Rexroth
Bosch Group

Rexroth IndraMotion MTX micro
MTX micro Trainer
Milling - First Steps

R911341439
Edition 01

Application Description

	Title	Rexroth IndraMotion MTX micro

Title Rexroth IndraMotion MTX micro
MTX micro Trainer
Milling - First Steps

Type of Documentation Application Description

Document Typecode DOK-MTXMIC-TRAIN*MILL*-AP01-EN-P

Internal File Reference RS-fadb37ec7b2ffb7c0a6846a500a9f19b-1-en-US-3

Purpose of Documentation This documentation describes the functions of the trainer for creating a milled part by using the Rexroth IndraMotion MTX micro.

Record of Revision

Edition	Release Date	Notes
Edition 01	07.2013	First edition

Note This document has been printed on chlorine-free bleached paper.

Table of Contents

Table of Contents

1   About this Documentation

1.1   Validity of the Documentation

Overview on Target Groups and Product Phases

The following illustration refers to the bordered activities, product phases and target groups of the present documentation.

Example: The target group "Machine operator" can "create the NC program" in the product phase "Operation" using this documentation.
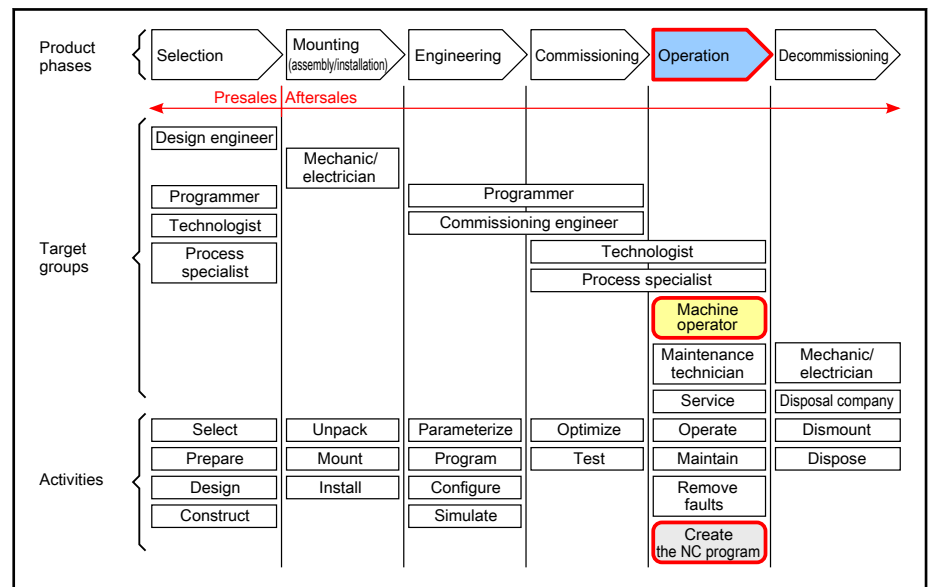
Fig.1-1:   Assigning this documentation to the target groups, product phases and target group activities

1.2   Required and Supplementing Documentation

1.2.1   Selecting

Documentation titles with type codes and parts numbers

Rexroth IndraMotion MTX micro 12VRS System Description
DOK-MTXMIC-SYS*DES*V12-PRxx-EN-P, R911334369
This documentation provides a system overview and describes the product properties of the Rexroth IndraMotion MTX micro.
Rexroth IndraControl VDP 80.1 Machine Operator Panel Operator Display
DOK-SUPPL*-VDP*80.1***-PRxx-EN-P, R911329156
This documentation contains a detailed description of the standard interface of the HMI operator panel.
Rexroth IndraDrive Drive Controllers HCQ, HCT
DOK-INDRV*-HCQ-T+HMQ-T-PRxx-EN-P , R911324185

xx   Corresponding version or edition
Tab.1-1:   MTX micro documentation overview - Selecting

About this Documentation

1.2.2   Configuring

Documentation titles with type codes and parts numbers

Rexroth IndraMotion MTX micro 13VRS Functional Description Basics
DOK-MTXMIC-NC*F*BA*V13-RExx-EN-P, R911336532
This documentation describes the basic functions of the Rexroth IndraMotion MTX micro. The basic commissioning steps and the functions of the control are given as description and handling instruction.
Rexroth IndraMotion MTX micro 13VRS Functional Description Extension
DOK-MTXMIC-NC*F*EX*V13-RExx-EN-P, R911337298
This documentation describes the extended functions of the Rexroth IndraMotion MTX micro. The basic commissioning steps and the functions of the control are given as description and handling instruction.
Rexroth IndraMotion MTX micro 13VRS Machine Parameters
DOK-MTXMIC-MA*PAR**V13-RExx-EN-P, R911336536
This documentation describes the design and adjustment of the available parameters.
Rexroth IndraMotion MTX 13VRS PLC Interface
DOK-MTX***-PLC*INT*V13-PRxx-EN-P, R911336344
This documentation describes interface signals and program function blocks for the integrated PLC.

xx   Corresponding version or edition
Tab. 1-2:   MTX micro documentation overview - Configuring

1.2.3   Commissioning

Documentation titles with type codes and parts numbers

Rexroth IndraMotion MTX micro Easy Setup for Standard Turning and Milling Machines
DOK-MTXMIC-EASY*******-COxx-EN-P, R911332281
This documentation provides an overview of the components of the IndraMotion MTX micro control system and supports the initial commissioning with handling instructions and examples.
Rexroth IndraWorks 13VRS Software Installation
DOK-IWORKS-SOFTINS*V13-COxx-EN-P, R911336880
This documentation describes the IndraWorks installation.
Rexroth IndraWorks 13VRS Engineering
DOK-IWORKS-ENGINEE*V13-APxx-EN-P, R911336870
This documentation describes the use of IndraWorks in which the Rexroth Engineering tools are integrated. It includes instructions on how to work with IndraWorks and how to operate the oscilloscope function.
Rexroth IndraMotion MTX 13VRS Commissioning
DOK-MTX***-STARTUP*V13-COxx-EN-P, R911336346
This documentation describes the commissioning of the IndraMotion MTX control. Apart from a complete overview, commissioning and configuration of the axes and the user interface as well as the PLC data are described.

About this Documentation

Rexroth IndraWorks 13VRS IndraLogic 2G PLC Programming System
DOK-IWORKS-IL2GPRO*V13-APxx-EN-P, R911336876
This documentation describes the PLC programming tool IndraLogic 2G and its use. It includes the basic use, first steps, visualization, menu items and editors.
Rexroth IndraWorks 13VRS, Basic Libraries, IndraLogic 2G
DOK-IL*2G*-BASLIB**V13-LIxx-EN-P, R911336285
This documentation describes the system-comprehensive PLC libraries.

xx   Corresponding version or edition
Tab.1-3:   MTX micro documentation overview - Commissioning

1.2.4   Operating

Documentation titles with type codes and parts numbers

Rexroth IndraMotion MTX micro 13VRS Standard NC Operation
DOK-MTXMIC-NC*OP***V13-APxx-EN-P, R911336348
This documentation describes the MMI operating software of the IndraMotion MTX micro.
Rexroth IndraMotion MTX micro 13VRS Programming Manual
DOK-MTXMIC-NC**PRO*V13-RExx-EN-P, R911336534
The following documentation provides information on the standard programming of the Rexroth IndraMotion MTX micro control.
Rexroth IndraMotion MTX 13VRS Standard NC Cycles
DOK-MTX***-NC*CYC**V13-PRxx-EN-P, R911336336
This documentation describes the application of the standard cycles of the different technologies for the Rexroth IndraMotion MTX control.
Rexroth IndraMotion MTX 12VRS Block Pre-Run
DOK-MTX***-BLK*RUN*V12-APxx-EN-P, R911334379
This documentation explains to the machine manufacturer how to setup the "Block pre-run" function at the machine for the end user.

xx   Corresponding version or edition
Tab.1-4:   MTX micro documentation overview - Operating

1.2.5   Maintaining

Documentation titles with type codes and parts numbers

Rexroth IndraMotion MTX 11VRS Diagnostic Messages
DOK-MTX***-DIAGMES*V11-RExx-EN-P, R911332311
This documentation provides an overview on errors, warnings and messages in the Rexroth IndraMotion MTX control.

xx   Corresponding version or edition
Tab.1-5:   MTX micro documentation overview - Maintenance

About this Documentation

1.2.6   Drive System: Commissioning and Project Planning

Documentation titles with type codes and parts numbers

Rexroth IndraDrive Drive Controllers HCQ, HCT
DOK-INDRV*-HCQ-T+HMQ-T-PRxx-EN-P , R911324185
It is used for the project planning of the drive systems Rexroth IndraDrive with the listed components
• HCQ02
• HCT02
Rexroth IndraDrive MPx-17 Functions
DOK-INDRV*-MP*-17VRS**-APxx-EN-P, R911331236
This documentation describes all the functional properties of the IndraDrive firmware in the variants MPB-17, MPM-17, MPC-17 and MPE-17.
Rexroth IndraDrive MPx-16 and MPx-17 Parameters
DOK-INDRV*-GEN1-PARA**-RExx-EN-P, R911328651
This documentation describes all parameters implemented in the firmware for drive controllers of the IndraDrive family. It supports the parameterization of the drive controllers.
• FWA-INDRV*-MPx-16VRS
• FWA-INDRV*-MPx-17VRS
Rexroth IndraDrive MPx-16 and MPx-17 Diagnostics
DOK-INDRV*-GEN1-DIAG**-RExx-EN-P, R911326538
This documentation describes the diagnostics implemented into the following firmwares:
• FWA-INDRV*-MPx-16VRS
• FWA-INDRV*-MPx-17VRS
It supports the operating crew as well as the programmer at troubleshooting.
Rexroth IndraDrive MPx-17 Version Notes
DOK-INDRV*-MP*-17VRS**-RNxx-EN-P, R911331588
The Version Notes contain an overview of the firmware function or the subject of the section. The Version Notes can contain, for example, general basics, the most important features of the function, overviews and examples of application.
The section "Notes on Application" contains the compatible and incompatible new functions and functional enhancements, as well as lists of new, modified and no longer existing parameters and diagnostic messages.
Rexroth IndraDyn S Synchronous Motors QSK061, -075
DOK-MOTOR*-QSK********-PRxx-EN-P, R911330321
This documentation…
• explains the features of the product, possibilities for use, operating conditions and operational limits of QSK motors
• contains technical data regarding available QSK motors
• provides information on product selection, handling and operation

xx   Corresponding version or edition

Tab. 1-6:   *MTX micro documentation overview - Drive system: Commissioning and Project Planning*

1.3   Information Representation

1.3.1   Safety Instructions

The safety instructions available in the user documentation contain certain signal words (Danger, Warning, Caution, Notice) and a signal alert symbols if necessary (acc. to ANSI Z535.6-2006).

The signal word draws attention to the safety instruction and indicates the risk potential.

The signal alert symbol (warning triangle with exclamation mark) positioned in front of the signal words Danger, Warning and Caution indicates hazards for individuals.

> **⚠ CAUTION**
>
> In case of non-compliance with this safety instruction, minor or moderate injury can occur.

> **NOTICE**
>
> In case of non-compliance with this safety instruction, material or property damage can occur.

1.3.2 Symbols Used

Note Notes are represented as follows:

> 🖝 This is a note for the user.

Tip Tips are represented as follows:

> 💡 This is a tip for the user.

1.3.3 Names and Abbreviations

Term	Explanation
IWE	IndraWorks Engineering
NC	Numerical Control
OEM	Original Equipment Manufacturer
sercos	sercos (Serial Realtime Communication System) interface is a world-wide standardized, digital interface to communicate between controls and drives
MP	Machine parameters

Tab. 1-7: Names and abbreviations used

DOK-MTXMIC-TRAIN*MILL*-AP01-EN-P   Bosch Rexroth AG   9/55
Rexroth IndraMotion MTX micro MTX micro Trainer Milling - First Steps

Introduction

2   Introduction

2.1   General Information

About this documentation   This documentation is intended for users wanting to manufacture workpieces at milling machines using the Bosch Rexroth control **IndraMotion MTX micro**. This application description explains how workpieces can be programmed and manufactured using the MTX micro.

Taking the example of a milled part, the entire process, starting from the drawing to the finished part, is explaining using the **IndraMotion MTX micro Trainer**. Download the software under www.boschrexroth.com/mtxtrainer. The user-friendly operating interface provides a graphically supported editor to create programs, an NC program simulation to visually check the NC program as well as comprehensive masks to set up the machine.

☞   Basic NC programming knowlege is required.

The following topics are covered:

- Creating tools and determining the zero points
- Operating steps for entering the NC program
- NC program structure
- Calling tools and technology
- Contour descriptions
- Cycle calls
- NC program simulation
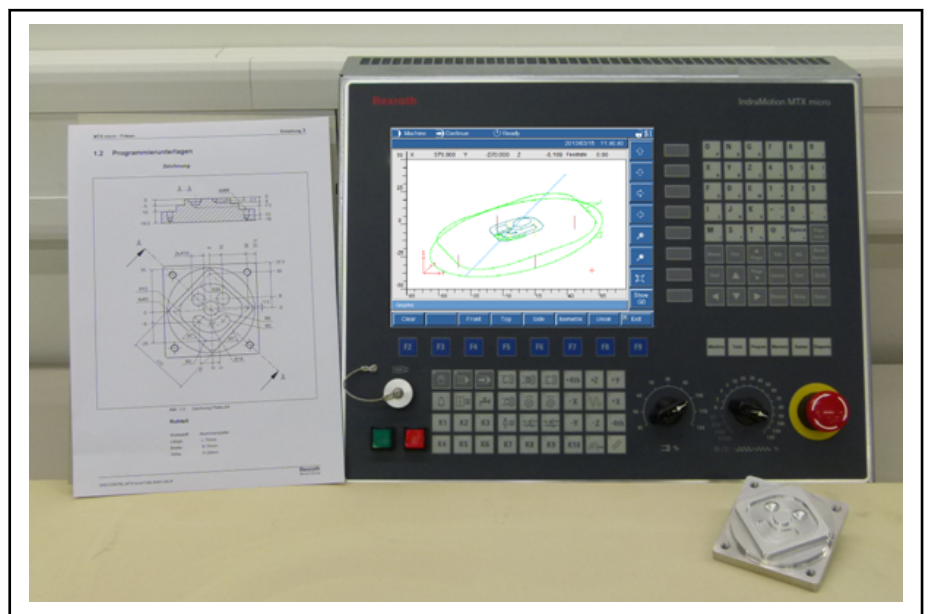
Fig.2-1:   From drawing to workpiece

2.2   Operation

The control panels to enter and simulate the programs are explained in this section.

