z are equal to 0. This point is known as the Machine Zero Point. On a lathe this is normally taken as the centre line of the spindle for the X axis (cross slide) and a known reference surface (such as the spindle face) for Z axis. Where these two axes cross is the Machine Zero Point (M).

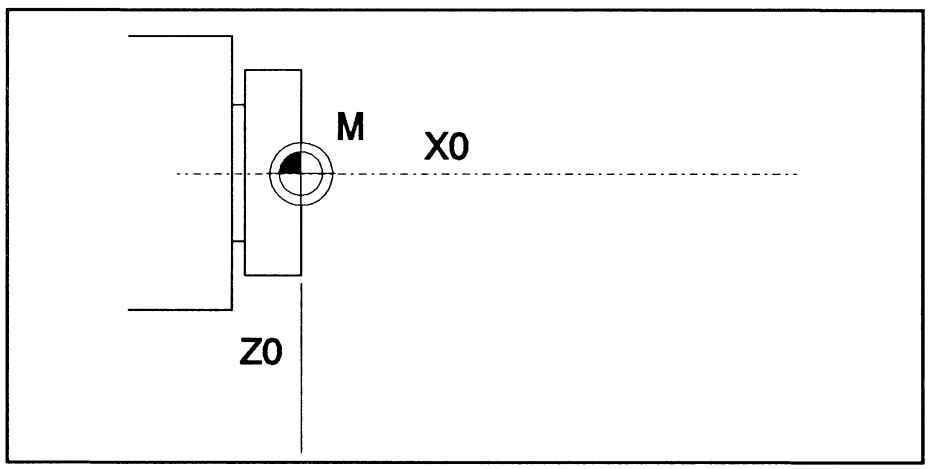


Figure 3.4: Machine Zero Point (M)

All positions for X and Z are related to this machine zero point. Although, it is easier when working to set a new zero point positioned on the work piece. This is the work piece zero point. See the section Work Piece Zero point for further details.

3.5 Machine Reference Point

The MACHINE REFERENCE POINT is a point which is used by the measuring system as a known reference. Its distance from the machine zero point is known after an X and Z reference procedure. See Chapter 4 The Reference Procedure. The axes distances displayed after an X and Z reference is the distance of a known point (usually one corner of the tool post, called the tool post zero point) from the machine zero point. See Figure 3.5 below.

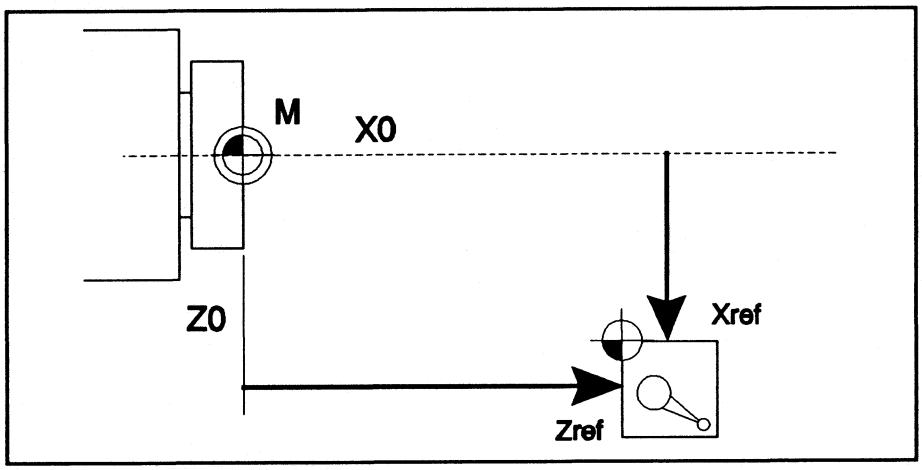


Figure 3.5: Location of Machine Reference Point (R)

3.6 Work Piece Zero Point

When defining the coordinates for a work piece it is normally easier to give X and Z dimensions from a zero point positioned on the work piece, rather than from the machine zero point.

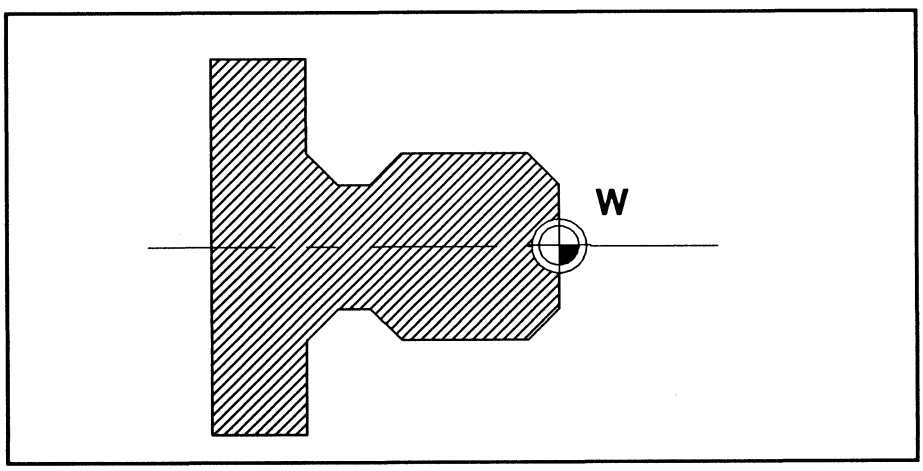


Figure 3.6: Work Piece Zero Point (W)

The machine zero point remains the absolute zero position, but X and Z dimensions are displayed and entered relative to the work piece zero point.

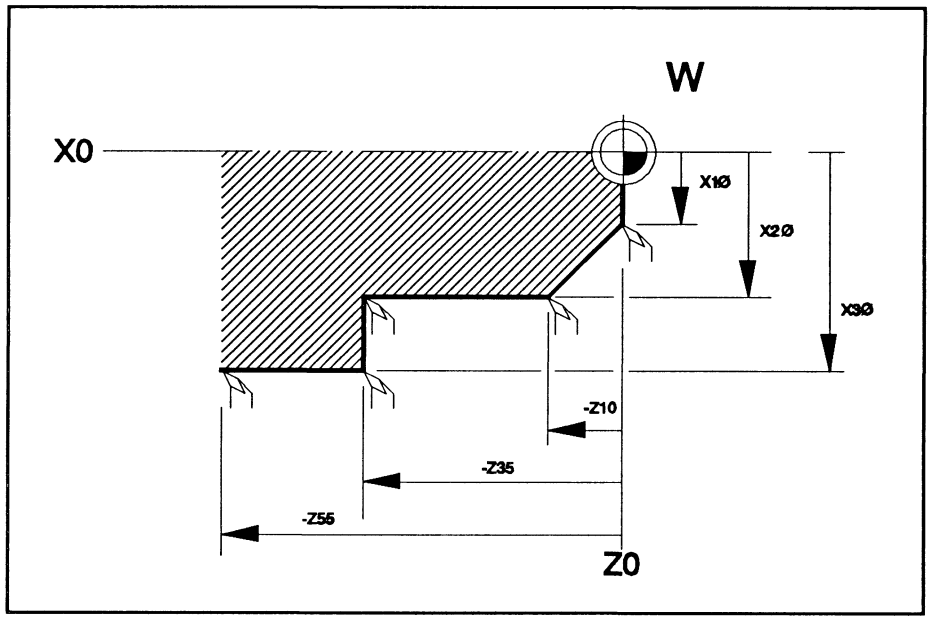


Figure 3.7: Part Defined as Coordinates

3.7 Measuring The Tools

Tool tips are positioned at the end of a tool shank. This puts a positioning error between the Machine Reference position and the tool tip. To remove this error you must measure the distances from the tool tip to the tool post zero point in both the X and Z directions. Then enter these distances into the MANUAL PLUS (see Chapter 6 TOOL MEASUREMENT for further details of measuring and entering tool lengths). After which positioning movements and displayed axes positions relate to the tool tip. See Figure 3.8

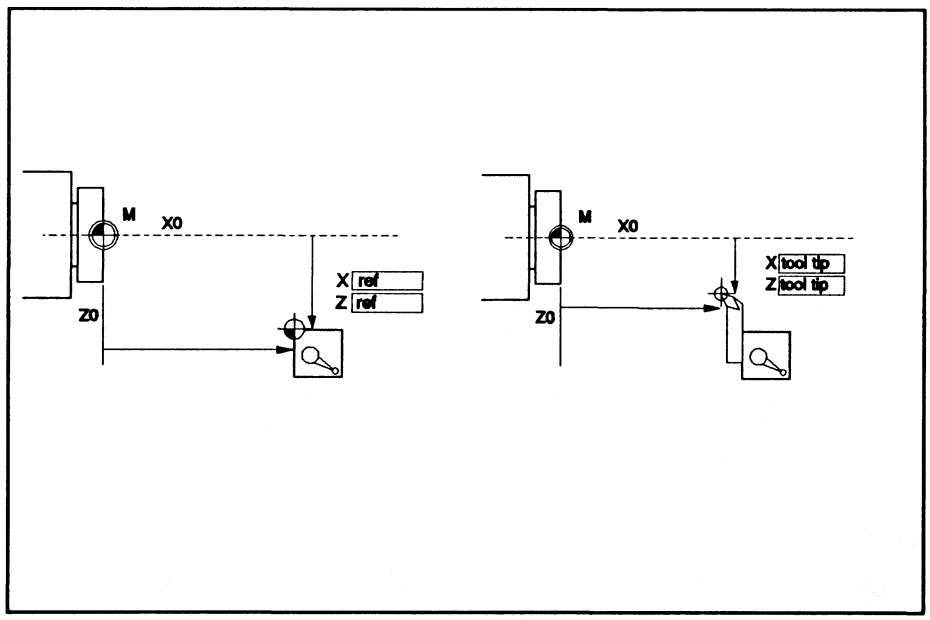


Figure 3.8: Tool Tip Distance from Machine Zero Point

3.7.1 Toolnose Radius Compensation

Cutting tools usually have a radius at the cutting tip. This radius produces dimensional errors when machining tapers, chamfers and radii. The error is caused by the difference between the real cutting point on the tool nose and the imaginary tool point define when measuring the tool. The reason for the errors, when the tool is measured, can be seen in the figure 3.9 below.

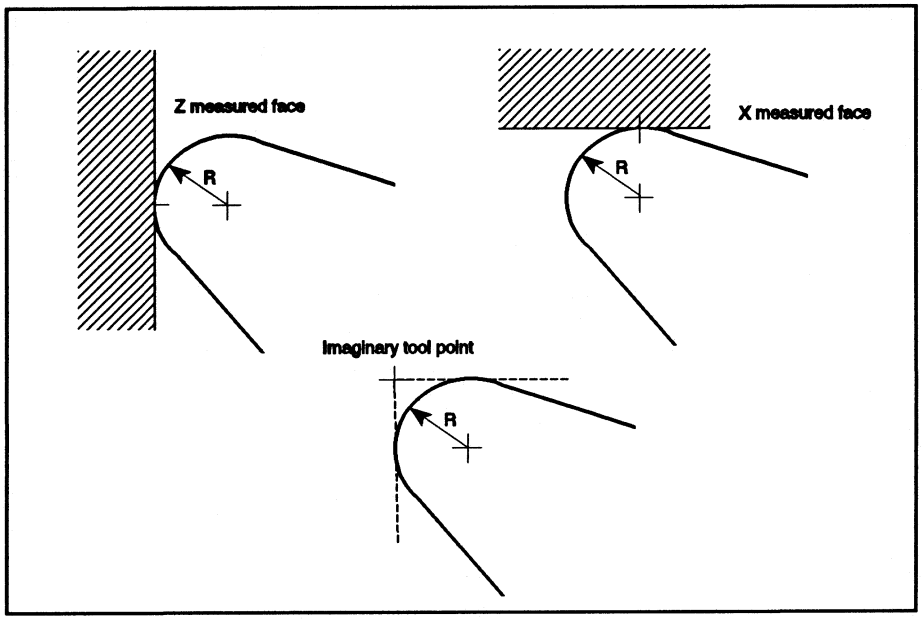


Figure 3.9: Measured Tool Point

When cutting a taper, chamfer or radius the real cutting point on the tool is not the measured tool point. Therefore, the tool radius produces a dimensional error. This error can be seen in Figure 3.10 below.

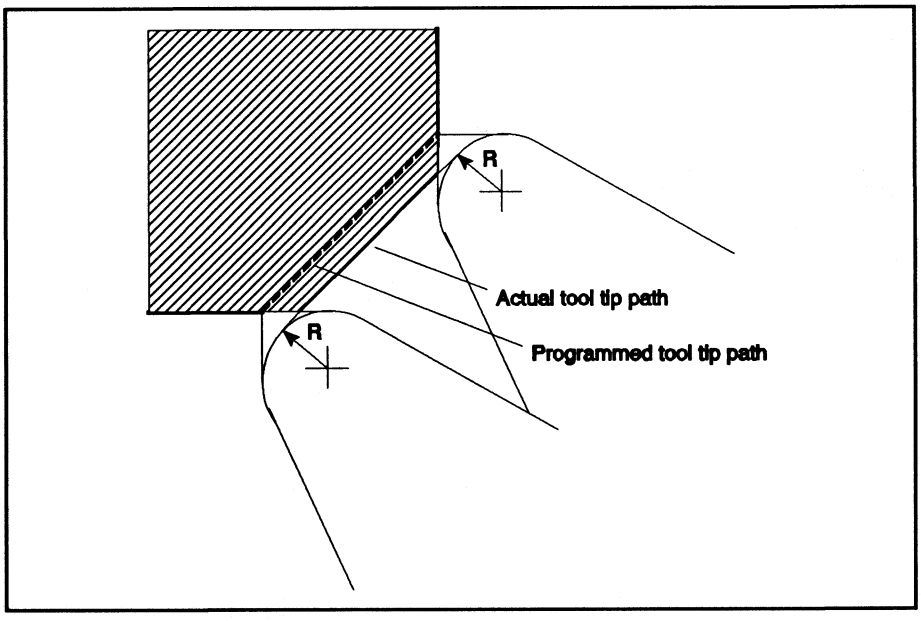


Figure 3.10: Tool Nose Radius Induced Errors

This type of error is automatically compensated when the tool radius is entered for the tool. See chapter 6 Tool Measurement for further information about measuring the tool length, tool nose radius and tool orientation

3.7.1.1 Toolnose Radius compensation for the available cycles

- Compensation at the begin of the cycle:

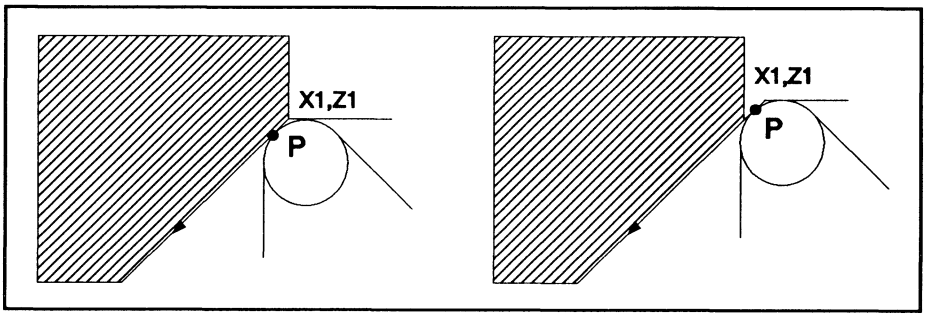


Figure 3.11: Tool Nose Radius at the begin

P is the point where tool starts cutting the taper or radius. If the tool has to start before point P, change the beginpoint or use the Chamfer/Rounding cycle (see appendix: Available Cycles).

- Compensation at the end of the cycle:

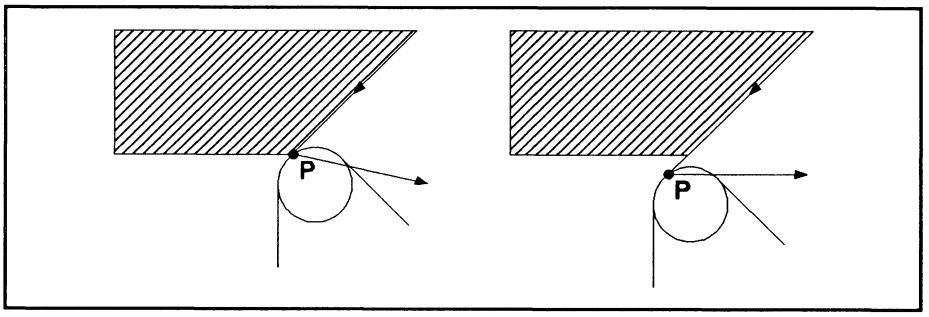


Figure 3.12: Tool Nose Radius at the end

P is the point where the tool leaves the taper, rounding. If the tool has to cut beyond point P, change the endpoint and chamfer size or use the Chamfer/Rounding cycle (see appendix: Available Cycles).

For information about which cycles have automatic toolnose radius compensation see appendix: Available Cycles

Note:

No tool radius or tool length can be entered for T0.

4. GETTING STARTED

4.1 Introduction

The first part of this chapter describes the sequence of events when turning the machine on. The remainder of the chapter describes the steps necessary for correct operation of the MANUAL PLUS.

4.2 Switching on the Machine

Power up the machine tool. The method of powering up the machine tool will depend upon the system used by the machine tool builder. Refer to the machine tool builder's documentation.

The MANUAL PLUS starts by showing a diagnostic screen as it completes its self tests. This is shown below. As each board is tested, it is marked by the word OK.

CAUTION

Diagnostic tests may fail due to power supply interference. If any diagnostic test fails, turn the power off, wait 30 seconds then turn on again. If the system fails the diagnostic test again isolate the machine and contact the appropriate maintenance personnel.

Once all tests have completed, the MANUAL PLUS initiates itself and displays the X Reference and Z Reference functions in the MANUAL OPERATE screen.

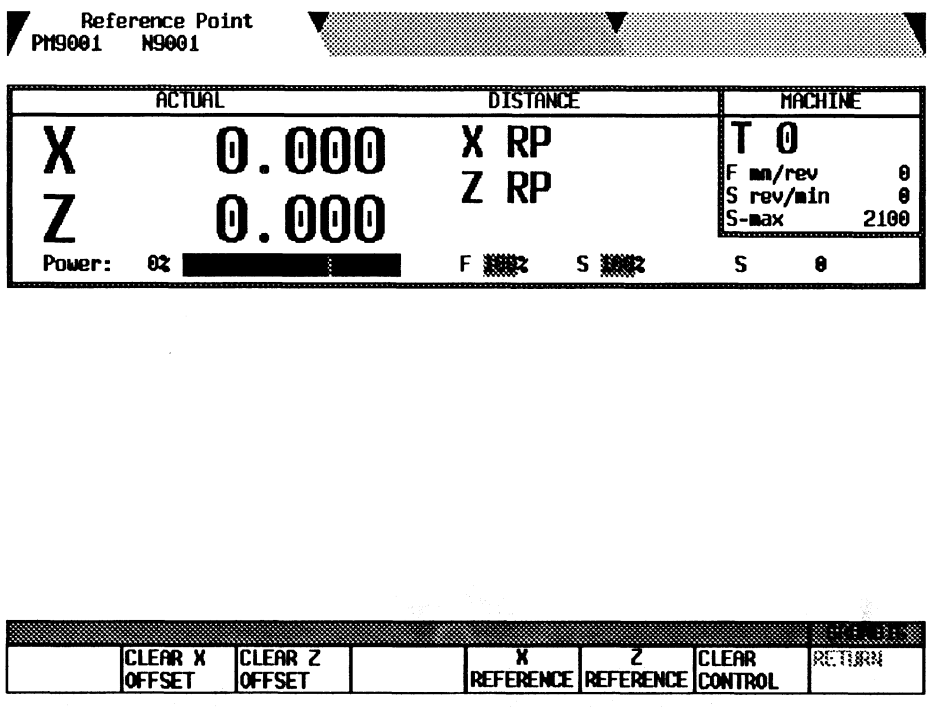


Figure 4.2: Initial MANUAL OPERATE Display

4.3 The Machine Reference Procedure

The X Reference and Z Reference procedure moves the axes to a known position. This position is called the machine reference point. It is usually set by limit switches on the slide ways. Once the machine reference point is reached the axes positions are known and the axes displays show the distance of the selected tool to the work piece zero point. After an axis reference procedure the machine zero point is known for the axis. The machine zero point for the X axis is the spindle centre line and for the Z axis maybe the spindle face (this will depend upon machine tool design and manufacturer's preferences).

X and Z Reference procedures also set internal 'limit switches'. If the axes are commanded to move beyond these limits, movement is only allowed until the limit distance is encountered. After which, further axis movement in that direction is inhibited. The axis must be moved in the opposite direction, away from the limits, to allow movement in both directions.

Note:

The X and Z positions at the reference point will show different values, depending on the actual toollength and preset axis values. The absolute position (distance to the machine zero point) is displayed when T0 is selected and the preset values are cleared. Absolute positions are mainly of interest during installation of the control system.
See Chapter 3 Machine Reference Point for a description of the machine reference point.

At start-up the X and Z Reference Function keys, functions F5 and F6, are displayed. You must always do an X and Z reference as the first action, immediately after powering-up the machine tool. To do an X and Z Reference from the main menu follow the sequence in Figure 4.3 below.

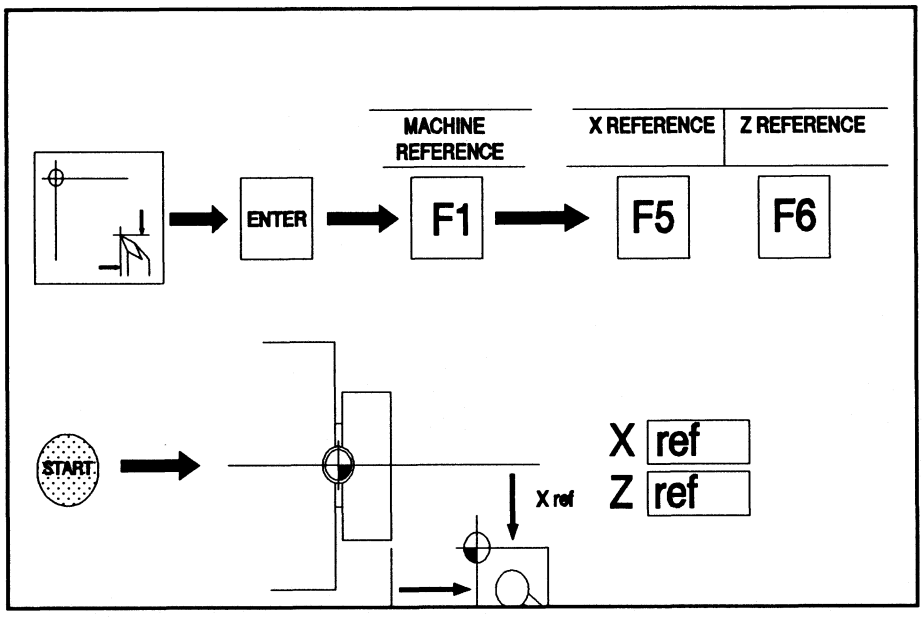


Figure 4.3: X and Z Reference Procedure

When in the reference procedure it is only possible to move the axes with the jog control or handwheels. You must complete an X and Z reference procedure before you can exit this start-up screen.

WARNING

The axes move at rapid directly to the machine reference positions. Ensure there are no obstacles in the way which may collide with the axes. Pay special attention to the tailstock.

You can perform a reference procedure for one axis then the other, to miss obstacles that may get in the way of axes movement.

5. MACHINE TOOL SETTINGS

5.1 Introduction

Before you can start machining on a traditional manual machine you must set certain machine settings, such as spindle speed and feed rates. Machining preparation with the MANUAL PLUS is no different. Only the values for spindle speed and feed rate are entered via the MANUAL PLUS rather than by selecting gear ranges with levers. This chapter describes how to enter these machining values. When the machine is switched off, the last values and rev/min or m/min setting are memorised. Figure 5.1 shows the machine status displays.

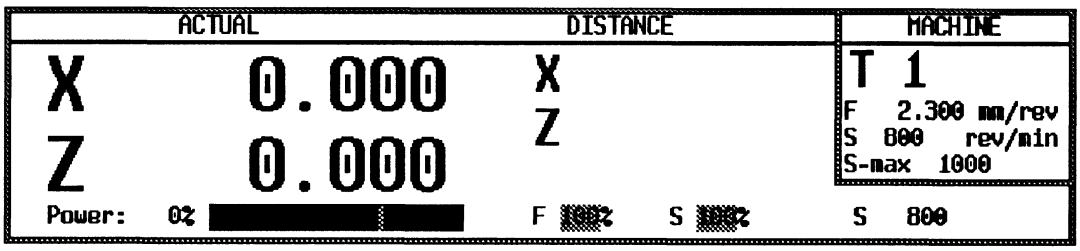


Figure 5.1: Machine Status Displays

5.1.1 S.F.T menu

Spindle speed, feed rate and tool number are all set in this menu. A list of all the tools currently in memory is displayed and Function keys gives access to the tool measuring and tool correction procedure.
Selecting the S, F and T menu, shown below, displays the data entry window. This window contains the machine settings, they are:

T	Tool number
S	Spindle Speed
F	Feed rate
D	Maximum spindle speed
A	Spindle Stop angle
S1	Power driven tool spindle speed

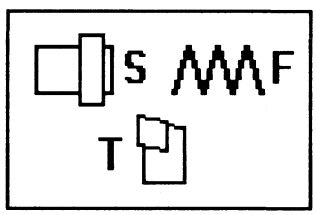


Figure 5.2: S.F.T Main Menu

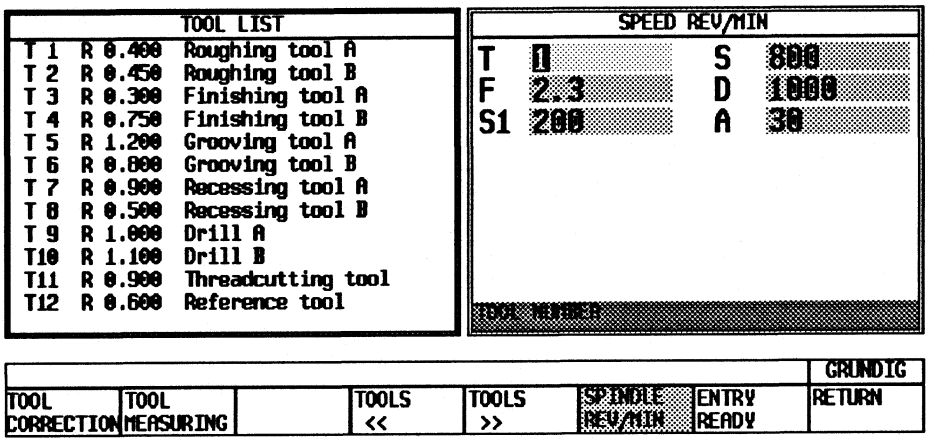


Figure 5.3: S.F and T Entry

Function keys

F1 TOOL CORRECTION — The Tool Correction procedure can be entered by pressing function key F1 TOOL CORRECTION (see chapter 6.14 Tool wear).

F2 TOOL MEASURING — The Tool Measuring procedure can be entered by pressing function key F2 TOOL MEASURING (see chapter 6 Tool measurement). This function is also used to enter a tool radius.

F4 TOOL << / F5 TOOL >> — Optional functions, only available with the extended tool table. Use these two function keys to step through the list of displayed tools.

F6 SPINDLE REV/MIN — This is a toggle action function key. Pressing it switches the spindle speed between meter/min and rev/min.

F7 ENTRY READY — Pressing F7 terminates data entry. Data is stored and made active when the START push button is pressed.

F8 RETURN — Pressing F8 exits the current menu and displays the previous menu.

The figure below explains how to enter the values, in this case the feed F.

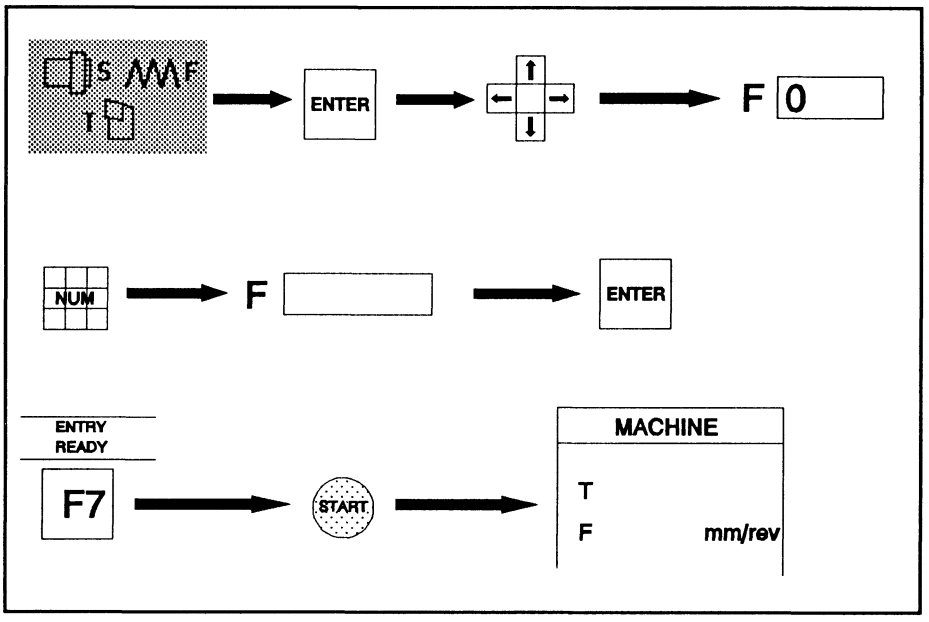


Figure 5.4: Example of setting the Feed Rate F

5.2 F= Feed Rate

F represents the FEED RATE setting and is specified in millimetres-per-spindle revolution (mm/rev). Determine the feed rate you require and enter the value for F.

The feed rate is tied to the spindle speed, the same as in traditional manual machine tools, but with one difference. The MANUAL PLUS has a feed rate override control which allows you to vary the feed rate during operation. Feed rate override changes the set feed rate in percentage points. The programmed feed rate is displayed in the MACHINE window, while the override percentage is displayed in the ACTUAL window. Both windows are in the MANUAL OPERATE screen. See Figure 5.5 below:

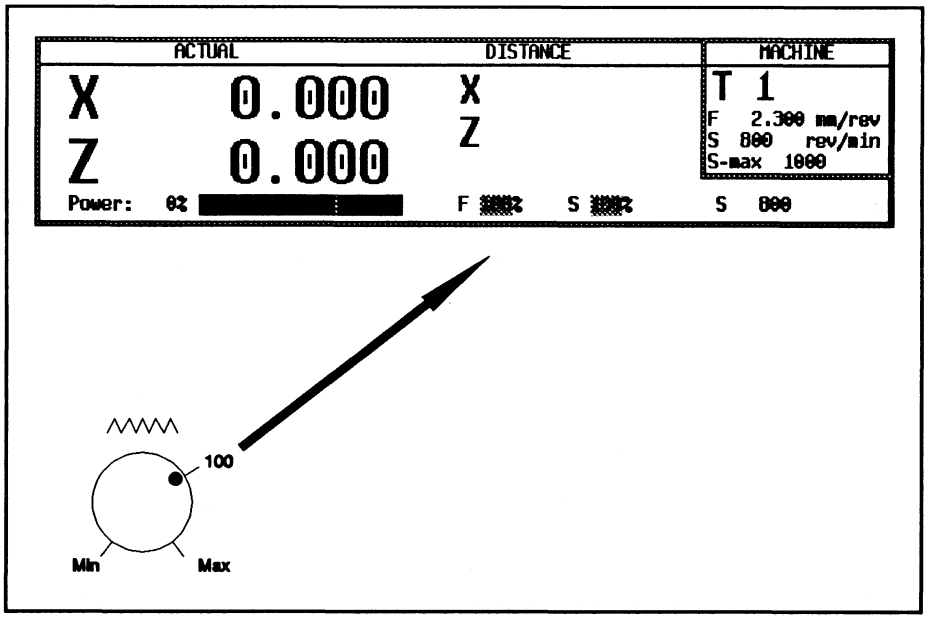


Figure 5.5: Feed Rate and Override Value Displays

5.3 S= Spindle Speed

S represents the spindle speed in revolutions/minute (rev/min) or as a constant cutting speed in metres/minute (m/min). Before you can start the spindle, as with a traditional manual machine tool, you must set the spindle speed.
Once set, use the spindle direction and stop controls on the machine control panel to start, stop and change spindle direction.

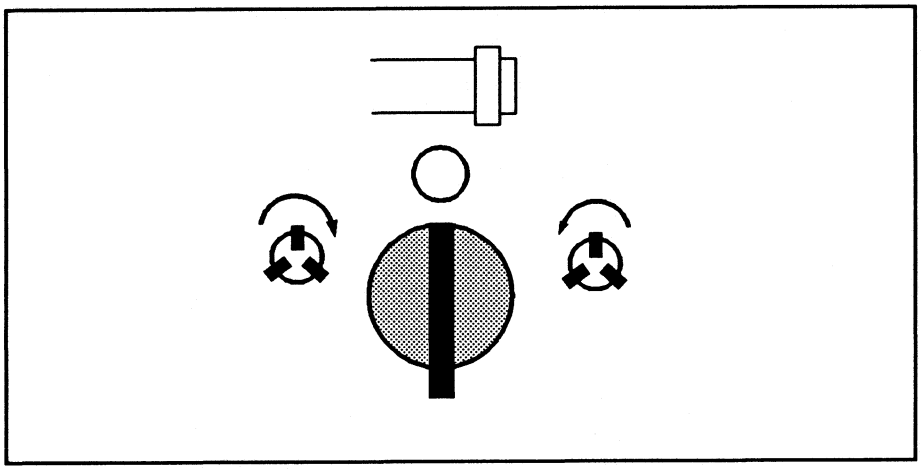


Figure 5.6: Example of Spindle Start Controls

The currently entered spindle speed and maximum limit set for spindle speed are displayed in the MACHINE window. The actual measured spindle speed is displayed in the ACTUAL window. Both these windows are in the MANUAL OPERATE screen. See Figure 5.5.

5.3.1 Revolutions per Minute (RPM settings)

To set the spindle speed in rev/min, determine the spindle speed you require. Switch on function key F6 SPINDLE REV/MIN and enter the speed S in rev/min.

5.3.2 Metres per Minute

MANUAL PLUS has another way of setting spindle speed, this is constant cutting speed. In constant cutting speed the speed of the tool tip across the work piece is maintained at a constant velocity in meters per minute. The effect is that as the diameter decrease, spindle speed increase to compensate for the reduced work piece circumference.

Note:

This mode can only be used successfully when the measuring system knows where the centre line of the spindle is positioned with respect to the tool tip. For more information about tool tip location see Chapter 6 TOOL MEASUREMENT and Chapter 4.3 The Reference Procedure.

Constant speed cutting must not be used when drilling.

Start by looking at information for the best cutting speed for the tool tip / work piece material combination. This will be expressed in m/min. Switch off function key F6 SPINDLE REV/MIN and enter the speed S in m/min.

5.4 T= Tool

T is used to represent a tool. The MANUAL PLUS can save and remember details of individual tools in a tool table.

Tools are set up in a holder and the distances of the tool tip from the tool post zero point are measured. See Section 3.7 Why Measure the Tool and Chapter 6 TOOL MEASUREMENT. Information about the tool nose radius and X and Z lengths are stored to the tool number. When the tool is used, the tool number is entered and all the information about the tool is made known to the MANUAL PLUS. See Chapter 6 TOOL MEASUREMENT for more details about entering tool data.

The currently active tool number is displayed in the MACHINE window in the MANUAL OPERATE screen.

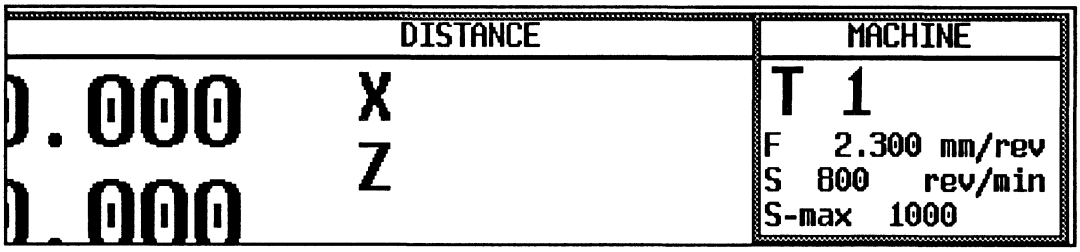


Figure 5.8: Tool Number Display

To select a tool follow the sequence in Figure 5.9 below.

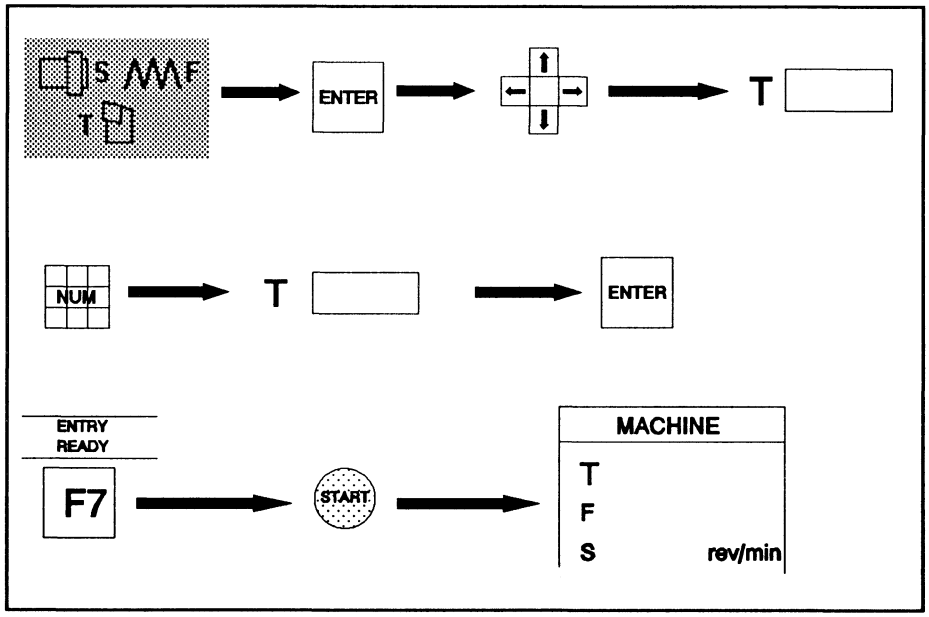


Figure 5.9: Selecting a Tool

5.5 D= Maximum Spindle Speed

In constant cutting speed mode the spindle speed will continue to increase as the tool tip moves towards the centre line of the spindle. For safety, and other reasons for example maximum chuck speed, it may be necessary to limit the maximum spindle speed (Note that this limitation is also effective when programming a spindle speed in rev/min). Enter the maximum spindle speed in rev/min using the following sequence.

