



PROGRAMMING AND OPERATION MANUAL

MANUAL MACHINE PLUS

CENTUR LINE

T12021A

INDÚSTRIAS ROMI S/A

MARKETING DIVISION:
Rua Coriolano, 710 Lapa
05047-900 São Paulo - SP - Brazil
Phone (11) 3873-3388
Telex 1183922
Facsimile (11) 3865-9510

HEADQUARTERS:
Avenida Pérola Byington, 56 Centro
13453-900 Santa Bárbara D'Oeste - SP - Brazil
Phone (19) 3455-9000
Telex 191054
Facsimile (19) 3455-2499

CONTENTS

PART I - PROGRAMMING AND OPERATION

1 - INITIAL PROCEDURES	2
1.1 - TURNING MACHINE ON	2
1.2 - TURNING MACHINE OFF	2
1.3 - REFERENCING MACHINE	2
2 - STARTING MANUAL MACHINE PLUS	3
2.1 - GO INTO MANUAL MACHINE PLUS MODE	3
2.2 - EXITING MANUAL MACHINE PLUS MODE	3
2.3 - DESCRIPTION OF MANUAL MACHINE PLUS STATUS MODE	3
3 - AXES MANUAL MOVEMENT	5
3.1 - MOVING AXES THROUGH CONTINUOUS JOG (JOYSTICK)	5
3.2 - MOVING AXES THROUGH ELECTRONIC HANDWHEELS	5
4 - MANUAL MOVEMENT MODES	6
4.1 - DEFINING MANUAL MOVEMENT HEADING	6
4.1.1 - Informing tool	6
4.1.2 - Informing feed option	7
4.1.3 - Informing rotation	7
4.2 - PARALLEL TRAVERSING OF AXES MODE	8
4.2.1 - Defining movement limitation	8
4.3 - TAPER TURNING MODE	9
4.4 - RADIUS TURNING MODE	10
4.4.1 - Defining radius through Xf / Zf + R information	11
4.4.2 - Defining radius through Xc / Zc + R + α information	11
4.4.3 - Defining radius through Xf / Zf + Xc / Zc information	11
5 - TOOL REFERENCING (PRESET)	12
5.1 - TOOL REFERENCING (PRESET) ON X AXIS	12
5.2 - "Z" AXIS REFERENCING (PRESET) PREPARATION	13
5.3 - TOOL REFERENCING (PRESET) ON Z AXIS	13
5.4 - TOOL RADIUS AND QUADRANT (CUTTING SIDE)	14
6 - SET WORKPIECE ZERO POINT	15
7 - MANUAL OPERATING CYCLES	16

7.1 - PROGRAMMING AND EXECUTING CYCLES FOR MANUAL OPERATIONS	16
7.2 - HEADING FOR MANUAL OPERATING CYCLES	17
7.3 - TURNING CYCLES	18
7.3.1 - Roughing Cycle A	18
7.3.2 - Roughing Cycle B	19
7.3.3 - Roughing Cycle C	20
7.3.4 - Roughing Cycle D	22
7.3.5 - Roughing Cycle E	23
7.3.6 - Free Contour Cycle	24
7.4 - DRILLING CYCLES	28
7.4.1 - Drilling Cycle	28
7.4.2 - Thread Tapping Cycle	29
7.5 - GROOVING CYCLE	30
7.6 - THREAD CUTTING CYCLE	31
7.6.1 - THREAD REPAIR	32
8 - PROGRAM EDITION	33
8.1 - CREATING A NEW PROGRAM	33
8.2 - SELECTING A PROGRAM	35
8.3 - INSERTING BLOCK OF DATA IN PROGRAM	35
8.4 - DELETING PROGRAM BLOCK DATA	36
8.5 - CHANGING PROGRAM BLOCK DATA	36
8.6 - COPYING PROGRAM BLOCK DATA	36
8.7 - INSERTING TOOL CHANGE BLOCK IN PROGRAM	37
9 - TEACH-IN MODE PROGRAM	38
10 - GRAPHIC SIMULATION	39
10.1 - SIMULATING A MANUAL CYCLE	39
10.2 - SIMULATING A PROGRAM	40
10.3 - ZOOM IN AND ZOON OUT DISPLAY	40
10.4 - DELETE PROFILE IN DISPLAY WINDOW	40
11 - MANUAL CYCLES OR PROGRAMS EXECUTION	41
11.1 - EXECUTING A MANUAL CYCLE	41
11.2 - EXECUTING A PROGRAM	41
11.2.1 - Restarting during Program Execution	42

PART II - MANUAL MACHINE PLUS TUTORIAL

INTRODUCTION	45
---------------------------	-----------

1 - CREATING THE PROGRAM	47
1.1 - CREATING THE NEW PROGRAM:	47
2 - FACING TOOL THROUGH TEACH-IN	47
2.1 - SELECTING TOOL:	47
2.2 - FACING TOOL:	47
3 - ROUGHING EXTERNAL CONTOUR	47
3.1 - ACTIVATING FREE CONTOUR CYCLE:	47
3.2 - DEFINING PROFILE:	47
4 - FINISHING EXTERNAL CONTOUR	50
4.1 - SELECTING TOOL:	50
4.2 - ACTIVATING FREE CONTOUR CYCLE:	50
4.3 - DEFINING PROFILE:	50
5 - GROOVING	51
5.1 - SELECTING TOOL:	51
5.2 - GROOVING	51
6 - EXTERNAL THREAD CUTTING	52
6.1 - SELECTING TOOL:	52
6.2 - CUTTING THREAD	52
7 - DRILLING THE WORKPIECE	53
7.1 - SELECTING TOOL:	53
7.2 - CUTTING THREAD	53
8 - MACHINING THE EXTERNAL PROFILE	54
8.1 - SELECTING TOOL:	54
8.2 - MACHINING INTERNAL PROFILE:	54
9 - DEFINING WORKPIECE CHANGE POINT THROUGH TEACH-IN	55
9.1 - DEFINING WORKPIECE CHANGE POINT:	55
10 - SAVING PROGRAM	55
10.1 - SAVING PROGRAM:	55
11 - SIMULATING PROGRAM	56
11.1 - SELECTING PROGRAM:	56
11.2 - GRAPHICALLY SIMULATING:	56
11.3 - EXITING GRAPHIC SIMULATION:	56

12 - EXECUTING THE PROGRAM	56
12.1 - EXECUTING THE PROGRAM:	56
12.2 - EXITING EXECUTING MODE:	56

PART I

PROGRAMMING AND OPERATION

1 - INITIAL PROCEDURES

This chapter describes the initial procedures such as how to turn machine on / off and how to reference axes.

NOTE: The procedures comprised in this chapter may also be found on Operation chapter of Programming and Operating Manual - CNC Siemens 802D - Centur Line.

1.1 - TURNING MACHINE ON

- Turn main switch on, by positioning lever in “**ON**”.
- Press “**CNC ON**” button located on machine panel and await for starting procedure.
- Deactivate **emergency button**.
- **Close** machine door.
- Press “**MACHINE ON**” button.
- Reference machine (according to item 1.3).

1.2 - TURNING MACHINE OFF

- Press **emergency button**.
- Turn main switch off, by positioning lever in “**OFF**”

1.3 - REFERENCING MACHINE

- Press “**REF POINT**” key.
- Press “**CYCLE START**” button.

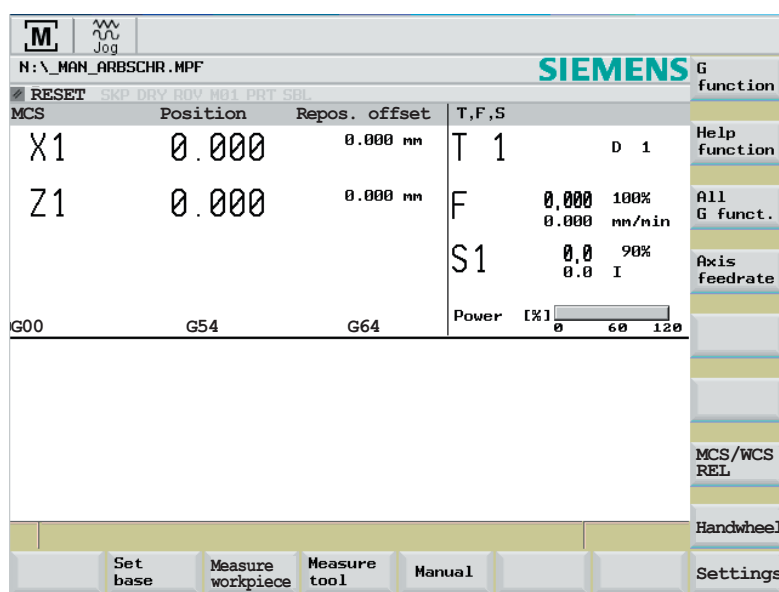


Figure 1-1. “POSITION” in CNC mode screen

2 -STARTING MANUAL MACHINE PLUS

This chapter shows Manual Machine Plus status screen as well as how to access it from programming and operating CNC mode.

2.1 - GOING INTO MANUAL MACHINE PLUS MODE

- Press “**JOG**” key
- Press [**MANUAL**] soft key.

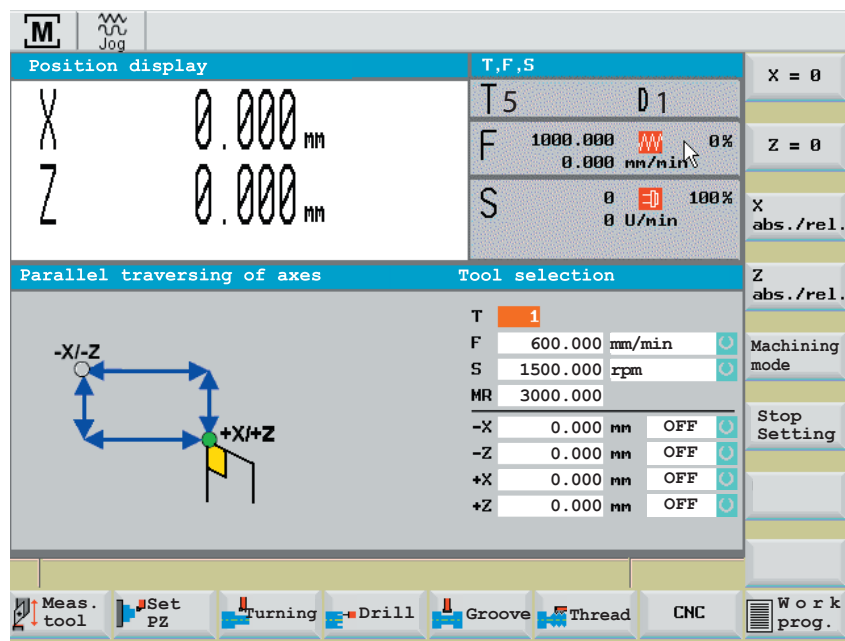


Figure 2-1. Manual Machine Plus Mode - Main Screen

2.2 - EXITING MANUAL MACHINE PLUS MODE

To exit Manual Machine Plus mode and return to CNC mode, proceed as follows:

- Press [**CNC**] soft key.

2.3 - DESCRIPTION OF MANUAL MACHINE PLUS STATUS SCREEN

The main screen for Manual Machine Plus, here called “status screen”, exhibits several information on current machine situation, that is, axes coordinate, rotation, feedrate, etc.

The meanings of main windows shown on this screen are described in the table below.

NOTE: The meaning of soft keys and “Parallel Traversing of Axes” will be shown in the respective chapters.

Display	Meaning
Values in front of X and Z axes (+/-)	- Current coordinate of respective axes
“S” window	- Programmed rotation value (RPM) or Cutting Speed (m/min) - Current percentage of spindle override switch
“F” window	- Programmed feed value (mm/rot or mm/min) - Current percentage of axes feedrate override switch
“T” window	- Tool number of the tool used
“D” window	- Tool offset applied
 [1] or  [10]	- Increment value for manual movement through electronic handwheels.
	- Indicates that axes are on rest position due to one of the factors: • Feedrate override is at position 0% • There is a machine alarm that blocks axes movement
  	- Spindle current status, respectively: • Spindle running clockwise (M03) • Spindle stopped (M05) • Spindle running counterclockwise (M04)

3 - AXES MANUAL MOVEMENT

This chapter describes the procedures related to axes manual movement either continuous or through electronic handwheels.

3.1 - MOVING AXES THROUGH CONTINUOUS JOG (JOYSTICK)

- Define the desired manual movement mode (see chapter 4).
- Place cursor on **subsequent field** to “F” field.
- Press  “**SELECT**” key to define feed option (**mm/min or mm/rot**)
- Place cursor on “**F**” field.
- **Enter** the desired **feed** value.
- Press “**INPUT**” key
- Place JOG **joystick** located at machine apron, on the desired position for moving (Z+, Z-, X+ or X-)

NOTE: For rapid traverse movement, press **RAPID** key, located at machine apron.

3.2 - MOVING AXES THROUGH ELECTRONIC HANDWHEELS

- Press, if necessary, the soft key [**Machining mode**] until manual movement mode “Parallel traversing of axes” is displayed (see more details on chapter 4)
- Press “**VAR**” key to select handwheel feed per pulse. [**1**] is equal to 0.001 and [**10**] is equal to 0.010 millimeter per pulse.
- **Rotate handwheels** located at machine apron to generate the desired movements.

NOTE: On machines equipped with Manual Machine Plus software, it is impossible to move axes with CNC panel electronic handwheel, because these machines have two handwheels in the electronic apron.

4 - MANUAL MOVEMENT MODES

This chapter describes the procedures related to the three modes of axes manual movement, which are called: Parallel traversing of axes, Taper turning and Radius turning.

4.1 - DEFINING MANUAL MOVEMENT HEADING

In all axes manual movement modes it is required to inform some data related to the machining process to execute, before effectively starting movement such as tool, rotation and feedrate.