Introduction

MTX micro Trainer

Fig.2-2:

Operation modes manual / MDI / automatic

Fig.2-3:

NC start / NC stop

Fig.2-4:

Operating areas

Fig.2-5:

Introduction

M-Key 1 to 8

Fig.2-6:

Introduction

2.3   Programming Documents

2.3.1   Drawing
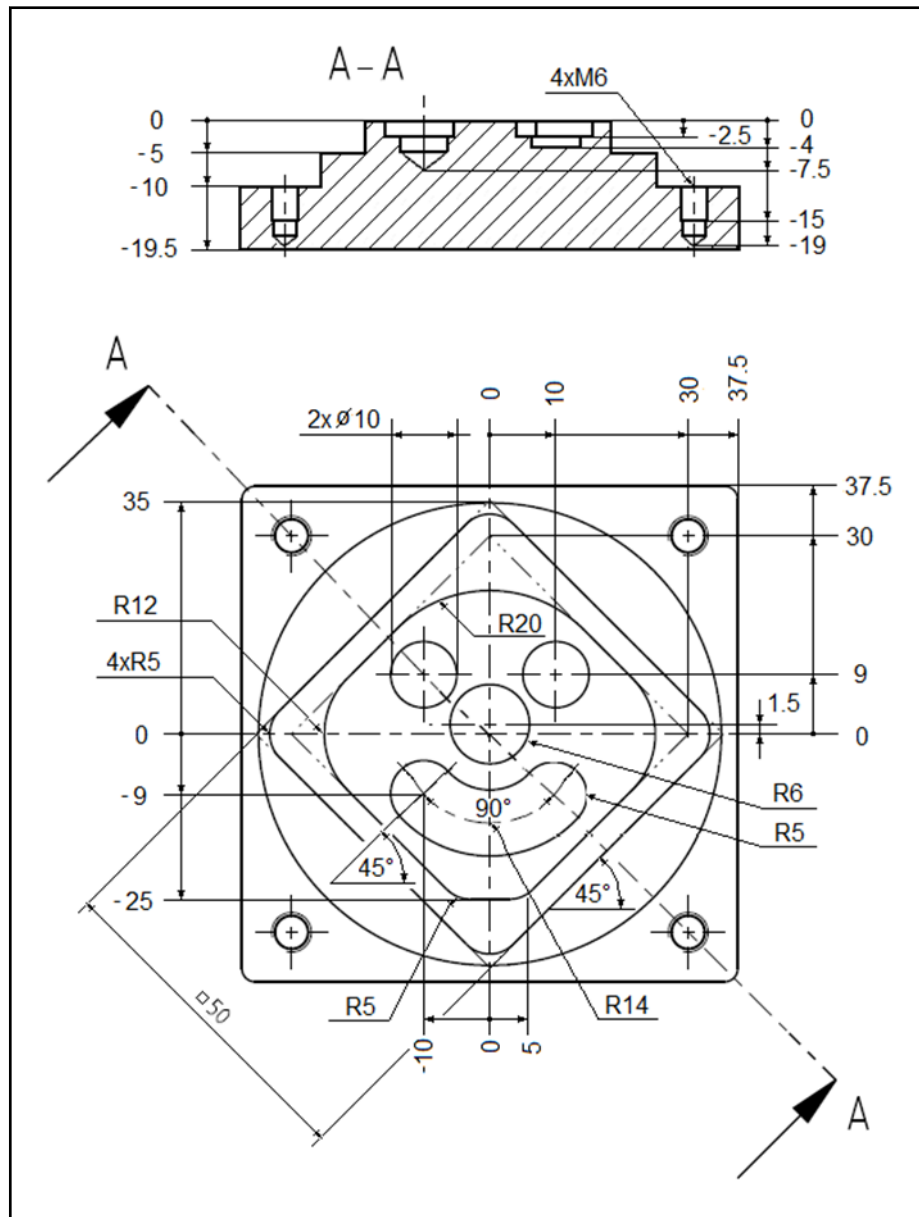
Fig.2-7:   Drawing Plate.dxf

2.3.2   Unmachined Part

Material: Aluminum plate

Length: L 75mm

Width: W 75mm

Height: H 20mm

2.3.3 Machining Plan

OP_No: Machining operation

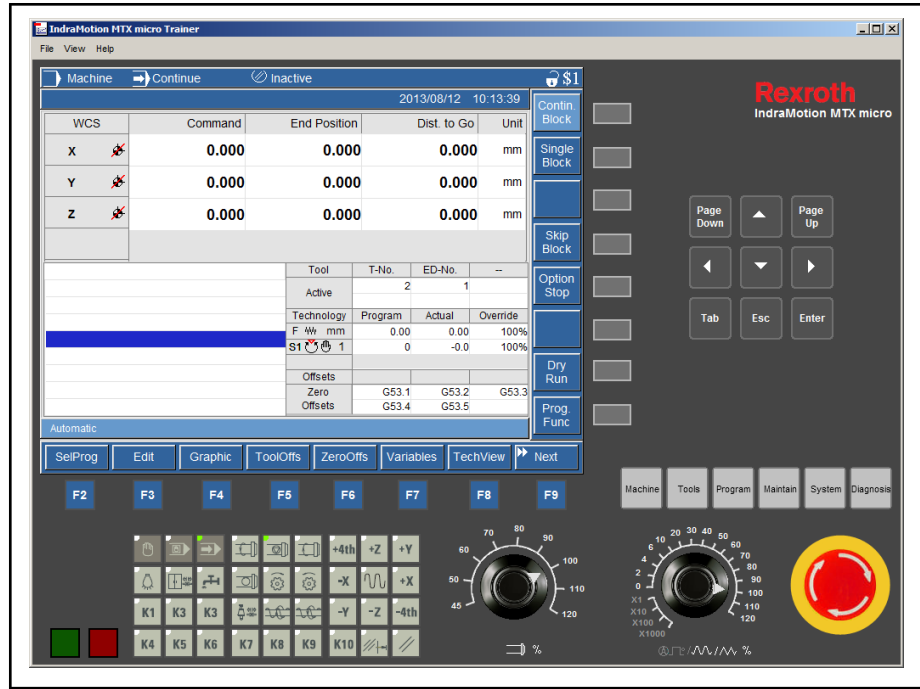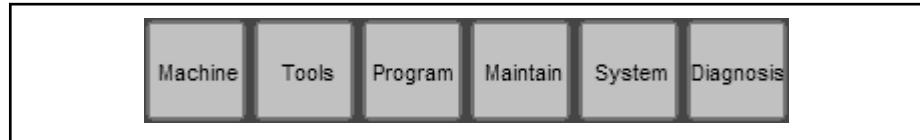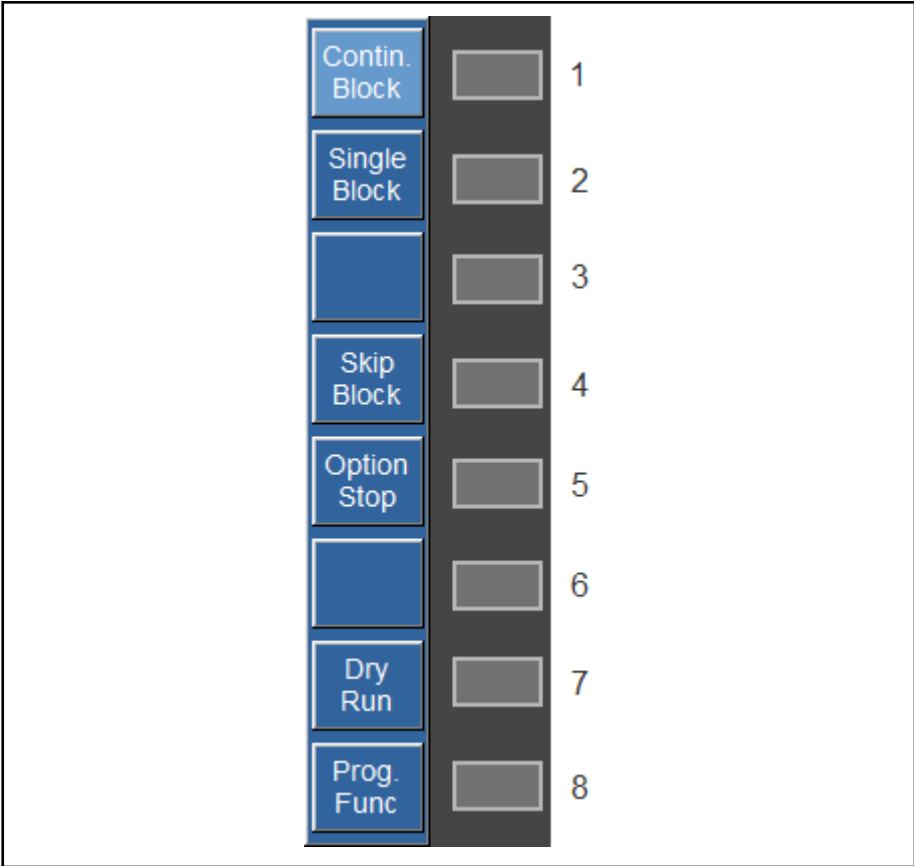| OP_No:	Machining operation	Remark	TNo.
OP 1	Surface milling	Finished size	1
OP 2	External contour circle roughing and finishing	Finishing allowance 0.1 mm	2
OP 3	External contour rhombus roughing and finishing	Finishing allowance 0.1 mm	2
OP 4	Pocket with island milling		17
OP 5	Curved slot roughing and finishing	Finishing allowance 0.1 mm	7
OP 6	Boring 2xD10	Depth: 15mm	6
OP 7	Machine thread 4xM6	Chamfer 0.1 x 45° tap hole D5	3/8/9

Tab.2-1:   Machining Plan

2.3.4 Tools

The following tools are required according to the machining plan:

Name	Magazine position	Type	Radius
T1	1	Milling head D80	40mm
T2	2	End face mill D50	25mm
T20	3	Boring groove milling tool D10	5mm
T17	4	Boring groove milling tool D6	3mm
T6	5	Twist drill D10 118°	0
T3	6	NC spot drill D12	0
T8	7	Twist drill D5 118°	0
T9	8	Tap M6	0

Tab.2-2:   Tool plan

Tool plan

2.4 Creating Tools

Changing to the "Tools" operating area

Press the "Tools" key.
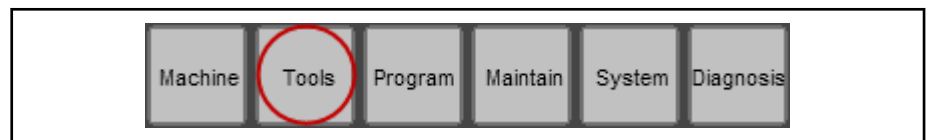
Fig.2-8:   Calling tools

Tool list display

The tool table of the milling machine is divided into two physical memories:

- Memory 1: The tool spindle
- Memory 2: The tool holding magazine

Introduction

The tool magazines are shown with their numbers in column S, the tool pocket locations with their numbers are shown in column P:

- Tool spindle memory 1, position 1
- Tool magazine memory 2, position 1 to 24

Action	M-key number	Display
Lock tool pocket location	1 (PB)	PB
Lock tool magazine	2 (TL)	TL

Tab.2-3:   Locking with M-key and display

The line highlighted in orange refers to the tool actively loaded in the spindle. The line highlighted in blue indicates the tool pocket selected to edit the tool data. Use "Cursor up" and "Cursor down" to highlight the previous or the next line.

The following figure shows that no tool is active in spindle position 1. The empty magazine position 4 is selected for editing.
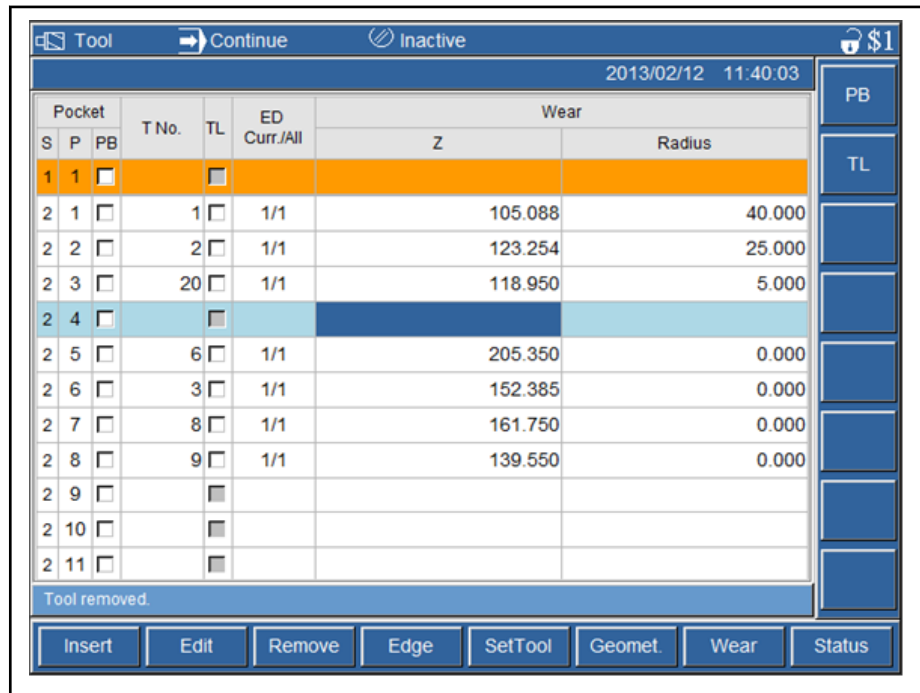
Fig.2-9:   Tool list

Selecting an empty tool pocket A tool pocket is empty if no TNo., no edge and no geometry data (Z, X, radius and Ori) are displayed. To create a new tool and an available tool pocket 4, select magazine position 4 in the tool list by moving the blue bar. To use more than one tool edge, e.g. a step drill, increase the number of edges in parameter "Offset Count (ED)".