S-max shows either maximum of the gearrange or the programmed maximum spindle speed.

WARNING

The MANUAL PLUS limits the analogue output to keep the spindle below the maximum set speed. In case of spindle over-speed refer to the machine tool builder's documentation for information about safety and machine responses.

5.6 A= Spindle Stop Angle

The angle A is the position of the spindle in degrees, used for oriented spindle stop.
The oriented spindle stop will put the chuck in the right position for opening and closing. This position depends on the type of chuck.

To find the suitable angle A, proceed as follows:
- Stop the spindle with oriented spindle stop.
- Rotate the spindle manually in the position for opening/closing the chuck.
- Read the spindle angle, displayed under actual spindle speed S. Use this value for A.

The oriented spindle stop can be executed at every spindle stop, or only when a switch is ON.
Refer to the machine tool builders documentation for details about this.

5.7 Gear Range

When more than one gear range is available, the operator has to select the appropriate range for the selected spindle speed. Refer to machine tool builder's documentation for information about gear ranges and range switching. The maximum speed (rev/min) of the actual range is displayed under S-max in the machine tool setting display.

The actual gear range is taken over in the program when a cycle is stored.
When the cycle is executed, the actual gear range will be compared to the original range by the machine tool interface. This can lead to a warning (E... "use gear range 1/2/3"), indicating the number of the gear range that was originaly used.
Refer to the machine tool builder's documentation for further information.

Note: The gear range, stored with a cycle, is only given to the interface when the spindle speed S is entered in this cycle.

5.8 S1= Power Driven Spindle Speed (option)

Power driven tools are used to perform drilling or milling operations. The spindle has to be oriented and clamped.

The speed of a power driven tool is entered with S1= in revolutions per minute.
It is displayed as S-tool on the same line as the spindle speed S. Usually, the speed is only displayed in this line when the driven tool is switched on. Refer to machine tool builder's documentation for details.

The feed for a movement with power driven tools is the value of F in mm per revolution of the tool.

The machine can have one of the following configurations:
a. The speed of the power driven tool is fixed or is set manually.
   The speed S1= is only used to inform the control system about the speed when running in mm per revolution of the tool.
   The speed override switch is effective for the calculated speed of the tool and must be set to 100%.
b. The speed of the power driven tool is controled electronically.
   Here, the speed S1= is used to control the speed of the power driven tool.
   The speed override switch will influence the speed of the tool.

For special applications, it is possible to let the power driven spindle run when the main spindle is also running. In this case, the feed F is in mm per revolutions of the main spindle.

6. TOOL MEASUREMENT

6.1 Introduction

This chapter describes why you must measure tools, how to measure tools and what happens to this information. You only need to measure tools if your machine tool has a turret type tool post or a changeable tool holder system. If you are going to unclamp and replace each tool manually in the tool holder read the sections; T0 (General purpose tool number) and Preset Axes in Chapter 7.

The MANUAL PLUS displays a position for each axis. This information is only useful when the displayed position is the real position of the tool tip relative to a known position on the work piece. When several tools are mounted in interchangeable tool holders, or on a turret tool post, there must be a way of automatically adjusting the displayed position to compensate for the variations in tool tip positions. Measuring the tool tip positions and storing this information in the tool table enables the MANUAL PLUS to adjust the displays whenever a tool is used.

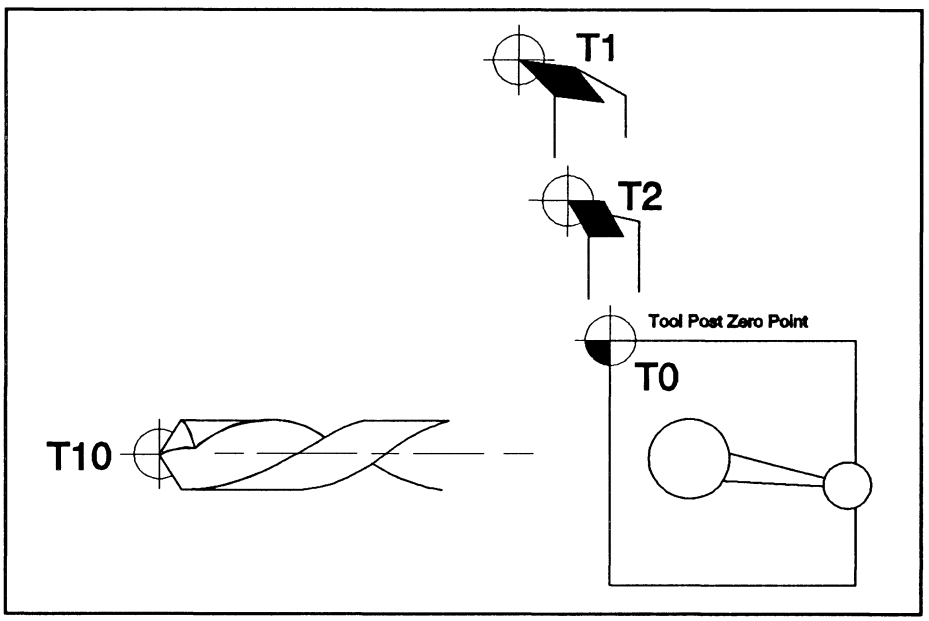


Figure 6.1: Variations in tool tip Positions

Each working tool is assigned one of the tool numbers in the Tool Table. When a tool is placed on the machine its tool number must be activated (by entering T=). During tool measurement the length of the tool and the tool nose radius are stored in the Tool table next to the tool number. Whenever a tool is used its tool number is entered (T=) into the MANUAL PLUS. Entering the tool number causes the MANUAL PLUS to read the stored tool lengths in the tool table and automatically change the displayed axes positions to show the real position of the tool tip.

6.2 Tool T0

Tool T0 is a special case tool number. No tool measurement values or tool nose radius can be entered for this tool. Use this tool number for example, for:

- conventional manual turning
- single one off cuts
- special tool (such as dedicated tool shapes)
- turning operations where tool nose compensation is not required (cycles without taper, chamfers or radii).

The axes displays must be set every time a new tool is used with T0, see section Preset Axes in Chapter 7.

6.3 Tool Measuring Method

The following tool measurement procedure has been developed to allow anyone to measure tools using only those tools and measuring systems available in a normal work shop.

There are two stages to measuring tools. The first stage is the selection of a reference tool and turning a reference face and diameter. The second stage must be completed for every tool measured. It gauges the work tools against the turned reference face and diameter.

6.4 The Reference Tool

It is advised to use the last tool number of the table for the reference tool (T48). A reference tool is a tool that is not used as a normal tool and which size is always correct. Place a tool in a holder and clearly mark this tool as the reference tool.

NEVER REMOVE THIS TOOL FROM THE HOLDER AND NEVER USE THIS TOOL AS A NORMAL WORK TOOL.

If the tool tip is damaged or the tool position in the holder changed, it will be necessary to repeat the reference tool measuring cycle again.

6.5 Setting the Tool Number

Every time a tool is placed on the tool post its tool number (T=) should be entered and made the active tool number. To set the tool number follow the procedure in Figure 6.2 below.

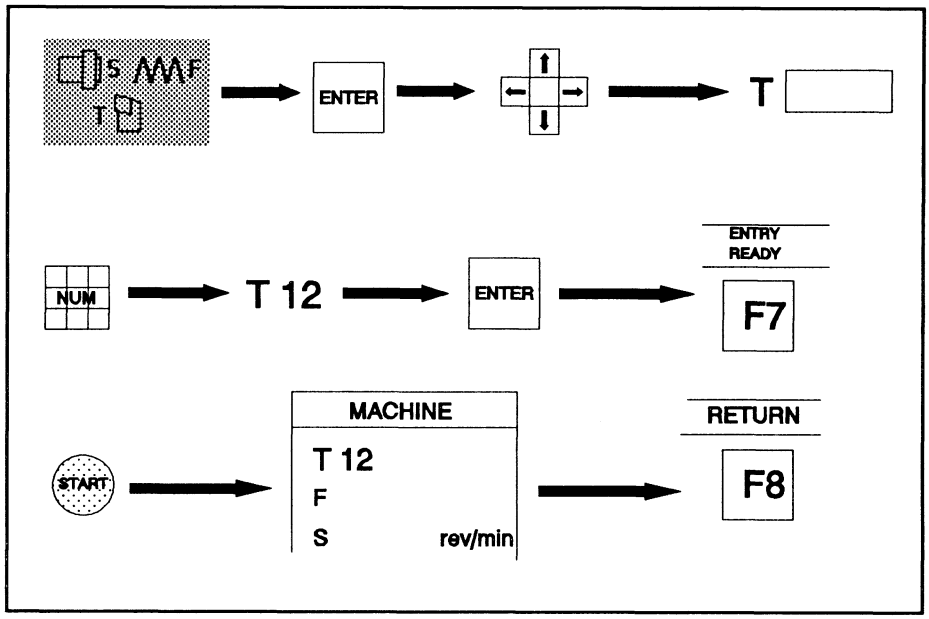


Figure 6.2: Setting the Tool Number

6.6 Setting the Reference Tool X and Z Lengths

The method described here requires a reference work piece. If using a pre-machined gauge follow all the actions, but skip the machining actions. The actions are:

1 Remove any preset axes values with "Clear Offsets" . See section 7.6.
2 Place the reference tool on the machine and activate tool T48.
3 Place a piece of material in the chuck (or the pre-machined reference gauge).
4 Select F2 Tool Measurement.
5 Machine a reference face (skip this if using a pre-machined gauge).
6 With the tool touching the face, measure and enter the distance of the reference face from the machine zero into the Z position entry box. Push F6 Take Over Z. The MANUAL PLUS now calculates the Z tool length and places it in the Tool Table.
7 Turn a reference diameter (skip this if using a pre-machined gauge).
8 With the tool touching the diameter, measure the diameter and enter this in the X position entry box. Push F5 Take Over X. The MANUAL PLUS now calculates the tool length and places it in the Tool Table.

Figures 6.3 and 6.4 below show the procedures, using a reference work piece, to set the X and Z position for the reference tool.

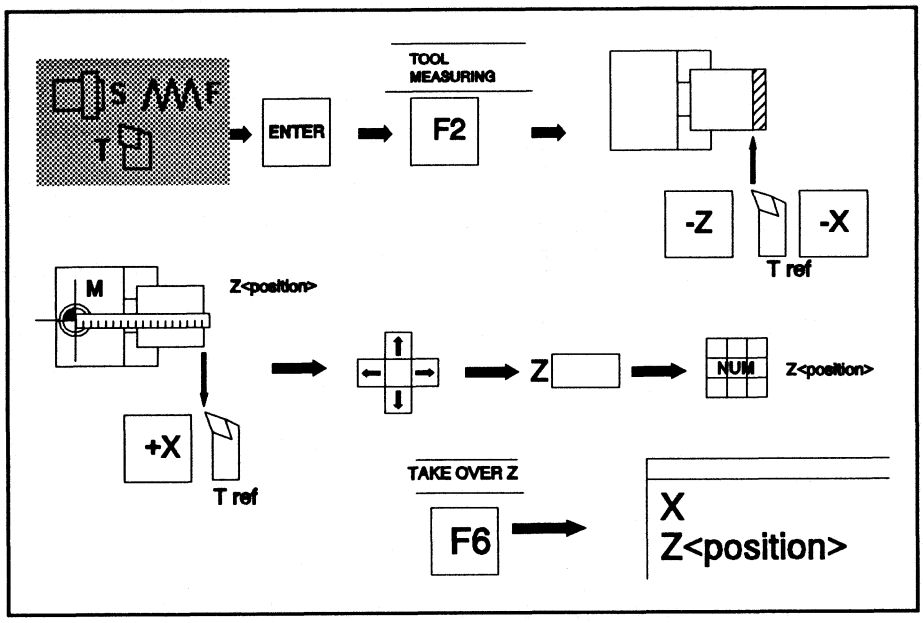


Figure 6.3: Measure the Reference Tool Z Length

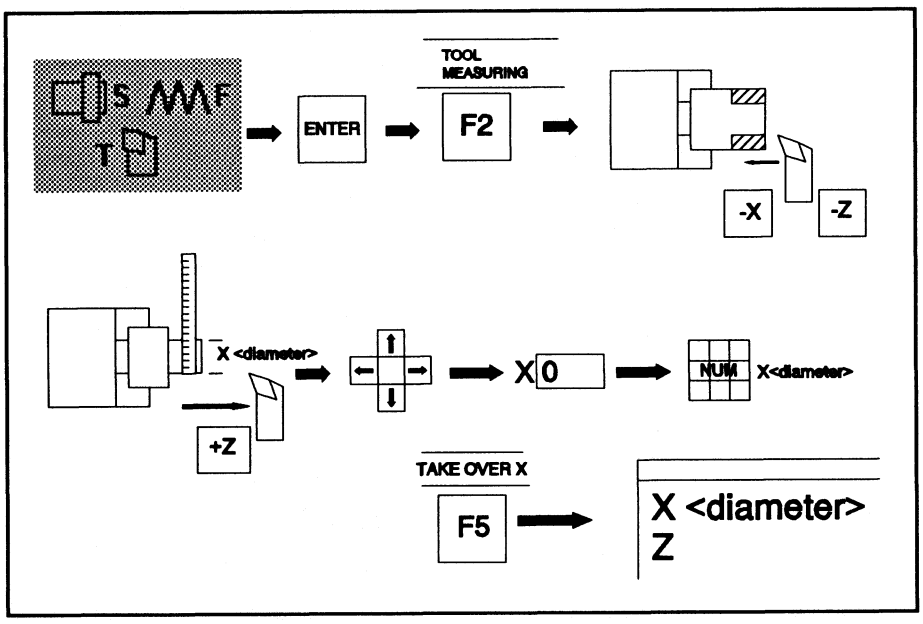


Figure 6.4: Measure the Reference Tool X Length

6.7 Measuring the Work Tools

First prepare a workpiece.

- Place a piece of material in the chuck.
- Machine the frontside and a diameter with a work tool.

Place the reference tool in the toolholder.

- Activate the tool number of the reference tool.
- Select F2 Tool Measurement in the F, S, T menu.
- Touch the face and enter the actual Z-position of the display for Z-reference.
- Touch the diameter and enter the actual X-position of the display for X-reference.

  Both values will be stored for later use. When the work piece zero point is changed with the function Preset Axis, the reference values are automatically adapted.

Place the tool to be measured in the tool holder.

- Return to the F.S.T entry and activate the tool number.
- Select F2 Tool Measurement in the F, S, T menu.
- Touch the face and press F6 TAKE OVER Z.
- Touch the diameter and press F5 TAKE OVER X.

Repeat the last procedure for each working tool.

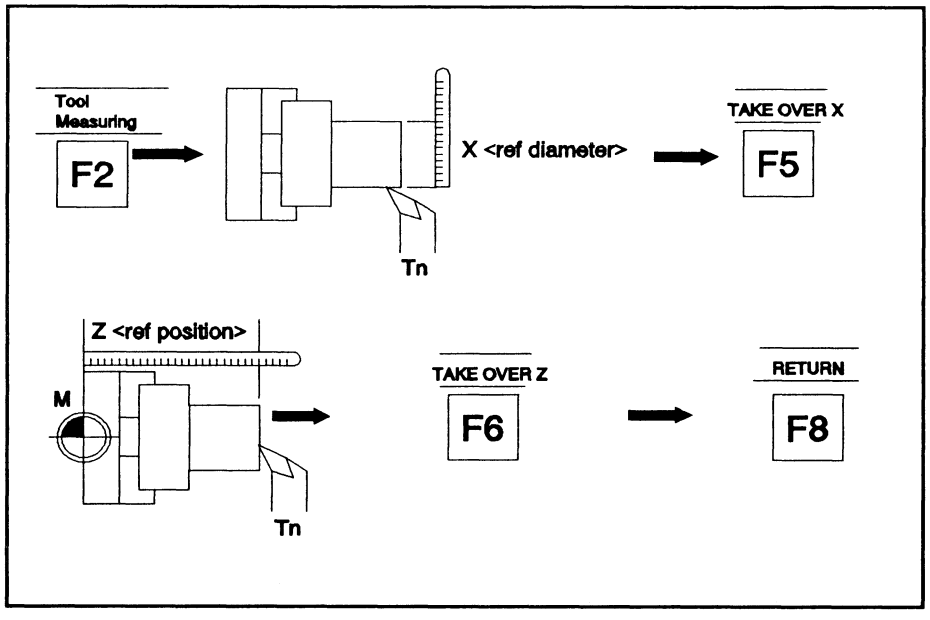


Figure 6.5: Measuring Working Tool Offsets

6.8 Measuring Internal Tools

You can measure the tip position of an internal turning tool, when only having a known outside diameter. Place a straight edge along the outside diameter and jog the tool outwards until it just touches the straight edge.

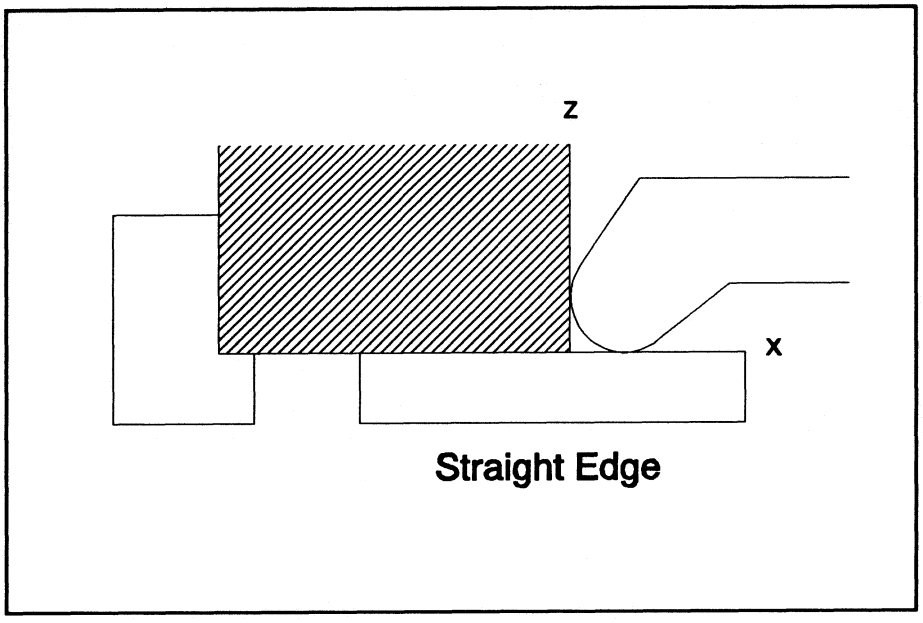


Figure 6.6: Measuring Internal Tools

6.9 Measuring Button Tools

Button tools are measured as normal tools on the side with which they are cutting.

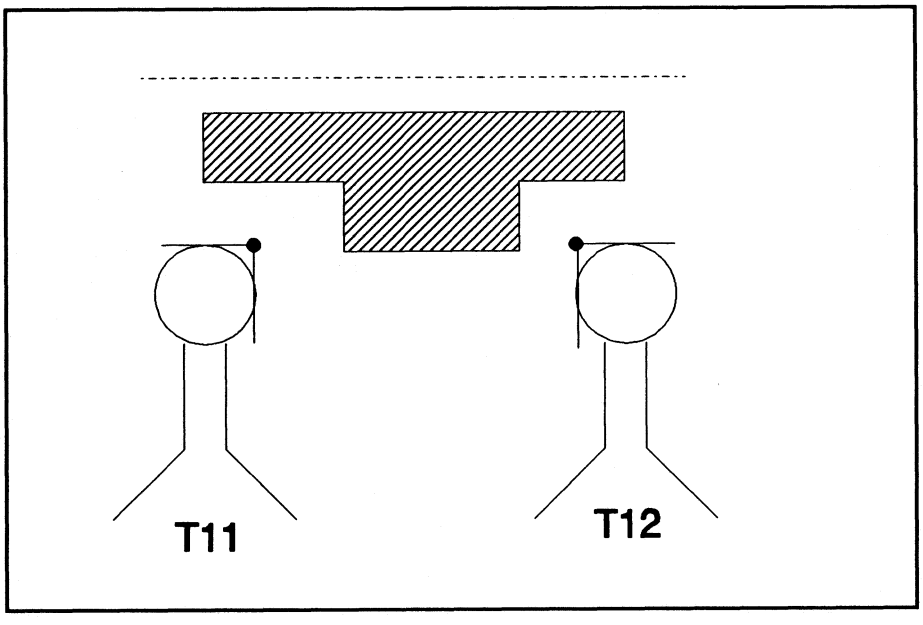


Figure 6.7: Measuring Button Tools

If a button too has to be used in two different cutting directions, use two tool numbers (i.e. T11, T12) to store the different tool length.

6.10 Measuring Thread Cutting Tools

Thread cutting tools are measured using the cutting face given in Figure 6.8 below.

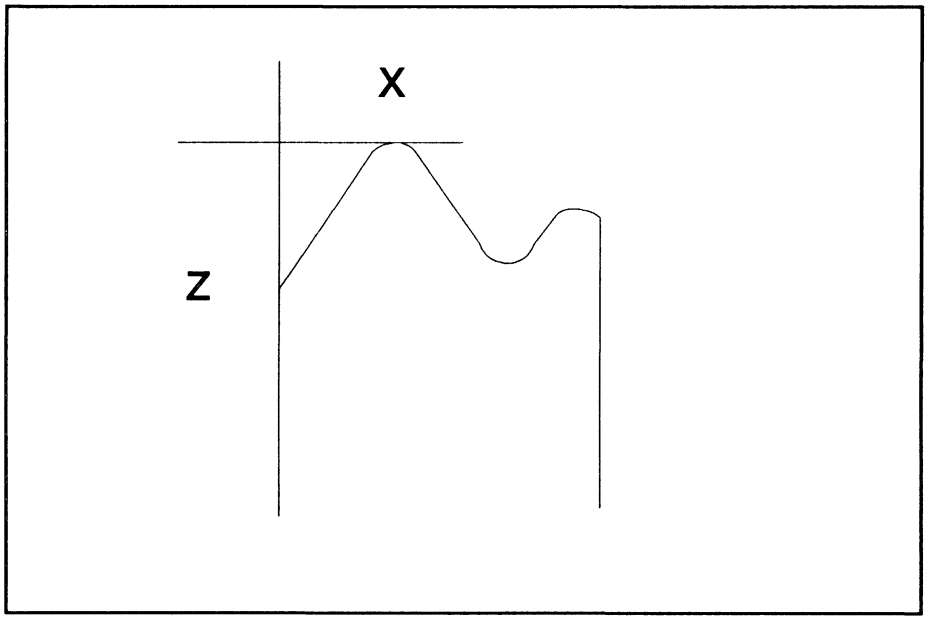


Figure 6.8: Measuring Thread Cutting Tools

6.11 Measuring Grooving Tools

Outside grooving tools are measured with one tip like outside tools. The width of the tool is given in the grooving cycles.
Face grooving tools are measured like inside tools.

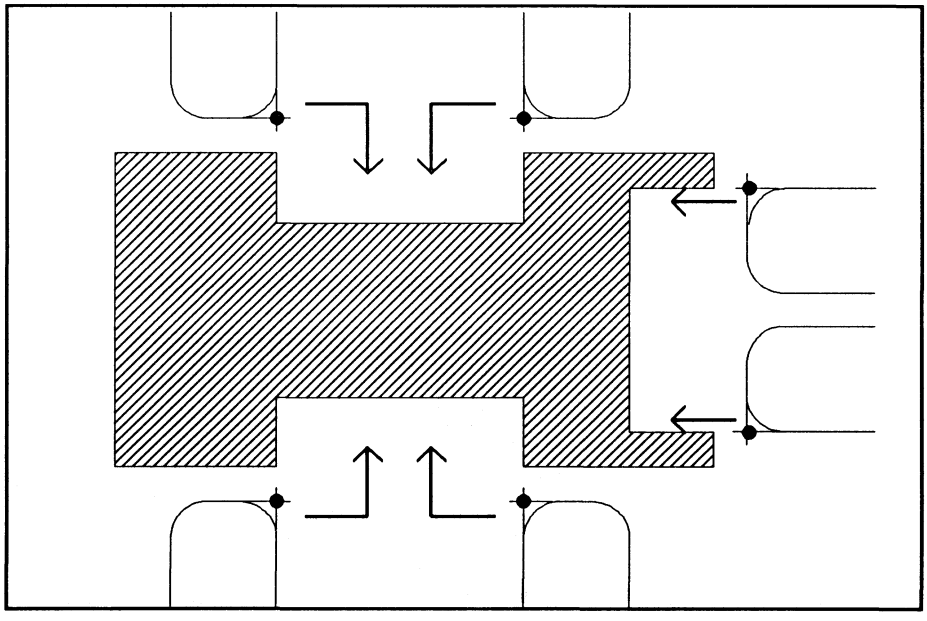


Figure 6.9: Grooving Tool Orientation

If for some reason the other tip of a grooving tool has to be programmed, it should be measured separately using another toolnumber.

6.12 Measuring a drill

Usually a drill is measured in the center, so that the drilling operation is executed on diameter X0.
In this case, constant cutting speed cannot be used. Only r.p.m. programming is allowed.
However it is possible to drill with constant cutting speed when:

1. The drill is measured on the outside diameter, like an inside turning tool.
2. The drilling operation is executed at the diameter of the drill.

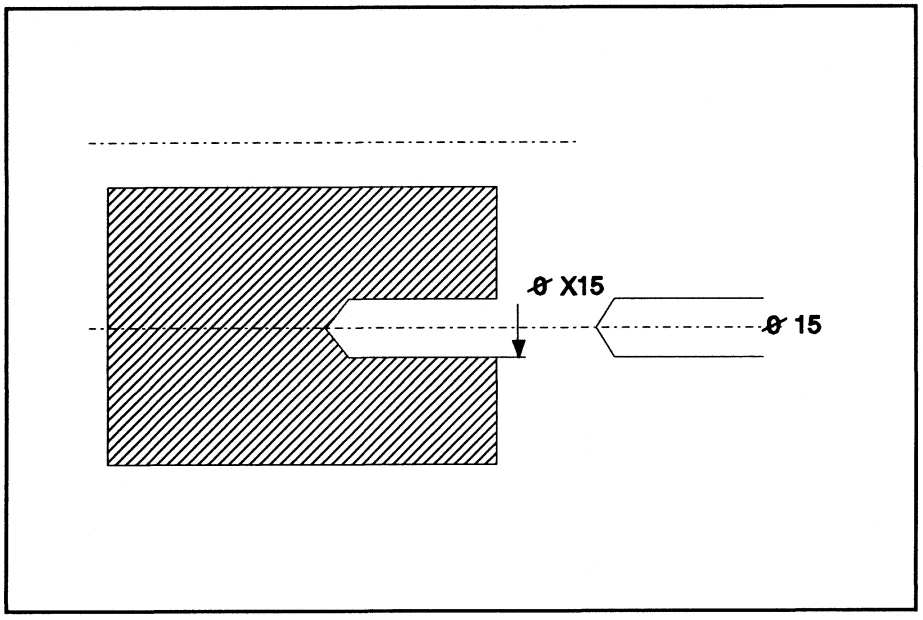


Figure 6.10: Measuring Drilling Tools

6.13 Tool Orientation

Several cycles use automatic tool radius compensation. This function automatically recognises 4 machining directions.
Outside- and inside-cutting, both right and left. These machining directions correspond with 4 tool orientations. The tools have to be measured as they are used.

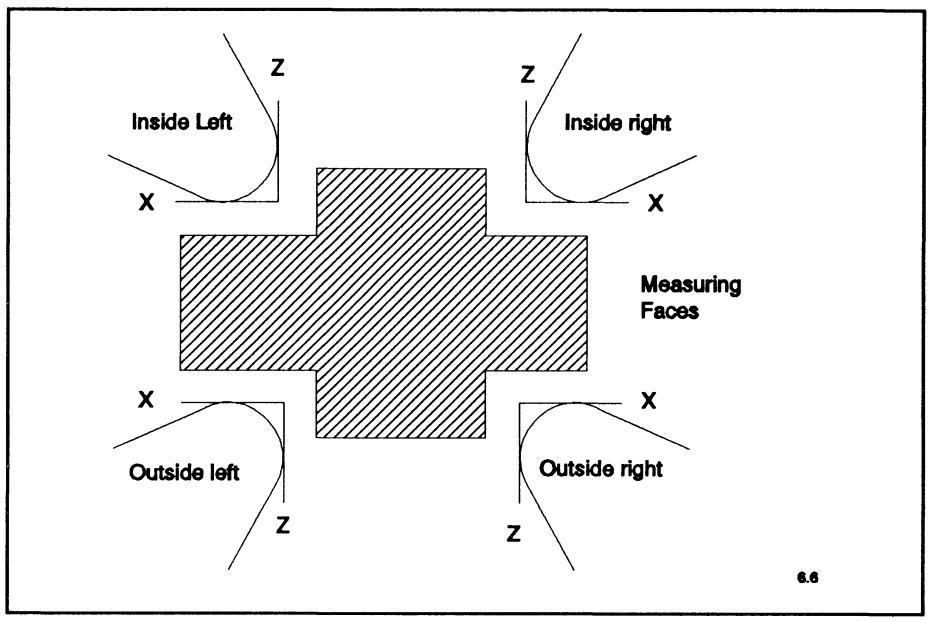


Figure 6.9: Tool Measurement Orientation

Note:
It is not possible to write a tool orientation in the tool memory.

6.14 Tool Wear Compensation

Compensating for tool wear in the MANUAL PLUS is simple.
The offsets of the actual tool can be corrected in the X- and Z-axis. This is done by moving the tool with the handwheels or the Jog keys after selection of the Tool Correction function. The corrected distance is stored in the U- or W-value of the actual tool in the tool memory.

The Tool Correction function can be entered via:

a. The F,S,T menu (see chapter 5)

b. After execution of a cycle in Teach In.

After pressing F1 TOOL CORRECTION the display shows the following function keys:

							GRUNDIG
	CLEAR X CORR.	CLEAR Z CORR.		X CORR. TOOL	Z CORR. TOOL		RETURN

Figure 6.10: Function keys for TOOL CORRECTION

Function keys.

F2 CLEAR X CORR. Clears the total tool wear correction in the X-axis of the actual tool.

F3 CLEAR Z CORR. Clears the total tool wear correction in the Z-axis of the actual tool.

F5 X CORR. TOOL Enables the tool wear correction function for the X-axis.

F6 Z CORR. TOOL Enables the tool wear correction function for the Z-axis.

F7 STORE Stores the correction values as indicated in the display under 'Distance'.

F8 RETURN Return to the previous situation. When F8 RETURN is pressed before F6 STORE, the corrections are not saved.

The procedure for tool wear correction in one axis is:

1. Select the axis to be corrected with function key F5 CORRECT X or F6 CORRECT Z.
2. Move and correct the tool with the handwheel.
   The display will show the amount under 'Distance'.
3. Press F7 STORE.
4. Leave the tool correction procedure by pressing F8 RETURN.

Note:
Tool wear correction has no effect on Tool T0.

6.15 Displaying the Tool Table

Tool information is stored in a table, in the CONTROL screen under the Toolmanagement menu. To display the tool table follow the sequence in Figure 6.11: below.

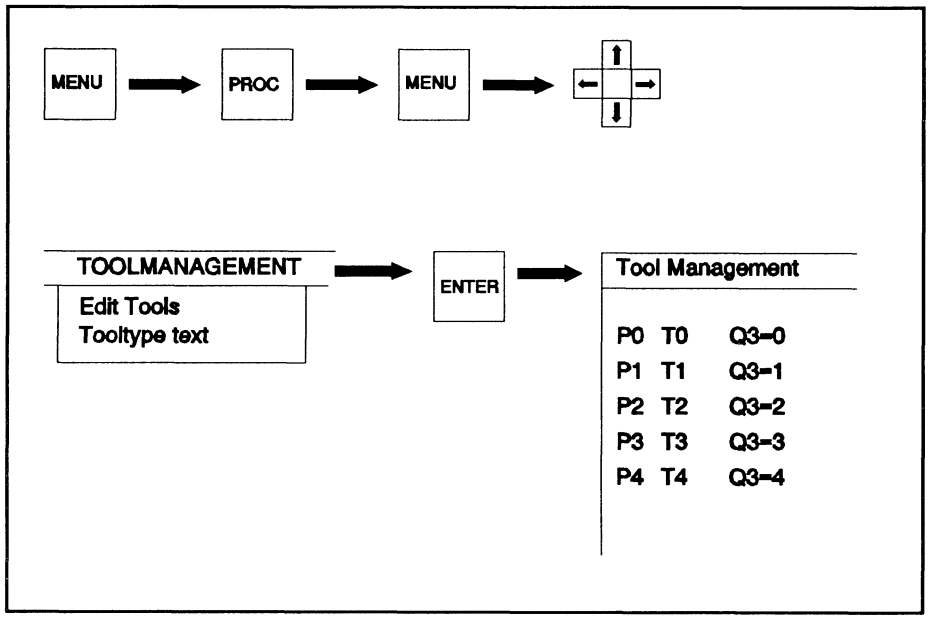


Figure 6.11: Displaying the Tool Table

The information displayed in the Tool Management table is:

P	Position of the tool in the Tool Management table.
T	Unique tool number assigned to the tool. Used to identify the tool and make it the active tool in MANUAL and CYCLE operations.
X	Measured X axis tool length.
Z	Measured Z axis tool length.
U	Tool wear X axis.
W	Tool wear Z axis.
R	Tool nose radius.
O	Tool orientation, automatically generated by the system, no entry, display only.
Q3	Tool type, related to the tool name in the Tool Type Table.

To be able to use a tool from the Tool Table, the Tool Table must contain the T number. The MANUAL PLUS is supplied with T numbers already entered (T1 to T48). Use the F2 EDIT TOOL function key to enter new T numbers and Q3 values (T1, T2, T3, etc and Q3=1, Q3=2 Q3=3, etc.) if tool information is lost. All other information is added to the table when a tool is selected and measured.

6.16 Displaying the Tool Type Text Table

The Tool type text table contains a text description of the tool in the Tool table. Each tool number is assigned a Q3 value, this value is the same in the Tool type table. The tool type text is displayed whenever the tool list is on the screen. To display the contents of the Tool type table follow the sequence in Figure 6.12 below.

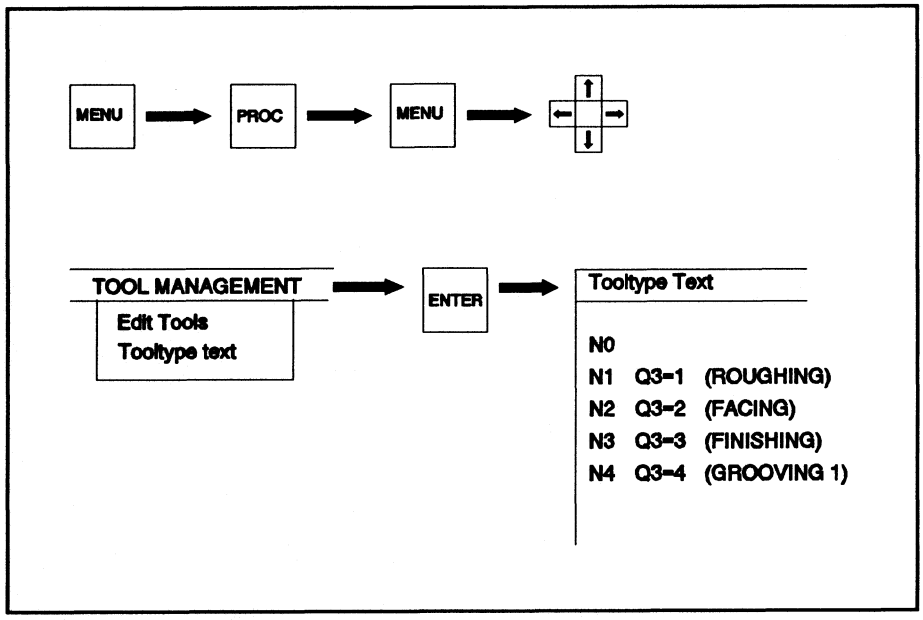


Figure 6.12: Displaying the Tool Type Table

It is possible to edit the text in the Tool type table using the edit function F3 TEXT ENTRY. Pressing F3 TEXT ENTRY displays the ASCII TEXT WINDOW. This window contains alphabetical characters not available on the MANUAL PLUS data entry panel.

Use the LEFT and RIGHT cursor keys to place the cursor at the correct position in the DATA ENTRY window (after Q3=..).

Pressing F7 CHANGE WINDOW changes the active window. The active window is indicated by the border changing colour and the cursor being positioned in the window. Pressing this Function key changes between the possible active windows. Press F7 CHANGE WINDOW to active the ASCII TEXT window. Use the cursor keys to highlight the desired character. Press ENTER to accept and enter the character in the DATA ENTRY window.

Tool texts can be up to 23 character long and must be enclosed by parenthesis (abc..). Once you have entered all the characters press F8 RETURN to leave the ASCII TEXT window then press STORE to place the text in the text table.

Note:

The values given for Q3 must be the same in both the Tool Table and Tool type table for the same P and N numbers.

Tool Table	*Tool Type Table*
P1 T1 Q3=1	*N1 Q3=1 (abc....)*
P2 T2 Q3=2	*N2 Q3=2 (def....)*
...	...

6.17 Pre-Measured Tools

For machine shops which have tool stores and hold pre-measured tools, the measured tool length can be entered directly into the MANUAL PLUS Tool table. Activate the tool and follow the procedure in Figure 6.13 below to enter pre-measured tool lengths.

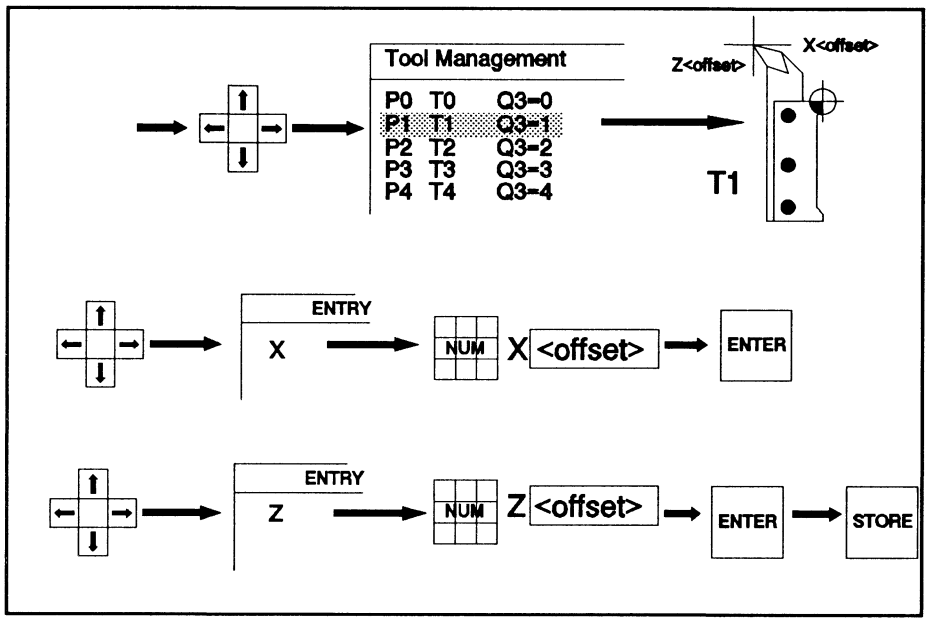


Figure 6.13: Entering Pre-measured Tool Lengths

Pre-measured tool lengths must be measured from the tool tip to the Tool Post Zero Point.