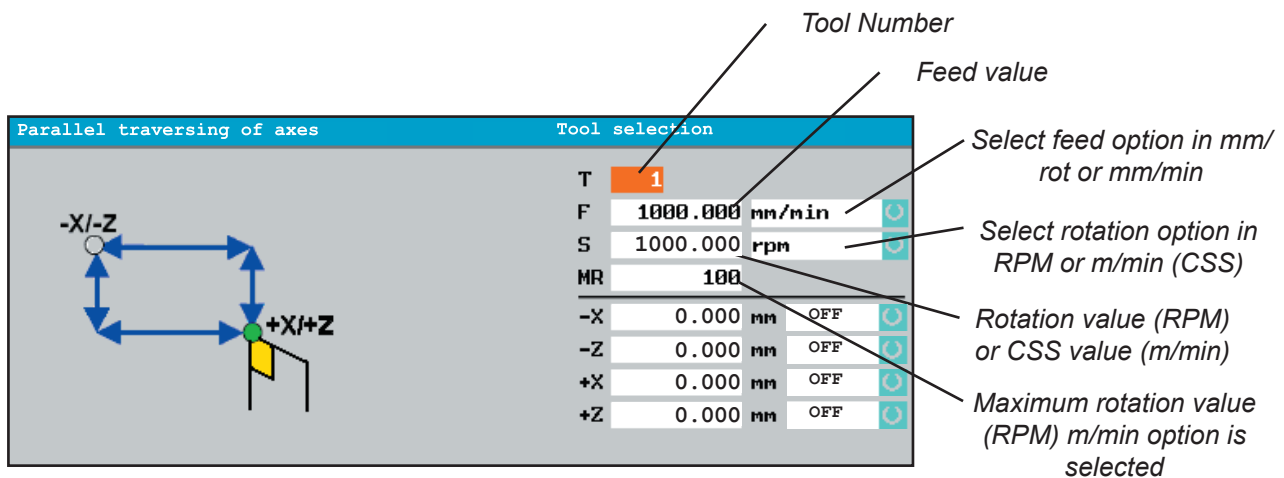


Figure 4-1. "Parallel traversing of axes" mode screen

4.1.1 - Informing tool

IMPORTANT: Before performing the procedure below, check if turret head is in a safe area where it can index with no collision with any machine devices (workpiece, chuck, tailstock, etc.).

To inform the tool to be used it is required to:

- Define the desired manual movement mode (see item 4.2)
- **Place** cursor on field related to the **T function**
- **Enter** the desired **tool** number. Example: 1 (for "T01")
- Press **"INPUT"** key (a message asking for tool change confirmation, according to figure 4-2 will be displayed)
- Press **"CYCLE START"** key for turret indexing and select the desired tool

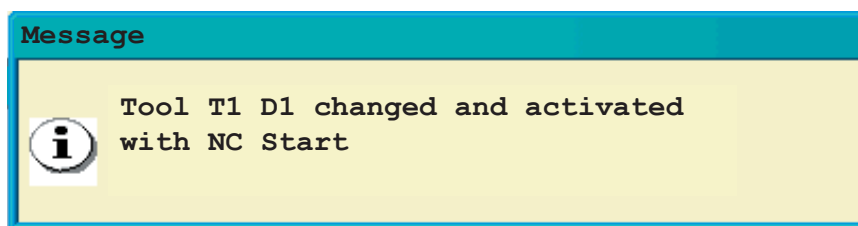


Figure 4-2. Tool change confirmation window

4.1.2 - Informing feed option

To inform feed to be used, it is required to:

- Define the desired manual movement mode (see item 4.2)
- Place cursor on **feed option** for selection (see figure 4.1)
- Press “**SELECT**” key to select the feed option to be used (mm/min or mm/rot).
Example: mm/rot
- Place cursor on field related to the **T function**
- **Enter** the desired **feed**, according to the selected feed. Example: 0.2 (0.2 mm/rot feed)
- Press “**INPUT**” key.

4.1.3 - Informing rotation

To inform rotation to be used, it is required to:

- Define the desired manual movement mode (see item 4.2)
- Place cursor on **feed option** for selection (see figure 4.1)
- Press “**SELECT**” key to select the rotation option to be used (RPM or m/min).
Example: RPM
- Place cursor on field related to the **S function**
- **Enter rotation** (RPM) or constant surface speed (m/min) desired value, according to rotation option selected. Example: 1500 (1500 RPM rotation)
- Press “**INPUT**” key

NOTE: If rotation option selected is m/min (CSS), it is also required to:

- Place cursor on **MR field**
- **Enter the maximum** admissible **rotation** value. Example: 3000 (3000 RPM maximum rotation)
- Press “**INPUT**” key

4.2 - PARALLEL TRAVERSING OF AXES MODE

This axes manual movement mode is used to execute orthogonal movements, that is, by using one axis at each time.

To use this resource it is required:

- Press, if necessary, the soft key [**Machining mode**] until manual movement mode “**Parallel traversing of axes**” is displayed
- **Fill in heading** data, according to item 4.1
- **Turn main spindle on**, if necessary, through lever located at electronic apron
- **Move axes** through Continuous Jog (see item 3.1) or electronic handwheels (see item 3.2)

4.2.1 - Defining movement limitations

Before moving axes, it is possible to preset a limit coordinate, either on negative or positive direction, so that it does not exceed this manual movement, for both Jog or electronic handwheels.

To use this resource it is required to:

- **Place cursor** on the field related to axis and direction where limitation will be defined. Example: -Z
- **Enter** the desired **limit** value: Example: -50 (for Z-50)
- Press “**INPUT**” key (the option “**ON**” will be displayed on the side field, which means that limit was activated)

NOTE: To deactivate limit for moving a certain axis, it is required to:

- Place **cursor** on field at the right of axis and direction where it is desired to deactivate limit, to which option “**ON**” is selected
- Press “**SELECT**” key to select the option “**OFF**”

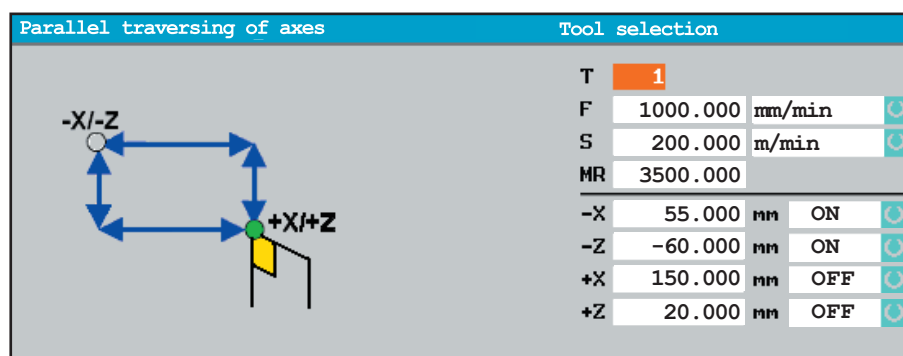


Figure 4-3. “Parallel traversing of axes” mode screen

4.3 - TAPER TURNING MODE

This axes manual movement mode is used to execute linear movements with two simultaneous axes, through previous information of the desired angle.

To use this resource it is required:

- Press soft key [**Machining mode**] until manual movement mode “**Taper turning**” is displayed
- **Fill in heading** data, according to item 4.1
- **Fill in**, if desired, **limitation** data for movement
- **Place** cursor on field “ α ”
- **Enter** the desired **angle** value for movement (see figure 4-4)
- Press “**INPUT**” key
- **Turn main spindle on**, through lever located at electronic apron
- **Move** axes through **JOYSTICK** located at electronic apron, reminding this positioning will indicate traversing direction (see figure 4-4)

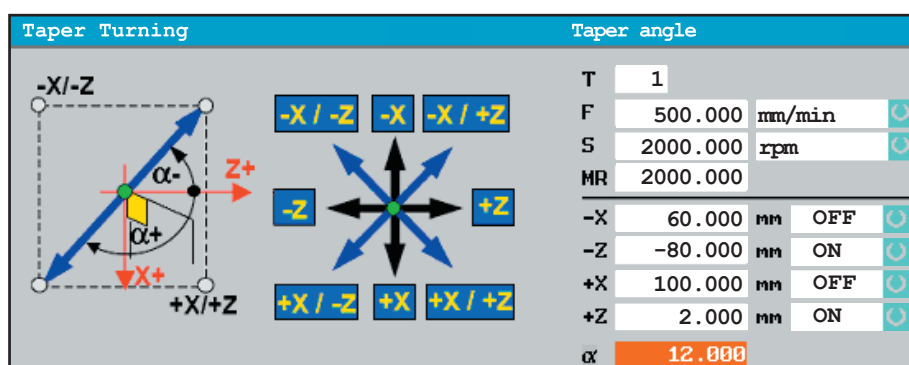


Figure 4-4. “Taper turning” mode screen

4.4 - RADIUS TURNING MODE

This axes manual movement mode is used to execute circular movements through previous information of desired radius or circle center coordinate.

To use this resource it is required:

- Press soft key [**Machining mode**] until manual movement mode “**Radius turning**” is displayed
- **Fill in heading** data, according to item 4.1
- **Place** cursor on field of **radius data** selection
- **Select** the desired information option (“ $X_f / Z_f + R$ ”, “ $X_c / Z_c + R + \alpha$ ” or “ $X_f / Z_f + X_c / Z_c$ ”)
- **Fill in** the required **fields**, according to selected radius information option, by pressing “**INPUT**” key to register data in the respective fields (see details on items 4.4.1, 4.4.2 and 4.4.3)
- **Place** cursor on the field that defines tool traversing direction (**clockwise or counterclockwise**)
- Press “**SELECT**” key to select axes traversing direction (**clockwise or counterclockwise**)
- **Turn main spindle on**, through lever located at electronic apron
- **Move** axes through **JOYSTICK** located at electronic apron, reminding this positioning will indicate traversing direction (see figure 4-5)

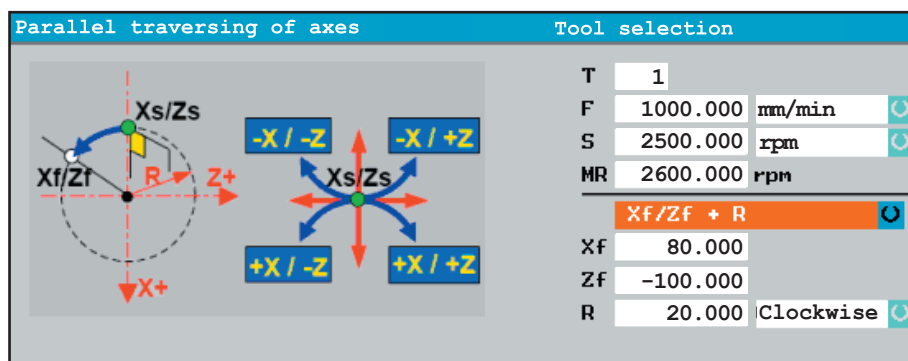


Figure 4-5. “Radius turning” mode screen

4.4.1 - Defining radius through Xf / Zf + R information

When selecting this type of information the following fields should be filled in:

Field	Description
Xf	X-axis Radius end coordinate
Zf	Z-axis Radius end coordinate in
R	Value of radius to be generated
CW / CCW	Axes traversing direction (clockwise or counterclockwise)

4.4.2 - Defining radius through Xc / Zc + R + α information

When selecting this type of information the following fields should be filled in:

Field	Description
Xc	X-axis Radius center coordinate
Zc	Z-axis Radius center coordinate
R	Value of radius to be generated
α	Radius total angular opening
CW / CCW	Axes traversing direction (clockwise or counterclockwise)

4.4.3 - Defining radius through Xf / Zf + Xc / Zc information

When selecting this type of information the following fields should be filled in:

Field	Description
Xf	X-axis Radius end coordinate
Zf	Z-axis Radius end coordinate
Xc	X-axis Radius center coordinate
Zc	Z-axis Radius center coordinate
CW / CCW	Axes traversing direction (clockwise or counterclockwise)

5 - TOOL REFERENCING (PRESET)

This chapter describes the procedures related to tool referencing, also known as “preset”, on X and Z axes.

5.1 – TOOL REFERENCING (PRESET) ON “X” AXIS

1) Manually withdraw turret to a safe position, to change the tool:

- Press “**JOG**” Key.
- Press “**[M] POSITION**” Key.
- Press “**[Machining Mode]**” Key.
- Press “**VAR**” Key to select the feed value by handwheels, **[1]**, that mean 1mm per turn or **[10]**, that mean 10mm per turn.
- Manually withdraw turret to a safe position, where it can index with no collision with any machine devices such as chuck, workpiece, taistock, etc.

2) Select the tool that will be done the reference:

- Place cursor corresponding to function “**T**” field.
- Enter number of tool. Example: 1 (for “**T01**”).
- Press “**INPUT**” key.
- Press “**CYCLE START**” Key to can index the turret and call the chose tool.

3) Touch the tool on knowed diameter and preset the tool:

- By handwheels touch the tool on knowed diameter (workpiece, chuck, etc), by using “**Parallel traversing of axes**” mode.
- Press “**[Meas Tool]**” soft key.
- Press “**[X]**” soft key.
- Place cursor corresponding to “**d1**” field.
- Enter diameter value on whih tool is touching. Example: **50**.
- Press “**INPUT**” key.
- Press “**[Set Length]**” soft key.
- Press “**[Cancel]**” soft key to exit.

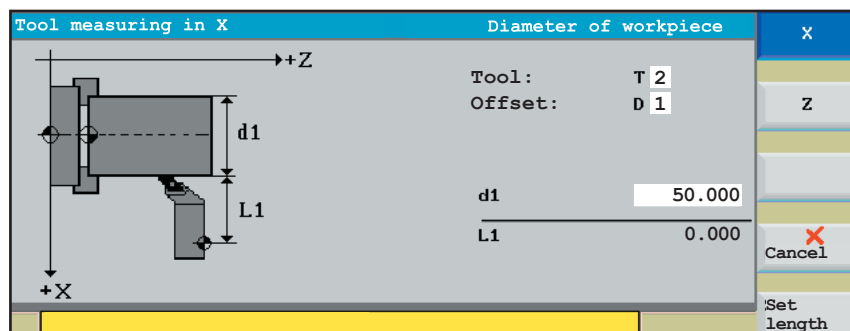


Figure 5-1. Tool referencing - X axis screen

5.2 “Z” AXIS REFERENCING (PRESET) PREPARATION

1) Manually withdraw turret to a safe position, to change the tool:

- Press “**JOG**” Key.
- Press “**[M] POSITION**” Key.
- Press “**[Machining Mode]**” Key.
- Press “**VAR**” Key to select the feed value by handwheels, **[1]**, that mean 1mm per turn or **[10]**, that mean 10mm per turn.
- Manually withdraw turret to a safe position, where it can index without collision with any machine devices such as chuck, workpiece, tailstock, etc.