Entering tool data Example: Press F3 "Insert" and enter the following tool data:

T Number 17

Z Offset 72.010

R Offset 3

Introduction
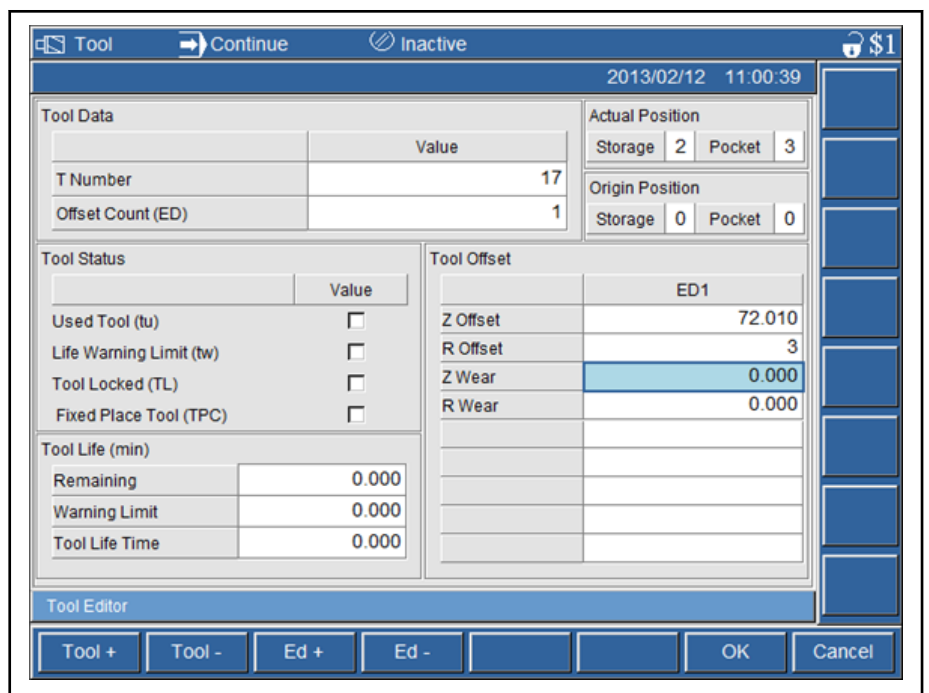
Fig.2-10:   Tool editor

Applying tool data   Complete the input and apply the tool data by pressing <F8>"Yes".
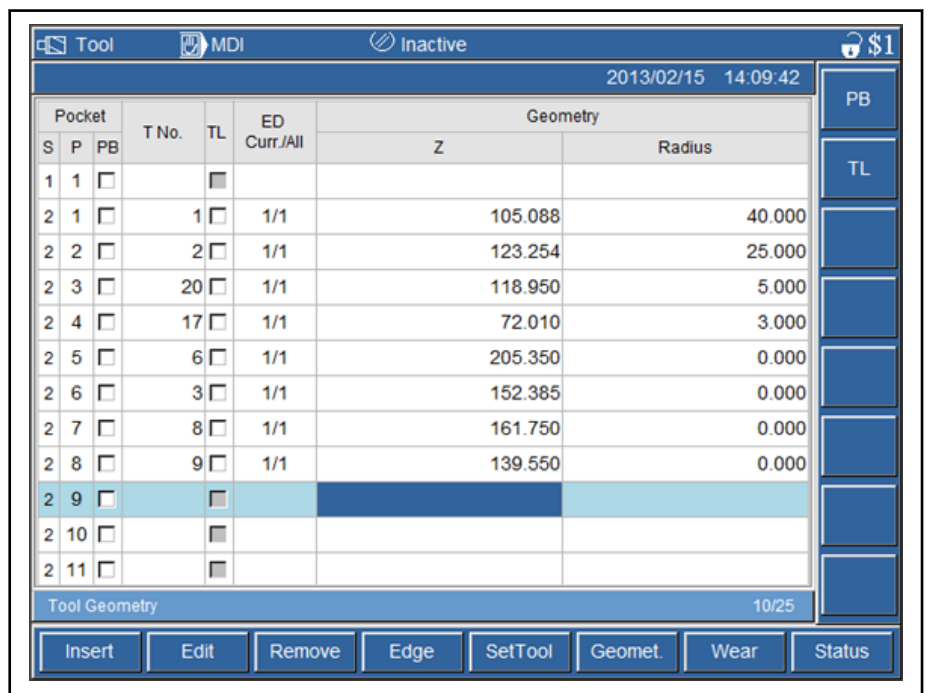
Fig.2-11:   Applying tool data

The tool T17 has been inserted in magazine position 4. Press <F5>"Edge" to change to the next edge in case of multi-edged tools. Refer to the display in column ED Curr/All.

Enter all other tools in the tool list according to the tool plan.

3   NC Program

3.1   General Information

Program flow chart   OP_No: Machining operation

Dividing machining in programming steps

| OP_No	Step	Program FACE	Note	TNo.
	1	Generate program header		
OP1	2	Tool change, technology input	G511	1
	3	Surface milling	G266	
OP2	4	Tool change, technology input	G511	2
	5	External contour circle with allowance roughing and finishing	G277	
OP3	6	External contour rhombus with allowance roughing and finishing	G276	
OP4	7	Tool change, technology input	G511	20
	8	Describing the pocket contour	N1000	
	9	Describing the island contour	N2000	
	10	Pocket with island milling	G281	
OP5	11	Tool change, technology input	G511	17
	12	Curved Slot Milling	G275	
OP6	13	Tool change, technology input	G511	6
	14	Boring 2xD10	G81	
OP7	15	Tool change, technology input	G511	3
	16	Spot drilling with chamfers 4x	G81 / G116	
	17	Tool change, technology input	G511	8
	18	Tap hole drilling 4xD5	G81 / G116	
	19	Tool change, technology input	G511	9
	20	Tapping 4xM6	G84 / G116	

Tab.3-1:

3.2   Creating the NC Program "FACE"

3.2.1   General Information

The entire machining is programmed in a machining program.

3.2.2   Creating the NC Program "FACE.npg"

General Information

First of all, the operating sequence to create a new NC program is described step by step.

Change to the "Program" machine operating area   Press "Program" on the control panel.
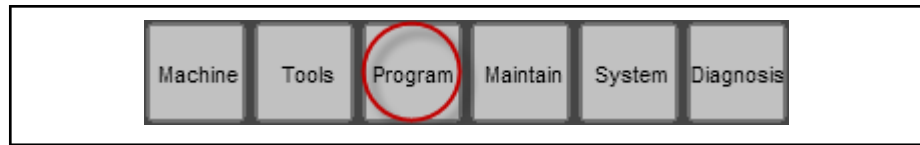
NC Program

Fig.3-1:

Opening editor for new NC program
Press <F2>"New", subsequently press <F3>"NCProg". The NC program editor opens with an empty window.

Saving the program
Press <F8>"Save". Enter a program name, e.g. "FACE.npg".

☞ **Note:** Note that the MTX micro file system is case sensitive. This refers to the names of directories and programs. For example, the program names "FACE.npg", "Face.npg" and "face.npg" would refer to three different programs.
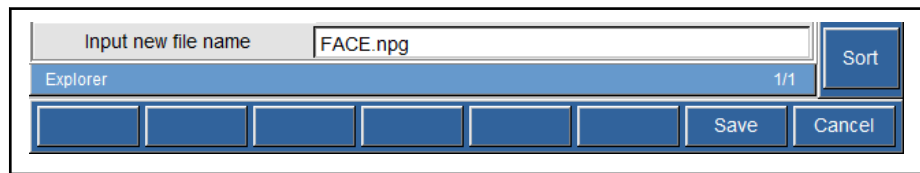
Fig.3-2:

Close the input with <ENTER> or <F8>"Save".

The program that is still empty is now saved under the name "FACE.npg" in the "/prog" directory.
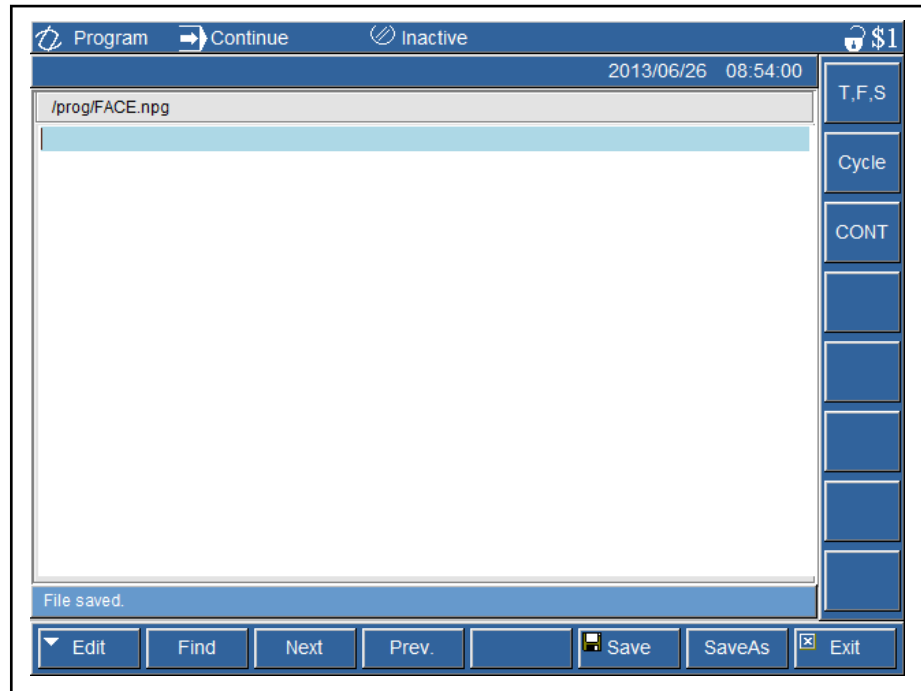
Fig.3-3:   Creatingn a new program that has no content yet

Step 1 - Creating the Program Header

Comment
Enter comments at the start of the program to track changes.

Enter the following comment lines (a comment line starts with "**;**" or "//"), as an example of this documentation:

NC Program

Program header

```
;Millingmachine:   MTX_micro
;NC program name: FACE.npg      Drawing:        FACE.dxf
;Material:         Aluminium    Stock           Dimension: 75mm x 75mm x 20mm
;Author:           Miller       Version: 1.0.0  Date: 14.02.2013
```

Close each comment line by clicking on <ENTER>.

Enter the following comment lines to structure the program:

Program start and end   *Program basis*

```
;-------------------- program start ----------------------------
;
N10
N990 M30
;-------------------- program end ------------------------------
```

Contour description   *Program for contour*

```
// description pocket contour
// begin
N1000
N
// end
;
// description island contour
N2000
N
// end
;--------------------------------------------------------------
```

☞   The program is located between N10 and N990 M30. Subsequently, the necessary contour descriptions are inserted.

Enter a comment that can be a user-defined character string. The semicolon can also be used within a block. For example:

```
N300 G0 ;rapid traverse
```

```
Program    →Continue    Inactive                            $1
                                      2013/06/26  09:21:51    T,F,S
/prog/FACE.npg
;Millingmachine:  MTX_micro                                  Cycle
;NC program name: FACE.npg    Drawing:         FACE.dxf
;Material:        Aluminium   Stock Dimension: 75mm x 75mm x 20mm
;Author:          Miller      Version: 1.0.0   Date: 26.06.2013   CONT
;-------------------- program start ----------------------------
;
N10
N990 M30
;-------------------- program end ------------------------------
// description pocket contour
// begin
N1000
N
// end
;
// description island contour
// begin
NC Editor                                              1/22
  Edit    Find    Next    Prev.        Save    SaveAs    Exit
```

Fig.3-4:

NC Program

3.2.3   OP1 - Face Milling

General Information

During the first machining operation, the workpiece is face milled. The cutter head T1 is swiveled in. MTX micro offers a comprehensive cycle package with graphic input masks for tool change and further machining.

Enter the comment for the first machining operation in the empty comment line after the "`program start`" comment.

```
;OP1 Face milling
```

Step 2 - Tool Change T1

Press <M-Key 1>"T,F,S", to open the input mask for the tool change cycle.

Fig.3-5:   Tool change cycle

Entering values in the cycle input masks

Select an input field in the cycle mask and enter the value. Confirm the entry with <ENTER>. The next input field is automatically selected.

The tool can be selected from the tool list if the required tool has already been entered in the list. Press <M-Key 3>"Search", after the input field for parameter "T" has been selected. Select the desired tool and confirm with <ENTER> or <F8>"Yes".

The cycle parameters are divided in mandatory and optional parameters. The mandatory parameters are identified by an "*" in the list. Optional parameters can be shown or hidden by pressing <M-Key 4>"Option". Many parameters have preassignments to facilitate the input of data. For more information, refer to the documentation about cycles.

☞   The cycle input mask can be opened by positioning the cursor on the line Nxx and by pressing "T,F,S" or "Cycle".

NC Program

Tool change T1 and technology

Specify all conditions for subsequent machining in the input mask:

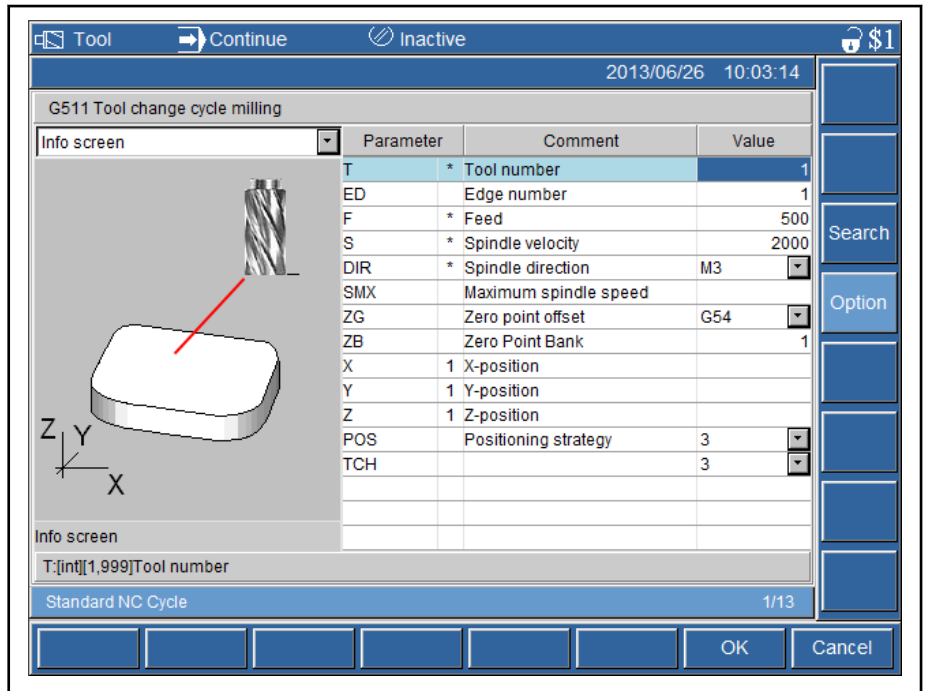|		
T	1	Tool, cutter head D80
F	500	Feed 500 mm/min
S	2000	Speed 2000 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-2:

Fig.3-6:   Tool change cycle with options

☞   **Note:** The input mask shows all parameters used in the cycle. Check the basic settings of the optional parameters.

Applying the cycle into the program

Complete the input by pressing <F8>"OK".

```
N10 G511(T1,F500,S2000,DIR M3,ZG G54)
```

The cycle G511 has been applied to the program. Insert a word wrap with <ENTER> at the end of the line.

```
;--------------------- program start ----------------------------
;OP1 Face milling
N10 G511(T1,F500,S2000,DIR M3,ZG G54)

N990 M30
;--------------------- program end ------------------------------
```

Fig.3-7:   NC program with cycle G511

Step 3 - Face Milling

Surface milling

During this machining operation, the plane surface is milled to finished dimension. Insert the NC block N20:

Selecting the group milling cycles

Use <M-Key 2>"Cycle" to navigate to the standard NC cycles.

NC Program

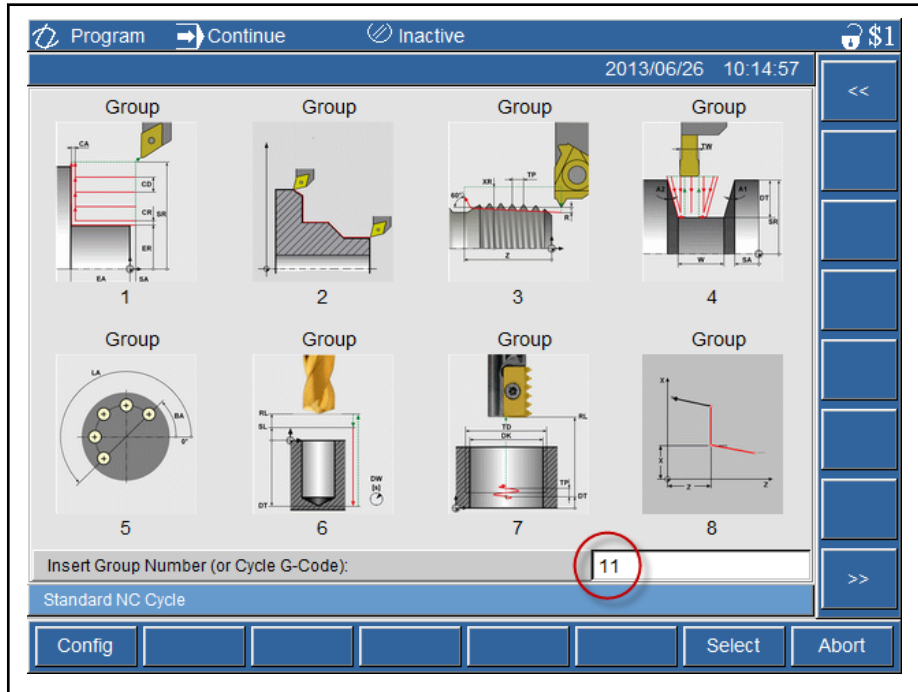Select cycle G266 in the group "11" (milling cycles).