7. MANUAL OPERATION

7.1 Introduction

This chapter describes how to use the controls on the MANUAL PLUS to operate the machine manually.

7.2 Work Method

As with a manual machine there are certain machine functions which must be set before machining material. They are:

- Set the spindle speed.
- Select the tool number (or leave T0 active).
- Set the feed rate (for the jog control).
- Preset axes positions.

7.3 Handwheel Operation

As the handwheels are rotated the axes follow, exactly the same as a conventional manual machine tool. You will also notice that the axes position displays change by the amount each axis moves. The handwheel gain, that is the distance moved by an axis for each rotation of the handwheel, can be changed by a selector switch on the machine control panel, giving three levels of positioning accuracy.

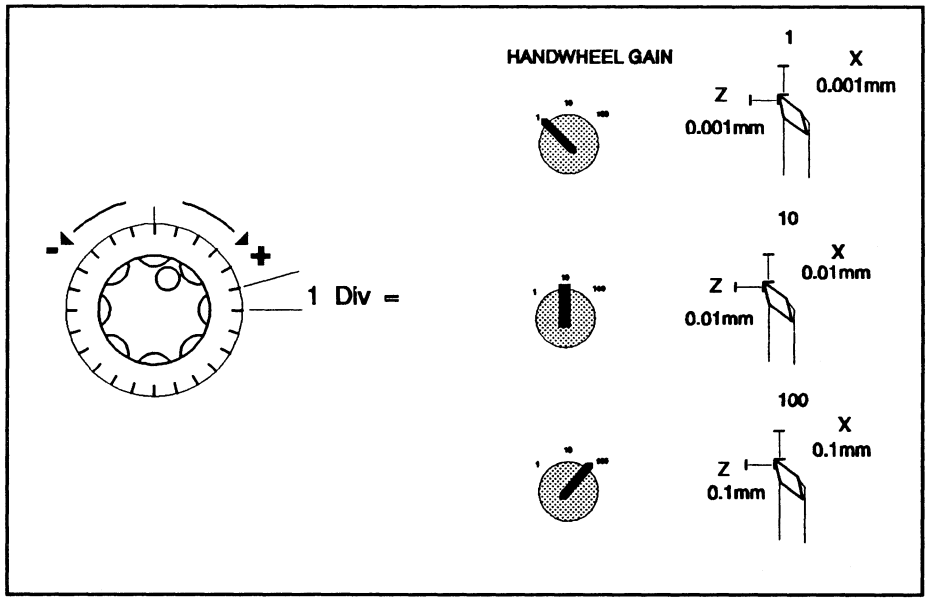


Figure 7.1 Handwheel Operation

Note:

After cycle stop the handwheels may not operate. Press the Clear push button on the Data Entry panel to enable the handwheels.

7.4 Jog Control Operation

The axes jog controls move the axes in the indicated direction at rapid jog speed or at the set feed rate. The jog rate is selected by a switch on the jog control. You will notice that as axes are moved the displayed axes positions change by the amount moved. The feed rate mode will only operate when the spindle is running (feed rate is in mm per spindle revolution).

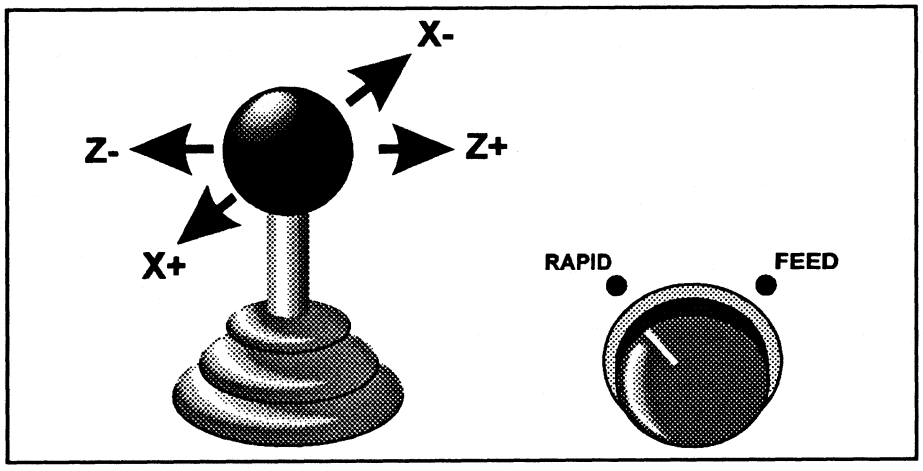


Figure 7.2: Example of Jog Selection

7.5 Preset The Axes

The Preset Axes function is used to preset the axes displays. Real measured positions of the tool tip can be entered, or work piece zero points set. Entering values for X and Z does not move the axes. The entered values for X and Z are placed in the position displays, creating a new workpiece zero point. The calculated zero offsets are displayed in the static picture. Figures 7.3 and 7.4 show the procedures to preset the X and Z axes.

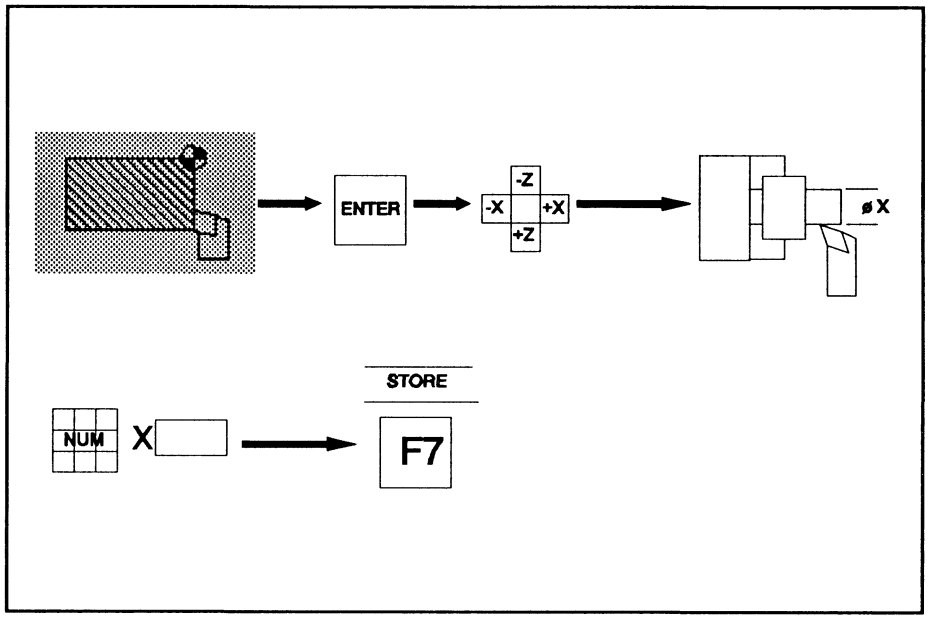


Figure 7.3: Preset X axis

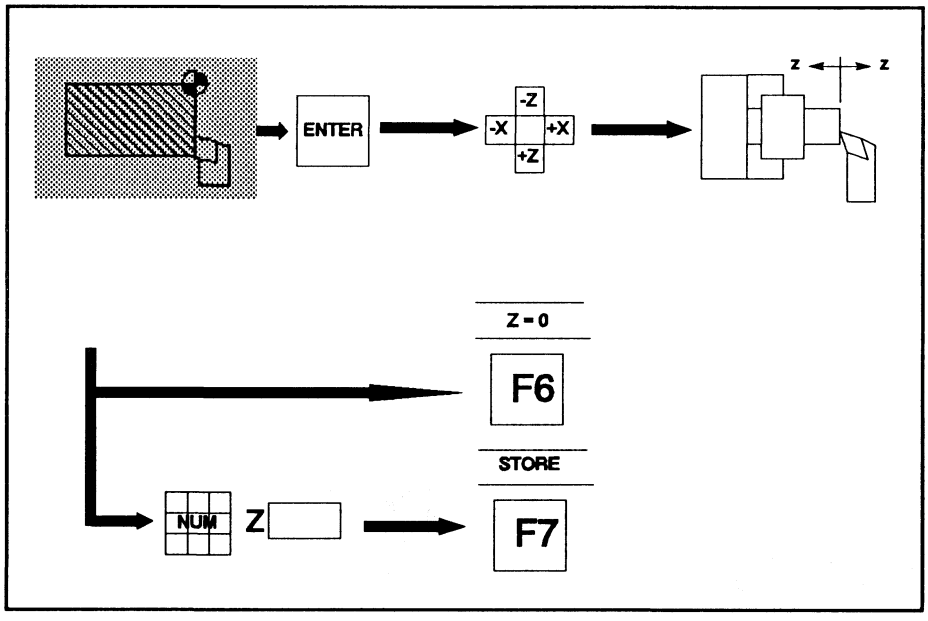


Figure 7.4: Preset Z axis

It is also possible to preset the spindle position with a value for A .
All spindle positions are then relative to this value:

- Spindle Stop position A in the F.S.T menu.
- Spindle position A7 in Thread Recut cycles (option).
- Spindle position A in the Bolt Hole Drilling cycles (option).

When the preset value of A is changed, the Spindle Stop position A in the F.S.T menu is adapted to.

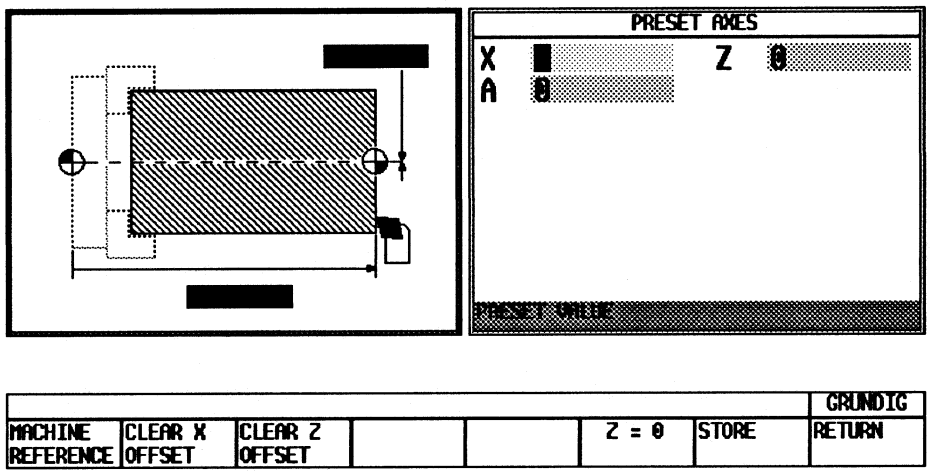


Figure 7.5: Preset Axes Entry

Function keys

F1 MACHINE REFERENCE — The X and Z Reference procedure can be entered by pressing function key F1 MACHINE REFERENCE (see chapter 4.3 The Machine Reference procedure). This procedure is only needed after start-up and after certain error conditions. Doing a reference procedure does not clear the Preset Axes values.

F2 CLEAR X OFFSET — Pressing F2 CLEAR X OFFSET sets the actual offset for the X-axis to zero. The actual offset is shown in the static picture.

F3 CLEAR Z OFFSET — Pressing F3 CLEAR Z OFFSET sets the actual offset for the Z-axis to zero. The actual offset is shown in the static picture.

F6 Z = 0 — Pressing F6 Z = 0 sets the Z axis display to zero. Z axis moved distances are then displayed as distances from this point. Pressing this key is the same as pressing F7 STORE. Any value entered for the X axis is saved and the Z axis display is zeroed. It is not necessary to press F7 STORE after pressing this function key.

F7 STORE — Pressing F7 stores the entered axes positions and places them in the axes displays.

F8 RETURN — Pressing F8 exits the current menu and displays the previous menu.

7.6 Cycle Operation

7.6.1 Executing a Single Cycle (Cycle Start)

Each cycle is executed as it programmed. Press the START push button on the Machine Control Panel to execute a cycle.

7.6.2 Cycle Stop

You can halt the execution of a cycle by pressing the CYCLE STOP push button. Depending upon the action being performed by the cycle, it may not stop immediately. Once a threading cutting cycle has started a cutting pass, it will complete the pass before stopping.

While a cycle is stopped it is possible to:

- move the axes using jog control or handwheels.
- press F8 CANCEL. This clears the remaining part of the cycle.
- press START. This starts execution of the cycle. Any jogged distances made during cycle stop are removed and the cycle starts execution from the point at which operation stopped.

Note:

Pressing CYCLE STOP (at any time) disables the handwheels. Press CLEAR to enable the handwheels.

7.6.3 Cycle Operation

The simplest way to use a machine tool fitted with a MANUAL PLUS is to use the handwheels and jog controls to move this axes. This does not, however, take advantage of the capabilities of the MANUAL PLUS. The MAIN MENU enables simple and complex machining operations to be entered and carried out under the control of the MANUAL PLUS.

This chapter only gives one example for each type of cycle available. Appendix available cycles contains a complete overview of all the cycles available.

7.6.4 What is a Cycle

A cycle is one complete executable operation. In the case of the MANUAL PLUS this is one machining operation selected and programmed from the cycle menu.

7.6.5 Work Method

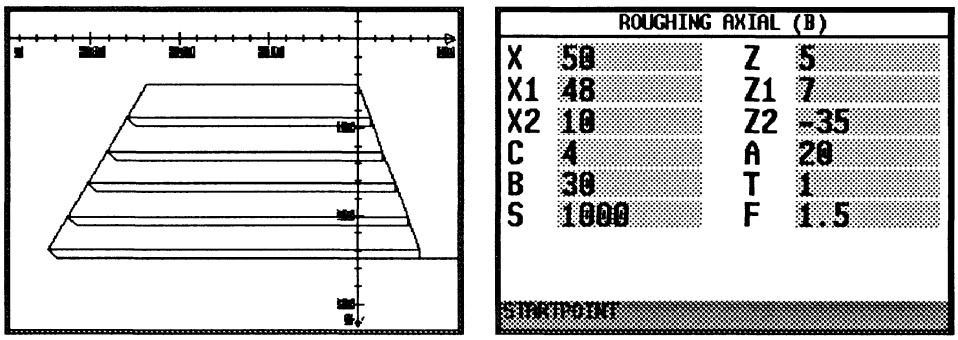
Before a cycle can be programmed it is necessary to set certain machine functions. The following method of working is suggested.

- Set the spindle speed.
- Select and set the tool number.
- Set the feed rate.
- Set the start position of the cycle (using the handwheels or jog controls).
- Select and program the cycle.

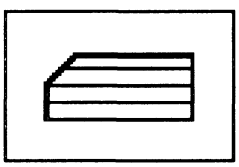
7.6.6 Graphic Cycle Check (option)

An option of the Manual Plus is the Graphic Cycle Check.
With this feature it is possible to check the programmed cycle graphically on its programmed contour and its cutterpath.

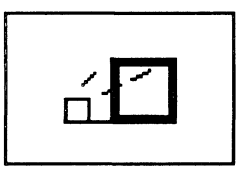
After activating this feature, the screen layout is changed. The window with the static support picture is changed to the graphic plot window. Automatically the programmed cycle is drawn. The programmed contour and the cutterpath of the cycle are shown.
With the axes scale it is possible to see if the cycle is executed at the right place and with the right dimensions.



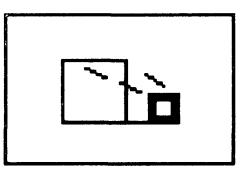
After the function key "ENTRY READY", the cycle can be executed on the machine with the "START" key, or or it can be simulated with the graphics function key F5.



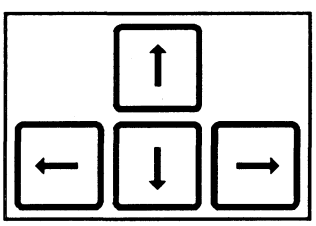
This function changes the screen-layout from data-entry to graphic plot (static picture window is changed to graphic plot window) and starts the drawing of the cycle.



This function key is active after the cycle is drawn in the graphics window. It can be used to expand the drawing step by step.



This function key is active after the cycle is drawn in the graphics window. It can be used to reduce the drawing step by step.



The cursor key's are used to shift the graphic drawing.

With the function key "RETURN" the data entry becomes active, and the cycle can be changed.

Note:

The cycles are simulated with the actual tool number of the machine status display.
When this tool is different from the tool number of the cycle, it can happen that errors are generated that do not occur during execution on the machine. This kind of errors are caused by the difference between the radius of the actual tool and the new tool of the cycle.
To overcome this, the cycle has to be started and the tool must be changed. Now, the cycle can be interrupted, canceled and simulated with the real tool.

The path of the tool is the real path, compensated for the tool radius. This means that tool path is always slightly beyond the contour. Thus, the amount of compensation can be seen in graphic simulation.

7.6.7 The Cycle Menu

Manual operations are selected from the graphical menu symbols in the MAIN MENU. Figure 7.8 below shows the MAIN MENU.

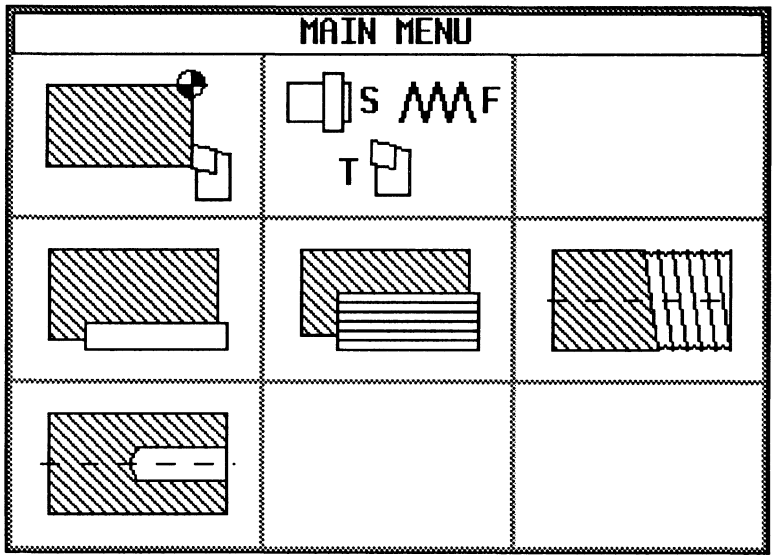
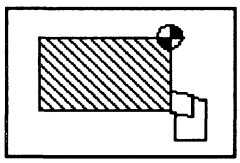


Figure 7.8: MANUAL MENU

The MAIN MENU has six graphical symbols. You can undertake all the standard machining operations from this menu. Each symbol gives access to further menus which cover presetting axes, setting spindle speed, feed rate, tool number, turning, facing, angle turning, circular turning, grooving, thread cutting and drilling.

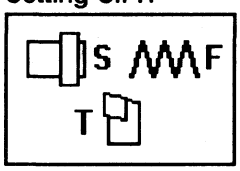
The 6 root menu symbols are:

Preset Axes



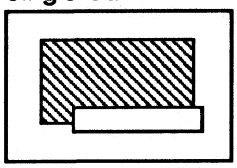
This enables new positions to be entered for the axes positions displays. The axes do not move, the axes position displays show the entered values. Used to preset the displays to show the real position of a tool tip. A function key gives access to the X and Z reference procedures.

Setting S.F.T



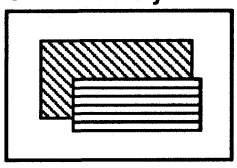
This gives access to the spindle speed, feed rate, tool number and via a function key to the tool measurement mode.

Single Cut Menu



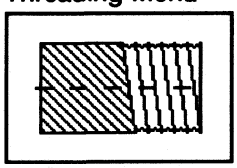
The single cut menu gives access to another menu which contains standard turning operations. Selecting one the symbols in this menu enables the programming of one machining cycle.

Clearance Cycle Menu



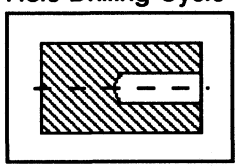
The clearance cycle menu gives access to another menu which contains standard turning operation. Unlike single cycles, clearance cycles may make several passes to complete one programmed operation. Function keys give access to extended and finishing contour cycles.

Threading Menu



The threading cycle menu gives access to another menu, from which undercuts to DIN standards, thread cutting cycles and thread recut cycles can be selected.

Hole Drilling Cycle



The drilling cycle menu gives access to another menu, from which the Drilling cycles or the Bolt Hole Drilling cycles can be selected.
After selection of a Drilling cycle, function keys give access to the extended drilling cycle which enables clearance retract actions for deep hole drilling.

7.6.8 Single Cut Menu

Single Cut menu displays further graphical menu symbols which set up and perform one tool movement. The graphics in the SUPPORT window shows the meaning of the data requested in the DATA ENTRY window for each parameter.

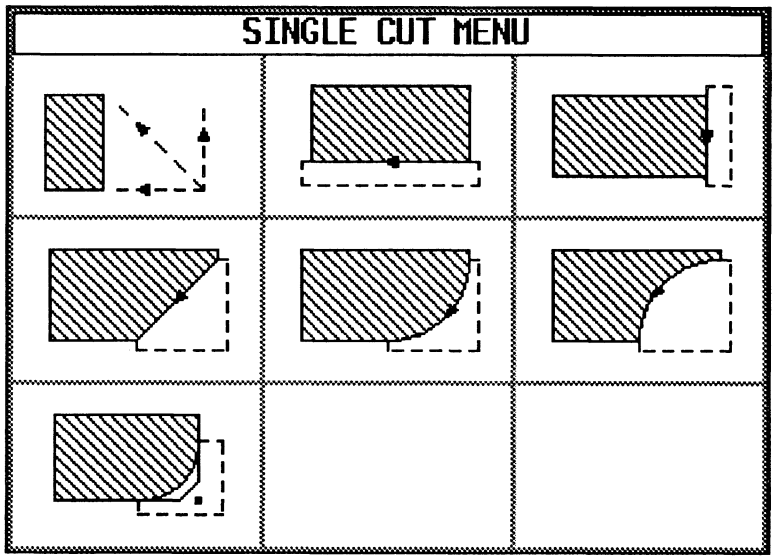


Figure 7.9: Single Cut Menu

7.6.8.1 Retract Function Key

All single cut machining cycles have a Function key to retract the tool after completion of the cut. This returns the tool to the start point of the cycle. Disabling this Function key leaves the tool at the cycle end point.

Note:

There is no tool tip radius compensation applied when retract is disabled. The MANUAL PLUS cannot calculate the compensated end point because it does not know the start point or direction of the next movement.

Example of CIRCULAR CW operation

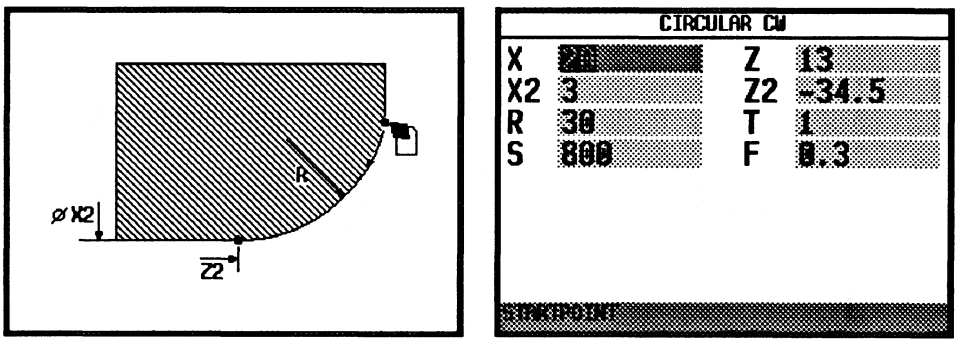


Figure 7.10: Circular CW SUPPORT an DATA ENTRY windows

7.6.9 Clearance Cycles Menu

Clearance cycles are used to remove large quantities of material before final contour machining and finishing. Clearance cycles may make one or several passes before completing execution. The number of passes depends upon the amount of material to be removed and the form being machined.

The Clearance Cycles menu displays a further three choices.

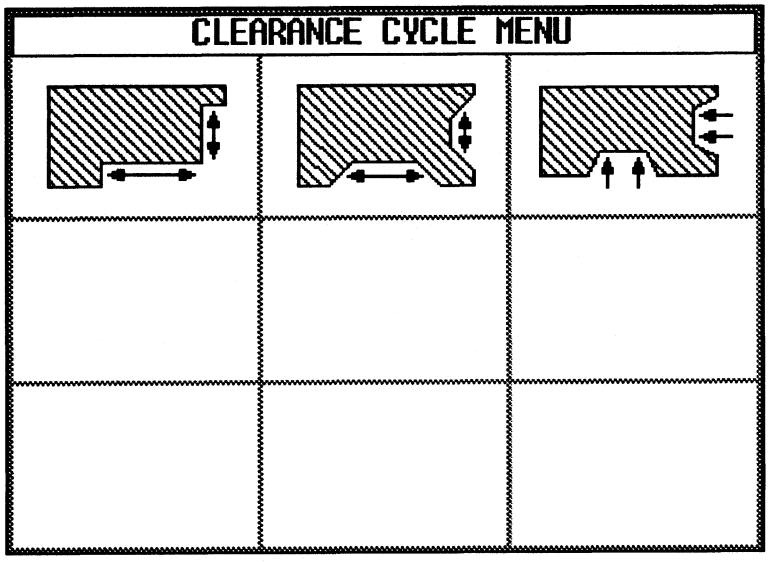


Figure 7.11: Clearance Cycle Menu

Example of CLEARANCE AXIAL cycle

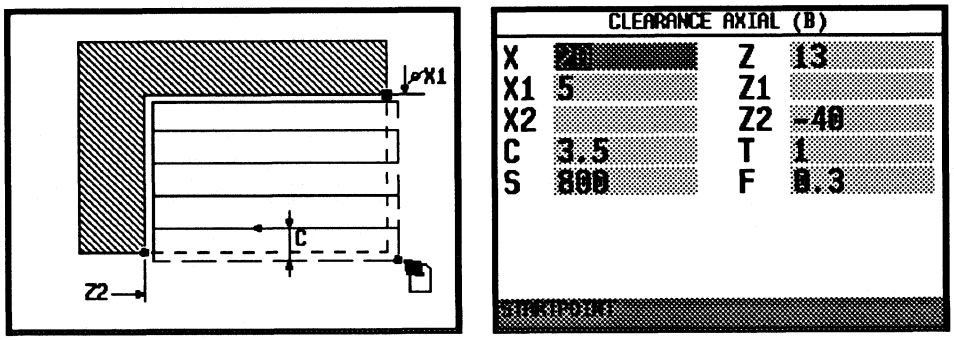


Figure 7.12: Clearance Cycle (axial)

7.6.10 Threading Cycles Menu

The threading cycles menu has four possibilities, two for different types of under cut and two for thread cutting.

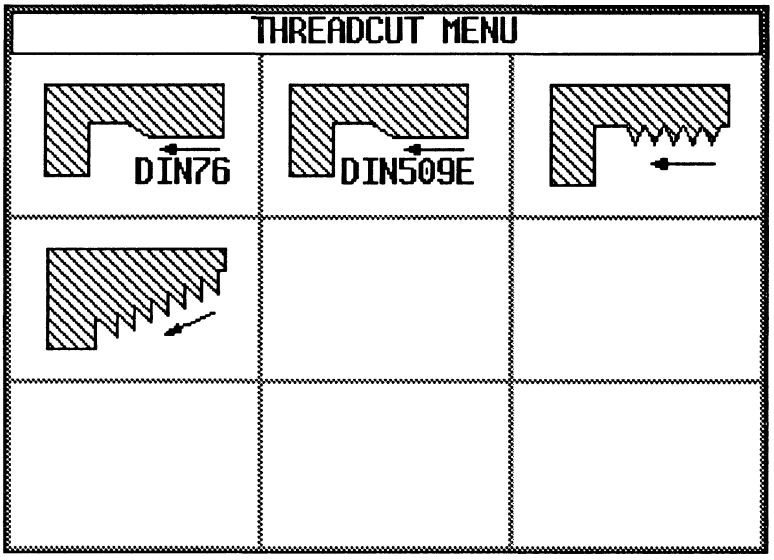


Figure 7.13: Threading Menu

Example of THREADING cycle

Thread cutting requires several passes to produce the correct thread profile and diameter. The thread cutting cycle requests data about the thread pitch, depth, length, lead-in and lead-out. The MANUAL PLUS then calculates the number of passes, depth of cuts and start position to machine the thread. Selecting thread cutting displays the following SUPPORT and DATA ENTRY window.

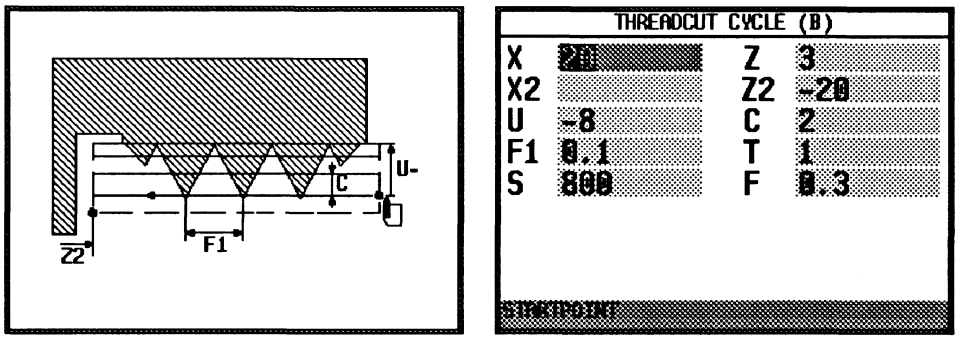


Figure 7.14: Threading Cycle

Automatic calculation number of passes.
When the first depth of cut C is not entered, the threadcutting cycles calculate a standard number of passes. The following table is used:

Pitch mm/rev	Nr of passes	Pitch	Passes	Pitch	Passes
0.50	4	1.50	7	3.0	11
0.75	4	1.75	8	3.5	12
1.00	5	2.00	9	4.0	13
1.25	6	2.50	10	4.5	14

7.6.10.1 Last Pass Function Key

It can be difficult to machine a thread to the right size. The function key F2 LAST PASS provides the possibility to repeat only the last pass of a threadcutting cycle. It is available when a threadcutting cycle can be executed and has to be pressed before starting the cycle. The size of the thread can be corrected by changing the startposition X1 or the depth U. It is also possible to use the tool correction function for this (see chapter 6.14 Tool Wear).

7.6.10.2 Thread recutting

When an existing thread should be recut, the start position of the thread does not correspond to the zero mark of the main spindle.

There are two methods for synchronising the thread to the main spindle:

A. Without Thread Recut cycles. The startpoint of the thread has to be found by trial and error. Proceed as follows:

- Select a spindle speed in rev/min.
- Select a threadcutting cycle and switch the LAST PASS function key on.
- Execute the cycle on a larger diameter by changing the start point or by using the Tool Correction function
- Observe the position of the tool in the thread during cutting.
- Correct the start point Z and repeat the cycle.
- When the tool and the thread are synchronised, ajust the start point X1 and cut the thread to depth.

B. With Thread Recut cycles (option). A position in the middle of the thread is measured. This position is indicated by Z7 and the angle A7. By means of this position, the cycle calculates the correct angle of the spindle in the start point. Proceed as follows:

- Stop the spindle with an oriented spindle stop. The actual spindle speed must display a value in degrees (decimal point).
- Select a Thread Recut cycle. Enter the known values, but not yet A7 and Z7.
- Move the tool by handwheel or Jog key in the middle of a full thread profile. The spindle may also be rotated to achieve this.
- Put the cursor on A7 or Z7 and press function key F5 TAKE OVER POSITION. A7 is the actual spindle angle.
- Move the tool away from the thread to the start position.
- Press F7 ENTRY READY and start the cycle.
  The cycle makes always one pass on the final depth.

7.7 Drilling Cycle Menu

The drilling cycle menu has 2 possibilities, one normal drilling cycle, and a cycle for bolt hole drilling with power driven tools (option).

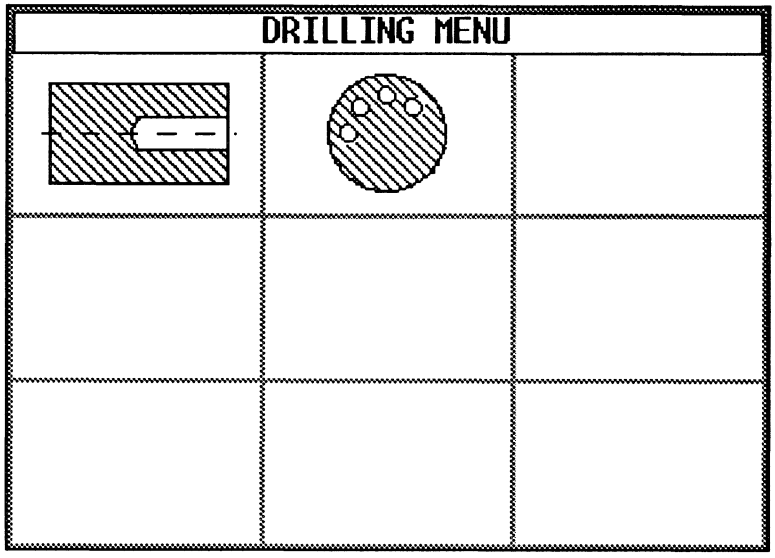


Figure 7.15: Drilling Menu

Example of a drilling cycle

When drilling holes it is often necessary to retract the drill to clear cut material and prevent drill breakage. The drilling cycle requests data about the size and depth of the hole. The MANUAL PLUS then calculates the number of actions necessary to drill the required hole. Selecting the drilling menu displays the following SUPPORT and DATA ENTRY window.

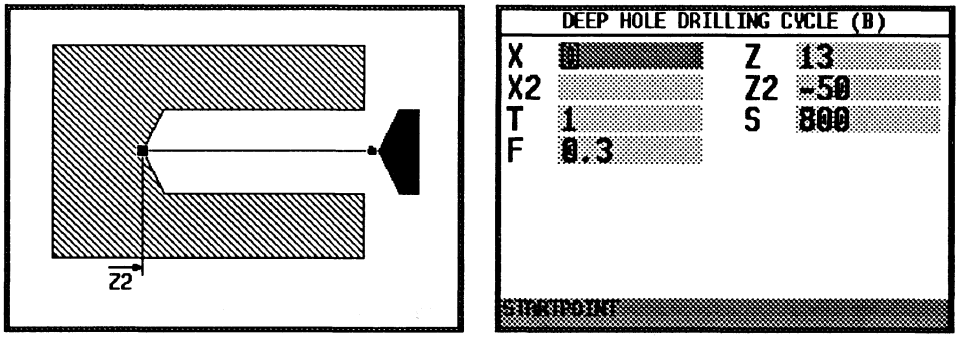


Figure 7.16: Drilling Cycle

8. TEACH-IN

8.1 Introduction

This chapter describes cycle programs, what they are and how to make them. Cycles in programs differ from single cycles in that each cycle requests data for spindle speed, feed rate, tool number and start position. These parameters can be set for each cycle throughout the program, to best suit the machining operation being undertaken by the cycle. Appendix available cycles contains all the cycles available in the MANUAL PLUS.

The MANUAL PLUS uses an enter-execute-store method for making cycle programs. The axes are positioned, a cycle is then selected, programmed, executed and placed in the cycle program. The axes are positioned for the next cycle and a cycle is selected, programmed, executed and stored again. In this way a work piece is machined as the cycle program is produced. The sequence is repeated until the complete work piece is machined. After which any number of work pieces can be produced using the resulting cycle program.

8.2 What is a Cycle Program?

A cycle program is one or more cycles grouped together to form a sequence of operations. Executing the program causes each cycle in the program to be executed one after the other. The cycle program can therefore contain instructions to machine a complete part. Starting with roughing operations, followed by contouring, drilling, thread cutting, grooving, etc.

8.3 Work Method

Before beginning to assemble a cycle program you must first determine the order of machining operations required to produce the finished part. There is also certain information which must be available in the MANUAL PLUS before programming begins. The following working method is recommended.

- Determine the machining operations and order to machine the part.
- The tools used to machine the part must be available in the tool list.
- Select and activate the cycle program which is to contain the cycles.
- Determine and set the start point for the part.
- Select and program the cycles required to machine the part.
- Execute each cycle as it is programmed.

8.4 Blocks and Block Numbers

Cycles are placed in blocks in the cycle program. Each block is a complete machining action. Blocks begin with the letter N followed by a number.

Block numbers do not have to follow in order, they are only for identification of the block. The following example would have been created by deleting and adding cycles, but will execute perfectly correctly. Cycle execution is in top to bottom sequence, NOT in block number order.

N9001 (Cycle Program Name)

N3 T cycle 1
N4 T cycle 2
N2 T cycle 3
N5 T cycle 4

Once the cycle program is complete it is possible with the F5 RENUMBER PROGRAM key to renumber all the blocks so they are in sequence. Note, blocks stay in the same location in the cycle program, they are only renumbered.

8.5 Cycle Number and Name

Each cycle program can contain up to 99 cycles. Cycle programs may contain enough cycles to machine a complete work piece or only part of the machining operations. You must therefore be able to identify each cycle program, as different cycle programs will contain different machining operations. The MANUAL PLUS has a fixed identification convention for cycle programs.

The MANUAL PLUS is supplied with up to 24 prepared empty cycles programs.

Each cycle program is given a number starting at N9001 (to N9024). Cycle programs can be stored on external devices and sent to the MANUAL PLUS via the communications options in the CONTROL screen. This is described in Chapter 10 Communication.

8.6 Making a Cycle Program

Cycle Programs are produced by selecting an existing cycle program and adding cycles (see Adding a Cycle) to the program.

Start by looking carefully at the work piece to determine the order of machining operations. Break the contour and work piece features into single movements and machining actions. Once the method of machining the work piece has been determined start by selecting the cycle program. Use the same procedures to determine the machining order and operations that you would use to machine a part manually

8.6.1 Selecting a Cycle Program

To produce a cycle program you must first select one of the prepared empty cycle programs. (An existing cycle program may be modified when the parts are similar). Figure 8.1 shows how to select a cycle program.