2) Select a turret position without tool:

- Place cursor corresponding to function “**T**” field.
- Enter number of turret position without tool. Example: **5** (for “T05”).
- Press “**INPUT**” key.
- Press “**CYCLE START**” Key to can index the turret and call that position.

3) Referencing the turret, touching it on any face that will be used as reference. For example: chuck face, workpiece face, jaws face, etc.

- By handwheels, touch the turret face on the chose reference face.
- Press “**[SET Z0 WO]**” soft key.
- Press “**[Z=0]**” soft key.
- Press “**[Back]**” soft key.

5.3 TOOL REFERENCING (PRESET) ON “Z” AXIS

1) Manually withdraw turret to a safe position, to change the tool:

- Press “**JOG**” Key.
- Press “**[M] POSITION]**” Key.
- Press “**[Machining Mode]**” Key.
- Press “**VAR**” Key to select the feed value by handwheels, **[1]**, that mean 1mm per turn or **[10]**, that mean 10mm per turn.
- Manually withdraw turret to a safe position, where it can index without collision with any machine devices such as chuck, workpiece, tailstock, etc.

2) Select the tool that will be done the reference:

- Place cursor corresponding to function “**T**” field.
- Enter number of tool. Example: **1** (for “T01”).
- Press “**INPUT**” key.
- Press “**CYCLE START**” Key to can index the turret and call the chose tool.

3) Referencing the tool, touching it on any face that was used as reference. For example: chuck face, workpiece face, jaws face, etc.

- By handwheels, touch the tool on the chose reference face.
- Press “[**Meas Tool**]” soft key.
- Press “[**Z**]” soft key.
- Place cursor corresponding to “a1” field.
- Enter: **0**
- Press “**INPUT**” key.
- Press [**Set Length**] soft key
- Press [**Cancel**] soft key to exit.

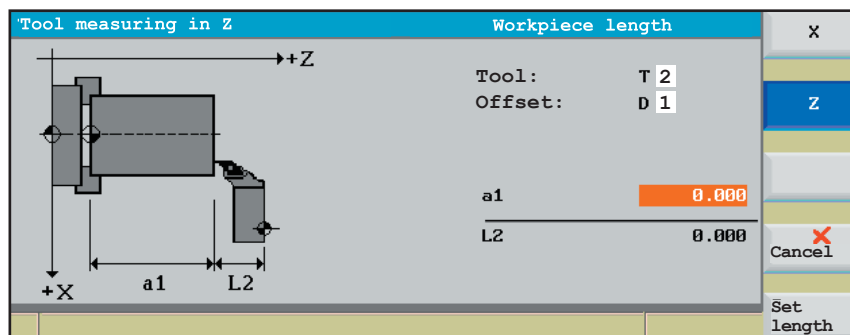


Figure 5-2. Tool referencing - Z axis screen

5.4 TOOL RADIUS AND QUADRANT (CUTTING SIDE)

Before presetting tool on “X” and “Z”, it is required to inform RADIUS and QUADRANT values in the offset of each one. For this it is required to:

- Press “**OFFSET PARAM.**” Key.
- Press “[**Tool list**]” soft key.
- Use cursor Keys (◀, ▶, ▲, ▼) to place cursor on tool “Radius” column (positioned in geometry table), which radius will be informed.
- Enter tool radius value. Example: **0.8**.
- Press “**INPUT**” key.
- Use cursor Keys (◀, ▶, ▲, ▼) to place cursor on tool field  whose cutting face will be informed.
- Enter cutting face value (according to item 7.4 – programming part of this manual). Example: **3** (right external turning).
- Press “**INPUT**” key.
- Press “[**M**] **POSITION**” Key to return to Manual Machine Plus software.

6 - SET WORKPIECE ZERO POINT

This chapter describes the procedure related to definition of the work coordinate system origin, known as workpiece zero.

IMPORTANT: Before performing this procedure, tool preset should be executed, according to chapter 5.

To set the workpiece zero point, proceed as follows:

- Go to “**Parallel traversing of axes**” mode (see item 4.2)
- Manually **withdraw turret** to a safe position, where it can index with no collision with any machine devices such as chuck, workpiece, tailstock, etc.
- **Index turret** using T field of “Parallel traversing of axes” mode (see item 4.1.1), by positioning it on the tool to be preset
- **Touch the tool** in a known diameter (workpiece, chuck, tailstock, etc), by using “Parallel traversing of axes” mode (see item 4.2)
- Press [**Set PZ**] soft key
- Press [**Z=0**] soft key (Z-axis absolute coordinate will shift to 0)
- Press [**Back**] soft key to exit this screen

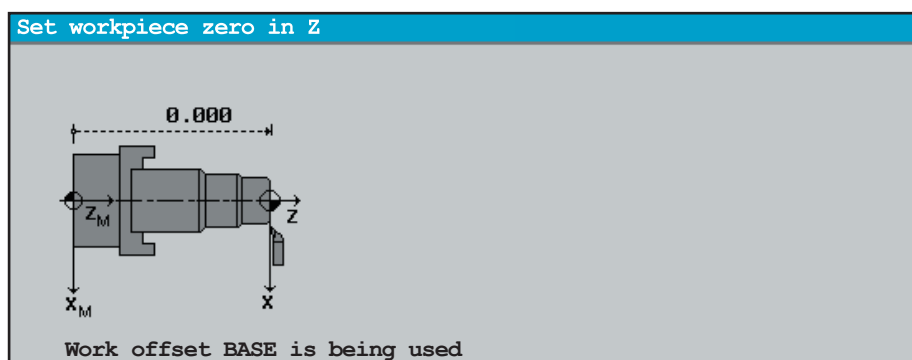


Figure 6-1. Set workpiece zero point - Z axis screen

7 - MANUAL OPERATION CYCLES

This chapter describes the procedures related to programming and execution of cycles that may be used for manual operations, that is, that will be immediately executed after their programming. They are: turning, grooving, threading, drilling and tapping cycles.

IMPORTANT: *In manual operation mode, cycles can not be saved within a program. To know how to use them in a worpiece program, refer to chapter 88.*

7.1 - PROGRAMMING AND EXECUTING CYCLES FOR MANUAL OPERATIONS

To operate these cycles, proceed as follows:

- Go to “**Parallel traversing of axes**” mode (see item 4.2)
- Manually **withdraw turret** to a safe position, where it can index with no collision with any machine devices such as chuck, workpiece, tailstock, etc.
- **Index turret** using T field of “Parallel traversing of axes” mode (see item 4.1.1), by positioning it on the tool to be preset
- **Approach tool** to workpiece, keeping a backlash on X and Z axes
- Press the **soft key** related to the desired **cycle**. Example: [Turning]
- Press, if necessary, the softkey related to the desired cycle. Example: [Roughing Cycle A]
- **Fill in** cycle **heading** data (see item 7.2)
- **Fill in** specific data of the selected **cycle** (see more details on next chapters)
- Graphically **Simulate** (see item 10.1)
- **Execute** cycle (see item 11.1)

NOTE: *To exit this cycle execution screen, press [**Cancel**] soft key*

7.2 - HEADING FOR MANUAL OPERATIONS CYCLES

When selecting one of cycles for manual operations, it is necessary to fill in a heading which is common for all of them.

See below a cycle example and data description related to their heading.

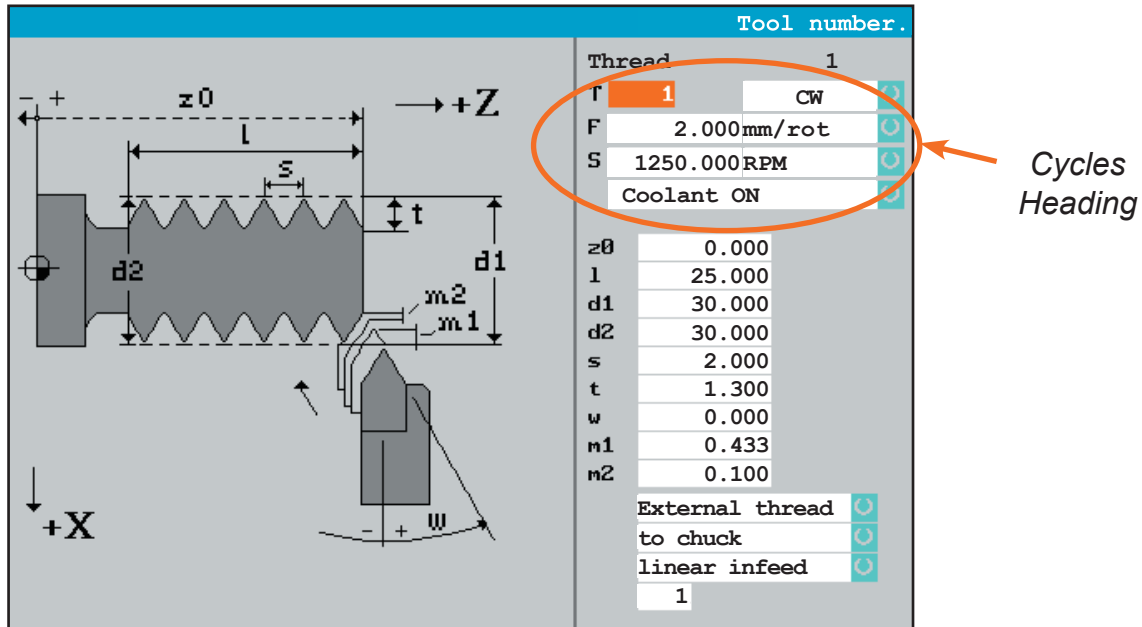


Figure 7-1. Threading cycle screen

Details on the arguments:

Argument	Description
T	Number of the tool to be used
D	Number of tool offset
CW or CCW	Spindle rotation direction - clockwise or counterclockwise respectively
F	Feedrate value in mm/min or mm/rot
mm/min or mm/rot	Define if feedrate (F) will be programmed in millimeter/minute or millimeter/rotation
S	Rotation value (RPM) or constant spindle speed (m/min)
RPM or m/min	Define programming through fixed rotation or constant spindle speed, respectively
Coolant OFF or Coolant ON	Operate with cutting coolant OFF or ON, respectively

7.3 - TURNING CYCLES

Turning cycles are composed by six canned cycles which are applied to parallel and transversal turning machining: Roughing A, Roughing B, Roughing C, Roughing D, Roughing E and Free Contour.

7.3.1 - Roughing Cycle A

Roughing Cycle A can be accessed through the following path:

- Press [**Turning**] soft key
- Press [**Roughing Cycle A**] soft key

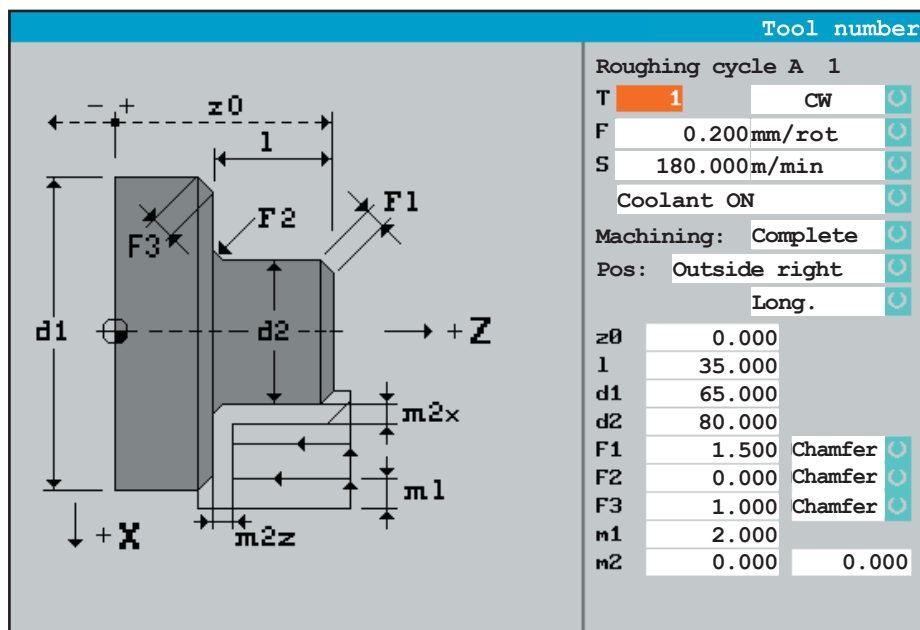


Figure 7-2. Roughing cycle type A screen

Details on the arguments:

Argument	Description
Roughing or Finishing or Complete	Defines which turning phase will be performed: roughing, finishing or both (complete)
Outside right or Inside right or Outside left	Defines machining type and cutting direction: outside and at chuck direction, inside and at chuck direction or outside and at chuck opposite direction, respectively
Long. or face	Defines cutting direction: parallel to Z axis or parallel do X axis, respectively
z0	Absolute coordinate of workpiece face (Z axis)
l	Total length of Z-axis turning from z0 (without signal)
d1	Workpiece outer diameter (absolute)

Argument	Description
d2	Workpiece inner diameter (absolute)
F1	Chamfer or radius value between face and workpiece inner diameter
F2	Chamfer or radius value between workpiece inner diameter and final Z
F3	Chamfer or radius value between final Z and workpiece inner dia.
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively
Chamfer or Radius	Defines if field F1, F2 or F3 will be a chamfer or radius, respectively