Fig.3-8:   Cycle group 11

👉   Use <M-Key 8>">>" to call other tool groups.

Selecting the cycle G266   Select the cycle G266.
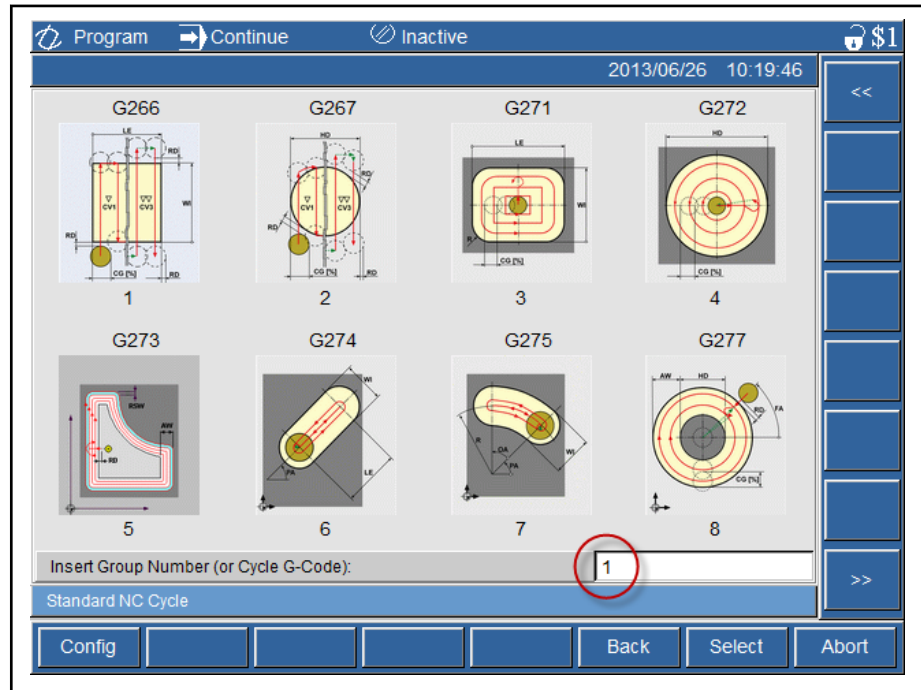

Fig.3-9:   Cycle G266 (1)

👉   Sampling of the zero point for the z-axis, 0.5 mm has to be deducted from the machine coordinates. As the unmachined part height is 20 mm, a machined part height of 19.5 mm results when face milling to Z0.

NC Program

Entering technology data for the face milling cycle

Use <F8>"Select" to open the cycle input mask.

Specify the parameters for the machining cycle G266 in the cycle input mask:

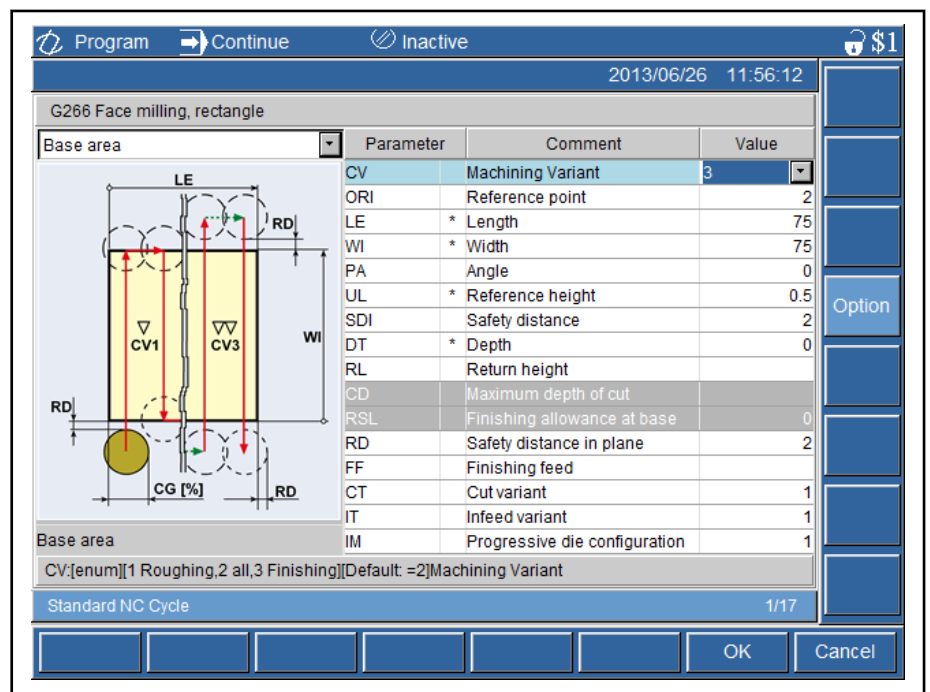|		
CV	3	Machining (cutting) variant
ORI	2	Reference point
LE	75	Length
WI	75	Width
UL	0.5	Reference height
SDI	2	Safety distance
DT	0	Depth
RD	2	Safety distance in plane
CG	100	Tool use in %

Tab.3-3:

Fig.3-10:   *Completing values in the input mask*

Complete the input by pressing <F8>"OK".

```
N20 G266(CV 3,ORI2,LE75,WI75,UL0.5,SDI2,DT0,RD2,CG100)
```

The modal cycle G266 is processed after the next positioning. Disable the modal call using G80, if no more position is to be machined. Insert the following NC blocks:

Face Milling, Rectangle
```
N30 G0 X0 Y0
```

Deleting the milling cycle, modal
```
N40 G80
```

NC Program

```
;--------------------- program start ----------------------------
;OP1 Face milling
N10 G511(T1,F500,S2000,DIR M3,ZG G54)
N20 G266(CV 3,ORI2,LE75,WI75,UL0.5,SDI2,DT0,RD2,CG100)
N30 G0 X0 Y0
N40 G80


N990 M30
;--------------------- program end ------------------------------
```

Fig.3-11:   NC program with cycle G266

3.2.4   OP2 - External Contour Circle - Roughing and Finishing

General Information

In the following machining operation, the external contour circle diameter 70 mm is roughened and subsequently finished with an allowance of 0.1 mm.

Enter a comment for this machining operation:

```
;------------------------------------------------------
;OP2 External contour circle, roughing and finishing
```

Step 4 - Tool Change T2

Tool change T2 and technology   The tool change is inserted by means of the T,F,S cycle.

The external contour circle is milled with the following parameters, using the tool T2 end face mill D50:

T	2	Tool, end face mill D50
F	250	Feed 250 mm/min
S	2000	Speed 2000 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-4:

```
N50 G511(T2,F250,S2500,DIR M3,ZG G54)
```

```
;----------------------------------------------------------------
;OP2 External contour circle, roughing and finishing
N50 G511(T2,F250,S2000,DIR M3,ZG G54)


N990 M30
;--------------------- program end ------------------------------
```

Fig.3-12:

Step 5 - External Contour Circle with Cycle G277

Non-uniform cutting conditions occur during circular traversing of square un-machined parts. Reason: The feed refers to the tool edge or contour in case of an active G45. Thus, the feed with G46 refers to the milling center point to achieve a smoother run and a better surface.

Insert the following NC block:

```
N60 G46 G0 X0 Y0 Z2
```

Cycle G277   Select cycle "8" Circular pin milling in the group "11" (milling cycles). Option-ally, make entries directly in the input line "G277".

Entering technology data for the circular pin cycle

☞ The feed for roughing has already been specified in the TFS cycle.

Specify the parameters for the machining cycle G277 in the cycle input mask:

CV	2	Machining (cutting) variant
HD	70	Circle diameter
UL	0	Reference height
SDI	0	Safety distance
DT	-10	Depth
RL	2	Retract height
RD	2	Safety distance in plane
AW	4	Machining allowance of unmachined part
RSW	0.1	Finishing allowance at the wall
RSL	0.1	Finishing allowance at the base
FF	150	Finishing feed
CG	80	Tool use in %

Tab.3-5:
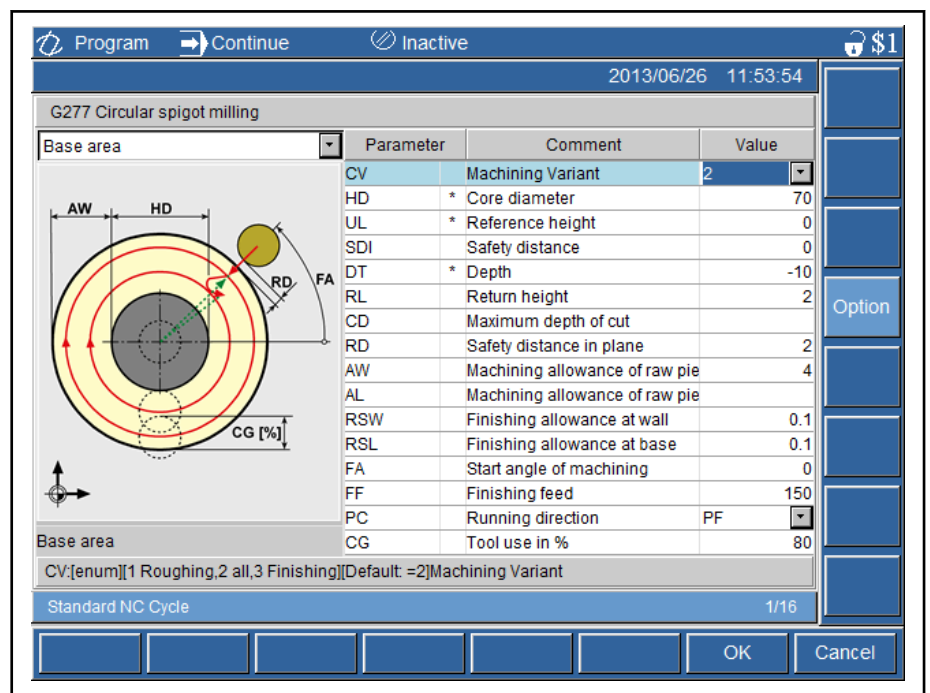
Fig.3-13:   Completing values in the input mask

Complete the input by pressing <F8>"OK".

```
N70
G277(HD70,UL0,SDI0,DT-10,RL2,RD2,AW4,RSW0.1,RSL0.1,FF150)
```

The modal cycle G277 is processed after the next positioning. Disable the modal call using G80, if no more position is to be machined. Insert the following NC blocks:

Circular pin milling

```
N80 G0 X0 Y0
```

NC Program

Deleting the milling cycle, modal   `N90 G80`

```
;----------------------------------------------------------------
;OP2 External contour circle, roughing and finishing
N50 G511(T2,F250,S2000,DIR M3,ZG G54)
N60 G46 G0 X0 Y0 Z2
N70 G277(HD70,UL0,SDI0,DT-10,RL2,RD2,AW4,RSW0.1,RSL0.1,FF150)
N80 G0 X0 Y0
N90 G80

N990 M30
;-------------------- program end ---------------------------------
```

Fig.3-14:

3.2.5   OP3 - External Contour Rhombus - Roughing and Finishing

General Information

During the next machining operating, the external contour rhombus is milled using the same tool T2. Enter a comment line for this machining operation:

`;----------------------------------------------------------`

`;OP3 External contour, rhombus milling`

Step 6 - External Contour Rhombus with Cycle G276

☞   The feed for roughing has to be programmed before or in the calling NC block as not tool change is required and thus, no TFS cycle has been completed.

Cycle G276   Select cycle "7" Rectangular spigot milling in the group "11" (milling cycles). Optionally, make entries directly in the input line "G276".

Entering technology data for the rectangular spigot milling cycle   Specify the parameters for the machining cycle G276 in the cycle input mask:

CV	2	Machining (cutting) variant
ORI	2	Reference point
LE	50	Length
WI	50	Width
R	5	Corner radius
PA	45	Angle
UL	0	Reference height
SDI	0	Safety distance
DT	-5	Depth
RL	2	Retract height
RD	2	Safety distance in plane
AW	4	Machining allowance of the raw piece at the wall
RSW	0.1	Finishing allowance at the wall
RSL	0.1	Finishing allowance at the base

NC Program

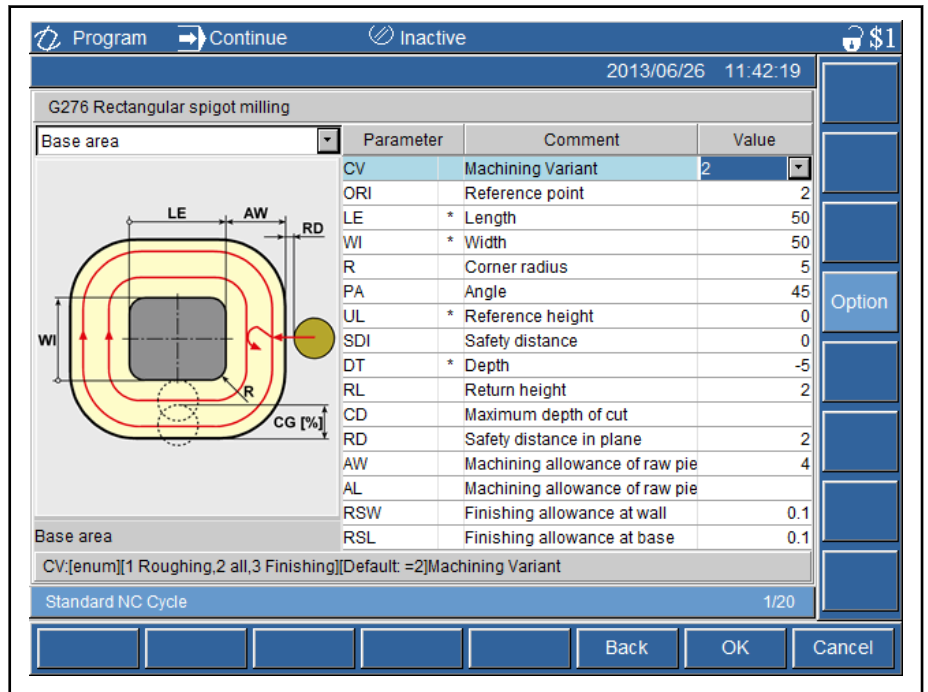|		
FF	150	Finishing feed
CG	80	Tool use in %

Tab.3-6:

Fig.3-15:

Complete the input with <F8>"OK" and add the feed for roughing.

```
N100              G276(ORI2,LE50,WI50,R5,PA45,UL0,SDI2,DT-5,
RL2,AW10,RSW0.1,RSL0.1,FF150) F250
```

The modal cycle G276 is processed after the next positioning. Disable the modal call using G80, if no more position is to be machined. Insert the following NC blocks:

Rectangle pin milling

```
N110 G0 X0 Y0
```

Deleting the milling cycle, modal

```
N120 G80
```

```
;-------------------------------------------------------------
;OP3 External contour, rhombus milling
N100 G276(ORI2,LE50,WI50,R5,PA45,UL0,SDI0,DT-5,RL2,RD2,AW4,RSW0.1,RSL0
N110 G0 X0 Y0
N120 G80


N990 M30
;-------------------- program end -----------------------------
```
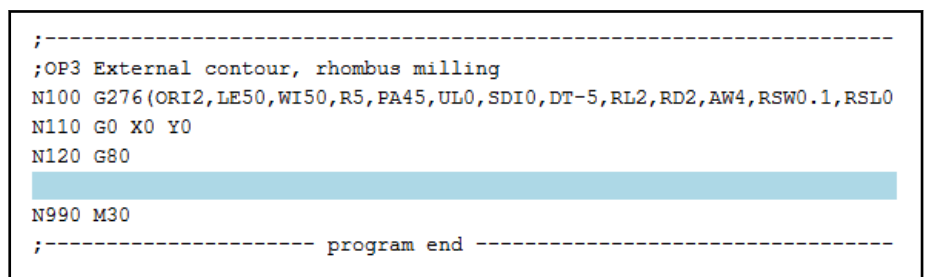
Fig.3-16:

3.2.6   OP4 – Pocket Milling with Islands

General Information

During the next machining operation, the contour pocket with island is milled.