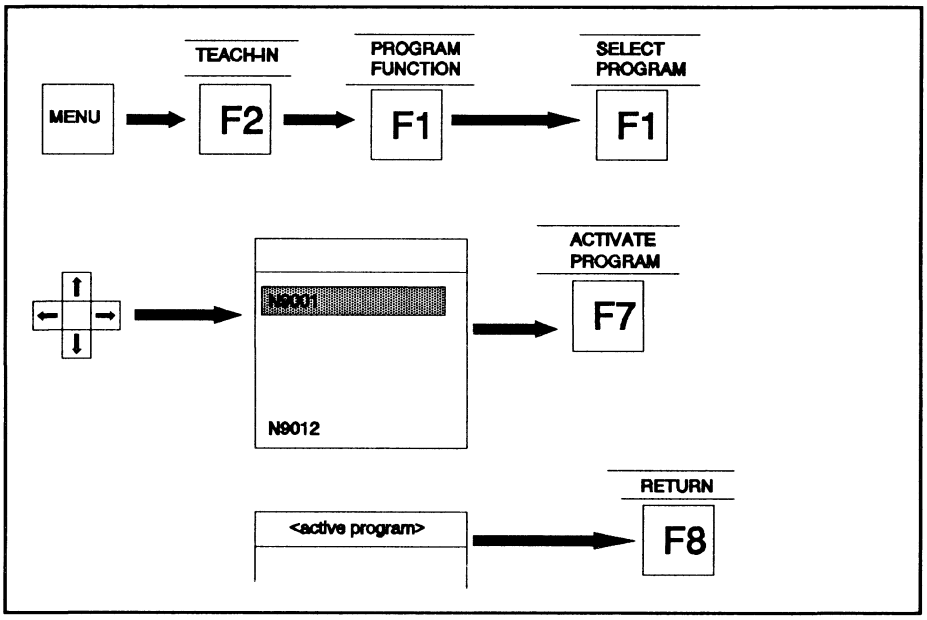


Figure 8.1: Selecting a Cycle Program

Numbering cycle programs does not give any indication as to what the cycle program does. Therefore, it is also possible to add a text description in the identification block. Follow the sequence given in Figure 8.2 below to add or change the cycle program name.

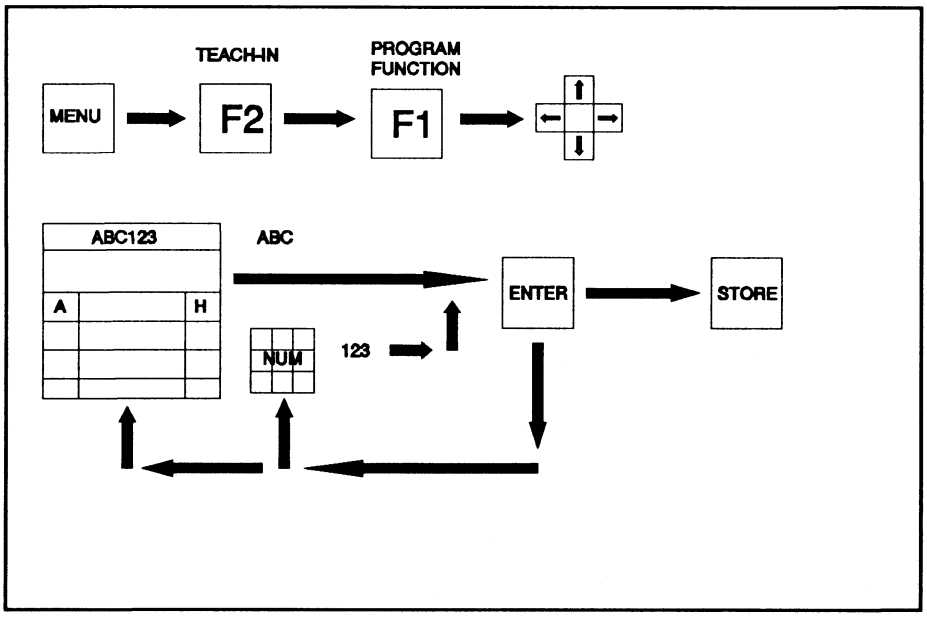


Figure 8.2: Adding a Cycle Name

8.6.2 Cycle Start point

The tool tip must be positioned to the required starting point of each cycle to produce the work piece profile. The first positioning movement is from the tool home position to the start position of the first cycle. This movement can be made with the handwheels or jog controls. It may also be necessary to position the tool tip between cycles to set new cycle start points. These start positions are then taken over by the cycle and used for every subsequent execution of the cycle program. Alternatively, when an exact start position is known, program a positioning cycle. At the end of the cycle program the axes are returned to their original tool home positions.

Caution:

The tool tip moves in a straight line to the start point at the beginning of the next cycle. Be careful when moving between cycles, such as facing and turning. It may require intermediate axes movements to position the tool at the start of the next cycle safely. Intermediate positioning movements should be stored as positioning cycles in the cycle program.

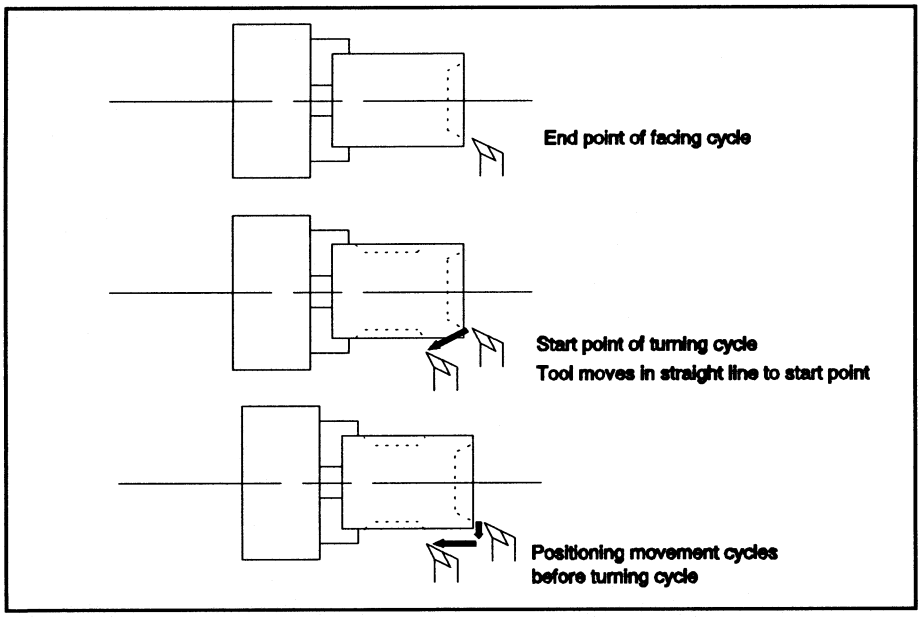


Figure 8.3: Positioning Movement Cycles

8.6.3 Adding a Cycle

Build up cycle programs by adding new cycles into the currently active cycle program.

First you must execute cycles as you define them by pressing the START push button before pressing F6 STORE. This executes the cycle before it is placed in the cycle program. By doing this each machining action is verified during cycle program generation.

Once the cycle has completed execution it can be placed in the cycle program by pressing F6 STORE, or if found to be incorrect modified.

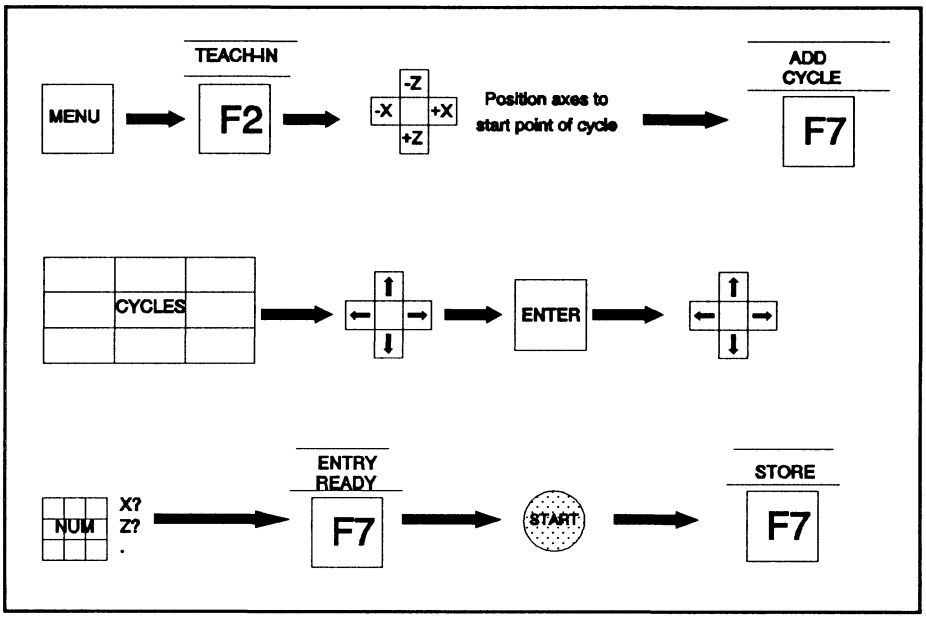


Figure 8.4: Adding a Cycle to the Cycle Program

8.6.4 Changing a Cycle

Change the contents of a cycle, either because it was entered incorrectly or to modify the cycle. This is particularly useful when similar parts are being machined with only minor changes. A new cycle program can be made from an existing one. Pressing F7 OVERWRITE replaces the contents of the existing cycle without having to execute the cycle first.

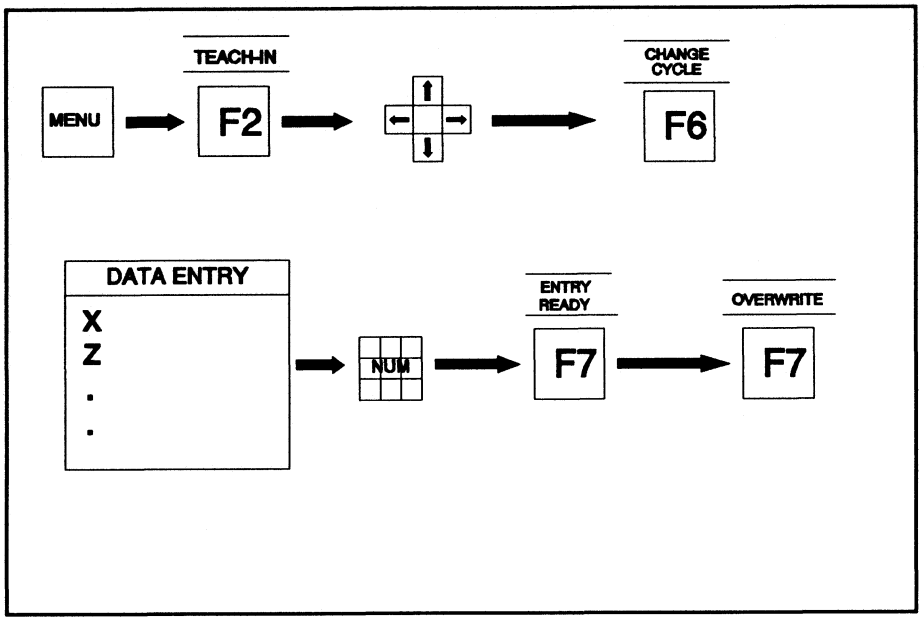


Figure 8.5: Changing the Contents of a Cycle

8.6.5 Deleting a Cycle

To delete a cycle from the cycle program follow the sequence in Figure 8.6 below. Deleting a cycle removes the cycle name and block number from the cycle program.

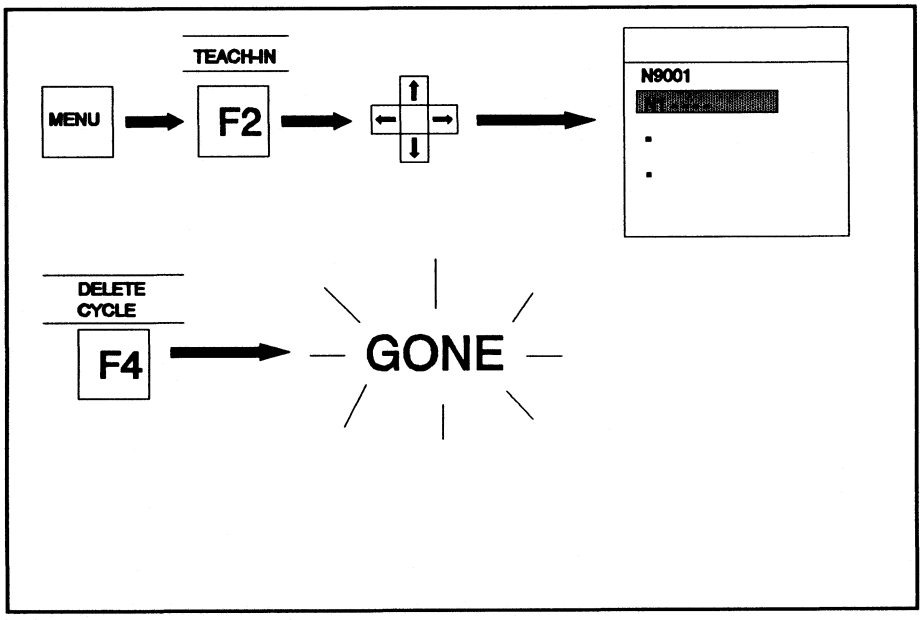


Figure 8.7: Deleting a Cycle from the Cycle Program

8.6.6 Clearing a Cycle Program

It is possible to clear the entire contents of the cycle program. The action of clearing a cycle program, deletes all information within the cycle program and also clears the program name. Follow the sequence in Figure 8.7 below to clear a cycle program

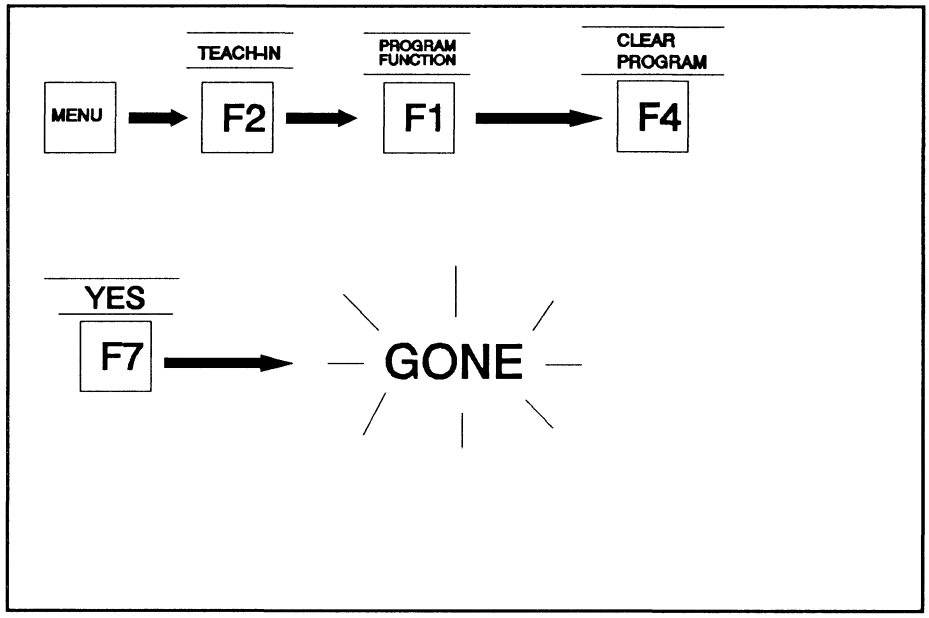


Figure 8.8: Clearing a Cycle Program.

8.7 Finishing a Contour

A contour can be composed by using cycles.

- Rapid to startpoint and retract:
  Rapid (G804)

- Single cut cycles for contour:
  Linear radial/axial (G800)
  Linear angle (G801)
  Circular clockwise (G802)
  Circular counter clockwise (G803)

- Finishing cycles for contour:
  Chamfer/Rounding (G815)
  Finishing axial (E) (G826)
  Finishing radial (E) (G827)
  Undercut DIN 76 (G850)
  Undercut DIN 509E (G851)

Tapered and circular movements need a correction for the toolnose radius. The finishing cycles have this correction and are used by preference.

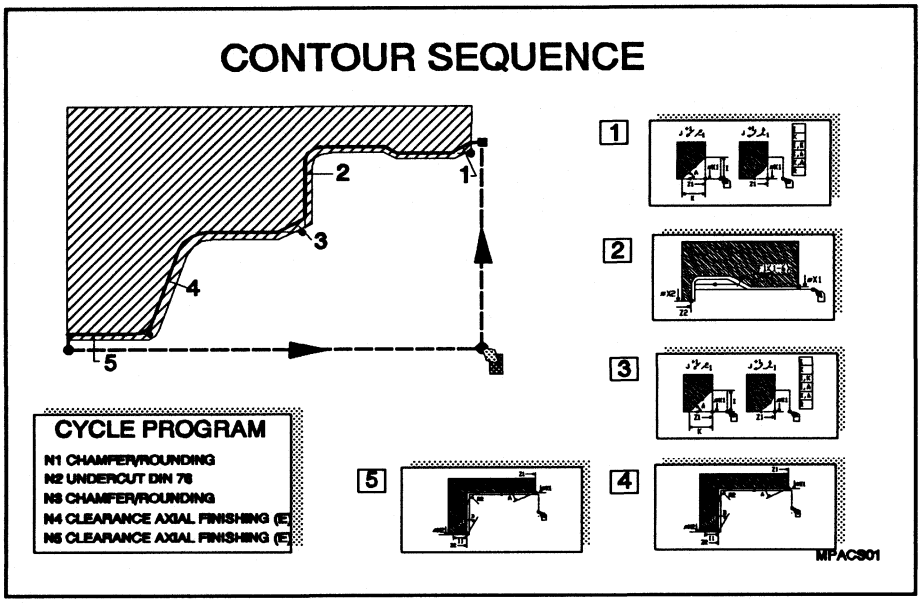


Figure 8.6: Contour finishing

9. CYCLE PROGRAM EXECUTION

9.1 Introduction

This chapter describes how to execute a cycle program. Function key F6 PROGRAM EXECUTION in the main MANUAL OPERATE screen enters the PROGRAM EXECUTION mode.

9.2 Work Method

The following working method is recommended when executing cycle programs.

- Select and activate the cycle program.
- Prepare and measure tools required.
- Set work piece zero point.
- Execute Cycle Program

9.3 Select and Activate Cycle Program

Before a cycle program can be executed you must first make it the active program. Follow the sequence in Figure 9.1 below to select and activate a cycle program.

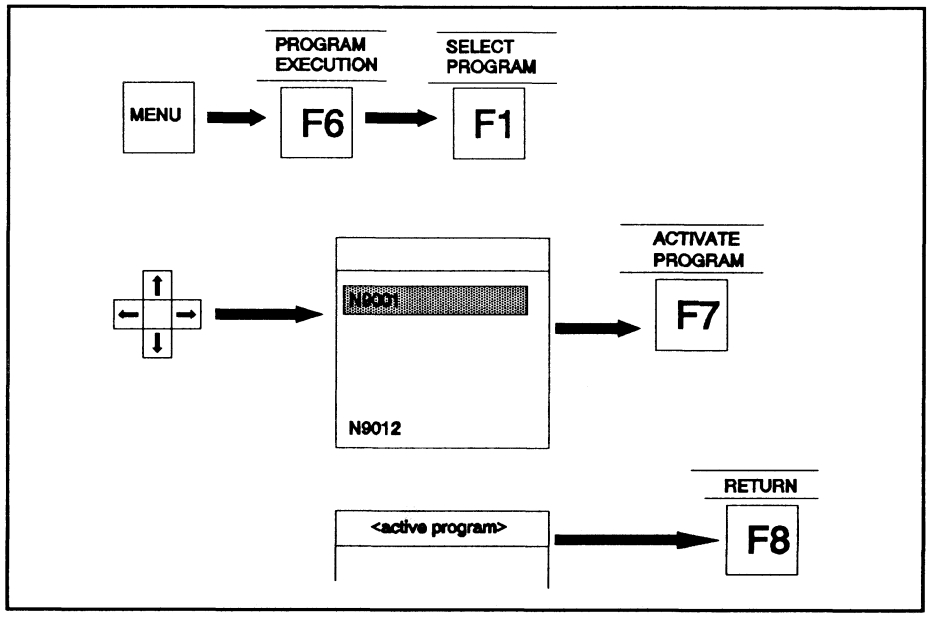


Figure 9.1: Selecting and Activating a Cycle Program

9.4 Tool Requirements

All the tools which are used to machine the part must be available in the tool table and measured. See Chapter 6 for details of measuring tools and tool table.

9.5 Setting Work Piece Zero Point

The sequence of positioning cycles contained in a cycle program are relative to the work piece zero point. To reproduce the same results observed during Teach-in, this position should be set to the original point given in Teach-in. To set the work piece zero point follow the procedure in Figure 9.2 below.

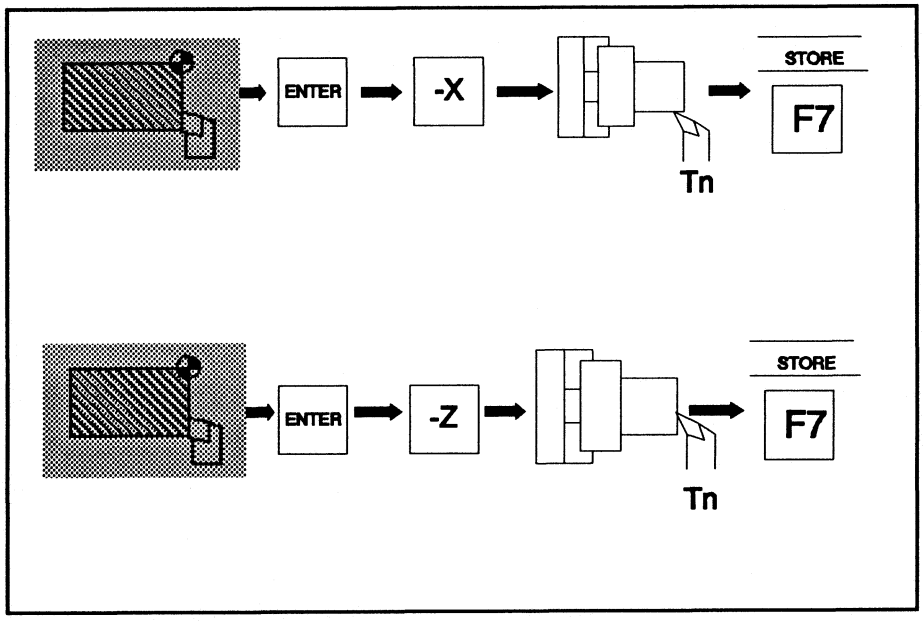


Figure 9.2: Setting the Cycle Program Start Point

9.6 Program Execution

To execute a cycle program follow the sequence in Figure 9.3 below.

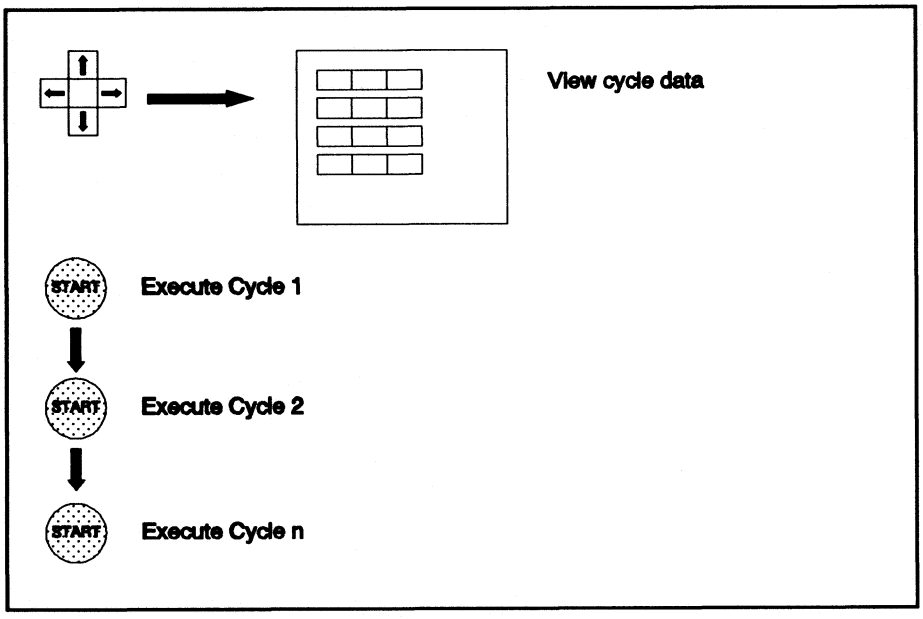


Figure 9.3: Executing a Cycle Program

9.7 Automatic Sequence (option)

This function key is active when machining cycle in program execution. Normally the MANUAL PLUS stops after each cycle. When the Automatic Sequence function is enabled, all subsequent cycles are executed without the necessity to press Start.
After execution of the last cycle, the cursor returns to the beginning of the program.

Note: remove any S or T commands in the automatic sequence to improve the contour smoothness.

9.8 Program Begin

Pressing this places the cursor at the beginning of the program.

10. COMMUNICATION

10.1 Introduction

This chapter describes the methods available for transferring information to and from the MANUAL PLUS. This enables cycle programs and tool tables to be stored externally from the MANUAL PLUS, extending the number of prepared cycle programs available. Additional cycle programs can be stored on external devices such as the MINI-PC option or a standard PC micro computer.

The procedures and sequences given in this chapter assume that the MANUAL PLUS has been correctly set-up for the external devices connected. See Appendix diagnostic (Operator Machine Constants) for more information of communication parameters.

The communication menu is contained in the CONTROL screen.

10.2 Sending

Sending data passes the contents of the selected memory to an external device for storage. To transfer data you must display the CONTROL screen. The sequence given in Figure 10.1 below transfers just the cycle program to the external device.

- To transfer the contents of individual memories, highlight the memory and follow the sequence in Figure 10.1 below. Memories which can be transferred are:

  - Main Program
  - Tool Memory
  - Constants Memory
  - Tool Type

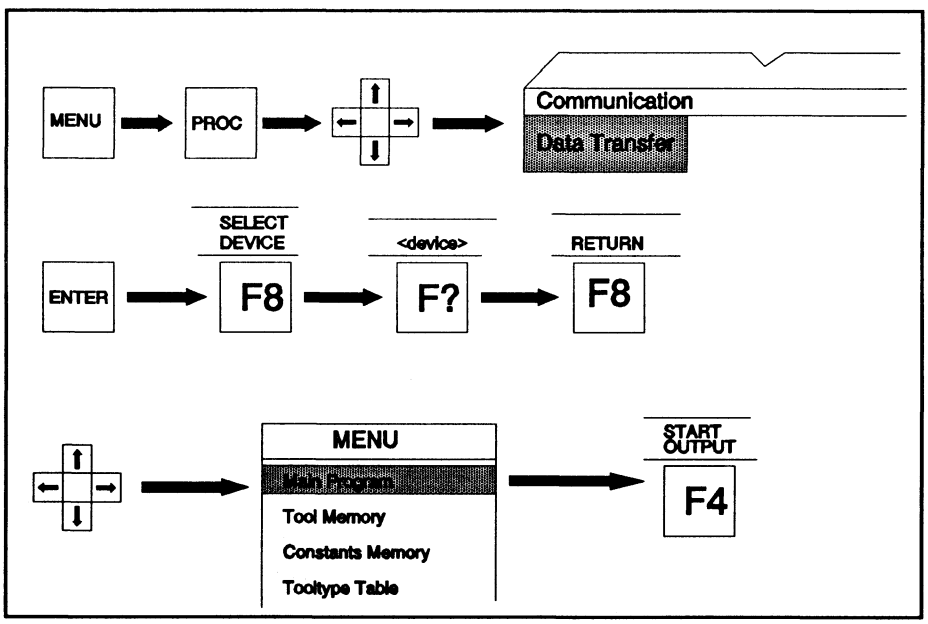


Figure 10.1: Sending Memories

10.3 Receiving

Receiving data passes memory data stored in an external device to the MANUAL PLUS. To transfer data to the MANUAL PLUS follow the sequence in Figure 10.2 below. The sequence given in Figure 10.2 below transfers the cycle programs to the MANUAL PLUS.

- To transfer the contents of individual memories, highlight the memory and follow the sequence in Figure 10.2 below. Memories which can be transferred are:

  - Main Program
  - Tool Memory
  - Constants Memory
  - Tool Type

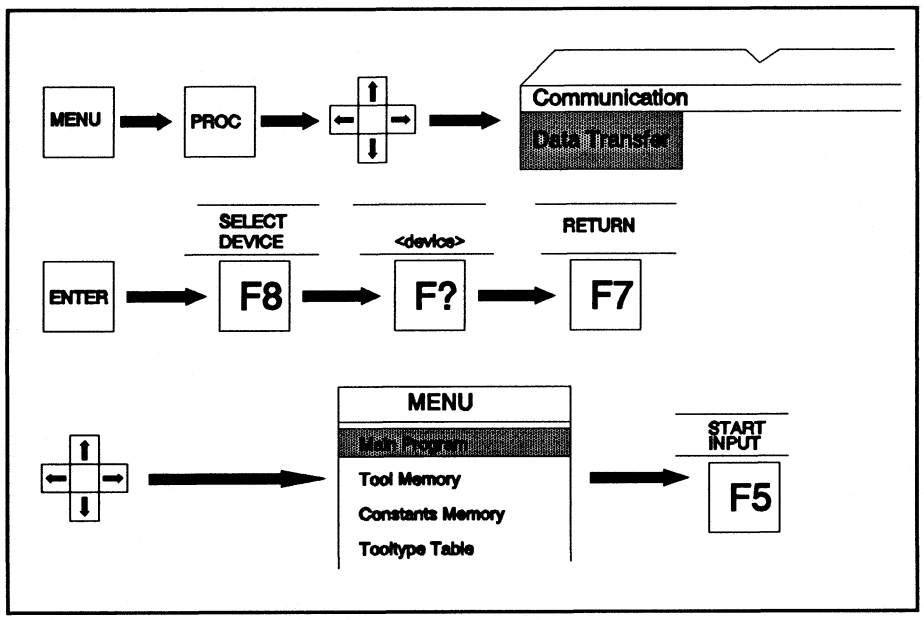


Figure 10.2: Receiving Memories

Note:

Only programs that were generated on the MANUAL PLUS can be loaded into the Main Program memory.

An active program cannot be overwritten by a new program with the same program number. Press CLEAR CONTROL to deactivate the current program (or make a different program the active program) allowing the program to be overwritten.

11. EXAMPLE

11.1 Introduction

This chapter contains an example work piece and a solution on how to make the cycles to machine the part.

Tools which are used and which are entered in the toolmemory are:
- T1 roughing
- T2 finishing

Work flow:
- clamp material in the chuck
- set zeropoint with preset axes
  (Z0 on the face of the material)
- measure tools T1 and T2
- change speed to m/min for constant cutting speed
- start producing the work piece, remove material cycle-by-cycle.

11.2 Example Work Piece

Figure 11.1 below shows the example work piece.

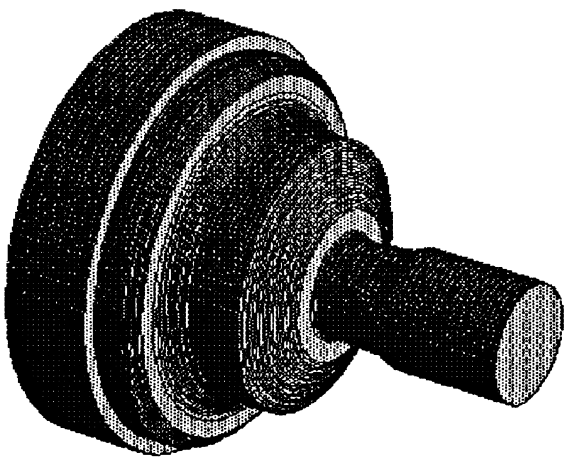


Figure 11.1: Example Workpiece

Workpiece dimensions to be machined

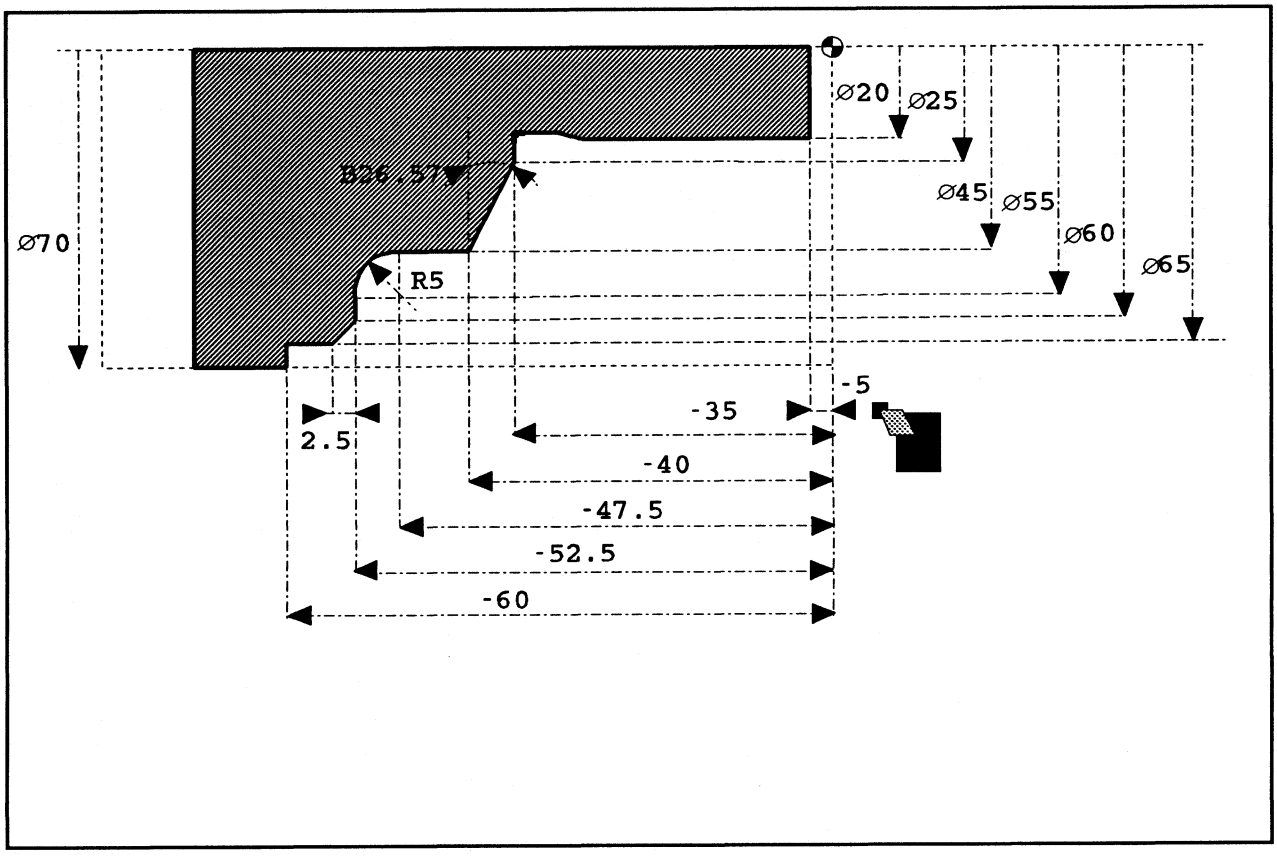


Figure 11.1: Workpiece to be machined

Volume to be machined

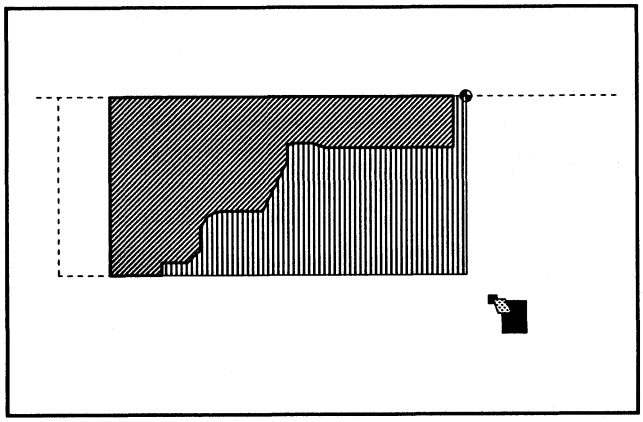
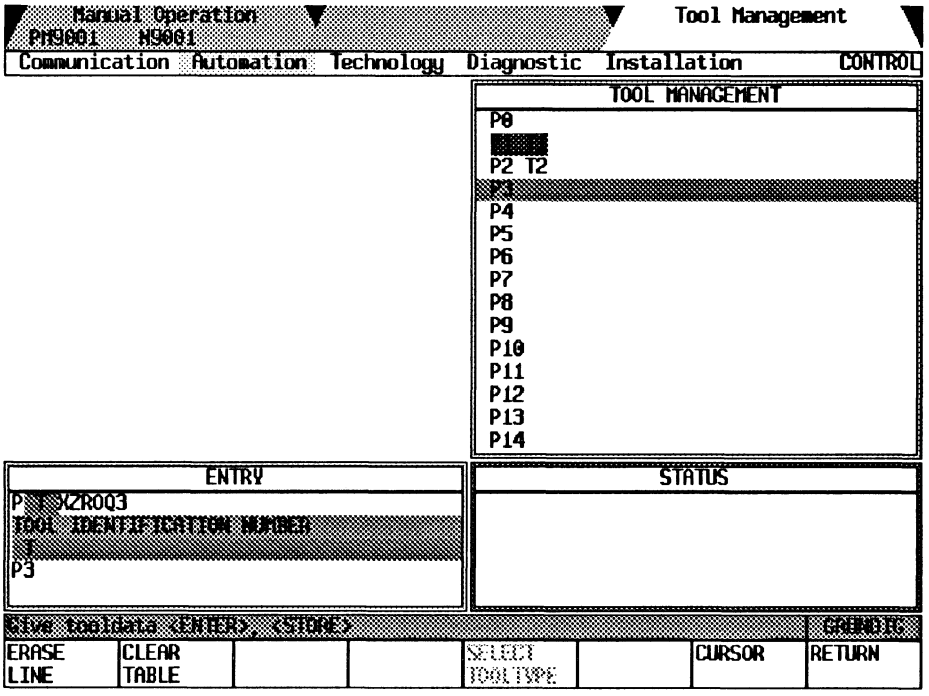


Figure 11.3: Machine volume

To begin the used tools (T1 for roughing and T2 for finishing) must be entered in the toolmemory (and measured if needed).