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.3.2 - Roughing Cycle B

Roughing Cycle B can be accessed through the following path:

- Press [**Turning**] soft key
- Press [**Roughing Cycle B**] soft key

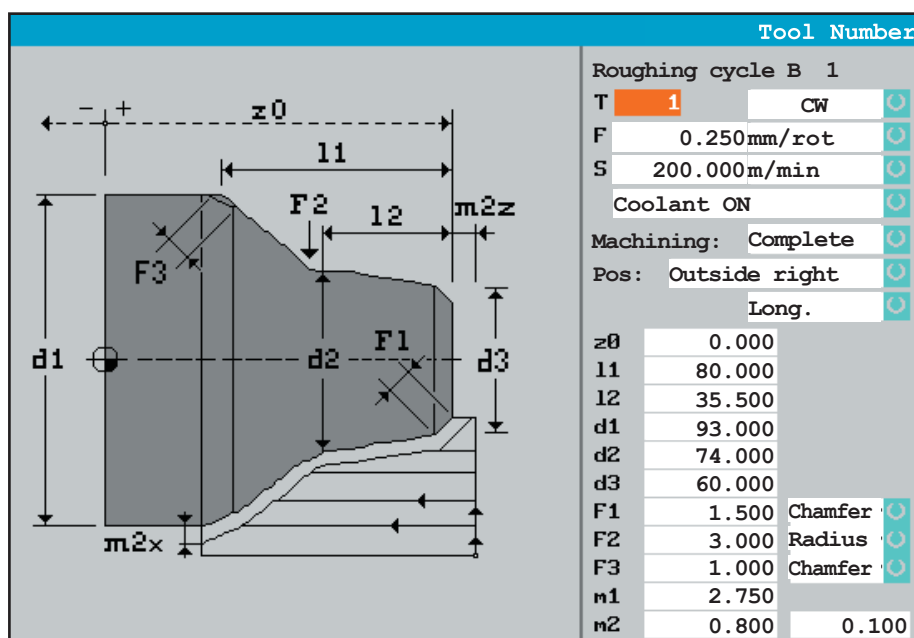


Figure 7-3. Roughing Cycle B screen

Details on the arguments:

Argument	Description
Roughing or Finishing or Complete	Defines which turning phase will be performed: roughing, finishing or both (complete)
Outside right or Inside right or Outside left	Defines machining type and cutting direction: outside and at chuck direction, inside and at chuck direction or outside and at chuck opposite direction, respectively
Long. or face	Defines cutting direction: parallel to Z axis or parallel do X axis, respectively
z0	Absolute coordinate of workpiece face (Z axis)
l1	Total length of Z-axis turning from z0 (without signal)
l2	Length of first linear segment from z0 (Z axis - without signal)
d1	Workpiece outer diameter (absolute)
d2	Workpiece intermediate point (absolute)
d3	Workpiece inner diameter (absolute)
F1	Chamfer or radius value between face and workpiece first linear segment
F2	Chamfer or radius value at intermediate point
F3	Chamfer or radius value between final Z and workpiece outside dia.
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively
Chamfer or Radius	Defines if field F1, F2 or F3 will be a chamfer or radius, respectively

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.3.3 - Roughing Cycle C

Roughing Cycle C can be accessed through the following path:

- Press [**Turning**] soft key
- Press [**Roughing Cycle C**] soft key

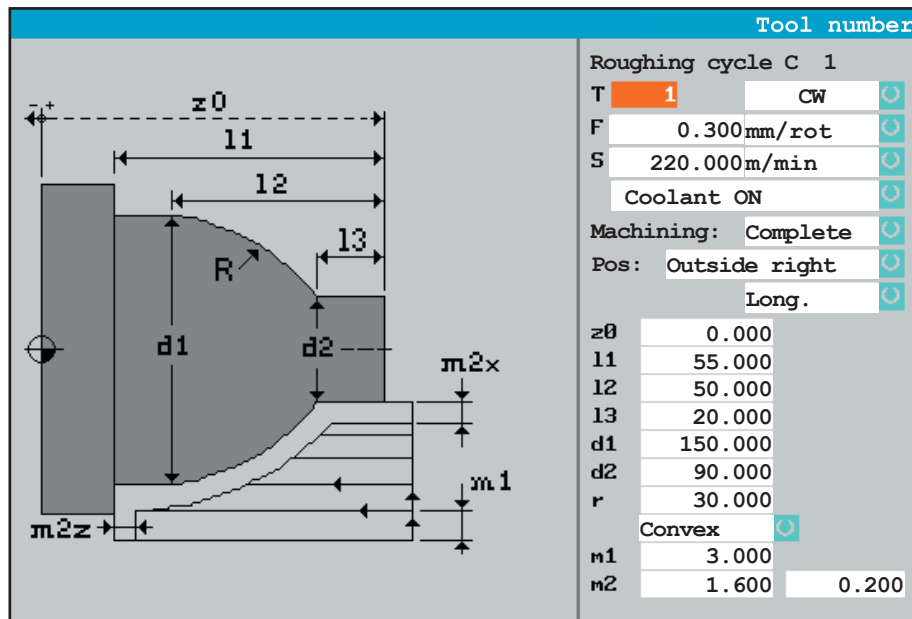


Figure 7-4. Roughing Cycle C screen

Details on the arguments:

Argument	Description
Roughing or Finishing or Complete	Defines which turning phase will be performed: roughing, finishing or both (complete)
Outside right or Inside right or Outside left	Defines machining type and cutting direction: outside and at chuck direction, inside and at chuck direction or outside and at chuck opposite direction, respectively
Long. or face	Defines cutting direction: parallel to Z axis or parallel do X axis, respectively
z0	Absolute coordinate of workpiece face (Z axis)
l1	Total length of Z-axis turning from z0 (without signal)
l2	Length of radius end point from z0 (without signal)
l3	Length of first linear segment from z0 (Z axis - without signal)
d1	Workpiece outer diameter (absolute)
d2	Workpiece inner diameter (absolute)
r	Radius value
Concave or Convex	Defines if radius will be concave or convex, respectively
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.3.4 - ROUGHING CYCLE D

Roughing Cycle D can be accessed through the following path:

- Press [**Turning**] soft key
- Press [**Roughing Cycle D**] soft key

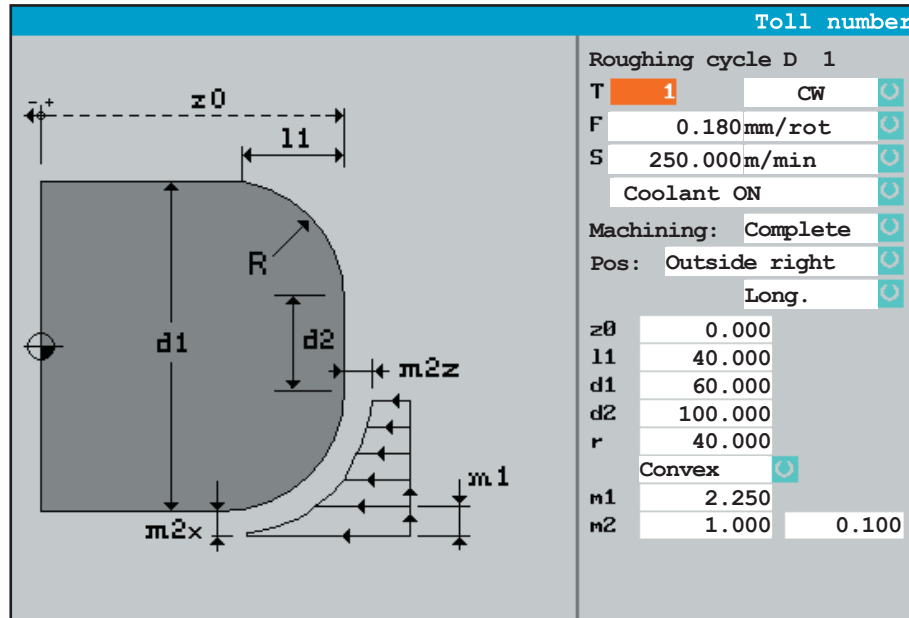


Figure 7-5. Roughing Cycle D Screen

Details on the arguments:

Argument	Description
Roughing or Finishing or Complete	Defines which turning phase will be performed: roughing, finishing or both (complete)
Outside right or Inside right or Outside left	Defines machining type and cutting direction: outside and at chuck direction, inside and at chuck direction or outside and at chuck opposite direction, respectively
Long. or face	Defines cutting direction: parallel to Z axis or parallel do X axis, respectively
z0	Absolute coordinate of workpiece face (Z axis)
l1	Length of radius end point from z0 (without signal)
d1	Workpiece outer diameter (absolute)
d2	Workpiece inner diameter (absolute)
r	Radius value
Concave or Convex	Defines if radius will be concave or convex, respectively
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.3.5 - Roughing Cycle E

Roughing Cycle E can be accessed through the following path:

- Press [**Turning**] soft key
- Press [**Roughing Cycle E**] soft key

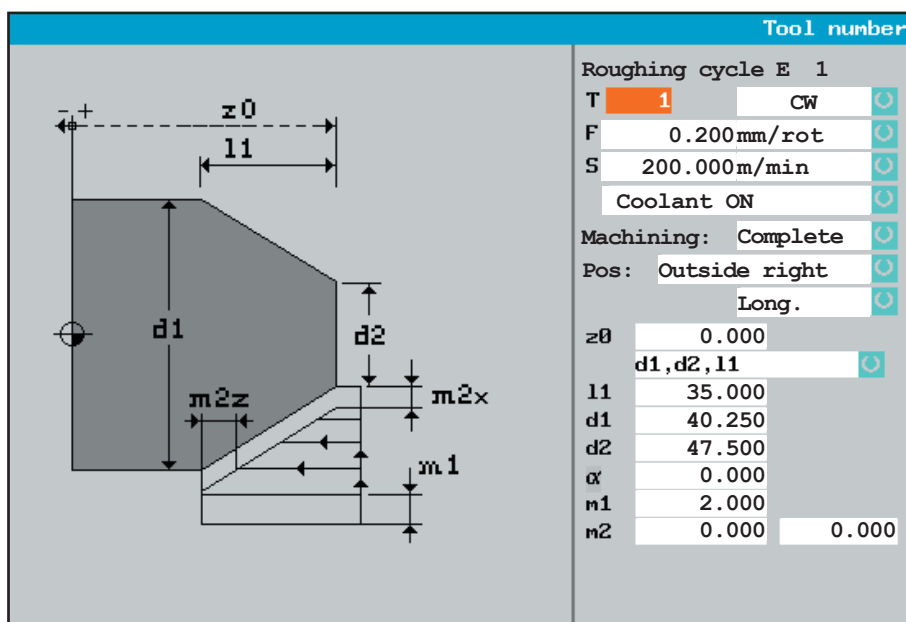


Figure 7-6. Roughing Cycle E screen

Details on the arguments:

Argument	Description
Roughing or Finishing or Complete	Defines which turning phase will be performed: roughing, finishing or both (complete)
Outside right or Inside right or Outside left	Defines machining type and cutting direction: outside and at chuck direction, inside and at chuck direction or outside and at chuck opposite direction, respectively
Long. or face	Defines cutting direction: parallel to Z axis or parallel do X axis, respectively
z0	Absolute coordinate of workpiece face (Z axis)
l1	Total length of Z-axis turning from z0 (without signal)

Argument	Description
d1, d2, l1 or angle, d1, l1, or ...	Defined which information will be used to generate the taper: d1, d2 and l1 or Angle, d1 and l1 or Angle, d2 and l1 or Angle (d1), d1 and d2 or Angle (d2) , d1 and d2
d1	Workpiece outer diameter (absolute)
d2	Workpiece inner diameter (absolute)
α	Angle value, when d1, d2 or l1 is omitted
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.3.6 - Free Contour Cycle

Free Contour Cycle can be accessed through the following path:

- Press [**Turning**] soft key
- Press [**Free contour**] soft key

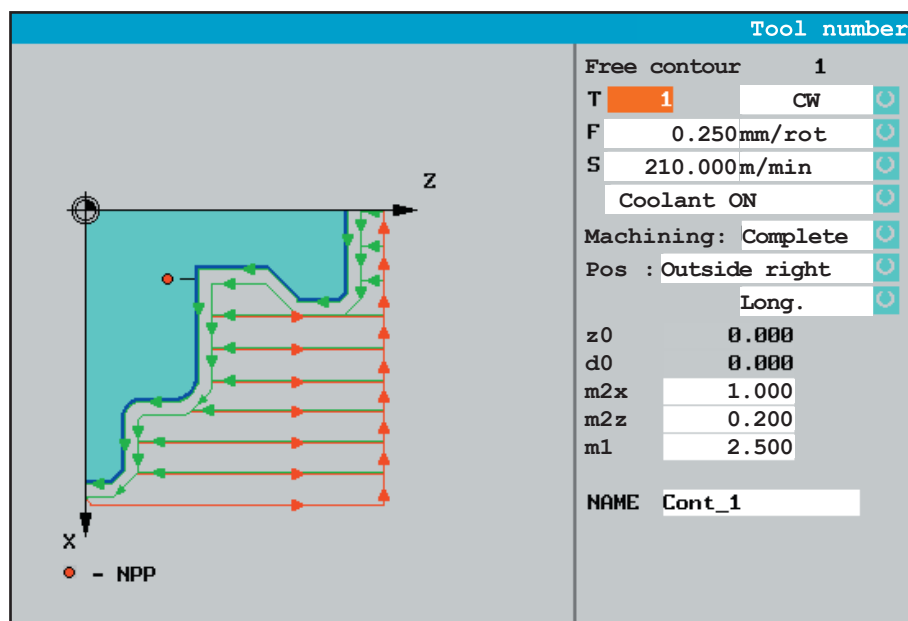


Figure 7-7. Free Contour Cycle Screen

Details on the arguments:

Argument	Description
Roughing or Finishing or Complete	Defines which turning phase will be performed: roughing, finishing or both (complete)
Outside right or Inside right or Outside left	Defines machining type and cutting direction: outside and at chuck direction, inside and at chuck direction or outside and at chuck opposite direction, respectively
Long. or face	Defines cutting direction: parallel to Z axis or parallel do X axis, respectively
z0	Absolute coordinate of workpiece face (Z axis). This value is automatically loaded.
d0	Workpiece outer diameter (absolute). This value is automatically loaded.
m2x	X-axis finishing allowance
m2z	Z-axis finishing allowance
m1	Maximum infeed depth (radius value)
NAME	Name of profile where contour will be defined. This field is automatically filled in.