Enter a comment line for this machining operation:

```
; -------------------------------------------------------------
;OP4 pocket milling with islands
```

NC Program

Step 7 - Tool Change T20

Tool change T2 and technology   Insert the tool change by means of the <M-Key 1>"T,F,S".

Tool T20 is loaded in a boring groove milling tool D10 and the pocket is machined with the following parameters:

T	20	Tool, boring groove milling tool D10
F	200	Feed 200 mm/min
S	2000	Speed 2000 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-7:

Complete the input by pressing <F8>"OK".

```
N130 G511(T20,F200,S2500,DIR M3,ZG G54)
```

The feed with G45 is referred to the cutting edge to achieve a smooth wall surface.

Insert the following NC block:

```
N140 G45
```

```
;------------------------------------------------------------------
;OP4 Mill contour pocket with isle contours
N130 G511(T20,F200,S2000,DIR M3,ZG G54)
N140 G45


N990 M30
;-------------------- program end --------------------------------
```

Fig.3-17:

Step 8 - Describing the Pocket Contour

The pocket contour is described in this program section. Starting and end block determine the start and the end of the entire contour path. The contour is described using coordinates, straight lines, circles, roundings and chamfers. The defined contour draft is used for roughing and finishing in this program.

Chamfers and roundings   *Modal active ON:*
- CHL(<Chamfer length>)
- RND(<Rounding radius>):

Modal active OFF:
- CHL()
- RND()

Block-by-block active:
- CHLL(<Chamfer length>)
- RNDL(<Rounding radius>)

The following numerical value in parentheses, specifies the chamfer length or the rounding radius.

NC Program

Defining the contour path of the pocket

In the program section ""`// description pocket contour`"", navigate to a position after the block number `N1000` and insert the description of the pocket contour.
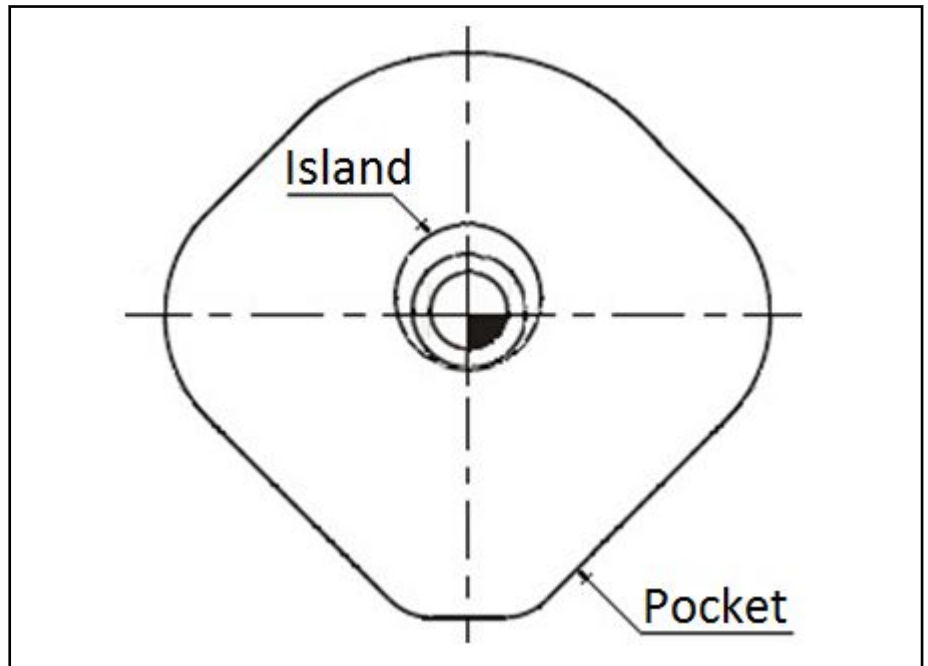

Fig.3-18:   Pocket contour

Insert the NC blocks `N1000` to `N1060`:

```
;--------------------- program end ----------------------------------
// description pocket contour
// begin
N1000 G1 X0 Y-25
N1010 X5 RNDL(5)
N1020 X30 Y0 RNDL(12)
N1030 X0 Y30 RNDL(20)
N1040 X-30 Y0 RNDL(12)
N1050 X-5 Y-25 RNDL(5)
N1060 X0
// end
;-------------------------------------------------------------------
```

Fig.3-19:

Step 9 - Describing the Island Contour

In the program section ""`// description island contour`"", navigate to a position after the block number `N2000` and insert the description of the island contour. Insert the following NC blocks:

Defining the contour path of the island

```
;-------------------------------------------------------------------
// description island contour
// begin
N2000 X0 Y7.5
N2010 G2 I0 J1.5
// end
;-------------------------------------------------------------------
```

Fig.3-20:

NC Program

Step 10 - Pocket Milling with Islands

Starting and end block of the island, the starting point of machining and the technology for machining has to be programmed. Three cycles are used for programmming.

Selecting the cycle G288 | **Navigate to the NC block N150. Select cycle G288 and enter the following parameters:**

Default		Deleting all previous island contours
S	2000	Starting block of the island contour description
E	2010	End block of the island contour description

Tab.3-8:
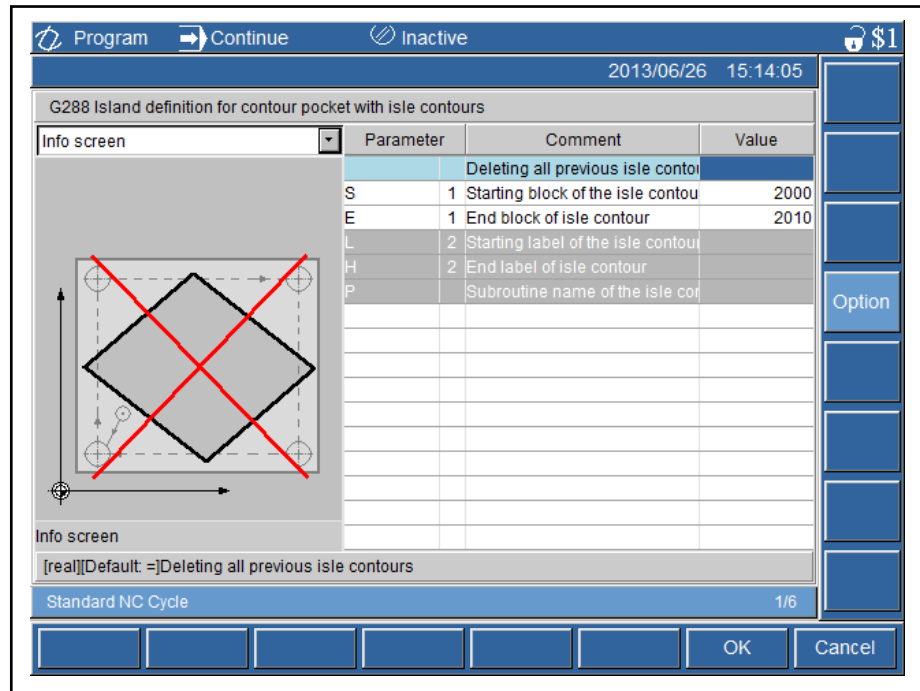
Fig.3-21:

Complete the input by pressing <F8>"OK".

```
;--------------------------------------------------------------------
;OP4 Mill contour pocket with isle contours
N130 G511(T20,F200,S2000,DIR M3,ZG G54)
N140 G45
N150 G288(S2000,E2010)

N990 M30
;-------------------- program end -----------------------------------
```

Fig.3-22:

Program the NC block N160, to define the starting block of machining:

NC Program

Selecting the cycle G289

Select cycle G289 and enter the following parameters:

Default		Deleting all previous starting points
MSP	-10	Starting point of the main axis
SSP	-10	Starting point of secondary axis

Tab.3-9:
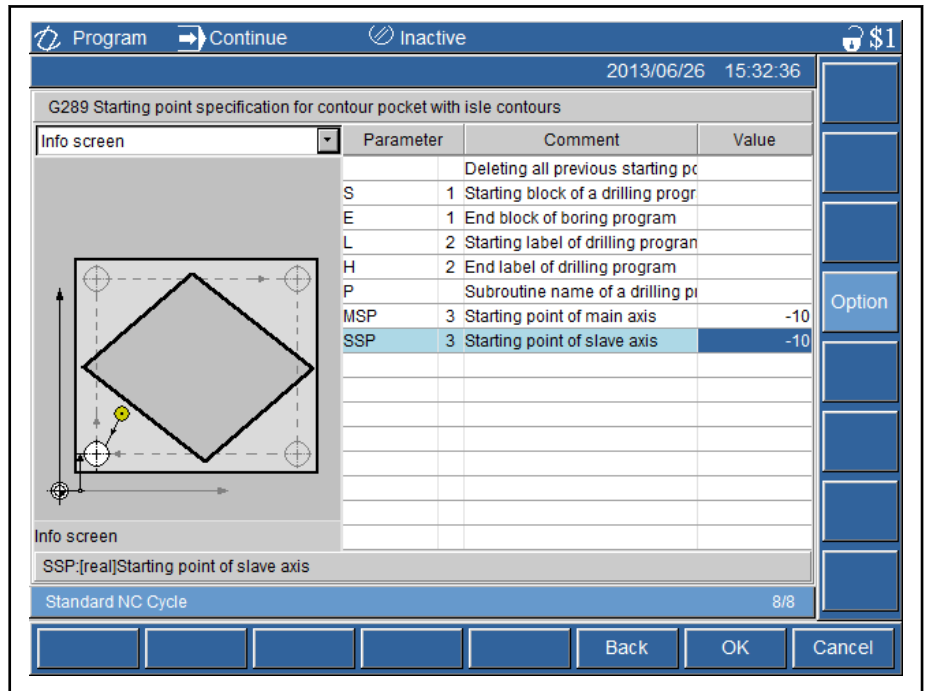
Fig.3-23:

Complete the input by pressing <F8>"OK".

```
;------------------------------------------------------------------
;OP4 Mill contour pocket with isle contours
N130 G511(T20,F200,S2000,DIR M3,ZG G54)
N140 G45
N150 G288(S2000,E2010)
N160 G289(MSP-10,SSP-10)

N990 M30
;-------------------- program end ---------------------------------
```

Fig.3-24:

Entering technology data for cycle G281

Specify the parameters for the machining cycle G281 in the cycle input mask:

S	1000	Starting block of the pocket contour description
E	1060	End block of the pocket contour description
CV	2	Machining (cutting) variant
CVT	1	Cutting variant
UL	0	Reference height
SDI	1	Safety distance
RD	0.1	Safety distance in plane

NC Program

DT	-2.5	Depth
RL	1	Retract height

Tab.3-10:
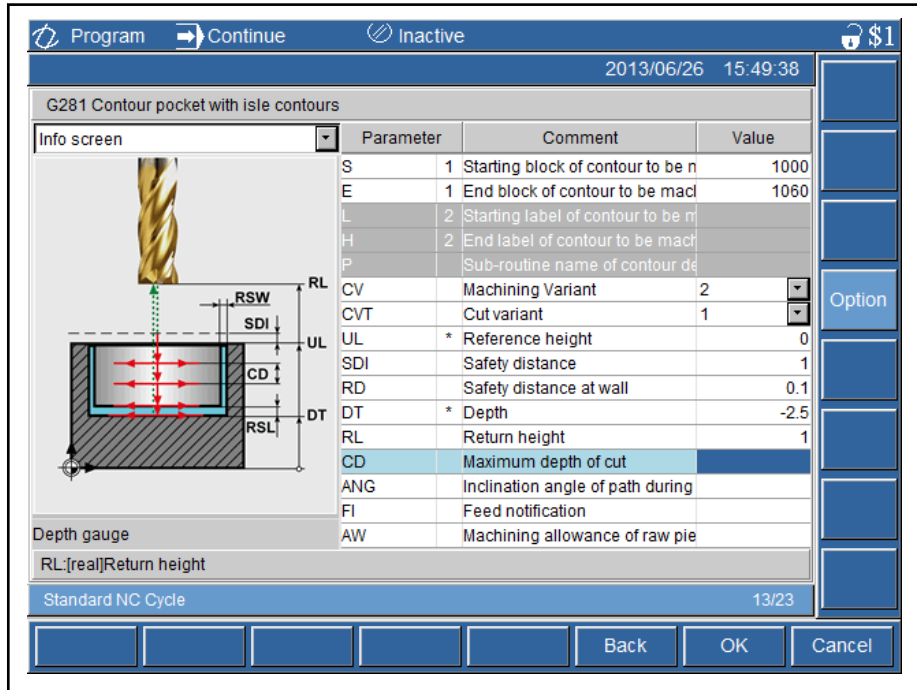
Fig.3-25:

Complete the input by pressing <F8>"OK".

```
;------------------------------------------------------------------
;OP4 Mill contour pocket with isle contours
N130 G511(T20,F200,S2000,DIR M3,ZG G54)
N140 G45
N150 G288(S2000,E2010)
N160 G289(MSP-10,SSP-10)
N170 G281(S1000,E1060,UL0,SDI1,RD0.1,DT-2.5,RL1)


N990 M30
;-------------------- program end ---------------------------------
```

Fig.3-26:

3.2.7   OP5 - Curved Slot - Roughing and Finishing

General Information

During this machining operation, the curved slot with an allowance of 0.1 mm is roughed and finished.