This workpiece will be made in Teach-In, so the programmed cycles are stored in a program. First step is to activate the Teach-In mode and added cycles in an empty program (i.e 9001):

TEACH-IN:

							GRUNDIG
	TEACH-IN				PROGRAM EXECUTION	CLEAR CONTROL	

ADD CYCLE:

							GRUNDIG
PROGRAM FUNCTIONS			DELETE CYCLE		CHANGE CYCLE	ADD CYCLE	RETURN

After entering all the parameters of the cycle always the following sequence must be followed:

ENTRY READY:

							GRUNDIG
				TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

START:

STORE:

							GRUNDIG
TOOL CORRECTION						STORE	RETURN

For the finishing allowance we take 0.5 mm for the X-size (= 1 mm on the diameter) and 0.1 mm for the Z-size.

On the following pages the programming steps are explaned.

Roughing the contour with T1

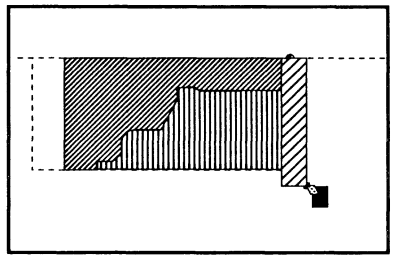
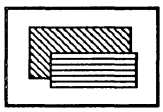


figure 73 Clearance radial

1. Clearance radial basic (Facing)

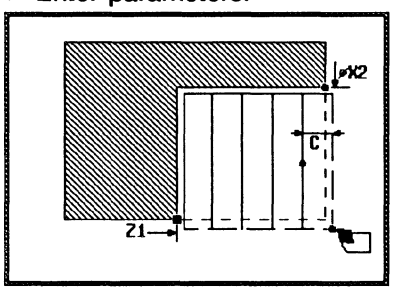
> Menu selection:

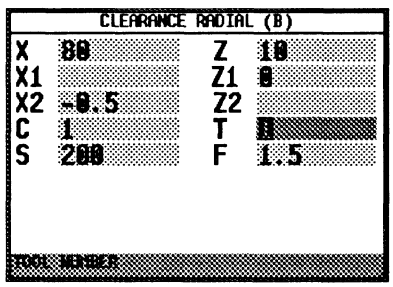


> Select RADIAL and BASIC (not EXTENDED):

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

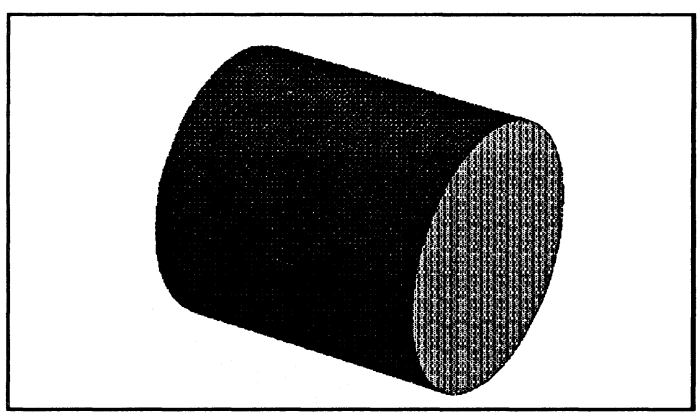
> Enter parameters:





> ENTRY READY, START, STORE (see above)

Result:



2. Clearance axial basic (Cylinder)

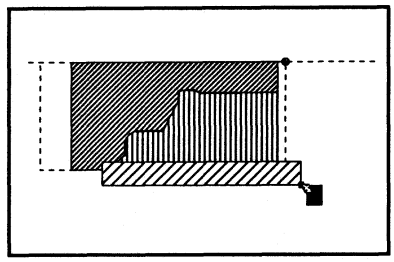
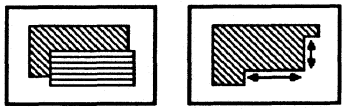


figure 77 Clearance axial

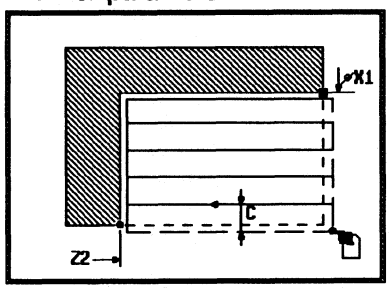
> Menu selection:

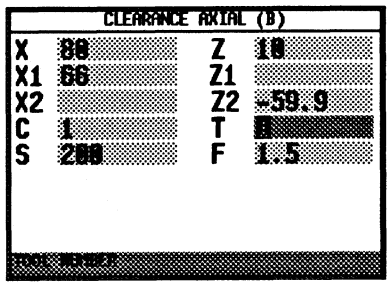


> Select AXIAL (not RADIAL) and BASIC (not EXTENDED):

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

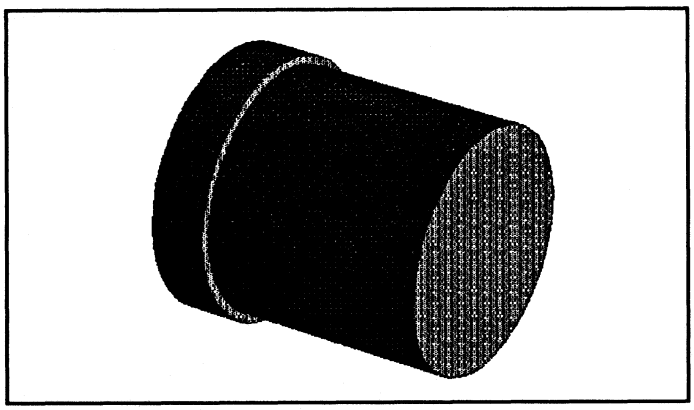
> Enter parameters:





> ENTRY READY, START, STORE (see above)

Result:



3. Clearance axial extended (cylinder with radius and fase)

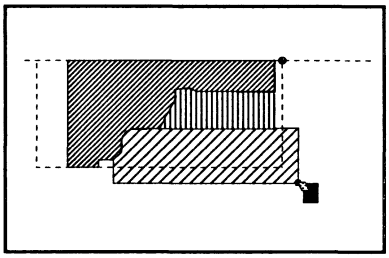
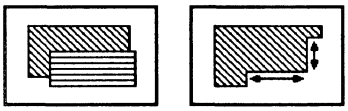


figure 81 Clearance axial

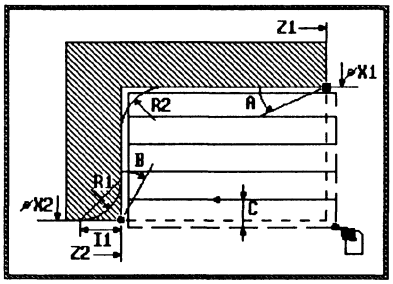
> Menu selection:



> Select AXIAL (not RADIAL) and EXTENDED:

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

> Enter parameters:



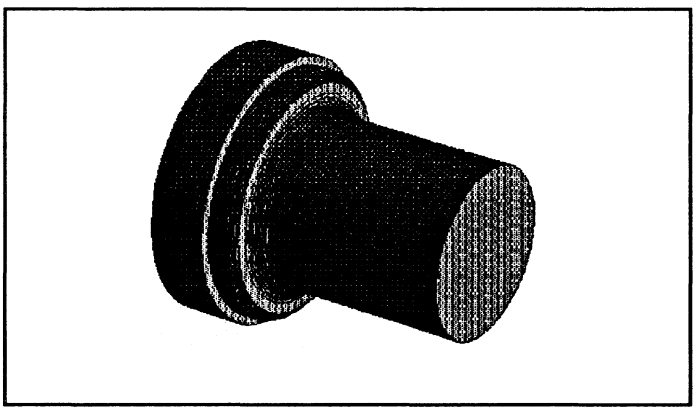
CLEARANCE AXIAL (E)

X	66	Z	5
X1	46	Z1	-39.9
X2	66	Z2	-52.4
C	1	A	
B		I1	2.5
R1		R2	5
T	1	S	200
F	1.5		

TOOL NUMBER

> ENTRY READY, START, STORE (see above)

Result:



4. Clearance axial extended (cylinder with fase)

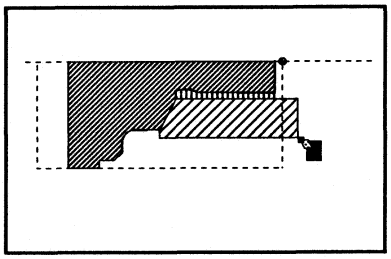
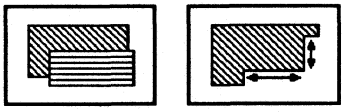


figure 85 Clearance axial

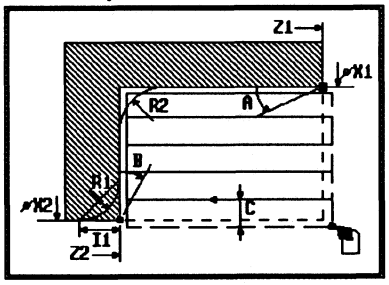
> Menu selection:



> Select AXIAL (not RADIAL) and EXTENDED:

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

> Enter parameters:

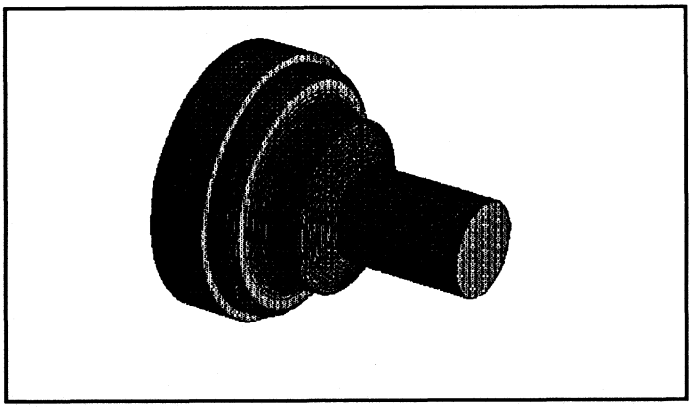


CLEARANCE AXIAL (E)

X	46	Z	5
X1	26	Z1	-34.9
X2	46	Z2	-39.9
C	1	A	
B	26.57	I1	
R1		R2	
T	1	S	200
F	1.5		

> ENTRY READY, START, STORE (see above)

Result:



5. Clearance axial basic (cylinder)

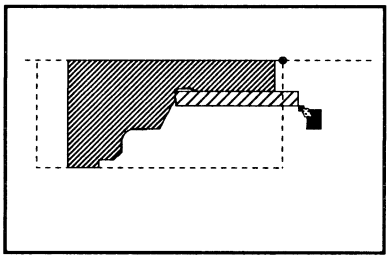
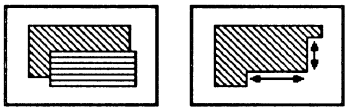


figure 89 Clearance axial

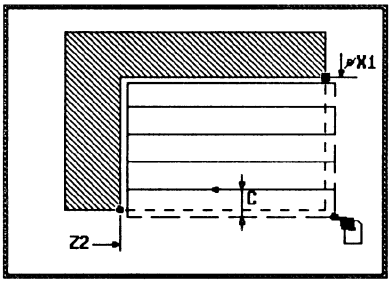
> Menu selection:

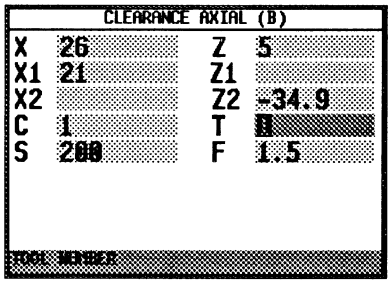


> Select AXIAL (not RADIAL) and BASIC (not EXTENDED):

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

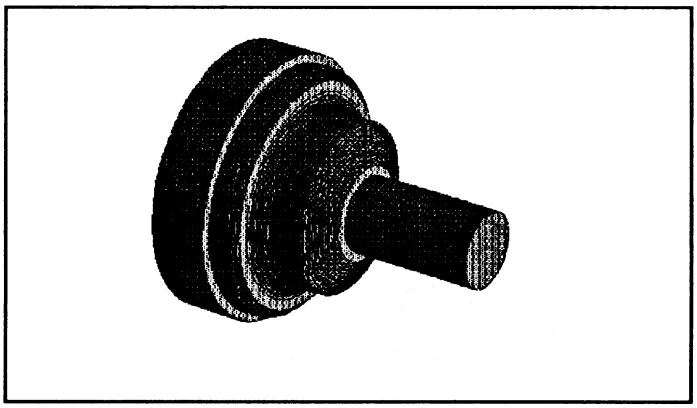
> Enter parameters:





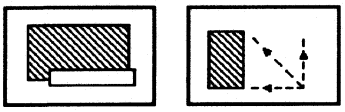
> ENTRY READY, START, STORE (see above)

Result:

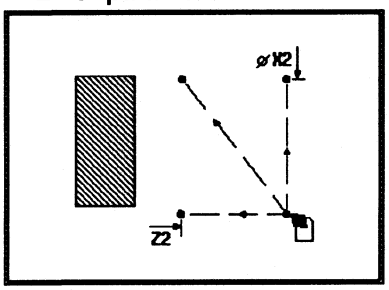


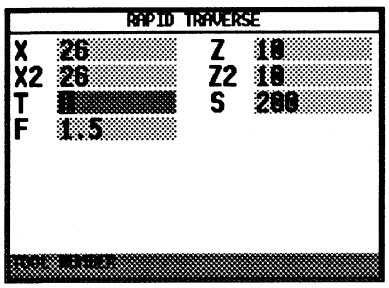
6. Rapid traverse for toolchange (roughing tool for finishing tool)

> Menu selection:



> Enter parameters:

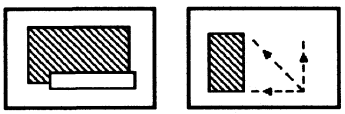




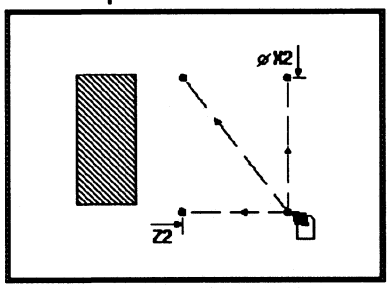
> ENTRY READY, START, STORE (see above)

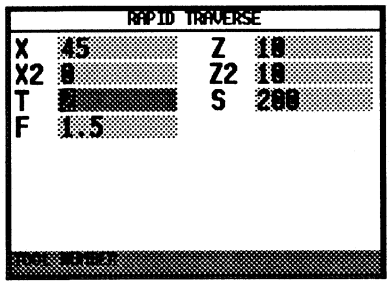
7. Rapid traverse to workpiece with T2

> Menu selection:



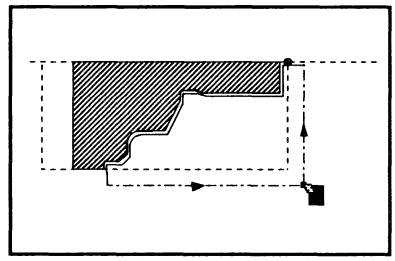
> Enter parameters:





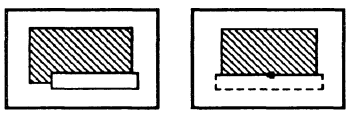
> ENTRY READY, START, STORE (see above)

Finishing contour with T2

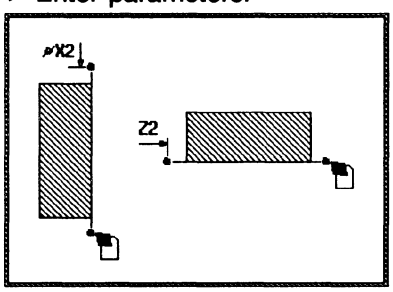


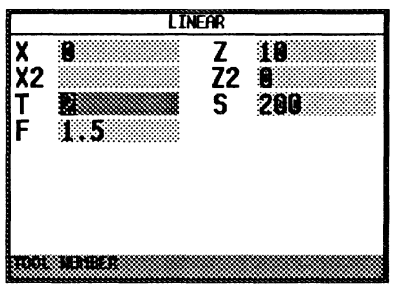
8. Linear horizontal

> Menu selection:



> Enter parameters:

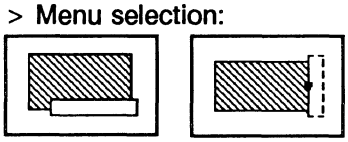




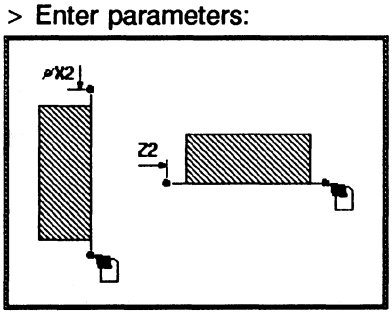
> ENTRY READY, START, STORE (see above)

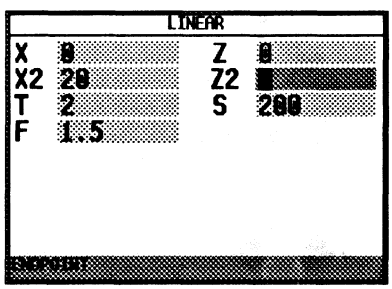
9. Linear vertical

> Menu selection:



> Enter parameters:

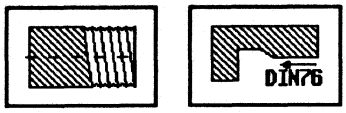




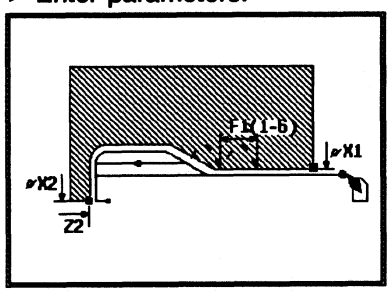
> ENTRY READY, START, STORE (see above)

10. Undercut DIN76

> Menu selection:



> Enter parameters:



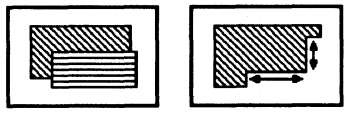
UNDERCUT DIN 76

X	20	Z	0
X1	20	Z1	
X2	25	Z2	-35
F1	2	T	2
S	200	F	1.5

> ENTRY READY, START, STORE (see above)

11. Clearance axial finishing extended

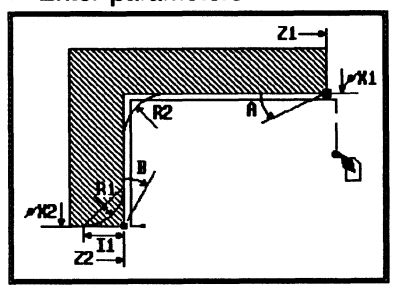
> Menu selection:



> Select Axial (not Radial) and Extended and Finishing:

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

> Enter parameters:



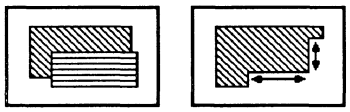
CLEARANCE AXIAL FINISHING (E)

X	25	Z	-35
X1	25	Z1	-35
X2	45	Z2	-40
A		B	26.57
I1	0	R1	
R2		T	2
S	200	F	1.5

> ENTRY READY, START, STORE (see above)

12. Clearance axial finishing extended

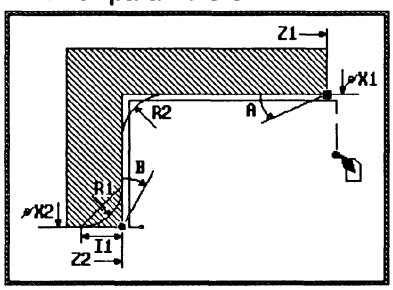
> Menu selection:



> Select Axial (not Radial) and Extended and Finishing:

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

> Enter parameters:



CLEARANCE AXIAL FINISHING (E)

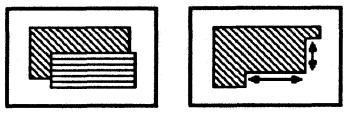
X	45	Z	-40
X1	45	Z1	-40
X2	65	Z2	-52.5
A		B	
I1	2.5	R1	
R2	5	T	2
S	200	F	1.5

TOOL NUMBER

> ENTRY READY, START, STORE (see above)

13. Area clearance axial finishing basic

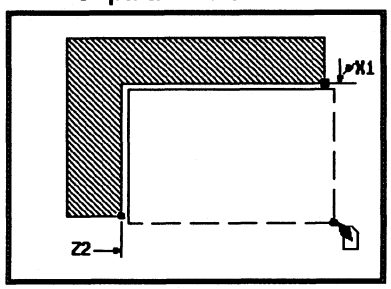
> Menu selection:

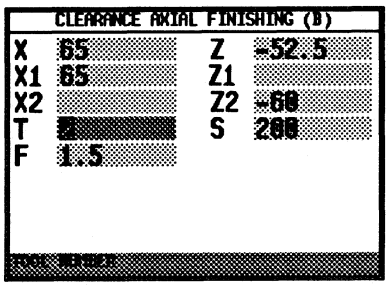


> Select Axial (not Radial) and Basic (not Extended) and Finishing:

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN

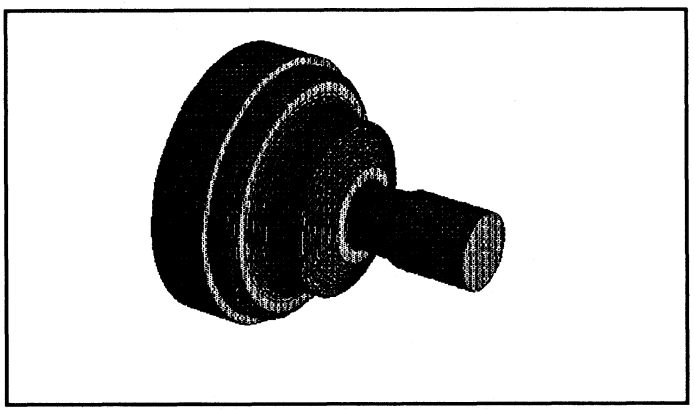
> Enter parameters:





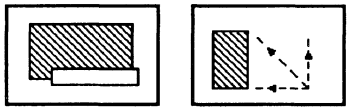
> ENTRY READY, START, STORE (see above)

Result:

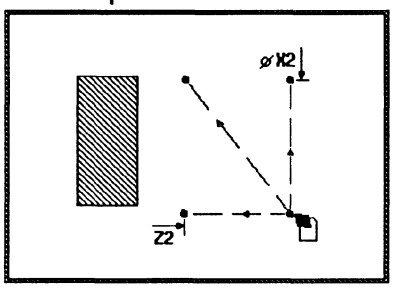


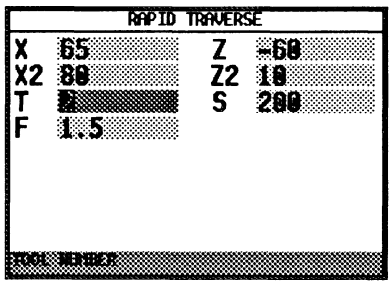
14. Rapid traverse
(Move tool to a save place)

> Menu selection:



> Enter parameters:





> ENTRY READY, START, STORE (see above)

Cycle list

N9001 G899 (DEMO PRODUCT /93)
N1 T1 CLEARANCE RADIAL (B)
N2 T1 CLEARANCE AXIAL (B)
N3 T1 CLEARANCE AXIAL (E)
N4 T1 CLEARANCE AXIAL (E)
N5 T1 CLEARANCE AXIAL (B)
N6 T1 RAPID TRAVERSE
N7 T2 RAPID TRAVERSE
N8 T2 LINEAR
N9 T2 LINEAR
N10 T2 UNDERCUT DIN76
N11 T2 CLEARANCE AXIAL FINISHING (E)
N12 T2 CLEARANCE AXIAL FINISHING (E)
N13 T2 CLEARANCE AXIAL FINISHING (B)
N14 T2 RAPID TRAVERSE

Program

```
N9001 G899 (DEMO PRODUCT /93)
N1 G821 X80 Z10 C1 F1.5 S200 T1 H96 X2=-0.5 Z1=0
N2 G820 X80 Z10 C1 F1.5 S200 T1 H96 X1=66 Z2=-59.9
N3 G822 X66 Z5 C1 F1.5 S200 T1 H96 X1=46 X2=66 Z1=-39.9 Z2=-52.4 I1=2.5 R2=5
N4 G822 X46 Z5 B26.57 C1 F1.5 S200 T1 H96 X1=26 X2=46 Z1=-34.9 Z2=-39.9
N5 G820 X26 Z5 C1 F1.5 S200 T1 H96 X1=21 Z2=-34.9
N6 G804 X26 Z10 F1.5 S200 T1 H96 X2=26 Z2=10
N7 G804 X45 Z10 F1.5 S200 T2 H96 X2=0 Z2=10
N8 G800 X0 Z10 F1.5 S200 T2 H96 Z2=0
N9 G800 X0 Z0 F1.5 S200 T2 H96 X2=20
N10 G850 X20 Z0 F1.5 S200 T2 H96 X1=20 X2=25 Z2=-35 F1=2
N11 G826 X25 Z-35 B26.57 F1.5 S200 T2 H96 X1=25 X2=45 Z1=-35 Z2=-40
N12 G826 X45 Z-40 F1.5 S200 T2 H96 X1=45 X2=65 Z1=-40 Z2=-52.5 I1=2.5 R2=5
N13 G824 X65 Z-52.5 F1.5 S200 T2 H96 X1=65 Z2=-60
N14 G804 X65 Z-60 F1.5 S200 T2 H96 X2=80 Z2=10
```

Contour roughing: N1 to N5
Contour finish : N8 to N14

Tools Used:
- T1 roughing
- T2 finishing

Work flow:
- clamp material in the chuck
- set zeropoint with preset axes
  (Z0 on the face of the material)
- measure tools T1 and T2
- change speed to m/min for constant cutting speed
- start producing the work piece, remove material cycle-by-cycle.

12. Remarks

For the MANUAL PLUS V264 and V350

- Undercut cycles cannot be executed with T0 set as the active tool.
- After a cycle stop or after graphic simulation the handwheels are disabled. Press the CLEAR button to activate the handwheels.
- After an external error E.. the CLEAR button must be pressed twice in order to re-activate the handwheels. When CLEAR is only pressed once, the error disappears but will be generated immediately when the operator starts turning the handwheel.
- Movements may be incorrect when programmed with unrealistic parameters (for example: a radius too large to fit the generated movement).
- When the programmed spindle speed is too high for the selected spindle gear the manual plus will generate an alarm P31.
  If you clear the alarm and switch the spindle on via the machine control panel, the spindle will be started and run with the maximum spindle speed for the selected gear range.
- Cycles for groove turning extended G832/833 have the following limitations:
  The radius R1 is only available in the endpoint of the groove. The startpoint never has a radius.
  In graphic simulation, the programmed contour is not shown. Only the cutterpath is shown.
- Cycles for groove turning axial extended G833 and G837 have the following limitations:
  When using radius R2 in the bottom, the groove can only have half of its maximum depth resulting from angles A and B. Deeper grooves give alarm P07.
  Deep grooves have to be cut with a tool with a large radius. Do not use R2.
  G832/G833 have no recalculation to equal depth of cut for all cuts. The last cut can be smaller than the previous cuts.
- After entry of a non-existing Tool T together with a Speed S in the F,S,T-menu an error P150 (TOOL NOT FOUND) will occur and the T and S are not activated. After solving error P150 and re-entering the T- and S-word the S-word will not be activated.
  This problem can be avoided by programming a slightly different S-word, i.e. S1001 i.s.o. S1000.
- During a graphic testrun in Teach-In mode, the dimensions of the actual tool are used i.s.o. of the dimensions of a programmed tool. This will result in a (slightly) wrong drawing of the contour if the tool nose radius of these tools is different.
- Grooving cycles (G840-G847) do not show the programmed contour in a graphic testrun. Only the cutter path is shown.
- The radii R1 and R2 and the chamfer I1 in the extended roughing and finishing cycles (G820-827, G830-837) must fit in the contour.
  The automatic radius compensation will increase the chamfer I1, the radius R1 and the radius R2 when it is on an outside corner. These enlarged sizes must also fit in the contour. If possible, change the startpoint or endpoint when I1,R1 or R2 does not fit (alarm P07). When the startpoint or endpoint cannot be changed, do the rouhing without radius compensation (tool radius R0) or with a smaller I1,R1 or R2. The contour will have a small error and should be finished using the Single Cut cycles (G801-804) with Return. These cycles can produce part of a radius.

Remarks

13. Appendix: Errors

This section describes operating and programming errors which may occur during normal operation of the MANUAL PLUS. If errors are displayed which are not given here call the appropriate service personnel.

O 134 VALUE > MAXIMUM
Programmed value for an address is greater than the specified maximum value. For example, entering: X101, X=101, X=100+1, when the maximum allowed value is 100.
Solution: enter the correct value

O 135 VALUE < MINIMUM
Programmed value for an address is smaller then the specified minimum value (R1= 0 or R2= 0).
Solution: enter the correct value

P 001 ILLEGAL WORD IN A BLOCK
In a block a word is programmed that cannot be used in combination with other programmed addresses. Go to the cycle you have programmed, in Appendix available cycles, to find a description of the error. The following cycles may generate a P01 error.

P 002 NECESSARY WORD MISSING FROM BLOCK
In the required block a word is missing that must be used in combination with other programmed addresses. Go to the cycle you have programmed, in Appendix available cycles, to find a description of the error. All cycles generate a P02 error.

P 007 PROGRAMMED DATA OUT OF RANGE
In a block with a circular movement where the programmed radius is more than 2 times the distance between begin and endpoint. Go to the cycle you have programmed, in available cycles, to find a description of the error.

P 020 TOOL RADIUS > PROGRAMMED RADIUS
Tool radius is larger than the radius of the programmed or calculated circle.

P 031 SPINDLE SPEED OUT OF RANGE
The spindle speed (rev/min) is higher than the maximum of the gear range.

P 034 IN BLOCK NUMBER ...
This error can occur together with another programming error. The number displayed is an internal number. It can help service engineers to find the cause of the programming error.

P 049 Orientation of the tool cannot be activated in the cycle (T0 is active)

P 116 Some parameters doesn't fit with each other. Go to the cycle you have programmed, in available cycles, to find a description of the error.
This can also be with respect to the dimension of the used tool

P 190 PROGRAMMED VALUE > MAXIMUM
A programmed value or a calculated intersection point gives too large positive position

P 191 PROGRAMMED VALUE < MINIMUM
A programmed value or a calculated intersection point gives too small negative position

O 201 START NOT ALLOWED
Push function key ENTRY READY before start.

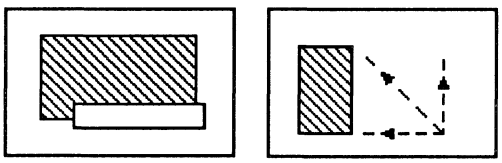
O 203 SPINDLE POSITION NOT KNOWN
"Take over position" in A7 or Z7 of a thread recut cycle is not possible when the spindle position is unknown.

14. : Appendix: Available Cycles

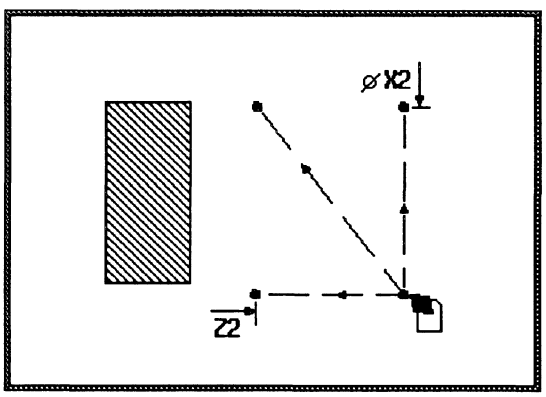
This gives all the machining cycles available in the MANUAL PLUS. Each cycle is given with its parameters for the basic, extended and finish modes.

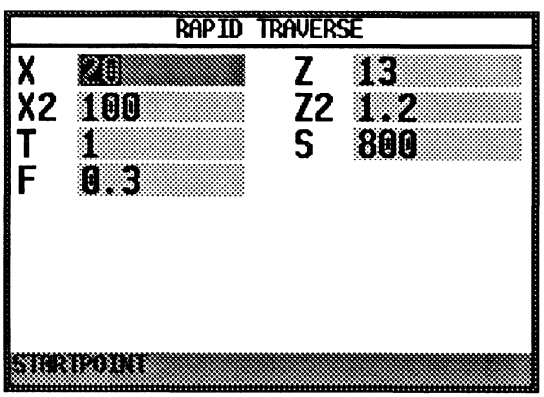
The G-number given to the right hand side of the cycle name is for reference. It is only important if you transfer a cycle program to an external device, such as a personal computer (PC). When cycle programs are displayed on a PC the cycle names are not shown. Instead the G-number and cycle contents are displayed. The G-number is given here to help locate the programmed cycle.

14.1 RAPID TRAVERSE G804



							GRUNDIG
	WITH RETRACT						RETURN





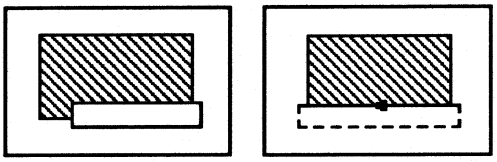
PARAMETERS:

X,Z	Startpoint of the rapid movement	(only Teach-In)
X2,Z2	Endpoint of the rapid movement	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

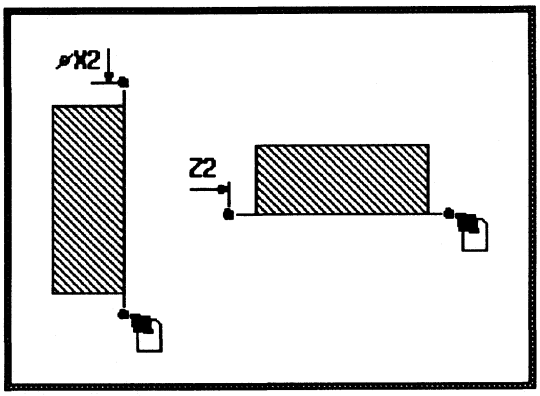
ERRORS:

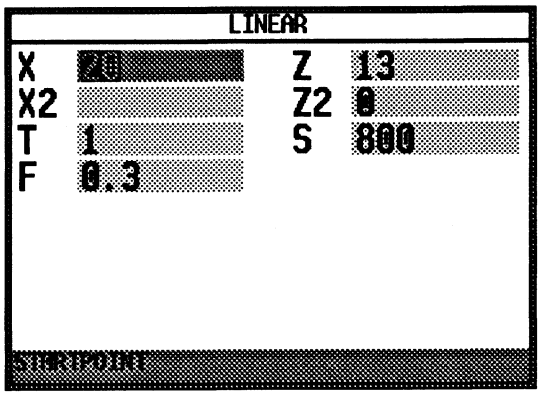
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.2 LINEAR G800



							GRUNDIG
	WITH RETRACT						RETURN





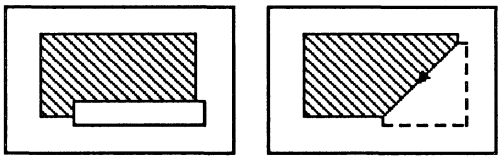
PARAMETERS:

X,Z	Startpoint of the linear movement	(only Teach-In)
X2,Z2	Endpoint of the linear movement	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

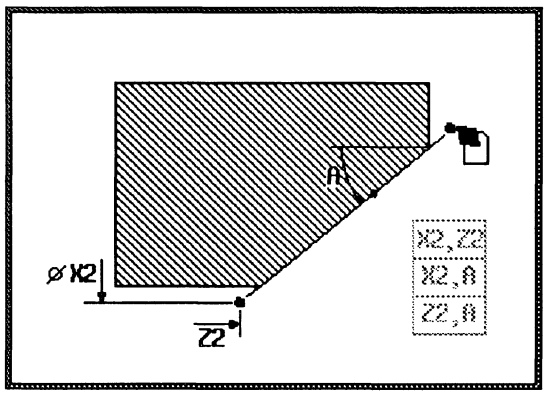
ERRORS:

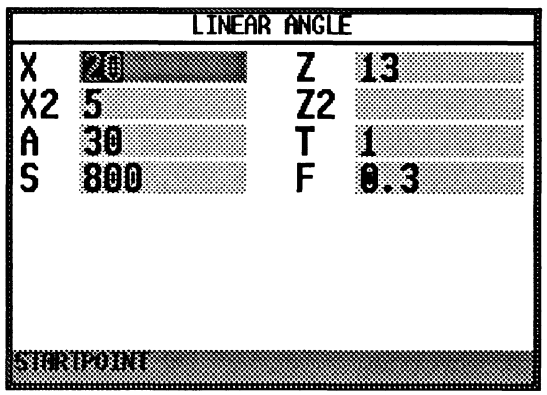
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.3 LINEAR ANGLE G801



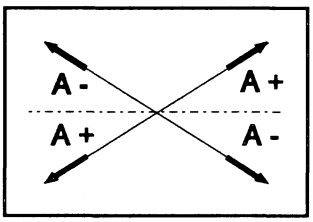
	WITH RETRACT						GRUNDIG RETURN





PARAMETERS:

X,Z	Startpoint of the linear movement	(only Teach-In)
X2,Z2	Endpoint of the linear movement	
A	Angle (min: +/-0.001 degr, max: +/-89.999 degr)	

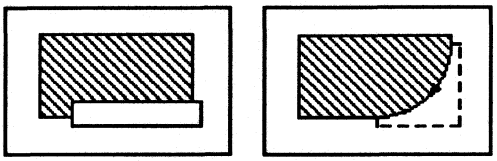


T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

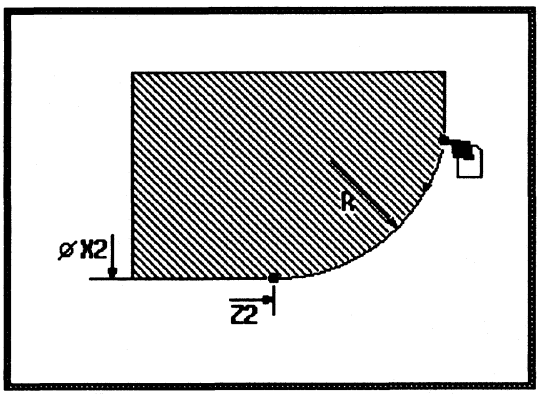
ERRORS:

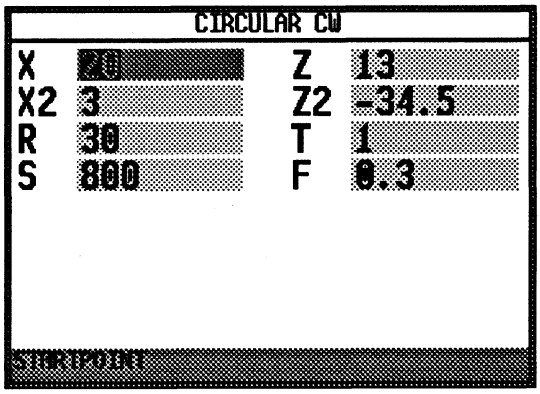
P01	X2,Z2,A programmed together
P02	One or more parameters are not programmed
P07	A0 programmed
P190	Intersection point with line A gives too large positive position
P191	Intersection point with line A gives too small negative position
O135	Value < minimun (i.e A < -90)
O134	Value > maximum (i.e A > +90)

14.4 CIRCULAR CLOCKWISE G802



							GRUNDIG
	WITH RETRACT						RETURN





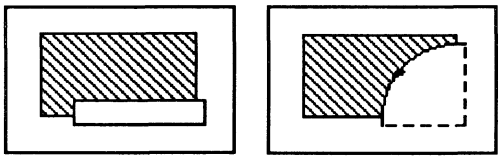
PARAMETERS:

X,Z	Startpoint of the circ. movement	(only Teach-In)
X2,Z2	Endpoint of the circ. movement	
R	Circle radius	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

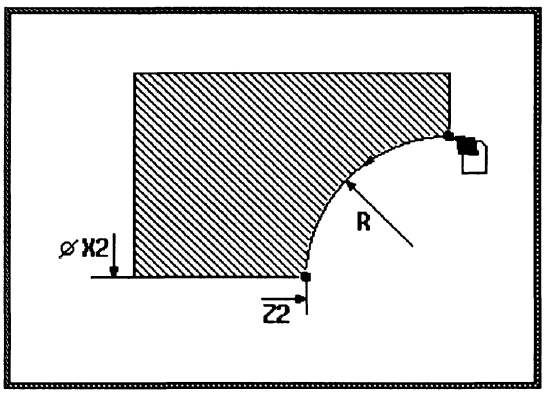
ERRORS:

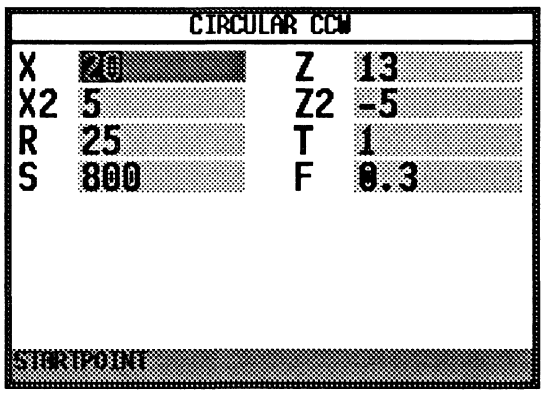
P02	One or more parameters are not programmed
P07	Programmed R does not fit between begin- and endpoint
O135	Value < minimun (i.e R < 0)
O134	Value > maximum (i.e T > 48)

14.5 CIRCULAR COUNTER CLOCKWISE G803



							GRUNDIG
	WITH RETRACT						RETURN





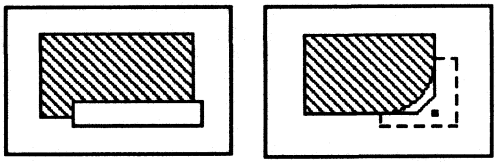
PARAMETERS:

X,Z	Startpoint of the circ. movement	(only Teach-In)
X2,Z2	Endpoint of the circ. movement	
R	Circle radius	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

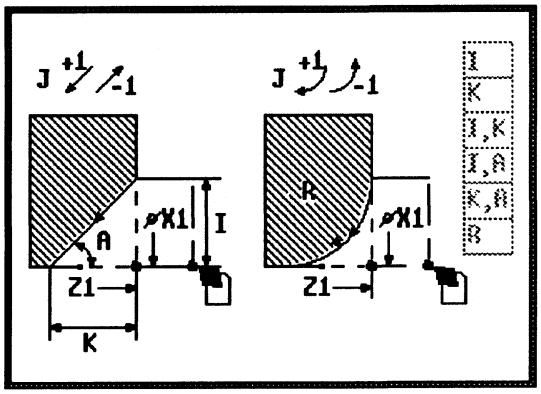
ERRORS:

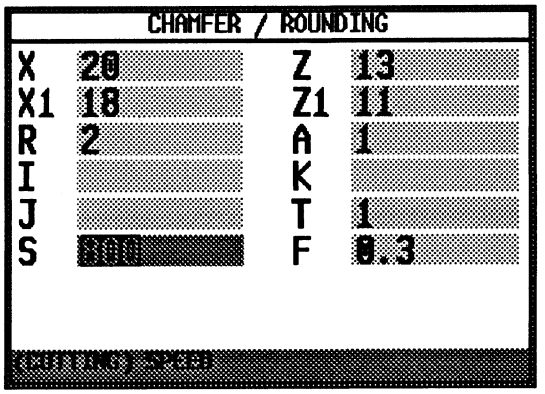
P02	One or more parameters are not programmed
P07	Programmed R does not fit between begin- and endpoint
O135	Value < minimun (i.e R < 0)
O134	Value > maximum (i.e T > 48)

14.6 CHAMFER / ROUNDING G815



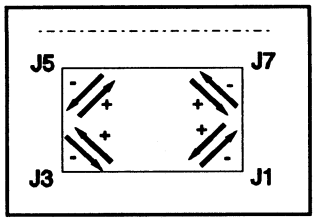
							GRUNDIG
	WITH RETRACT						RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Cornerpoint of the chamfer/rounding	
R	Rounding Radius (optional)	
A	Angle of the chamfer (optional)	
I	Chamfer length (optional)	
K	Chamfer length (optional)	
J	Cutting direction in X-axis (optional) + = positive X-axis - = negative X-axis Corner orientation 1,3,5,7 If not programmed, J is derived from the position of start- and cornerpoint.	



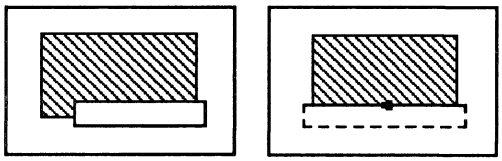
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

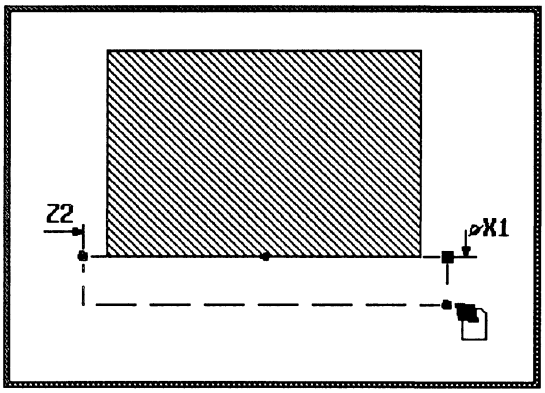
P01	Wrong combination of I, K, A, R
P02	One or more parameters (not optional) are not programmed
P07	Wrong value of J
P190	Intersection point with line A gives too large positive position
P191	Intersection point with line A gives too small negative position
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

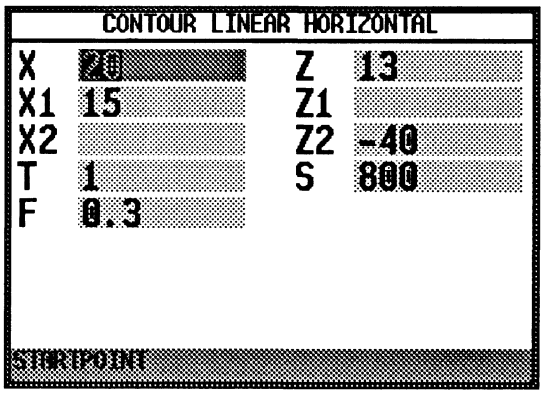
REMARK:
Automatic tooltip radiuscompensation

14.7 CONTOUR LINEAR HORIZONTAL G810



							GRUNDIG
	WITH RETRACT			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





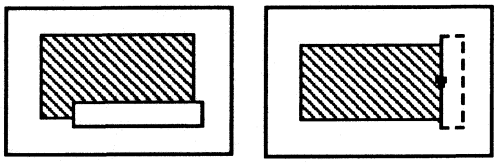
PARAMETERS:

X,Z	Startpoint of the single cut cycle	(only Teach-In)
X1	Beginpoint of the linear movement	
Z2	Endpoint of the linear movement	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

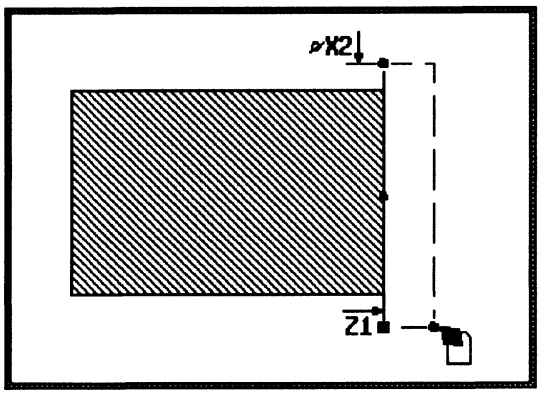
ERRORS:

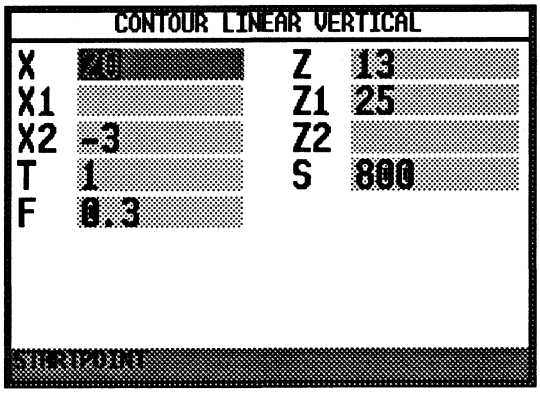
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.8 CONTOUR LINEAR VERTICAL G814



							GRUNDIG
	WITH RETRACT			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





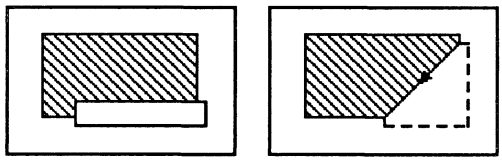
PARAMETERS:

X,Z	Startpoint of the single cut cycle	(only Teach-In)
Z1	Beginpoint of the linear movement	
X2	Endpoint of the linear movement	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

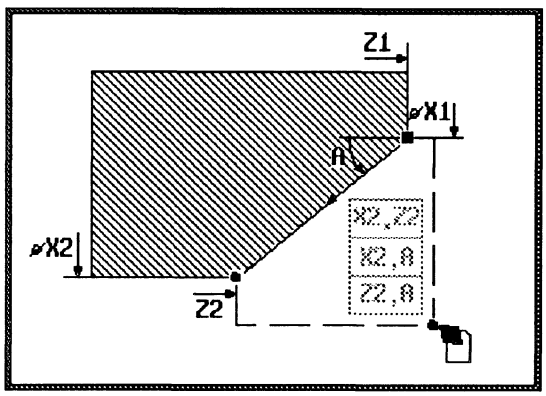
ERRORS:

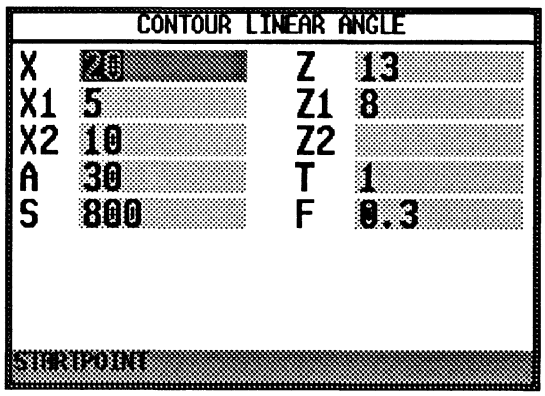
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.9 CONTOUR LINEAR ANGLE G811



							GRUNDIG
	WITH RETRACT			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the single cut cycle	(only Teach-In)
X1,Z1	Beginpoint of the linear movement	
X2,Z2	Endpoint of the linear movement	
A	Angle (min: 0.001 degr, max: +89.999 degr)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

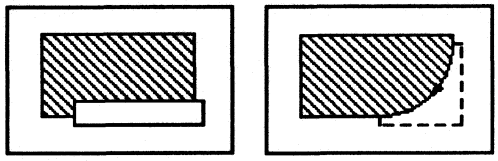
P01	X2,Z2,A together programmed
P02	One or more parameters are not programmed
P07	X1 = X2 (minimum difference = .001 mm)
	Z1 = Z2 (minimum difference = .001 mm)
P190	Intersection point with line A gives too large positive position
P191	Intersection point with line A gives too small negative position
O135	Value < minimun (i.e A < 0)
O134	Value > maximum (i.e A > +90)

REMARK:

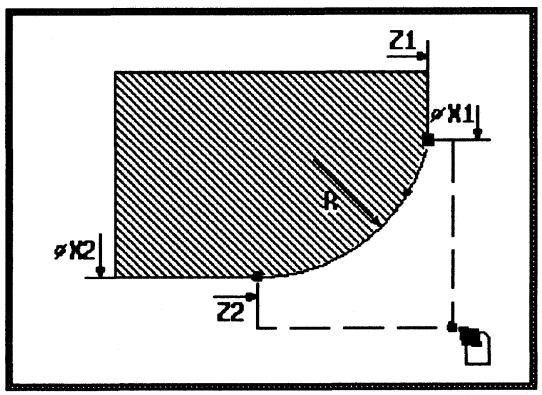
Automatic tooltip radiuscompensation

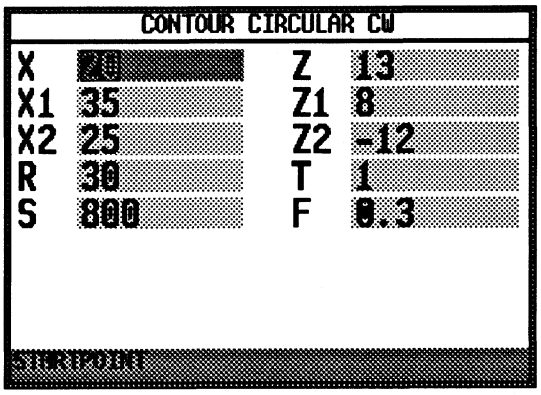
The startpoint X,Z must be beyond the begin and endpoint, otherwise the tool can collide with the material during retract to the startpoint.

14.10 CONTOUR CIRCULAR CLOCKWISE G812



							GRUNDIG
	WITH RETRACT			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the single cut cycle	(only Teach-In)
X1,Z1	Beginpoint of the circ. movement	
X2,Z2	Endpoint of the circ. movement	
R	Circle radius	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

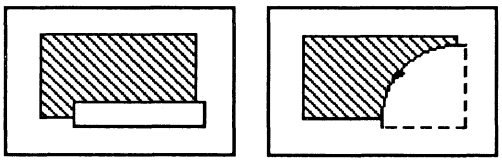
ERRORS:

P02	One or more parameters are not programmed
P07	Programmed R does not fit between begin- and endpoint
O135	Value < minimum (i.e T < 0)
O134	Value > maximum (i.e T > 48)

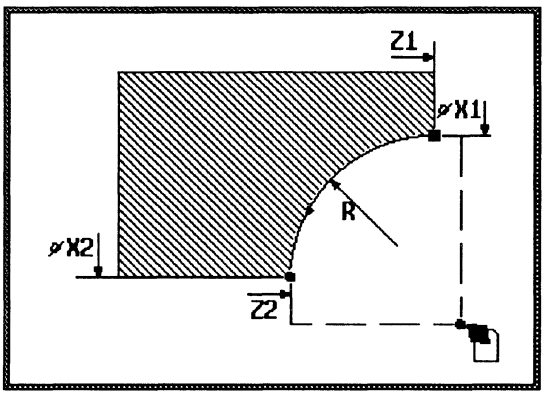
REMARK:

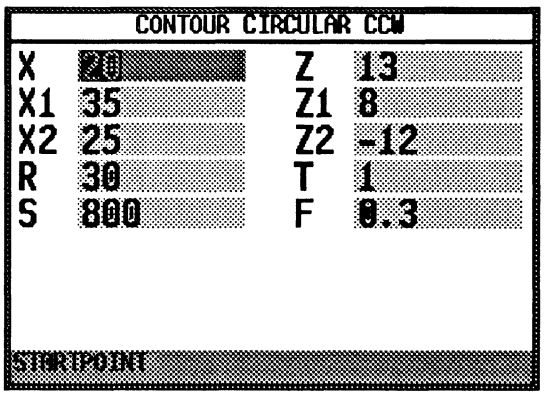
Automatic tooltip radiuscompensation

14.11 CONTOUR CIRCULAR COUNTER CLOCKWISE G813



							GRUNDIG
	WITH RETRACT			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

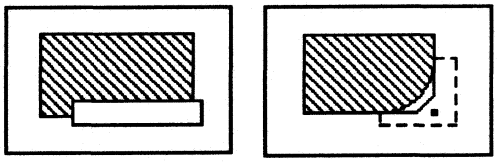
X,Z	Startpoint of the single cut cycle	(only Teach-In)
X1,Z1	Beginpoint of the circ. movement	
X2,Z2	Endpoint of the circ. movement	
R	Circle radius	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

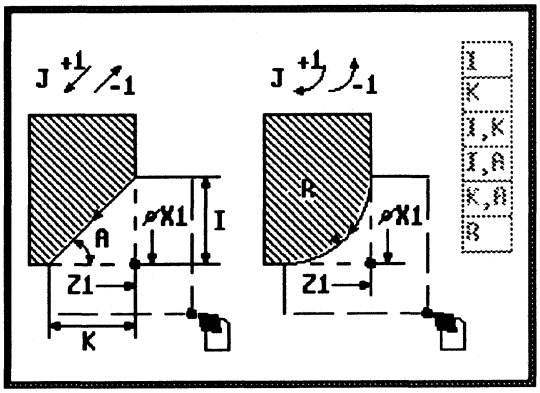
P02	One or more parameters are not programmed
P07	Programmed R does not fit between begin- and endpoint
P20	Programmed R equal or smaller than the tool radius This alarm also generates message P34
O135	Value < minimun (i.e R < 0)
O134	Value > maximum (i.e T > 48)

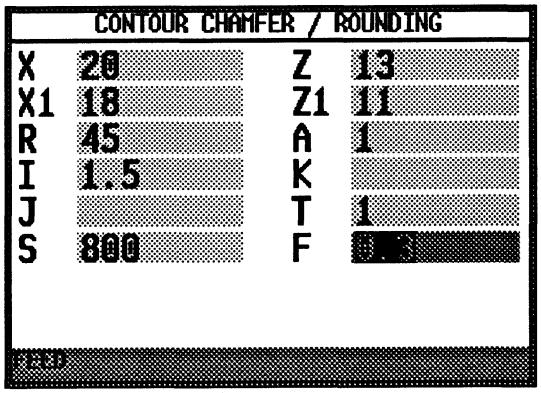
REMARK:
Automatic tooltip radiuscompensation

14.12 CONTOUR CHAMFER / ROUNDING G816



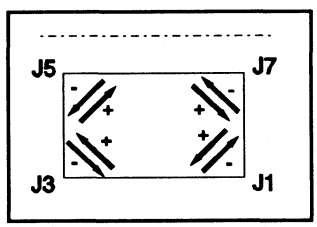
							GRUNDIG
	WITH RETRACT			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Cornerpoint of the fase/rounding	
R	Rounding Radius (optional)	
A	Angle of the chamfer (optional)	
I	Chamfer length (optional)	
K	Chamfer length (optional)	
J	Cutting direction in X-axis (optional) + = positive X-axis - = negative X-axis Corner orientation 1,3,5,7 If not programmed, J is derived from the position of start- and cornerpoint.	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev (only Teach-In)	

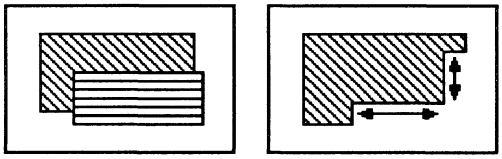


ERRORS:

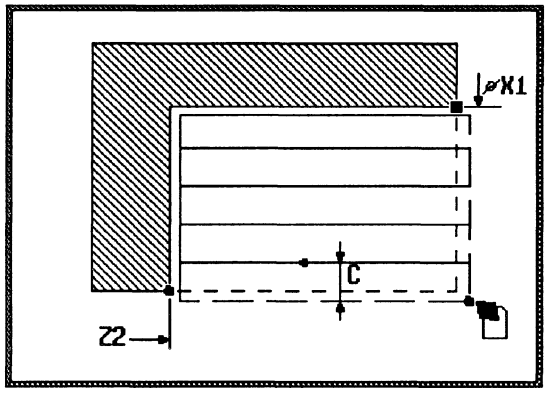
P01	Wrong combination of I, K, A, R
P02	One or more parameters (not optional) are not programmed
P07	Wrong value of J
P190	Intersection point with line A gives too large positive position
P191	Intersection point with line A gives too small negative position
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

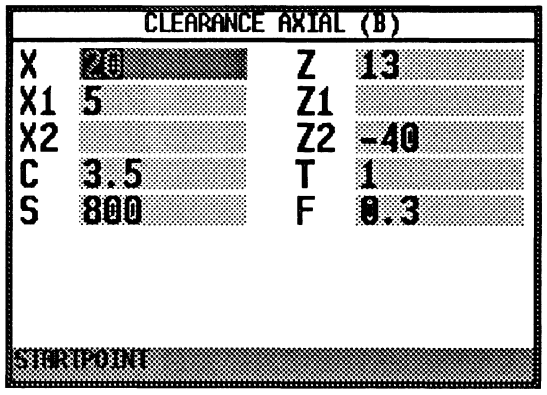
REMARK:
Automatic tooltip radiuscompensation

14.13 CLEARANCE AXIAL (BASIC) G820



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





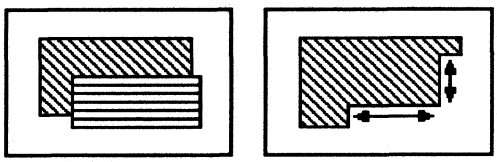
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1	Beginpoint of the contour	
Z2	Endpoint of the contour	
C	Cutting depth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev (only Teach-In)	

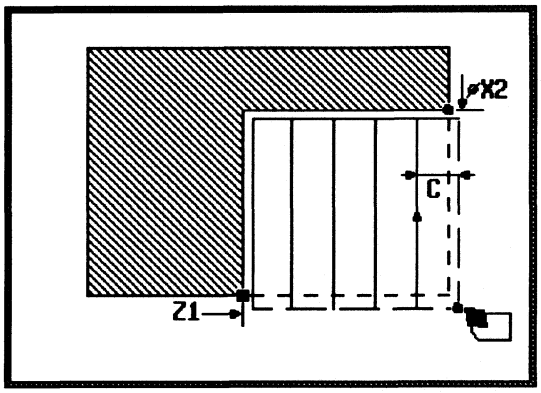
ERRORS:

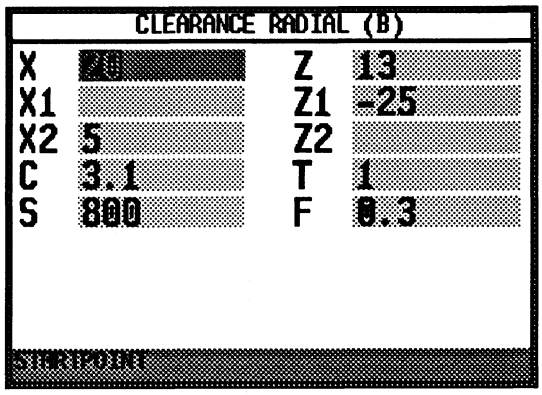
P02 One or more parameters (not optional) are not programmed
O135 Value < minimun (i.e T < 0)
O134 Value > maximum (i.e T > 48)

14.14 CLEARANCE RADIAL (BASIC) G821



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





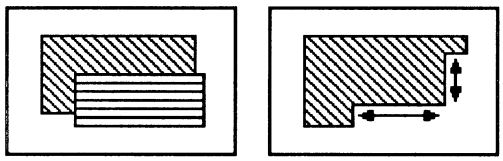
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
Z1	Beginpoint of the contour	
X2	Endpoint of the contour	
C	Cutting depth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

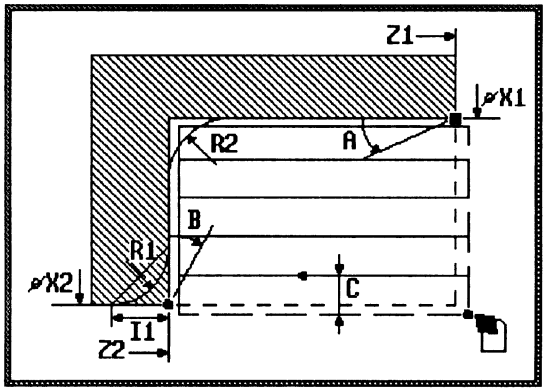
ERRORS:

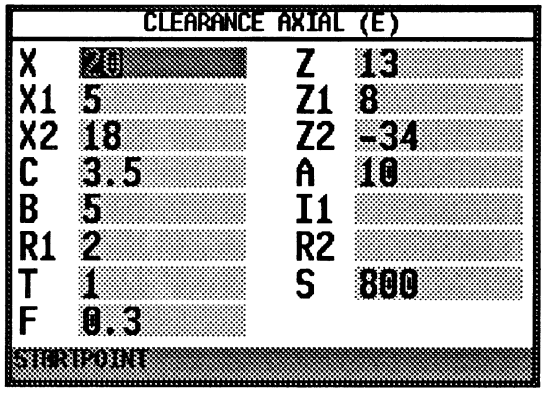
P02	One or more parameters (not optional) are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.15 CLEARANCE AXIAL (EXTENDED) G822



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

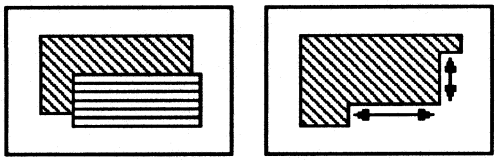
X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
A,B	First and second angle (min: 0 degr, max: 90 degr) (optional)	
C	Cutting depth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

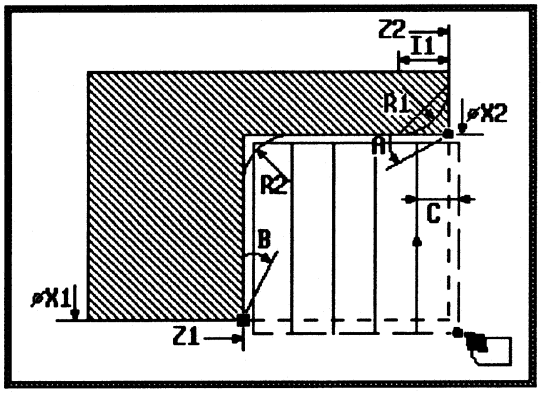
P01	R1 and I1 are programmed
	B=90 and I1 is programmed
	B=90 and R1 is programmed
P02	One or more parameters (not optional) are not programmed
P07	R2 too large for contour
	I1 too large for contour
	R1 too large for contour
	Intersectionpoint line A and B is beyond the area between X1,Z1 and X2,Z2
	Startpoint Z on wrong side of Z1
P20	Toolradius >= R2
O135	Value < minimun (i.e R1=0, R2=0)
O134	Value > maximum (i.e B > 90)

REMARK:
Automatic tooltip radiuscompensation

14.16 CLEARANCE RADIAL (EXTENDED) G823



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN



CLEARANCE RADIAL (E)

X	20	Z	13
X1	18	Z1	-21
X2	5	Z2	8
C	2.8	A	12
B	5	I1	
R1		R2	4
T	1	S	800
F	0.3		

STARTPOINT

PARAMETERS:

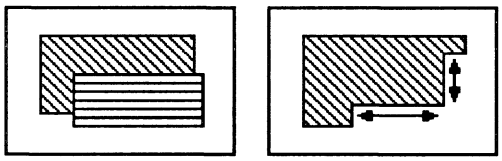
X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
A,B	First and second angle (min: 0 degr, max: 90 degr) (optional)	
C	Cutting depth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

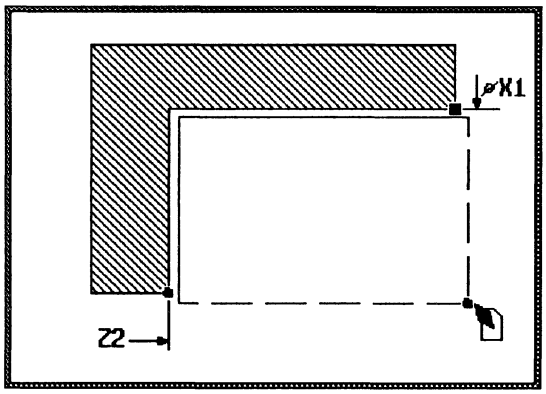
P01	R1 and I1 are programmed
	A=90 and I1 is programmed
	A=90 and R1 is programmed
P02	One or more parameters (not optional) are not programmed
P07	R2 too large for contour
	I1 too large for contour
	R1 too large for contour
	Intersectionpoint line A and B is beyond the area between X1,Z1 and X2,Z2
	Startpoint X on wrong side of X1
P20	Toolradius >= R2
O135	Value < minimun (i.e R1=0, R2=0)
O134	Value > maximum (i.e B > 90)

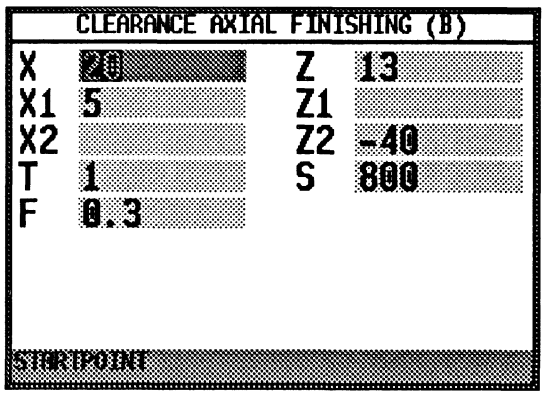
REMARK:
Automatic tooltip radiuscompensation

14.17 CLEARANCE AXIAL FINISHING (BASIC) G824



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





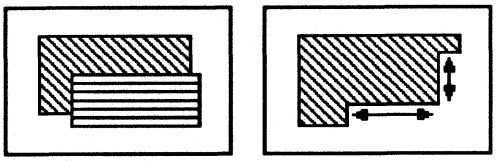
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
Z1	Beginpoint of the contour	
X2	Endpoint of the contour	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

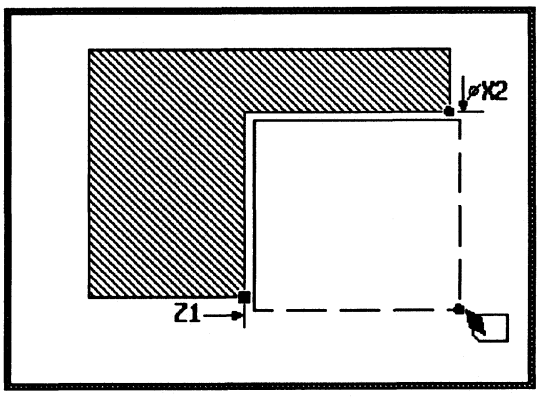
ERRORS:

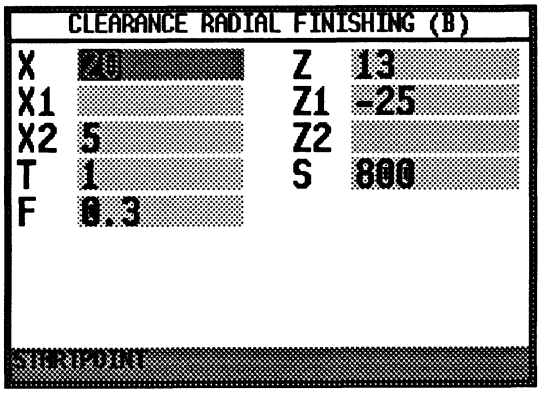
P02	One or more parameters (not optional) are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.18 CLEARANCE RADIAL FINISHING (BASIC) G825



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





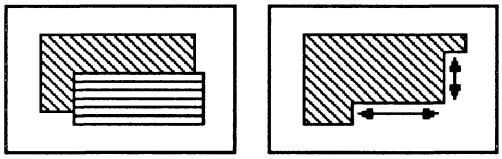
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
Z1	Beginpoint of the contour	
X2	Endpoint of the contour	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

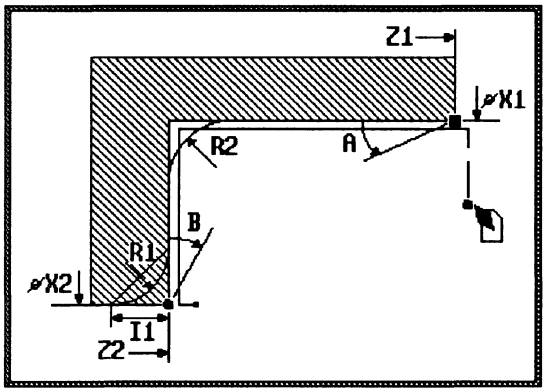
ERRORS:

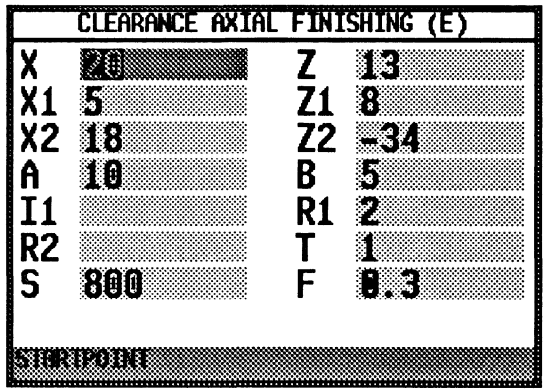
P02	One or more parameters (not optional) are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.19 CLEARANCE AXIAL FINISHING (EXTENDED) G826



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
A,B	First and second angle (min: 0 degr, max: 90 degr) (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

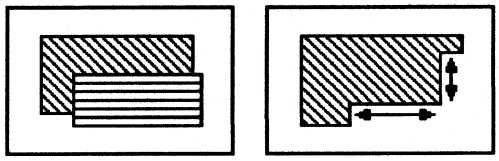
P01	R1 and I1 are programmed
	B=90 and I1 is programmed
	B=90 and R1 is programmed
P02	One or more parameters (not optional) are not programmed
P07	R2 too large for contour
	I1 too large for contour
	R1 too large for contour
	Intersectionpoint line A and B is beyond the area between X1,Z1 and X2,Z2
	Startpoint Z on wrong side of Z1
P20	Toolradius >= R2
O135	Value < minimun (i.e R1=0, R2=0)
O134	Value > maximum (i.e B > 90)

REMARK:

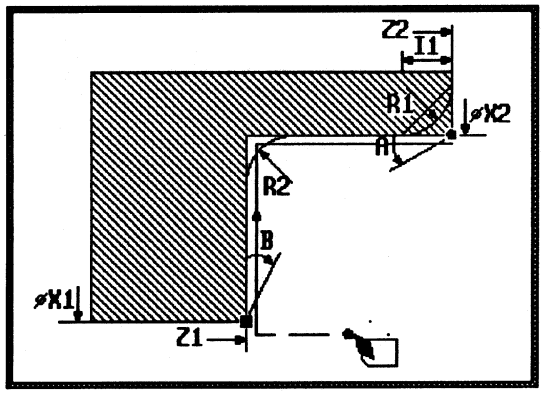
Automatic tooltip radiuscompensation

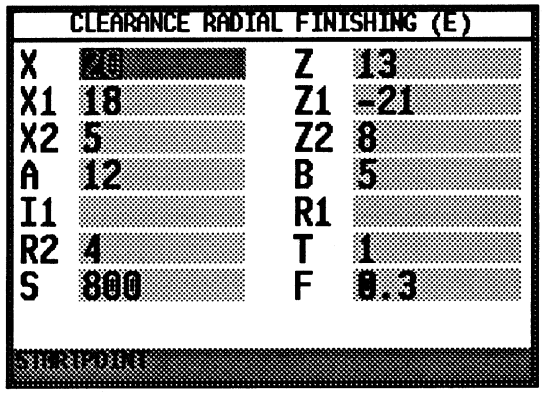
When the startpoint X is under the beginpoint X1, the tool first moves to the beginpoint Z1.

14.20 CLEARANCE RADIAL FINISHING (EXTENDED) G827



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
A,B	First and second angle (min: 0 degr, max: 90 degr) (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

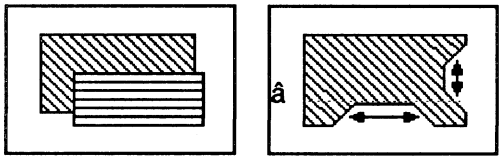
P01	R1 and I1 are programmed
	A=90 and I1 is programmed
	A=90 and R1 is programmed
P02	One or more parameters (not optional) are not programmed
P07	R2 too large for contour
	I1 too large for contour
	R1 too large for contour
	Intersectionpoint line A and B is beyond the area between X1,Z1 and X2,Z2
	Startpoint X on wrong side of X1
P20	Toolradius >= R2
O135	Value < minimun (i.e R1=0, R2=0)
O134	Value > maximum (i.e B > 90)

REMARK:

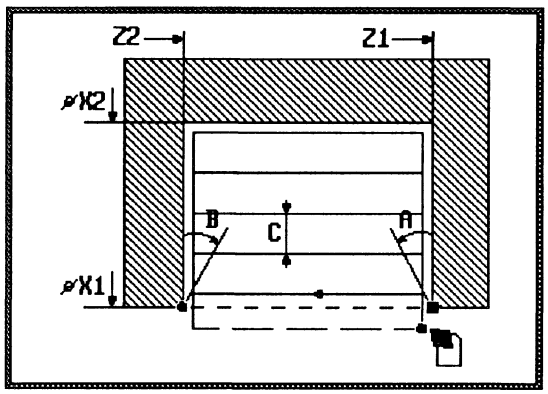
Automatic tooltip radiuscompensation

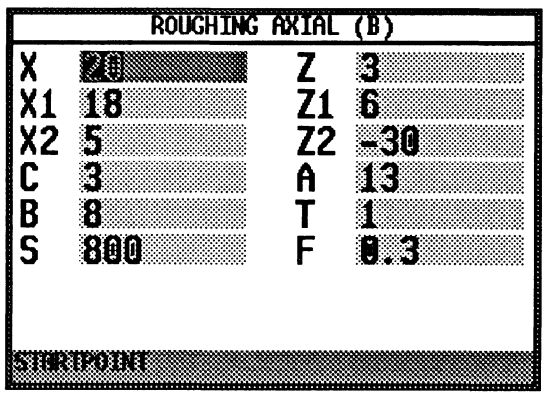
When the startpoint Z is left from the beginpoint Z1, the tool first moves to the beginpoint X1.