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.3.6.1 - Defining machining profile

To access screen where machining profile will be defined, it is required to:

- Fill in data related to Contour Cyle (see item 7.3.6)
- Press [**Machined contour**] soft key
- Draw the profile according to the example below:

Example of profile definition:

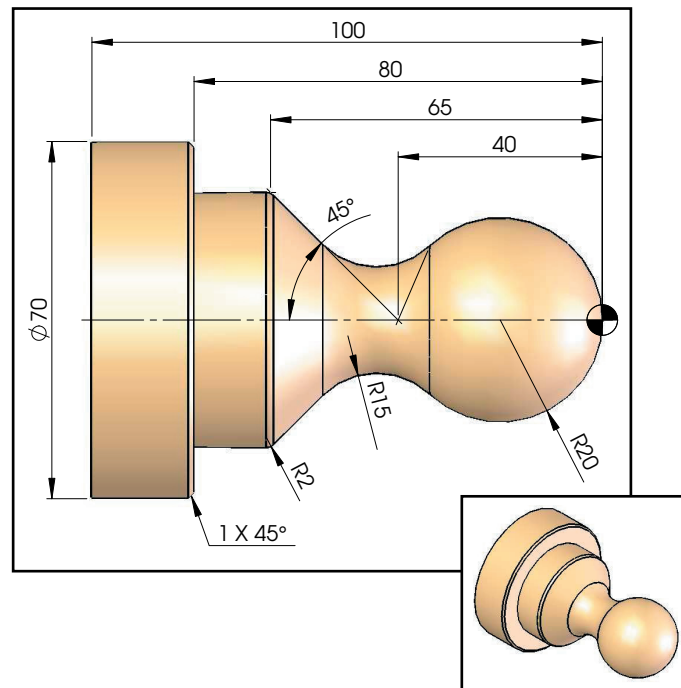


Figure 7-8. Example of profile definition

a) Reference point:

- Select / inform data:
Programmed level: G18
Diameter (DIAMON)
X 0.000 ABS
Z 0.000 ABS
Start point approach: G1
- Press [Accept element]

b) Contour drawing:

- Press soft key 
- Select option 
- Enter data:
R 20.000
X 0.000 ABS
Z -40.000 ABS
RD 12.000
- Press [Accept element]
- Press soft key 
- Enter data:
Z -65.000 ABS
 $\alpha 1$ 135.000 °
RD 2.000
- Press soft key [Accept element]

- Press soft key 
- Enter data:
Z -80.000 ABS
- Press soft key [Accept element]
- Press soft key 
- Enter data:
X 70.000 ABS
FS 2.000
- Press soft key [Accept element]
- Press soft key 
- Enter data:
Z -2.000 INC
- Press soft key [Accept element]
- Press soft key [Transfer]

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.4 - DRILLING CYCLES

Turning cycles are composed by two canned cycles for drilling and tapping operations.

7.4.1 - Drilling Cycle

Drilling cycle can be accessed through the following path:

- Press [**Drill**] soft key
- Press soft key [**Drilling centered**]

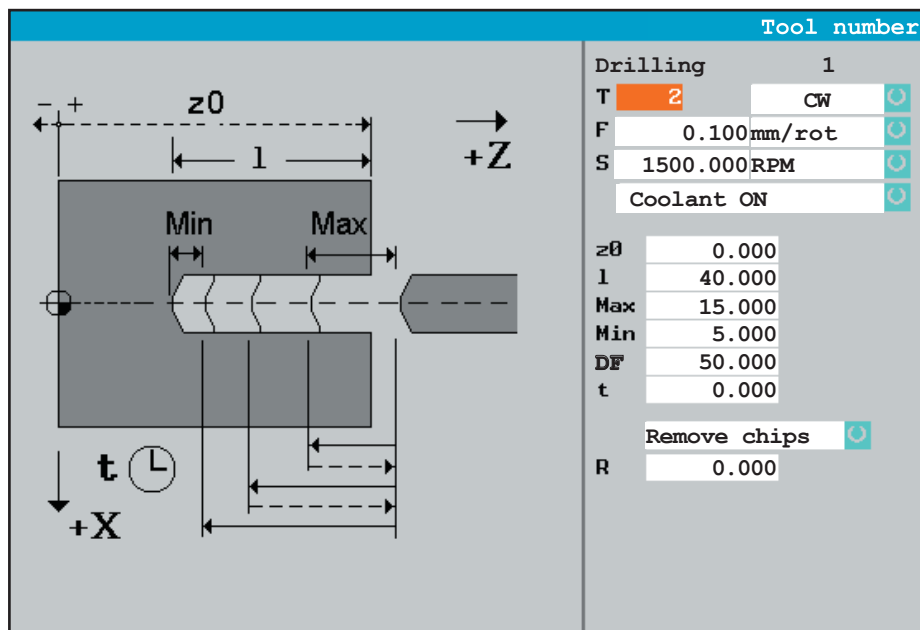


Figure 7-9. Drilling Cycle Screen

Details on the arguments:

Argument	Description
z0	Absolute coordinate of workpiece face (Z axis)
l	Total length of drilling from z0 (without signal)
Max	Maximum infeed depth for the first infeed
Min	Minimum infeed depth for subsequent infeeds
DF	Degression factor for infeed depth from second infeed, being: 1 = 100% (equal infeeds), 0.5 = 50% (second infeed is 50% of first infeed value and so on, until maximum infeed is reached (Min)
t	Dwell time on reaching the drilling depth (seconds)
Chip removal or Chip breaking	Defines how drilling will be executed: through chip removal or chip breaking process
R	Return travel distance for chip breaking. This field is not shown if "Remove chips" option is selected

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.4.2 - Thread Tapping Cycle

Thread Tapping Cycle can be accessed through the following path:

- Press [**Drill**]
- Press [**Tapping**] soft key

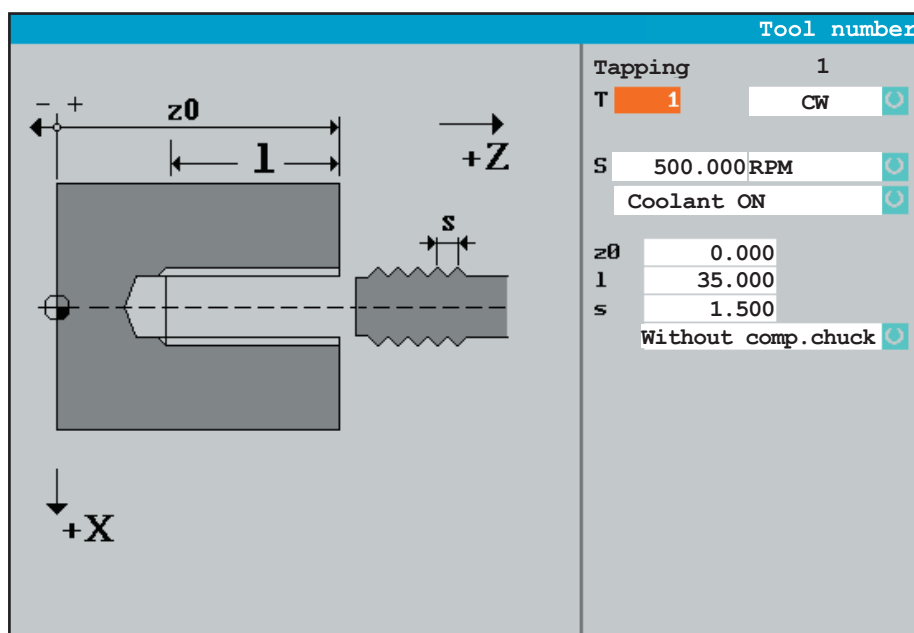


Figure 7-10. Thread Tapping Cycle Screen

Details on the arguments:

Argument	Description
z0	Absolute coordinate of workpiece face (Z axis)
l	Total drilling depth from z0 (without signal)
s	Thread pitch (mm/rotation)
without comp.chuck or with comp. chuck	Defines if threading will be executed with rigid tap or floating tap, respectively

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.5 - GROOVING CYCLE

Grooving cycle is a canned cycle for internal or external grooving operations, single or multiple equidistant grooves.

IMPORTANT: To use this cycle tool must have two tool offsets (D1 and D2). Tool references (preset) in X axis are equal on two offsets and different in Z axis, and the first corresponds to the tool left corner reference and the second one corresponds to the right corner reference.

Grooving cycle can be accessed through the following path:

- Press [**Groove**]

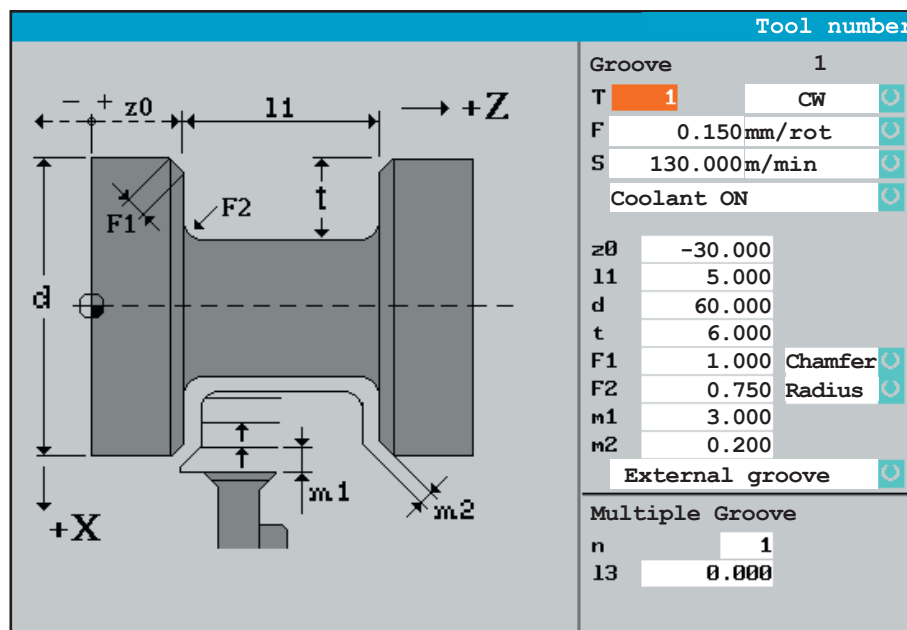


Figure 7-10. Grooving Cycle Screen

Details on the arguments:

Argument	Description
z0	Absolute coordinate of first groove left side (Z axis)
l1	Groove width
d	Absolute coordinate of groove starting diameter (X axis)
t	Groove depth from “d” diameter (radius - without signal)
F1	Chamfer or radius value in groove starting diameter
F2	Chamfer or radius value in groove ending depth
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively
Chamfer or Radius	Defines if F1 or F2 will be a chamfer or radius respectively
External groove or Internal groove	Defines if groove will be external or internal respectively
n	Defines the amount of groove to be executed
l3	Distance among groove, in case of multiple grooves

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.6 - THREAD CUTTING CYCLE

Thread cutting is a canned cycle for internal or external threads.

Thread Cutting Cycle can be accessed through the following path:

- Press [**Thread**] soft key

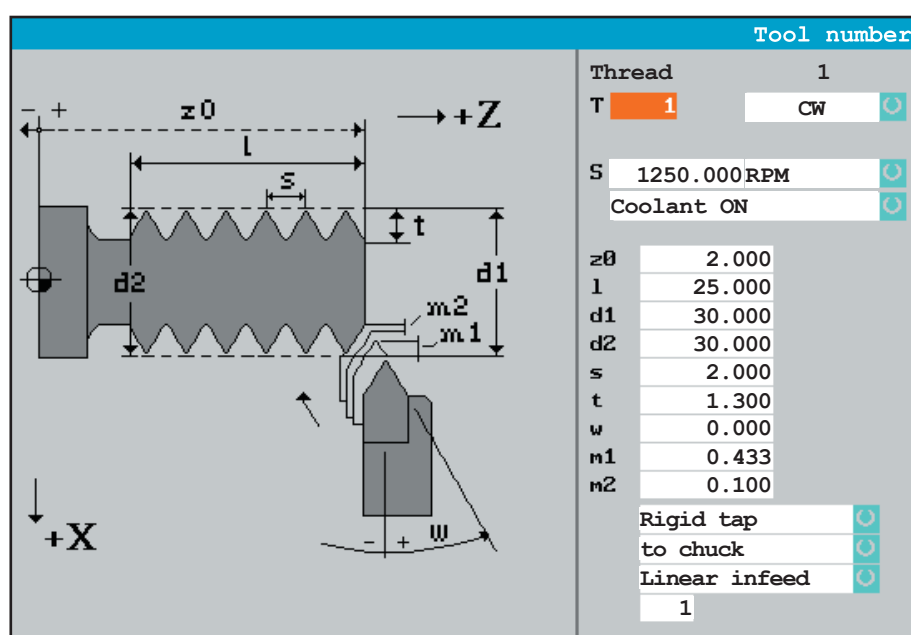


Figure 7-11. Thread cutting cycle screen

Details on the arguments:

Argument	Description
z0	Absolute coordinate of thread start position (Z axis)
l	Total thread length from z0 (without signal)
d1	Absolute coordinate of thread starting diameter on z0 point (X Axis)
d2	Absolute coordinate of thread starting diameter on l point (X axis)
s	Thread pitch (mm/rotation). In case of multiple start thread, pitch should be multiplied by the number of thread starts
t	Thread lead height (radius)
w	Angle

Argument	Description
m1	Maximum infeed depth (radius value)
m2	Finishing allowance on X and Z, respectively
External or Internal	Defines if groove will be external or internal, respectively
To chuck or From chuck	Defines if cycle will be for a single groove (n=1) or multiple grooves (n = 2, 3, ...)
Linear infeed or Degressive infeed	Distance between grooves, in case of multiple grooves
Number of thread starts	Defines if thread will be of one (1) or multiple starts (2 or more)

NOTES:

- To exit this screen, press [**Cancel**] soft key
- To simulate and execute this cycle, procedures described on chapter 9 and 10 should be respectively performed.