Enter a comment for this machining operation:

; --

;OP5 Curved slot, roughing and finishing

Step 11 - Tool Change T17

Tool change T17 and technology

Insert the tool change by means of the <M-Key 1>"T,F,S".

NC Program

The external contour circle is milled with the following parameters, using the tool T17 end face mill D6:

T	17	Tool, boring groove milling tool D6
F	200	Feed 200 mm/min
S	2500	Speed 2500 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-11:

```
;------------------------------------------------------------------
;OP5 Curved slot, roughing and finishing
N180 G511(T17,F200,S2500,DIR M3,ZG G54)


N990 M30
;-------------------- program end -----------------------------------
```

Fig.3-27:

Step 12 – Curved Slot - Roughing and Finishing

Cycle G275   Select cycle "G275" in the input line.

Entering technology data for the curved slot cycle   Specify the parameters for the machining cycle G275 in the cycle input mask:

CV	2	Machining (cutting) variant
WI	10	Width
R	5	Machining (cutting) variant
PA	225	Starting angle
OA	90	Slot angle
UL	-2.5	Reference height
SDI	1	Safety distance
DT	-4	Depth
RL	1	Retract height
IM	1	Plunging strategy
RSW	0.1	Finishing allowance at the wall
RSL	0.1	Finishing allowance at the base
FF	180	Finishing feed

Tab.3-12:

NC Program
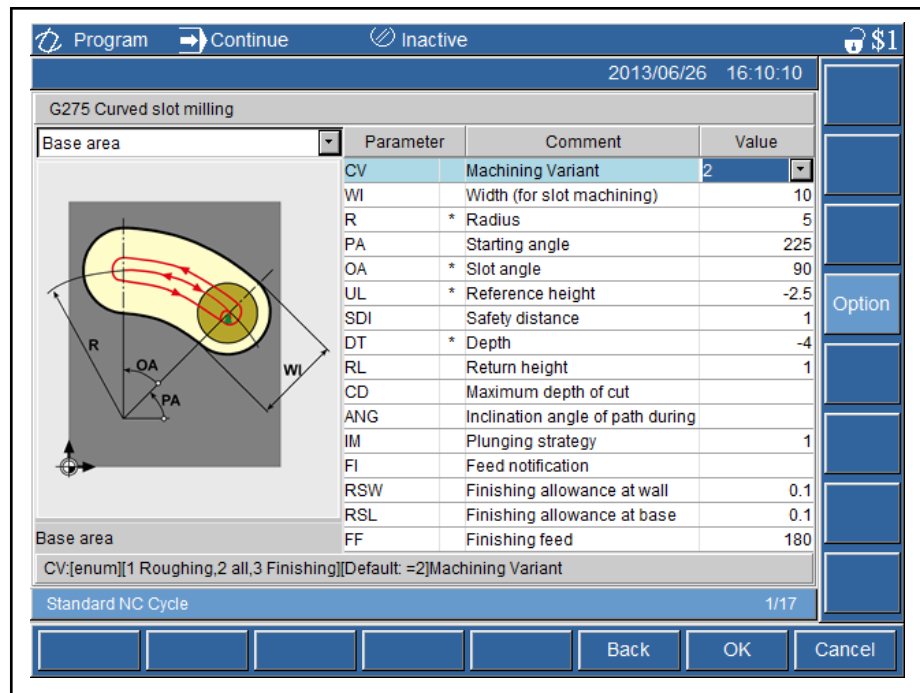
Fig.3-28:

Complete the input by pressing <F8>"OK".

```
;-----------------------------------------------------------------
;OP5 Curved slot, roughing and finishing
N180 G511(T17,F200,S2500,DIR M3,ZG G54)
N190 G275(WI10,R14,PA225,OA90,UL-2.5,SDI1,DT-4,RL1,FI100,RSW0.1,RSL0.1



N990 M30
;-------------------- program end ---------------------------------
```

Fig.3-29:

The modal cycle G275 is processed after the next positioning. Disable the modal call using G80, if no more position is to be machined. Insert the following NC blocks:

Curved slot milling `N200 G0 X-10 Y-9`

Deleting the milling cycle, modal `N210 G80`

```
;-----------------------------------------------------------------
;OP5 Curved slot, roughing and finishing
N180 G511(T17,F200,S2500,DIR M3,ZG G54)
N190 G275(WI10,R14,PA225,OA90,UL-2.5,SDI1,DT-4,RL1,FI100,RSW0.1,RSL0.1
N200 X-10 Y-9
N210 G80



N990 M30
;-------------------- program end ---------------------------------
```

Fig.3-30:

3.2.8   OP6 – Boring 2xD10

General Information

During this machining operation, two borings with diameter 10 are machined in the pocket.

Enter a comment for this machining operation:

```
;----------------------------------------------------------
;OP6 Boring 2x10
```

Step 13 - Tool Change T6

Tool change T6 and technology   Insert the tool change by means of the <M-Key 1>"T,F,S".

The two borings are machined using the following parameters:

T	6	Tool, drill D10
F	250	Feed 250 mm/min
S	1500	Speed 1500 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-13:

```
N220 G511(T6,F250,S1500,DIR M3,ZG G54)
```

```
;----------------------------------------------------------------
;OP6 Boring 2x10
N220 G511(T6,F250,S1500,DIR M3,ZG G54)

N990 M30
;-------------------- program end --------------------------------
```

Fig.3-31:

Step 14 - Boring 2x10

Cycle G81   Select cycle "1" G81 in the group "6" (boring cycles). Optionally, make entries directly in the input line "G81".

Entering technology data for the cycle G81   **Specify the parameters for the machining cycle G81 in the cycle input mask:**

IX	Z	Drill axis
SL	-2	Safety distance
DT	-7.5	Depth
DW	0	Dwell time
RL	1	Retract height

Tab.3-14:

NC Program
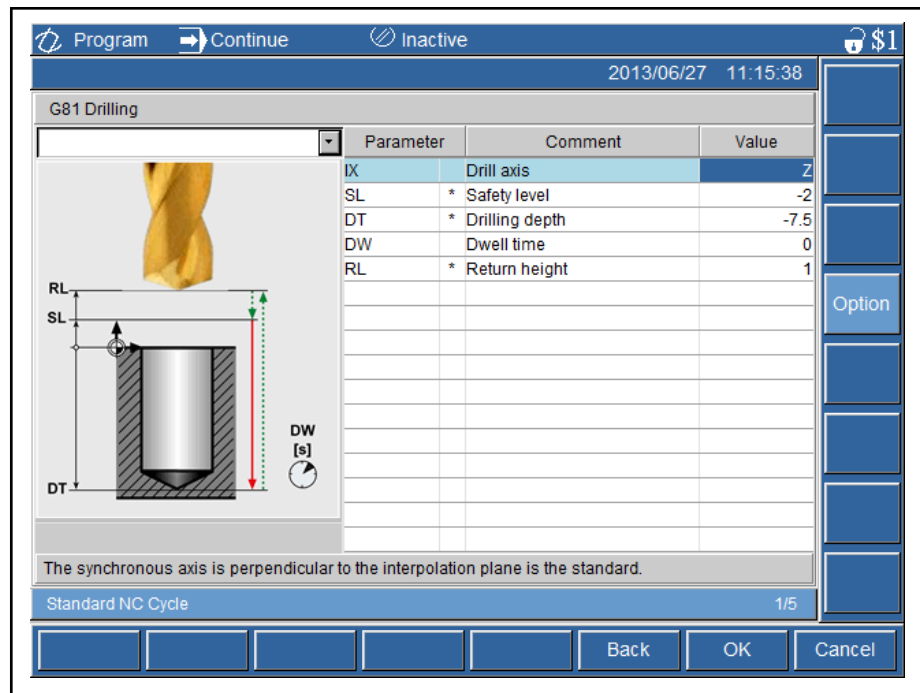
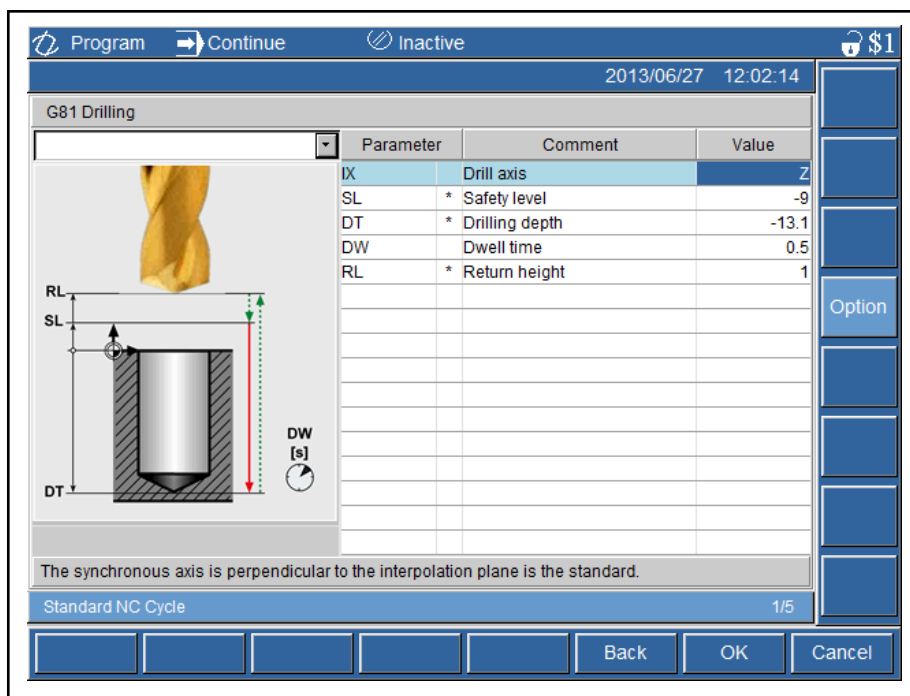*Fig.3-32:*

Complete the input by pressing <F8>"OK".

```
N230 G81(IX Z,SL-2,DT-7.5,RL1)
```

The modal cycle G81 is processed after the next positioning. Disable the modal call using G80, if no more position is to be machined. Insert the following NC blocks:

Boring position 1   `N240 G0 X10 Y9`

Boring position 2   `N250 G0 X-10 Y9`

Deleting boring cycle, modal   `N260 G80`

```
;----------------------------------------------------------------
;OP6 Boring 2x10
N220 G511(T6,F250,S1500,DIR M3,ZG G54)
N230 G81(IX Z,SL-2,DT-7.5,RL1)
N240 G0 X10 Y9
N250 G0 X-10 Y9
N260 G80

N990 M30
;-------------------- program end --------------------------------
```

Fig.3-33:

3.2.9   OP7 – Thread 4xM6

General Information

During this machining operation, the four threads M6 are machined. Different tools are required. Sequence of operations: Spot drilling to chamfer depth, tap drilling and thread cutting.

Enter a comment for this machining operation:

```
;----------------------------------------------------------------
;OP7 Machine thread 4xM6
```

Step 15 - Tool Change T3

Tool change T3 and technology
Insert the tool change by means of the <M-Key 1>"T,F,S". An NC spot drill 90° is used for spot drilling to chamfer depth.

T	3	Tool, NC sport drill D10
F	250	Feed 250 mm/min
S	2000	Speed 2000 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-15:

```
N270 G511(T3,F250,S2000,DIR M3,ZG G54)
```

```
;-----------------------------------------------------------------
;OP7 Manufacture thread 4xM6
N270 G511(T3,F250,S2000,DIR M3,ZG G54)

N990 M30
;---------------------- program end ------------------------------
```

Fig.3-34:

Step 16 - Spot Drilling to Chamfer Depth

Cycle G81
Select cycle "1" G81 in the group "6" (boring cycles). Optionally, make entries directly in the input line "G81".

Entering technology data for the cycle G81
Specify the parameters for the machining cycle G81 in the cycle input mask:

IX	Z	Drill axis
SL	-9	Safety distance
DT	-13.1	Depth
DW	0.5	Dwell time
RL	1	Retract height

Tab.3-16:

NC Program

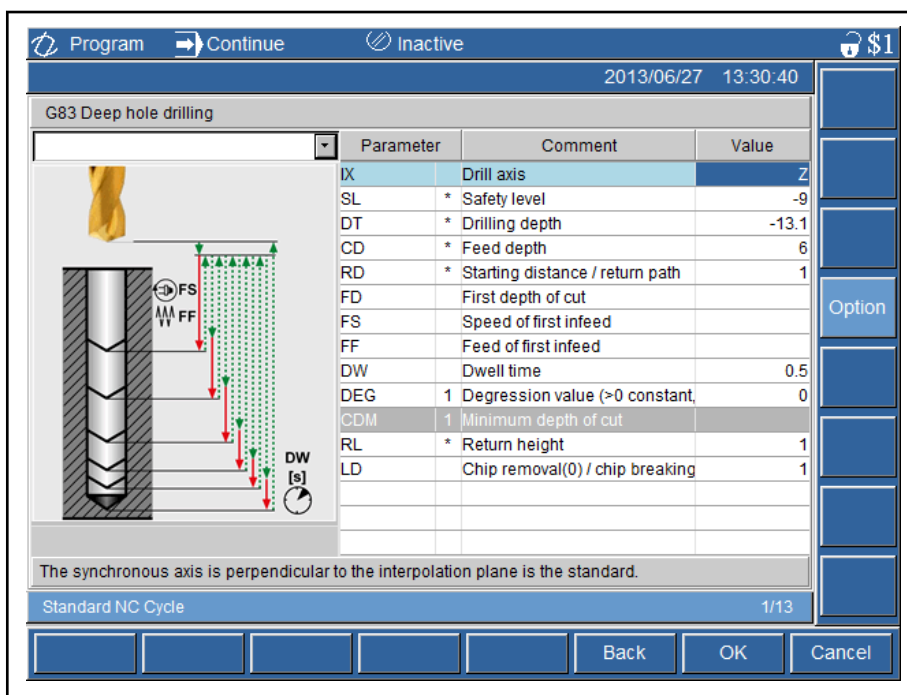*Fig.3-35:*

Complete the input by pressing <F8>"OK".

```
N280 G81(IX Z,SL-9,DT-13.1,DW0.5,RL1)
```

The modal cycle G81 is processed after the next positioning. The point pattern G116 is used.

Select cycle "6" G116 Hole Pattern Frame in the group "5" (pattern of holes). Optionally, make entries directly in the input line "G116".

Entering technology data for the cycle G116

Specify the parameters for the machining cycle G116 in the cycle input mask:

MSP	-30	Starting point: main axis
SSP	-30	Starting point: secondary axis
TA	0	Angle
PD	60	Hole distance
NH	2	Number of drillings
VO	60	Boring distances, vertical
NL	2	Number of borings, vertical
MA	0	Angular offset

Tab.3-17:

NC Program

Fig.3-36:

Complete the input by pressing <F8>"OK".

```
N290 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
```

Insert the following NC block:

Deleting boring cycle, modal   ```N300 G80```

Step 17 - Tool Change T8

Tool change T8 and technology   Insert the tool change by means of the <M-Key 1>"T,F,S".

A twist drill D5 is used for boring the tap hole.