14.21 ROUGHING AXIAL (BASIC) G830



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

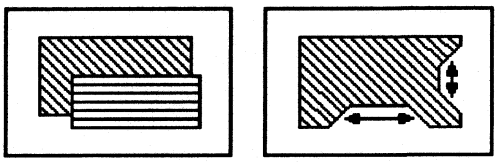
X,Z	Startpoint of the cycle (min: 0)	(only Teach-In)
X1,Z1	Beginpoint of the contour (min: 0)	
X2,Z2	Endpoint of the contour (min: 0)	
C	Cutting depth	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

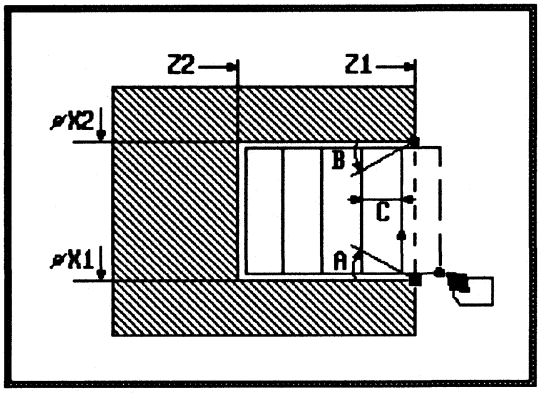
P02	One or more parameters (not optional) are not programmed
P07	X1 positive and X2 negative or X1 negative and X2 positive Startpoint X,Z is in or on the material
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

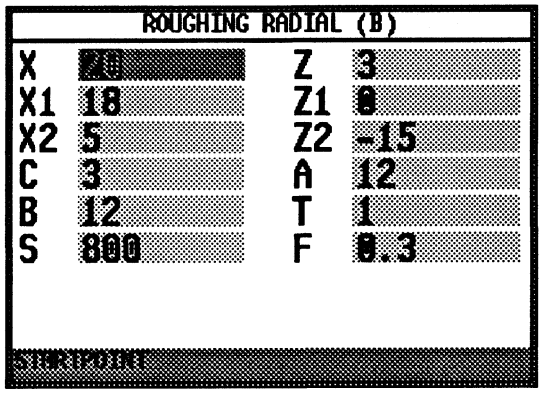
REMARK:
Automatic tooltip radiuscompensation

14.22 ROUGHING RADIAL (BASIC) G831



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
C	Cutting depth	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

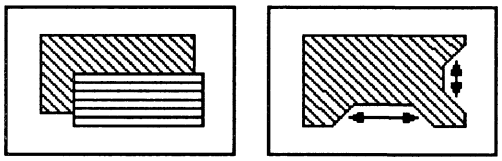
ERRORS:

P02	One or more parameters (not optional) are not programmed
P07	X1 < X2 (only X1 > X2 operations are permitted)
	Z1 < Z2 (only Z1 > Z2 operations are permitted)
	Lines of contour with A and B are intersecting lines
	Startpoint X,Z is in or on the material
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

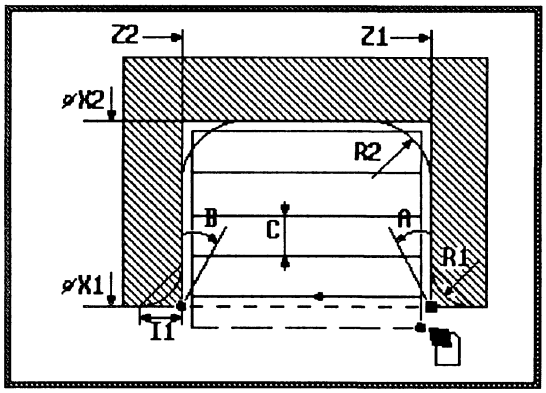
REMARK:

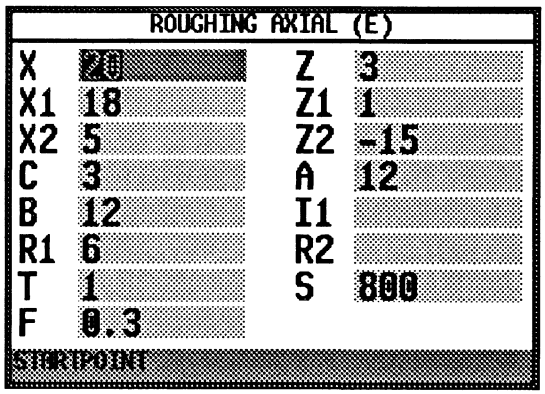
Partly automatic tooltip radiuscompensation only for the begin of the cycle (angle A)
Substract 2* tool radius from X1 to obtain correct compensation on line A.
No tooltip radiuscompensation on line B.
The depth must exceed the tool radius when I1 or R1 not programmed.

14.23 ROUGHING AXIAL (EXTENDED) G832



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle (min: 0)	(only Teach-In)
X1,Z1	Beginpoint of the contour (min: 0)	
X2,Z2	Endpoint of the contour (min: 0)	
C	Cutting depth	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (min: > 0) (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

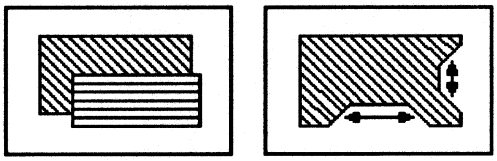
ERRORS:

P01	R1 and I1 are programmed
P02	One or more parameters (not optional) are not programmed
P07	X1 positive and X2 negative or X1 negative and X2 positive
	Startpoint X,Z is in or on the material
	I1 too large for contour
	R1 too large for contour
	R2 too large for contour
O135	Value < minimun (i.e R1 = 0)
O134	Value > maximum (i.e B > 89)

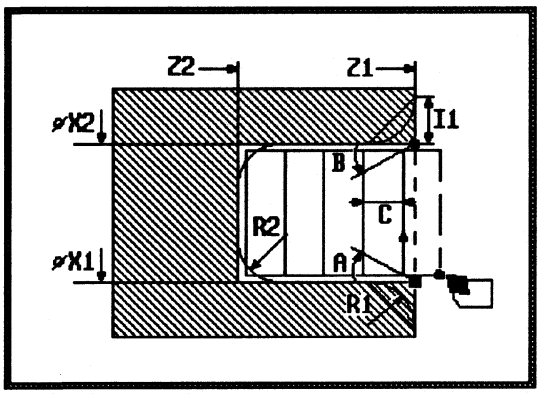
REMARK:

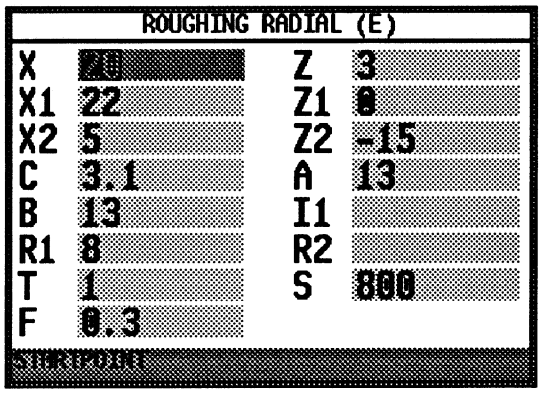
Automatic tooltip radiuscompensation
Rounding R1 is not executed on side A
Rounding R2 is executed as straight line on side A

14.24 ROUGHING RADIAL (EXTENDED) G833



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
C	Cutting depth	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (min: > 0) (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

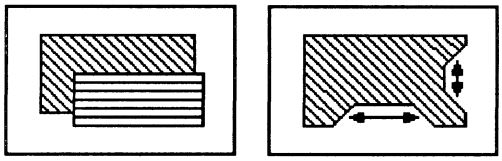
ERRORS:

P01	R1 and I1 are programmed, R2 is to large for contour
P02	One or more parameters (not optional) are not programmed
P07	X1 < X2 (only X1 > X2 operations are permitted)
	Z1 < Z2 (only Z1 > Z2 operations are permitted)
	The combination of A,B,R2,R1,I1 doesn't fit in the programmed contour
	Startpoint X,Z is in or on the material
O135	Value < minimun (i.e R1 = 0)
O134	Value > maximum (i.e B > 89)

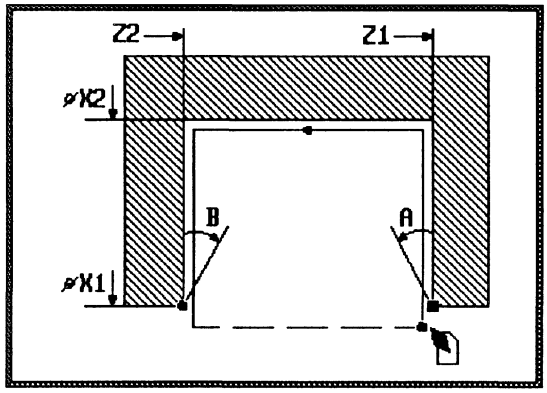
REMARK:

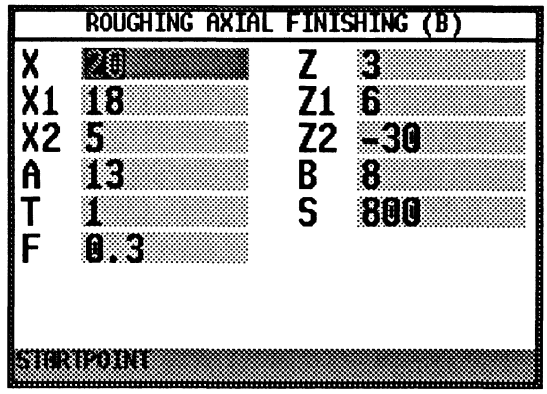
Partly automatic tooltip radiuscompensation only for the begin of the cycle (angle A)
Substract 2* tool radius from X1 to obtain correct compensation on line A.
No tooltip radiuscompensation on line B.
The depth must exceed the tool radius when I1 or R1 not programmed.
Rounding R1 is not executed on side A
Rounding R2 is executed as straight line on side A

14.25 ROUGHING AXIAL FINISHING (BASIC) G834



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

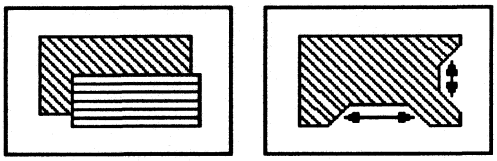
X,Z	Startpoint of the cycle (min: 0)	(only Teach-In)
X1,Z1	Beginpoint of the contour (min: 0)	
X2,Z2	Endpoint of the contour (min: 0)	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

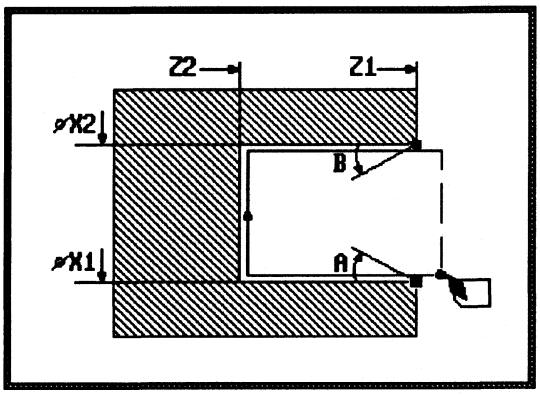
P02	One or more parameters (not optional) are not programmed
P07	X1 positive and X2 negative or X1 negative and X2 positive Startpoint X,Z is in or on the material
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

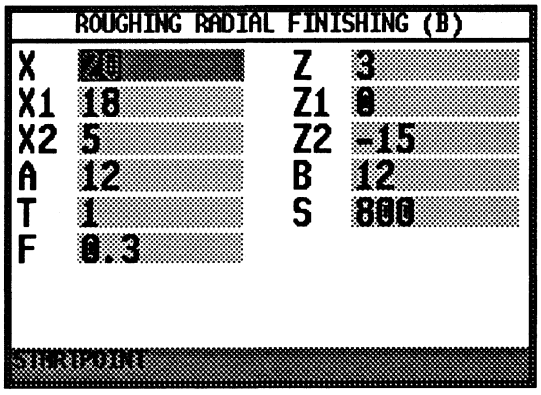
REMARK:
Automatic tooltip radiuscompensation

14.26 ROUGHING RADIAL FINISHING (BASIC) G835



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

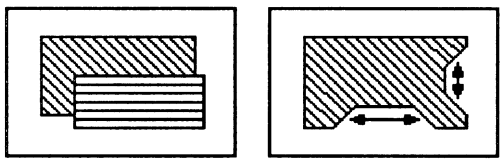
P02	One or more parameters (not optional) are not programmed
P07	X1 < X2 (only X1 > X2 operations are permitted)
	Z1 < Z2 (only Z1 > Z2 operations are permitted)
	Lines of contour with A and B are intersecting lines
	Startpoint X,Z is in or on the material
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

REMARK:

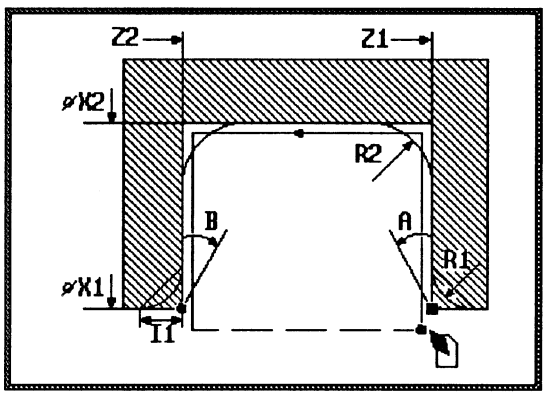
Automatic tooltip radiuscompensation

The depth must exceed the tool radius when I1 or R1 not programmed

14.27 ROUGHING AXIAL FINISHING (EXTENDED) G836



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN



ROUGHING AXIAL FINISHING (E)

X	20	Z	3
X1	18	Z1	1
X2	5	Z2	-15
A	12	B	12
I1		R1	6
R2		T	1
S	800	F	0.3

STARTPOINT

PARAMETERS:

X,Z	Startpoint of the cycle (min: 0)	(only Teach-In)
X1,Z1	Beginpoint of the contour (min: 0)	
X2,Z2	Endpoint of the contour (min: 0)	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

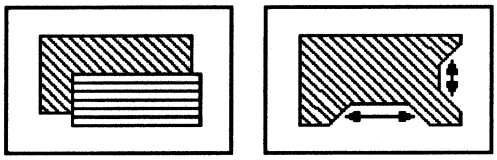
ERRORS:

P01	R1 and I1 are programmed
	R2 is to large for contour
P02	One or more parameters (not optional) are not programmed
P07	X1 positive and X2 negative or X1 negative and X2 positive
	Startpoint X,Z is in or on the material
	I1 too large for contour
	R1 too large for contour
	R2 too large for contour
O135	Value < minimun (i.e R1 = 0)
O134	Value > maximum (i.e B > 89)

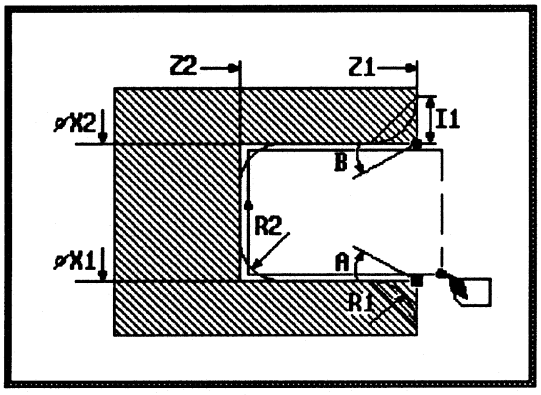
REMARK:

Automatic tooltip radiuscompensation

14.28 ROUGHING RADIAL FINISHING (EXTENDED) G837



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN



ROUGHING RADIAL FINISHING (E)

X	26	Z	3
X1	22	Z1	0
X2	5	Z2	-15
A	13	B	13
I1		R1	6
R2		T	1
S	800	F	0.3

STARTPOINT

PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
A,B	First and second angle (min: 0 degr, max: 89 degr)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

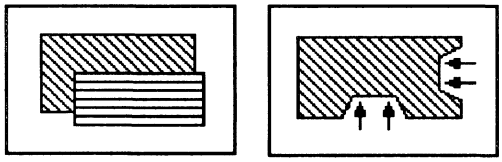
P01	R1 and I1 are programmed
	R2 is to large for contour
P02	One or more parameters (not optional) are not programmed
P07	X1 < X2 (only X1 > X2 operations are permitted)
	Z1 < Z2 (only Z1 > Z2 operations are permitted)
	Operation is not in the +X direction
	The combination of A,B,R2,R1,I1 doesn't fit in the programmed contour
	Startpoint X,Z is in or on the material
O135	Value < minimun (i.e R1 = 0)
O134	Value > maximum (i.e B > 89)

REMARK:

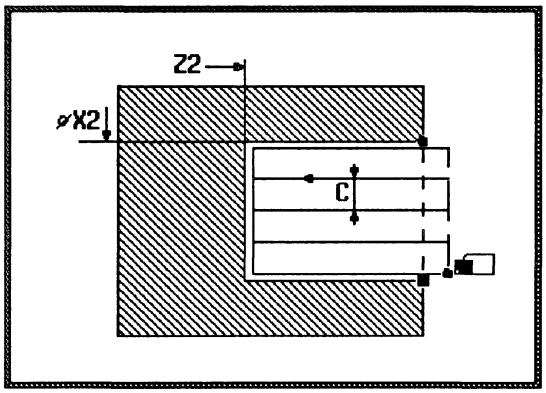
Automatic tooltip radiuscompensation

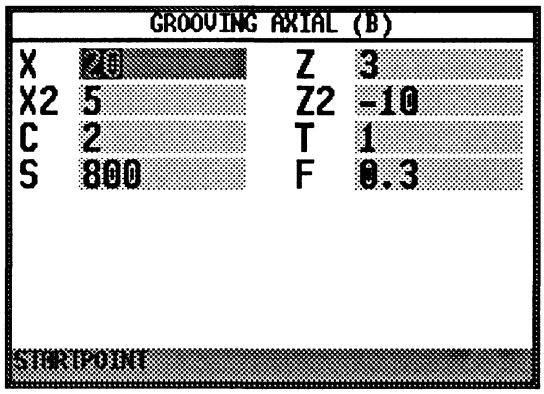
The depth must exceed the tool radius when I1 or R1 not programmed.

14.29 GROOVING AXIAL (BASIC) G840



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





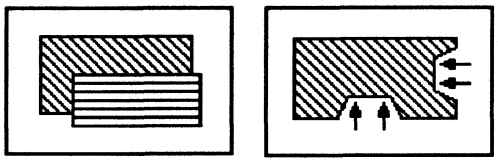
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X2,Z2	Endpoint of the contour	
C	Toolwidth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

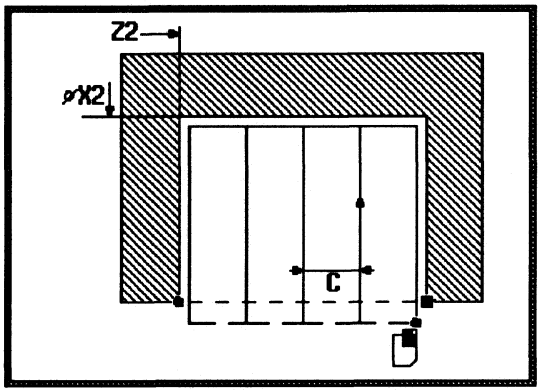
ERRORS:

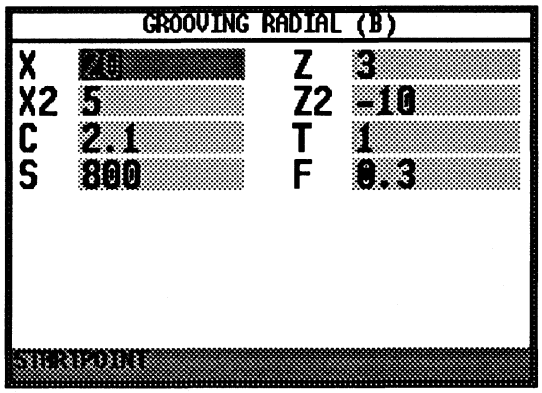
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of groove
P20	Toolwidth C < 2* toolradius R
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.30 GROOVING RADIAL (BASIC) G841



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





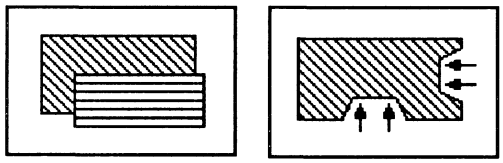
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X2,Z2	Endpoint of the contour	
C	Toolwidth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

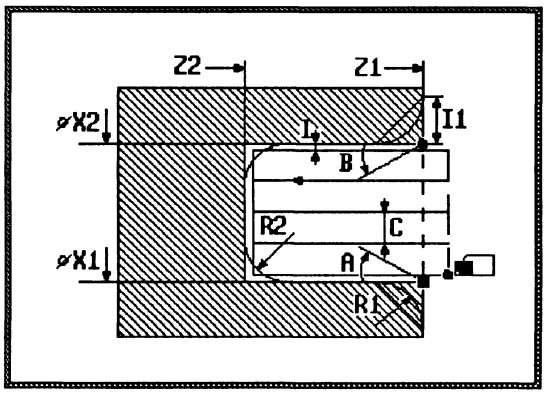
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of groove
P20	Toolwidth C < 2* toolradius R
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

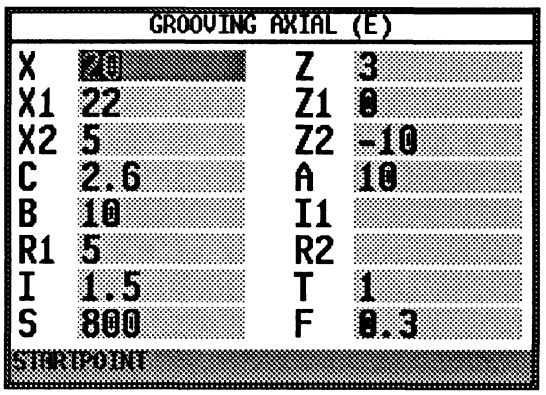
14.31 GROOVING AXIAL (EXTENDED) G842



Y

							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
C	Toolwidth	
A,B	First and second Angle (min: 0 degr, max: 89 degr) (optional)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
I	Finishing radial (min: 0, max: < 2) (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

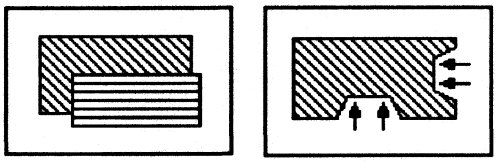
P01	R1 and I1 are programmed
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of bottom Depth of groove is not sufficient for chamfer and roundings (I1/R1, R2)
P20	Toolwidth C < 2* toolradius R
P116	I1 or R1 > (C - toolradius)
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

REMARK:

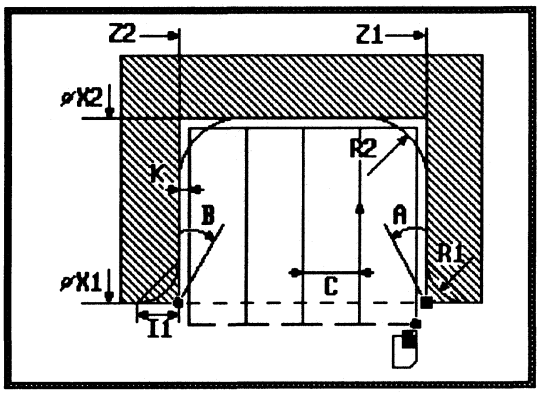
Automatic tooltip radiuscompensation

During roughing, the radius R2 is executed as a straight line

14.32 GROOVING RADIAL (EXTENDED) G843



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN



GROOVING RADIAL (E)

X	20	Z	3
X1	18	Z1	1
X2	5	Z2	-15
C	2.1	A	10
B	10	I1	
R1	5	R2	
K	1.2	T	1
S	800	F	0.3

STARTPOINT

PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
C	Toolwidth	
A,B	First and second Angle (min: 0 degr, max: 89 degr) (optional)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
K	Finishing axial (min: 0, max: < 2) (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

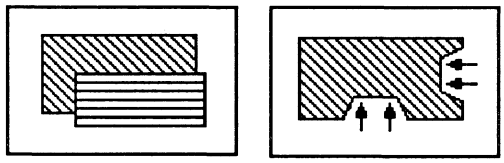
ERRORS:

P01	I1 and R1 are programmed
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of bottom Depth of groove is not sufficient for chamfer and roundings (I1/R1, R2)
P20	Toolwidth C < 2* toolradius R
P116	I1 or R1 > (C - toolradius)
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

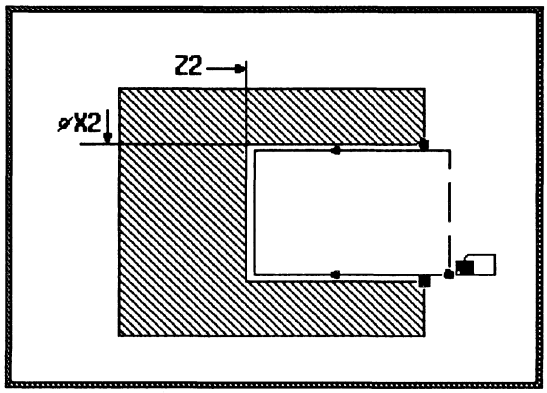
REMARK:

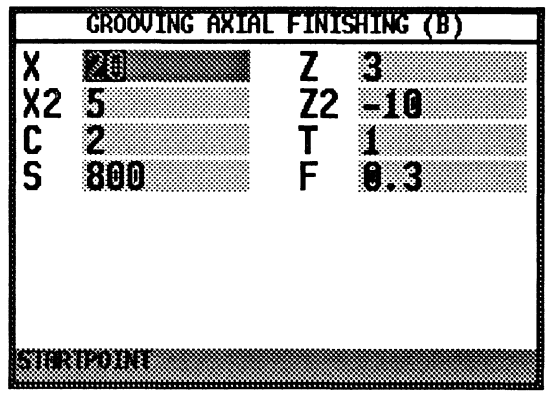
Automatic tooltip radiuscompensation
During roughing, the radius R2 is executed as a straight line

14.33 GROOVING AXIAL FINISHING (BASIC) G844



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





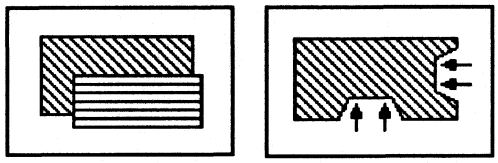
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X2,Z2	Endpoint of the contour	
C	Toolwidth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

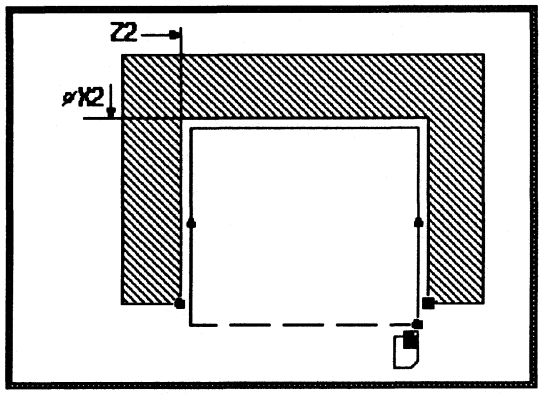
ERRORS:

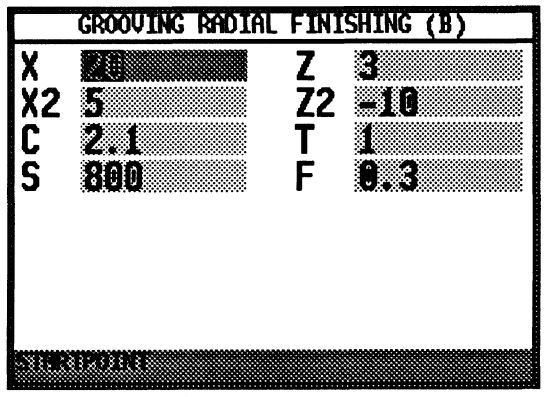
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of groove
P20	Toolwidth C < 2* toolradius R
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.34 GROOVING RADIAL FINISHING (BASIC) G845



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





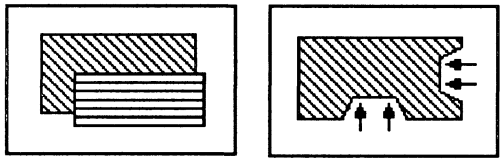
PARAMETERS:

X,Z	Startpoint of the cycle	(only Teach-In)
X2,Z2	Endpoint of the contour	
C	Toolwidth	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

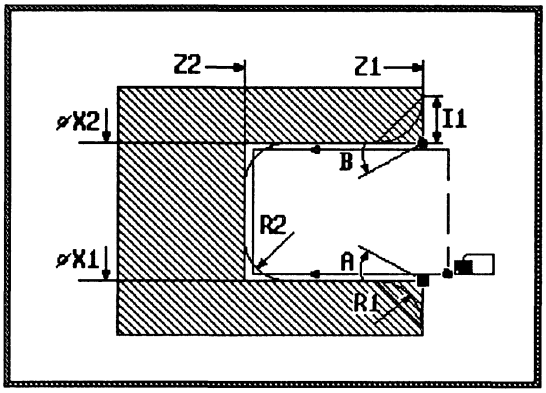
ERRORS:

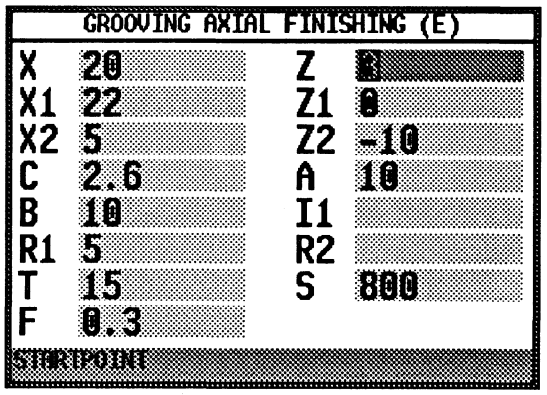
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of groove
P20	Toolwidth C < 2* toolradius R
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.35 GROOVING AXIAL FINISHING (EXTENDED) G846



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

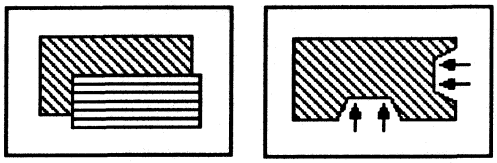
X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
C	Toolwidth	
A,B	First and second Angle (min: 0 degr, max: 89 degr) (optional)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

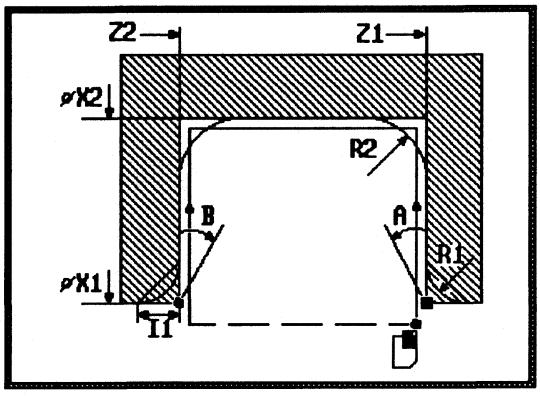
P01	R1 and I1 are programmed
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of bottom Depth of groove is not sufficient for chamfer and roundings (I1/R1, R2)
P20	Toolwidth C < 2* toolradius R
P116	I1 or R1 > (C - toolradius)
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

REMARK:
Automatic tooltip radiuscompensation

14.36 GROOVING RADIAL FINISHING (EXTENDED) G847



							GRUNDIG
RADIAL	EXTENDED	FINISHING CUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN



GROOVING RADIAL FINISHING (E)

X	20	Z	0
X1	18	Z1	1
X2	5	Z2	-15
C	2.1	A	10
B	10	I1	
R1	5	R2	
T	12	S	800
F	0.3		

STARTPOINT

PARAMETERS:

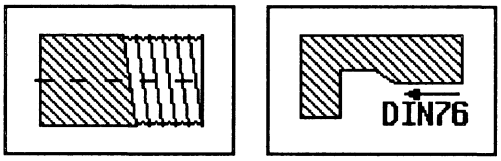
X,Z	Startpoint of the cycle	(only Teach-In)
X1,Z1	Beginpoint of the contour	
X2,Z2	Endpoint of the contour	
C	Toolwidth	
A,B	First and second Angle (min: 0 degr, max: 89 degr) (optional)	
I1	First chamfer length (optional)	
R1,R2	First and second radius (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

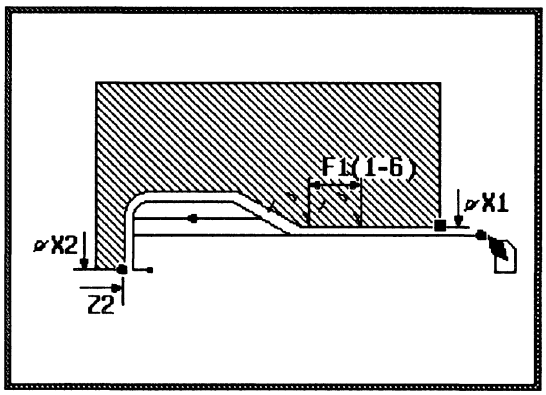
P01	R1 and I1 are programmed
P02	One or more parameters (not optional) are not programmed
P07	C is larger then width of bottom Depth of groove is not sufficient for chamfer and roundings (I1/R1, R2)
P20	Toolwidth C < 2* toolradius R
P116	I1 or R1 > (C - toolradius)
O135	Value < minimun (i.e B < 0)
O134	Value > maximum (i.e B > 89)

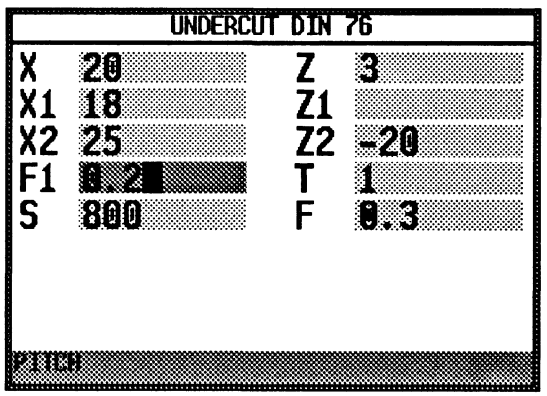
REMARK:
Automatic tooltip radiuscompensation

14.37 UNDERCUT DIN 76 G850



							GRUNDIG
				TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the undercut	(only Teach-In)
X1	Beginpoint of the undercut	
X2,Z2	Endpoint of the undercut	
F1	Pitch (min: 1, max: 6)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

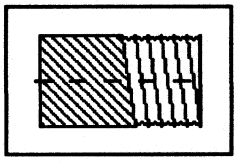
ERRORS:

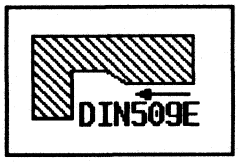
P02	One or more parameters are not programmed
P20	Toolradius > programmed radius (toolradius is greater then allowed for the undercut)
P49	Orientation cannot be activated in the cycle (T0 is active. use only T1 - T48)
O135	Value < minimun (i.e F1 < 1)
O134	Value > maximum (i.e F1 > 6)

REMARK:

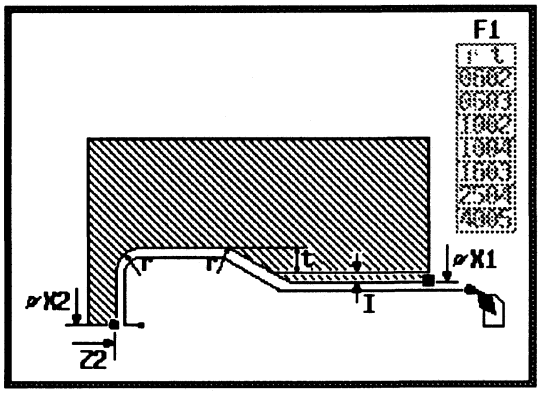
In Inch mode, the thread undercut cycle cannot be used. Use grooving cycles G830 - G837 to make undercuts in inch mode.

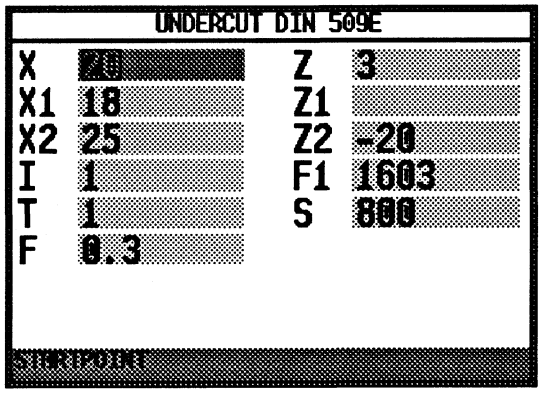
14.38 UNDERCUT DIN 509E G851





							GRUNDIG
				TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the undercut	(only Teach-In)
X1	Beginpoint of the undercut	
X2,Z2	Endpoint of the undercut	
F1	Size of radius and depth values: 602,603,1002,1004,1603,2504,4005	
I	Finishing allowance (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev (only Teach-In)	

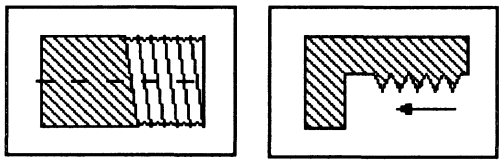
ERRORS:

P02	One or more parameters are not programmed
P07	F1 value is not allowed
P20	Toolradius > programmed radius (toolradius is larger then allowed for the undercut)
P49	Orientation cannot be activated in the cycle (T0 is active. use only T1 - T48)
O135	Value < minimun (i.e F1 < 602)
O134	Value > maximum (i.e F1 > 4005)

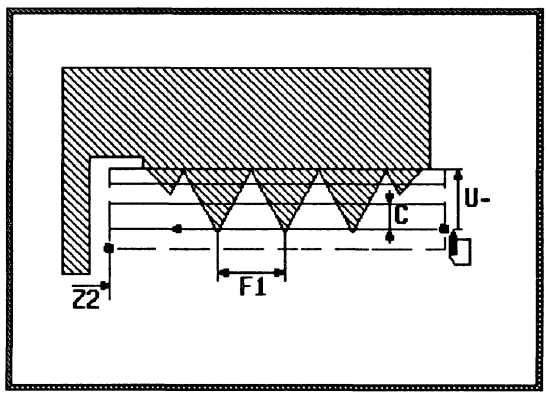
REMARK:

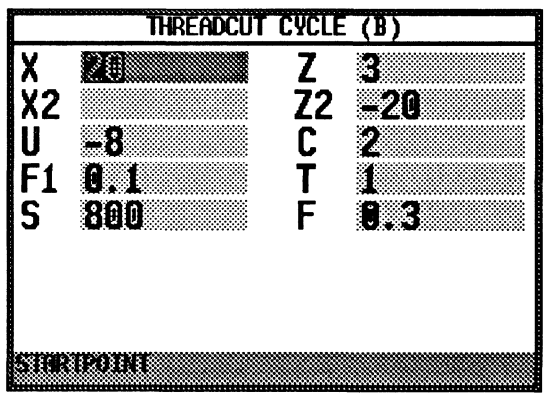
In Inch mode, the grinding undercut cycle can be used, but F1 has to be entered in mm. During execution, the values of F1 are recalculated to inch.