7.6.1 - THREAD REPAIR

This resource is used when it is required to repair a certain thread, despite this workpiece has been removed from and hold again into the machine.

To perform Thread Repair, proceed as follows:

- Go to “**Parallel traversing of axes**” mode (see item 4.2)
- Manually **withdraw turret** to a safe position, where it can index with no collision with any machine devices such as chuck, workpiece, tailstock, etc.
- **Index turret** using T field of “Parallel traversing of axes” mode (see item 4.1.1), by positioning it on the thread tool to be used
- **Place tool** inside one of the thread fillets, by using thr “Parallel traversing of axes” mode (see chapter 4.2)
- Press [**Thread**] soft key
- **Fill in data** according to previous item (7.6)
- Press [**Thread repair**] soft key
- Press [**Accept angle**] soft key
- Press [**OK**] soft key
- **Place tool outside thread**, keeping a backlash on X and Z axes, by using “Parallel traversing of axes” mode (see item 4.2)
- Press “**CYCLE START**” key

8 - PROGRAM EDITION

This chapter describes the procedures related to workpiece program edition.

IMPORTANT: *Unlike manual operation mode, cycles mentioned in this chapter can be saved in machine memory.*

8.1 - CREATING A NEW PROGRAM

To create a new program, proceed as follows:

1. Press [**Work prog.**] soft key
2. Press [**File**] soft key
3. Press [**Open**] soft key
4. Place cursor on field "**Name**"
5. **Enter** a **name** for the new program. Example: WORKPIECE_01
6. Press [**OK**] soft key. A prompt window appears as shown in the figure 8-1:
7. Press [**OK**] soft key

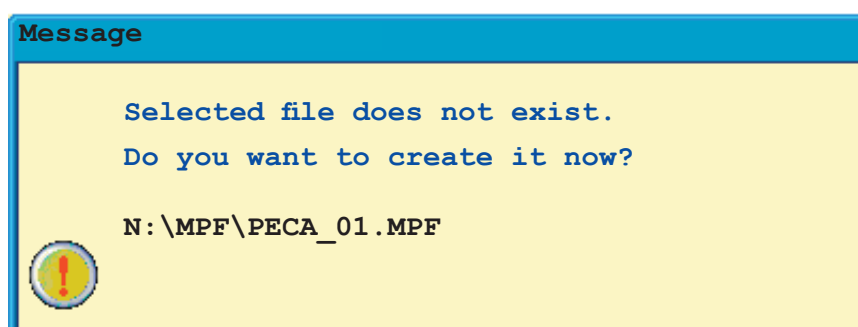


Figure 8-1. Prompt window for confirming program creation

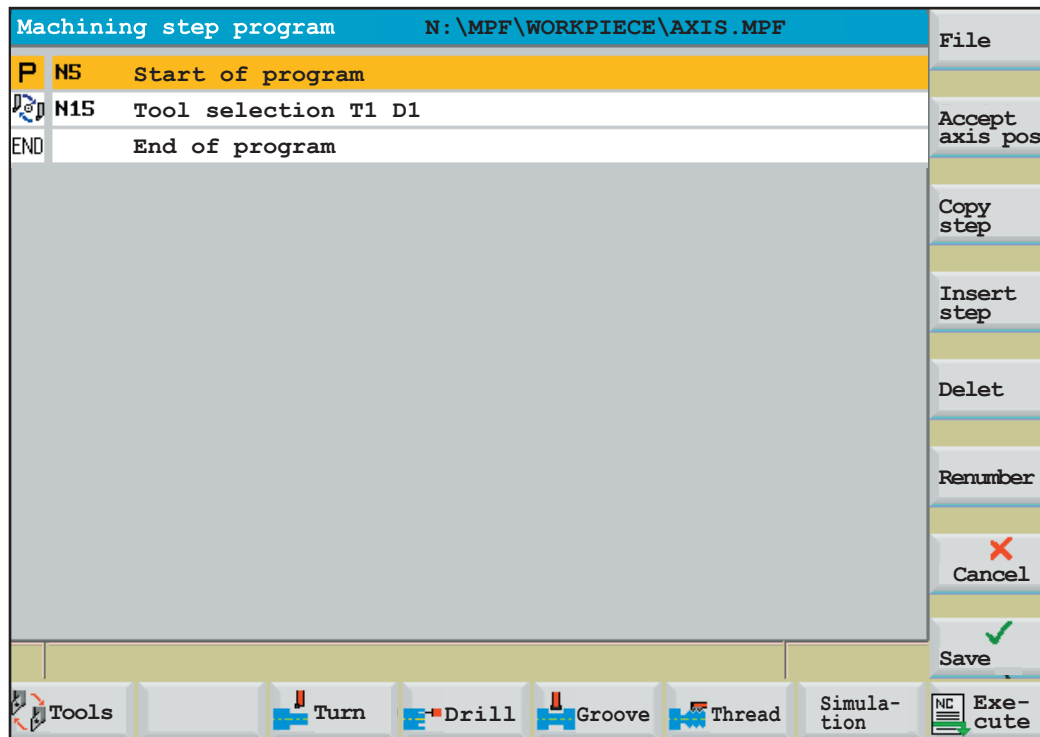


Figure 8-2. Program edition screen

NOTES:

- All programs are started by a tool selection block which is automatically inserted, taking into consideration the tool positioned on turret when program is created.
- To create a program inside a preexisting folder, soon after inserting program name (item 5), it is required to proceed as follows:
 - Press “**TAB**” key
 - **Place** cursor on the desired **folder**
 - Press “**INPUT**” key
 - **Continue** the above procedure from **item 6**
- Folders (directories) can only be created in CNC mode. To know how to created them, refer to Programming and Operating Manual - CNC Siemens 802D - Centur Line.
- When finishing program, press [**SAVE**] soft key to exit edition screen
- Programs created within Manual Machine Plus mode can not be edited in CNC mode.

8.2 - SELECTING A PROGRAM

To select a program already existing in machine memory, proceed as follows:

- Press [**Work prog.**] soft key
- Press [**File**] soft key
- Press [**Open**] soft key
- Press “**TAB**” key
- Place cursor on folder where the program is contained
- Press “**INPUT**” key
- **Place** cursor on desired program
- Press [**OK**] soft key

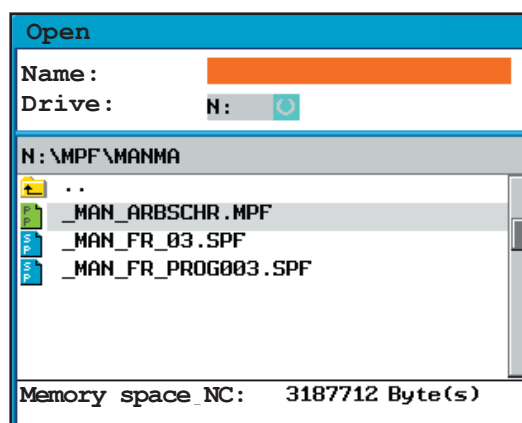


Figure 8-3. Window for program creation or selection

8.3 - INSERTING BLOCK OF DATA IN PROGRAM

To insert a block of data in an already existing program in machine memory, proceed as follows:

- **Select** the desired **programa** (see item 8.2)
- **Place** cursor on **previous block** to where a new program block will be inserted
- Press **soft key** related to the desired **cycle**. Example: [Turning]
- Press, if necessary, the softkey related to the desired cycle. Example: [Roughing cycle A]
- **Fill in** data of cycle **heading** (see item 7.2)
- **Fill in** specific **data** of selected **cycle** (see more details on items 7.3, 7.4, 7.5 and 7.6)
- Press [**OK**] soft key

8.4 - DELETING PROGRAM BLOCK DATA

To delete a block of data in an already existing program in machine memory, proceed as follows:

- Select the desired program (see item 8.2)
- Place cursor on block to be deleted
- Press [**Delete**] soft key

8.5 - CHANGING PROGRAM BLOCK DATA

To change a block of data in an already existing program in the machine memory, proceed as follows:

- Select the desired program (see item 8.2)
- Place cursor on the block to be changed
- Press key  (cursor at right)
- Change the desired information
- Press [**OK**] soft key

8.6 - COPYING PROGRAM BLOCK DATA

To copy a block of data in an already existing program in machine memory, proceed as follows:

- Select the desired program (see item 8.2)
- Place cursor on block to be copied
- Press soft key [**Copy step**]
- Place cursor on previous block to where copied block will be pasted
- Press [**Insert step**] soft key

8.7 - INSERTING TOOL CHANGE BLOCK IN PROGRAM

To insert a tool change block in an already existing block, proceed as follows:

- Select the desired program (see item 8.2)
- Place cursor on previous block to where tool change block will be inserted
- Press [**Tools**] soft key
- Enter number of the desired tool. Example: 2 (for T02)
- Press “**INPUT**” key
- Place cursor on field **G75**
- Press “**SELECT**” key to activate or deactivate axes automatic movement to tool change point before indexing turret
- Press [**Transfer.**] soft key
- Press [**Cancel**] soft key

NOTE: When finishing program, [**SAVE**] soft key must be pressed to exit this edition screen.

Tool selection						
T 6 D 1 ☐ ☒ G75						
Type	T	D _z	Length 1	Length 2	Radius	Free Angle
	1	1	0.000	6.019	0.000	0.000
	2	1	0.000	0.000	0.000	0.000
	6	1	0.000	0.000	0.000	0.000
	9	1	0.000	0.000	0.000	0.000

Figure 8-4. Tool selection screen

9 - TEACH-IN MODE PROGRAM

Teach-in is the programming mode where operator manually executes movements, registering them in the machine, thus creating a program.

To use teach-in mode, proceed as follows:

1. Select the desired program (see item 8.2)
2. Place cursor on previous block to where blocks to be elaborated will be inserted through teach-in mode
3. Press [**Turning**] soft key
4. Press [**Teach In**] soft key
5. Fill in heading data of the desired manual movement mode, according to item 4.1
6. Turn main spindle on through lever located at electronic apron
7. **Move axes** through one of manual movement modes (parallel traversing of axes, taper turning or radius turning), according to chapter 4
8. Press [**Save block**] soft key
9. Press [ **Save**] soft key to save as cutting movement (G01, G02 or G03), or [ **Save**] to save as rapid movement (G00)

Note: Execute items 7, 8 and 9 according to the number of manual operations to be performed.

NOTE: To exit teach-in mode, press [**Finish Teach-in**] soft key.

10 - GRAPHIC SIMULATION

Graphic simulation is a resource used to check if programmed profile is equivalent to the desired workpiece.

10.1 - SIMULATING A MANUAL CYCLE

To graphically simulate a manual cycle, it is required to:

- **Fill in** the necessary data of the desired **manual cycle** (see chapter 7)
- Press [**Simulation**] soft key
- Press [**Zoom Auto**] soft key
- Press “**CYCLE START**” key

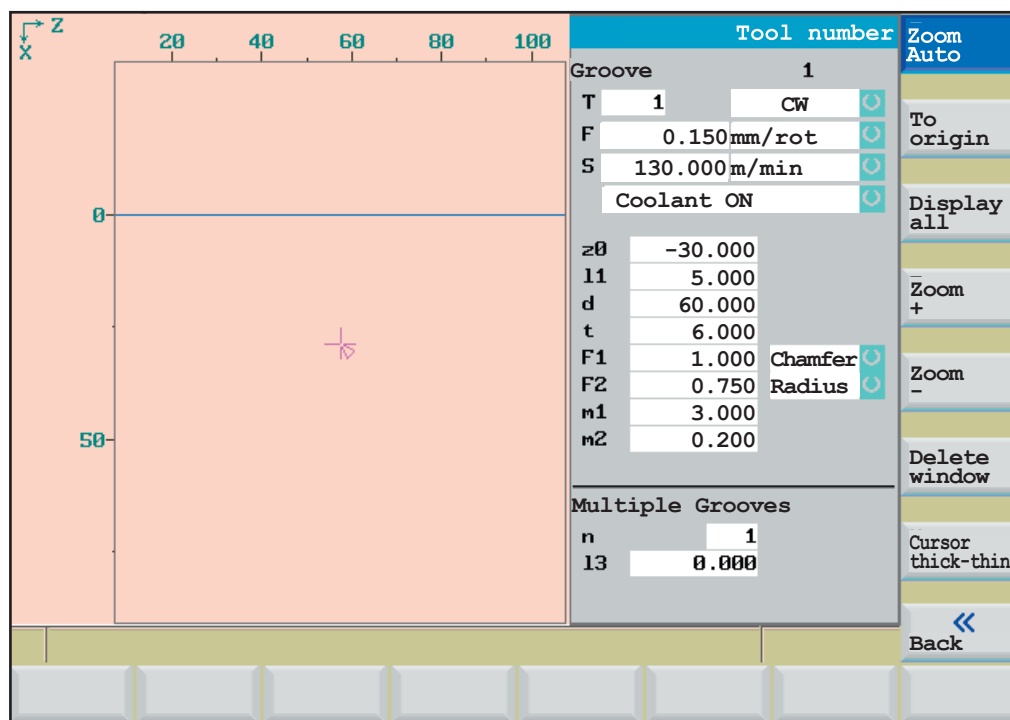


Figure 10-1. Manual cycle simulation

NOTE: To exit graphic simulation mode, press [**Back**] soft key.