T	8	Tool, twist drill D5
F	300	Feed 300 mm/min
S	2000	Speed 2000 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-18:

```
N310 G511(T8,F300,S2000,DIR M3,ZG G54)
```

Step 18 - Drilling the Tap Hole for M6

Cycle G83   Select cycle "3" G83 Deep hole drilling in the group "6" (boring cycles). Optionally, make entries directly in the input line "G83".

Entering technology data for the cycle G83   Specify the parameters for the machining cycle G83 in the cycle input mask:

IZ	Z	Drill axis
SL	-9	Safety distance
DT	-13.1	Drilling depth
CD	6	Infeed depth

NC Program

RD	1	Starting distance/Retract distance
DW	0.5	Dwell time
DEG	0	Degression value
RL	1	Retract height
LD	1	Chip removal (0) / chip breaking (1)

Tab.3-19:

Fig.3-37:

Complete the input by pressing <F8>"OK".

```
N320 G83(IX Z,SL-9,DT-19,CD6,RD0.1,DW0.5,RL1,LD1)
```

The modal cycle G83 is processed after the next positioning. The point pattern G116 is used.

Select cycle "6" G116 Hole Pattern Frame in the group "5" (pattern of holes). Optionally, make entries directly in the input line "G116".

Entering technology data for the cycle G116

Specify the parameters for the machining cycle G116 in the cycle input mask:

MSP	-30	Starting point: main axis
SSP	-30	Starting point: secondary axis
TA	0	Angle
PD	60	Hole distance
NH	2	Number of drillings
VO	60	Boring distances, vertical
NL	2	Number of borings, vertical
MA	0	Angular offset

Tab.3-20:

Complete the input by pressing <F8>"OK".

NC Program

```
N330 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
```

Insert the following NC block:

Deleting boring cycle, modal
```
N340 G80
```

Step 19 - Tool Change T9

Tool change T8 and technology   Insert the tool change by means of the <M-Key 1>"T,F,S".

Tap M6 is used for thread cutting:

T	9	Tool, tap M6
F	0	Feed 0 mm/min
S	0	Speed 0 1/min
M	3	Direction of rotation, clockwise
ZG	G54	Workpiece zero point G54

Tab.3-21:
```
N350 G511(T9,F0,S0,DIR M3,ZG G54)
```

Step 20 - Thread M6 Tapping

Cycle G84   Select cycle "4" G84 Floating tapping in the group "6" (boring cycles). Option-ally, make entries directly in the input line "G84".

Entering technology data for the cycle G84   Specify the parameters for the machining cycle G84 in the cycle input mask:

IX	Z	Drill axis
SL	-8	Safety distance
DT	-18	Thread depth
TP 1	1	Thread pitch
MS	500	Speed
RL	1	Retract height

Tab.3-22:

NC Program
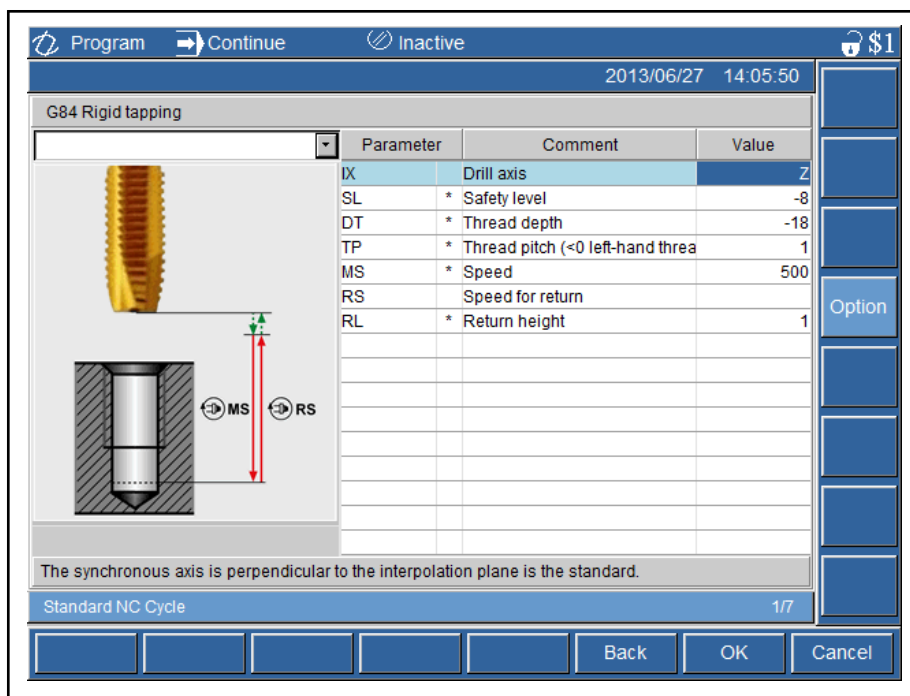
Fig.3-38:

Complete the input by pressing <F8>"OK".

```
N360 G84(IX Z,SL-8,DT-18,TP1,MS500,RL1)
```

The modal cycle G84 is processed after the next positioning. The point pattern G116 is used.

Select cycle "6" G116 Hole Pattern Frame in the group "5" (pattern of holes). Optionally, make entries directly in the input line "G116".

Entering technology data for the cycle G116

Specify the parameters for the machining cycle G116 in the cycle input mask:

MSP	-30	Starting point: main axis
SSP	-30	Starting point: secondary axis
TA	0	Angle
PD	60	Hole distance
NH	2	Number of drillings
VO	60	Boring distances, vertical
NL	2	Number of borings, vertical
MA	0	Angular offset

Tab.3-23:

Complete the input by pressing <F8>"OK".

```
N370 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
```

Insert the following NC blocks:

Deleting thread cutting cycle, modal

```
N380 G80 M5
```

Remove tool

```
N390 T0 M6
```

NC Program

```
;-----------------------------------------------------------------
;OP7 Manufacture thread 4xM6
N270 G511(T3,F250,S2000,DIR M3,ZG G54)
N280 G81(IX Z,SL-9,DT-13.1,DW0.5,RL1)
N290 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
N300 G80
N310 G511(T8,F300,S2000,DIR M3,ZG G54)
N320 G83(IX Z,SL-9,DT-13.1,CD6,RD1,DW0.5,RL1,LD1)
N330 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
N340 G80
N350 G511(T9,F0,S0,DIR M3,ZG G54)
N360 G84(IX Z,SL-8,DT-18,TP1,MS500,RL1)
N370 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
N380 G80 M5
N390 T0 M6
N990 M30
;-------------------- program end --------------------------------
```

Fig.3-39:

Saving the NC program   Save the program with <F7>"Save" before exiting the editor.
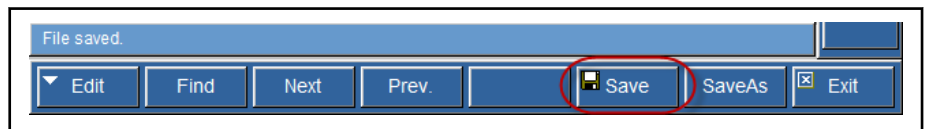
Fig.3-40:

3.3   Complete Program "FACE"

Program listing FACE

```
;Milling machine: MTX_micro
;NC program name: FACE.npg   drawing:           FACE.dxf
;Material:          Aluminium  Stock Dimension: 75mm x 75mm x 20mm
;Author:            Miller     Version: 1.0.0   Date: 26.06.2013
;----------------------- program start ---------------------------
;OP1 Face milling
N10 G511(T1,F500,S2000,DIR M3,ZG G54)
N20 G266(CV 3,ORI2,LE75,WI75,UL0.5,SDI2,DT0,RD2,CG100)
N30 G0 X0 Y0
N40 G80
;-----------------------------------------------------------------
;OP2 External contour circle, roughing and finishing
N50 G511(T2,F250,S2000,DIR M3,ZG G54)
N60 G46 G0 X0 Y0 Z2
N70 G277(HD70,UL0,SDI0,DT-10,RL2,RD2,AW4,RSW0.1,RSL0.1,FF150)
N80 G0 X0 Y0
N90 G80
;-----------------------------------------------------------------
;OP3 External contour, rhombus milling
N100 G276(ORI2,LE50,WI50,R5,PA45,UL0,SDI0,DT-5,RL2,RD2,AW4,RSW0.1,RSL0.1,FF150
N110 G0 X0 Y0
N120 G80
;-----------------------------------------------------------------
;OP4 Mill contour pocket with isle contours
N130 G511(T20,F200,S2000,DIR M3,ZG G54)
N140 G45
N150 G288(S2000,E2010)
N160 G289(MSP-10,SSP-10)
N170 G281(S1000,E1060,UL0,SDI1,RD0.1,DT-2.5,RL1)
;-----------------------------------------------------------------
;OP5 Curved slot, roughing and finishing
N180 G511(T17,F200,S2500,DIR M3,ZG G54)
N190 G275(WI10,R5,PA225,OA90,UL-2.5,SDI1,DT-4,RL1,RSW0.1,RSL0.1,FF180)
N200 X-10 Y-9
N210 G80
;-----------------------------------------------------------------
;OP6 Boring 2x10
N220 G511(T6,F250,S1500,DIR M3,ZG G54)
N230 G81(IX Z,SL-2,DT-7.5,RL1)
```

NC Program

```
N240 G0 X10 Y9
N250 G0 X-10 Y9
N260 G80
;-------------------------------------------------------------------
;OP7 Manufacture thread 4xM6
N270 G511(T3,F250,S2000,DIR M3,ZG G54)
N280 G81(IX Z,SL-9,DT-13.1,DW0.5,RL1)
N290 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
N300 G80
N310 G511(T8,F300,S2000,DIR M3,ZG G54)
N320 G83(IX Z,SL-9,DT-13.1,CD6,RD1,DW0.5,RL1,LD1)
N330 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
N340 G80
N350 G511(T9,F0,S0,DIR M3,ZG G54)
N360 G84(IX Z,SL-8,DT-18,TP1,MS500,RL1)
N370 G116(MSP-30,SSP-30,TA0,PD60,NH2,VO60,NL2)
N380 G80 M5
N390 T0 M6
N990 M30
;--------------------- program end --------------------------------
// description pocket contour
// begin
N1000 G1 X0 Y-25
N1010 X5 RNDL(5)
N1020 X30 Y0 RNDL(12)
N1030 X0 Y30 RNDL(20)
N1040 X-30 Y0 RNDL(12)
N1050 X-5 Y-25 RNDL(5)
N1060 X0
// end
;-------------------------------------------------------------------
// description island contour
// begin
N2000 X0 Y7.5
N2010 G2 I0 J1.5
// end
;-------------------------------------------------------------------
```

3.4 Simulating the NC Program "FACE"

3.4.1 Entering the Zero Point Offset

The following values for the zero point offset can be calculated for clamping of the unmachined part by taking the allowances of the unmachined part and the clamping device into consideration:

G54.1 X = 217.5 Y = 193.5 Z = 214.5 mm

NC Program
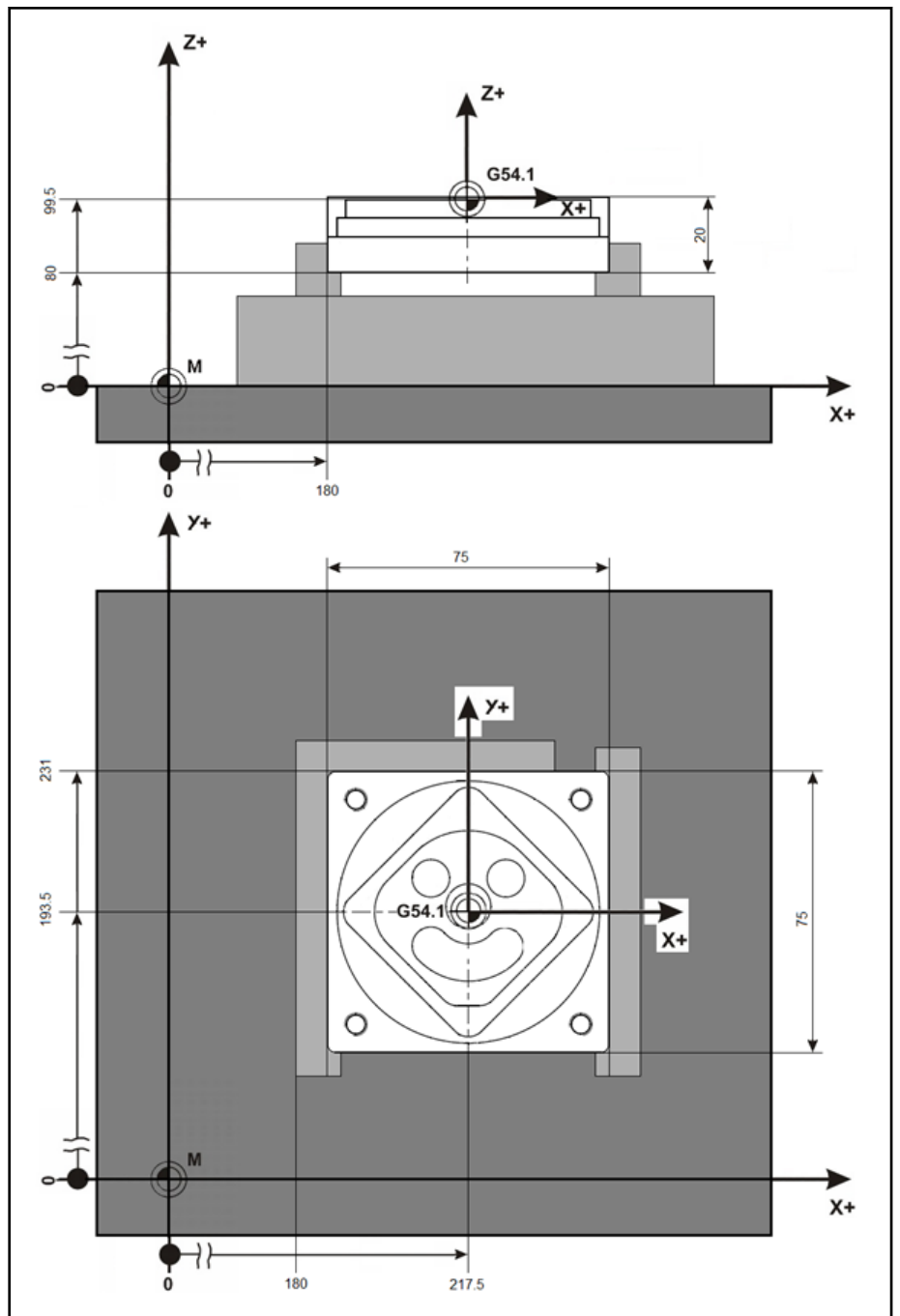
Clamping sketch

Fig.3-41:   Clamping sketch

Changinging the operating area   Press the key "Machine" at the control panel.
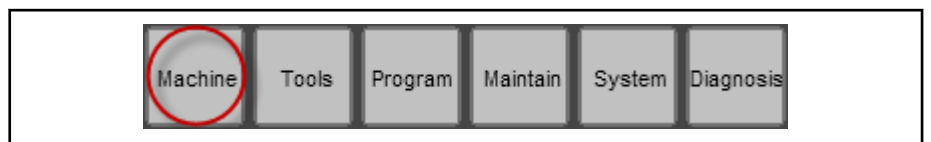
Fig.3-42:

Selecting the zero point table and enter the offsets   Press <F6>"ZeroOffs".