14.39 THREADCUT CYCLE (BASIC) G860



							GRUNDIG
	EXTENDED	RECUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





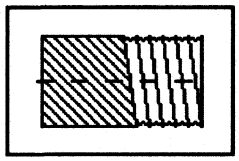
PARAMETERS:

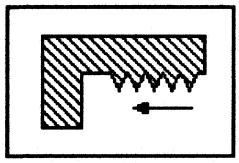
X,Z	Startpoint of the threadcut	(only Teach-In)
Z2	Endpoint of the threadcut	
U	Thread depth (optional)	
	U-999: inside thread with calculation	
	U999: outside thread with calculation	
	U not programmed: outside thread with calculation	
	(outside: U = -0.6495 * F1)	
	(inside: U = 0.6403 * F1)	
C	Cutting depth (optional)	
F1	Pitch	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

ERRORS:

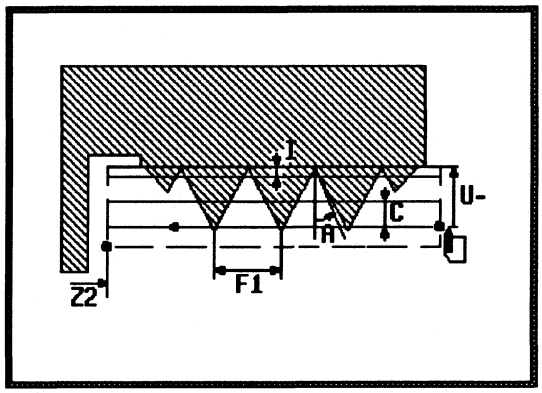
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

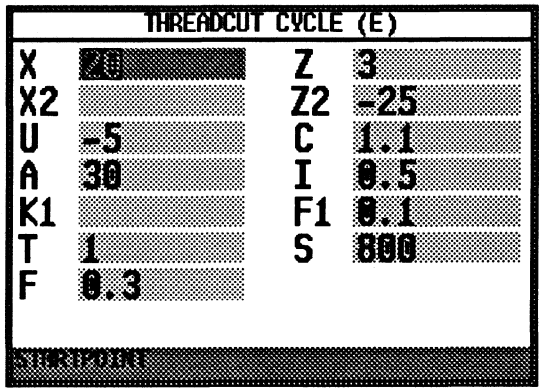
14.40 THREADCUT CYCLE (EXTENDED) G861





							GRUNDIG
	EXTENDED	RECUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Startpoint of the threadcut	(only Teach-In)
Z2	Endpoint of the threadcut	
U	Thread depth (optional) U-999: inside thread with calculation U999: outside thread with calculation U not programmed: outside thread with calculation (outside: U = -0.6495 * F1) (inside: U = 0.6403 * F1)	
C	Cutting depth	
A	In feed angle (min: 0 degr., max: 45 degr.) (optional)	
I	Finishing radial (optional)	
K1	Number of multiple threads (min: 1, max: 99) (optional)	
F1	Pitch	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

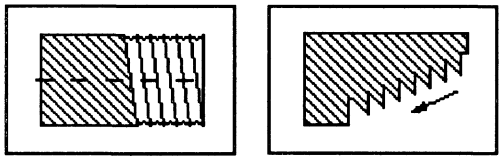
ERRORS:

P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

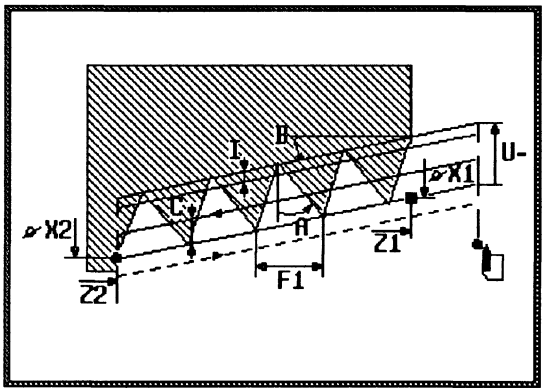
REMARK:

If Cutting depth C > Thread depth U then C = U

14.41 TAPERED THREADCUT CYCLE G863



							GRUNDIG
		RECUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN



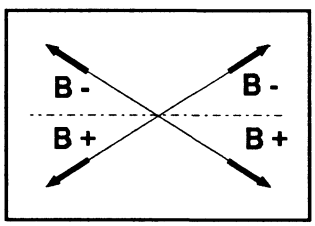
TAPERED THREADCUT CYCLE

X	20	Z	3
X1	18	Z1	0
X2	21	Z2	-20
U	5	C	2
A		B	3
I	1	K1	
F1	0.1	T	1
S	800	F	0.3

STARTPOINT

PARAMETERS:

X,Z	Startpoint of the threadcut	(only Teach-In)
X1,Z1	Beginpoint if the thread contour (X1 = optional)	
X2, Z2	Endpoint of the threadcut (X2 = optional)	
U	Thread depth (default: see cylind. threadcutting)	(optional)
C	Cutting depth	
A	In feed angle (min: 0 degr., max: 45 degr.)	(optional)
B	Taper angle (optional) -45 degr - +45 degr Use B instead of X1, or X2 Only possible X1,X2 or X1,B or X2,B	

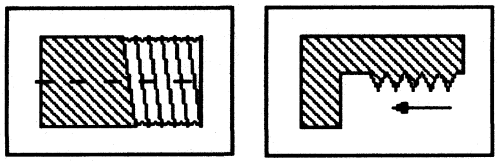


I	Finishing radial	(optional)
K1	Number of multiple threads (min: 1, max: 99)	(optional)
F1	Pitch	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

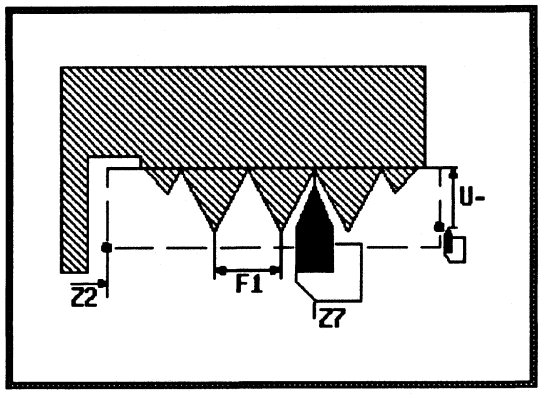
ERRORS:

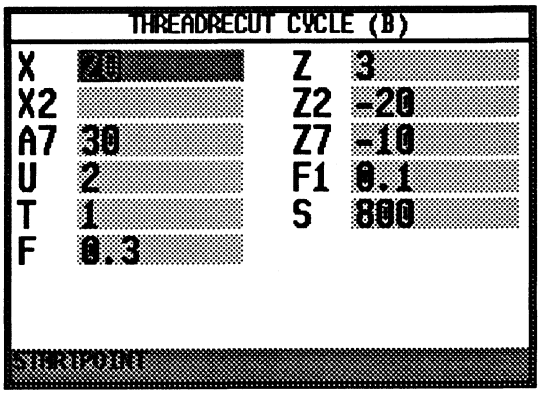
P01	Wrong combination of X1, X2, B
P02	One or more parameters are not programmed
P07	Taper angle > 45 degr or < -45 degr
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.42 THREAD RE-CUT CYCLE (BASIC) G864 (Option)



							GRUNDIG
	EXTENDED	RECUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





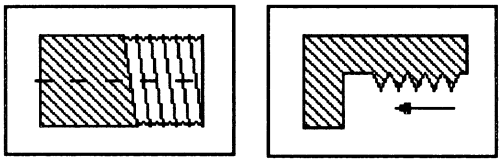
PARAMETERS:

X,Z	Startpoint of the tool in the thread	(only Teach-In)
Z2	Endpoint of the thread contour	
A7	Measured spindle angle (taken over with function key)	
Z7	Measured Z position in the thread (taken over with function key)	
U	Thread depth (optional)	
	U-999: inside thread with calculation	
	U999: outside thread with calculation	
	U not programmed: outside thread with calculation	
	(outside: U = -0.6495 * F1)	
	(inside: U = 0.6403 * F1)	
F1	Pitch	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

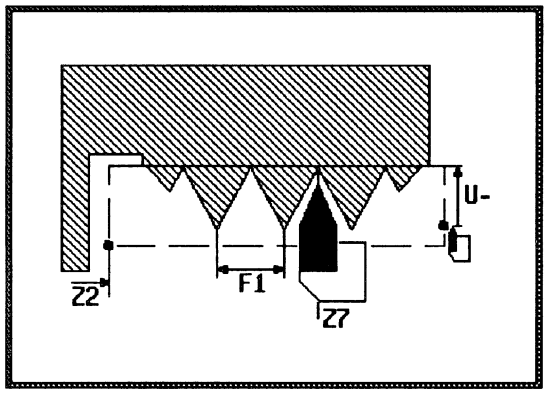
ERRORS:

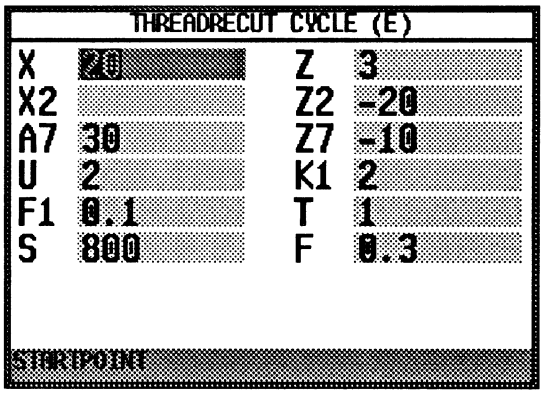
P02 One or more parameters are not programmed
O135 Value < minimun (i.e T < 0)
O134 Value > maximum (i.e T > 48)
O203 "Take over position" in A7 or Z7 is not possible when the spindle position is unknown.

14.43 THREAD RE-CUT CYCLE (EXTENDED) G865 (Option)



							GRUNDIG
	EXTENDED	RECUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





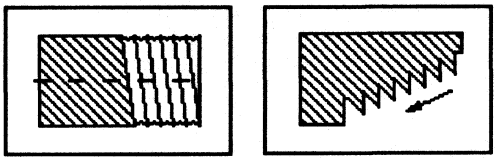
PARAMETERS:

X,Z	Startpoint of the tool in the thread	(only Teach-In)
Z2	Endpoint of the thread contour	
A7	Measured spindle angle (taken over with function key)	
Z7	Measured Z position in the thread (taken over with function key)	
U	Thread depth (optional) U-999: inside thread with calculation U999: outside thread with calculation U not programmed: outside thread with calculation (outside: U = -0.6495 * F1) (inside: U = 0.6403 * F1)	
K1	Number of multiple threads (min: 1, max: 99) (optional)	
F1	Pitch	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

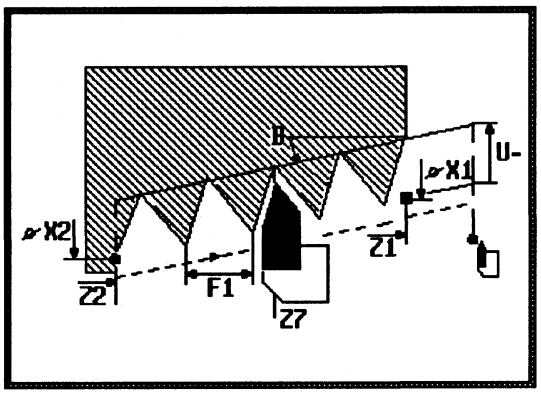
ERRORS:

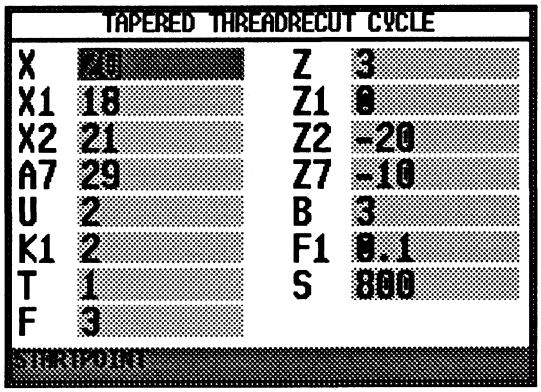
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)
O203	"Take over position" in A7 or Z7 is not possible when the spindle position is unknown.

14.44 TAPERED THREAD RE-CUT CYCLE G867 (Option)



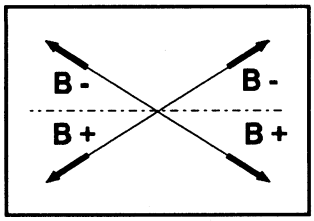
							GRUNDIG
		RECUT		TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

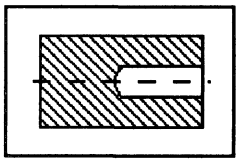
X,Z	Startpoint of the tool in the thread	(only Teach-In)
X1,Z1	Beginpoint of the thread contour	
X2,Z2	Endpoint of the thread contour (X2=optional)	
A7	Measured spindle angle (taken over with function key)	
Z7	Measured Z position in the thread (taken over with function key)	
U	Thread depth (default: see cylind. threadcutting) (optional)	
B	Taper angle (optional) -45 degr - +45 degr Use B instead of X1, or X2 Only possible X1,X2 or X1,B or X2,B	
K1	Number of multiple threads (optional)	
F1	Pitch	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)



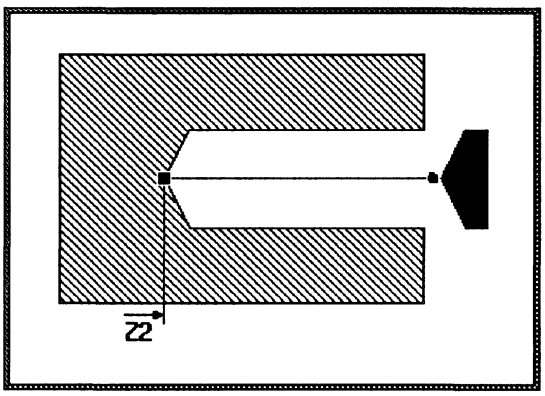
ERRORS:

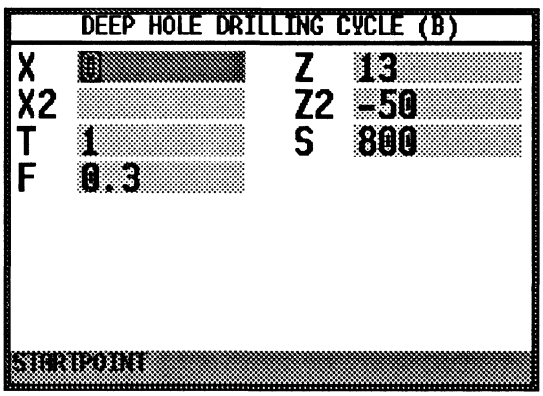
P01	Wrong combination of X1, X2, B
P02	One or more parameters are not programmed
P07	Taper angle > 45 degr or < -45 degr
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)
O203	"Take over position" in A7 or Z7 is not possible when the spindle position is unknown.

14.45 DEEP HOLE DRILLING CYCLE (BASIC) G870



							GRUNDIG
	EXTENDED			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





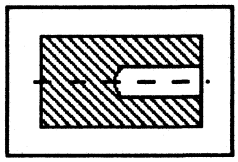
PARAMETERS:

X,Z	Startpoint of the drilling cycle	(only Teach-In)
Z2	Endpoint of the drilling cycle	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

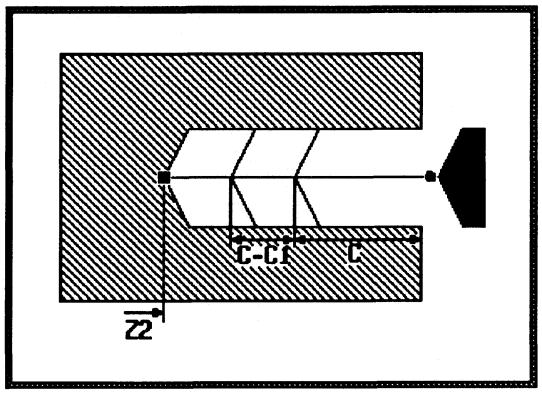
ERRORS:

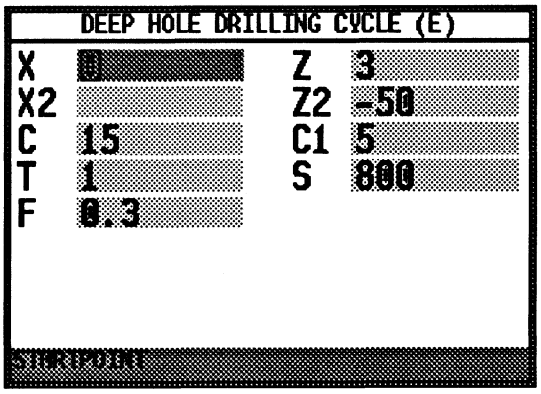
P02	One or more parameters are not programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.46 DEEP HOLE DRILLING CYCLE (EXTENDED) G871



							GRUNDIG
	EXTENDED			TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





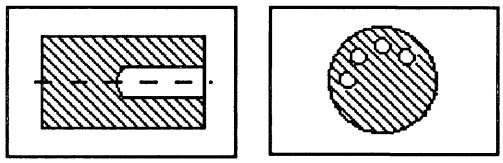
PARAMETERS:

X,Z	Startpoint of the drilling cycle	(only Teach-In)
Z2	Endpoint of the drilling cycle	
C	Cutting depth (optional)	
C1	Degression (optional)	
T	Toolnumber	(only Teach-In)
S	Spindle speed in rev/min or m/min	(only Teach-In)
F	Feed in mm/rev	(only Teach-In)

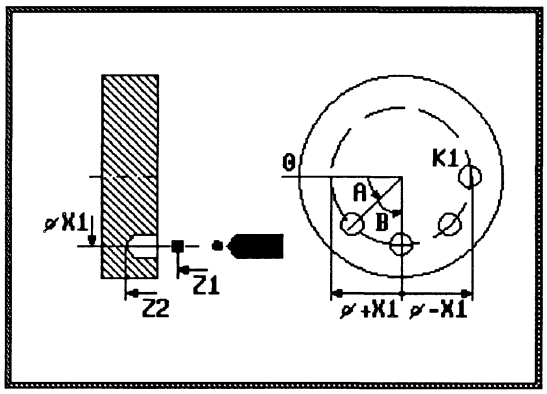
ERRORS:

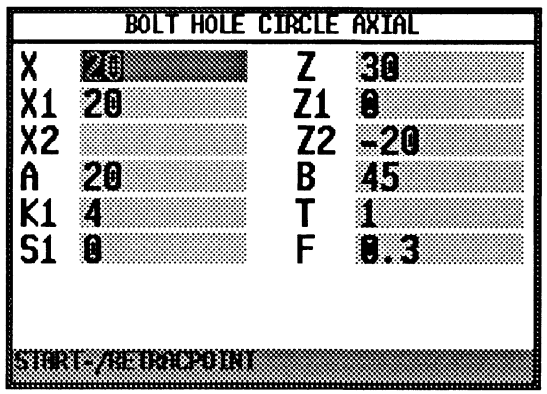
P02	C1 without C programmed
O135	Value < minimun (i.e T < 0)
O134	Value > maximum (i.e T > 48)

14.47 BOLT HOLE CIRCLE AXIAL G872 (Option)



							GRUNDIG
RADIAL				TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Cycle start/retrac position	(only Teach-In)
X1	Beginposition circle pattern (optional) (not programmed: = X)	
Z1	Beginposition for drilling (optional) (not programmed: = Z)	
Z2	Endposition hole(s) (optional) (not programmed: = Z1)	
A	Start angle of the circle (-360 - +360) (optional) (not programmed: = 0)	
B	Angle between the holes (-360 - +360) (optional) (not programmed: = 0)	
K1	Number of holes (max: 99) (optional) (not programmed: 1 hole)	
T	Toolnumber	(only Teach-In)
S1=	Speed rotating tool (rev/min)	
F	Feed in mm/rev	(only Teach-In)

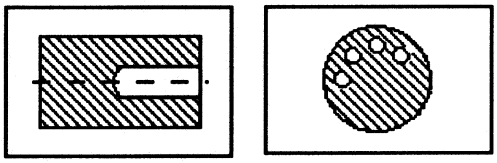
ERRORS:

O135 Value < minimun (i.e K1 < 1)
O134 Value > maximum (i.e K1 > 99)

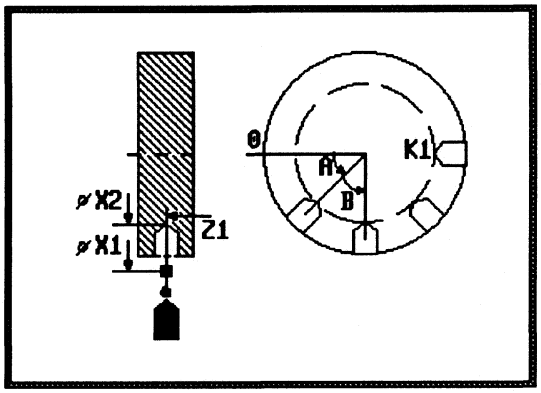
REMARK:

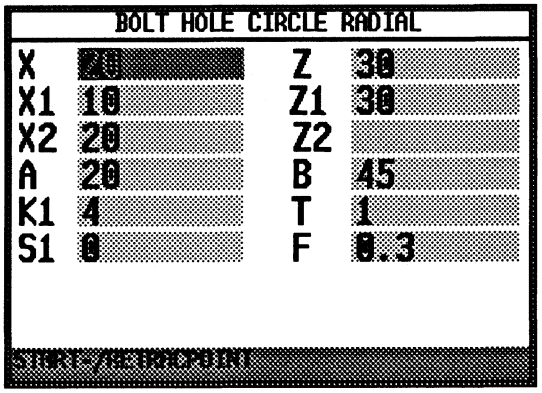
The cycle can be used for positioning the spindle. Enter only X and Z for this purpose. Machining can then be done with another cycle. Typical application is groove milling with cycles G830 - G837.

14.48 BOLT HOLE CIRCLE RADIAL G873 (Option)



							GRUNDIG
RADIAL				TAKE OVER POSITION	SPINDLE REV/MIN	ENTRY READY	RETURN





PARAMETERS:

X,Z	Cycle start/retrac position	(only Teach-In)
X1	Beginposition for drilling (optional) (not programmed: = X)	
Z1	Beginposition for circle pattern (optional) (not programmed: = Z)	
X2	Endposition hole(s) (optional) (not programmed: = X1)	
A	Start angle of the circle (-360 - +360) (optional) (not programmed: = 0)	
B	Angle between the holes (-360 - +360) (optional) (not programmed: = 0)	
K1	Number of holes (1 - 99) (optional) (not programmed: 1 hole)	
T	Toolnumber	(only Teach-In)
S1 =	Speed rotating tool (rev/min)	
F	Feed in mm/rev	(only Teach-In)

ERRORS:

O135 Value < minimun (i.e K1 < 1)
O134 Value > maximum (i.e K1 > 99)

15. : Appendix: Diagnostics

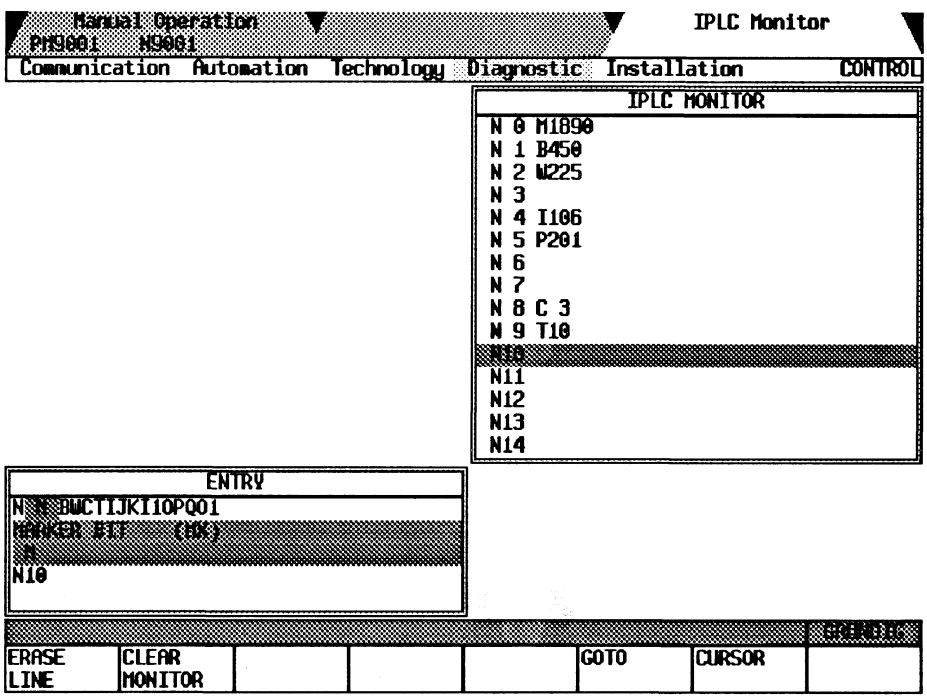
The MANUAL PLUS has a number of diagnostic facilities built in. The most obvious are the start-up diagnostics displayed during power-up. Further diagnostics are available in the CONTROL screen, under the Diagnostic option. Topics in the Diagnostic menu are:

IPLC Monitor
Configuration Check
Test Serial Port
Keyboard Test

15.1 The IPLC Monitor

The IPLC is the MANUAL PLUS internal logic controller. Responsible for interfacing between the MANUAL PLUS controller and the machine tool. The IPLC Monitor option enables the status of the IPLC inputs and outputs to be viewed.

The ENTRY window contains parameters which enables specific information to be viewed in the IPLC monitor. This information is entered and displayed in the block numbers in the IPLC MONITOR window.



IPLC MONITOR and ENTRY windows

15.2 Displaying the IPLC Monitor

The IPLC MONITOR and ENTRY windows in the CONTROL screen selects the IPLC information to be displayed. To see the actual status of the selected IPLC items you have to change to the OPERATE screen. Once Diagnostics has been selected the OPERATE screen displays the status of the selected IPLC items.
To clear the IPLC status displayed in the OPERATE screen you must go to the CONTROL screen and activate a different menu option.

15.3 Configuration Check

Entering this menu displays the controller functions in the Module group window. The larger information window displays the status of the diagnostics tests.

Use the cursor keys to high a function in the Module group window, then press ENTER. The results of the power-up diagnostics are then displayed in the information window for the highlighted module.

15.4 Test Serial Ports

The serial ports test is a loop test and requires special test plugs to be connected to the serial ports.

15.5 Key Board Test

The Keyboard test checks all the keys on the data entry panel. Highlight Keyboard test using the cursor keys, then press ENTER. The MANUAL PLUS displays an empty information window. Along the bottom of the screen one function key is active F2 START TEST. This function key is a toggle action, pressing once starts the test, pressing again stops the test.

Press F2 START TEST to active the keyboard test. As each key on the data entry panel is pressed its function is echoed in the information window. You must press F2 STOP TEST before you can leave the keyboard test.

15.6 Operator Machine Constants

This section gives the Operator Machine Constants. For a complete list of all machine constants see the MANUAL PLUS Installation Manual.

Operator Machine Constants control such items as the colour of the borders and communication parameters.

MACHINE CONSTANTS OVERVIEW

MC 0018 ACTIVE WINDOW BORDERCOLOR (0-15)
This machine constant changes the colour of the border around any active window.

Assign	1 for RED	9 for intensive RED
	2 for GREEN	10 for intensive GREEN
	3 for YELLOW	11 for intensive YELLOW
	4 for BLUE	12 for intensive BLUE
	5 for MAGENTA	13 for intensive MAGENTA
	6 for CYAN	14 for intensive CYAN
	7 for WHITE	15 for intensive WHITE
	8 for BLACK	0 for intensive BLACK

MC 0021 This machine constant should not be changed.

COMMUNICATION (Devices)

DEVICE 1

MC 0900 DEV1:CHANNEL (0=NONE,1=RS232C,2=RS422)
Sets the channel for synchronous peripheral 'device 1'.

Assign 0= Not present
1= RS232C
2= RS422

MC 0901 DEV1: BAUDRATE (110-19200)
Sets the send and receive baud rate for (device 1). Only 110, 300, 600, 1200, 2400, 4800, 9600 and 19200 baud rates can be used. Other values will give unpredictable results.

MC 0903 DEV1: NUMBER OF STOPBITS (1 OR 2)
The number of stop bits required by the external data peripheral.

Assign 1 for 1 stopbit
2 for 2 stopbits

MC 0904 DEV1: LEADER/TRAILER LENGTH (0-120)
Some peripherals (eg. paper tape devices) need an amount of NUL characters before and after a transmission.
This MC sets the number of NUL characters.

MC 0905 DEV1:DATA CARRIER (0=ASCII,1=ISO,2=EIA)
The CNC supports the use of three character sets, referred to as ASCII, ISO and EIA.

The code to be used for coding the data carrier information is given by the external data peripheral.

Assign 0 for data operation with ASCII-code
1 for data operation with ISO-code
2 for data operation with EIA-code

MC 0906 DEV1:AUTO CODE RECOGNITION (0=OFF 1=ON)
Enables the MANUAL PLUS to recognize the code sent by the data peripheral. The code is displayed and the data loaded, provided it is ASCII, ISO or EIA-code.
When the automatic code recognition is disabled, only data code in accordance with the code defined by MC905 will be read.

Assign 0 for no automatic recognition
1 for automatic recognition

MC 0907 DEV1: FLOW CONTROL (0=RTS,1=RTS-F,2=XON)
Sets of the flow control protocol.

Assign 0= RTS/CTS handshake. The MANUAL PLUS will toggle CTS for each character to be sent by the peripheral.

1= RTS/CTS flow control. The MANUAL PLUS will set CTS to inactive when its internal input buffer is almost full. The CTS line will be made active when enough room is available again. The peripheral must use a similar protocol when it sends to the MANUAL PLUS.

2= XON/XOFF flow control. The MANUAL PLUS will send an XOFF character when its internal input buffer is almost full. An XON character will be sent when enough room is available again. The peripheral must use a similar protocol when it sends to the MANUAL PLUS.

MC 0908 DEV1: CHECK DTR (0=NO, 1=YES)
The interface signal DTR can be used to check if a cable and/or a peripheral device is actually connected. The DTR signal must then be active, otherwise the MANUAL PLUS will not transfer data but error D41 will occur.

When MC908 is 1, the control circuit checks DTR continuously for an active level. If this is undesirable, the machine constant should be set to 0. This selects a simple three-wire connection, in which only the circuits TxD, RxD and GND are used. This should only be used in combination with the XON/XOFF protocol.

DEVICE 2

MC 0911 DEV2: BAUDRATE (110-19200)
Sets the send and receive baud rate for (device 2). Only 110, 300, 600, 1200, 2400, 4800, 9600 and 19200 baud rates can be used. Other values will give unpredictable results.

MC 0913 DEV2: NUMBER OF STOPBITS (1 OR 2)
The number of stop bits required by the external data peripheral.

Assign 1 for 1 stopbit
2 for 2 stopbits

MC 0914 DEV2: LEADER/TRAILER LENGTH (0-120)
Some peripherals (eg. paper tape devices) need an amount of NUL characters before and after a transmission.
This MC sets the number of NUL characters.

MC 0915 DEV2:DATA CARRIER (0=ASCII,1=ISO,2=EIA)
The MANUAL PLUS supports the use of three character sets, referred to as ASCII, ISO and EIA.

The code to be used for coding the data carrier information is given by the external data peripheral.

Assign 0 for data operation with ASCII-code
1 for data operation with ISO-code
2 for data operation with EIA-code

MC 0916 DEV2:AUTO CODE RECOGNITION (0=OFF 1=ON)
Enables the MANUAL PLUS to recognize the code sent by the data peripheral. The code is displayed and the data loaded, provided it is ASCII, ISO or EIA-code.
When the automatic code recognition is disabled, only data code in accordance with the code defined by MC915 will be read.

Assign 0 for no automatic recognition
1 for automatic recognition

MC 0917 DEV2: FLOW CONTROL (0=RTS,1=RTS-F,2=XON)
Sets of the flow control protocol.

Assign 0= RTS/CTS handshake. The MANUAL PLUS will toggle CTS for each character to be sent by the peripheral.

1= RTS/CTS flow control. The MANUAL PLUS will set CTS to inactive when its internal input buffer is almost full. The CTS line will be made active when enough room is available again. The peripheral must use a similar protocol when it sends to the MANUAL PLUS.

2= XON/XOFF flow control. The MANUAL PLUS will send an XOFF character when its internal input buffer is almost full. An XON character will be sent when enough room is available again. The peripheral must use a similar protocol when it sends to the MANUAL PLUS.

MC 0918 DEV2: CHECK DTR (0=NO, 1=YES)
The interface signal DTR can be used to check if a cable and/or a peripheral device is actually connected. The DTR signal must then be active, otherwise the MANUAL PLUS will not transfer data but error D41 will occur.

When MC918 is 1, the control circuit checks DTR continuously for an active level. If this is undesirable, the machine constant should be set to 0. This selects a simple three-wire connection, in which only the circuits TxD, RxD and GND are used. This should only be used in combination with the XON/XOFF protocol.

DEVICE 3

MC 0921 DEV3: BAUDRATE (110-19200)
Sets the send and receive baud rate for (device 3). Only 110, 300, 600, 1200, 2400, 4800, 9600 and 19200 baud rates can be used. Other values will give unpredictable results.

MC 0923 DEV3: NUMBER OF STOPBITS (1 OR 2)
The number of stop bits required by the external data peripheral.

Assign 1 for 1 stopbit
2 for 2 stopbits

MC 0924 DEV3: LEADER/TRAILER LENGTH (0-120)
Some peripherals (eg. paper tape devices) need an amount of NUL characters before and after a transmission.
This MC sets the number of NUL characters.

MC 0925 DEV3:DATA CARRIER (0=ASCII,1=ISO,2=EIA
The MANUAL PLUS supports the use of three character sets, referred to as ASCII, ISO and EIA.

The code to be used for coding the data carrier information is given by the external data peripheral.

Assign 0 for data operation with ASCII-code
1 for data operation with ISO-code
2 for data operation with EIA-code

MC 0926 DEV3:AUTO CODE RECOGNITION (0=OFF 1=ON)
Enables the MANUAL PLUS to recognize the code sent by the data peripheral. The code is displayed and the data loaded, provided it is ASCII, ISO or EIA-code.
When the automatic code recognition is disabled, only data code in accordance with the code defined by MC925 will be read.

Assign 0 for no automatic recognition
1 for automatic recognition

MC 0927 DEV3: FLOW CONTROL (0=RTS,1=RTS-F,2=XON)
Sets of the flow control protocol.

Assign 0= RTS/CTS handshake. The MANUAL PLUS will toggle CTS for each character to be sent by the peripheral.

1= RTS/CTS flow control. The MANUAL PLUS will set CTS to inactive when its internal input buffer is almost full. The CTS line will be made active when enough room is available again. The peripheral must use a similar protocol when it sends to the MANUAL PLUS.

2= XON/XOFF flow control. The MANUAL PLUS will send an XOFF character when its internal input buffer is almost full. An XON character will be sent when enough room is available again. The peripheral must use a similar protocol when it sends to the MANUAL PLUS.

MC 0928 DEV3: CHECK DTR (0=NO, 1=YES)
The interface signal DTR can be used to check if a cable and/or a peripheral device is actually connected. The DTR signal must then be active, otherwise the MANUAL PLUS will not transfer data but error D41 will occur.

When MC928 is 1, the control circuit checks DTR continuously for an active level. If this is undesirable, the machine constant should be set to 0. This selects a simple three-wire connection, in which only the circuits TxD, RxD and GND are used. This should only be used in combination with the XON/XOFF protocol.

LSV/2

LSV/2 is an synchronous data link protocol, to be used for point-to-point connections between computers. The purpose of this protocol is the reliable transfer of "messages" from the sending station to the receiving station. The "message" consists of a variable number of ASCII characters.

When the LSV/2 protocol is used (MC780 is set to 1) for communication, the following Machine Constants are important:

MC 0931 LSV/2 BAUDRATE (110-19200)
Sets the baudrate for the LSV/2 protocol.
Only the values 110, 300, 600, 1200, 2400, 4800, 9600 and 19200 are allowed.
Intermediate values will give unpredictable results.

MC 0932 LSV/2 CHARACTER SET (0=ASCII,1=ISO)

Assign 0 for data operation with ASCII-code
1 for data operation with ISO-code

MC 0933 LSV/2 TIMEOUT PERIOD (0-128S)
After the MANUAL PLUS has sent ENQ and after the transfer of the LSV/2 message DLE STX .....DLE ETX BCC, the MANUAL PLUS waits for an answer during the defined time out for DNC (from 0 to 128 sec.). When no answer is received within this time, ENQ is sent again.

MC 0934 LSV/2 NR.OF REPEATS (0=NO LIMIT,1-12)
The number of ENQ-repeats after the time out for DNC may be from 0 to 12.

MC 0935 LSV/2 DELAY TIME (0-128MS)
Waiting time for the MANUAL PLUS in milliseconds before sending data, after the last received character.
The delay is used when a MANUAL PLUS is connected to a MANUAL PLUS type prior to M600. In all other situations the delay value must be 0, because this will delay the communication.

MC 0936 LSV/2 CHECK DTR (0=NO, 1=YES)
The interface signal DTR can be used to check if a cable and/or a peripheral device is actually connected. The DTR signal must then be active, otherwise the MANUAL PLUS will not transfer data but report error D81.

To enable checking of the DTR line, set the MC to 1.

16. INDEX