10.2 - SIMULATING A PROGRAM

To graphically simulate a program, it is required to:

- **Select** the desired **program** (see item 8.2)
- Press [**Simulation**] soft key
- Press [**Zoom Auto**] soft key
- Press “**CYCLE START**” key

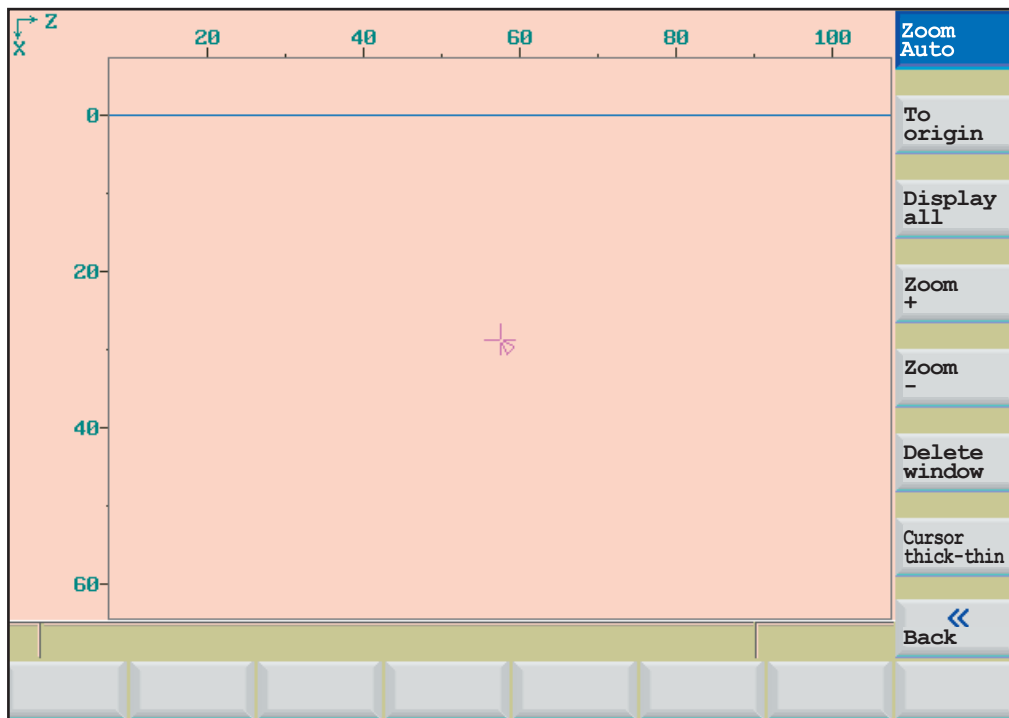


Figure 10-2. Machining Simulation

NOTE: To exit graphic simulation, press [**Back**] soft key.

10.3 - ZOOM IN AND ZOOM OUT DISPLAY

To enlarge or reduce workpiece display scale on graphic screen, proceed as follows:

- **Simulate program** according to the previous procedure (chapter 10)
- **Place** cursor on the **area** where scale will be modified
- Press soft key [**Zoom +**] to enlarge or [**Zoom -**] to reduce scale

10.4 - DELETE PROFILE IN DISPLAY WINDOW

To delete a profile on graphic screen display window, proceed as follows:

- Press [**Delete window**]

11 - MANUAL CYCLES OR PROGRAMS EXECUTION

IMPORTANT: Before executing a manual cycle or program, make sure they were already duly simulated, tools and workpiece were referenced and jaws were duly prepared.

11.1 - EXECUTING A MANUAL CYCLE

To execute a manual cycle, it is required to:

- **Fill in** necessary data of desired **manual cycle** (see chapter 7)
- Press [**OK**] soft key
- Press “**CYCLE START**” key

NOTES:

- To exit manual cycle execution mode, press [**Cancel**] soft key
- To execute “single block” cycle, press “**Single Block**”.

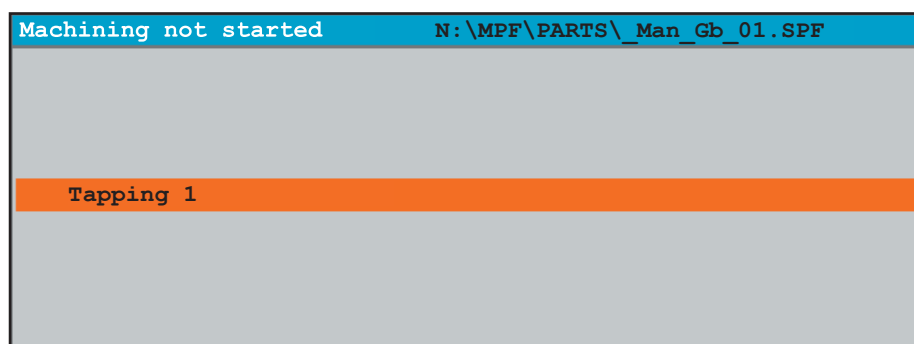


Figure 11-1. Window for Manual cycle execution

11.2 - EXECUTING A PROGRAM

To execute a program it is required to:

- **Select** the desired **program** (see item 8.2)
- Press [**Execute**] soft key
- Press “**CYCLE START**” key

NOTES:

- To exit program execution mode, press [**Cancel**] soft key
- To execute single block cycle, press “**Single Block**” key.

Machining step program		N: \MPF\PART\PART1.MPF
P	N5	Start of program
	N15	Tool selection T1 D1
	N20	Pos.w. rapid trav. X=0.000 Z=0.000 T1 D1
	N15	Free contour Cont_3 T=1 D=1
END		End of program

Figure 11-2. Program execution screen

11.2.1 - Restarting during Program Execution

To restart during program execution, it is required to:

- **Select** the desired **program** (see item 8.2)
- **Place** cursor on block corresponding to the desired tool for restarting execution
- Press key 
- Press [**Exec. here**] soft key
- Press “**CYCLE START**” key

NOTES:

- To execute the whole program again, it is required to exit this execution mode and repeat procedure on item 11.2
- To exit this execution mode, press [**Cancel**] soft key
- To execute “single block” cycle, press “**Single Block**” key.

PART II

MANUAL MACHINE PLUS

TUTORIAL

1 - CREATING THE PROGRAM

1.1 - CREATING THE NEW PROGRAM:

- Press [**Work prog.**] soft key
- Press [**File**] soft key
- Press [**Open**] soft key
- Place cursor on field “**Name**”
- Enter: **TUTORIAL**
- Press “**INPUT**” key
- Press twice [**OK**] soft key

Machining step program		N:\MPF\TUTORIAL.MPF
P	N5	Start or program
	N15	Tool selection T7 D1
END		End of program

Figure 1-1. Program edition screen

2 - FACING TOOL THROUGH TEACH-IN

2.1 - SELECTING TOOL:

- Manually **move axes** up to a safe point
- Place cursor on “**Tool selection**”
- Press key 
- **Place** cursor on field “**T**”
- Enter: **1** (for tool T01)
- Press “**INPUT**” key
- Press [**Accept**] soft key
- Press “**CYCLE START**” key (turret will be indexed on position 01)

2.2 - FACING TOOL:

- Press [**Turning**] soft key
- Press [**Teach In**] soft key
- Press [**Machining mode**] soft key until “**Parallel traversing of axes**” is displayed
- **Fill in data** according to figure 2-1
- **Turn main spindle on** through lever



Figure 2-1. "Parallel traversing of axes" mode screen

- Manually place tool on **X55 Z0** coordinate
- Press [**Save block**] soft key
- Press [ **Save**] soft key
- Face tool, by moving X axis up to **X-0.8** coordinate without moving Z axis
- Press [**Save block**] soft key
- Press [ **Save**] soft key
- Move Z axis up to **Z2** coordinate without moving X axis
- Press [**Save block**] soft key
- Press [ **Save**] soft key
- Move X axis up to **X52** coordinate without moving Z axis
- Press [**Save block**] soft key
- Press [ **Save**] soft key
- **Turn main spindle off** through lever
- Press [**Finish Teach-in**] soft key

3 - ROUGHING EXTERNAL CONTOUR

3.1 - ACTIVATING FREE CONTOUR CYCLE:

- Press [**Turning**] soft key
- Press [**Free contour**] soft key
- Fill in cycle **data**, according to figure3-1

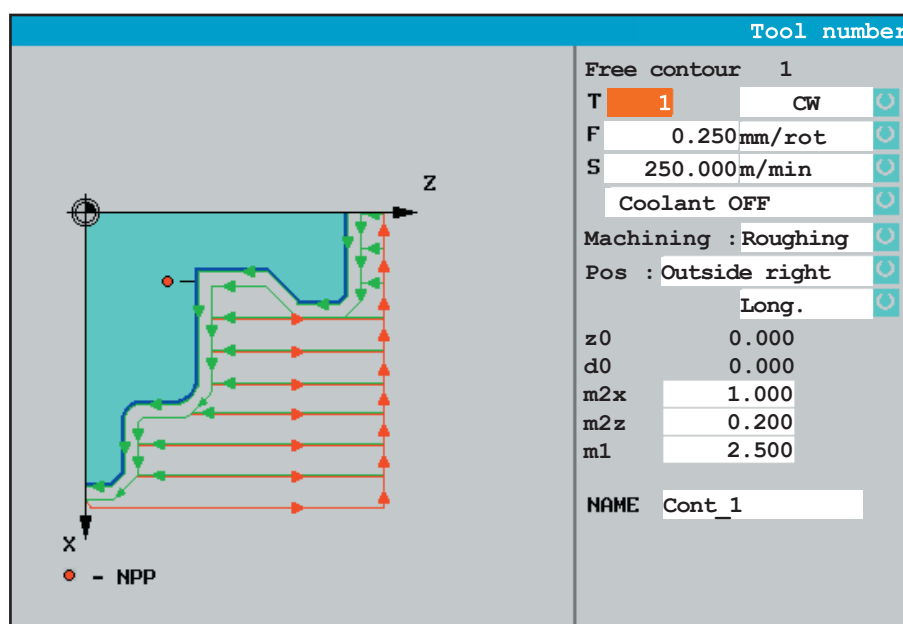


Figure 3-1. Free Contour Cycle Screen

3.2 - DEFINING PROFILE:

3.2.1 - Defining profile start point:

- Press [**Machined contour**] soft key
- Insert **start point** coordinate, according to 3-2
- Press [**Accept element**] soft key

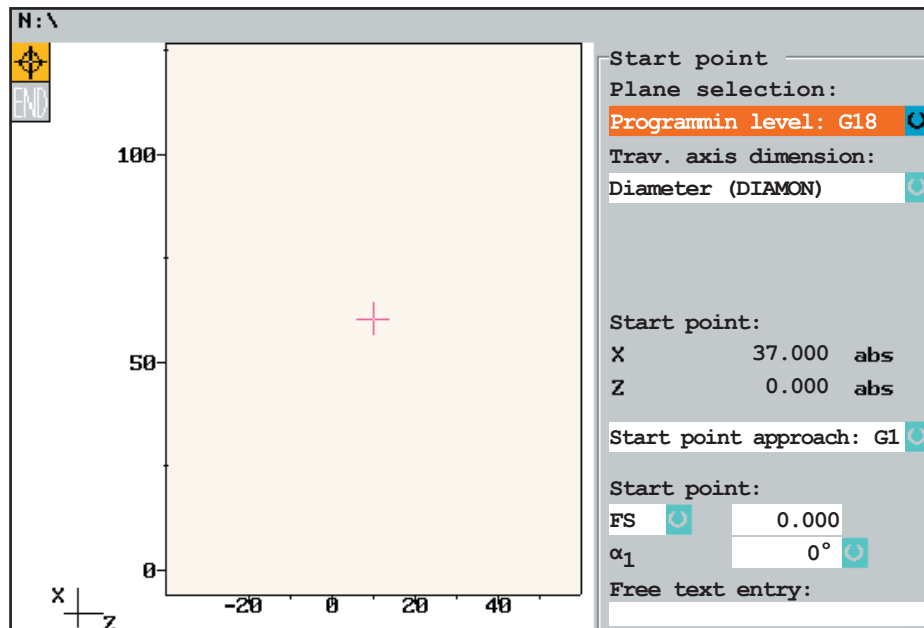


Figure 3-2. Free Contour Cycle Screen

3.2.2 - Creating profile:

- Press soft key 
- Enter data:
X 40.000 ABS
Z -1.500 ABS
- Press [**Accept element**] soft key

- Press soft key 
- Enter data:
Z -20.870 ABS
RD 20.000
- Press [**Accept element**] soft key

- Press soft key 
- Select option 
- Enter data:
R 10.000
X 40.000 ABS
Z -35.040 ABS
RD 1.000
- Press [**Accept element**] soft key

- Press soft key 
- Enter data:
X 36.000 ABS
α1 210.000 °
- Press [**Accept element**] soft key

- Press soft key 
- Enter data:
Z -65.000 ABS
RD 4.000
- Press [**Accept element**] soft key

- Press soft key 
- Enter data:
X 50.000 ABS
α1 105.000 °
RD 1.000
- Press [**Accept element**] soft key

- Press soft key 
- Enter data:
Z -90.000 ABS
- Press [**Accept element**] soft key

- Press [**Transfer**] soft key
- Press [**OK**] soft key

4 - FINISHING EXTERNAL CONTOUR

4.1 - SELECTING TOOL:

- Press [**Tools**] soft key
- **Place** cursor on field “**T**”
- Enter: **2** (for tool T02)
- Press “**INPUT**” key
- Press [**Transfer.**] soft key
- Press [**Cancel**] soft key

4.2 - ACTIVATING FREE CONTOUR CYCLE:

- Press [**Turning**] soft key
- Press [**Free contour**] soft key
- Fill in cycle data, according to figure 4-1