Select the input field for G54.1 using the cursor keys.

NC Program

Please enter the following values:

- Column X: Value 217.5
- Column Y: Value 193.5
- Column Z: Value 99.5
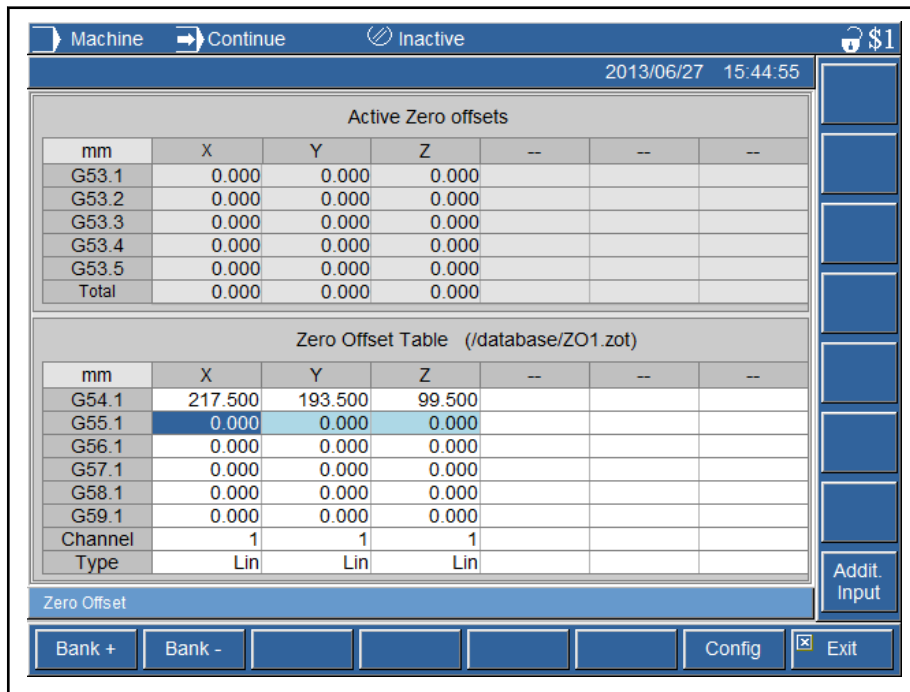
Fig.3-43:   Zero point table

Press <F9>"Exit".

☞   **Note:** The values used for zero points are only examples. The actual values have to be determined and entered at the machine.

3.4.2   Operating the Simulation

☞   Axis motions, tool change, etc. are not automatically disabled when simulating the NC programs at the machine.

        Please note the operating instructions of your machine!

The program "FACE" can be simulated to check for programming and input errors. The required operation of the **IndraMotion MTX micro Trainer** is described as follows:

Changinging the operating area   Press the key "Machine" at the control panel.
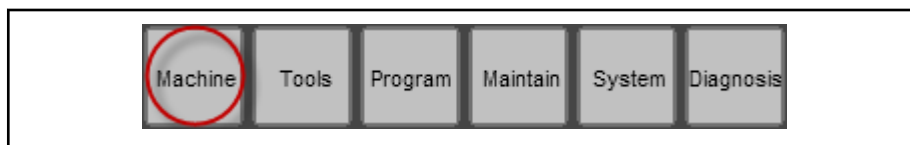
Fig.3-44:

Changing the operation mode   Press the key <Automatic> key on the control panel:
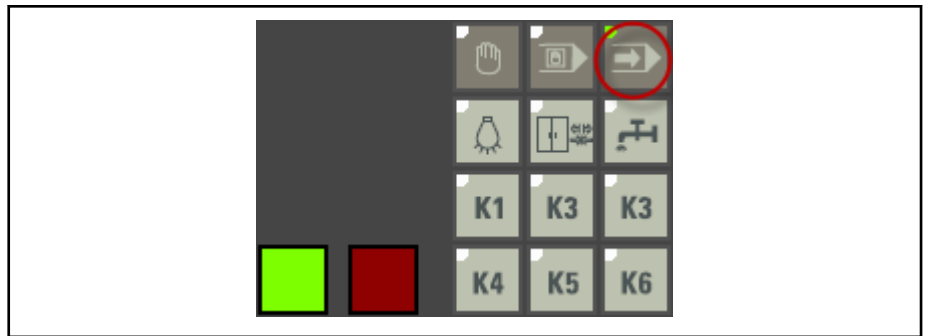
NC Program

Fig.3-45:

Selecting program   Press <F2>"SelProg":
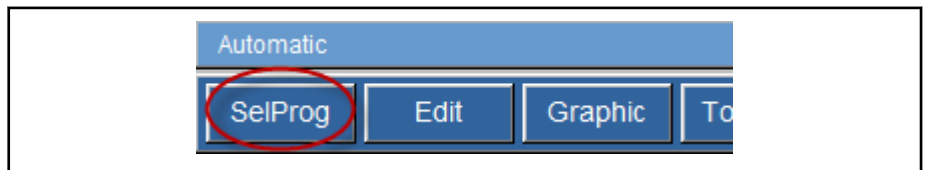
Fig.3-46:

Select the program "FACE":

☞   **Note:** Navigate using the <TAB> key as well as the cursor keys
              <up> and <down>.
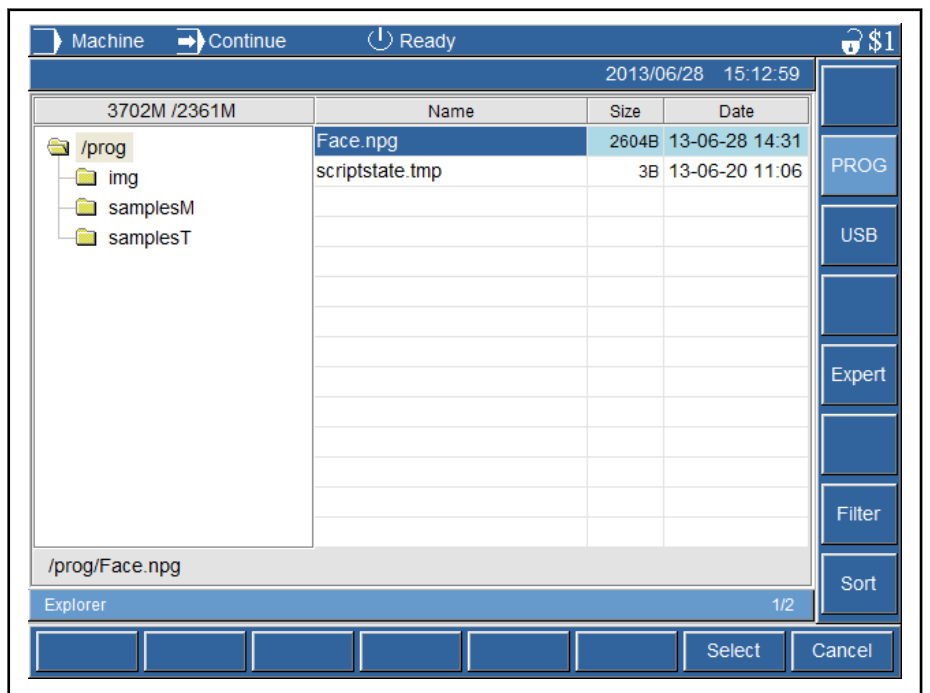
Fig.3-47:

Confirm the selection with <F8>"Select".

Start program   The program selected for processing is now shown on the program display.
The green key <NC Start> flashes on the control panel.

NC Program
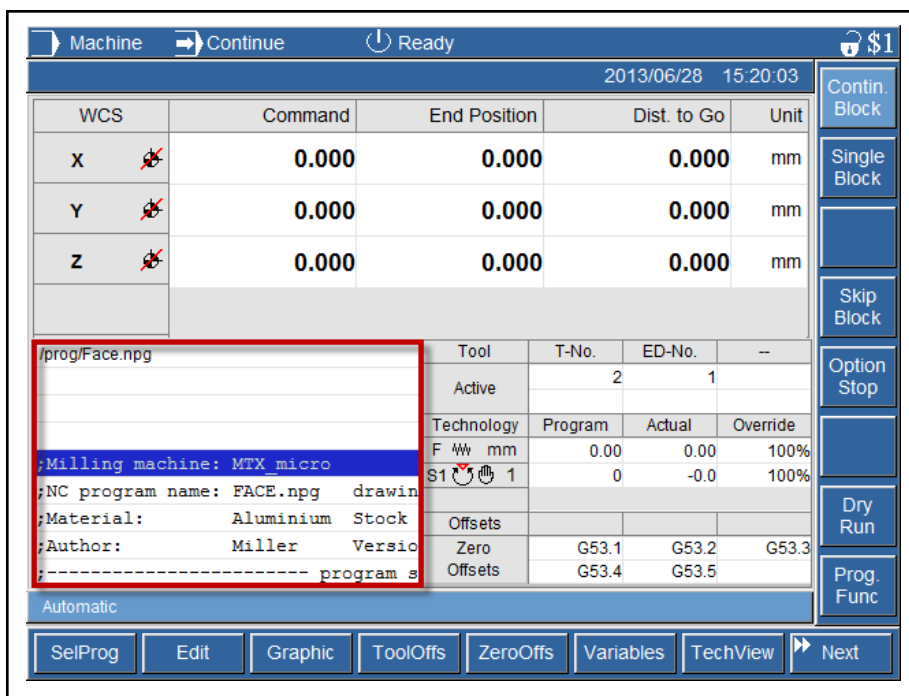
Fig.3-48:

Setting the graphic display   Press <F4>"Graphic", the graphic display opens. Press <F7>"Isometric" to view the coordinate system in 3D:
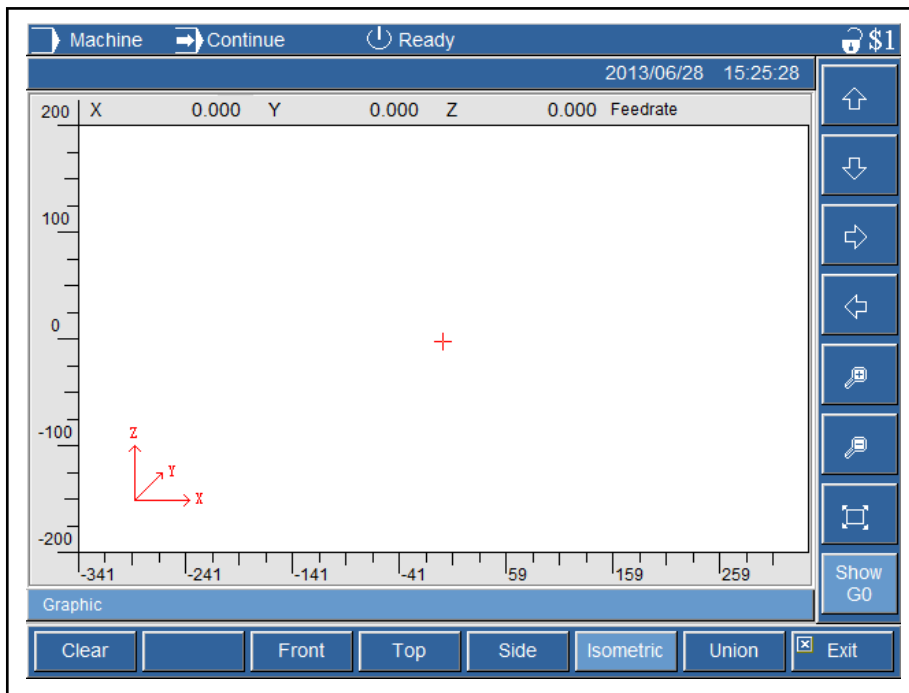
Fig.3-49:

Start simulation   Press the key <Start> at the control panel.

While the graphic is drawn, enable "Autozoom" by pressing the M-key 7. Using "Autozoom", scale and picture detail are adjusted automatically so that all programmed motions are visible. Using the direction keys <M-Key 1> to <M-Key 4>, the graphics can be moved in the display window and can be zoomed in and zoomed out using the <M-Key 5> and <M-Key 6>. Using <M-

NC Program

Key 8>"Show G0", the rapid traverse motions (highlighted in red) can be shown or hidden.
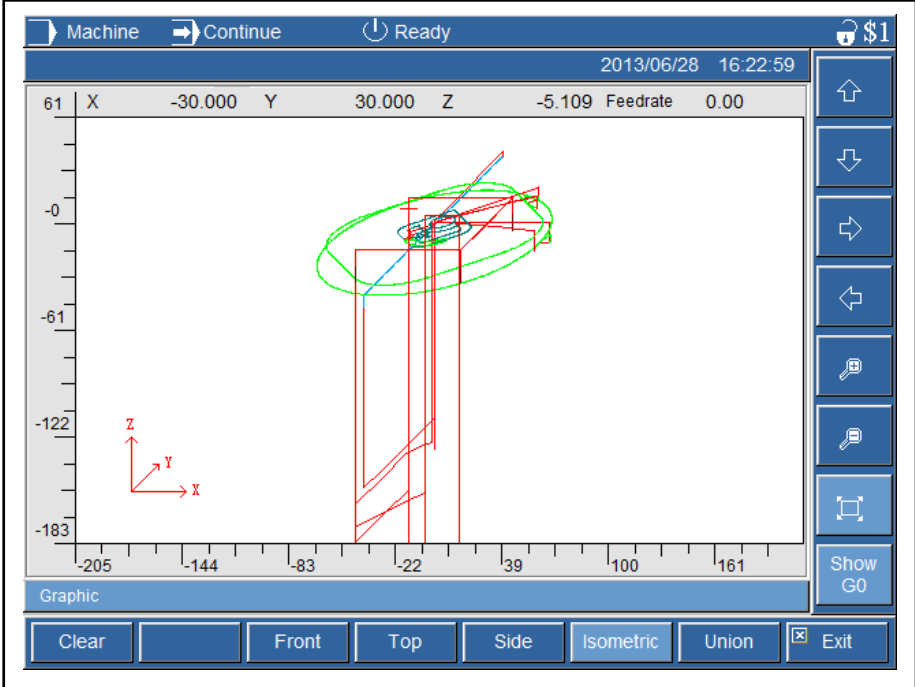
Fig.3-50:

4   Service and Support

Our worldwide service network provides an optimized and efficient support. Our experts offer you advice and assistance should you have any queries. You can contact us **24/7**.

Service Germany

Our technology-oriented Competence Center in Lohr, Germany, is responsible for all your service-related queries for electric drive and controls.

Contact the **Service Helpdesk & Hotline** under:

Phone:	**+49 9352 40 5060**
Fax:	**+49 9352 18 4941**
E-mail:	service.svc@boschrexroth.de
Internet:	http://www.boschrexroth.com

Additional information on service, repair (e.g. delivery addresses) and training can be found on our internet sites.

Service worldwide

Outside Germany, please contact your local service office first. For hotline numbers, refer to the sales office addresses on the internet.

Preparing information

To be able to help you more quickly and efficiently, please have the following information ready:

- Detailed description of malfunction and circumstances resulting in the malfunction
- Type plate name of the affected products, in particular type codes and serial numbers
- Your contact data (phone and fax number as well as your email address)

Index

Notes

R911341439

DOK-MTXMIC-TRAIN*MILL*-AP01-EN-P