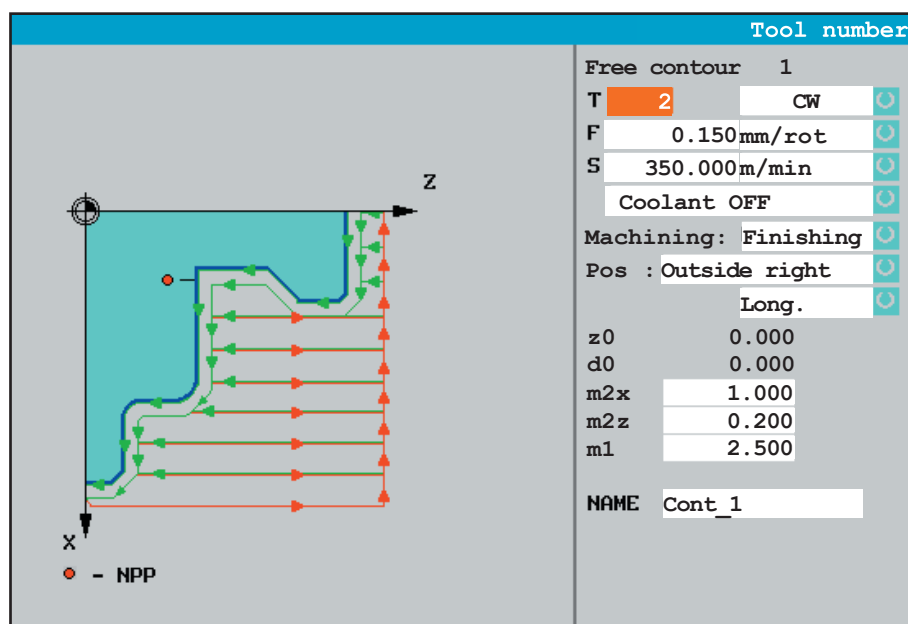


Figure 4-1. Free Contour Cycle Screen

4.3 - DEFINING PROFILE:

- **Place** cursor on “**NAME**”
- Press soft key [**Contour list**]
- **Select contour** executed in roughing. Example: Cont_1
- Press twice [**OK**] soft key

5 - GROOVING

5.1 - SELECTING TOOL:

- Press [**Tools**] soft key
- **Place** cursor on field “**T**”
- Enter: **3** (for tool T03)
- Press “**INPUT**” key
- Press [**Transfer.**] soft key
- Press [**Cancel**] soft key

5.2 - GROOVING

- Press [**Groove**] soft key
- **Fill in cycle data**, according to figure 5-1
- Press [**OK**] soft key

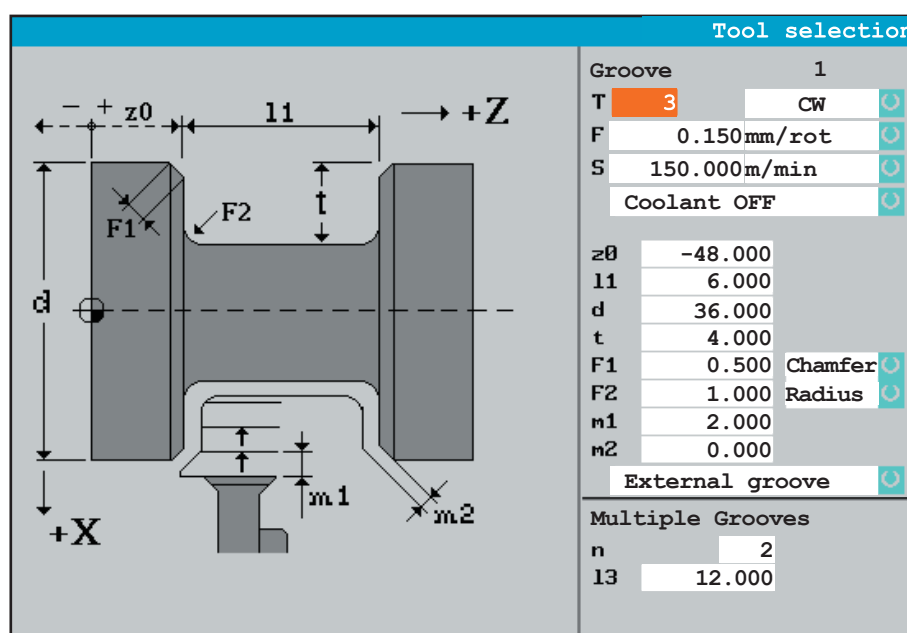


Figure 7-10. Grooving Cycle Screen

6 - EXTERNAL THREAD CUTTING

6.1 - SELECTING TOOL:

- Press [**Tools**] soft key
- **Place** cursor on field “**T**”
- Enter: **4** (for tool T04)
- Press “**INPUT**” key
- Press [**Transfer.**] soft key
- Press [**Cancel**] soft key

6.2 - CUTTING THREAD

- Press [**Thread**] soft key
- **Fill in cycle data**, according to figure 6-1
- Press [**OK**] soft key

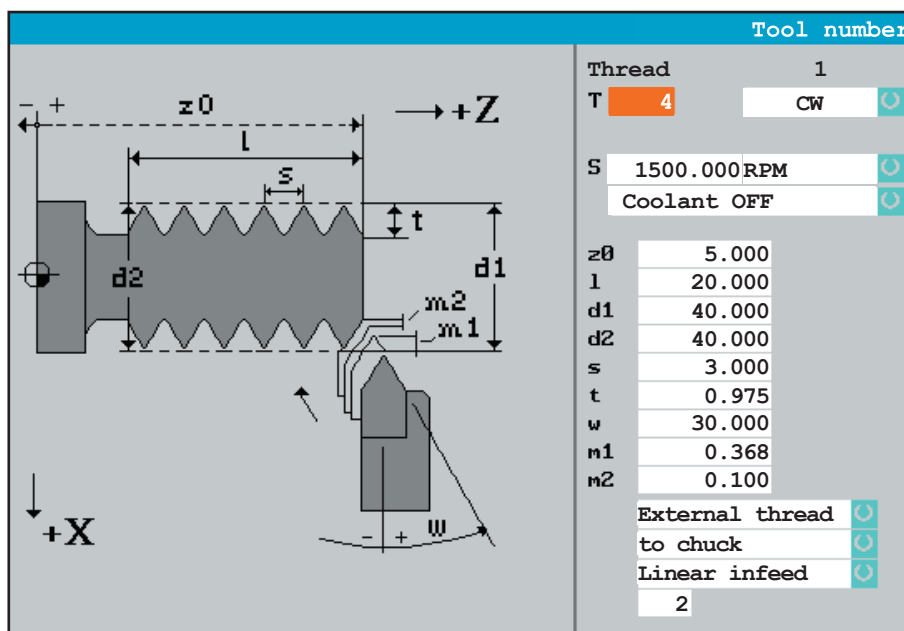


Figure 6-1. Threading Cycle Screen

7 - DRILLING THE WORKPIECE

7.1 - SELECTING TOOL:

- Press [**Tools**] soft key
- **Place** cursor on field “**T**”
- Enter: **5** (for tool T05)
- Press “**INPUT**” key
- Press [**Transfer.**] soft key
- Press [**Cancel**] soft key

7.2 - CUTTING THREAD

- Press [**Drilling**] soft key
- Press soft key [**Drilling centered**]
- **Fill in cycle data**, according to figure 7-1
- Press [**OK**] soft key

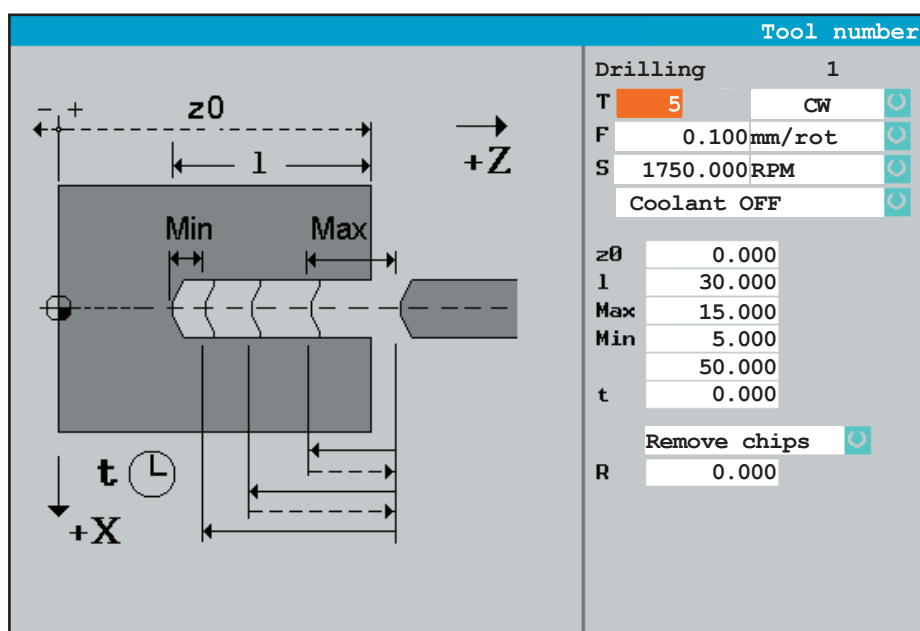


Figure 7-1. Drilling Cycle Screen

8 - MACHINING THE INTERNAL PROFILE

8.1 - SELECTING TOOL:

- Press [**Tools**] soft key
- **Place** cursor on field “**T**”
- Enter: **6** (for tool T06)
- Press “**INPUT**” key
- Press [**Transfer.**] soft key
- Press [**Cancel**] soft key

8.2 - MACHINING INTERNAL PROFILE:

- Press [**Turning**] soft key
- Press soft key [**Roughing cycle A**]
- **Fill in cycle data**, according to figure 8-1
- Press [**OK**] soft key

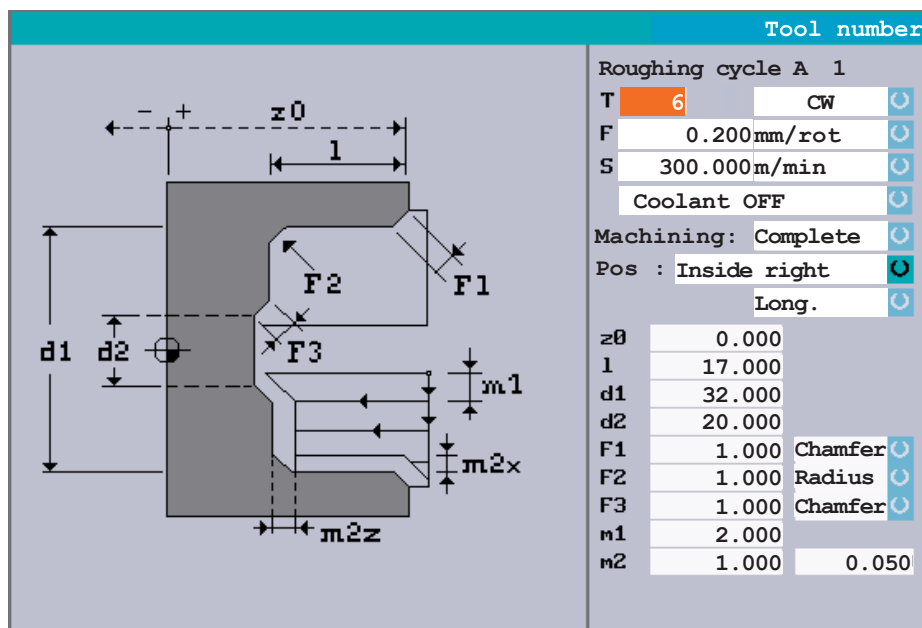


Figure 8-1. Roughing Cycle A Screen

9 - DEFINING WORKPIECE CHANGE POINT THROUGH TEACH-IN

9.1 - DEFINING WORKPIECE CHANGE POINT:

- Manually **place tool** at a point distant from workpiece. Example: X200 Z250
- Press [**Turning**] soft key
- Press [**Teach in**] soft key
- Press [**Machining mode**] soft key until “**Parallel traversing of axes**” is displayed
- **Place** cursor on field “**T**”
- Enter: **6** (for tool T06)
- Press “**INPUT**” key
- Press “**CYCLE START**” key
- **Fill in data** according to figure 9-1
- Press [**Save block**] soft key
- Press [ **Save**] soft key
- Press [**Teach-in exit**] soft key

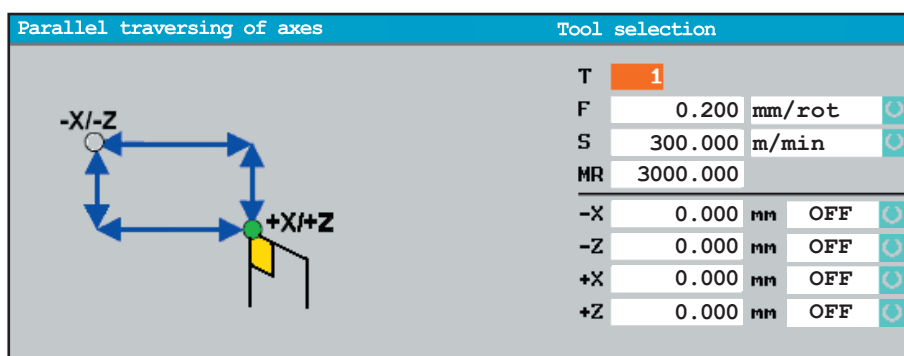


Figura 9-1. “Parallel traversing of axes” Mode Screen

10 - SAVING PROGRAM

10.1 - SAVING PROGRAM:

- Press [**Back**] soft key
- Press [**Save**] soft key

11 - SIMULATING PROGRAM

11.1 - SELECTING PROGRAM:

- Press [**Work prog.**] soft key
- Press [**File**] soft key
- Press [**Open**]
- Place cursor on “**TUTORIAL**” program
- Press [**OK**] soft key

11.2 - GRAPHICALLY SIMULATING:

- Press [**Simulation**] soft key
- Press [**Zoom Auto**] soft key
- Press “**CYCLE START**” key

11.3 - EXITING GRAPHIC SIMULATION:

- When finishing simulation, press [**Back**] soft key

12 - EXECUTING THE PROGRAM

IMPORTANT: Before executing the program, make sure it was already duly simulated, tools and workpiece were referenced and jaws were duly prepared.

12.1 - EXECUTING THE PROGRAM:

- Press [**Execute**] soft key
- Press “**CYCLE START**” key

12.2 - EXITING EXECUTION MODE:

- When finishing program execution, press [**Cancel**] soft key